Tekla Structures 2023

Create models

April 2023

Contents

1 Get to know Tekla Structures basic working methods

Before starting to create models and drawings, get to know some basic working methods that you will need to work efficiently both in the Tekla Structures models and in the drawings.

We recommend you to familiarize yourself how to

- work with grids (page 20) and views
- define the work area (page 48) and coordinates (page 51) that affect your work
- zoom and rotate (page 81) the model
- create (page 196), select (page 113) and move (page 140) objects
- position objects by using snapping (page 83)
- filter objects (page 150) both in the modeling mode and in the drawing mode.

See also

Create parts and modify part properties (page 196)

1.1 Set up the workspace

Before starting to model, check that your Tekla Structures workspace is set up correctly.

1. Define the units and decimals you will use. (page 20)
2. Modify the grid to suit your needs. (page 20) Create a modular grid if needed.
3. Create some views (page 31) to examine the model from different angles and elevations.

4. Resize the work area to suit your project. (page 48)
5. Get familiar with the coordinate system (page 51). If you are modeling sloped structures, shift the work plane accordingly. (page 53)

Change units and decimals

You can define which units and how many decimals Tekla Structures uses. The settings are model-specific. Note that these settings do not have any effect on drawings or reports, or on the **Inquire** and **Measure** tools.

1. On the **File** menu, click **Settings** --> **Options** , and go to the **Units and decimals** settings.
2. Modify the units and decimals to suit your needs.

   The number located to the right of each option indicates the number of decimals. The number of decimals affects the input and storage accuracy. Always use a sufficient number of decimals.

   - The settings on the **Modeling** tab affect the data that is used when you are modeling, for example copying, moving, creating grids, creating points, and so on.
   - The settings on the **Catalogs** tab affect the profile and material data, for example catalogs.
   - The settings on the **Analysis results** tab affect the output data.

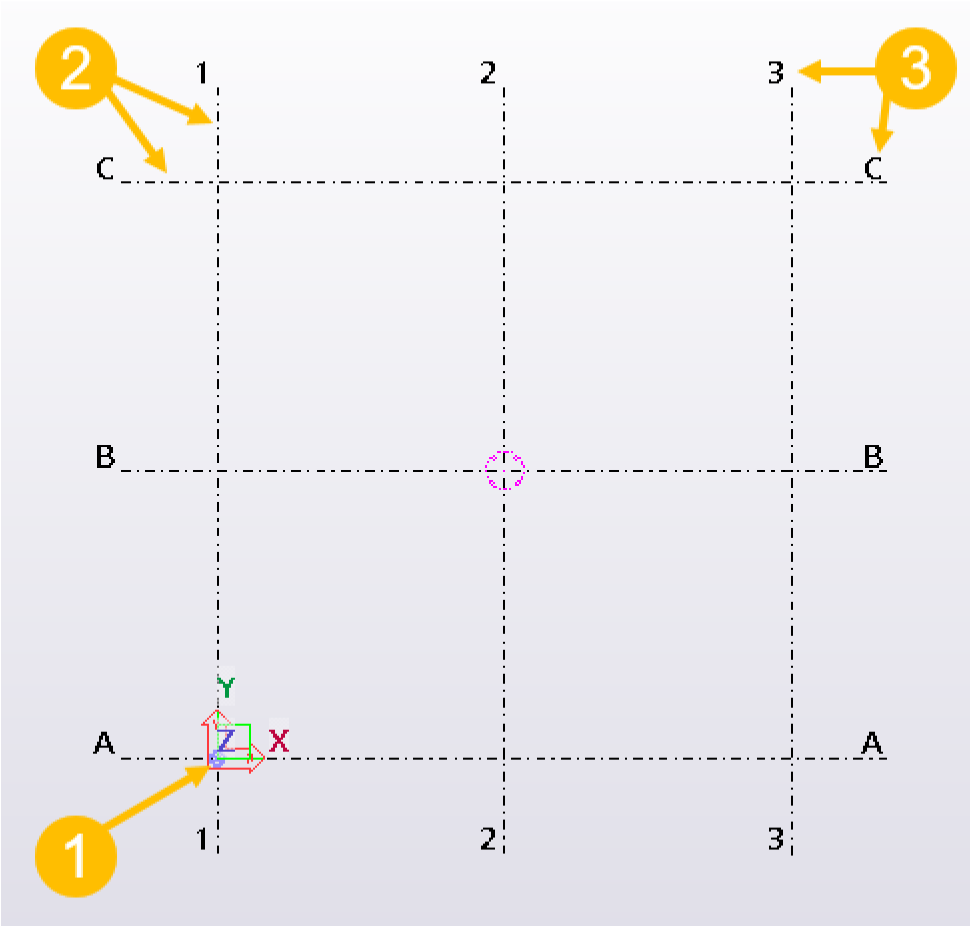
     Reinforcement area and **Mass/Length** also affect cross section area and weight per unit length in the rebar catalog.
   - The settings on the **MEP** tab affect the data that is used in mechanical, electrical, and plumbing design and construction.
3. Click **OK** to save the changes.

Work with grids

A *grid* represents a three-dimensional complex of horizontal and vertical planes. The grid is shown on the view plane using dash-and-dot lines. You can create both rectangular and radial grids. Use grids as an aid in locating objects in a model. You can make rectangular grids and *grid lines* act magnetically so that the objects on the grid lines of a rectangular grid follow if you move the grid line.

- Create, modify, or delete a grid (page 23)
- Add a single grid line (page 28)
- Modify a single grid line (page 28)

Grid terminology



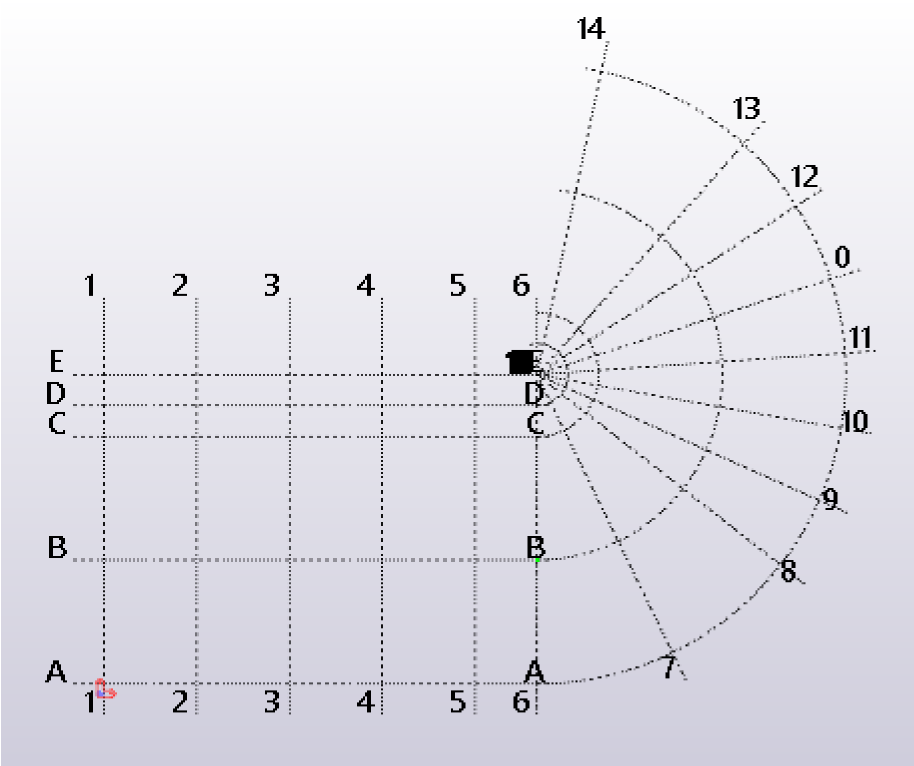
(1) Grid origin is the point where the zero points of each coordinate axis intersect

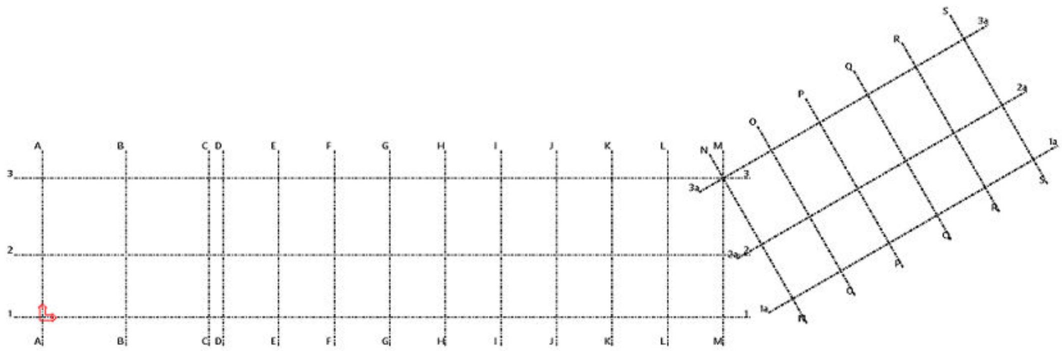
(2) Grid line extensions define how far the grid lines extend in each direction

(3) Grid labels are the names of the grid lines shown in views

Several grids in one model

You can have more than one grid in a model. You can create a large-scale grid for the entire structure, and smaller grids for some detailed sections. Create as many grids as needed, so that you can easily place objects in your model.

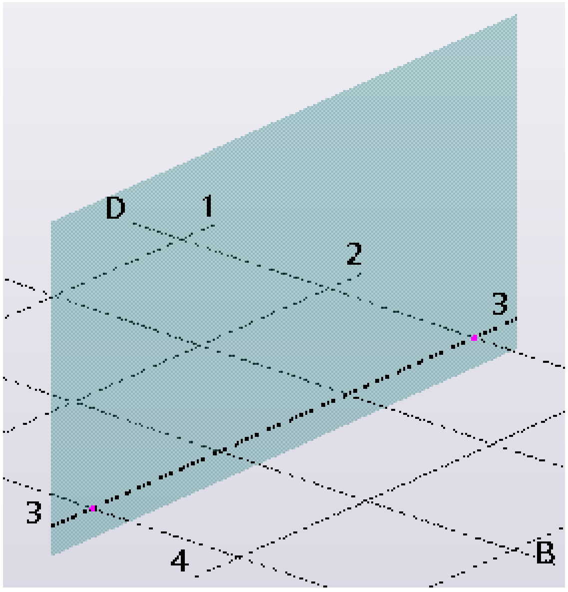


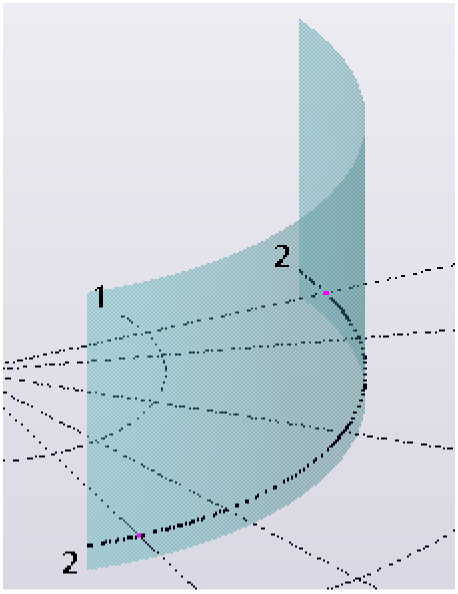


Single grid lines

You can create single grid lines and attach them to an existing grid.

Single grid lines have handles (page 307). If the **Select grid line** selection switch is active and you select a grid line, the handles appear in magenta. If you move the handles to make a skewed grid, you can do this only on the local XY plane (page 31) of the grid.



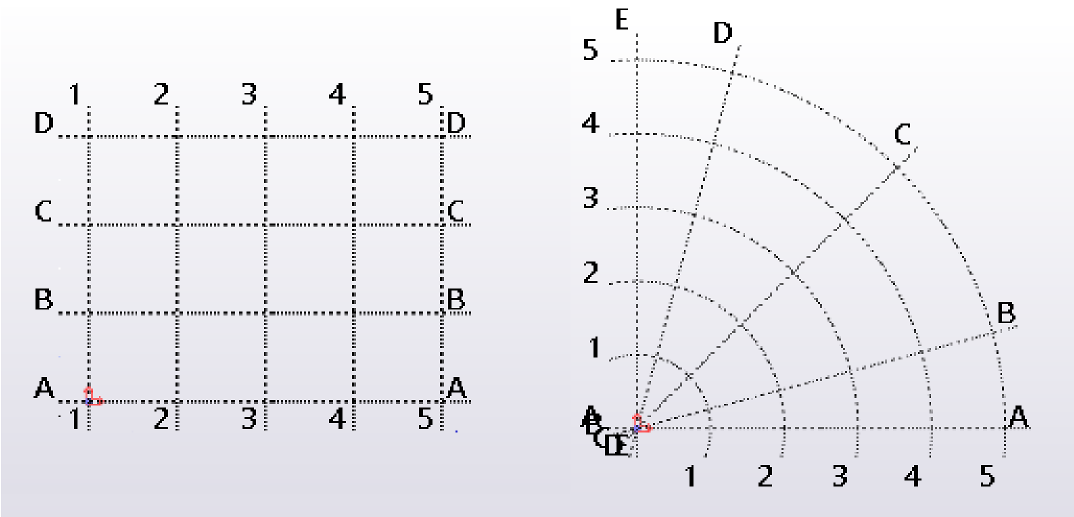


Create, modify, or delete a grid

When you create a new model, Tekla Structures automatically creates a rectangular grid and a view according to the saved standard properties. If needed, you can later modify the grid properties, or create new rectangular and radial grids. You can modify an existing grid, or delete it. Use the grid properties in the property pane to view and modify the grids.

Create a rectangular grid or a radial grid

You can create both rectangular and radial grids and modify their properties in the property pane.



1. Select whether to create a rectangular or a radial grid.
    - To create a rectangular grid: on the **Edit** tab, click **Grid** --> **Create rectangular grid** .
    - To create a radial grid: on the **Edit** tab, click **Grid** --> **Create radial grid** .

    Tekla Structures shows a preview of the grid. You can modify the grid properties in the property pane before inserting the grid. The grid preview changes according to the changes in the property pane.
2. Pick a point in the model to indicate the origin of the grid, or click the middle mouse button to accept the grid property values in the property pane.
    - If you pick an origin point to the grid, the grid is created using the properties in the property pane, and the origin is the location you picked.
    - If you click the middle mouse button, the grid is created using the properties in the property pane, and to the origin defined in the property pane.

      The coordinates of the origin are shown in the property pane in the **Origin** section as the **X**, **Y**, and **Z** values.

NOTE When working with very large grids, always having the grid labels visible might slow down Tekla Structures. To hide the grid labels when you zoom in, use the advanced option XS_ADJUST_GRID_LABELS.

Create a radial grid (alternative method)

This is an alternative method to create a radial grid. You can create a radial grid using a component called **Radial Grid**. Note that the curved grid lines created by the **Radial Grid** component are not truly curved but straight.

1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
2. Start typing `radial grid` in the search box.
3. Double-click **Radial Grid** to open the properties dialog box.
4. Modify the grid properties.

   In the coordinate properties

   - **X** defines the location of the curved grid lines and the distance between the grid lines.

     The first value is the radius of the innermost curved grid line.

   - **Y (degrees)** defines the location of the straight grid lines and the distance between the grid lines in degrees.

     The first value defines how the grid is rotated. The grid is rotated counter-clockwise from the x axis in the current work plane.
5. Click **OK**.
6. Pick a point to indicate the origin of the grid.

   The grid is created automatically.

Modify a grid

Double-click an existing grid to modify it.

1. Ensure that the **Select grid** selection switch is active.
2. Double-click a grid line.

   Depending on the type of the grid, the **Rectangular grid** or the **Radial grid** properties open in the property pane.
3. Change the grid properties.
4. Click **Modify** to apply the changes.

Delete a grid

When you delete an entire grid, ensure that you do not have any other objects selected. Otherwise Tekla Structures only deletes the objects, not the grid.

1. Ensure that **only** the **Select grid** selection switch is active.
2. Select the grid.

3. Press **Delete**.
4. Confirm that you want to delete the grid.

Grid properties

Use the **Rectangular grid** or the **Radial grid** properties in the property pane to view and modify the properties of a grid. To open the properties, double-click the grid. The file name extension of a rectangular grid property file is `*.grd`. The file name extension of a radial grid property file is `*.rgrd`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description

Coordinates	In the **Rectangular grid** properties:
	
	Coordinates of the grid in the x, y, and z directions.
	
	X: grid lines vertical to the work plane
	
	Y: grid lines horizontal to the work plane
	
	Z: elevations in the structure
	
	You can enter a maximum of 1024 characters. Use a zero at the start to represent a grid on the 0,0 coordinate and spaces as separators for coordinates.
	
	The x and y coordinates are relative, which means that the entries for **X** and **Y** are always relative to the previous entry. The **Z** coordinates are absolute, which means that entries for z are absolute distances from the work plane origin.
	
	You can either define the coordinates individually, or you can define several grid lines with equal spacing. Both of the following coordinate entries create three grid lines with the spacing of 4000:
	
	`0 4000 4000`
	
	`0 2*4000`

	In the **Radial grid** properties: Radial and angular values of the grid lines. Elevations is the value of the grid in z direction. **Radial**: curved grid lines. If you enter only one value for radius, it must be >0. **Angular**: straight grid lines. You can enter negative angle values, if needed. **Elevations**: elevations in the structure

Setting	Description

	Use a zero at the start to represent a grid on the 0,0 coordinate and spaces as separators for coordinates.
	
	The **Radial** and **Angular** coordinates are relative, which means that the entries are always relative to the previous entry. The **Elevations** are absolute, which means that entries are absolute distances from the work plane origin.
	
	You can either define the coordinates individually, or you can define several grid lines with equal spacing. Both of the following coordinate entries create three grid lines with the spacing of 4000:
	
	`0 4000 4000`
	
	`0 2*4000`

Labels	In the **Rectangular grid** properties: Names of the grid lines shown in views. Names in the **X** box are associated with the grid lines parallel to the y axis and vice versa. The **Z** box is for the names of levels parallel to the work plane. If you wish, you can leave the label boxes empty.
	In the **Radial grid** properties: Names of the grid lines shown in views. Names in the **Radial** box are associated with the curved grid lines. Names in the **Angular** box are associated with the straight grid lines. The **Elevations** box is for the names of levels parallel to the work plane. If you wish, you can leave the label boxes empty.
Line extensions	In the **Rectangular grid** properties: Define how far the grid lines extend in the directions **Left/Below** and **Right/Above**.
	In the **Radial grid** properties: Define how far the grid lines extend in the directions **Start** and **End**.
Origin	Coordinates of the grid origin in the x, y, and z directions. These values offset the grid from the work plane origin, not from the global model origin.

Setting	Description
Magnetic grid plane	In the **Rectangular grid** properties: Select whether objects are bind to grid lines (page 28). If you bind the objects to grid lines, the objects follow if you move the grid line.
Grid color	Use the color picker to pick the grid color.
Label font size and color	Define the label font size. Use the color picker to pick the label color.
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (UDAs) of the grid. These can be used for customizing drawing grid labels.

Add a single grid line

You can add new grid lines either between existing grid lines or between two freely chosen points that you define in the model.

Add a grid line between existing grid lines

You can add new grid lines between existing grid lines.

1. Ensure that the **Direct modification** switch is active.
2. Ensure that the **Select grid** selection switch is active.
3. Select an existing grid to attach the grid line to.
4. Click the symbol between two existing grid lines or outside the grid.

   Tekla Structures creates the grid line and gives it a label using the labels of the adjacent grid lines. For example, a new grid line between the grid lines 1 and 2 receives the label 12*.

Add a grid line between two points

You can add new grid lines between two picked points.

1. On the **Edit** tab, click **Grid** --> **Add grid line** .
2. Select an existing grid to attach the grid line to.
3. Pick the start point of the grid line.
4. Pick the end point of the grid line.

   Tekla Structures creates the grid line.

Modify a single grid line

You can modify single grid line properties. You can also move grid lines or change the grid line labels.

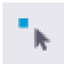
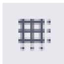
Modify grid line properties

You can edit the properties of a single grid line in the property pane.

1. Ensure that the  **Select grid line** selection switch is active.
2. If the property pane is not open, double-click the grid line to open the **Grid line** properties.
3. Change the properties as needed.
4. Click **Modify** to apply the changes.

Move a grid line

Use direct modification to move single grid lines.

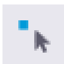
1. Ensure that the  **Direct modification** switch is active.
2. Ensure that the  **Select grid** selection switch is active.
3. Select the grid.
4. Select the grid line you want to move.
5. Drag the grid line to a new location.

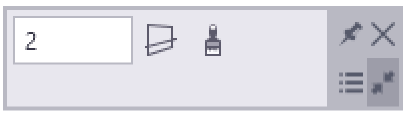
   You can also use the keyboard to enter a numeric location.

   To start with the negative sign (-), use the numeric keypad. To enter an absolute coordinate, first enter $, then the value. Press **Enter** to confirm.

Change a grid line label

Use the contextual toolbar to change the label of a single grid line.

1. Ensure that the  **Direct modification** switch is active.
2. Ensure that the  **Select grid line** selection switch is active.
3. Select a grid line.
4. On the contextual toolbar, enter a new label.



Stretch, shrink, or incline a grid line

Use direct modification to stretch, shrink, or incline single grid lines in rectangular grids.

Note that this works only for the single grid lines that you have added in rectangular grids by using the **Add grid line** command.

1. Ensure that the **Direct modification** switch is active.
2. Ensure that the **Select grid** selection switch is active.
3. Select the rectangular grid.
4. Select the grid line.
5. Drag a grid line handle to a new location.

Turn grid line stretching off

If you move the outermost grid lines in rectangular grids using the line handles, Tekla Structures stretches or shrinks the perpendicular, crossing grid lines accordingly by default. You can switch this off temporarily.

1. Ensure that the **Direct modification** switch is active.
2. Ensure that the **Select grid** selection switch is active.
3. Select the grid line.
4. On the contextual toolbar, click the **Turn grid line stretching off** button.

Grid line properties

Use the **Grid line** properties in the property pane to view and modify the properties of a single grid line. To open the properties, double-click a single grid line. The file name extension of a grid line property file is `.grdp`.

The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals** .

Setting	Description
Label	Name of the grid line.
Depth in view plane	Height of the grid plane perpendicular to the view plane.

Setting	Description
Line extension Left/Below	Define how far the grid lines extend in the directions **Left/Below** and **Right/ Above**.
Line extension Right/Above	
Magnetic	Select whether objects are bind to straight grid lines. If you bind the objects to grid lines, the objects follow if you move the grid line.
Visible in drawing	Select whether grid lines are made visible in drawings.
Grid line automatic dimensioning	Select whether single grid lines are used in grid dimensioning.
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (UDAs) of the grid line.

Delete a single grid line

You can delete grid lines in two different ways. The easiest way is by using direct modification.

Delete a grid line using direct modification

Use direct modification to quickly delete single grid lines.

1. Ensure that the **Direct modification** switch is active.
2. Select the grid line you want to delete.
3. Press **Delete**.

Delete a grid line (alternative method)

This is the alternative way of deleting single grid lines.

1. Ensure that the **Select grid line** selection switch is active.
2. Select the grid line you want to delete.
3. Ensure that you do not have any other objects selected.

   If you also have other objects selected, Tekla Structures only deletes the objects, not the grid line.
4. Right-click and select **Delete** from the pop-up menu.
5. Confirm that you want to delete the grid line.

Work with views

A *view* is a representation of a model from a specific location. Each view is represented in its own window within Tekla Structures. Selecting a part in a view highlights the part in all open views.

- Create model views (page 33)
- Open, save, modify, or delete a view (page 43)
- Switch between views (page 45)
- Change the color settings for dimensions, labels, and model background (page 69)

View plane

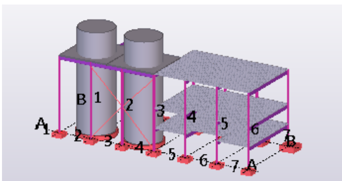
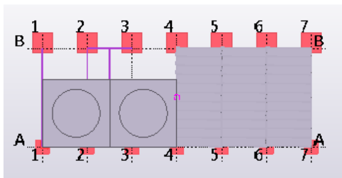
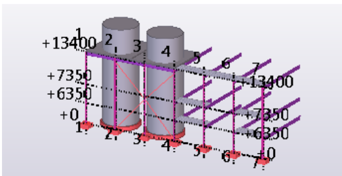
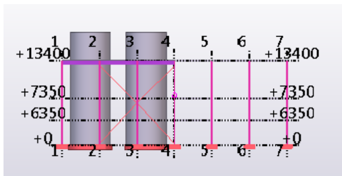
Each view has a view plane on which the grids (page 20) are visible and points (page 631) are represented as blue crosses. Points that are located outside the view plane are red. You can move the view plane (page 33) like any other object.

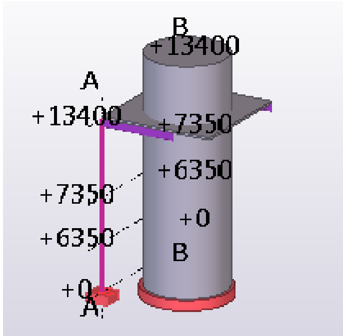
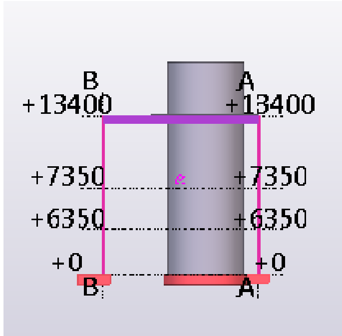
Basic views

Basic views are those parallel to the global basic planes (xy, xz, and zy). In basic views, two axes always define the view plane and the axes appear in the plane name. The third axis is perpendicular to the view plane. It does not appear in the plane name. In the basic plane view, the model is shown from the direction of the third axis.

When you create basic views (page 34), you must define the view plane's distance (the view plane coordinate) from the global origin in the direction of the third axis.

Examples of basic views:

Plane	3D view	Plane view
XY		
XZ		

Plane	3D view	Plane view
ZY		

Other views

For other view types, you either define the view plane and coordinate by picking points, or the points are defined automatically, depending on the creation method.

Should I model in a 3D or plane view?

Views can be 3D or plane views. 3D, plane, and also elevation views provide different type of information, which is useful for different tasks.

One common technique is to open several views:

- A 3D view to see a real-life version of the model
- A plane view, where you can add and connect parts
- An elevation view to check the level

If you are working with several screens, maximize your work area by putting the different views on different screens.

You can easily switch between the 3D and plane view (page 31) by using the keyboard shortcut **Ctrl+P**.

Move the view plane

You can move the view plane like any other object. When you move it, Tekla Structures only uses the vector that is perpendicular to the view plane.

1. Click the view.
2. Right-click and select **Move** --> **Linear**.
3. Pick the start point of the translation vector, or enter its coordinates.
4. Pick the end point of the translation vector, or enter its coordinates.
5. Click **Move** to move the view plane.

Create model views

You can create model views in several ways in Tekla Structures.

For example, you can create views

- of the entire model (3D view) (page 34)
- of selected parts (page 39)
- of selected components (page 40)
- along grid lines (axis views) (page 35)
- to the surface of an object (page 40)

Each view has properties which define its appearance. You can change the appearance of a view after you have created it. To view and modify the properties of each view (page 46), double-click anywhere on the model background to open the **View Properties** dialog box.

If you create views along grid lines, you can view and modify the properties (page 48) using the **Creation of Views Along Grid Lines** dialog box.

If you need to reopen views for later use, you can name and save (page 43) them. Tekla Structures only saves the named views when you exit the model.

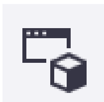
Create a basic view of the model
You can create a basic view along two coordinate axes. Use this view for the overall viewing of the model.

1. On the **View** tab, click  **New view** --> **Basic view** .
2. Select a view plane from the **Plane** list.

   The view plane is defined by two axes.
3. In the **Coordinate** box, enter the view level.

   This value defines the distance of the view plane from the global origin, perpendicular to the view plane.
4. Click **Create**.

Create a view using two points
You can create a view using two points you pick: the origin and a point in the direction of the x axis.

1. On the **View** tab, click  **New view** --> **Using two points** .
2. Pick a point to indicate the origin of the view plane.
3. Pick a second point to indicate the direction of the x axis.

   The y axis is perpendicular to the view plane on which you picked the second point.

Create a view using three points

You can create a view using three points you pick: the origin, a point in the direction of the x axis, and the third point in the direction of the y axis.

1. On the **View** tab, click **New view** --> **Using three points** .
2. Pick a point to indicate the origin of the view plane.
3. Pick a second point to indicate the direction of the x axis.
4. Pick a third point to indicate the direction of the y axis.

Create a view of the work plane

You can create a view of the work plane using the current view properties.

- On the **View** tab, click **New view** --> **On work plane** .

Create grid views

You can create views along the grid lines you select.

Before you start, create a view that contains a grid, and check the grid properties. Note that with radial grid you can create the grid views only on the straight grid lines, not on the arched grid lines.

If the grid properties are incorrect in some way, Tekla Structures may cut the views at the wrong elevations or they may be named incorrectly. If you change the grid labels or the elevation or grids later on, the views will not be automatically renamed.

1. Select the grid.
2. On the **View** tab, click **New view** --> **Along grid lines** .
3. Modify the grid view properties if needed.
   a. In the **Number of views** list, select how many views you want to create.
   b. In the **View name prefix** box, enter a prefix.
   c. In the **View properties** list, define which view properties (applied or saved) you want to use.
4. Click **Create**.

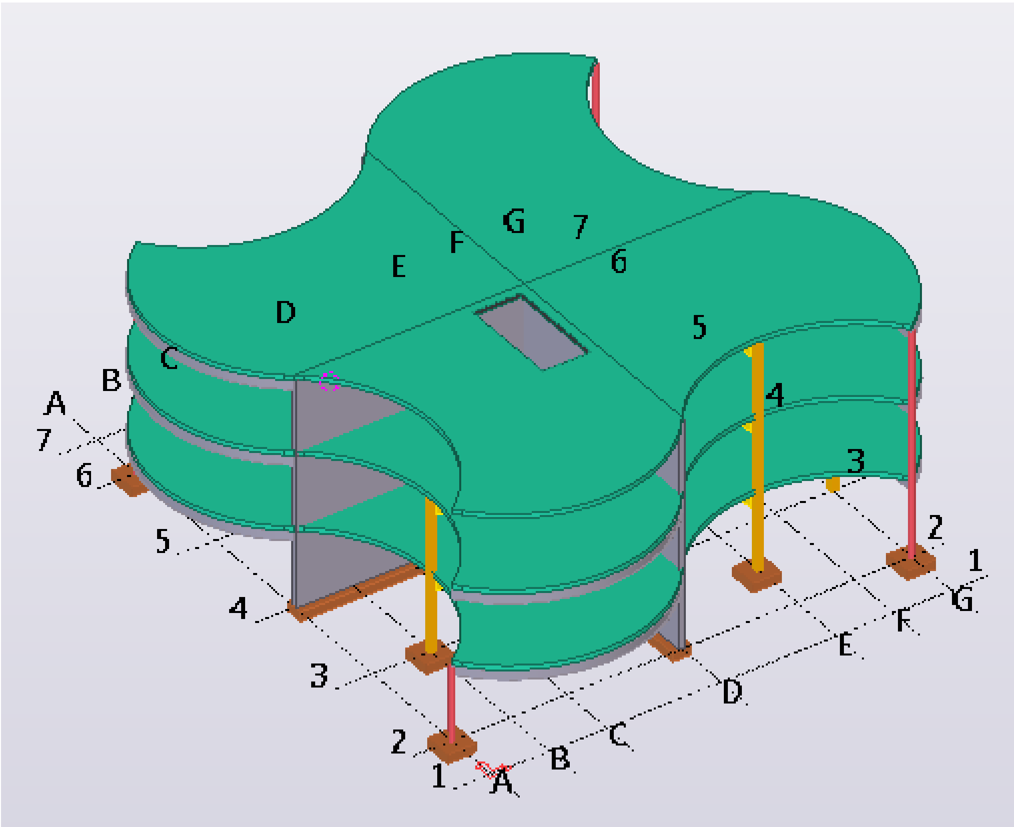
   The **Views** dialog box opens.

5. Click the arrow buttons to move views from the **Named views** list to the **Visible views** list.

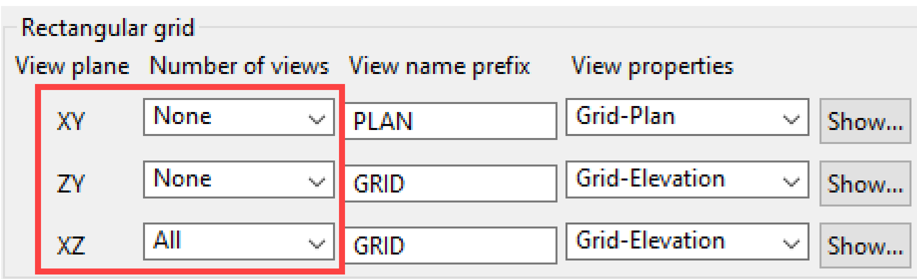
   The views will not be visible until you move them to the **Visible views** list.

Example

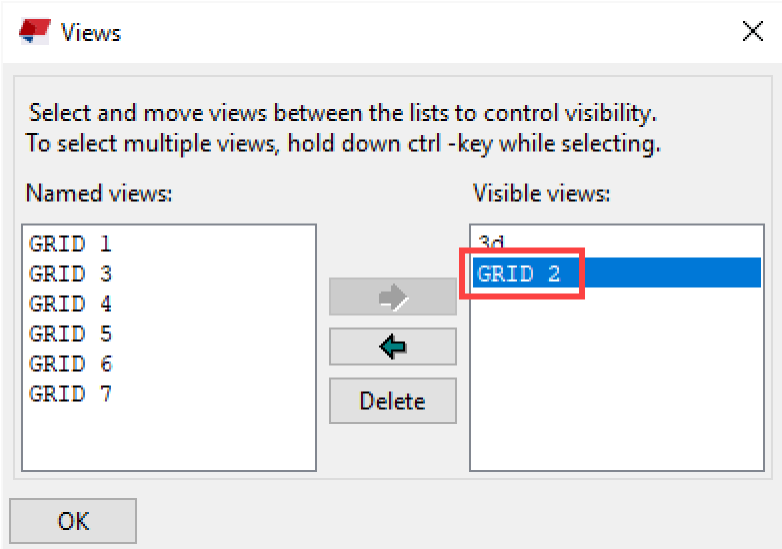
This example shows how to create vertical views of the grid lines 1–7 on the following model:



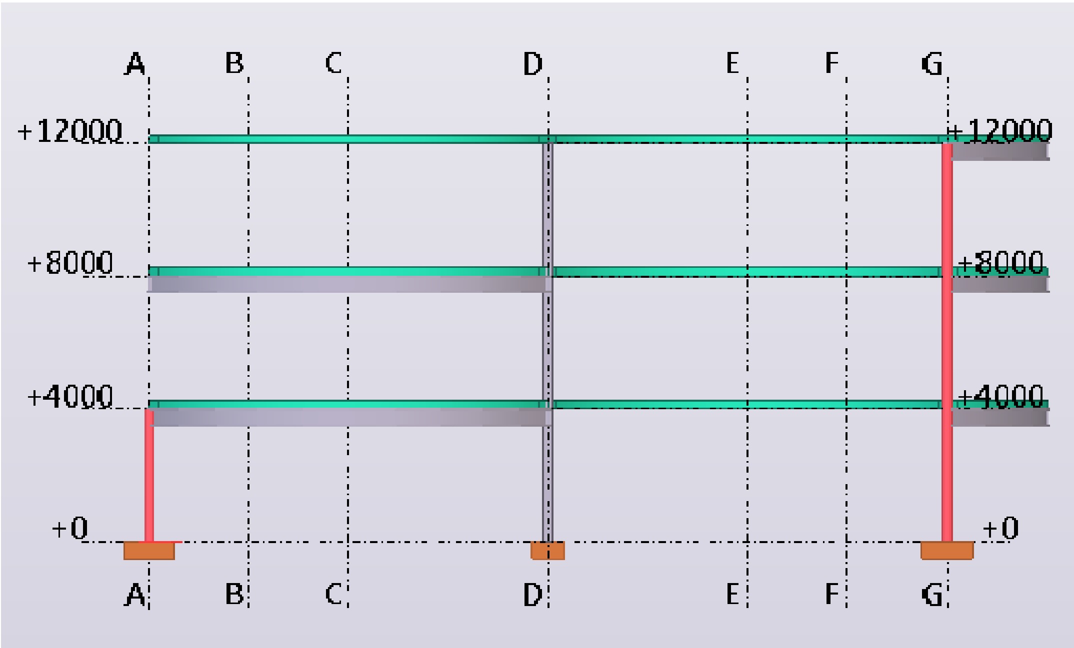
In the **Creation of views along grid lines** dialog box, select **All** for the view plane XZ and **None** for the view planes XY and ZY. Use the default settings for the view name prefix and the view properties.



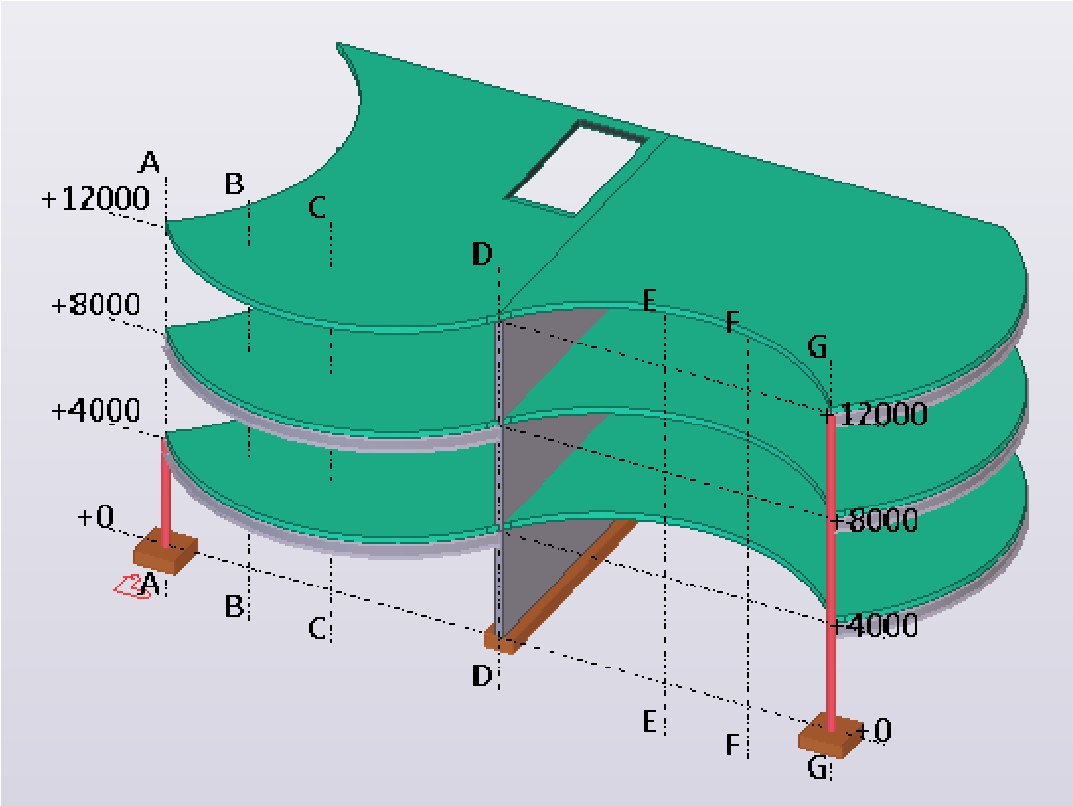
After creating the grid views, move the view named **Grid 2** to the **Visible views** list:



The grid view is displayed as a plane view in a new window:

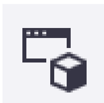


Rotate the view to see it in 3D:



Create a view on a plane

You can create a view on the work plane or almost any plane of an existing part.

1. On the **View** tab, click  **New view** --> **On plane** .

   When you hover the mouse cursor over model objects, Tekla Structures highlights the available planes in light blue.

2. Select the desired plane.

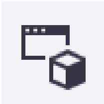
TIP Alternatively, to create a view on the front, top, back, or bottom plane of a part, you can also use the following commands available on the **View** tab:

- **On part front plane**
- **On part top plane**
- **On part back plane**
- **On part bottom plane**

To use these commands, first select a command and then the part.

Create a 3D view of a part

When you need to see a specific part clearly, create a 3D view of the part. The part is placed in the center of the view.

1. On the **View** tab, click  **New view** --> **3D view of part** .
2. Select the part.

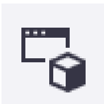
   Tekla Structures creates the view using the properties defined in the `part_basic_view` property file. The view plane y axis is the global z axis of the model. The x axis is the projection of the part's local x axis onto the global xy plane.

   Tekla Structures fits the work area to include the selected part.

   If you want Tekla Structures to rotate the view once whenever you create a new 3D view, select the Basic view auto rotation setting in the **File** menu.

Create default part views

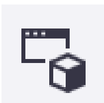
You can create four views of a part: front, top, end, and 3D view. Tekla Structures creates these views all at once with the same command. The front, top, and end views are plane views by default.

1. On the **View** tab, click  **New view** --> **Default part views** .
2. Select the part.

   Tekla Structures creates the four default views all at once, using the properties defined in `part_front_view`, `part_top_view`, `part_end_view`, and `part_persp_view` property files.

Create an undeformed part view

You can create a view that shows a deformed part in undeformed form. This works only for beams and columns.

1. On the **View** tab, click  **New view** --> **Undeformed part view** .
2. Select the part.

   For example, select a warped beam. Tekla Structures displays the beam in a separate view in undeformed form.

Create a 3D view of a component

When you need to see a specific component clearly, create a 3D view of the component. The component is placed in the center of the view.

1. On the **View** tab, click **New view** --> **3D view of component** .
2. Select the component.

   Tekla Structures creates the view, using the properties defined in the `component_basic_view` property file. The view plane y axis is the global z axis of the model. The x axis is the projection of the first secondary part local x axis onto the global xy plane. Work area depth is 1 m in all directions.

   The component you select automatically defines the work area.

   If you want Tekla Structures to rotate the view once whenever you create a new 3D view, select the Basic view auto rotation setting in the **File** menu.

Create default component views

You can create four views of a component: front, top, end, and 3D view. Tekla Structures creates these views all at once with the same command. The front, top, and end views are plane views by default.

1. On the **View** tab, click **New view** --> **Default views of component** .
2. Select the component.

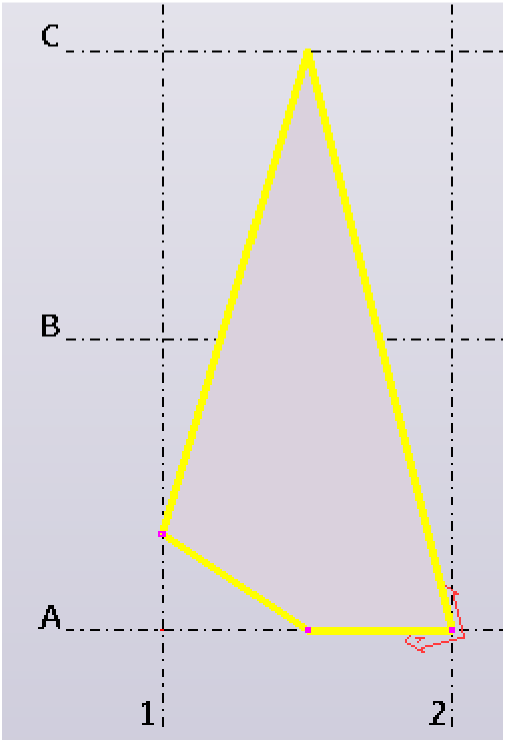
   Tekla Structures creates the four default views all at once, using the properties defined in the `component_front_view`, `component_top_view`, `component_end_view`, and `component_persp_view` property files.

Create a surface view

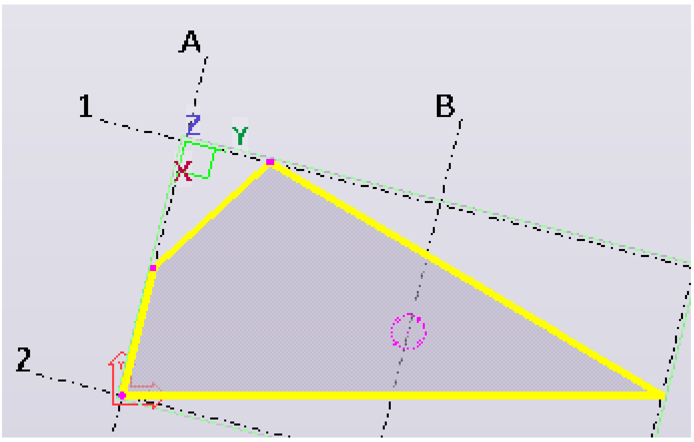
Use the **Create surface view** macro to create an automatically aligned surface view. This can be useful when modeling bolt groups, stiffener plates, and hole penetrations on complex geometry.

To be able to select the surface of a part, ensure that you are using a model view that shows part faces. On the **View** tab, click **Rendering**, and select either the **Parts grayscale** or the **Parts rendered** option.

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 **Create surface view** to start the macro.
4. Select the surface of the part.



Tekla Structures creates a new temporary view and moves the work plane typically along the longest edge of the part face. You can model in the surface view and see your modeling work being done in your original 3D view at the same time.



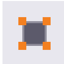
5. Press **Esc** to stop the macro.
6. To return the work plane back to the origin:
   a. Repeat steps 1–2 to open the **Applications** list.
   b. Double-click the **Work plane global** macro.

      The work plane returns back to the origin and is aligned with the global x,y, and z planes of the model.

Create a surface view along selected edge

Use the **Create surface view wEdge** macro to create a surface view and align the work plane along the edge you select. This can be useful when modeling bolt groups, stiffener plates, and hole penetrations on complex geometry.

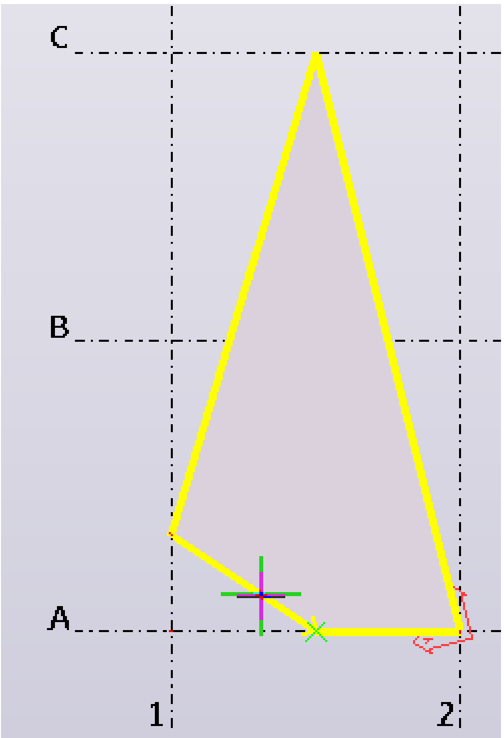
To be able to select the surface of a part, ensure that you are using a model view that shows part faces. On the **View** tab, click **Rendering**, and select either the **Parts grayscale** or the **Parts rendered** option.

1. Ensure that the  **Snap to geometry lines / points** snap switch (page 84) is active.

   This allows you to pick along an edge to define the direction.

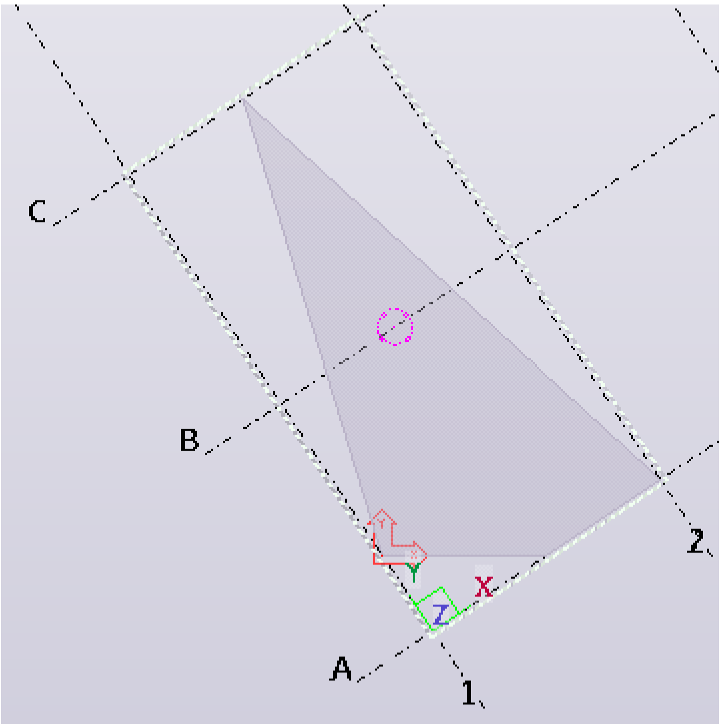
2. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
3. Click the arrow next to **Applications** to open the applications list.
4. Double-click **Create surface view wEdge** to start the macro.
5. Select the surface of the part.

   When you hover the mouse pointer over the part edges, a yellow arrow symbol is displayed to indicate the possible edges you can align the view to. The head of the arrow represents the positive direction of the x axis. The view will be rotated in this direction to form the flat horizontal edge of the view. The origin of the view and work plane will be at the start of the arrow snap line.

6. Pick the desired edge.

   Tekla Structures creates a new temporary view, and the selected edge forms the x axis of the view. You can model in the surface view and see your modeling work being done in your original 3D view at the same time.

7. Press **Esc** to stop the macro.
8. To return the work plane back to the origin:
   a. Repeat steps 2–3 to open the **Applications** list.
   b. Double-click the **Work plane global** macro.

      The work plane returns back to the origin and is aligned with the global x,y, and z planes of the model.

Open, save, modify, or delete a view

When you create views, you can have up to nine views on the screen at the same time. If you need views for later use, you can name and save them. You can modify the properties of an existing view by using the **View Properties** dialog box.

Open a view

You can have up to nine views on the screen at the same time. If you are unable to open a view, check how many views you already have open - you may need to close some of them first.

1. On the **View** tab, click  **View list** to open the **Views** dialog box.

   Tekla Structures lists all invisible named views on the left, and all visible views on the right.

2. Select a view from the **Named views** list and click the right arrow to move it to the **Visible views** list.

   You can also double-click a view to open it. If the view does not appear, check how many views you already have open.

3. To open multiple views, use the **Shift** and **Ctrl** keys when you select views from the list.

Save a view

If you need to reopen views later on, give each view a unique name. When you exit the model, Tekla Structures only saves the named views. Temporary views disappear when you close them.

Before you start, create one or more views (page 33) in the model.

1. Double-click the view to open the **View Properties** dialog box.

2. Enter a unique name in the **Name** box.

   Temporary views have a default name in parentheses. Do not use parentheses when naming a view, or the view will not be saved for later use.

   NOTE In multi-user mode, it is very important to give views unique names. If several users have different views with the same name, the view settings of one user may accidentally override the settings of another user.

   ---

3. Click **Modify**.

   Tekla Structures will automatically save all named views when you close the model.

Modify a view

You can modify a view simply by double-clicking it.

1. Double-click the view to open the **View Properties** dialog box.

2. Modify the view properties (page 46).

   For example, to rename the view, enter a new name in the **Name** box.

3. Click **Modify**.

Delete a view

You can permanently delete named views.

1. On the **View** tab, click **View list** to open the **Views** dialog box.

   Tekla Structures lists all invisible named views (page 31) on the left, and all visible views on the right.

2. Select the view you want to delete.
3. Click **Delete**.

   Tekla Structures deletes the view permanently. If the view was visible during the deletion, it will remain visible until you close it.

4. To delete multiple views, use **Shift** or **Ctrl** when you select views from the list.

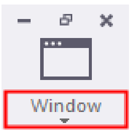
Switch between views

You can easily switch between all open views while modeling. You can also switch between the 3D and plane view to examine the current view from different perspectives, or switch between view angles to show the model in different angles.

Switch between open views

To switch between open views, do one of the following:

- Use the keyboard shortcut **Ctrl+Tab**.
- Click **Window** and select a view from the list.

  The views are listed in alphabetical order.

- Right-click a view, then select **Next window** from the pop-up menu.

  The next open view becomes active.

Switch between 3D and plane view

Use the **Switch to 3D or plane** command to examine the current view from different perspectives.

- On the **View** tab, click **Switch to 3D or plane**.

  You can also press **Ctrl+P**.

Switch between view angles

Use the **View angle** command to show a view in different view angles.

1. Select the view, and click **View angle** on the contextual toolbar.
2. Select a top, back, right, bottom, front, or left view.

3. To return to the original 3D view, click the button in the middle of the view angle options.

Update and refresh views

Use the **Update window** and **Redraw** commands to refresh a single view or all views at once.

- **Update**: Removes temporary graphics (such as measured distances) but does not redraw the view. Faster than redrawing.
- **Redraw**: Redraws the view completely and shows all previously hidden objects.

To	Do this
Update the current view	Right-click the view and select **Update window**.
Update all of the views	On the **View** tab, click **Redraw** --> **Erase temporary graphics**. 
Redraw the current view	Right-click the view and select **Redraw view**.
Redraw all of the views	On the **View** tab, click .

See also

Switch between views (page 45)

View properties

Use the **View Properties** dialog box to view and modify the model view properties.

Option	Description
Name	The name of the view.
Angle	Whether the view angle is **Plane** or **3D**.
Projection	The projection type of views. **Orthogonal**: All objects are of equal size (no perspective). When you zoom, text and point size remains the

Option	Description
	same. In addition, the zoom remains on object faces. **Perspective**: Distant objects appear smaller than close ones, as do text and points. You can zoom, rotate, or fly through the model.
Rotation	How the view is rotated around the z and x axes. Rotation is view-specific. The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals** .
Share	This option is available only if the model has been shared with Tekla Model Sharing. In Tekla Model Sharing, views are not shared by default. Views are shared if they have a name, and the **Share** option is set to **Shared**.
Color and transparency in all views	The color and transparency setting that is used in all views (according to the status of the objects in the model).
Representation...	Opens the **Object Representation** dialog box for defining color and transparency settings.
View depth	The thickness of the displayed slice of model. You can define the depth separately upwards and downwards from the view plane. Only objects positioned within the view depth are visible in the model. The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals** .
Display...	Opens the **Display** dialog box for defining which objects are displayed (page 647) in the view and how.
Visible object group	Which object group is displayed in the view.
Object group...	Opens the **Object Group - View Filter** dialog box for creating and modifying object groups.

See also

Open, save, modify, or delete a view (page 43)

Grid view properties

Use the **Creation of Views Along Grid Lines** dialog box to view and modify the properties of grid views.

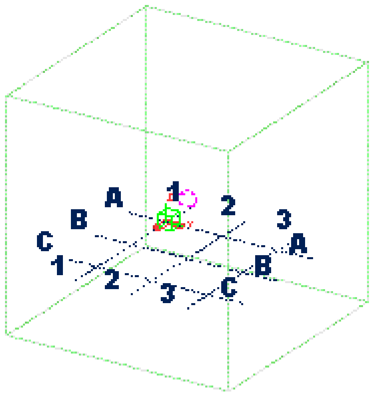
Option	Description
View plane	The plane of the view defined by two axes similarly to the default view.
Number of views	Defines which grid lines the views will be created of. **None** does not create any views. **One (First)** only creates the view closest to the grid origin. **One (Last)** only creates the view furthest from the grid origin. **All** creates all views in grid planes in the relevant direction.
View name prefix	The prefix to be used with the grid label in the view name. This name overrides the name in the view properties. View names consist of a prefix and a grid label, e.g. PLAN +3000. If the **View name prefix** box is left empty, no prefix is used. Tekla Structures adds a dash and a running number to the view name if view names are otherwise identical.
View properties	Defines which view properties (applied or saved) will be used. Each view plane has its own view properties. You can load the properties from the current view properties with the option **<applied values>** or from saved view properties. The **Show** button displays the current view properties.

See also

Create model views (page 33)

Define the work area

Tekla Structures indicates the work area of a view using dashed lines. Objects outside the work area exist, but they are not visible. You can shrink and expand the work area to suit particular situations, for example to concentrate on a particular area of the model. You can temporarily hide the work area box.



Fit work area to entire model

You can resize the work area to include all model objects, either in all views or in selected views only.

1. On the **View** tab, click **Work area**  and select one of the following:

   - **To entire model in all views**

     Fits the work area to include all model objects in all visible views.
   - **To entire model in selected views**

     Fits the work area to include all model objects in the selected views.

Fit work area to selected parts

You can resize the work area to include only selected parts, either in all views or in selected views only.

1. Select the objects you want to include.
2. On the **View** tab, click **Work area**  and select one of the following:

   - **To selected parts in all views**

     Fits the work area to include the selected model objects in all views.
   - **To selected parts in selected views**

     Fits the work area to include the selected model objects in the selected views.

Fit work area using two points

You can resize the work area based on two corner points you pick on the view plane. The depth of the work area is the same as the view depth.

1. On the **View** tab, click **Work area**  and select **Using two points**.
2. Pick the first point.
3. Pick the second point.

Hide the work area box

You can temporarily hide the work area box in a view. This can be useful, for example, when creating screenshots for presentations.

1. Hold down the **Ctrl** and **Shift** keys simultaneously.
2. On the **View** tab, click **Redraw** --> **Redraw all views**.

3. To make the box visible again, click **Redraw** --> **Redraw all views** again.

TIP Alternatively, use the advanced option XS_HIDE_WORKAREA.

If you place objects are outside the work area

If you place new objects outside the work area, or copy or move objects outside the work area, Tekla Structures displays a warning message **Objects outside the work area**. You can expand the work area to include the new objects.

If you do not wish to see thew warning message again you have the option to hide future warnings of the same type. You can also have Tekla Structures show these warnings again.

- To hide future warnings of the same type, select the **Do not show this message again** check box.
- To re-display the warnings, hold down the **Shift** key while running a command that should normally induce a warning message. For example, if you deliberately copy or move objects outside the work area while holding down the **Shift** key, the associated warning message reappears.

If you cannot see all objects

The visibility of objects in a view depends on a number of different settings. If you cannot see all the desired objects in a model view, check the following settings:

- work area
- view depth
- view filter
- view and representation settings
- color and transparency settings

Note that work area and view depth are like two virtual boxes. Objects that have their handles partially or totally inside both boxes are visible. Newly created objects are also visible outside the view depth but never outside the work area. When you redraw a view, only the objects inside the view depth are displayed.

See also

Define the work area (page 48)

View properties (page 46)

Filter objects (page 150)

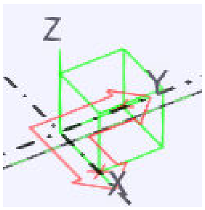
Change the color and transparency of model objects by using object representation (page 656)

Coordinate system

Tekla Structures uses two coordinate systems: the global and the local coordinate system. The local coordinate system is also known as the work plane.

Global coordinate system

The green cube symbol represents the global coordinate system and lies at the global point of origin (x=0, y=0, z=0). The global coordinate system is static and it cannot be changed.

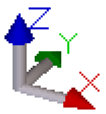


Do not place the model far away from the origin. If you create model objects far away from the origin, snapping to points (page 83) in the model views may become inaccurate. The further away from the origin you model, the less precise all computations become.

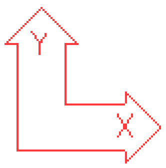
If you need to use another coordinate system for inserting reference models or exporting IFC models, you can use base points (page 57). When you use base points, you can keep the coordinates small and locate the model wherever needed.

Local coordinate system (Work plane)

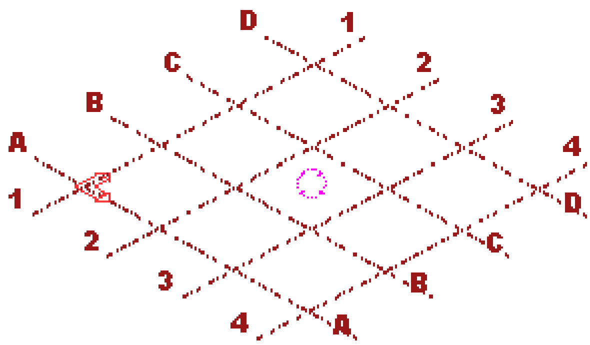
The work plane represents the local coordinate system. Most of the commands that are dependent on the coordinate system use the work plane coordinates. For example creating points, part positioning, and copying always comply with the work plane coordinate system. The coordinate symbol, which is located in the lower right corner of the model view, follows the work plane.



The work plane is model specific, so it is the same in all views. The red work plane arrow symbol shows the xy plane. The z direction follows the right-hand rule (page 53).



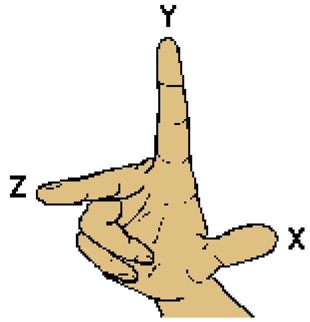
You can change the local coordinate system by shifting the work plane (page 53). The work plane also has its own red grid, which can be used for positioning parts. You can show or hide (page 53) this grid as needed.



To control which work plane or base point you currently have in use in the model, use the work plane handler toolbar (page 68).

Right-hand rule

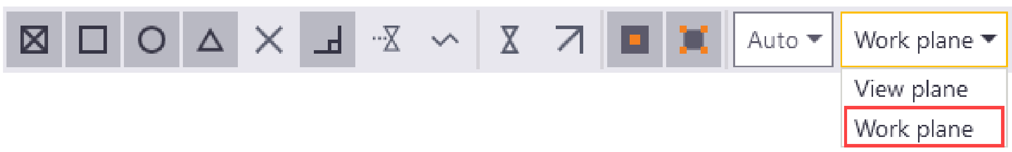
Right-hand rule indicates the direction of the coordinate axes. When you hold the thumb, index finger, and middle finger of your right hand so that they form three right angles, then the thumb symbolizes the x axis, the index finger the y axis, and the middle finger the z axis.



Show or hide the work plane grid

The work plane grid is hidden by default. Use the options on the **Snapping** toolbar to show or hide the work plane grid.

1. To show the grid, select **Work plane** from the second list.



2. To hide the grid, select **View plane** from the same list.

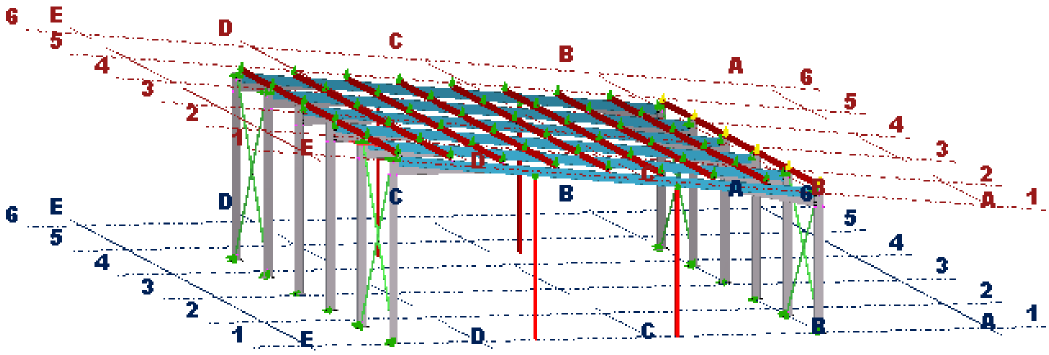
See also

Shift the work plane (page 53)

Shift the work plane

You can set the work plane to any position by picking points or by selecting a plane. This makes it easier to place parts accurately when modeling sloped parts.

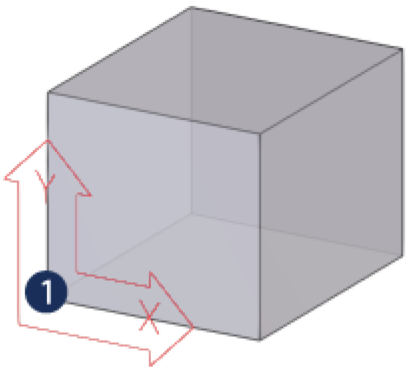
For example, you can shift the work plane to the slope of the roof to make it easier to model horizontal bracing and purlins in a sloped roof.



Set work plane to any part plane
Use the **Work plane tool** command to set the work plane to any part plane or grid intersection. Note that you can select only native Tekla Structures object planes. You cannot select reference model object planes.

1. On the **View** tab, click **Work plane** --> **Work plane tool**.

2. Move your mouse over the model to preview the work plane direction.
3. Click the left mouse button to set the work plane.

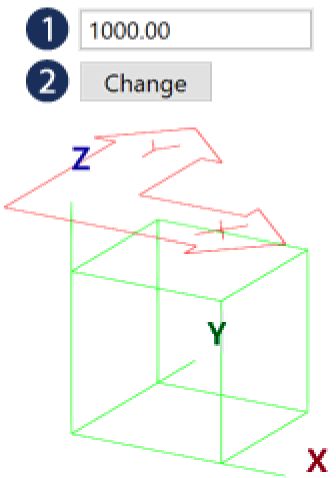


Set work plane parallel to xyz plane
You can set the work plane parallel to the xy, xz, or zy plane.

1. On the **View** tab, click **Work plane** and select **Parallel to XY(Z) plane**.

2. In the **Plane** list box, select the plane parallel to the work plane.
3. Enter the depth coordinate.

   The depth coordinate defines the distance of the work plane from the global origin along a line perpendicular to the plane parallel to the third axis.
4. Click **Change**.



Set work plane using one point
You can set the work plane using one picked point. The work plane stays parallel to the current work plane, but moves to a new position. The x and y directions are unchanged.

1. On the **View** tab, click **Work plane** and select **Using one point**.

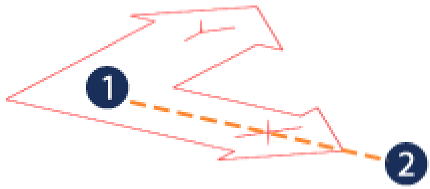
2. Pick the new position of the work plane.

Set work plane using two points
You can set the work plane using two picked points. The first point you pick is the origin, the second defines the x direction of the work plane. The y direction remains the same as the previous work plane.

1. On the **View** tab, click **Work plane** and select **Using two points**.

2. Pick the origin of the work plane.
3. Pick a point in the work plane, in the positive x direction.

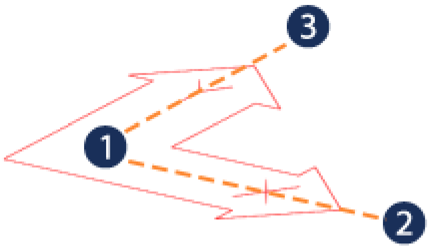


Set work plane using three points

You can set the work plane using three picked points. The first point you pick is the origin, the second defines the x direction, and the third defines the y direction of the work plane. Tekla Structures fixes the z direction according to the right-hand rule.

1. On the **View** tab, click **Work plane** and select **Using three points**.

2. Pick the origin for the work plane.
3. Pick a point in the positive x direction.
4. Pick a point in the positive y direction.

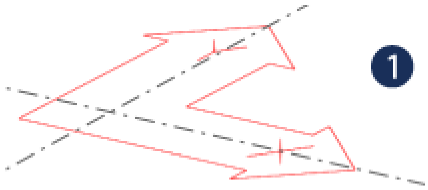


Set work plane parallel to view plane

You can set the work plane to be the same as the view plane of a selected view.

1. On the **View** tab, click **Work plane** and select **Parallel to view plane**.

2. Select the view.



Restore the default work plane

Remember to change back to the default work plane when you have finished modeling sloped structures.

1. On the **View** tab, click **Work plane** --> **Parallel to XY(Z) plane**.

2. In the **Plane** list, select **XY**.
3. In the **Depth coordinate** box, enter **0**.
4. Click **Change**.

Base points

The base points (control points) allow you to use a coordinate system based on the civil origin or other coordinate system for interoperability and collaboration. For example, you can use base points when inserting reference models, exporting IFC models, in drawings, in **Layout manager**, and in reports and templates.

Civil origin is the datum point or fundamental benchmark point of the national land survey network.

When you use base points, you can keep the coordinates small and locate the model wherever needed. You can create as many base points as you need, and select one of those to be the project base point.

Note the following:

- Reference models should not have any additional lines to the origin.
- Reference models should not include objects that are very far from each other because otherwise using the model may get difficult.
- Tekla Structures native objects, including reference models, should not be inserted very far from the Tekla Structures model origin.

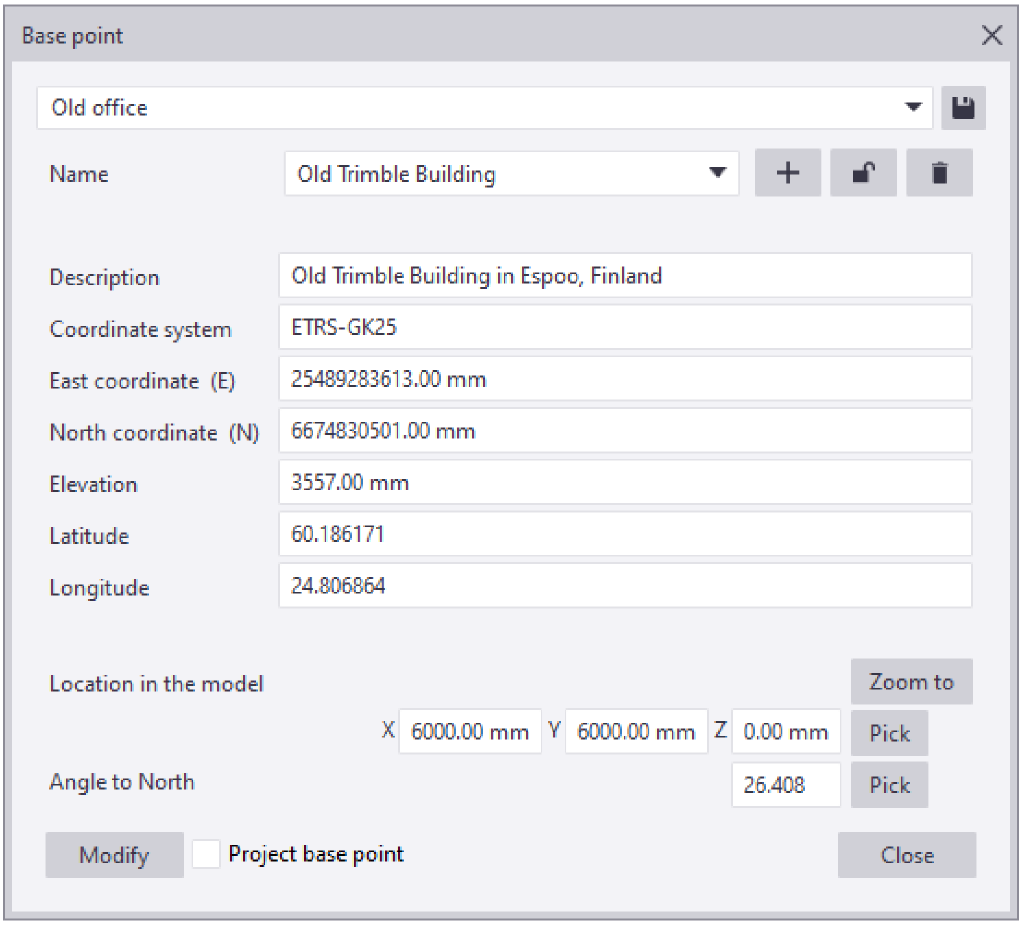
Define a base point

You can define base points in **Project properties**. In case you need to import or export a reference model, you need to know the coordinates of the reference model that you are importing, or the coordinates that you want to use in the resulting IFC export file.

TIP You can inquire the point coordinates in Tekla Structures: On the ribbon, click the down arrow next to , and then select **Point coordinates**. For more information, see Inquire object properties (page 686).

---

1. Open a Tekla Structures.
2. Click **File** --> **Project properties** --> **Base points** to open the **Base point** dialog box.
3. Fill in the needed information:



You can load existing base point settings. To do this, copy your settings file <settings name>.basePoint.json from the \attributes folder under the model where you have the base point settings you want to use to the \attributes folder under the current model folder. In the **Base point** dialog box, select the base point settings from the list.

Note that base point settings also work with project and firm folders: Copy the base point settings file to the \attributes folder under the firm and project folders.

Name, **Description**	Enter a name and a description for the base point. Giving a name is obligatory.
Coordinate System	Enter the name of the coordinate system you are using.
East coordinate (E)	Enter the **East coordinate (E)** that represents the X coordinate related to the civil origin.
North coordinate (N)	Enter the **North coordinate (N)** that represents the Y coordinate related to the civil origin.
Elevation	Enter the **Elevation**that represents the Z coordinate related to the civil origin.
Latitude,**Longitude**	Enter the **Latitude** and **Longitude** of the base point to be used in the IFC export. **Latitude** and **Longitude** is additional information, which some software can use. In the IFC file, this is written to `IFCSITE` information. If the total number of digits in **Longitude** is more than 15, the value is rounded up to the nearest if it is > 99.99999999999999999. To convert **Latitude** and **Longitude** information between decimal format and degree/minute/second (DMS) format, see Convert Latitude/Longitude to decimal.
Location in the model	Pick or enter a location for the base point in the Tekla Structures model. The distance is measured from the model origin. The base point location in the model can have a maximum value of +/- 10 km from the model origin. Modeling is meant to be done near the model origin, and the offset is to be given with **East coordinate** and **North coordinate** values.
Angle to North	Pick or enter the **Angle to North**, which is the angle between Y and North directions. The maximum number of decimals for the angle is 13.
Project base point	If you want to set a coordinate system as the project base point, select a base point from the list at the top and then select the **Project base point** check box.

4. To save the base point settings, type a unique name for the settings and click **Save**. To use the same base point settings in another model, copy your settings file `<settings name>.basePoint.json` from the `\attributes` folder under the current model folder to the `\attributes`

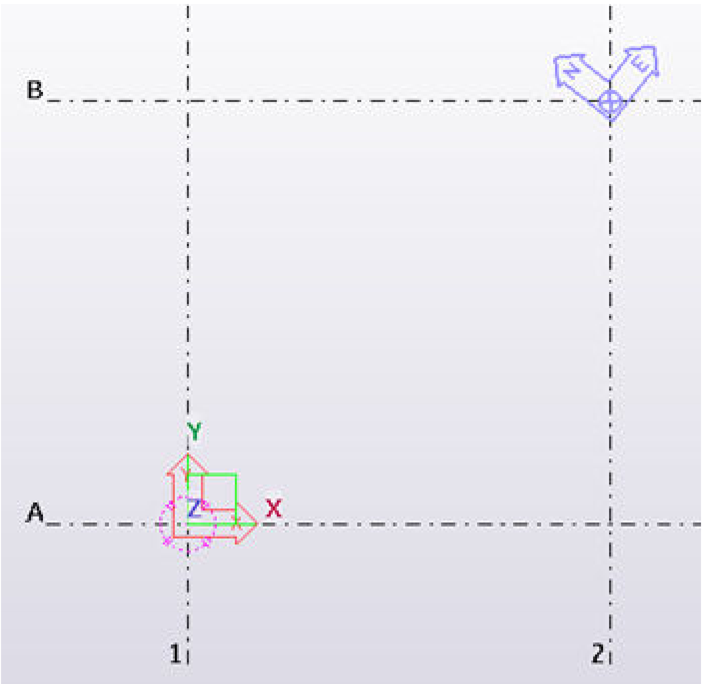
folder under the model where you wish to use the same base point settings. The settings also work with project and firm folders: Copy the base point settings file to the `\attributes` folder under the firm and project folders.

5. To avoid unwanted changes by others working in the same model, lock the base point by clicking the **Lock/Unlock** button next to the base point name. The button changes to . To unlock the base point, click the button again.

6. Click **Modify** to save the base point.

   A blue symbol is added in the model.

   If you later on make changes to the base point, the base point location in the model changes according to the location or rotation changes you make in the **Base point** dialog box when you press **Enter** or click another input field, and there is no need to click **Modify**. Note that if the base point is locked, you cannot modify it.



   Now you can insert a reference model, or export an IFC model using the defined base point, for example.

Set a coordinate system as the project base point

One base point can be set to be as the project base point. Model origin is the default project base point value if the model does not contain any base points, or if none of the existing base points has been set to be the project base point.

You can check and change the current project base point through **File** --> **Project properties** --> **Location by**.

Note that it is not recommended to change the project base point temporarily during a project.

1. Click **File** --> **Project properties**.

   You can see the current project base point in the **Location by** box.

2. To change the project base point, click **Edit**, and select a new project base point from the **Location by** list.

3. Click **Apply**.

TIP You can also set a base point as the project base point in the **Base point** dialog box by selecting a base point from the list at the top and then selecting the **Project base point** check box.

Insert a reference model using a base point

Before you can insert a reference at the base points, you need to create a base point in your model. To create the base point, you need to know the coordinates of the reference model that you are importing.

1. Open the **Reference Models** list by clicking the **Reference Models** button in the side pane .

2. In the **Reference Models** list, click the **Add model** button.

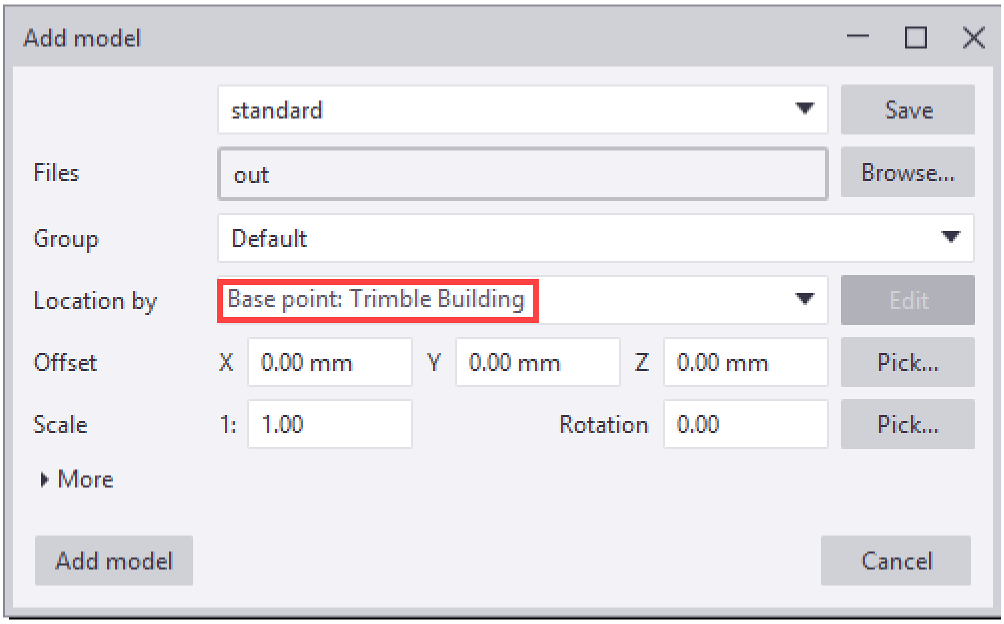
3. In the **Add model** dialog box, if you have any previously created reference model properties files, load the desired file by selecting the file from the properties file list at the top.

4. Browse for the reference model by clicking **Browse...**.

5. In **Group**, select a group for the reference model, or enter a name for a new group.

   If you do not enter a name for the group, the reference model is inserted in the **Default** group.

6. In **Location by**, select the base point you want to use.



7. Click the **Add model** button. Tekla Structures inserts the reference model relative to the selected base point by using the coordinate system values, elevation and angle in the base point definition in the model **Project properties**. For example, both IFC2x3 and IFC4 reference model inserting support base points.

Export an IFC model using a base point

Before you can export an IFC file using a base point, you need to create a base point in your model.

1. Click **File** --> **Export** --> **IFC** to open the **Export to IFC** dialog box.
2. In **Location by**, select a base point that you have created.
3. Fill in other necessary IFC export information.
4. Click **Export**. The base point option exports the IFC model relative to the base point using the coordinate system values, elevation, latitude, longitude and angle in the base point definition in the model **Project properties**.

Base points in drawings

It is possible to use base-point-defined coordinate system values in drawings. If you change the project base point Z or the elevation value, the level value will change accordingly when a drawing is opened.

- Base point data can be used in drawing and view level to set the coordinate system. 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.

- This setting affects level marks and attributes that end with `_BASEPOINT`.
- When the base point is set at the drawing level, the `_BASEPOINT` template attributes can be used in drawing templates.

You can set **Location by** in drawing view properties to use model origin, project base point or any base point defined coordinate system. **Location by** uses the project base point as the default value.

The datum level only affects the attributes `TOP_LEVEL` and `TOP_LEVEL_UNFORMATTED` when **Location by** is set to **Model origin** or to the project base point that is in the model origin.

To change the **Location by** value:

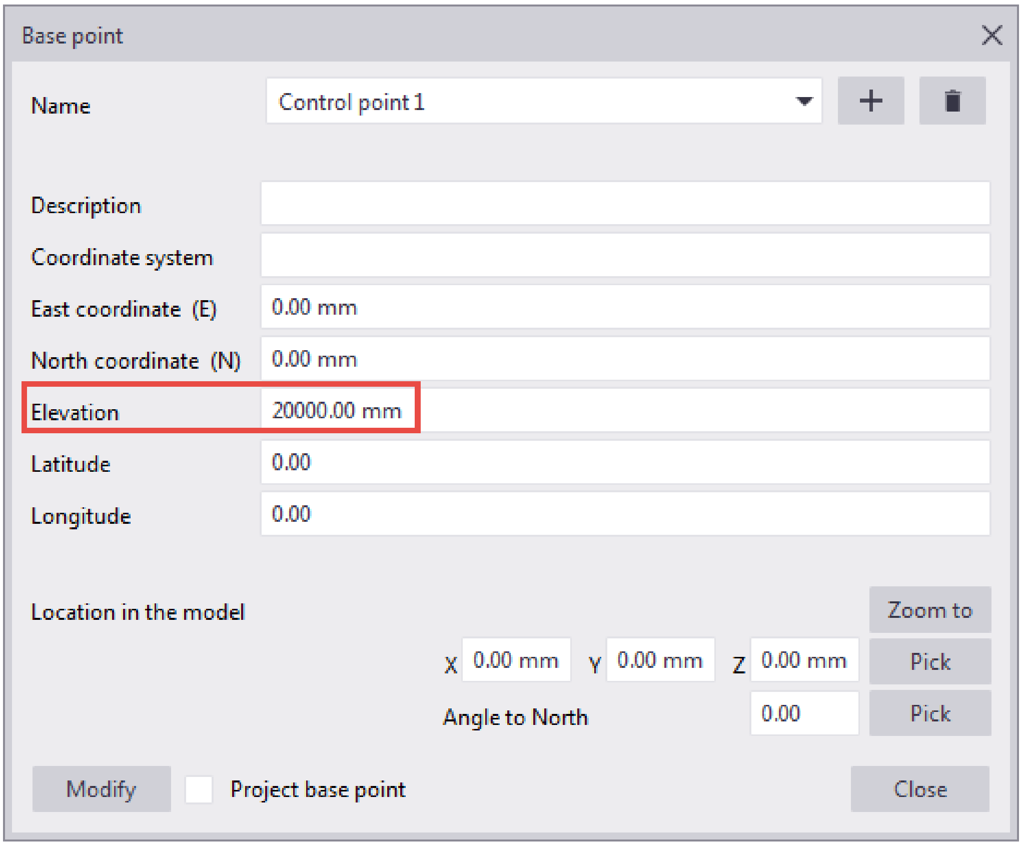
1. In an open drawing, double-click the drawing view frame to open the **View Properties** dialog box.
2. On the **Attributes 2** tab, set **Location by** to a new base point, or to the model origin.
3. Click **Modify**.

Example of base point usage in a drawing

In the following example, do the following:

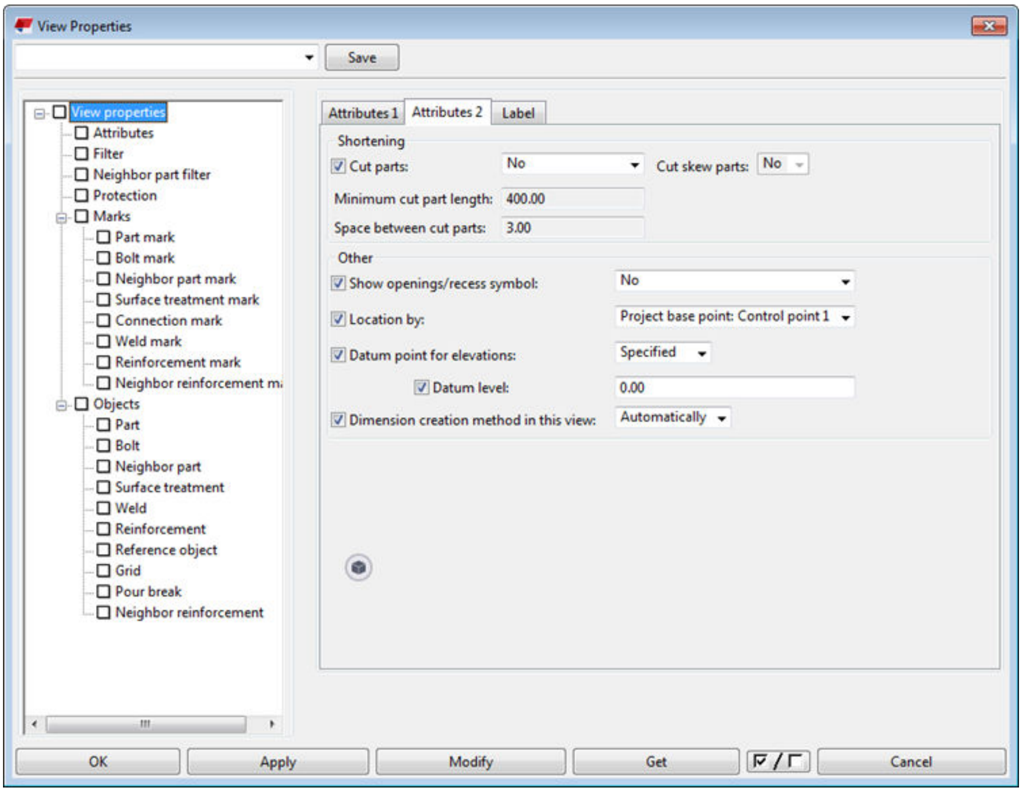
1. Create a slab with a thickness of 200 mm with the slab's top at the level 0 in the model.

2. Create a new base point "Control point 1" with elevation 20000 mm.



3. Create a GA drawing in plan view.
4. Open the GA drawing, double-click the view frame to open the **View properties** dialog box.

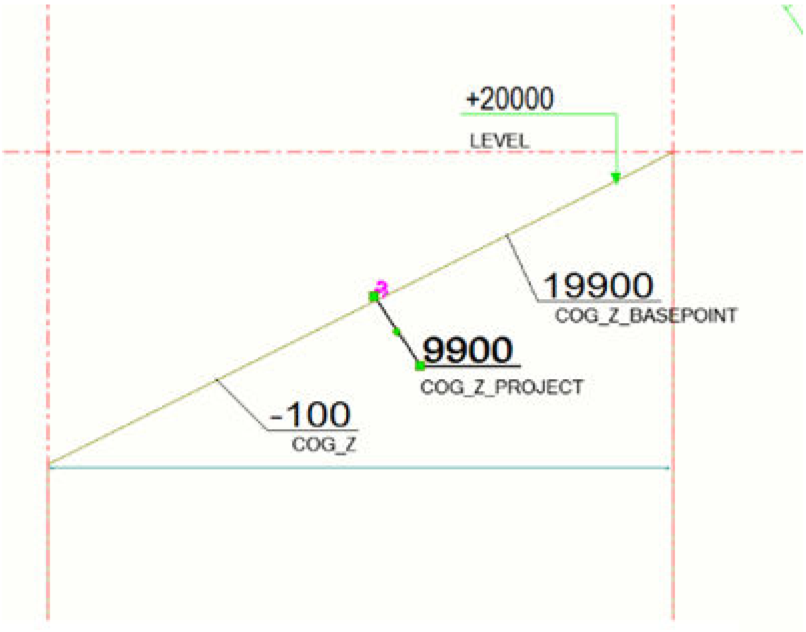
5. On the **Attributes 2** tab, you set **Location by** to the new base point (Project base point) "Base point 2" and click **Modify**.



6. Add a level mark using the following template attributes:
   - `COG_Z`
   - `COG_Z_PROJECT`
   - `COG_Z_BASEPOINT`

7. Reopen the drawing.

   Note that changing the value does not update the template attribute automatically but after reopening the drawing.



Base points in Layout manager

You can use base points in **Layout manager** when defining the location of layout points.

- You can use base points as location coordinates when exporting and importing layout points.
- When you add, modify or delete base points, you need to reopen or refresh **Layout manager** to make the changed base point data available in **Layout manager**.

Base point in reports and templates

You can inquire project base point and current base point value in reports and templates.

The following table lists the template attributes where you can use `_PROJECT` and `_BASEPOINT` at the end, for example, `ASSEMBLY_BOTTOM_LEVEL_PROJECT` or `ASSEMBLY_BOTTOM_LEVEL_BASEPOINT`. Note that `_BASEPOINT` uses the current base point in the same way as the workplane uses the current workplane. If there is no current base point defined, `_BASEPOINT` provides values relative to model origin (Global).

Content type	Attributes

`ASSEMBLY`, `CAST_UNIT` and `PART`	`ASSEMBLY_BOTTOM_LEVEL`
	`ASSEMBLY_BOTTOM_LEVEL_UNFORMATTED`
	`ASSEMBLY_TOP_LEVEL`
	`ASSEMBLY_TOP_LEVEL_UNFORMATTED`

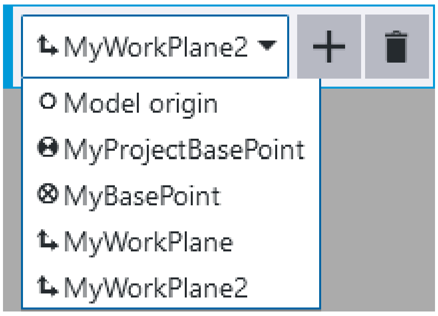
Content type	Attributes
	BOTTOM_LEVEL BOTTOM_LEVEL_UNFORMATTED BOUNDING_BOX_MIN_X BOUNDING_BOX_MIN_Y BOUNDING_BOX_MIN_Z BOUNDING_BOX_MAX_X BOUNDING_BOX_MAX_Y BOUNDING_BOX_MAX_Z BOUNDING_BOX_X BOUNDING_BOX_Y BOUNDING_BOX_Z COG_X COG_Y COG_Z START_X START_Y START_Z END_X END_Y END_Z TOP_LEVEL TOP_LEVEL_UNFORMATTED LOCATION_BREAKDOWN_STRUCTURE.LBS_FLOOR_ELEVATION ASSEMBLY.LOCATION_BREAKDOWN_STRUCTURE.LBS_FLOOR_ELEVATION
REFERENCE MODEL, REFERENCE OBJECT and REFERENCE_ ASSEMBLY	BOUNDING_BOX_MIN_X BOUNDING_BOX_MIN_Y BOUNDING_BOX_MIN_Z BOUNDING_BOX_MAX_X BOUNDING_BOX_MAX_Y BOUNDING_BOX_MAX_Z LOCATION_BREAKDOWN_STRUCTURE.LBS_FLOOR_ELEVATION

Content type	Attributes
POUR OBJECT	BOTTOM_LEVEL BOTTOM_LEVEL_UNFORMATTED TOP_LEVEL TOP_LEVEL_UNFORMATTED LOCATION_BREAKDOWN_STRUCTURE.LBS_FLOOR_ELEVATION
CONNECTION	ORIGIN_X ORIGIN_Y ORIGIN_Z
HIERARCHIC OBJECT	LOCATION_BREAKDOWN_STRUCTURE.LBS_FLOOR_ELEVATION

Select the work plane

If you have defined base points or saved work planes, you can use the **Work plane handler** toolbar to select which work plane you currently have in use in the model.

By default, the **Work plane handler** toolbar is located at the bottom of the screen.



You can select among the following work planes:

- Model origin (if project base point is set somewhere else)
- Project base point
- All base points (page 57) you have defined
- All work planes (page 53) you have set and saved

The symbol for model origin is different if the project base point is set to model origin, or if the project base point is set somewhere else.

When a model is reopened, the last used work plane or base point is used.

Add a work plane to the toolbar

1. Set a work plane in the model.

2. On the **Work plane handler** toolbar, enter a name for the work plane in the **Select work plane** box.

3. Click the **Save work plane** button to add the new work plane to the list.

   If needed, you can rename your work plane by double-clicking it and entering a new name.

4. To remove a work plane from the list, click the **Remove work plane** button.

By default, the **Work plane handler** toolbar is located at the bottom of the screen. If you are unable to find the toolbar, click **File** --> **Settings**, and in the **Toolbars** list ensure that the **Work plane handler toolbar** is selected.

See also

Coordinate system (page 51)

Change the color settings for dimensions, labels, and model background

You can define which color you want to use for dimensions, labels, and background in the model. For example, if you set the background color to black, you may need to adjust the other color settings as well to ensure that the text and dimensions will be visible.

Change the color settings in the **Advanced options** dialog box by using RGB values on a scale of 0.0 to 1.0. Separate the values with spaces. For example, the color code for yellow is `1.0 1.0 0.0`.

TIP Alternatively, if you wish to change the color settings at one go without using the advanced options, you can use the Background Color Tool, which is available in Tekla Warehouse.

To change the drawing mode color, go to **File** --> **Settings** --> **Color mode** and select one of the options.

Find RGB values for colors

To find the right RGB values for colors, use, for example, the following tools:

- Background Color Selector tool, which is available in Tekla Warehouse
- Color picker for Tekla Structures tool, which is available in Tekla User Assistance

Change the model background color

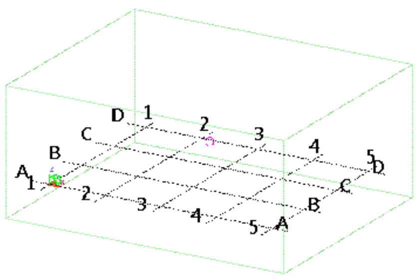
Set the background color using a combination of four different advanced options. You can control the color of each corner of the background separately.

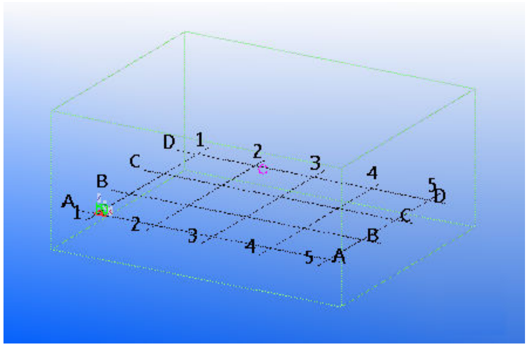
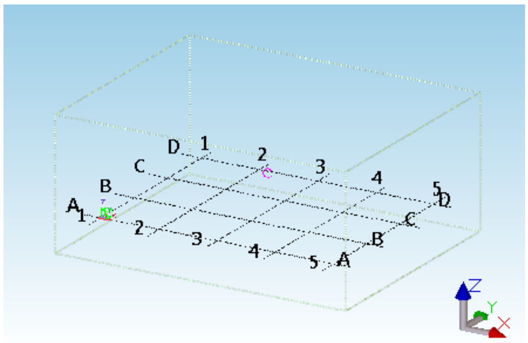
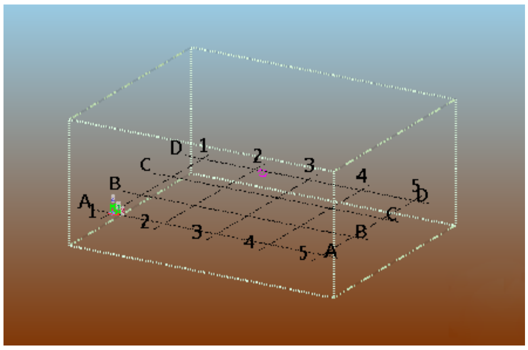
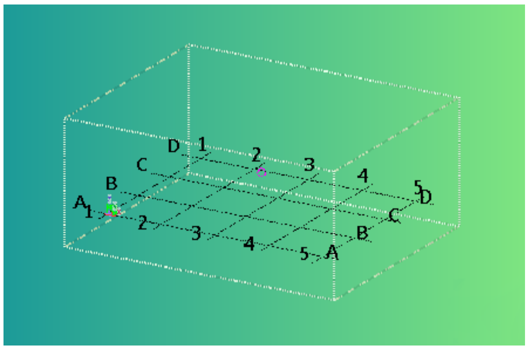
1. On the **File** menu, click **Settings** --> **Advanced options** , and go to the **Model views** category.
2. Set the background color using the following advanced options:
   - XS_BACKGROUND_COLOR1
   - XS_BACKGROUND_COLOR2
   - XS_BACKGROUND_COLOR3
   - XS_BACKGROUND_COLOR4

   To use a single-colored background, set the same color code for all four corners of the background. To use the default background color, leave the boxes empty.
3. Click **OK** to save the changes.
4. Close and reopen the view to see the changes.

Examples

Below are some examples of possible background colors that you can define. The first RGB value refers to the advanced option XS_BACKGROUND_COLOR1, the second value to the advanced option XS_BACKGROUND_COLOR2, and so on.

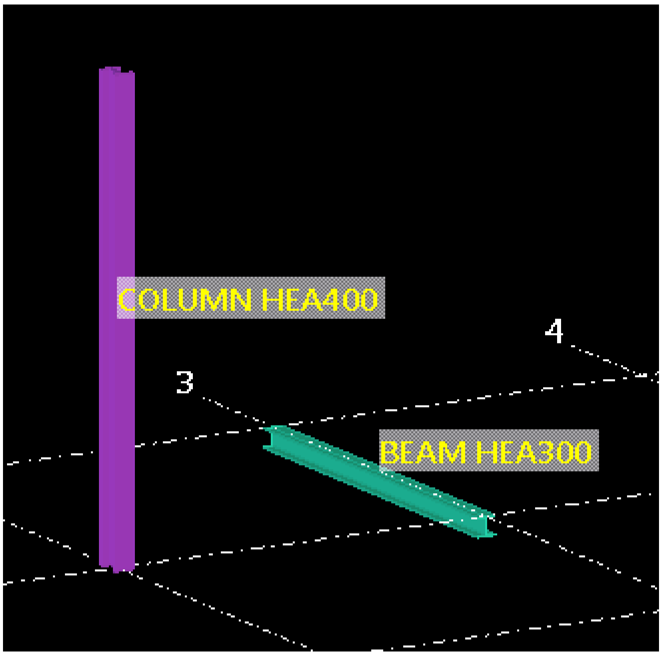
RGB values	Result
`1.0 1.0 1.0` `1.0 1.0 1.0` `1.0 1.0 1.0` `1.0 1.0 1.0`	

RGB values	Result
0.98 0.98 0.99 0.99 0.99 0.99 0.00 0.37 0.99 0.21 0.46 0.88	
0.6 0.8 0.9 0.6 0.8 0.9 1.0 1.0 1.0 1.0 1.0 1.0	
0.6 0.8 0.9 0.6 0.8 0.9 0.5 0.2 0.0 0.5 0.2 0.0	
0.1 0.6 0.6 0.5 0.9 0.5 0.1 0.6 0.6 0.5 0.9 0.5	

TIP You can change the drawing mode background to black with the advanced option XS_BLACK_DRAWING_BACKGROUND.

Change the color of dimensions, part labels, and bolts

You can define which color you want to use for dimensions, part labels, and bolts that are using the representation option **Fast** in the model.



1. On the **File** menu, click **Settings** --> **Advanced options** .
2. Search for the color setting you want to change.

Color setting	Advanced option
Dimension lines	XS_VIEW_DIM_LINE_COLOR
Dimension text	XS_VIEW_DIM_TEXT_COLOR
Part labels	XS_VIEW_PART_LABEL_COLOR
Work plane grid	XS_GRID_COLOR_FOR_WORK_PLANE
Bolts using the representation option **Fast**	XS_VIEW_FAST_BOLT_COLOR

TIP To quickly find all color related advanced options, type `color` in the **Search** box and press **Enter**. Make sure the **In all categories** check box is selected.

3. Define the color using RGB color codes.
4. Click **OK** to save the changes.

   You may be required to restart Tekla Structures.
5. Close and reopen the view to see the changes.

Change the rendering mode for model views

Tekla Structures model view rendering can use either DirectX technology or the legacy OpenGL technology.

By default, Tekla Structures uses the DirectX rendering. The DirectX rendering adds a subtle shading effect to Tekla Structures objects, making the 3D visualizations clearer and more defined.

If you want to use the legacy OpenGL rendering, switch the **Use legacy rendering** option **on** in **File** --> **Settings** --> **Switches**.

The rendering setting is model view specific, which means that you can use different rendering options in different model views. If you switch between the rendering options, you need to reopen the model view to activate the rendering option.

DirectX rendering engine

The DirectX rendering mode is better optimized for modern graphics cards, and the graphics performance is better on the recommended NVIDIA GeForce GTX graphics cards than with graphics cards that have a lower end or no graphics processing unit (GPU). For more information on the graphics cards, see Tekla Structures hardware recommendations.

If you want to measure the performance of your DirectX 3D rendering engine, you can use the TeklaMark tool from Tekla Warehouse. The tool tests how fast your computer handles graphical information typically used in Tekla Structures, for example, processor speed, load time, average draw time, and per frame details. For more information, see the TeklaMark support article that shows graphs of the collected performance data on different hardware setups.

NOTE If you use Tekla Structures via remote connections, the DirectX rendering may not work as expected: parts that you have created may not show in the model, or the model is working slowly. If you experience such problems, switch the DirectX rendering off.

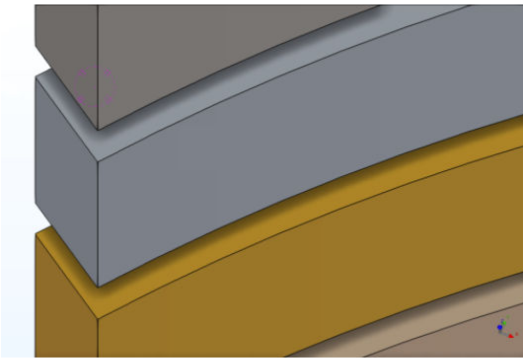
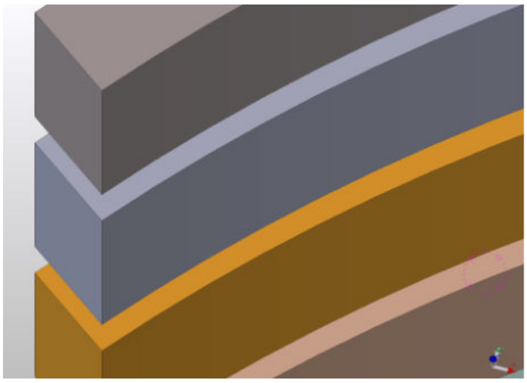
Use the following advanced options to fine-tune the DirectX rendering:

- XS_USE_ANTI_ALIASING_IN_DX
- XS_SHOW_SHADOW_FOR_ORTHO_IN_DX
- XS_SHOW_SHADOW_FOR_PERSPECTIVE_IN_DX
- XS_SHOW_STATISTICS_IN_DX

DirectX rendering examples

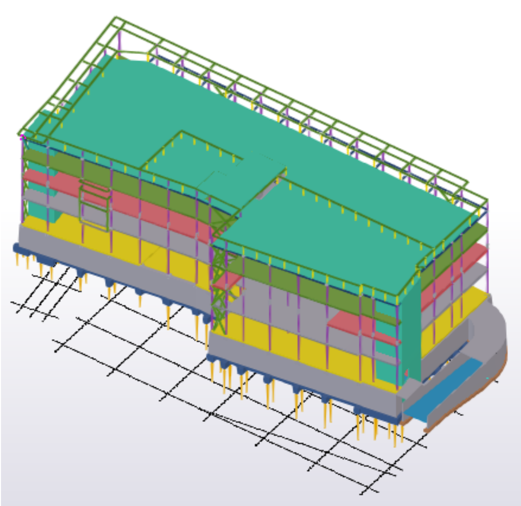
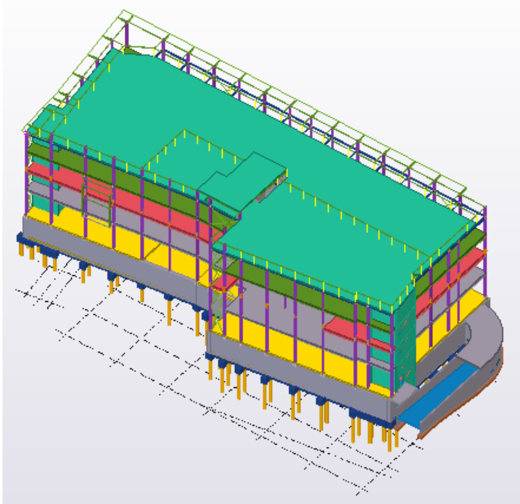
Visualization of distance

In DirectX rendered model views, the visualization of the distances uses subtle shadows and ambient occlusion. This gives a better understanding of the structure and distances.

DirectX	OpenGL
	

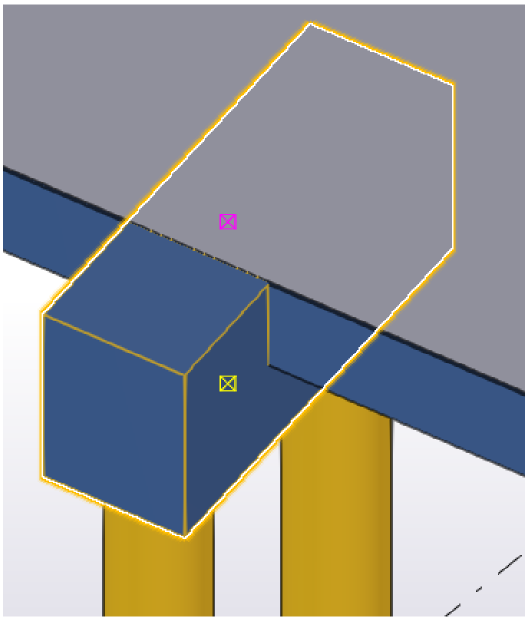
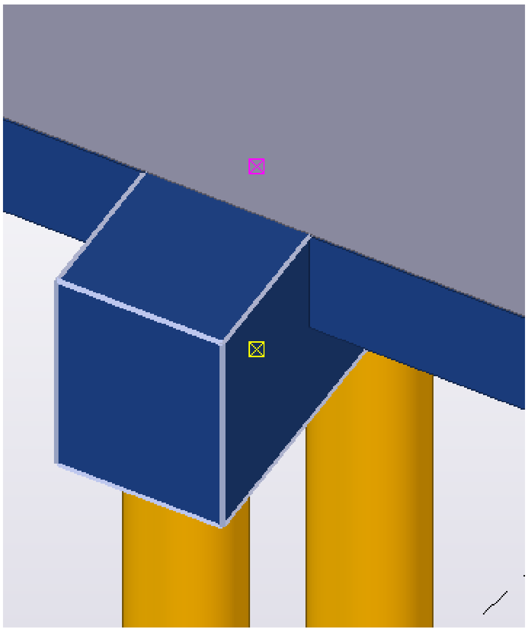
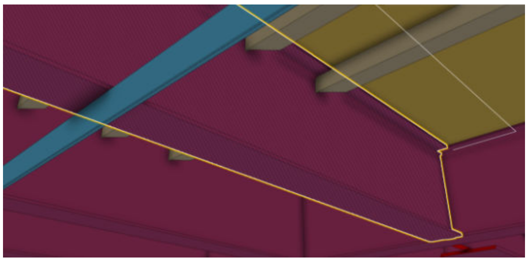
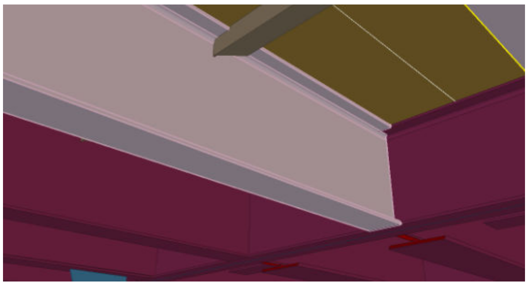
Depth accuracy

In DirectX rendered model views, the accuracy in depth buffer is improved so that when the model is zoomed, parts are not shown through other parts' faces as often as previously.

DirectX	OpenGL
	

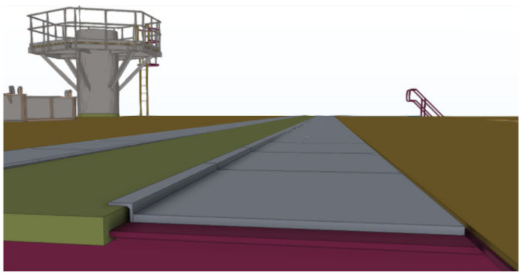
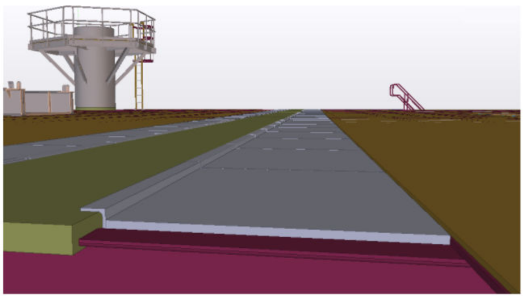
Dynamic states

In DirectX rendered model views, in dynamic states, such as in selection and in preselection highlight, the selection is clearer and the highlight is less intrusive.

DirectX	OpenGL
	
	

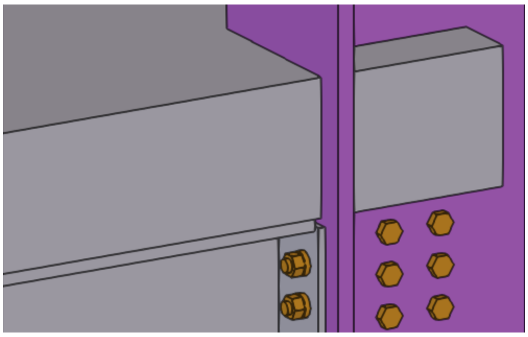
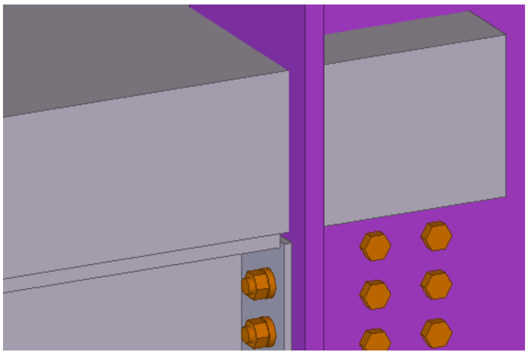
Antialiasing for higher fidelity

In DirectX rendered model views, the image quality is by default better than in OpenGL rendered model view, with less flickering.

DirectX	OpenGL
	

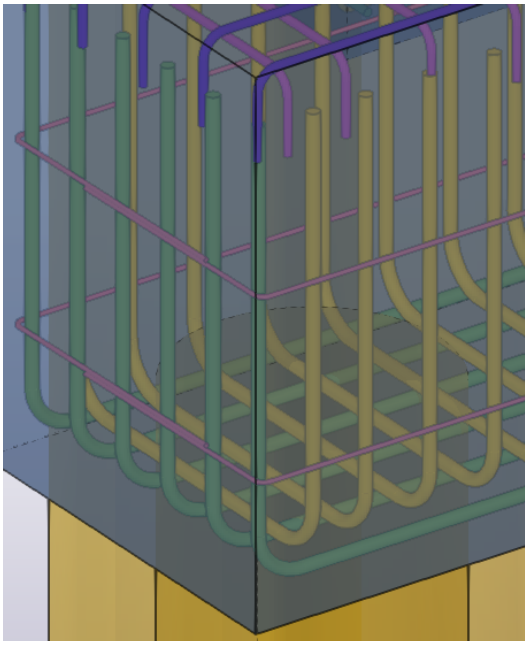
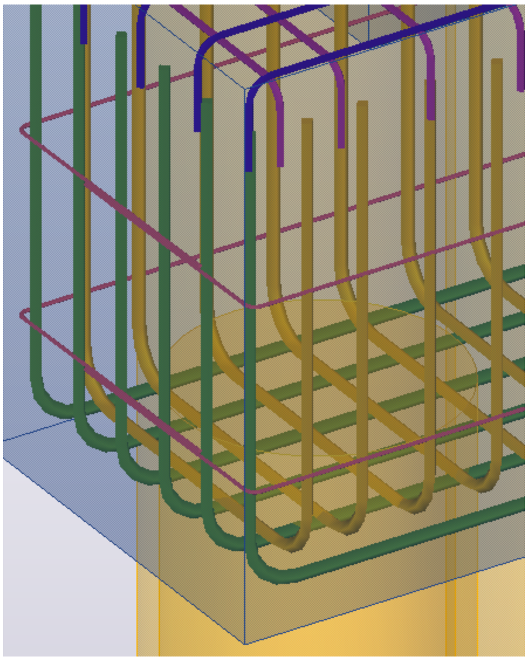
High fidelity edge line

In DirectX rendered model views, there are no flickering zigzag lines but continuous, smooth edges.

DirectX	OpenGL
	

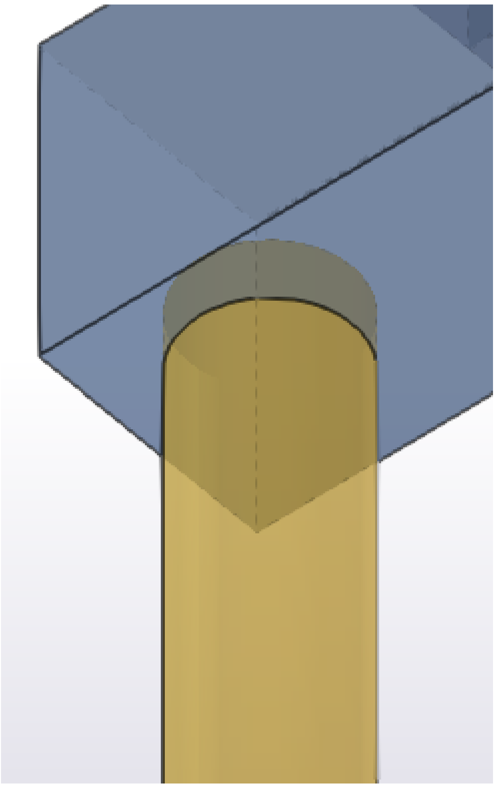
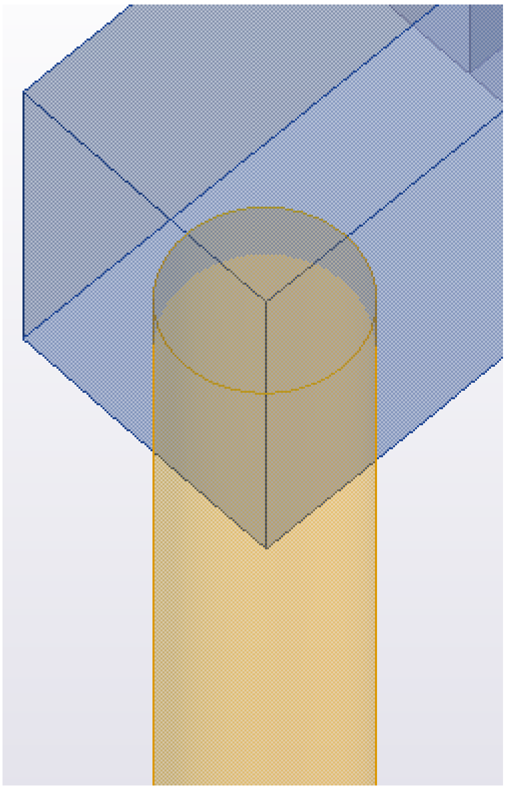
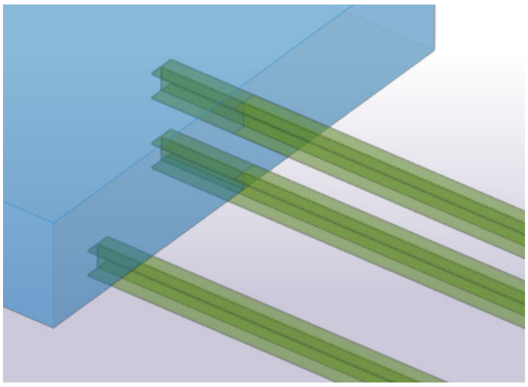
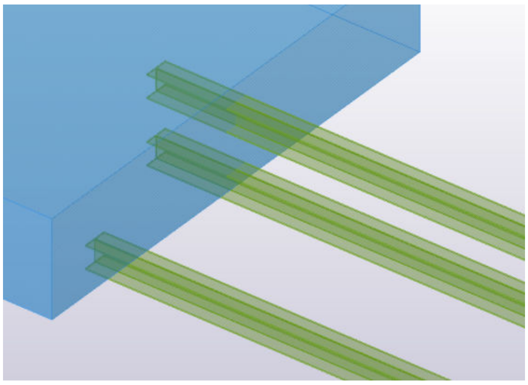
Accurate reinforcing bars

In DirectX rendered model views, reinforcing bars have edge lines. When you zoom in, the reinforcing bars are shown as round.

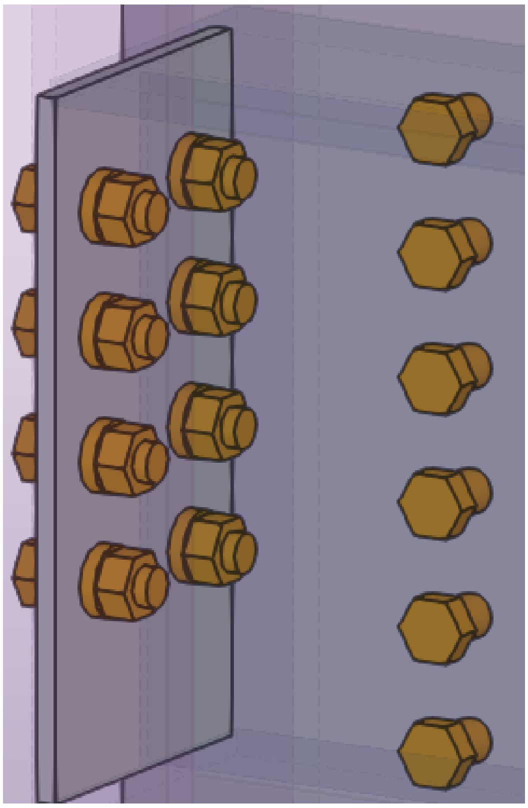
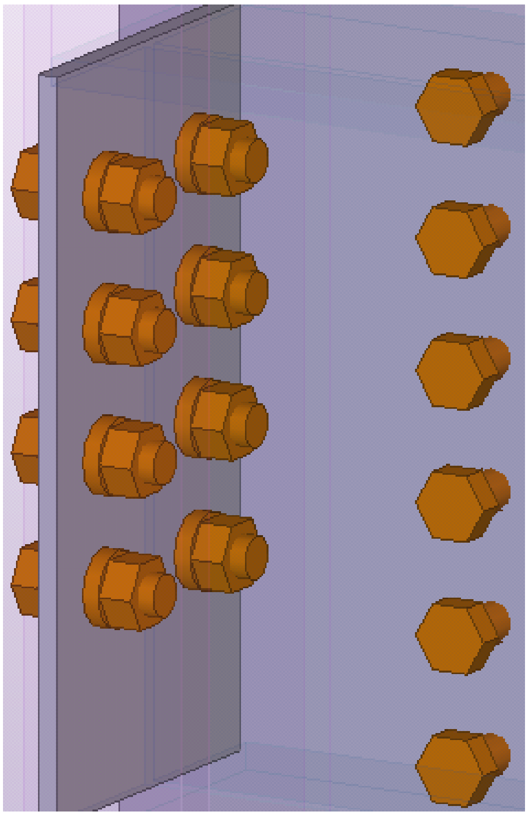
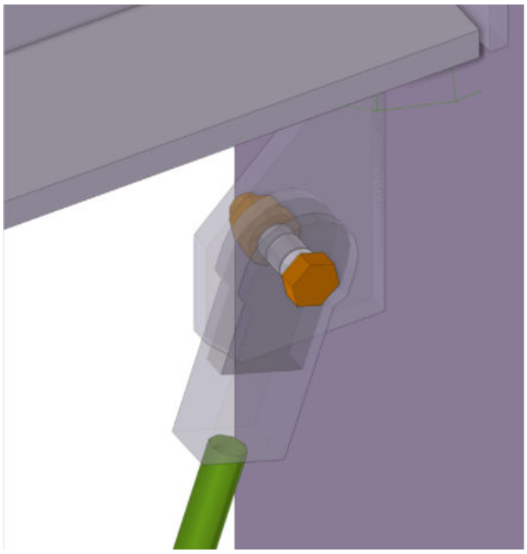
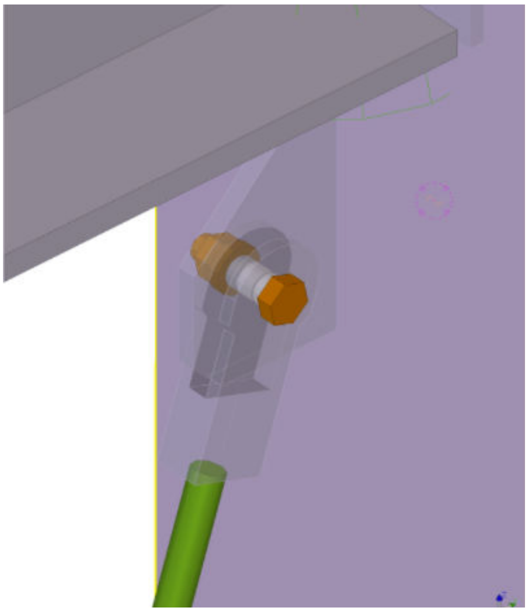
DirectX	OpenGL
	

Automatic edge lines for intersecting material in transparent view

In DirectX rendered model views, you can see where there are intersecting materials in the model.

DirectX	OpenGL
	
	

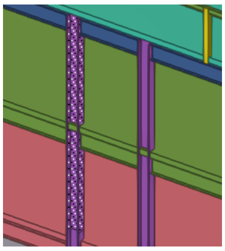
Accuracy and clarity on details

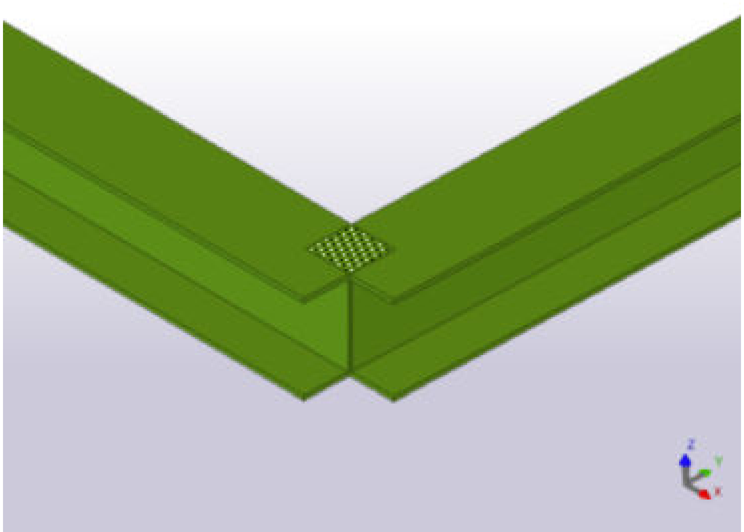
DirectX	OpenGL
	
	

Automatic hatching for overlapping surfaces on same plane

In DirectX rendered model views, duplicate objects or overlapping parts are visualized with a hatch in non-transparent views (**Ctrl+4** for parts and **Shift+4** for components).

Switch the hatching on in **File menu** --> **Settings** --> **Switches** --> **Hatching of overlapping surfaces**.

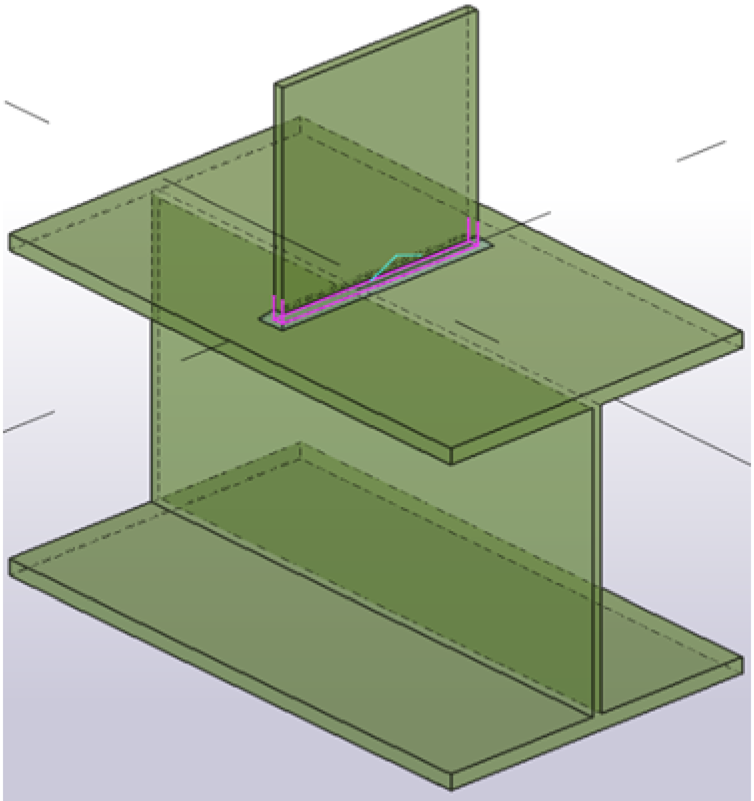


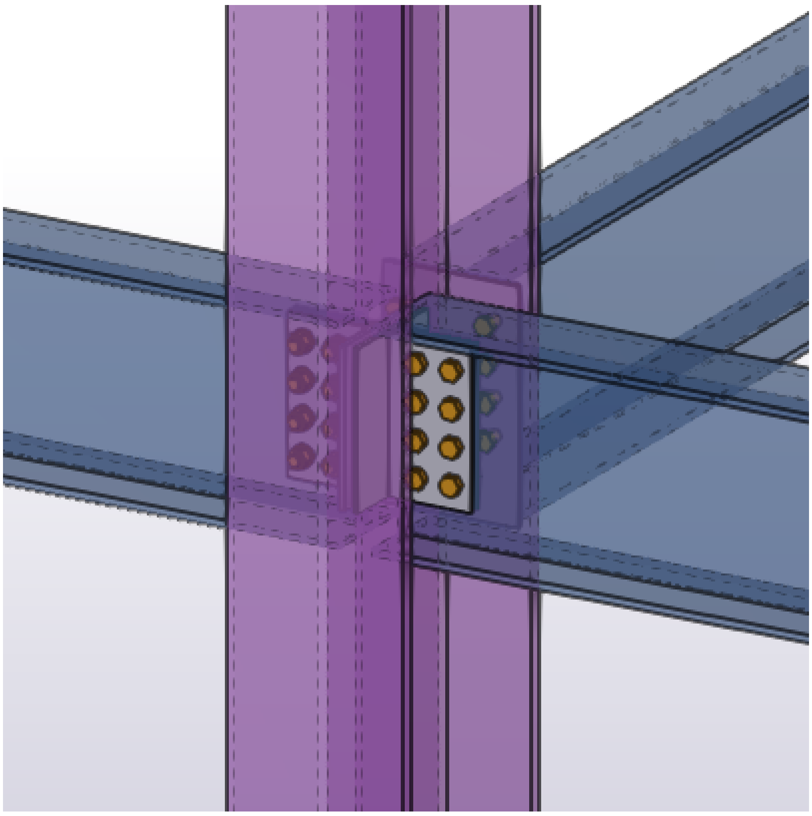


Dashed hidden lines

In DirectX rendered model views, the part edge lines that are hidden behind another part are shown as dashed lines in transparent views (**Ctrl+1, 2, 3, or 5** for parts and **Shift+1, 2, 3, or 5** for components).

Switch the dashed lines on in **File menu** --> **Settings** --> **Switches** --> **Dashed line for hidden line**.





1.2 Zoom and rotate the model

The commands on the **View** tab allow you to focus on a particular area, or pull out for a wider view. You can use a mouse, command, keyboard shortcut, or a combination of these.

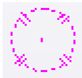
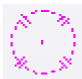
Zoom in and out

You can use a variety of tools to zoom in and out in the model. By default, the mouse pointer position determines the center point of zooming.

To	Do this
Zoom in	Scroll forward with the mouse wheel. Alternatively, press **Page Up**.
Zoom out	Scroll backward with the mouse wheel. Alternatively, press **Page Down**.
Zoom to selected objects	1. Select the objects. 2. On the **View** tab, click **Zoom** --> **Zoom selected**. Alternatively, press **Shift+Space**.
Zoom with menu commands	On the **View** tab, click **Zoom** and select one of the zoom commands.
Keep the center point of zooming in the middle of the view	On the **File** menu, click **Settings** and select **Centered zooms**.
Define the zoom ratio	Use these advanced options: XS_ZOOM_STEP_RATIO XS_ZOOM_STEP_RATIO_IN_ MOUSEWHEEL_MODE XS_ZOOM_STEP_RATIO_IN_SCROLL_ MODE

Rotate the model

You can use either the middle or left mouse button, or keyboard, to rotate the model in a view.

To	Do this
Rotate using the **middle** mouse button	1. On the **View** tab, click **Navigate** --> **Set view point**. You can also press **V**. 2. To set the view point, pick a position in the view. The following symbol appears in the model:  3. Hold down the **Ctrl** key, and click and drag the model with the middle mouse button. Tekla Structures rotates the model around the view point you defined in step 2.
Rotate using the **left** mouse button	1. On the **View** tab, click **Navigate** --> **Rotate with mouse**. You can also press **Ctrl+R**. 2. To set the view point, pick a position in the view. The following symbol appears in the model:  3. Click and drag the model with the left mouse button. Tekla Structures rotates the model around the view point you defined in step 2.
Rotate using keyboard	Use the keyboard shortcuts **Ctrl +arrow keys** and **Shift+arrow keys**. **Ctrl+arrow keys** rotates the model in 15 degree increments. **Shift+arrow keys** rotates the model in 5 degree increments.

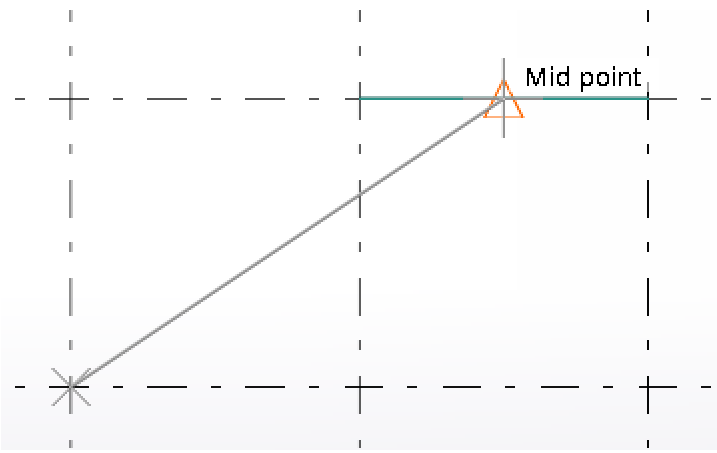
Pan the model

You can use either the middle or left mouse button to pan the model in a view.

To	Do this
Move the model using the **middle** mouse button	1. On the **File** menu, click **Settings** and check that the **Middle button pan** check box is selected. 2. Hold down the middle mouse button and drag the model.
Move the model using the **left** mouse button	1. To activate dynamic panning, go to the **View** tab and click **Navigate --> Pan**. You can also press **P**. The mouse pointer changes to a hand: 2. Hold down the left mouse button and drag the model. 3. To stop panning, press **Esc**.

1.3 Snap to positions

Most commands ask you to pick points to place objects in the model or drawing. This is called *snapping*. When you are creating a new object, Tekla Structures displays snap symbols and snap tooltips for the available snap points and a light gray line between the snap point and the last point picked.



Use the snap switches (page 84) on the Snapping toolbar to control which positions you can snap to.

For example, you can snap to

- different points, such as end points and midpoints
- centers
- intersections
- lines and edges
- dimensions and mark lines, drawing layout items and drawing frames

If you want to use exact distances or coordinates when snapping to positions, use numeric snapping (page 87).

With the combination of different snapping tools, you can, for example, snap to the closest orthogonal point (page 99) on the plane, both in the model and in the drawings. Additionally, you can follow a line and pick a point at a specified distance along the line, or create a temporary reference point to use as a local origin, both in the model and in the drawings.

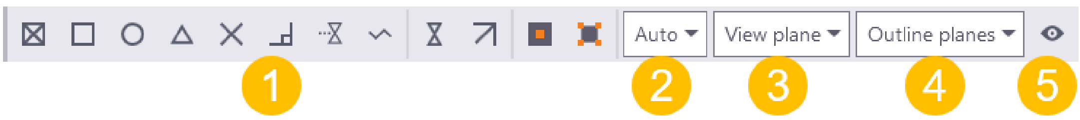
Tekla Structures displays snap dimensions in the model, which means you can easily create objects of a desired length. Use the advanced option XS_DISPLAY_DIMENSIONS_WHEN_CREATING_OBJECTS to switch the snap dimensions on or off.

TIP Use the snapping keyboard shortcuts to speed up your work.

Snap to points by using snap switches

Use the snap switches to control which positions you can pick in the model or drawing. By using snap switches, you can position objects precisely without having to know the coordinates. You can use snap switches whenever Tekla Structures prompts you to pick a point.

Click the snap switches on the **Snapping** toolbar to switch them on or off. If there is more than one point available to snap to, press the **Tab** key to cycle forward through the snap points, and **Shift+Tab** to cycle backwards through them. Click the left mouse button to select the appropriate point.



(1) Use the snap switches (page 84) to control which positions you can pick when placing objects. Snap switches define exact locations on objects, for example, end points, midpoints, and intersections.

(2) Use the first list to define the snap depth.

(3) Use the second list to switch between the view plane and work plane (page 53).

(4) Use the third list to set the plane type (page 960). The plane type defines what planes you can select in the model.

(5) You can hide selected switches from the toolbar.

Alternatively, you can control the snap switches with **Quick Launch**. Start typing the name of the snap switch, for example, `snap`, and click the name of the snap switch in the search results list to activate the switch.

Snap zone

Each object has a snap zone. It 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.

You can set the snap zone using the advanced option XS_PIXEL_TOLERANCE.

Snap priority

If you pick and hit several positions simultaneously, Tekla Structures automatically snaps to the point with the highest snap priority. To control which positions you can pick, use snap switches. Snap switches define the snap priority of positions.

Snap depth

The first list on the **Snapping** toolbar defines the depth of each position you pick. You have the following options:

- **Plane**: You can snap to positions either on the view plane (page 31) or the work plane (page 51), depending on what you have selected in the second list on the **Snapping** toolbar.
- **Auto**: In perspective views, this option works like the **3D** option. In non-perspective views, it works like the **Plane** option.
- **3D**: You can snap to positions in the entire 3D space.

Visual cues in snapping

Tekla Structures indicates where you can snap in the model and which snap switches can be used to snap to certain positions.

When you start a command that requires picking points and you move the mouse pointer over objects, the mouse pointer locks to a snap point and Tekla Structures displays in the model

- a snap symbol

  The snap symbol changes according to the possible snap point. Tekla Structures automatically highlights the points where you can snap to.

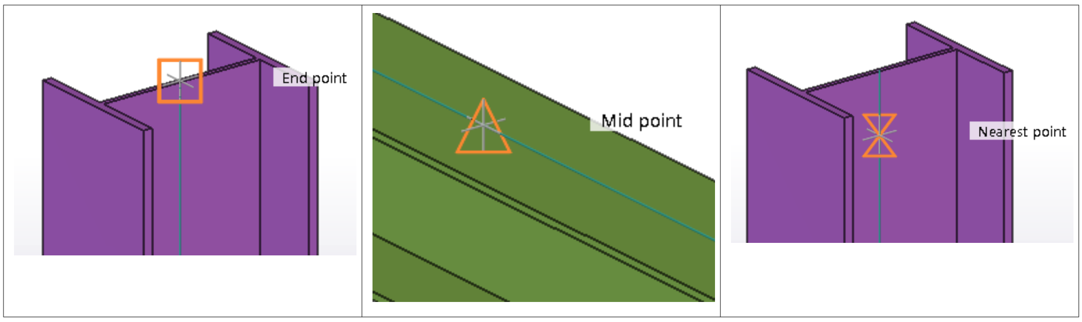
  Use the snap settings (page 104) to show or hide the snap symbols.

- a snap tooltip which shows the name of the possible snap point

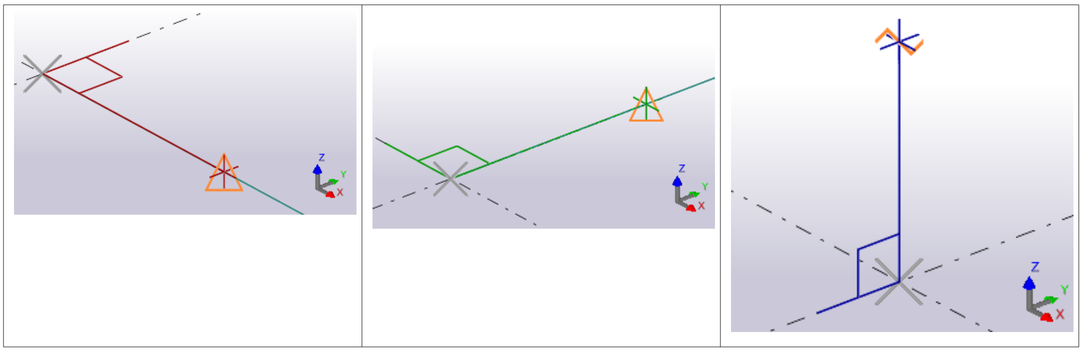
  To show or hide the snap tooltips, click **File** --> **Settings** and select the **Snap tooltips** check box.

- a turquoise reference line or geometry line of the object. The turquoise line shows the line or the edge to which the snap point belongs to.

For example:



Additionally, Tekla Structures indicates in which direction the picked points are located. When the Ortho (page 99) tool is active, Tekla Structures displays a rubber band line between the last point picked and the snap point. The color of the cursor and the rubber band line follow the color of the work plane axis: red for x-axis, green for y-axis and blue for z-axis. For any other direction the color of the rubber band line and the cursor is black.



As a general rule of thumb, you can only snap to something that is visible.

For example, if you use the rendering options **Parts rendered** or **Components rendered** (**Ctrl/Shift +4**), the object surfaces are displayed and the objects are not transparent. This means that you cannot snap to the object's geometry lines or reference lines which cannot be seen through the object.

Override the current snap switch settings

You can temporarily override the current snap switch settings, and activate only the selected snap switch. The selected snap switch overrides the other snap settings for the next point you pick.

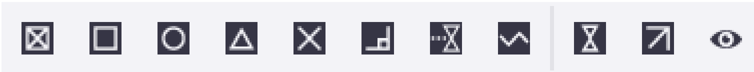
1. Run a command that asks you to pick a point.

   For example, start creating a beam.

2. To override the current snap switches, do one of the following:

   - Right-click to show a list of snap options, and then select one of the options.
   - Click **File** --> **Settings**, and in the list of toolbars select **Snap override toolbar**.

     A new toolbar appears. Click a button to activate the selected snap switch.

   - Use Quick Launch to activate a snap override switch. Type `override` in the **Quick Launch** box and select the needed override switch in the search results list.

Snap to points by using exact distance or coordinates - numeric snapping

You can enter exact distances and coordinates when snapping to a position. This is called *numeric snapping*.

Enter a distance or coordinates

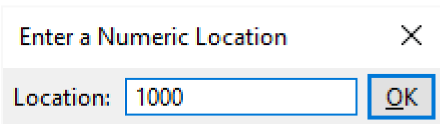
Use the **Enter a Numeric Location** dialog box to specify the distance or coordinates to a position you want to snap to.

1. Run a command that requires you to pick points.

   For example, start creating a beam.

2. Pick the first point.
3. Move the mouse pointer to indicate the direction of snapping.
4. Enter a distance or coordinates by using the keyboard.

   For example, type `1000` as the distance from the last point picked. When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog box automatically.

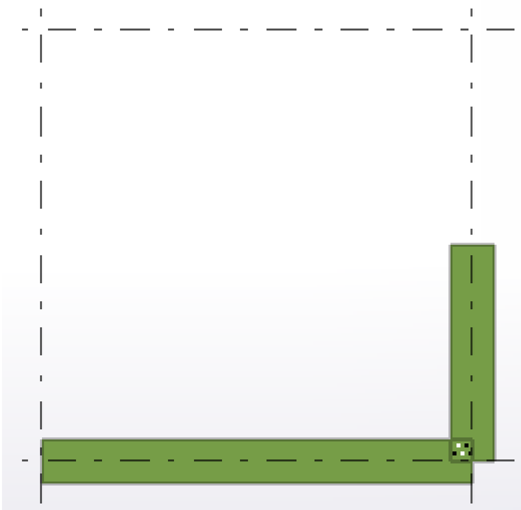
   

5. After entering the distance or coordinates, click **OK** or press **Enter** to snap to the position.

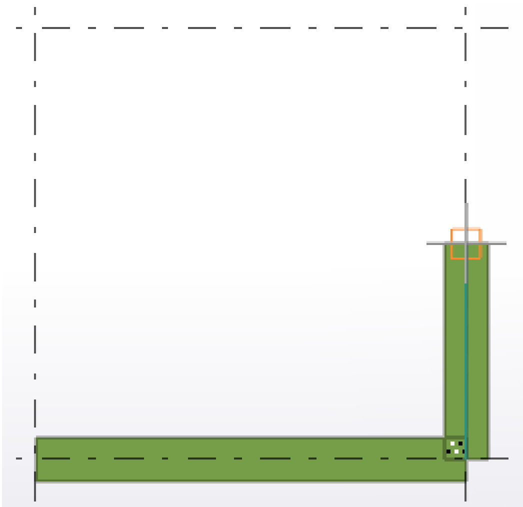
Snapping example: Track along a line towards a snap point

Tracking means that you follow a line and pick a point at a specified distance along the line. You usually use tracking in combination with numeric coordinates and other snapping tools, such as snap switches and orthogonal snapping. This example shows how to pick a point at a specified distance along a line. Use the **Enter a Numeric Location** dialog box to specify the distance from the last point picked.

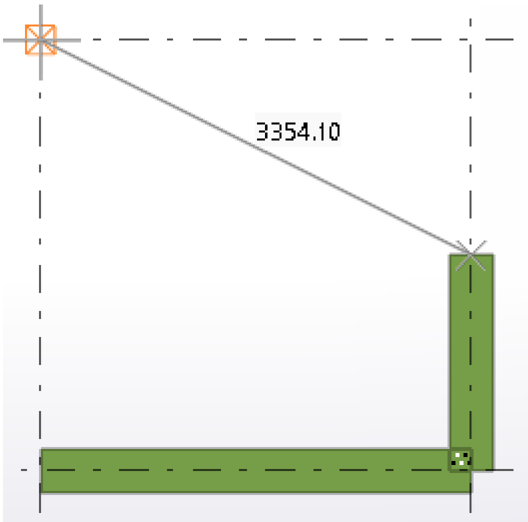
1. Create two beams and place them as shown below:



2. Activate the beam command, to create one more beam.
3. Pick the first point.

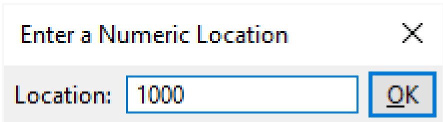


4. Move the mouse pointer over the grid line intersection so that it locks onto the snap point, but **do not** click the mouse button.



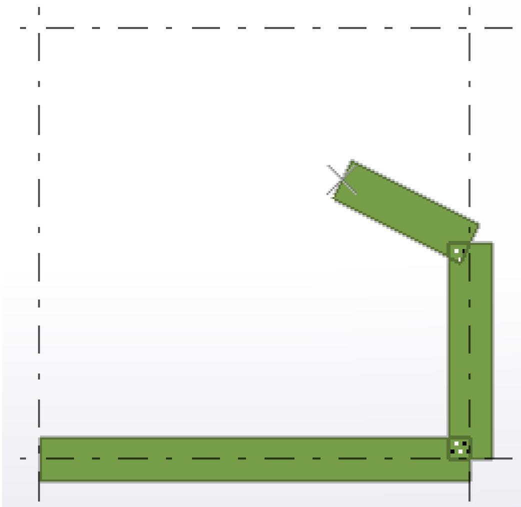
5. Type `1000`.

   When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog box.

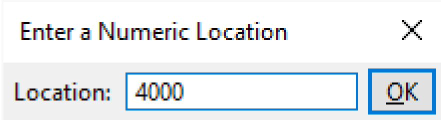
6. Click **OK** to confirm the distance.

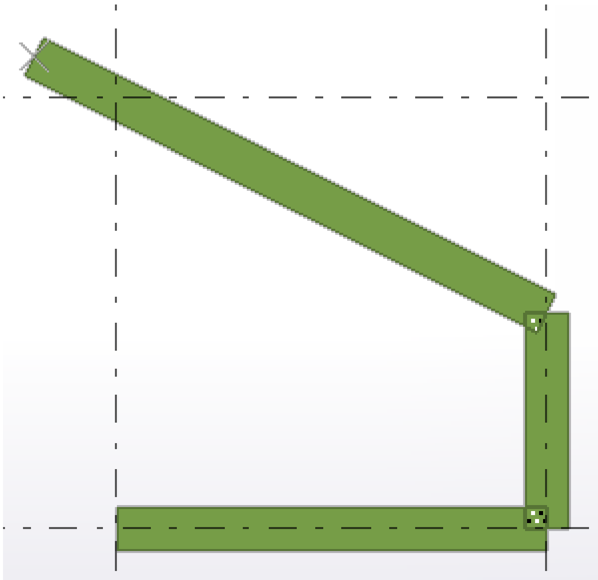
   Tekla Structures creates a beam, which is 1000 units long and positioned between the points you defined:

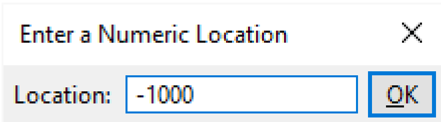
You can also:

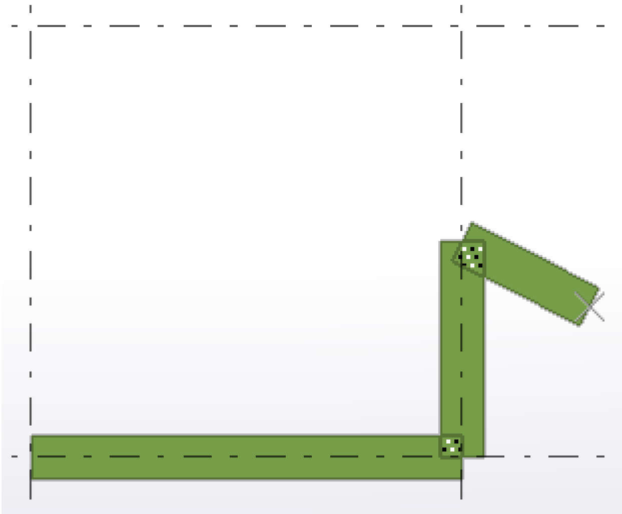
- Track beyond the snap point, for example 4000 units from the first point:





- Track in the opposite direction by entering a negative value, for example `-1000`:





For an example of how to use numeric snapping in drawings, see Place a sketch object at a specified distance.

Change the snapping mode

Tekla Structures has three snapping modes: relative, absolute, and global. Use the advanced option `XS_KEYIN_DEFAULT_MODE` to indicate the default snapping mode.

1. On the **File** menu, click **Settings** --> **Advanced options** and go to the **Modeling properties** category.
2. Set the advanced option XS_KEYIN_DEFAULT_MODE to `RELATIVE`, `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.
3. Click **OK** to save the changes.
4. If you want to temporarily override the default snapping mode, enter a special character in front of the coordinates when entering a numeric location.

    By default, the special characters are:

    - @ for relative coordinates
    - $ for absolute coordinates
    - ! for global coordinates

    Alternatively, you can start numeric snapping and bring up the special characters by typing R, A, or G. R is for relative coordinates, A is for absolute coordinates, and G is for global coordinates.

    NOTE If you want to change the special character for any of the three snapping modes, use the advanced options XS_KEYIN_RELATIVE_PREFIX, XS_KEYIN_ABSOLUTE_PREFIX, and XS_KEYIN_GLOBAL_PREFIX.

Options for coordinates

The table below explains the types of information you can enter in the **Enter a Numeric Location** dialog box.

Note that Tekla Structures has three *snapping modes*: relative, absolute, and global. You can temporarily override the default snapping mode by using a special character in front of the coordinates in the **Enter a Numeric Location** dialog box.

You can enter	Description	Special character
One coordinate	A distance to an indicated direction.	
Two coordinates	If you omit the last coordinate (z) or angle, Tekla Structures assumes that the value is 0. In drawings, Tekla Structures ignores the third coordinate.	
Three coordinates		
Cartesian coordinates	The x, y, and z coordinates of a position separated by commas. For example, `100,-50,-200`.	, (comma)
Polar coordinates	A distance, an angle on the xy plane, and an angle from the xy plane separated by angle brackets. For example, `1000<90<45`. Angles increase in the counterclockwise direction.	<
Relative coordinates	The coordinates relative to the last position picked. For example, `@1000,500` or `@500<30`.	@
Absolute coordinates	The coordinates based on the origin of the work plane. For example, `$0,0,1000`.	$
Global coordinates	The coordinates relative to the global origin and the global x and y directions. For example, `6000,12000,0`. This is useful, for example, when you have set the work plane to a part plane and want to snap to a position defined in the global coordinate system without changing the work plane to global.	!
Coordinate axis prefixes	When using direct modification, with relative and absolute coordinates, you can also use axis prefixes to	x y z

You can enter	Description	Special character
	allow snapping in the prefixed directions only. For example, `@z500` or `$y6000,z-500`. Axis prefixes cannot be used with global coordinates. If any of the entered coordinate values has an axis prefix, the other values need to have prefixes, too. The axis prefixes are not case-sensitive, and the prefixed values can be entered in any order.	

Snap to lines, edges, and extension lines

You can snap to lines when you model objects that should be lined up with an existing object or with a grid line. You can also snap to the extension lines of part reference lines, or to the extension lines of nearby objects' reference lines.

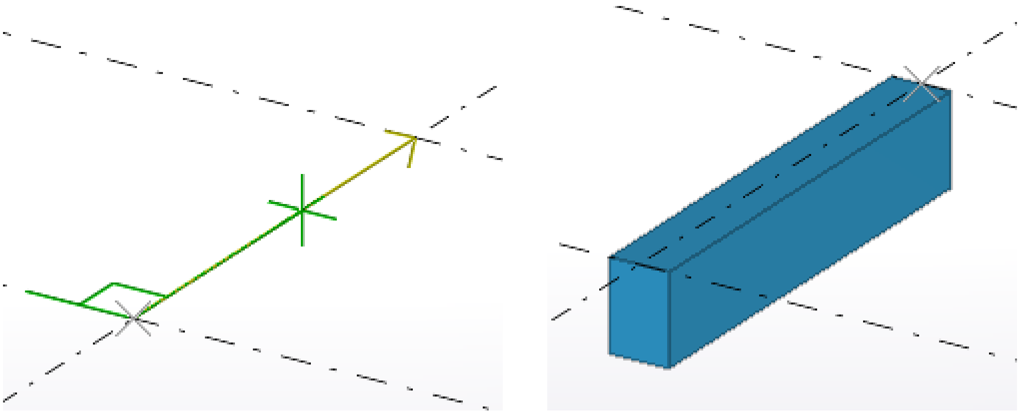
Snap to a line or an edge

Use the **Snap to line** snap switch when you need to snap to another line in the model. You can snap to grid lines, reference lines, and the edges of existing objects.

Use the **Snap to line** snap switch when you need to create, for example, several beams one after another on a grid line. With the **Snap to line** snap switch you do not need separately pick the start point and the end point of the beam.

1. Ensure that the  **Snap to line** snap switch is active.
2. Run a command that requires you to pick two or more points.

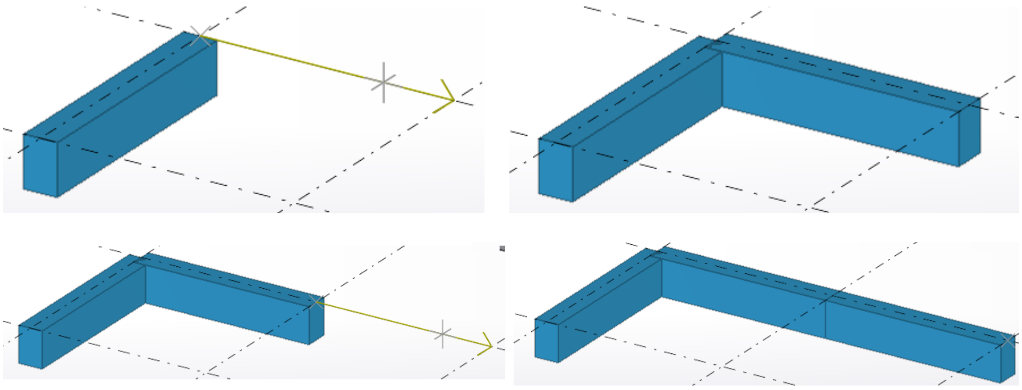
   For example, start creating a beam. When you move the mouse pointer over a grid line or a nearby object, Tekla Structures automatically picks both ends of the line. The yellow arrow symbol indicates the direction of the points.



3. To switch direction, move the mouse pointer closer to the opposite end of the line.

4. Click the left mouse button to confirm the snap position.

   Tekla Structures creates the object. For example:



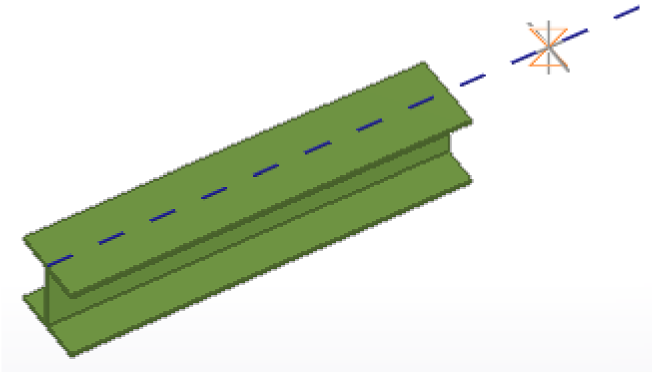
NOTE If you use the **Snap to line** snap switch with a command that requires only one point to be picked, for example when creating a column, only the start point of the line is used to position the part.

Snap to extension lines

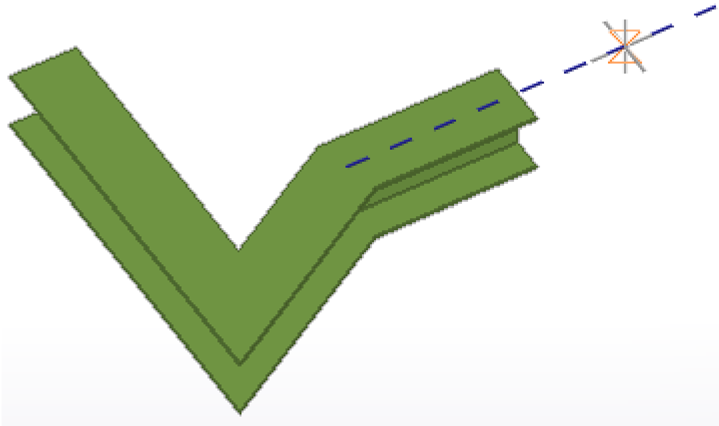
Use the **Snap to extension lines** snap switch when you need to snap to the extensions of the part reference lines, which are the lines between part handles, or to the extensions of nearby objects' reference lines. The extension line is shown as a blue, dashed line.

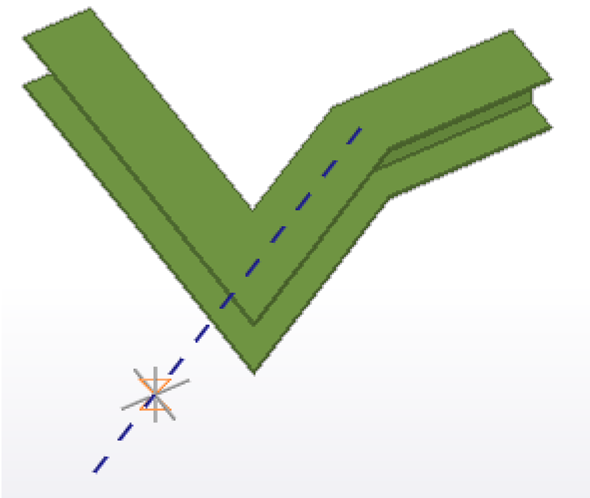
The **Snap to extension lines** snap switch works with beams, polybeams, plates, and slabs.

- With beams, the extension line is the line that passes through both of the part handles.

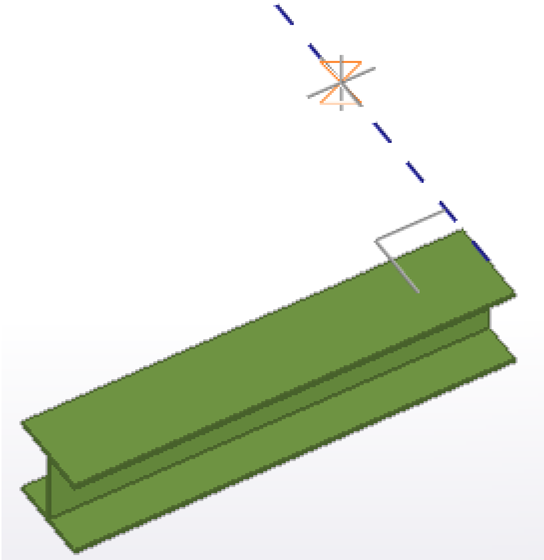


- With polybeams, plates, and slabs, the extension lines are the lines that pass through to the consecutive part handles.





- With beams and polybeams, you can snap to the line that passes through the handle at the end of the beam, and which is perpendicular to the direction of the beam.

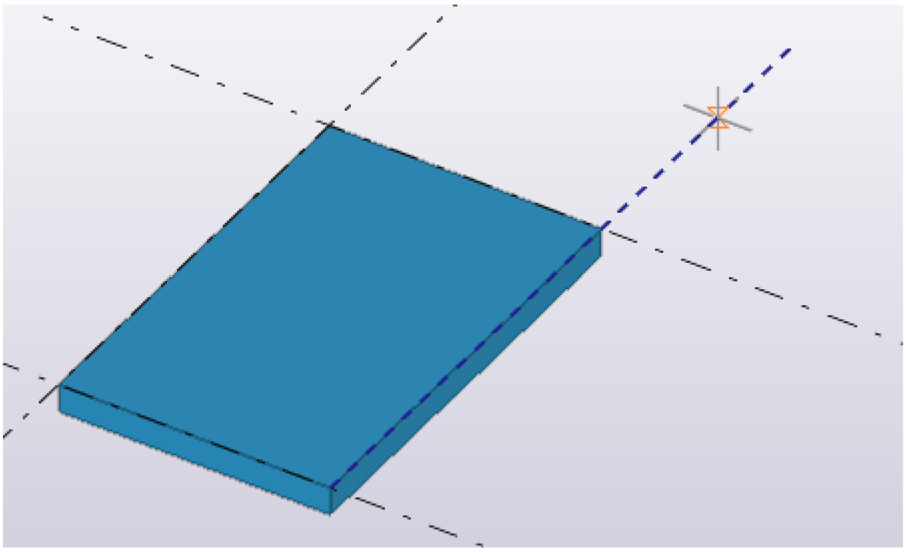


- When you snap to the extension lines of nearby objects, the extension line snaps to the direction of the nearby object and the extension line indicates the direction that is followed in snapping. Snapping to the extension lines of nearby objects can be useful, for example, when you want to align objects with one another.

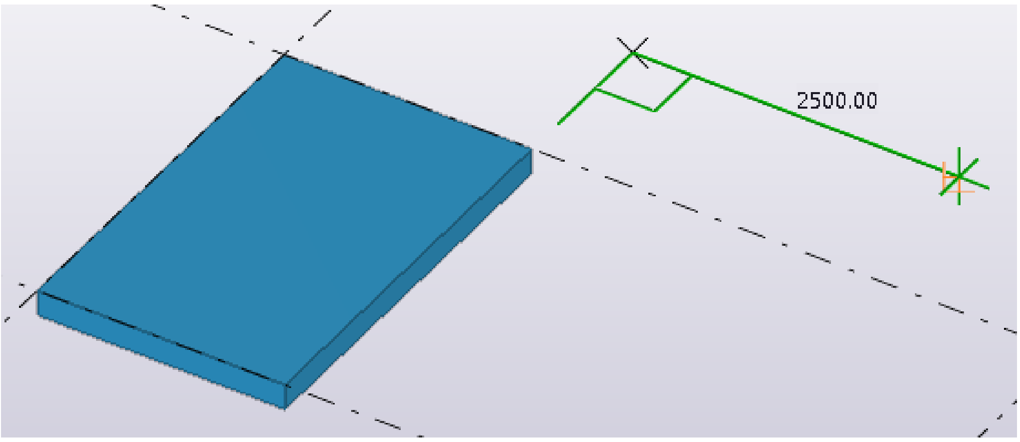
NOTE The **Snap to reference lines and points** and **Snap to geometry lines and points** snap switches do not affect the **Snap to extension lines** snap switch.

1. Ensure that the correct snap switches are active:
   - Switch on **Snap to extension lines**.
   - Switch on either **Snap to intersection points** or **Snap to nearest points (points on line)** if you are snapping to the intersection of an extension line and a grid line.
   - Switch off **Snap to end points** if you are working in 3D.
2. Run a command that requires you to pick points.

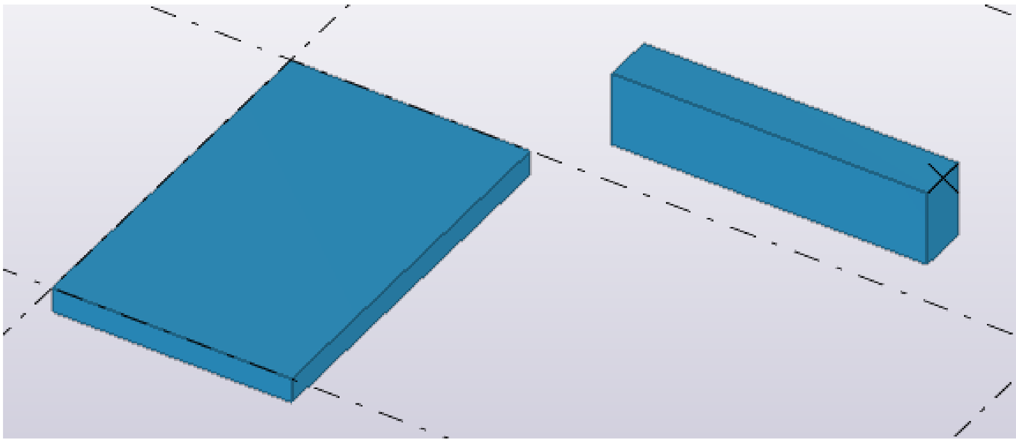
   For example, start creating a beam, a plate, or slab.
3. Move the mouse pointer over an existing object to see the extension lines.

   When a line is found, you can move the pointer further away while keeping the snap.
4. Pick the rest of the points.



Tekla Structures creates the object:



Lock X, Y, or Z coordinate on a line

You can lock the x, y, and z coordinates on a line. This is useful when you need to determine a point to pick and the needed point does not exist on the line. When a coordinate is locked, you can only snap to points in that direction.

1. Run a command that requires you to pick positions.

   For example, start creating a beam.

2. Lock a coordinate:

   - To lock the x coordinate, press **X**.
   - To lock the y coordinate, press **Y**.
   - To lock the z coordinate, press **Z**.

   Now you can only snap to points in the chosen direction.

   Tekla Structures indicates the locked coordinate with the letters **X**, **Y**, or **Z** in the status bar at the bottom of the Tekla Structures main window.

3. To unlock the coordinate, press the same letter (**X**, **Y**, or **Z**) again.

Align objects using a snap grid

A snap grid makes it easier to align objects in a model, because it allows you to snap to positions only at set intervals (page 104). Use a snap grid when you pick points using the **Snap to any position** snap switch.

1. On the **File** menu, click **Settings** --> **Snap settings** .
2. Define the grid spacing intervals in the **Spacing** boxes.

   For example, if the spacing of the x coordinate is 500, you can snap to positions at intervals of 500 units in the x direction.
3. If needed, define offsets for the snap grid origin in the **Origin** boxes.
4. To activate the snap grid, select the **Active (when free snap is on)** check box.
5. Click **OK**.

   Now when you pick points using the **Snap to any position** snap switch, you can only snap to positions at set intervals. The snap grid itself is invisible in the model.

Snap in orthogonal directions

You can snap to orthogonal points in models and in drawings using the **Ortho** tool. If you create objects that require you to pick multiple points, you can snap in orthogonal directions relative to the two previously picked points.

NOTE The **Ortho** tool has the lowest priority among snap points.

Even if you have activated the **Ortho** tool, but Tekla Structures detects any other possible snap point than an orthogonal point, Tekla Structures uses the found snap point instead of the orthogonal snap point. If there are no other possible snap points found, then Tekla Structures uses the orthogonal snap point.

Activate the Ortho tool

Before you can snap in orthogonal directions, ensure that the **Ortho** tool is active. The letter **O** in the status bar at the bottom of the Tekla Structures main window indicates that **Ortho** is active.

If **Ortho** is not active

- press **O** to activate it
- alternatively, click **File** --> **Settings** and select the **Ortho** check box.

Snap to orthogonal points

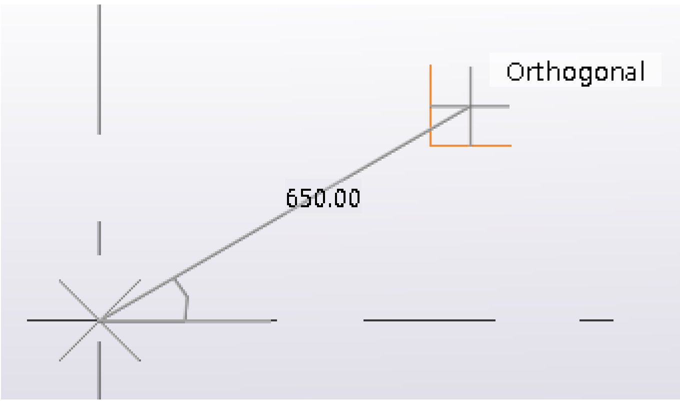
Use the **Ortho** tool to snap to the closest orthogonal point on the plane (0, 45, 90, 135, 180 degrees, and so on). The mouse pointer automatically snaps to positions at even distances in the given direction. This can be useful, for example, if you need to place marks in a consistent manner in exact locations in a drawing.

1. Ensure that the **Ortho** tool is active.
   - Press **O** to activate **Ortho** if the tool is not active.
   - Alternatively, on the **File** menu, click **Settings** and select the **Ortho** check box.
2. Run a command that requires you to pick points.

   For example, start creating a beam. Tekla Structures displays an angle symbol to indicate the direction of snapping.

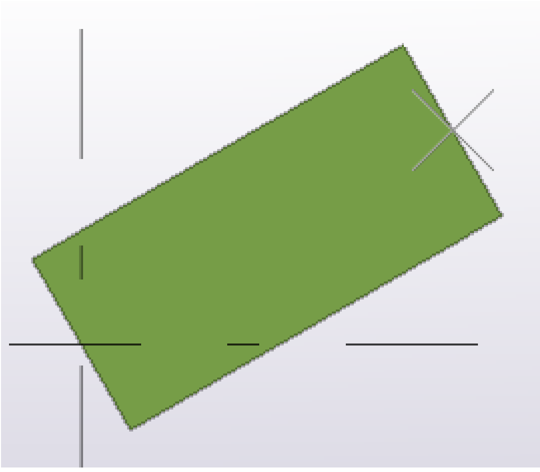
   The snapping precision depends on the current zoom level.

   The angle interval (page 104) depends on the settings in the **Model snap settings** dialog box.

3. Click the left mouse button to confirm the snap position.

Tekla Structures creates the object. For example:



Snap in orthogonal direction relative to previously picked points

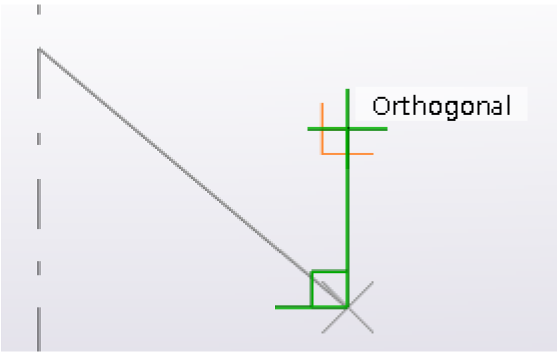
When you create objects that require you to pick more than two points, for example when you create a polybeam or a contour plate, you can snap in orthogonal directions relative to the two previously picked points. This can be useful, for example, if you need to create a rectangular slab that is on the view plane but not along the x and y axes.

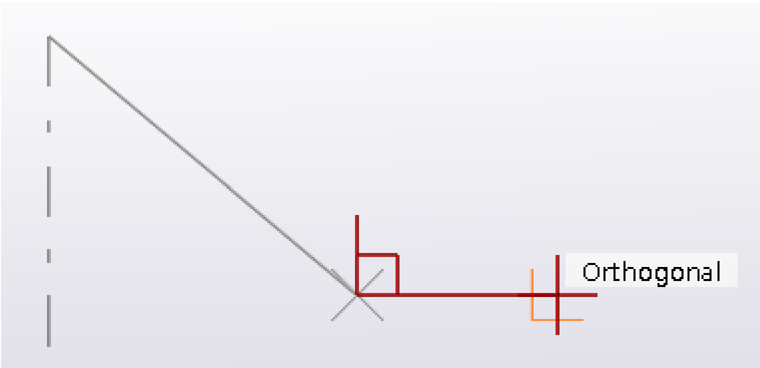
1. Ensure that the **Ortho** tool is active.
   - On the **File** menu, click **Settings** and select the **Ortho** check box.
   - Alternatively, you can press **O**.
2. Run a command that requires you to pick multiple points.

   For example, start creating a polybeam or a rectangular slab.
3. Pick the first two points.

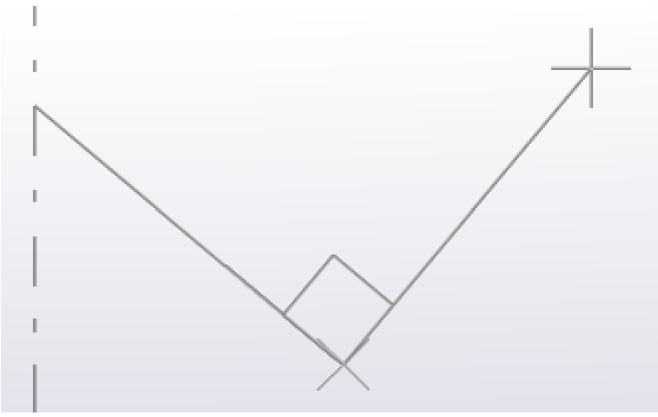
   Tekla Structures displays an angle symbol to indicate the direction of snapping.
4. Move the mouse pointer in the model to see the angle symbol.

When the snap is orthogonal to a work plane axis, the color of the angle symbol follows the color of the work plane axis: red for x-axis, green for y-axis and blue for z-axis.





When the snap is orthogonal to the previous points, the color of the angle symbol is black.



5. Pick the rest of the points.

Tekla Structures creates the object. For example:

Set a temporary reference point

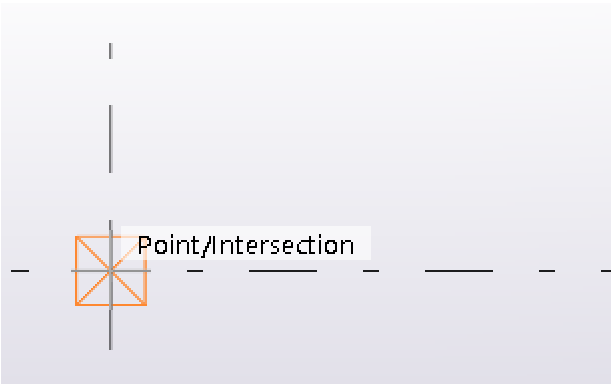
You can set a temporary reference point to be used as a local origin when snapping in models and drawings. Typically the **Ortho** tool and the **Snap to perpendicular points** snap switch use the reference point information.

The reference point information is automatically set to the last picked point, and shown as a gray cross. When you interrupt a command, the reference point information, meaning the last picked point, is cleared. If you need to use the reference point, set the temporary reference point manually.

1. Run a command that requires you to pick points.

   For example, start creating a beam.

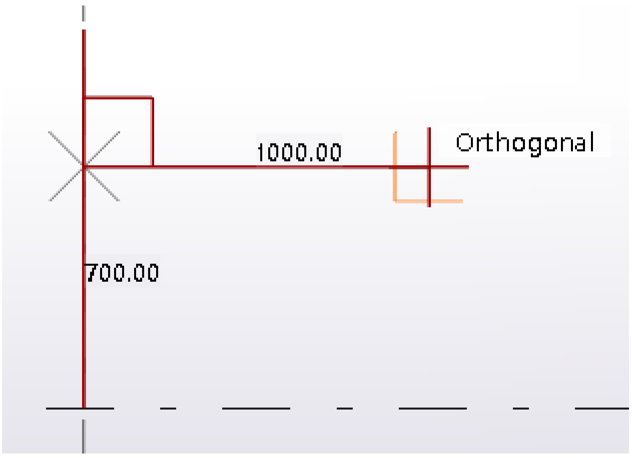
2. Pick the start point.

3. Hold down the **Ctrl** key and pick a position.

   Alternatively, right-click and select **Define temporary snap reference point** and pick a position.

   A gray cross indicates that this position is now a temporary reference point. You can continue snapping from the temporary reference point.

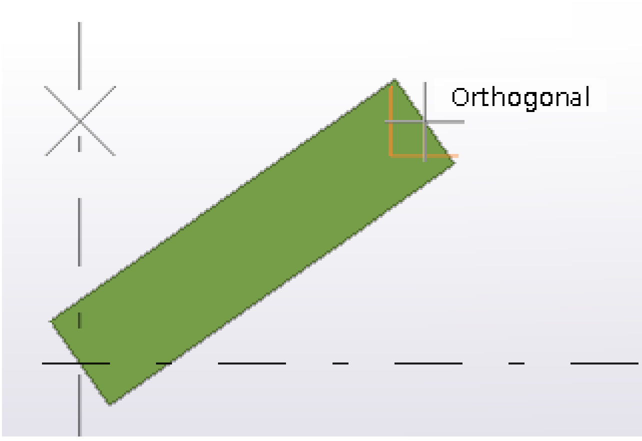


4. Repeat step 3 to create as many reference points as needed.

   If you use the **Define temporary snap reference point** command, you need to activate it for every reference point you pick.

5. Release the **Ctrl** key and pick the end point.

   Tekla Structures creates the object between the start point and the end point. For example:



Settings for the Ortho tool

Use the settings (page 104) in the **Snap settings** dialog box to set the angle interval for **Ortho**. Use the **Angle interval** or the **Custom angles** option.

The default angle interval value is 90 degrees.

Snap settings

Use the **Model snap settings** dialog box in **File menu** --> **Settings** --> **Snap settings** to view and modify the snap settings in the model. The **Drawing snap settings** dialog box has the same options for drawings. These settings are user specific.

Option	Description
Symbol	Show or hide the snap symbols. Select the check box to show the snap symbols, and clear the check box to hide them.
Active (when free snap is on)	Select the check box to activate the snap grid (page 99).
Spacing	Define grid spacing intervals for the snap grid origin. For example, if the spacing of the x coordinate is 500, you are can snap to positions at intervals of 500 units in the x direction.
Origin	Define offsets for the snap grid origin.
Angle interval	Set the angle interval for the **Ortho** tool. This setting is used when you snap to orthogonal points (page 100). For example, if you set the interval to **10**, the **Ortho** tool will snap to angles at intervals of 10 degrees in the model or drawing.
Custom angles	Define custom angles for the **Ortho** tool. This setting is used when you snap to orthogonal points (page 100). Separate the values with empty spaces. For example, if you enter `12.5 60`, the **Ortho** tool will snap to the angles 12.5 and 60 in the model or drawing.

See also

Snap to points by using snap switches (page 84)

1.4 Create model objects

When you model in Tekla Structures, you create and work with different types of model objects. In most cases, a model object represents a building object that will exist in the real building or structure, or that will be closely related to it. A model object can also be a modeling aid that represents information that is only relevant when you are creating the model. Model objects are either created in the model or imported to it.

You can create different types of model objects, such as parts and items, bolts, reinforcement, and cuts, by using the commands on the ribbon.

Some ribbon commands have a keyboard shortcut that speed up your modeling work. You can customize the keyboard shortcuts and assign your own shortcuts for the most used commands.

Alternatively, you can use Quick Launch or the property pane to start many of the commands that create model objects.

After you have created the model objects, you can view and modify the model object properties by using the property pane.

Examples of model objects

Examples of model objects include:

- Parts (page 196) and items (page 300)
- Bolts (page 341) and welds (page 359)
- Reinforcement (page 469) and embeds
- Surface treatment (page 402) and surfaces (page 415)
- Cuts (page 389), fittings (page 375), bolt holes (page 353), and chamfers (page 397)
- Pour breaks (page 458)
- Loads

Model objects may also be created by components (page 759).

To resize and reshape model objects, use the direct modification (page 107) handles.

The following modeling aids can be used in the model:

- Grids (page 23) and grid lines (page 28)
- Construction objects (page 621) and points
- Reference models

You can combine model objects to bigger entities by creating assemblies, cast units (page 430), and pour units (page 453).

You can manage model objects by using object groups (page 662), Organizer, and other planning tools.

Create or delete a model object

1. Run a command that creates a model object, such as a part.

- On the ribbon: click a command. For example, click to create a steel beam.
- By using **Quick Launch**: enter a search term. For example, type `steel beam` to find the **Create steel beam** command.
- In the property pane: ensure that you have nothing selected in the model. Click the **Object type list** button and select from the list the object you want to create.

2. Pick points (page 83) to place the model object in the model.

   Tekla Structures creates the model object using the current properties of the object type.

3. Follow the status bar messages to get instructions on how to proceed.
4. To create more model objects with the same properties, pick more points.

   The command runs until you end it or start another command.

5. If you want to delete a model object, select the object and press **Delete**.

1.5 Resize and reshape model objects

You can resize, reshape, and move model objects by using direct modification handles. When you select an object, Tekla Structures displays the handles and dimensions that are specific for that model object.

Direct modification can be used with the following object types:

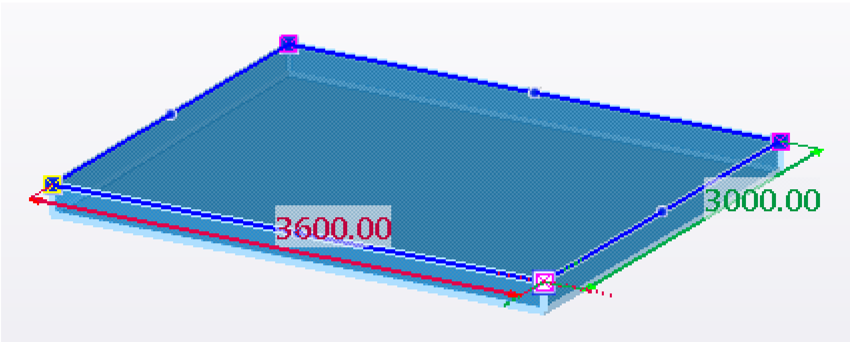
- Parts
- Construction objects
- Grids and grid lines
- Line cuts and polygon cuts
- Reinforcement
- Rebar set guidelines, modifiers, and leg faces
- Pour breaks
- Custom parts
- Loads

1. Ensure that **Direct modification** is switched on.

   To switch direct modification on or off, click or press **D**.

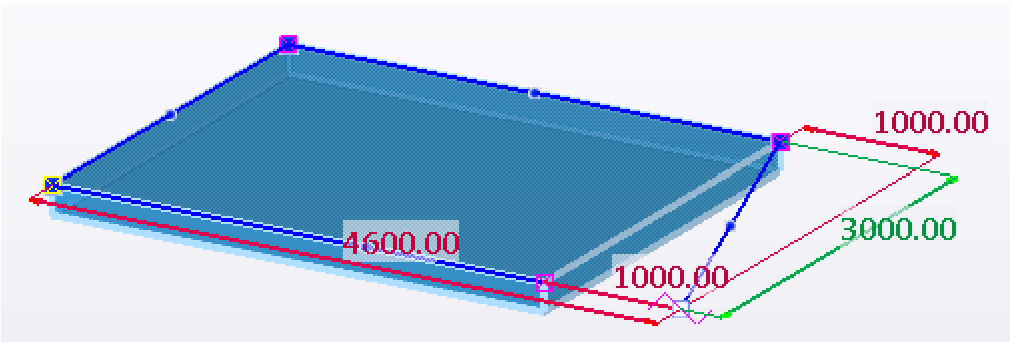
2. Click the object to select it.

Tekla Structures displays the handles that you can use to modify the object.

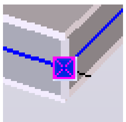
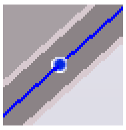
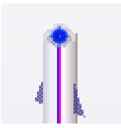


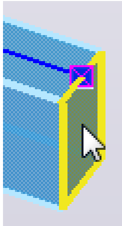
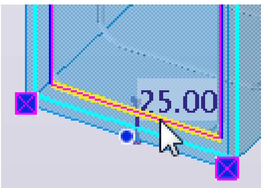
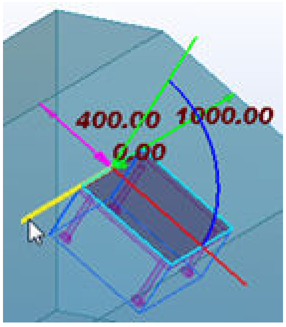
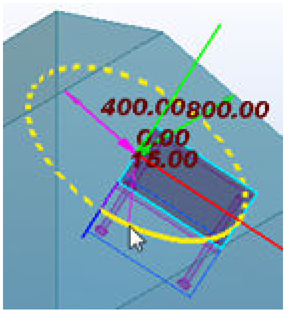
The relevant dimensions are shown when you move the mouse pointer slowly over the object's edges. The dimension colors follow the colors of the work plane coordinate axes: red in the X direction, green in the Y direction, and blue in the Z direction. Diagonal dimensions are magenta.

3. To reshape the object, drag any of the handles.



Here are some examples of direct modification handles:

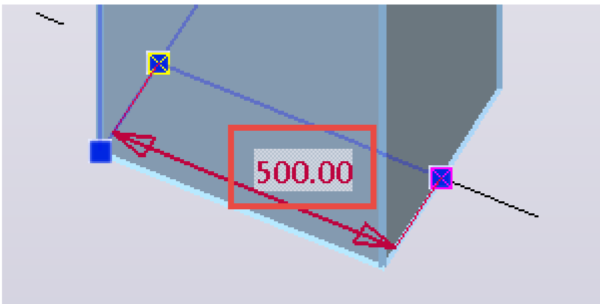
Handle	Description
	Reference point handle
	Midpoint handle
	End point handle (for reinforcing bars only)

Handle	Description
	Plane handle
	Line handle
	Axis handle (for items (page 300) and custom parts only)
	Rotation handle (for items and custom parts only)

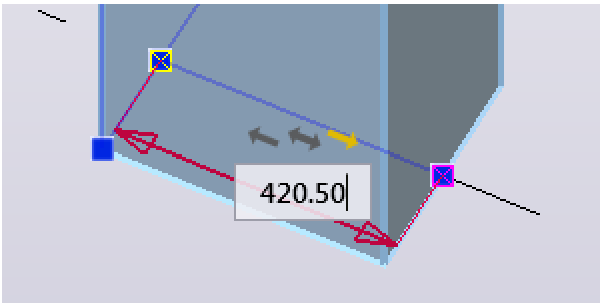
TIP You can use the snap switches (page 84) when dragging a handle. To temporarily disable the snap switches, hold down the **Shift** key when dragging a handle.

4. To give an exact value for a dimension, modify the dimension value.

   a. Click a dimension to select it.

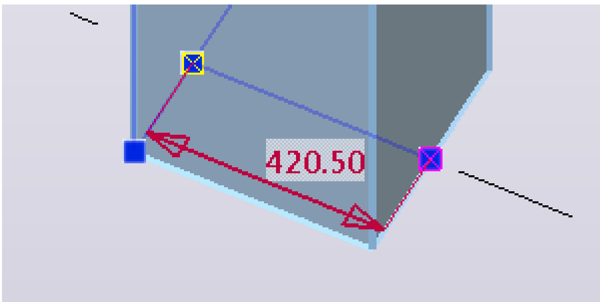


b. Type a new value.



The yellow arrow defines the direction in which the object is extended or shortened. You can change the direction by clicking the arrows.

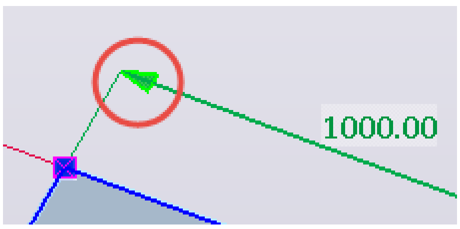
c. Press **Enter** to confirm the new value.



5. To change a dimension only from one end, move the dimension arrowheads.

   You can either drag an arrowhead to a new location, or you can type an exact distance or coordinates.

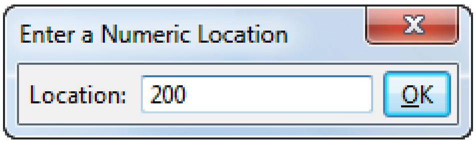
   a. Select the dimension arrowhead you want to move. For example:



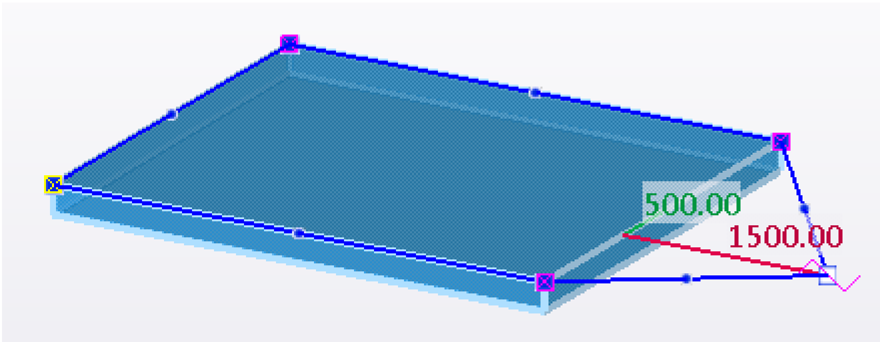
To change a dimension at both ends, select both arrowheads.

b. Type the distance or coordinates.

When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog box. Click **OK** to confirm the dimension.

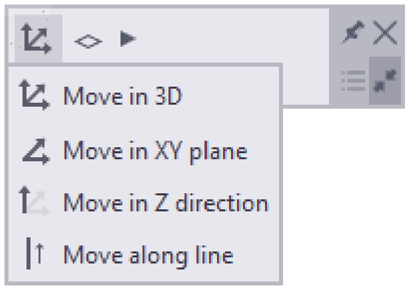


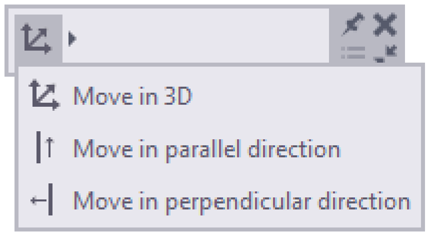
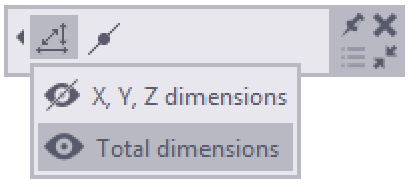
6. To add a new object corner, drag a midpoint handle . For example:



7. To display more modification options, select a handle.

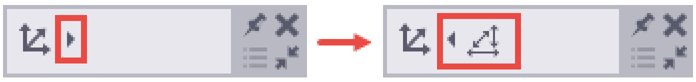
A contextual toolbar appears with more options. The availability of the options depends on the object and handle you have selected.

Click this button	To do this	Location
	Move a handle to any location in the 3D space.	
	Move a handle in the XY plane only.	
	Move a handle in the Z direction only.	
	Move a handle along the reference line only.	

Click this button	To do this	Location
	Move a handle in the parallel direction only.	
	Move a handle in the perpendicular direction only.	
	Move a handle parallel to a certain plane only. Select the plane and drag the handle to a new location. This option can be useful when working with a sloped roof, for example.	
	Control the visibility of direct modification dimensions. Click the eye symbol to show or hide dimensions. • **X, Y, Z dimensions**: All orthogonal dimensions in the work plane directions X, Y, and Z are displayed. • **Total dimensions**: Only the total	

Click this button	To do this	Location
	length is displayed.	
	Show or hide midpoint handles.	
	Add a new point at the end of an object. Only available for objects that pass through multiple points, such as polybeams, panels, strip footings, and rebar set modifiers.	

NOTE Some of these options are located in an expandable section on the contextual toolbar. Click the small triangle symbol on the contextual toolbar to show or hide the options:



8. To delete a handle, select it and press **Delete**.

See also

Modify a construction object (page 628)

Modify a single grid line (page 28)

Modify a single reinforcing bar, bar group, or mesh (page 544)

Modify a rebar set (page 521)

Modify a pour break (page 463)

Add custom components to a model (page 845)

1.6 Select objects

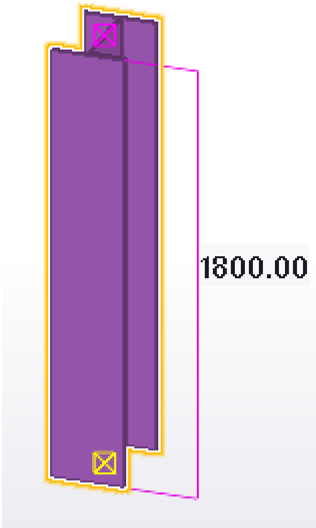
Many Tekla Structures commands require to select objects. You can make single selections and area selections. Tekla Structures highlights the selected objects. The number of selected objects and handles is displayed in the bottom right corner of the status bar. For example:

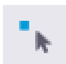
Use the different commands and methods to select the objects. To control which type of objects can be selected, use the selecting toolbar and the selection switches.

Select single objects

1. Make sure that the correct selection switches are active.
2. Click an object to select it.

   Tekla Structures displays dimensions and dimension lines for column, beam, reinforcing bar group, and for rebar set. If you want to hide the dimensions, use the advanced options XS_DISPLAY_DIMENSIONS_WHEN_SELECTING_OBJECTS and XS_DISPLAY_DIMENSIONS_WHEN_SELECTING_REBARS.

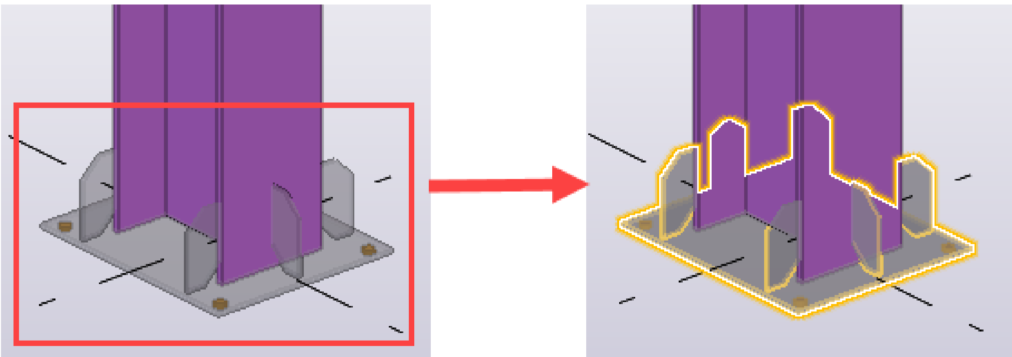


NOTE When direct modification (page 107)  is switched on, the object dimensions and dimension lines are always hidden. Only the direct modification dimensions are then shown. This makes it easier to know which dimensions can be edited.

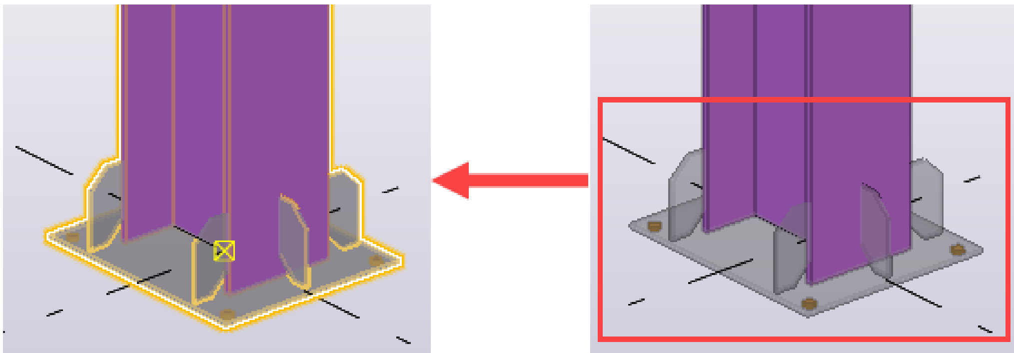
Select multiple objects using area selection

You can select multiple objects using area selection. By default, the dragging direction affects the selection of objects.

1. Make sure that the correct selection switches are active.
2. To select all objects that are completely within a rectangular area, hold down the left mouse button and drag the mouse from **left to right**.



3. To select all objects that are at least partly within a rectangular area, hold down the left mouse button and drag the mouse from **right to left**.

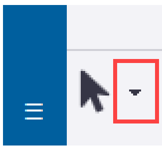


4. If you want to change how area selection works, click **File menu** --> **Settings** and select or clear the **Crossing selection** check box.

   By default, the option is switched off. When the option is **off**, the dragging direction affects the selection of objects. When the option is **on**, all objects that fall at least partially inside the rectangular area are selected, regardless of the dragging direction.

Select all objects

To select all objects at once, do one of the following:

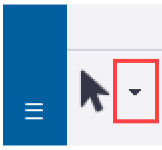
- On the ribbon, click the small down arrow next to the arrow button , and then click **Select all objects**.
- Press **Ctrl+A**.

Note that

- the **Select all objects** command respects the selection filter so that only the objects that are matching the current selection filter are selected.
- the **Select all objects** command selects also those objects that are hidden by a view filter, work area, or that have been hidden by using the **Hide** command, if they match the selection filter.

Select previous objects

Sometimes you need to select again the same objects that you selected previously but have then deselected. To select previously selected objects, do one of the following:

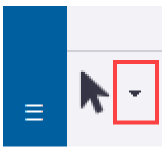
- On the ribbon, click the small down arrow next to the arrow button , and then click **Select previous objects**.
- Press **Alt+P**.

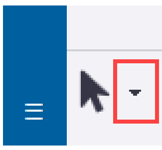
Select objects by identifier

If you know the GUID (globally unique identifier), or the ID of an object, or the IFC GUID of a reference object, you can use the **Select by identifier** command to locate the objects in a model or in a drawing.

You can often find information about the object GUID or ID, for example, in reports and log files. By using the **Select by identifier** command you can quickly find the objects in a model or in a drawing, instead of defining a view filter or a selection filter with the specific GUID or ID. You can use IFC GUIDs to find IFC reference objects. This is useful if you need to track updates and changes in IFC reference models.

Additionally, you can use the **Select by identifier** command to inquire the GUIDs of selected objects, instead of using the traditional inquiry (page 686).

To	Do this
Find objects based on the object GUID, ID, or IFC GUID identifier	1. Do one of the following: • In the modeling mode: on the ribbon, click the small down arrow next to the arrow button , and then click **Select by identifier**.

To	Do this
	• In the drawing mode: in **Quick Launch**, type **Select by identifier**. The **Select by identifier** dialog box opens. 2. Copy the object identifier, for example from a log file, to the dialog box. You can enter multiple identifiers in the dialog box. Either enter each identifier on its own row, or separate them with semicolon `;`. 3. To define the search, select the needed check boxes. • **Reference objects**: Tekla Structures selects IFC objects based on their GUID or IFC GUID. • **Keep selection**: Tekla Structures keeps the currently selected object and appends it with new selection. • **Zoom to selected**: Tekla Structures selects the object and zooms to it. 4. Click **Select**. Tekla Structures selects the objects based on the GUID in the model or in the drawing. If there are identifiers that are not found in the model or in the drawing, they are listed in the status bar as `identifier?`.
Find a model object in a drawing	You can select an object in a model, get its identifier, and then find it in a drawing based on the identifier. 1. In the modeling mode: on the ribbon, click the small down arrow next to the arrow button , and then click **Select by identifier**. The **Select by identifier** dialog box opens. 2. Select an object or objects in the model.

To	Do this
	3. Click **Get**. The **Select by identifier** dialog box lists the identifiers of the selected objects. If you want to get IFC GUIDs, make sure that the **Reference objects** check box is selected. 4. Keep the dialog box open. 5. Open a drawing. 6. In the drawing mode, click **Select** to find the objects in the drawing. You can then continue working with the found objects.
Find a drawing object in a model	You can select an object in a drawing, get its identifier, and then find it in a model based on the identifier. 1. In the drawing mode: in **Quick Launch**, type **Select by identifier**. The **Select by identifier** dialog box opens. 2. Select an object or objects in the drawing. 3. Click **Get**. The **Select by identifier** dialog box lists the identifiers of the selected objects. 4. Keep the dialog box open. 5. Close the drawing. 6. In the modeling mode, click **Select** to find the objects in the model. You can then continue working with the found objects.

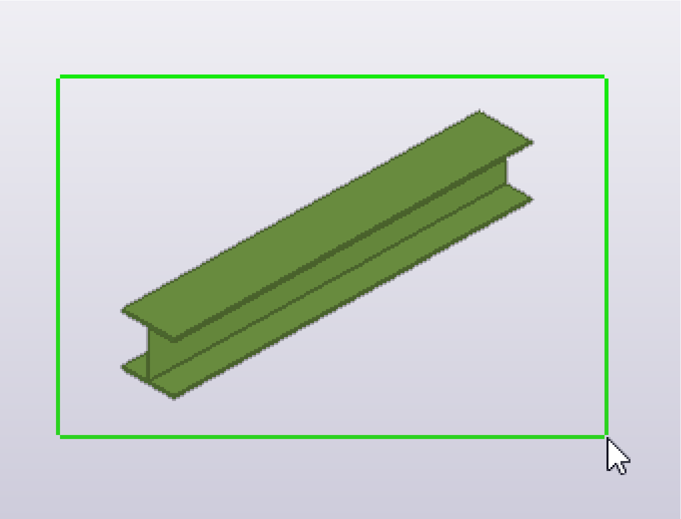
Select handles

Sometimes you need to select only the handles of a part, for example when moving the part.

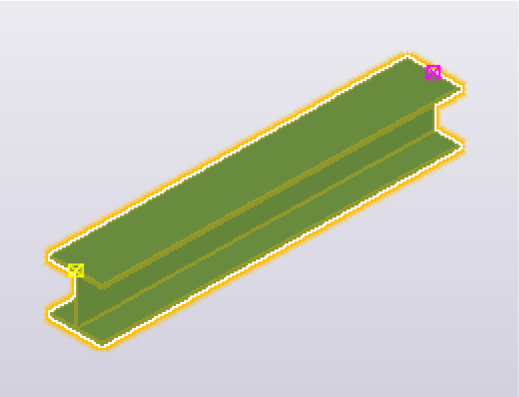
Before you start, make sure that **Crossing selection** is switched off, and that the **Direct modification** switch  is not active.

1. On the **File** menu, click **Settings** and make sure that **Crossing selection** is switched off. If the **Crossing selection** is not switched off, selecting the handles with the **Alt** key does not work.

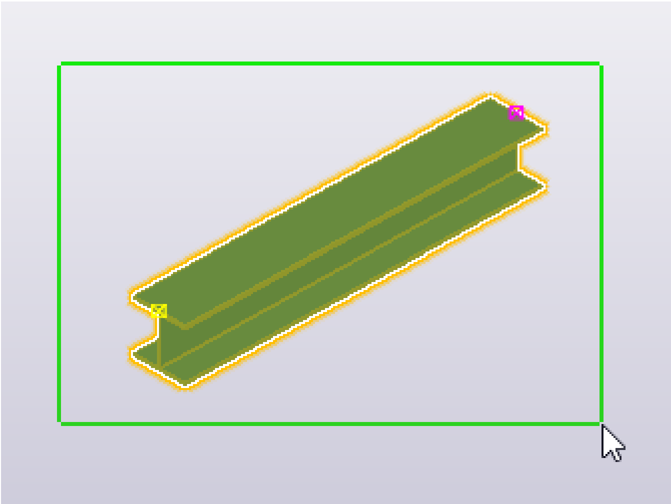
2. Make sure that the correct selection switches are active.
3. Hold down the left mouse button and drag the mouse from left to right to include the entire part.



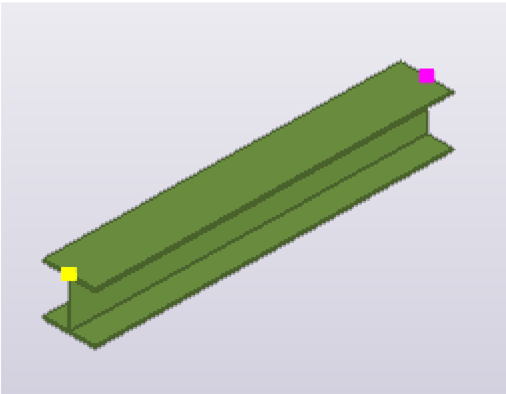
The part becomes selected:



4. Hold down the **Alt** key and drag the mouse from left to right again.



Now only the part handles are selected:



NOTE When direct modification (page 107)  is switched on, Tekla Structures also displays the direct modification handles for reference points, corners, segments, and segment midpoints of the selected part. These handles are blue.

Modify the selection

You can add objects to the current selection, or remove objects from the selection.

1. To add objects to the current selection, press the **Shift** key and select more objects.
2. To switch the selection of an object on or off, press the **Ctrl** key during the selection.

   Tekla Structures deselects the objects that were already selected and selects those that were previously not selected.

3. To clear the selection of all objects and handles, click somewhere else.

   For example, click on the empty background of the current view.

Select assemblies, cast units, and nested objects

You can select either assemblies or cast units, or single objects in nested assemblies or nested components.

Select assemblies and cast units

Use the **Select assemblies** selection switch to select assemblies and cast units (page 430).

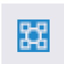
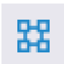
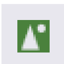
1. Ensure that the  **Select assemblies** selection switch is active.
2. Select a part.

   Tekla Structures selects the entire cast unit or assembly that contains the selected part.

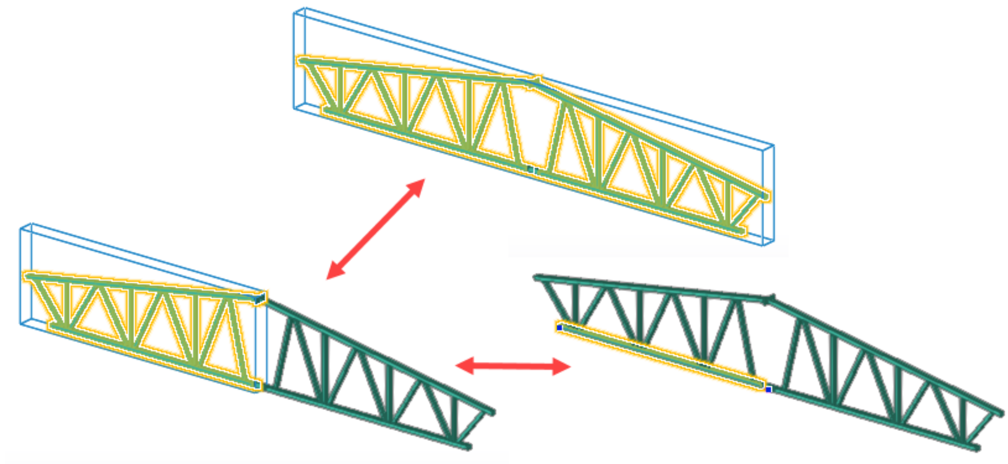
Select nested objects

You can select nested assemblies and components. The active selection switch defines on which level you start and toward which direction you move in the component or assembly hierarchy. The status bar shows the steps you take in the hierarchy.

1. Ensure that the correct selection switch is active.

   - : to start from the assemblies on the highest level, move to their sub-assemblies, and finally select single parts, bolts, and so on
   - : to start from single objects and move to higher levels of nested assemblies
   - : to start from the components on the highest level, move to their sub-components, and finally select single parts, bolts, and so on
   - : to start from single objects and move to higher levels of nested components
2. Place the mouse pointer on any part in the assembly or in the component.
3. Hold down the **Shift** key.
4. Scroll with the mouse wheel.

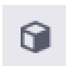
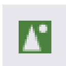
The blue highlighting box indicates the assembly or the component that you can select.



Select reference models, reference model objects and assemblies

You can select either entire reference models, or single objects and assemblies that are part of a reference model. The use of selection switches differs in each case.

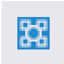
Select an entire reference model

1. Activate the  **Select reference models** selection switch.
2. Activate the  **Select components** selection switch.
3. Select the reference model.

Select a reference model object

1. Activate the  **Select reference models** selection switch.
2. Activate the  **Select objects in components** selection switch.
3. Select the desired object in the reference model.

Select a reference model assembly

1. Activate the  **Select reference models** selection switch.
2. Activate the  **Select assemblies** selection switch.
3. Select the desired assembly in the reference model.

Tips for selecting objects

Here are listed some tips that can help when you are selecting objects.

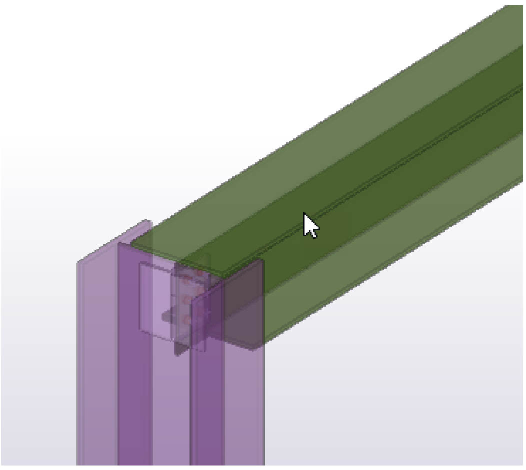
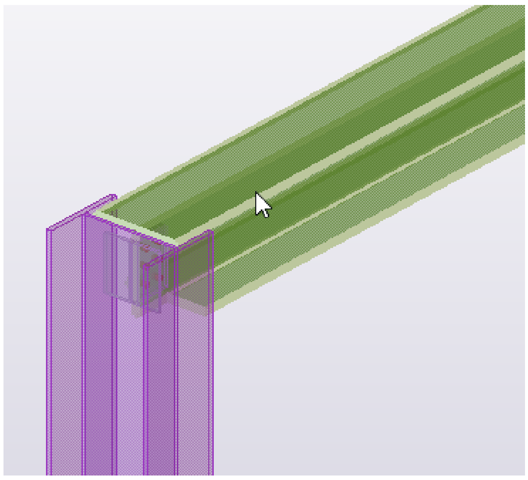
Switch rollover highlight on or off

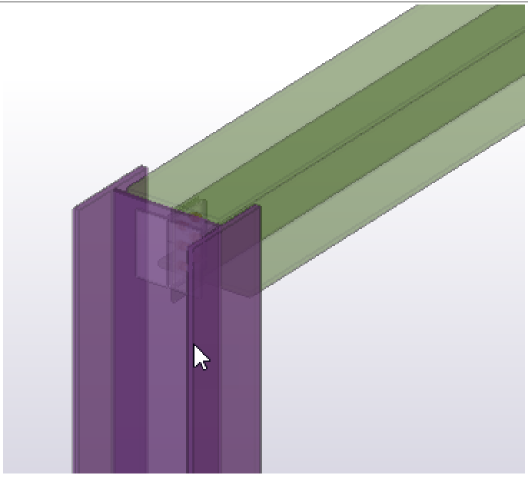
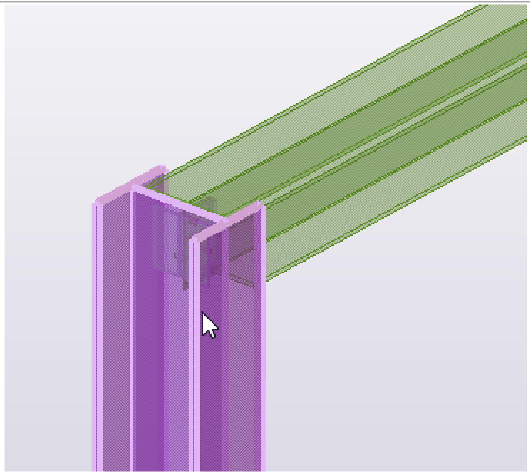
By default, Tekla Structures highlights the objects that you can select. You can switch the highlighting on or off.

To switch the rollover highlight on or off, on the **File** menu, click **Settings** and select or clear the **Rollover highlight** check box. Alternatively, you can press **H**.

Depending on the rendering engine (page 72) you are using, OpenGL or DirectX, Tekla Structures highlights the objects differently when rollover highlight is on.

In the example below, the rendering of parts (page 642) is set to **Parts shaded wireframe**.

DirectX	OpenGL
Tekla Structures highlights the objects by showing them with darker object color. For example: 	Tekla Structures highlights the objects by showing them with light edge line color. For example: 

DirectX	OpenGL
	

Select on right-click

You can change the settings so that you can select objects also with the right mouse button.

1. On the **File** menu, click **Settings** and select the following check boxes:
    - **Select on right-click**
    - **Rollover highlight**
2. Right-click an object to select it. Tekla Structures highlights the object and shows the related shortcut menu.

If you cannot select objects

If you cannot select desired objects in the model, check the selection switches and the filter settings.

- Check that you have switched on all the needed selection switches.
- If you still cannot select the objects, check the selection filter settings. You can select a different filter or modify the current filter.

Interrupt object selection

You can have Tekla Structures interrupt the object selection process if the selection takes over a defined period of time. For example, if you are working on a large model and you accidentally select all or part of the model, you can interrupt the selection if it takes over 5000 milliseconds (5 seconds) to complete.

1. Define the time after which Tekla Structures asks if you want to interrupt object selection.
    a. On the **File** menu, click **Settings** --> **Advanced options** and go to the **Modeling properties** category.

b. Modify the advanced option XS_OBJECT_SELECTION_CONFIRMATION.

   The default value is 5000 milliseconds.

c. Click **OK**.

2. Select (page 113) all or part of the model.
3. When Tekla Structures asks if you want to interrupt object selection, click **Cancel**.

1.7 Copy and move objects

The basic functionality of copying and moving objects is the same in models and drawings. You can copy and move objects linearly, with rotation, and with mirroring.

- Copy objects (page 127)
- Move objects (page 140)
- Rotate objects (page 145)
- Mirror objects (page 149)

Tips for copying and moving objects

By copying objects the modeling work becomes faster, more efficient, and more consistent.

- **Copy carefully**

  Always copy the objects carefully, and ensure that you are copying the intended objects.

- **Control over copying**

  At first, when you familiarize yourself with snapping, we recommend to use the **Copy Special - Linear - Pick** method to have more control over the copying action.

- **Compare copied objects**

  Any comparison between the copied objects should be done before running numbering. The numbering itself works as the final check.

- **Select a suitable command for copying**
  - To ensure that the object is copied in the intended plane, use the **Copy special** --> **Linear** command. The **Copy - Linear** dialog box can be used as an explicit check to confirm that copying distance is in the intended direction and round values.
  - To copy objects, such as reinforcement, between similar objects, use the **Copy special** --> **To another object** command. Always ensure that the object you copy from and the object you copy to are of similar type and have similar shape. For example, a polygonal slab and a rectangular

column have different types of part handles, and their front faces are of different shape and location.

  - To copy objects around a specified line on the work plane, use the **Copy special** --> **Rotate** command. When you use this command, copy carefully and always check the outcome. If the results are not as expected, copy in smaller chunks, for example, one component at a time.
  - To copy objects from one model to another, use the **Copy special** --> **From another model** command. The copying is based on the phase numbers in the original model. Successful copying requires that you have set the objects correctly, without any additional objects, in a specific phase in the source model. Otherwise all the objects included in the phase are copied.

    Note that if you copy objects from another model, only the model objects are copied. The drawings are not copied.
  - The source object for the copying determines the object orientation.

    When the objects are copied with the **Copy** command, the object orientation of the target object remains the same as the orientation of the source object.

    When the objects are copied with the **Copy special** --> **To another object** command, the object orientation is defined in relation to the source object's internal coordinate system, and this orientation will be translated into the target object's internal coordinate system.

    Custom components have their own logic for object orientation. It may be more straightforward to add the component in the model than to use copying, especially if the target object's main geometry differs considerably from the source object's geometry.

- **Duplicate objects**

  After copying and moving, check that the outcome is as expected and there are no accidentally created duplicates in the model.

  Two objects are considered duplicates if they have the same properties and location. Tekla Structures checks for duplicate, overlapping objects when you copy and move objects or create new objects in the same location as an existing object. If duplicates are found, you can choose whether to keep or delete them. If you choose to keep the duplicates, they are hard to detect later on.

  Use the advanced option XS_DUPLICATE_CHECK_LIMIT_FOR_COPY_AND_MOVE to define the maximum number of objects that can be counted as duplicates while copying or moving objects.

  NOTE Tekla Structures does not check for duplicates when you copy objects using a modeling tool, such as the **Array of objects (29)** component.

- **Assemblies and cast units**

  If you copy or move objects from an assembly or cast unit, Tekla Structures copies the assembly structure if possible. For example, sub-assemblies are copied as sub-assemblies if a parent object is found.

  When selecting the correct content to be copied, first use model selection filters (page 151), and secondly assembly, part, or component selection switches.

  To easily select all objects within an assembly or a cast unit according to the selection filter, hold down the **Alt** key and click any object in the assembly or cast unit.

- **Drawing objects**

  You can copy and move drawing objects between drawing views that have different scales.

- **Reinforcement and surface treatment**

  If you copy or move reinforcement or surface treatments (page 402), and want them to adapt to the part they are copied or moved to:

  - The reinforcement handle or surface treatment handles must be in part corners.
  - The parts between which you copy or move must have the same number of cross section corners.
  - Circular parts must have the same cross section dimensions.

- **Copy and move efficiently**

  You can keep the **Move** and **Copy** dialog boxes open if you are going to use them often.

  After you have run the **Copy - Linear**, **Copy - Mirror**, **Copy - Rotate**, or the **Move - Linear**, **Move - Mirror** or **Move - Rotate** command, interrupt the command and leave the dialog box open. When you want to continue copying or moving, click the dialog box to activate it and continue to copy or move objects.

Copy objects

You can copy objects in a number of different ways. When you copy an object, Tekla Structures copies all objects connected to it, including the components.

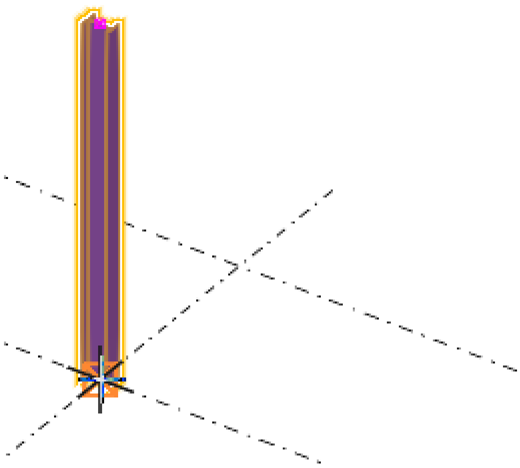
Copy by picking two points

The basic way to copy objects in a model or drawing is by defining the origin and one or more destination points.

1. Select the objects you want to copy.
2. Run the **Copy** command:

- In the model, on the **Edit** tab, click **Copy**.
- In the drawing, on the **Drawing** tab, click **Copy** --> **Copy**.

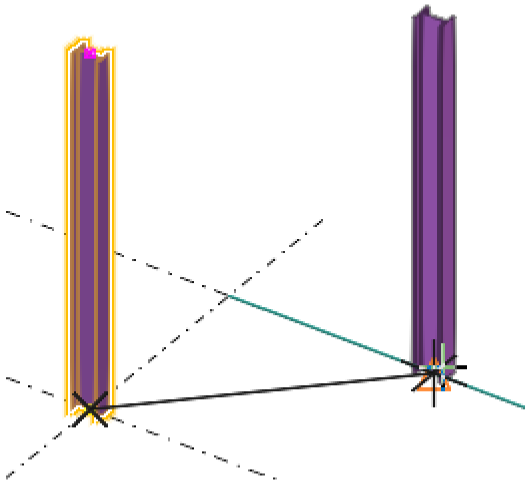
3. Pick the origin for copying.



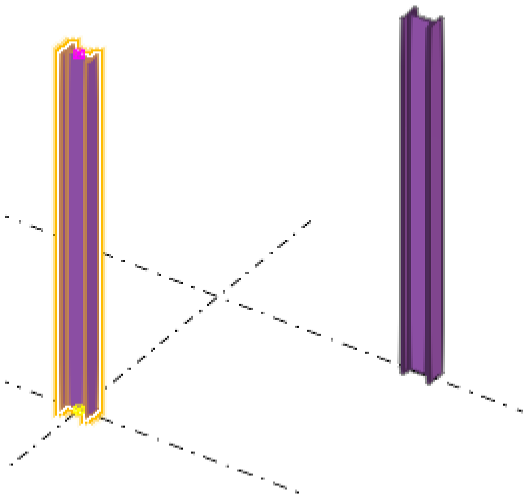
In the model, Tekla Structures displays a rubber band line between the first picked point and the cursor position. This is a preview of the position where the objects will be copied. Move the cursor to see how the preview changes.

Note that Tekla Structures always displays the preview in the position where the objects will be copied, not in the position where the cursor is when you pick the destination point.

4. Pick one or more destination points.



The objects are copied immediately. The **Copy** command remains active.



5. If you want to undo the latest copy operation, click the **Undo** button on the top left corner of the Tekla Structures main window.

   The **Copy** command still remains active.

6. To stop copying, press **Esc**.

NOTE If you want to limit the number of objects shown in the preview, use the advanced option `XS_PREVIEW_LIMIT`. The default value is 1000. When the value is 0, the preview is off.

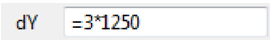
Copy linearly

In the model, you can create multiple copies of an object in the same linear direction.

1. Select the objects you want to copy.
2. On the **Edit** tab, click  **Copy special** --> **Linear**.

   The **Copy - Linear** dialog box opens.
3. Pick two points, or enter the coordinates in the **dX**, **dY**, and **dZ** boxes.

   You can also use a formula to calculate the x, y, and z displacements. For example:

4. Enter the number of copies.
5. Click **Copy**.
6. To stop copying, press **Esc**.

   TIP If the dialog box is open but the command is no longer active, click the **Pick...** button to re-activate the command.

Copy by specifying a distance from origin

You can copy objects to a new position in the model or drawing by specifying a distance from the origin. Use the **Enter a Numeric Location** dialog box to specify the distance.

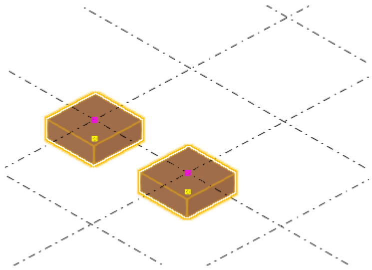
1. Select the objects you want to copy.
2. Run the **Copy** command:
   - In the model, on the **Edit** tab, click  **Copy**.
   - In the drawing, on the **Drawing** tab, click  **Copy** --> **Copy**.
3. Pick the origin for copying.
4. Move the cursor in the direction you want to copy the objects, but do not pick the point.
5. Type the distance.

   When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog box automatically.
6. Click **OK**.

Copy using drag-and-drop

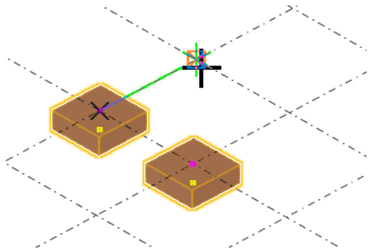
You can copy objects by using drag-and-drop.

1. On the **File** menu, click **Settings** and select the **Drag & drop** check box to activate the command.
2. Select the objects you want to copy.

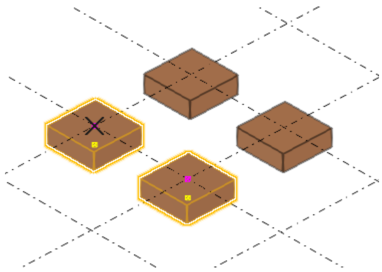


3. Hold down the **Ctrl** key and drag the objects to a new location.

   The point you start dragging from (center, corner, or middle point) affects the object's alignment in the new location.



   Tekla Structures copies the objects:



NOTE To copy grid labels in a drawing, first select the grid label and then either activate the  **Select grid line** selection switch or select the grid label handle.

Copy objects to another object

In the model, you can copy reinforcement, surface treatments, and single part welds between similar objects and make them adaptive to the object they are copied to. This is useful, for example, when you detail previously modeled parts. The objects that you can copy between can have different dimensions, length, and rotation. If you copy objects from an assembly or cast unit, Tekla Structures copies the assembly structure as well, if possible. For example, sub-assemblies are copied as sub-assemblies if a parent object is found.

TIP Instead of copying objects from an assembly or cast unit to other identical assemblies or cast units, you can use the **Batch editor** tool. **Batch editor** detects matching objects in the target assemblies or cast units, and edits the matching objects by modifying their geometry and properties.

If you copy reinforcement or surface treatments and want them to adapt to the part they are copied to, note the limitations:

- The reinforcement or surface treatment handles must be in part corners.
- The parts between which you copy must have the same number of cross section corners.
- Circular parts must have the same cross section dimensions.

1. Select the objects you want to copy.
2. On the **Edit** tab, click **Copy special** --> **To another object**.
3. Select the object to copy from (source object).
4. Select the objects to copy to (target object).

Copy all content to another object

In the model, you can copy objects from an assembly or cast unit to other similar assemblies or cast units without individually selecting each object to copy. This is useful, for example, when you have detailed an assembly and want to copy all details to another similar assembly.

TIP Instead of copying objects from an assembly or cast unit to other identical assemblies or cast units, you can use the **Batch editor** tool. **Batch editor** detects matching objects in the target assemblies or cast units, and edits the matching objects by modifying their geometry and properties.

---

1. Ensure that the **Select assemblies** selection switch is active.
2. Select the assembly or cast unit to copy from (source object).
3. On the **Edit** tab, click **Copy special** --> **All content to another object**.
4. Select the assemblies or cast units to copy to (target objects).

As a result, Tekla Structures copies the following objects:

- Secondary parts
- Reinforcement, bolts, and welds
- Cuts, fittings, and edge chamfers
- Sub-assemblies
- Components

NOTE Tekla Structures does not copy pour breaks, or secondary parts created by a component that has also created the assembly main part. If some of the objects to be copied already exist in the assembly or cast unit to copy to, Tekla Structures may create duplicate objects. Tekla Structures warns you about duplicate secondary parts, reinforcement, and sub-assemblies, but not about duplicate bolts, welds, cuts, or components.

Copy to another plane

In the model, you can copy objects from the first plane you specify to the second, third, and so on, plane you specify. The position of the copied objects relative to the second, third, and so on, plane remains the same as the position of the original objects relative to the first plane.

1. Select the objects you want to copy.
2. On the **Edit** tab, click **Copy special** --> **To another plane**.
3. Pick the point of origin of the first plane.
4. Pick a point on the first plane in the positive x direction.
5. Pick a point on the first plane in the positive y direction.
6. Repeat steps 3–5 for all destination planes.

Copy from another model

You can copy objects from another model based on phase numbers. Note that Tekla Structures copies secondary parts from the model only if they belong to the same phase as their main part. This also applies to component objects.

1. On the **Edit** tab, click **Copy special** --> **From another model**.

   The **Copy from Model** dialog box opens.
2. In the **Model directories** list, select the model to copy from.

   This is the source model. Note that the target model must have been created using the same or newer version of Tekla Structures as the source model. You cannot copy from a newer version to an older version.

3. In the **Phase numbers** box, enter the numbers of the phases from which to copy objects, separated by spaces.

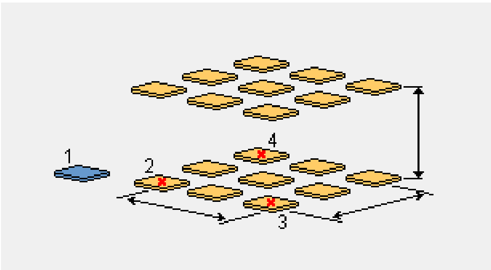
   For example, `2 7`.

4. Click **Copy**.
5. Close the dialog box.

NOTE If you copy pour breaks from another model, the copied pour breaks automatically adapt to the target model. Always check that the copied pour breaks have adapted correctly.

Copy objects using linear array tool

Use **Linear array tool** to copy selected objects linearly along multiple directions at defined intervals or spacing. Tekla Structures does not check for duplicates when you copy objects using this method.



How to use Linear array tool

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Search for **Linear array tool**, and then double-click to open it.
3. Select the **Copy method**. The options are:
   - **Selected objects only**

     This is the default. Only the selected objects are copied.
   - **All associated objects**

     Selected objects and all objects associated with them are copied. For example, cuts and fittings applied to a part.
   - **Advanced**

     This option is similar to **All associated objects**, but works better with modifications. For example, when you have stairs that have posts welded to the steps, and you modify the distance between steps.
4. Select the **Copy origin**. The options are:

- **Object to be copied**

  This is the default. Copies are relative to the input objects.

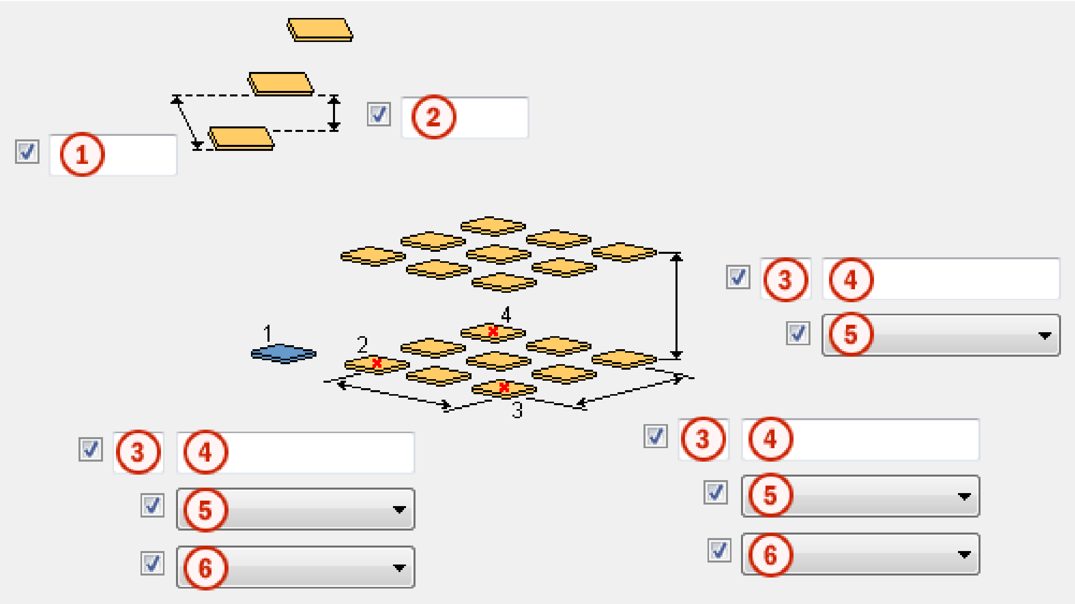
- **Origin point**

  Copies are relative to the input origin point.

5. Define the settings.
6. Select the objects to copy.
7. Click **OK** to close the dialog box.
8. Click the middle mouse button.
9. Pick origin point.
10. Pick axis direction X.
11. Pick axis direction Y.

    The selected objects are copied.

How to define the settings

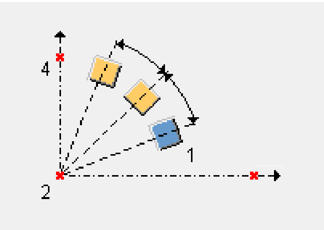


1	Offset along the Y axis. The default value is 0 mm.
2	Offset along the Z axis. The default value is 0 mm.
3	Number of copies. The default value is 0. If you leave this empty, the number of copies is taken from the **Space between copies** field.

4	Space between copies. The default value is 0 mm. Use the space character to separate values. Enter a value for each space between copies. This option is not available if you select **Equal** as the spacing method.
5	Copy direction. The options are: • **Normal** (default) Spacing values are calculated from the origin in positive direction along the axis. • **Reverse** Spacing values are calculated from the origin in negative direction along the axis. • **Centered** Copies are centered on the origin. • **Mirror** Spacing values are calculated from the origin in both positive and negative direction. Mirrored copying doubles the number of copies.
6	Spacing method. The options are: • **Equal** (default) Copies are equally spaced based on the length of the X or Y axis. • **Specified** Copies are spaced according to the number and spacing values given.

Copy objects using radial array tool

Use **Radial array tool** to copy selected objects radially along multiple directions at defined intervals or spacing. Tekla Structures does not check for duplicates when you copy objects using this method.



How to use Radial array tool

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Search for **Radial array tool**, and then double-click to open it.

3. Select the **Copy method**. The options are:
    - **Selected objects only**

      This is the default. Only the selected objects are copied.
    - **All associated objects**

      Selected objects and all objects associated with them are copied. For example, cuts, welds, and bolts.
    - **Advanced**

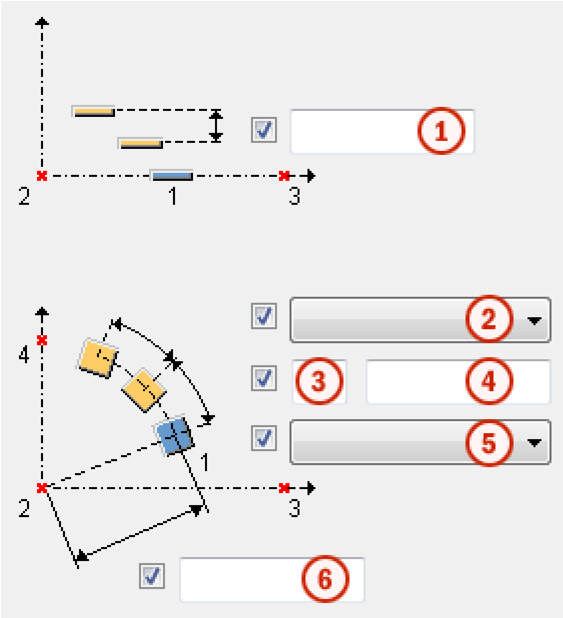
      This option is similar to **All associated objects**, but works better with modifications. For example, when you have stairs that have posts welded to the steps, and you modify the distance between steps.
4. Select the **Rotate copies** option.

   The default is **Yes**.
5. Define the rotation axis.

   The default is **X**.
6. Define the settings.
7. Select the objects to copy.
8. Click **OK** to close the dialog box.
9. Click the middle mouse button.
10. Pick origin point.
11. Pick axis direction X.
12. Pick axis direction Y.

    The selected objects are copied.

How to define the settings



1	Distance between copies. The default value is 0.
2	Rotation. The options are: • **Angle** (default) The copies are rotated by angle. • **Distance** The copies are rotated by distance.
3	Number of angles or distances. The default value is 0. If you leave this empty, the number of copies is taken from the **Space between copies** field.
4	Space between copies. Use the space character to separate values. Enter a value for each space between copies.
5	Copy direction. The options are: • **Normal** (default) Spacing values are calculated from the origin in positive direction along the axis. • **Reverse** Spacing values are calculated from the origin in negative direction along the axis. • **Centered** Copies are centered on the origin.

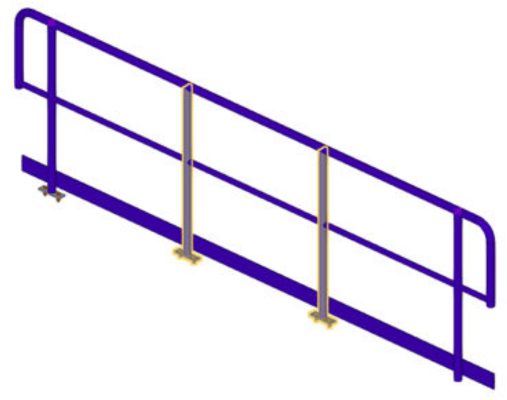
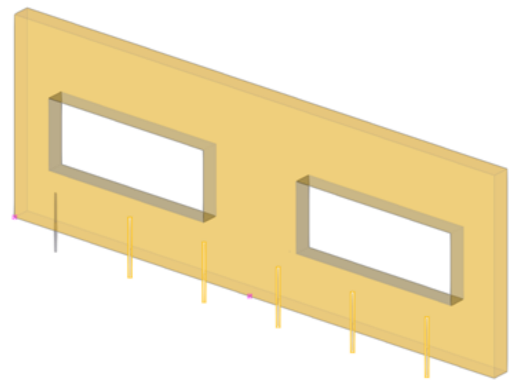
	• **Mirror** Spacing values are calculated from the origin in both positive and negative direction. Mirrored copying doubles the number of copies.
6	Radial distance. The radial distance should be equivalent to the distance you picked when applying the component. If the radial distance is smaller or greater than the picked distance, the spacing between the copied objects is not the same as given in the **Space between copies** box (4). Tekla Structures calculates the rotation angle according to the dialog box values (spacing and radial distance), and the rotation angle overrides the spacing given in the dialog box.

Copy objects using Array of objects (29) component

Use the **Array of objects (29)** component to copy model objects along a line. If you modify the original object, Tekla Structures also changes the copied objects.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Search for the **Array of objects (29)** component, and then double-click to open it.
3. Define the settings:
    - **Number of copies**: Enter the number of copies you want to create.
    - **Spacing values**: Define the spacing of the objects.
    - **Copy to the opposite direction**: Select **Yes** if you want to copy in the direction opposite to the points you pick.
    - **Start point for copying**: Choose either the object to be copied or the first input point.
    - **Copy at equal distances (Ignore spacing values)**: Select **Yes** if you want to create the objects at equal distances. **Spacing value** will be ignored.
4. Click **OK** to save the settings.
5. Select the objects to copy.
6. Click the middle mouse button to finish selecting.
7. Pick a point to indicate the start of the line along which to arrange copied objects.
8. Pick a point to indicate the end of the line.

Examples

Example	Description
	An array of steel objects.
	An array of concrete objects.

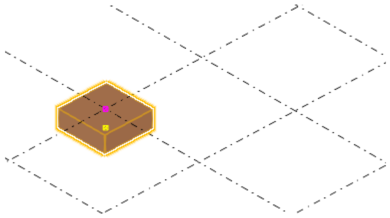
Move objects

You can move objects in a number of different ways, especially in models. When you move an object, Tekla Structures also copies all objects connected to it, including the components.

Move by picking two points

The basic way to move objects in a model or drawing is by defining the origin and one or more destination points.

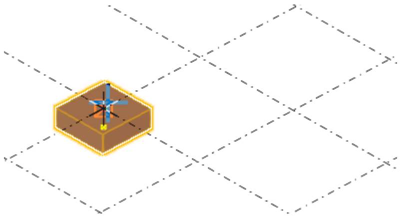
1. Select the objects you want to move.



2. Run the **Move** command:

   - In the model, on the **Edit** tab, click  **Move**.
   - In the drawing, on the **Drawing** tab, click  **Move --> Move**

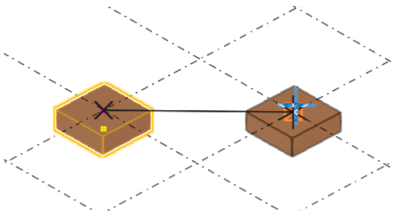
3. Pick the origin for moving.

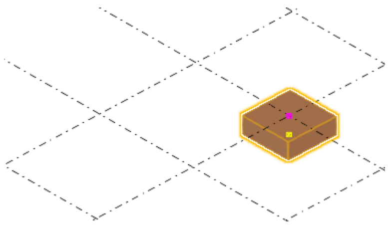
   In the model, Tekla Structures displays a rubber band line between the first picked point and the cursor position. This is a preview of the position where the objects will be moved. Move the cursor to see how the preview changes.

   Note that Tekla Structures always displays the preview in the position where the objects will be moved, not in the position where the cursor is when you pick the destination point.

4. Pick the destination point.

   The objects are moved immediately. The **Move** command does not remain active.

NOTE If you want to limit the number of objects shown in the preview, use the advanced option `XS_PREVIEW_LIMIT`. The default value is 1000. When the value is 0, the preview is off.

Move linearly

You can move objects linearly to a new position in the model.

1. Select the objects you want to move.
2. On the **Edit** tab, click **Move special** --> **Linear**.

   The **Move - Linear** dialog box opens.
3. Pick two points in the model, or enter the coordinates in the **dX**, **dY**, and **dZ** boxes.

   You can also use a formula to calculate the x, y, and z displacements. For example:

   dY =3*1250
4. Click **Move**.

TIP If the dialog box is open but the command is not active anymore, click the **Pick...** button to re-activate the command.

Move by specifying a distance from origin

You can move objects to a new position in the model or drawing by specifying a distance from the origin. Use the **Enter a Numeric Location** dialog box to specify the distance.

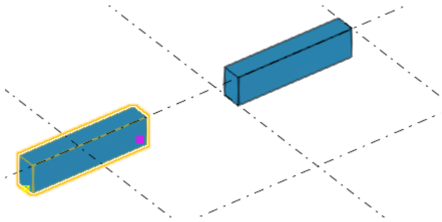
1. Select the objects you want to move.
2. Run the **Move** command:
   - In the model, on the **Edit** tab, click **Move**.
   - In the drawing, on the **Drawing** tab, click **Move** --> **Move**.
3. Pick the origin for moving.
4. Move the cursor in the direction you want to move the objects, but do not pick the point.
5. Type the distance.

   When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog box automatically.
6. Click **OK**.

Move using drag-and-drop

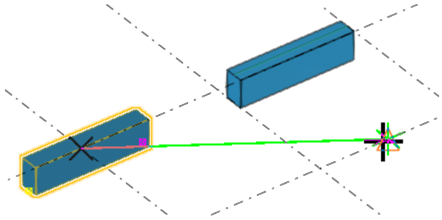
You can move objects by dragging them to a new location.

1. On the **File** menu, click **Settings** and select the **Drag & drop** check box to activate the command.
2. Select the objects you want to move.

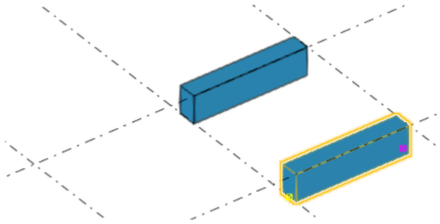


3. Drag the objects to a new location.

   The point you start dragging from (center, corner, or middle point) affects the object's alignment in the new location.

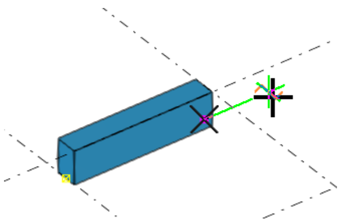
   The objects are moved immediately.

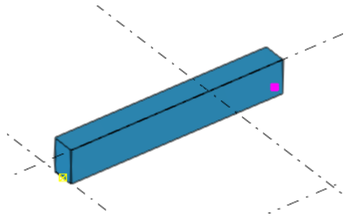
4. To move an end point using drag-and-drop:

   a. Select the handle.

   b. Hold down the left mouse button, and drag the handle to a new location.

      The end point is moved accordingly:



Note that for some objects you may need to switch on **Smart select** to drag from handles without selecting them first. To switch it on, click **File** --> **Settings** and select the **Smart select** check box.

NOTE To move grid labels in a drawing, first select the grid label and then either activate the  **Select grid line** selection switch or select the grid label handle.

Move to another plane

In the model, you can move objects from the first plane you specify to the second, third, and so on, plane you specify. The moved objects remain in the same position in the second plane, third, and so on, plane as the original objects in the first plane.

1. Select the objects you want to move.
2. On the **Edit** tab, click  **Move special** --> **To another plane**.
3. Pick the point of origin of the first plane.
4. Pick a point on the first plane in the positive x direction.
5. Pick a point on the first plane in the positive y direction.
6. Repeat steps 3–5 for the destination planes.

Move objects to another object

In a model, you can move objects from an object to other, similar objects. This is useful, for example, when you detail previously modeled parts. The objects that you move between can have different dimensions, length, and rotation.

If you move reinforcement or surface treatments and want them to adapt to the part they are moved to, note the limitations:

- The reinforcement or surface treatment handles must be in part corners.
- The parts between which you move must have the same number of cross section corners.
- Circular parts must have the same cross section dimensions.

1. Select the objects you want to move.

2. On the **Edit** tab, click **Move special** --> **To another object**.
3. Select the object to move from (source object).
4. Select the objects to move to (target object).

Rotate objects

You can copy or move an object in a model by rotating it around any line you choose. In a drawing, you can copy or move an object by rotating it around a given line on the work plane.

NOTE Positive rotation is according to the right-hand rule (page 53) (clockwise when looking from the start point of the rotation axis).

Rotate around a line

Use the **line** option in the **Rotate** dialog box when you want to copy and rotate, or move and rotate objects around any given line in the model.

1. Select the objects you want to copy or move.
2. Activate the rotation command.

    - To copy and rotate, go to the **Edit** tab and click **Copy special** --> **Rotate**.

      The **Copy - Rotate** dialog box opens.

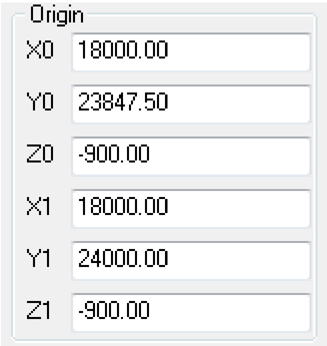
    - To move and rotate, go to the **Edit** tab and click **Move special** --> **Rotate**.

      The **Move - Rotate** dialog box opens.
3. In the **Around** list, select **line**.
4. Pick the start point of the rotation axis, or enter its coordinates.
5. Pick the end point of the rotation axis, or enter its coordinates.
6. If you are copying, enter the number of copies.
7. If needed, enter the **dZ** value, which is the difference in position between the original and copied object in the z direction.
8. Enter the rotation angle.
9. Click **Copy** or **Move**.

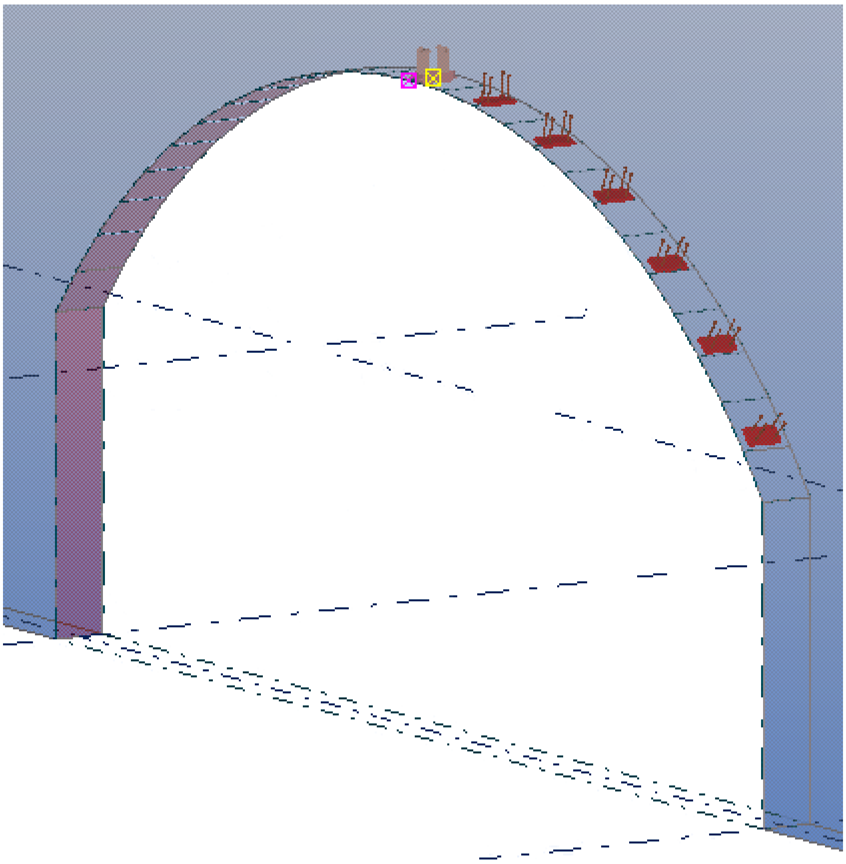
   The objects are rotated accordingly.

Example

In this example, a fitting plate is copied and rotated around a construction line that is located at the following coordinates.



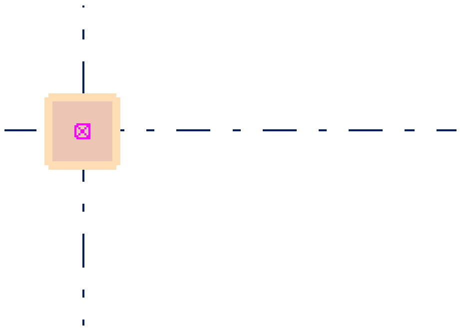
As a result, the copied fitting plates follow the curve of the concrete panel.



Rotate around the z axis

Use the **Z** option in the **Rotate** dialog box when you want to copy and rotate, or move and rotate objects around the z axis in the model.

1. Select the objects you want to copy or move. For example:



2. Activate the rotation command.

   - To copy and rotate, go to the **Edit** tab and click  **Copy special** --> **Rotate**.

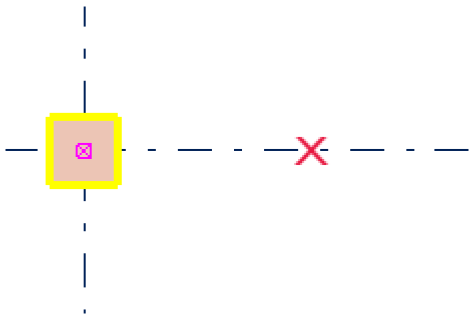
     The **Copy - Rotate** dialog box opens.

   - To move and rotate, go to the **Edit** tab and click  **Move special** --> **Rotate**.

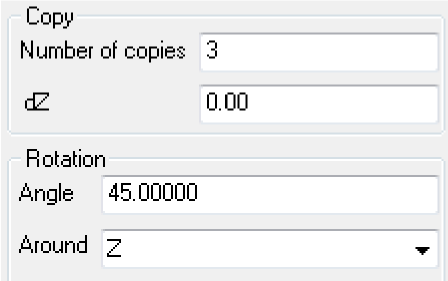
     The **Move - Rotate** dialog box opens.

3. Select **Z** in the **Around** list.
4. Pick a point to define the rotation axis, or enter its coordinates.

   In the example below, the red cross indicates the picked point.

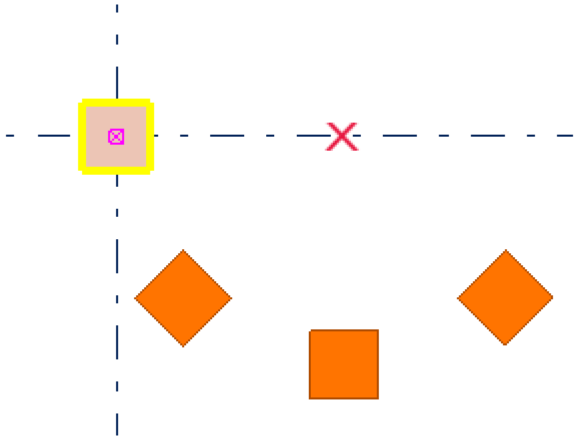


5. If you are copying, enter the number of copies.
6. If needed, enter the **dZ** value, which is the difference in position between the original and copied object in the z direction.
7. Enter the rotation angle. For example:



8. Click **Copy** or **Move**.

   The objects are rotated accordingly.



Rotate drawing objects

Use this option when you want to rotate drawing objects on the work plane.

1. Select the objects you want to copy or move.
2. Activate the rotation command.

   - To copy and rotate, go to the **Drawing** tab and click  **Copy** --> **Rotate**.

     The **Copy - Rotate** dialog box opens.

   - To move and rotate, go to the **Drawing** tab and click  **Move** --> **Rotate**.

     The **Move - Rotate** dialog box opens.

3. Pick a point, or enter its coordinates.
4. If you are copying, enter the number of copies.
5. Enter the rotation angle.
6. Click **Copy** or **Move**.

Rotation settings

Use the **Copy - Rotate** and **Move - Rotate** dialog boxes to view and modify the settings that are used when you rotate objects in Tekla Structures. The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals** .

Option	Description
X0	The x and y coordinates of the start point of the rotation axis.
Y0	
Origin angle	The angle of the rotation axis when rotating around a line on the work plane.
Number of copies	Define the number of copies created.
dZ	The difference in position between the original and copied object in the z direction.
Rotation angle	The rotation angle between the original and new position.
Around	Define whether the rotation axis is a **line** on the work plane or in the **z** direction.

Mirror objects

When you copy or move objects, you can mirror them through a plane that is perpendicular to the work plane and passes through a line you specify.

Note that Tekla Structures cannot create mirrored copies of object properties. For example, the **Copy special** --> **Mirror** command does not fully mirror objects if they include components with asymmetrically positioned parts, or reinforcement objects with asymmetric properties such as spacing.

Mirror model objects

Use this method to copy and mirror, or move and mirror objects in a model.

1. Select the objects you want to copy or move.
2. Activate the mirroring command.

   - To copy and mirror, go to the **Edit** tab and click  **Copy special** --> **Mirror**.

     The **Copy - Mirror** dialog box opens.

- To move and mirror, go to the **Edit** tab and click  **Move special** --> **Mirror**.

  The **Move - Mirror** dialog box opens.

3. Pick the start point of the mirroring plane, or enter its coordinates and angle.
4. Pick the end point of the mirroring plane, or enter its coordinates and angle.

   The first picked point (X0, Y0) is the origin, and the second picked point calculates the angle around the origin.
5. Click **Copy** or **Move**.

Mirror drawing objects

Use this method to copy and mirror, or move and mirror objects in a drawing.

1. Select the objects you want to copy or move.
2. Activate the desired mirroring command.
   - To copy and mirror, go to the **Drawing** tab and click **Copy** --> **Mirror**.

     The **Copy - Mirror** dialog box opens.
   - To move and mirror, go to the **Drawing** tab and click **Move** --> **Mirror**.

     The **Move - Mirror** dialog box opens.
3. Pick the start point of the mirroring plane, or enter its coordinates in the dialog box.
4. Pick the end point of the mirroring plane, or enter its coordinates in the dialog box.
5. Enter the angle in the dialog box.
6. Click **Copy** or **Move**.

1.8 Filter objects

Use filters to restrict what can be selected or what is visible in a view. You can create filters of your own, or you can use any of the standard filters available in Tekla Structures.

Here are some examples of what filters can be used for:

- **To select a large number of objects**

  Use selection filters when you need to change a specific object property that is common for many objects. The rest of the objects will not be affected, even if you try to include them in the selection.

- **To check the model**

  Use view filters to ensure that beams are called beams, columns are called columns, and so on. You can highlight several groups of objects, one at a time, to check that all required objects are included in a given group.

- **To hide objects**

  Use view filters to temporarily hide the columns in a view so that it easier to select all the beams, for example.

- **To find objects**

  You can create a selection filter to find all locations where ½” reinforcing bars are in the model, for example. Once the filter is active, you can make an area selection that includes the entire model. All specified reinforcing bars become selected, but the other objects will not be affected.

See also

Use existing filters (page 151)

Create new filters (page 153)

Filtering techniques (page 161)

Examples of filters (page 181)

Use existing filters

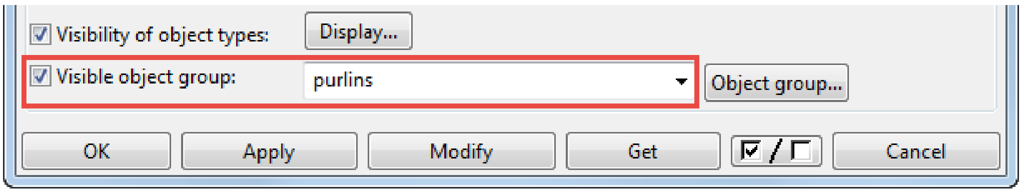
Before creating new custom filters, check out the existing view and selection filters available in Tekla Structures.

How to use a view filter

Use view filters to define which objects are displayed in a model view.

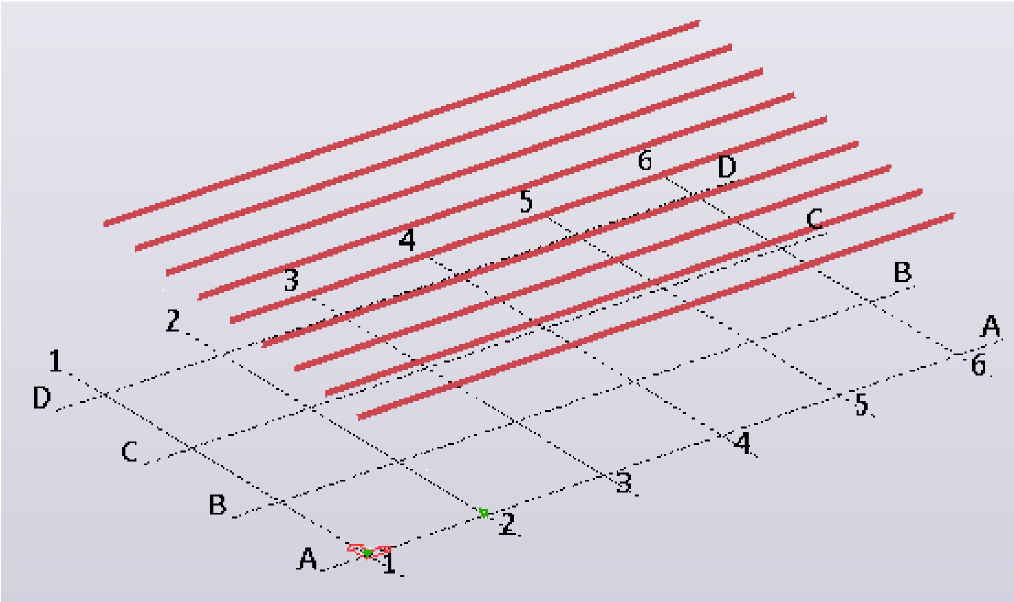
1. Double-click the view to open the **View Properties** dialog box.
2. Select a filter from the **Visible object group** list.

   For example, select **purlins**.



3. Click **Modify**.

Now only the objects defined by the filter are visible. For example, the purlins:



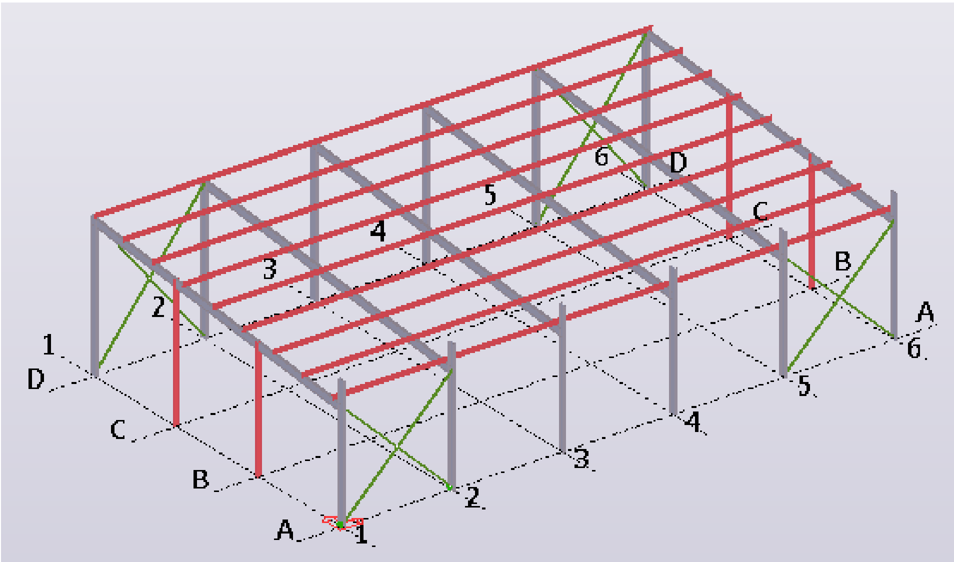
4. To stop using the filter:

   a. Double-click the view to open the **View Properties** dialog box.

   b. In the **Visible object group** list, select the **standard** filter.

   c. Click **Modify**.

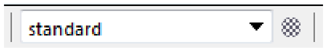
      All objects are visible again:



NOTE [If you cannot see all desired objects (page 50)](#), note that the work area, view depth, view setup, and object representation settings also affect the visibility of objects.

How to use a selection filter

Use selection filters to define which objects can be selected in the model. Note that an object must be visible in the model to be selectable.

1. On the **Selecting** toolbar, select a filter from the 

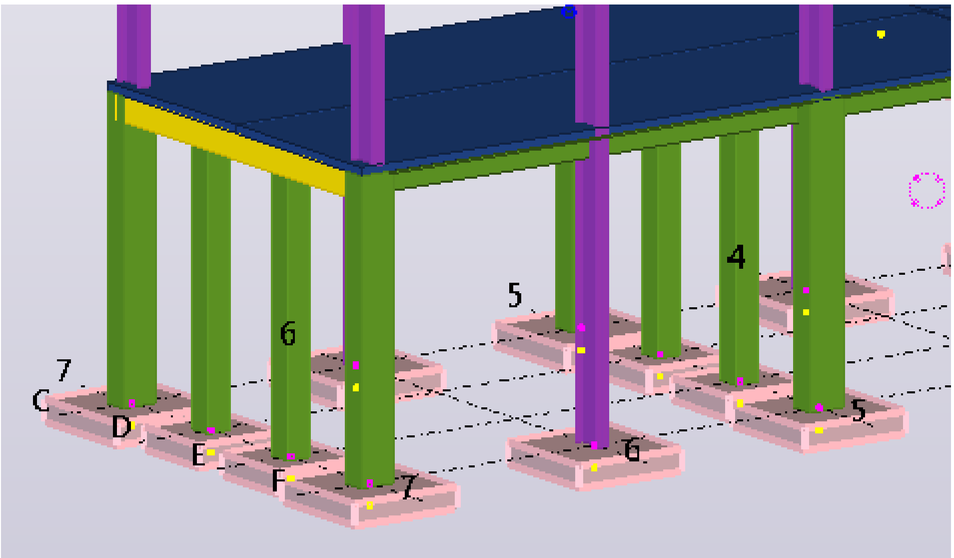
list.

   The list is by default located at the bottom of the Tekla Structures main window.

   For example, select the **Name - Footing** filter.

2. Select the desired objects in the model.

   You can select multiple objects or even the entire model at once. Now that the filter is active, only objects defined by the filter become selected. For example, if the **Name - Footing** filter is active, only footings are selectable and the rest of the objects stay intact:

3. If you cannot select all objects defined by the selection filter, check your view filter settings and ensure that you have switched on all the needed selection switches.

4. To stop using the filter, go to the **Selecting** toolbar and select the **standard** filter.

   All objects are selectable again.

Create new filters

You can create custom filters to define which objects are visible and selectable in the model and drawings. Add new filter rules, one on each row, to define which objects should be included or excluded.

Create a view filter

You can create your own custom filters to define which objects are visible in a model.

1. Double-click the view to open the **View Properties** dialog box.
2. Click **Object group**.

   The **Object Group - View Filter** dialog box opens, showing the currently active filter.
3. Click **New filter** to create a new filter from scratch.
4. Click **Add row** to add a new filter rule.
5. In the **Category** list, select an object category.

   You have the following options:

   - Part
   - Component
   - Bolt
   - Weld
   - Reinforcing bar
   - Surface
   - Assembly
   - Construction object
   - Load
   - Template
   - Reference assembly
   - Reference object
   - Location breakdown structure
   - Pour object
   - Pour break
   - Pour unit
   - Task
   - Object

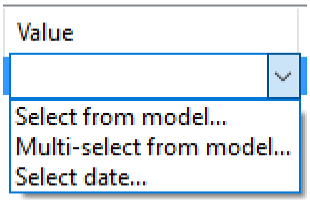
6. In the **Property** list, select a suitable object property (page 164).

   The options vary depending on the object category you chose in step 5.

7. In the **Condition** list, select a suitable condition (page 161).

8. In the **Value** list, type a value.

   Alternatively, to use the current value of an existing object, click **Select from model...** and select the desired object from the model. To use the values of multiple objects, click **Multi-select from model**, select the objects from the model, and then click the middle mouse button. For date values, also the **Select date...** option is available.

   The values can be complete strings, such as the profile name `UC310*97`. You can also use incomplete strings together with wildcards (page 181). For example, the value `UC*` will match with all parts whose profile name begins with the characters `UC*`. Empty values are matched to empty object properties.

   If you use multiple values, separate the strings with blank spaces (for example, `12 5`). If a value consists of multiple strings, enclose the entire value in quotation marks (for example, `"custom panel"`), or use a question mark (for example, `custom?panel`) to replace the space.

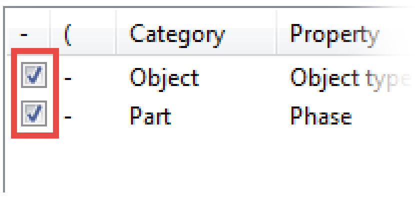
9. Repeat steps 4–8 to create as many filter rules as needed.

   You can apply several filter rules at the same time.

10. Use the **And/Or** options and brackets (page 161) to define how multiple rows work together.

11. Select the check boxes next to all filter rules that you want to enable.

    If the check box is selected, the filter rule is enabled and effective. For example:

    By default, each new rule is disabled.

12. Define the filter type.

    a. Click >> to display more settings.

b. Select or clear the check boxes to define where the filter will be used.

   For example, the same filter may be used both as a view filter and as a selection filter.

13. Enter a unique name in the box next to the Save as button.

NOTE
- Filters are case sensitive.
- Do not use spaces in filter names.
- We recommend that you use _ (underscores) in your naming convention.
- To have the filter appear at the top of the list, right after the standard filter, use capital letters in the filter name.

---

14. Click **Save as** to save the filter.
15. To apply the filter to the current view, click **Modify**.

Create a selection filter

You can create your own custom filters to help you select objects in a model.

1. On the **Selecting** toolbar, click to open the **Object Group - Selection Filter** dialog box.

2. Follow the instructions above on how to create a view filter.

   The same instructions apply to selection filters.

Create a drawing filter

For general arrangement drawings, you can create drawing filters that affect the whole drawing, not just a specific view. Drawing filters select objects in the whole drawing.

You can use drawing filters together with saved object property files when you create and apply object level settings in the whole drawing. For example, you might create a filter that selects all beams, then save an object property file that defines that the part color is blue, and then create and apply an object level settings file that changes all beams to blue in the whole drawing.

1. In a GA drawing, on the **Drawing** tab, click **Properties** --> **Drawing**.
2. Click **Filter**.
3. Click **New filter** to create a new filter from scratch.
4. Click **Add row** to add a new filter rule.
5. In the **Category** list, select an object category.

You have the following options:

- Part
- Component
- Bolt
- Weld
- Reinforcing bar
- Surface
- Assembly
- Construction object
- Template
- Reference assembly
- Reference object
- Location breakdown structure
- Pour object
- Pour break
- Pour unit
- Task
- Object

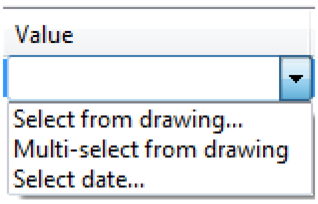
6. In the **Property** list, select a suitable object property (page 164).

   The options vary depending on the object category you chose in step 5.

7. In the **Condition** list, select a suitable condition (page 161).

8. In the **Value** list, type a value.

   Alternatively, to use the current value of an existing object, click **Select from drawing** and select the desired object from the drawing. To use the values of multiple objects, click **Multi-select from drawing**, select the objects from the drawing, and then click the middle mouse button. For date values, also the **Select date...** option is available.

   The values can be complete strings, such as the profile name `UC310*97`. You can also use incomplete strings together with wildcards (page 181). For example, the value `UC*` will match with all parts whose profile name begins with the characters `UC*`. Empty values are matched to empty object properties.

If you use multiple values, separate the strings with blank spaces (for example, `12 5`). If a value consists of multiple strings, enclose the entire value in quotation marks (for example, `"custom panel"`), or use a question mark (for example, `custom?panel`) to replace the space.

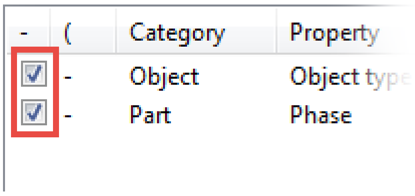
9. Repeat steps 4–8 to create as many filter rules as needed.

   You can apply several filter rules at the same time.

10. Use the **And/Or** options and brackets (page 161) to define how multiple rows work together.

11. Select the check boxes next to all filter rules that you want to enable.

    If the check box is selected, the filter rule is enabled and effective. For example:

    By default, each new rule is disabled.

12. Define the filter type.

    a. Click >> to display more settings.

    b. Select or clear the check boxes to define where the filter will be used.

       For example, the same drawing filter may be used both as a model view filter and as a model selection filter, and also as an Organizer filter.

13. Enter a unique name in the box next to the **Save as** button.

NOTE
- Filters are case sensitive.
- Do not use spaces in filter names.
- We recommend that you use _ (underscores) in your naming convention.
- To have the filter appear at the top of the list, right after the standard filter, use capital letters in the filter name.

---

14. Click **Save as** to save the filter.

15. When you are done, click **Cancel** to close the filter properties dialog box.

Create a drawing view filter

You can create your own custom view filters to help you select a specific group of view objects in a drawing view.

You can use drawing view filters for changing the appearance of a certain object group, or for selecting which objects are hidden in a drawing view.

You can also use drawing view filters together with saved object property files when you create and apply object level settings in the selected view. For example, you might create a view filter that selects all columns in a view, then save an object property file that defines that the part color is red, and then create and apply an object level settings file that changes all columns to red in the selected view.

1. Open a drawing.
2. Double-click a drawing view frame.
3. Click **Filter**.
4. Click **New filter** to create a new filter from scratch.
5. Click **Add row** to add a new filter rule.
6. In the **Category** list, select an object category.

   You have the following options:

   - Part
   - Component
   - Bolt
   - Weld
   - Reinforcing bar
   - Surface
   - Assembly
   - Construction object
   - Template
   - Reference assembly
   - Reference object
   - Location breakdown structure
   - Pour object
   - Pour break
   - Pour unit
   - Task
   - Object

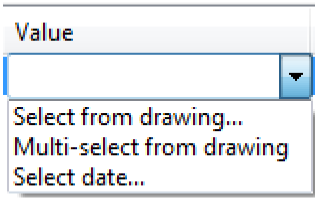
7. In the **Property** list, select a suitable object property (page 164).

   The options vary depending on the object category you chose in step 5.

8. In the **Condition** list, select a suitable condition (page 161).

9. In the **Value** list, type a value.

   Alternatively, to use the current value of an existing object, click **Select from drawing** and select the desired object from the drawing. To use the values of multiple objects, click **Multi-select from drawing**, select the objects from the drawing, and then click the middle mouse button. For date values, also the **Select date...** option is available.

   The values can be complete strings, such as the profile name `UC310*97`. You can also use incomplete strings together with wildcards (page 181). For example, the value `UC*` will match with all parts whose profile name begins with the characters `UC*`. Empty values are matched to empty object properties.

   If you use multiple values, separate the strings with blank spaces (for example, `12 5`). If a value consists of multiple strings, enclose the entire value in quotation marks (for example, `"custom panel"`), or use a question mark (for example, `custom?panel`) to replace the space.

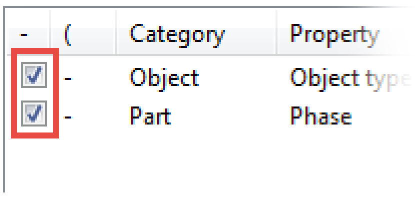
10. Repeat steps 4–8 to create as many filter rules as needed.

    You can apply several filter rules at the same time.

11. Use the **And/Or** options and brackets (page 161) to define how multiple rows work together.

12. Select the check boxes next to all filter rules that you want to enable.

    If the check box is selected, the filter rule is enabled and effective. For example:

    By default, each new rule is disabled.

13. Define the filter type.

    a. Click  to display more settings.

b. Select or clear the check boxes to define where the filter will be used.

   For example, the same drawing view filter may be used both as a model view filter and as a model selection filter, and also as an Organizer filter.

14. Enter a unique name in the box next to the **Save as** button.

NOTE
- Filters are case sensitive.
- Do not use spaces in filter names.
- We recommend that you use _ (underscores) in your naming convention.
- To have the filter appear at the top of the list, right after the standard filter, use capital letters in the filter name.

---

15. When you are done, click **Cancel** to close the filter properties dialog box.

Create a drawing selection filter

You can create your own custom filters to help you select objects in a drawing.

You can use selection filters in drawings if you want to hide certain parts from drawings or drawing views, or change the part color or representation for certain parts.

Also, if you have some different looking part marks for different types of parts, you can select the specific parts using a selection filter and then only modify part marks for those parts.

1. In an open drawing, on the **Selecting** toolbar, click  (**Ctrl+G**).

   The **Selection filter** dialog box opens.

2. Follow the instructions above on how to create a drawing or drawing view filter.

   The same instructions apply to drawing selection filters.

3. Click **Apply** or **OK** to select the parts according to the filter.

Filtering techniques

By using conditions, brackets, and the **And/Or** options you can create filters that can be as complex as needed.

Conditions

Use conditions to define how filter criteria relate to one another. Note that when you create filters (page 153), you are always defining what should be **shown** (or be selectable) in the model or drawing. Therefore, if you enter "Component name does not contain `gusset`", you are telling Tekla Structures to show all components whose name **does not contain** the word "gusset". Tekla Structures then hides all components which have the word "gusset" in their name.

Condition	Description
Equals	Use this condition when the filter value must be matched exactly. For example, "Part name equals `BEAM`."
Does not equal	Filters out objects that contain the value you enter. For example, "Part profile does not equal `BL200*20`" means that Tekla Structures will hide (or not select) the objects whose profile is BL200*20. The rest of the objects will be shown (or selected).
Begins with	Finds all objects that start with the value you enter. For example, "Component name begins with `purlin`."
Does not begin with	Filters out objects that start with the value you enter. For example, "Component name does not begin with `shear`" means that Tekla Structures will hide (or not select) the objects whose name begins with the word "shear". The rest of the objects will be shown (or selected).
Ends with	Finds all objects that end with the value you enter. For example, "Component name ends with `plate`."
Does not end with	Filters out objects that end with the value you enter. For example, "Component name does not end with `angle`" means that Tekla Structures will hide (or not select) the objects whose name ends with the word "angle". The rest of the objects will be shown or selected.
Contains	Finds all objects that include the value you enter. For example, "Component name contains `plate`" would find `base plate` and `shear plate simple`.
Does not contain	Filters out objects that include the value you enter. For example, "Component name does not contain

Condition	Description
	gusset" means that Tekla Structures will hide (or not select) the objects whose name contains the word "gusset". The rest of the objects will be shown or selected.
Greater than	Finds all objects that exceed the value you enter. For example, "Template attribute LENGTH is greater than 5000." This property can only be used with numerical data, such as the part start number, class, phase, or LENGTH.
Greater or equal	Finds all objects that match or exceed the value you enter. This property can only be used with numerical data, such as the part start number, class, phase, or LENGTH.
Less than	Finds all objects that are less than the value you enter. This property can only be used with numerical data, such as the part start number, class, phase, or LENGTH.
Less or equal	Finds all objects that match or are less than the value you enter. This property can only be used with numerical data, such as the part start number, class, phase, or LENGTH.
Later than	Available for dates only. The date must be later than the one you have defined. For example, "Object's approval date later than 4/10/2017."
Later than or equal	Available for dates only. The date must be later than or equal to the one you have defined.
Earlier than	Available for dates only. The date must be earlier than the one you have defined. For example, "Object's approval date earlier than 2/18/2017."
Earlier than or equal	Available for dates only. The date must be earlier than or equal to the one you have defined.

And/Or options

Use the **And/Or** options when you create filter rules that consist of several rows.

Option	Description
And	Use to find objects that match both values. When you create filter rules between objects whose **Category** settings differ, use the **And** option when possible to avoid potential problems with more complex rules.
Or	Use to find objects that match either value.
empty (= And)	Empty means the same as **And**.

Brackets

You can use single, double, and triple brackets to create more complex filter rules.

Example 1. Use the format "(A and (B or C)" to find objects that match the first filter rule and **either** of the last two rules.

-	(	Category	Property	Condition	Value	)	And/Or
☑	-	Part	Name	Equals	BRACING	-	And
☑	(	Part	Phase	Equals	1	-	Or
☑	-	Part	Phase	Equals	3	)	Or

Example 2. Use the format "(A and B) or C" to find objects that match both of the first two rules **or** the third.

-	(	Category	Property	Condition	Value	)	And/Or
☑	(	Part	Name	Equals	COLUMN	-	And
☑	-	Part	Profile	Equals	IPE300	)	Or
☑	-	Part	Material	Equals	S235JR	-	Or

Object properties in filtering

You can select from a wide variety of object properties when creating new filters. In the tables below, the properties are listed according to their object category. In addition to these, almost all categories contain user-defined attributes and template attributes that can also be used in filtering.

Category: Object

Use the **Object** category to filter objects based on their object level properties.

Property	Description
GUID	To filter objects based on their globally unique identifier (GUID). For example, "Object GUID begins with `ID7554C9EB-C8B4`."
Phase	To filter objects based on their phase number. For example, "Object phase does not equal `3`."
Object type	To filter objects based on their type. Select an object type from the **Value** list, or use the **Select from model...** or **Multi-select from model** option. NOTE: We recommend that you include one filter rule for the **Object type** property in each filter you create. This ensures that only objects of this type are filtered in. If you leave the object type out, the filtering outcome will be different, and objects that do not match the category in further filter rules may be filtered in. The following object types can be selected from the list: • Assembly • Bolt group • Connection • Part • Pour break • Pour object • Pour unit • Reference object • Reinforcing bar • Surface • Surface treatment • Weld The following object types are shown as numeric values only:

Property	Description
	• 1 = point • 9 = fitting • 11 = polygon cut • 12 = line cut • 24 = construction line • 30 = construction plane • 38 = added material • 42 = construction circle • 43 = construction arc • 48 = reference model • 76 = analysis part • 79 = edge chamfer • 103 = construction polycurve
Is component	To filter objects based on whether or not they are components. The options are **Yes** and **No**. For example, "Object is component equals `Yes`."

Some object types are not visible directly, but only when the objects that make them up are visible. For example, assemblies are visible when parts are visible, and pour units are visible when pour objects are visible. Therefore, using the object type **Assembly** or **Pour unit** alone in a view filter in the model or in drawings does not show anything. However, selection filters are able to select object types like assemblies and pour units.

Category: Part

Use the **Part** category to filter parts (page 196) based on their common properties.

Property	Description
Name	To filter objects based on their name. For example, "Part name equals `SLAB`."
Profile	To filter objects based on their profile. For example, "Part profile does not equal `L20*2`."
Material	To filter objects based on their material grade. For example, "Part material equals `C25/30`."

Property	Description
Finish	To filter objects based on how the part surface has been treated. For example, "Part finish equals "`FP - Fire proofing`"."
Prefix	To filter objects based on their numbering prefix. For example, "Part prefix equals `P`."
Start number	To filter objects based on their start number. For example, "Part start number greater than `100`."
Numbering series	To filter objects based on their numbering series information. For example, "Part numbering series equals `TP/1`." Note that position number separator can be a dot (.), comma (,), slash (/), or hyphen (-), depending on what you have defined in **File menu** --> **Settings** --> **Options** --> **Numbering**.
Position number	To filter objects based on their position number. For example, "Part position number does not equal `P/5`." Note that position number separator can be a dot (.), comma (,), slash (/), or hyphen (-), depending on what you have defined in **File menu** --> **Settings** --> **Options** --> **Numbering**.
Class	To filter objects based on their class number. For example, "Part class equals `210`."
Phase	To filter objects based on their phase number. For example, "Part phase equals `1 2`."
Lot	To filter objects based on their lot number. For example, "Part lot greater than `1`."
Primary part	To filter objects based on whether they are main or secondary parts in an assembly or cast unit. **1** = primary part, **0** = secondary part. For example, "Part primary part equals `1`."

Property	Description
Pour phase	To filter parts based on their pour phase. For example, "Pour phase does not equal `0`."

Category: Component

Use the **Component** category to filter components (page 759) based on their common properties.

Property	Description
Name	To filter components based on their name. For example, "Component name equals "`shear plate simple`"."
Connection code	To filter components based on their connection code, which can be either a text string or a number. For example, "Component connection code equals `200_2`."
Running number	To filter components based on their unique running number. For example, "Component running number less than `150`."
Phase	To filter components based on their phase number. For example, "Component phase equals `2`."
Is conceptual	To filter components based on their type. Components can be either detailed or conceptual. **Yes** = conceptual, **No** = detailed. For example, "Component is conceptual equals `Yes`."

Category: Bolt

Use the **Bolt** category to filter bolts (page 341) based on their common properties.

Property	Description
Size	To filter bolts based on their diameter. For example, "Bolt size less than `20.00`."
Standard	To filter bolts based on their bolt assembly standard/grade. For example, "Bolt standard equals `7990`."

Property	Description
Site/Workshop	To filter bolts based on their assembly type. **Site** = 0, **Workshop** = 1. For example, "Bolt Site/Workshop equals `1`."
Phase	To filter bolts based on their phase number. For example, "Bolt phase equals `3 4`."
Length	To filter bolts based on their length. For example, "Bolt length greater than `50.00`."
Hole 1 type ... **Hole 5 type**	To filter bolts based on their hole types in the bolted parts 1...5. The options are **Slotted**, **Oversized**, **Tapped**, **No hole**, and **Regular**.

Category: Weld

Use the **Weld** property to filter welds (page 359) based on their common properties.

Property	Description
Size above line **Size below line**	To filter welds based on their size. For example, "Weld size above line equals `5.00`."
Reference text	To filter welds based on their reference text, which is a user-definable value in the **Weld** properties. For example, "Weld reference text contains `12345`."
Phase	To filter welds based on their phase number. For example, "Weld phase equals `3`."
Type above line **Type below line**	To filter welds based on their weld type (page 367). Select the type from the **Value** list.
Length above line **Length below line**	To filter welds based on their length value. For example, "Weld length greater than `0.00`."
Welding site	To filter welds based on where they should be made. The options are **Site** and **Workshop**.
Position number	To filter welds based on their unique position number. For example, "Weld position number is greater than `100`."

Property	Description
Angle above line **Angle below line**	To filter welds based on the angle of weld preparation, bevels, or groove. For example, "Weld angle below line greater than `0.000`."
Contour above line **Contour below line**	To filter welds based on their fill type contour. The options are **None**, **Flush**, **Convex**, and **Concave**. For example, "Weld contour above line does not equal `None`."
Effective throat above line **Effective throat below line**	To filter welds based on their weld size used in weld strength calculation. For example, "Weld effective throat above line equals `0.500`."
Finish above line **Finish below line**	To filter welds based on how they have been treated. The options are **None**, **Grind**, **Machine**, **Chip**, **Finished weld**, and **Smooth transition**.
Increment amount above line **Increment amount below line**	To filter welds based on their amount of increments. For example, "Weld increment amount above line greater than `0`."
Intermittent type	To filter welds based on their shape. The options are **Continuous**, **Intermittent**, and **Staggered intermittent**.
Pitch above line **Pitch below line**	To filter welds based on their spacing of weld increments.
Root face thickness above line **Root face thickness below line**	To filter welds based on their root face thickness, in other words, the height of the narrowest part inside the root opening.
Root opening above line **Root opening below line**	To filter welds based on the space found between the welded parts.
Size prefix above line **Size prefix below line**	To filter welds based on their weld size prefix. For example, "Weld size prefix above line equals `a`." The standard ISO 2553 prefixes are `a` (Design throat thickness), `s` (Penetration throat thickness), and `z` (Leg length).

Property	Description
User defined cross-section	To filter welds based on whether or not they contain user-defined cross-sections. The options are **Yes** and **No**.
Electrode classification	To filter welds based on their weld electrode classification. The options are (empty), **35**, **52**, **50**, **E60XX**, **E70XX**, **E80XX**, and **E90XX**.
Electrode strength	To filter welds based on their electrode strength. For example, "Weld electrode strength greater than `0.000`."
Electrode coefficient	To filter welds based on their electrode coefficient.
Process type	To filter welds based on their welding process type. The options are **SMAW**, **SAW**, **GMAW**, **FCAW**, **ESW**, and **EGW**.
NDT inspection	To filter welds based on their non-destructive testing and inspection level. The options are **A**, **B**, **C**, **D**, and **E**.
Is around weld	To filter welds based on whether only one edge or the entire perimeter of a face is welded. **No** = edge, **Yes** = around.

Category: Reinforcing bar

Use the **Reinforcing bar** category to filter reinforcing bars (page 496) based on their common properties.

Property	Description
Name	To filter reinforcing bars based on their name. For example, "Reinforcing bar name equals `"STIRRUP"`."
Class	To filter reinforcing bars based on their class number. For example, "Reinforcing bar class equals `3`."
Size	To filter reinforcing bars based on their size. The size property depends on the environment and may contain letters and special characters. For example, in the US imperial environment, "Reinforcing bar size equals `#18`."

Property	Description
Diameter	To filter reinforcing bars based on their diameter. Diameter is the nominal diameter of the bar, not actual. For example, "Reinforcing bar diameter less than 12."
Length	To filter reinforcing bars based on their total length. For example, "Reinforcing bar length greater than 5000.00."
Material	To filter reinforcing bars based on their material grade. For example, "Reinforcing bar material does not equal Undefined."
Prefix	To filter reinforcing bars based on their numbering prefix. For example, "Reinforcing bar prefix equals R."
Start number	To filter reinforcing bars based on their start number. For example, "Reinforcing bar start number greater than 1."
Numbering series	To filter reinforcing bars based on their numbering series information. For example, "Reinforcing bar numbering series equals R/1."
Position number	To filter reinforcing bars based on their position number. For example, "Reinforcing bar position number equals R/3."
Phase	To filter reinforcing bars based on their phase number. For example, "Reinforcing bar phase equals 2."
Shape	To filter reinforcing bars based on their bending shape (page 588). For example, "Reinforcing bar shape does not equal 2_1."
Layer	To filter rebar set bars based on their bar layer information. For example, "Rebar set bar layer equals B3" (the third layer of bottom bars).

Category: Surface

Use the **Surface** category to filter surfaces (page 415) based on their common properties.

Property	Description
Name	To filter surfaces based on their name. For example, "Surface name equals `SURFACE`."
Type	To filter surfaces based on their type. The options are **Formwork** and **Concrete finish**.
Class	To filter surfaces based on their class number. For example, "Surface class does not equal `13`."
Phase	To filter surfaces based on their phase number. For example, "Surface phase equals `3 4`."

Category: Assembly

Use the **Assembly** category to filter assemblies, cast units (page 430), and rebar assemblies (page 614) based on their common properties.

Property	Description
Name	To filter assemblies and cast units based on their name. For example, "Assembly name does not contain `RAFTER`."
GUID	To filter assemblies based on their globally unique identifier (GUID). For example, "Assembly GUID equals `ID89F414A7-ECA6-4B14-99CB-6985B84E64CB`."
Prefix	To filter assemblies and cast units based on their numbering prefix. For example, "Assembly prefix equals `A`."
Start number	To filter assemblies and cast units based on their start number. For example, "Assembly start number greater than `1`."
Position number	To filter assemblies and cast units based on their position number. For example, "Assembly position number equals `A/13`."
Phase	To filter assemblies and cast units based on their phase number. For

Property	Description
	example, "Assembly phase does not equal `1`."
Assembly level	To filter assemblies and cast units based on their position in the assembly hierarchy. The larger the value, the lower the position in the assembly hierarchy. 0 is the top-most level, and 1 is the first sub-assembly level. For example, to check if the model contains sub-assemblies, use the filter rule "Assembly level greater than or equal to `1`."
Assembly type	To filter assemblies and cast units based on their type. • 0 = precast • 1 = cast in place • 2 = steel • 3 = timber • 4 = rebar • 6 = miscellaneous
Assembly series	To filter assemblies and cast units based on their numbering series information. For example, "Assembly series equals `C/1`."

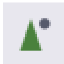
Category: Construction object

Use the **Construction object** category to filter construction objects based on their common properties.

Property	Description
Phase	To filter construction objects based on their phase number. For example, "Construction object phase does not equal `1`."
Type	To filter construction objects based on their type. The options are **Line**, **Arc**, **Circle**, **Plane**, and **Polycurve**.

Category: Load

Use the **Load** category to filter loads based on their common properties.

Property	Description
Load group	To filter loads based on which load group they belong to. For example, "Load group does not equal `DefaultGroup`."
Load type	To filter loads based on their type. The options are **line**, **point**, **area**, **uniform**, and **temperature**. Note that wind loads are handled as area loads in filtering. Use the  **Select components** and  **Select objects in components** selection switches to select wind loads.
Phase	To filter loads based on their phase number. For example, "Load phase does not equal `1`."

Category: Template

Use the **Template** category to filter parts and other objects by using template attributes.

With this category, you can type the name of any template attribute or user-defined attribute directly in the **Property** box, even if it is not on the list. Use the `ASSEMBLY.`, `CAST_UNIT.`, or `POUR_UNIT.` prefix in front of the property name to access higher hierarchy level attributes, and the `USERDEFINED.` prefix to access user-defined attributes.

For example, to filter objects that are one hierarchy level below the cast-unit with the user-defined attribute **User field 1**, type `CAST_UNIT.USERDEFINED.USER_FIELD_1` in the **Property** box.

In some cases, you can filter objects based on properties of other objects from lower hierarchy levels. This is possible when there is only one lower level object to be accessed. For example, there is only one main part in each assembly or cast unit, so you can access the main part properties from the assembly or cast unit level using the `MAINPART` prefix. Similarly, there can only be one pour object in each pour unit, so you can access the pour object properties from the pour unit level using the `POUR_OBJECT` prefix.

For example, to filter objects in assemblies whose main part has a certain name, type `ASSEMBLY.MAINPART.NAME` in the **Property** box.

For example, to filter all reinforcing bars belonging to pour units with a certain pour object type, type `POUR_UNIT.POUR_OBJECT.POUR_TYPE` in the **Property** box.

TIP To check which unit Tekla Structures uses for a particular template attribute, use the **Select from model...** option in the **Value** list in the filtering dialog box.

Category: Reference assembly

Use the **Reference assembly** category to filter reference model assemblies based on their common properties.

Property	Description
Creation	
GUID	To filter reference model assemblies based on their globally unique identifier (GUID). For example, "Reference assembly GUID equals `IDA51E6BFF-DAB9-4A56-970C-7486EF17B7B7`."
Phase	To filter reference model assemblies based on their phase number. For example, "Reference assembly phase equals `2`."
Lot	To filter reference model assemblies based on their lot number. For example, "Reference assembly lot greater than `1`."
Description	To filter reference model assemblies based on their description, which is a user-definable value in the **Reference object** dialog box. For example, "Reference assembly description contains `"architect model"`."
Info text	To filter reference model assemblies based on their info text, which is a user-definable value in the **Reference object** dialog box. For example, "Reference assembly info text contains `revised`."
Locked	To filter reference model assemblies based on whether or not they are locked. **0** = No, **1** = Yes, **2** = Organization.
Logical name	To filter reference model assemblies based on their logical name, which is

Property	Description
	a user-definable value in the **Reference object** dialog box. For example, "Reference assembly logical name equals "`MEP heating system`"."

Category: Reference object

Use the **Reference object** category to filter reference model objects based on their common properties.

Property	Description
Creation	
GUID	To filter reference model objects based on their globally unique identifier (GUID).
Phase	To filter reference model objects based on their phase number. For example, "Reference object phase does not equal `1`."
Lot	To filter reference model objects based on their lot number. For example, "Reference object lot equals `1`."
Description	To filter reference model objects based on their description, which is a user-definable value in the **Reference object** dialog box. For example, "Reference object description contains "`architect model`"."
Info text	To filter reference model objects based on their info text, which is a user-definable value in the **Reference object** dialog box. For example, "Reference object info text contains `revised`."
Locked	To filter reference model objects based on whether or not they are locked. **0** = No, **1** = Yes, **2** = Organization.
Logical name	To filter reference model objects based on their logical name, which is a user-definable value in the **Reference object** dialog box. For

Property	Description
	example, "Reference object logical name contains "3rd floor"."

TIP You can filter reference model object attributes using the **Template** category and the EXTERNAL. prefix in the **Property** box. For example, "Reference object EXTERNAL.Material equals A572."

Category: Location breakdown structure

Use the **Location breakdown structure** category to filter objects based on their location categories, which can be defined in **Organizer**.

Property	Description
Site	To filter objects based on which site category they belong to. For example, "Location breakdown structure site equals "Site 2"."
Building	To filter objects based on which building category they belong to. For example, "Location breakdown structure building does not equal "Building A"."
Section	To filter objects based on which section category they belong to. For example, "Location breakdown structure section equals Ramp."
Floor	To filter objects based on which floor they are located on. For example, "Location breakdown structure floor equals "Floor 4"."

Category: Pour object

Use the **Pour object** category to filter pour objects (page 451) based on their common properties.

Property	Description
Pour number	To filter pour objects based on their pour number. For example, "Pour number equals 5."
Pour type	To filter pour objects based on their type. For example, "Pour type equals WALL."
Concrete mixture	To filter pour objects based on the properties of their concrete mix, for example, maximum grain size of

Property	Description
	aggregate, and/or plasticity of fresh concrete.
Material	To filter pour objects based on their material grade. For example, "Material equals `C35/45` ."
Pour phase	To filter pour objects based on their pour phase. For example, "Pour phase does not equal `0`."

Category: Pour break

Use the **Pour break** category to filter pour breaks (page 458) based on their common properties.

Property	Description
Creation	
ID number	To filter pour breaks based on their ID number. For example, "ID number equals `25237`."
Phase	To filter pour breaks based on their phase. For example, "Pour break phase equals `2 3`."
Pour break type	To filter pour breaks based on their type. For example, "Pour break type equals `"Watertight construction joint"`."

Category: Pour unit

Use the **Pour unit** category to filter pour units based on their common properties.

Property	Description
Name	To filter pour units based on their name. For example, "Pour unit name contains `beam`."
GUID	To filter pour units based on their globally unique identifier (GUID). For example, "Pour unit GUID contains `8505`."

Category: Task

Use the **Task** category to filter scheduled tasks based on their common properties.

Property	Description
Name	To filter scheduled tasks based on their name. For example, "Task name contains `floors`."
Planned start date	To filter scheduled tasks based on their planned start date. For example, "Task planned start date earlier than `Review date`."
Planned end date	To filter scheduled tasks based on their planned start date. For example, "Task planned end date later than or equal to `10/13/2017`."
Actual start date	To filter scheduled tasks based on their actual start date.
Actual end date	To filter scheduled tasks based on their actual end date.
Completeness	To filter scheduled tasks based on their completeness. The value is a percentage. For example, "Task completeness is `75`."
Critical	To filter scheduled tasks based on how critical they are. A task can be critical only if it has been imported from external software. **1** = Critical, **0** = Not critical. Note that this property is not visible in **Task manager**.
Local	To filter scheduled tasks based on whether they were created in **Task manager** or imported from external software. **1** = Created in Task Manager, **0** = Imported.
Contractor	To filter scheduled tasks based on the contractor. For example, "Task contractor equals `"Contractor A"`."
Scenario	To filter scheduled tasks based on the scenario they belong to. For example, "Task scenario equals `"Scenario 1"`."

Property	Description
Task type	To filter scheduled tasks based on their type. For example, "Task type does not equal "`A - Floor tiling`"."

Wildcards

A wildcard is a symbol that stands for one or more characters. You can use wildcards to shorten strings, for example in filtering.

Wildcard	Description	Example
* (asterisk)	Matches any number of characters	`HE*` matches all parts with a profile name that begins with the characters "HE". You can also this symbol at the beginning of a word: `*BRAC*`.
? (question mark)	Matches a single character	`HE?400` matches parts with profile names such as HEA400, HEB400, and HEC400.
[ ] (square brackets)	Matches whatever is enclosed in the brackets	`L[78]X4X1/2` matches parts with the profile names L7X4X1/2 and L8X4X1/2.

NOTE The characters * and ? may also be used in object names in Tekla Structures. If the object name you want to filter contains * or ?, you need to enclose these characters in square brackets. For example, to find the profile P100*10, enter `P100[*]10` in the filter field.

See also

Filter objects (page 150)

Examples of filters

Here are some examples of filters that you can create. The same filtering techniques can be used for view, selection, and drawing filters.

Filter parts based on their name

Create a filter that only shows parts with a certain name.

1. Create a new view filter. (page 153)
2. Click **Add row** three times to add three filter rules.
3. In the first filter rule, define that the object type must be part:

a. In the **Category** list, select **Object**.

b. In the **Property** list, select **Object type**.

c. In the **Condition** list, select **Equals**.

d. In the **Value** list, select **Part**.

e. In the **And/Or** list, select **And**.

4. In the second and third filter rules, define that the part name must be BEAM or COLUMN:

   a. In the **Category** list, select **Part**.

   b. In the **Property** list, select **Name**.

   c. In the **Condition** list, select **Equals**.

   d. In the **Value** box, enter the part names, `BEAM` and `COLUMN`.

   e. In the **And/Or** list, select **Or**.

5. Include the second and third filter rule in brackets. The filter is now looking for parts that are named either BEAM or COLUMN.

6. Enter a unique name in the box next to the **Save as** button.

7. Click **Save as**.

-	(	Category	Property	Condition	Value	)	And/Or
☑	-	Object	Object type	Equals	Part	-	And
☑	(	Part	Name	Equals	COLUMN	-	Or
☑	-	Part	Name	Equals	BEAM	)	

Filter main parts

Create a filter that selects the main parts only.

1. Create a new selection filter. (page 153)

2. Click **Add row** twice to add two filter rules.

3. In the first filter rule, define that the object type must be part:

   a. In the **Category** list, select **Object**.

   b. In the **Property** list, select **Object type**.

   c. In the **Condition** list, select **Equals**.

   d. In the **Value** list, select **Part**.

   e. In the **And/Or** list, select **And**.

4. In the second filter rule, define that you only want to include main parts:

   a. In the **Category** list, select **Part**.

   b. In the **Property** list, select **Primary part**.
   c. In the **Condition** list, select **Equals**.
   d. In the **Value** box, enter `1`.

      In this context, `1` means the main parts, and `0` would mean the secondary parts.
5. Enter a unique name in the box next to the **Save as** button.
6. Click **Save as**.

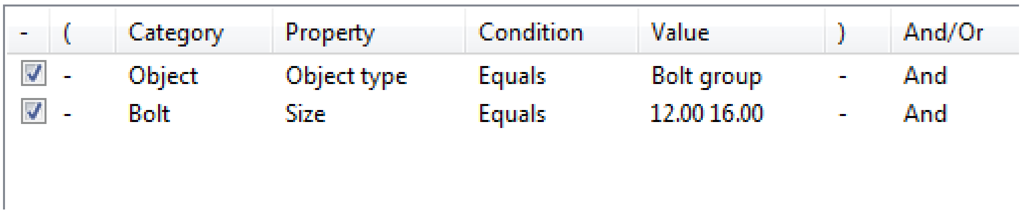
-	(	Category	Property	Condition	Value	)	And/Or
☑	-	Object	Object type	Equals	Part	-	And
☑	-	Part	Primary part	Equals	1	-	And

Filter bolts based on their size

Create a filter that only shows bolts of certain sizes.

1. Create a new view filter. (page 153)
2. Click **Add row** twice to add two filter rules.
3. In the first filter rule, define that the object type must be bolt:
   a. In the **Category** list, select **Object**.
   b. In the **Property** list, select **Object type**.
   c. In the **Condition** list, select **Equals**.
   d. In the **Value** list, select **Bolt group**.
   e. In the **And/Or** list, select **And**.
4. In the second filter rule, define that the bolt size must be 12.00 or 16.00:
   a. In the **Category** list, select **Bolt**.
   b. In the **Property** list, select **Size**.
   c. In the **Condition** list, select **Equals**.
   d. In the **Value** box, enter the bolt sizes, `12.00` and `16.00`.

      Separate the strings with a blank space.
5. Enter a unique name in the box next to the **Save as** button.
6. Click **Save as**.



Filter parts based on their assembly type

Create a filter based on assembly types. For example, you can create a filter that only shows cast-in-place and precast columns. Steel columns and any other columns or parts are hidden. The same filtering technique can be used for steel, concrete, timber, and miscellaneous parts.

1. Create a new filter. (page 153)
2. Click **Add row** four times to add four filter rules.
3. In the first filter rule, define that the object type must be part:
    a. In the **Category** list, select **Object**.
    b. In the **Property** list, select **Object type**.
    c. In the **Condition** list, select **Equals**.
    d. In the **Value** list, select **Part**.
    e. In the **And/Or** list, select **And**.
4. In the second filter rule, define that the part name must be COLUMN:
    a. In the **Category** list, select **Part**.
    b. In the **Property** list, select **Name**.
    c. In the **Condition** list, select **Equals**.
    d. In the **Value** box, enter the part name, `COLUMN`.
    e. In the **And/Or** list, select **And**.
5. Include the first and second filter rule in brackets.
6. In the third and fourth filter rules, define that the assembly type must be precast or cast in place:
    a. In the **Category** list, select **Assembly**.
    b. In the **Property** list, select **Assembly type**.
    c. In the **Value** box, enter the assembly types, `0` and `1`.

Value	Assembly type
0	precast
1	cast in place
2	steel

Value	Assembly type
3	timber
4	rebar
6	miscellaneous

d. In the **And/Or** list, select **Or**.

7. Include the third and fourth filter rule in brackets. The filter is now looking for concrete parts that are named COLUMN.
8. Enter a unique name in the box next to the **Save as** button.
9. Click **Save as**.

-	(	Category	Property	Condition	Value	)	And/Or
☑	(	Object	Object type	Equals	Part	-	And
☑	-	Part	Name	Equals	COLUMN	)	And
☑	(	Assembly	Assembly type	Equals	1	-	Or
☑	-	Assembly	Assembly type	Equals	0	)	

Filter sub-assemblies

Create a filter that only selects parts that belong to a sub-assembly.

1. Create a selection filter. (page 153)
2. Click **Add row** to add a new filter rule.
3. In the **Category** list, select **Template**.
4. In the **Property** list, select `ASSEMBLY.HIERARCHY_LEVEL`.
5. In the **Condition** list, select **Does not equal**.
6. In the **Value** box, enter `0`.

   In this context, `0` means that the part does not belong to any sub-assembly, and `1` would mean that the part does. The filter will show only those parts whose value is **not** `0`.
7. Enter a unique name in the box next to the **Save as** button.
8. Click **Save as**.

-	(	Category	Property	Condition	Value	)	And/Or
☑	-	Template	ASSEMBLY.HIERARCHY_LEVEL	Does not equal	0	-	And

Filter objects based on their class

Create a filter based on object types and class. The following example filter can be used to select or show parts and reinforcement in certain classes.

1. Create a new view filter. (page 153)
2. Click **Add row** three times to add three filter rules.
3. In the first and second filter rules, define that the object type must be part or reinforcement.
    a. In the **Category** list, select **Object**.
    b. In the **Property** list, select **Object type**.
    c. In the **Condition** list, select **Equals**.
    d. In the **Value** list, select **Part** for the first rule and **Reinforcing bar** for the second rule.
    e. In the **And/Or** list, select **Or** for the first rule and **And** for the second rule.
4. Include the first and second filter rule in brackets.
5. In the third filter rule, use the template attribute `CLASS_ATTR` to define that the object class must be less than or equal to 5:
    a. In the **Category** list, select **Template**.
    b. In the **Property** list, type `CLASS_ATTR`.
    c. In the **Condition** list, select **Less or equal**.
    d. In the **Value** list, enter `5`.

    The filter is now looking for parts and reinforcement that belong to classes 0–5.
6. Enter a unique name in the box next to the **Save as** button.
7. Click **Save as**.

-	(	Category	Property	Condition	Value	)	And/Or
☑	(	Object	Object type	Equals	Part	-	Or
☑	-	Object	Object type	Equals	Reinforcing bar	)	And
☑	-	Template	CLASS_ATTR	Less or equal	5	-	

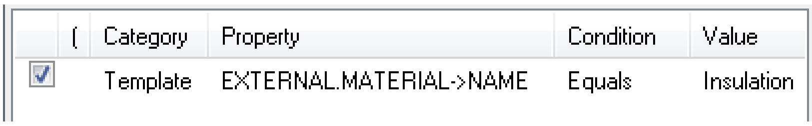
Filter reference model objects

Create a filter based on reference model object properties.

1. Create an empty view or selection filter. (page 153)
2. Click **Add row** to add a new filter rule.
3. In the **Category** list, select **Template**.
4. In the **Property** list, select the desired template attribute or enter one of your own (page 164).

TIP To find the attribute name used by the reference model, select a reference model object, right-click and select one of the **Inquire** commands. Find the property name in the **Inquire object** dialog box and copy it.

5. Add the prefix `EXTERNAL.` in front of the template attribute name.
6. In the **Condition** list, select **Equals**.
7. In the **Value** box, enter the desired value or click **Select from model...** to select the object in the model.
8. Enter a unique name in the box next to the **Save as** button.
9. Click **Save as**.



	(	Category	Property	Condition	Value
☑		Template	EXTERNAL.MATERIAL->NAME	Equals	Insulation

Filter parts within component

Create a filter that selects all parts within a component.

1. Create an empty selection filter. (page 153)
2. Click **Add row** twice to add two filter rules.
3. In the first filter rule, define that the object must be a component:
   a. In the **Category** list, select **Object**.
   b. In the **Property** list, select **Is component**.
   c. In the **Condition** list, select **Equals**.
   d. In the **Value** list, select **Yes**.
   e. In the **And/Or** list, select **And**.
4. In the second filter rule, define that the object type must be part:
   a. In the **Category** list, select **Object**.
   b. In the **Property** list, select **Object type**.
   c. In the **Condition** list, select **Equals**.
   d. In the **Value** list, select **Part**.
5. Enter a unique name in the box next to the **Save as** button.
6. Click **Save as**.

-	(	Category	Property	Condition	Value	)	And/Or
☑	-	Object	Is component	Equals	Yes	-	And
☑	-	Object	Object type	Equals	Part	-	And

Filter reinforcement in pour units based on pour object type

Create a filter that only shows all reinforcement that belongs to pour units with a pour object of a certain type.

1. Ensure that you have XS_ENABLE_POUR_MANAGEMENT set to `TRUE`.
2. Calculate the pour units. (page 453)
3. Create a new view filter. (page 153)
4. Click **Add row** twice to add two filter rules.
5. In the first filter rule, define the pour object type.
   a. In the **Category** list, select **Template**.
   b. In the **Property** box, enter `POUR_UNIT.POUR_OBJECT.POUR_TYPE`.
   c. In the **Condition** list, select **Equals**.
   d. In the **Value** box, enter the pour object type, for example `MyType`, or click **Select from model...** to select the object in the model.
   e. In the **And/Or** list, select **And**.
6. In the second filter rule, define that the object type must be reinforcement:
   a. In the **Category** list, select **Object**.
   b. In the **Property** list, select **Object type**.
   c. In the **Condition** list, select **Equals**.
   d. In the **Value** list, select **Reinforcing bar**.
7. Enter a unique name in the box next to the **Save as** button.
8. Click **Save as**.

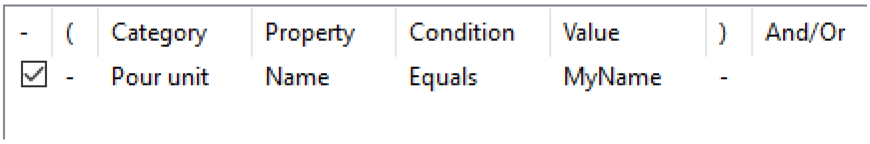
-	(	Category	Property	Condition	Value	)	And/Or
☑	-	Template	POUR_UNIT.POUR_OBJECT.POUR_TYPE	Equals	MyType	-	And
☑	-	Object	Object type	Equals	Reinforcing bar	-	

Filter all contents of a pour unit

Create a filter that selects all contents of a pour unit with a certain name.

1. Ensure that you have XS_ENABLE_POUR_MANAGEMENT set to `TRUE`.
2. Calculate the pour units. (page 453)

3. [Create a selection filter. (page 153)](#)
4. Click **Add row** to add a new filter rule.
5. In the **Category** list, select **Pour unit**.
6. In the **Property** list, select **Name**.
7. In the **Condition** list, select **Equals**.
8. In the **Value** box, enter the pour unit name, for example `MyName`.
9. Enter a unique name in the box next to the **Save as** button.
10. Click **Save as**.



Copy and remove filters

You can copy custom filters to another model by manually copying filter files to the `attributes` folder under the desired model folder. You can also manually remove unneeded filters from the same folder. If you want to make a filter available in all models, copy the file to your company's project or firm folder.

Copy a filter to another model

1. Select the filter you want to copy.

   The filters you have created are located in the `attributes` folder under the current model folder. You can recognize different filter types based on their file name extension:

File name extension	Filter type
`.VObjGrp`	Model view filter
`.SObjGrp`	Model selection filter
`.PObjGrp`	Object group filter
`.vf`	Drawing view filter
`.vnf`	Drawing view level neighbor part filter
`.wdf`	Single-part drawing filter
`.wdnf`	Single-part drawing neighbor part filter
`.adf`	Assembly drawing filter
`.adnf`	Assembly drawing neighbor part filter
`.cuf`	Cast unit drawing filter

File name extension	Filter type
`.cunf`	Cast unit drawing neighbor part filter
`.gdf`	General arrangement drawing filter
`.gdnf`	General arrangement drawing neighbor part filter
`.dsf`	Drawing selection filter

2. To make the filter available in another model, copy the file to the `attributes` folder of the destination model folder.
3. To make the filter available in all models, copy the file to your company's project or firm folder.
4. Restart Tekla Structures.

Remove a filter

1. Remove the filter file from the model's `attributes` folder.
2. Restart Tekla Structures.

Select values from the model

You can select object properties and dates directly from the model. This can be useful when creating view filters, selection filters, and object groups.

Before you start, create an empty view or selection filter, or an object group.

1. Create an empty view or selection filter (page 153), or an object group (page 662).
2. Click **Add row**.
3. Select options from the **Category** and **Property** lists.
4. In the **Value** list, select one of the options.

   The availability of options depends on your selection in the **Property** list. You can select dates from the model only if the property is a date.

   a. To select an object property, click **Select from model...** and then select an object.

   b. To select a date, click **Select date...** to open the **Select Date** dialog box, and then select one of the options.

      You can either select a date from the calendar, select the review date, or define the number of days before or after the review date. The review date is the same as **Review date** in the **Project Status Visualization** dialog box.

1.9 Tips for large models

Modeling item	Tips
Coordinate system (page 51)	• Do not place the model far away from the origin. The further away from the origin you model, the less precise all the computations become. • Mark global coordinates as labels instead of actually using them during modeling. • If you need to use building site coordinates, drop the first digits if they are always the same. For example, instead of coordinate 758 375 6800, use 375 6800. • Base points allow you to use another coordinate system needed for interoperability and collaboration. You can use another coordinate system for inserting reference models and exporting IFC models. When you use base points, you can keep the coordinates small and locate the model wherever needed. You can create as many base points as you need, and select one of those to be the project base point. For more information, see Base points (page 57).
Work area (page 48) and visibility	• Keep the work area as small as possible. • Show only the required parts in views. • Use view filters to control the visibility of parts.
Views (page 31)	• Close unnecessary views. • Close all views when you save large models.
Selection switches	• Switch on the **Select reference models** selection switch only when necessary. The speed of

Modeling item	Tips
	zooming and rotating may be affected by this switch especially in large and complicated models that contain reference models.
Round objects	• Create holes with the **Create bolts** command instead of using part cuts with round beams. • Use studs to model small straight round objects instead of small round beams. • Model lifting hooks and other embeds with reinforcement bars instead of round polybeams.
Hollow core profiles	• Use simple fixed (non-parametric) profiles. • Use chamfers for curved corners.
Custom components (page 823)	• Do not create overly complex custom components. When used in great numbers they consume a lot of memory.
Numbering (page 712)	• Do not number the whole model in one go. Numbering all objects in large models may take a considerable amount of time.
Model database	• If your model file is getting large, repairing the model database can help to reduce the file size considerably and therefore help with memory problems.
Firm and Project folders	• Save `Firm` and `Project` folders locally on the hard drive of your computer instead of a network drive. This saves time if network speed is slow. When working in the multi-user mode, ensure that the folders are synchronized on all users' hard drives so that important data is not lost or changed.

1.10 Create model templates

Model templates allow you to start a model with predefined company templates and settings. This can be especially useful for sub-contractors.

Only single-user models can be created with model templates. If you wish to create a multi-user model using a model template, create the model in single-user mode and then switch to multi-user mode.

By default, the model template folder is saved in your environment folder. Use the advanced option XS_MODEL_TEMPLATE_DIRECTORY to define a different location.

Create a new model template

You can create your own model templates and use them for creating new models. You can select which catalogs, custom components, model subfolders, drawing templates, and report templates from the model are included in the model template.

1. Create a new model.

   Always start by creating a new empty model. This is because old models that have been used in live projects cannot be completely cleaned. They may contain excess information that increases the size of the model even if you delete all objects and drawings from the model.

2. Add the desired part properties, drawing properties, profiles, materials, custom components, sketches, and so on, in the model.

   You can copy the needed attribute files from another model, for example.

3. On the **File** menu, click **Save as** --> **Save**.

   You need to save the model to include custom components in the `xslib.db1` file. If you do not save the model, custom components will not be included in the model template.

4. On the **File** menu, click **Save as** --> **Save as model template**.
5. Enter a name for the model template.
6. Select which catalogs, drawing templates, report templates, and model subfolders to include in the model template.

   You can only select files and folders that are available in the model folder. Catalogs are typically located in an `environment` folder and they are included in the model folder only if they have been modified.

7. If you want to open the destination folder after creating the model template, select the check box.
8. Click **OK**.

   You can now use the model template for creating new models.

9. When you create new models in **File** --> **New**, you can mark the important model templates as favorites, or hide the templates that you do not need.
    a. Select a model template in the list.
    b. Right-click and select **Favorite** or **Hidden**.

       If you marked a template as **Favorite**, it is placed on top of the template list. Alternatively, use the star icon on the template to mark it as **Favorite**, or to remove the marking.

       If you marked a template as **Hidden**, it is removed from the template list. Select the **Show hidden items** check box to show it again.

Modify an existing model template

To modify an existing template, save the model as a new template. Alternatively, you can modify the template by copying new or updated files directly to the model template folder.

1. Create a model using the existing model template.
2. Make the needed changes.
3. Save it as a new model template.

Download model templates

You can download, share, and store model templates using Tekla Warehouse.

Model template options

Use the **Save as model template** dialog box to define which files and folders are included in the model template.

Setting	Files and folders included
Profiles	`profdb.bin` `profitab.inp`
Materials	`matdb.bin`

Components and sketches	`ComponentCatalog.txt`
	`ComponentCatalogTreeView.txt`
	`xslib.db1`
	`thumbnail_bitmap.arc`
	`*.dat` files
	`CustomComponentDialogFiles` folder

Setting	Files and folders included
Attribute definitions	Includes all attribute definitions of the current model.
Bolts and bolt assemblies	screwdb.db assdb.db
Reinforcement	rebar_database.inp RebarShapeRules.xml rebardatabase_config.inp rebardatabase_schedule_config.inp
Meshes	mesh_database.inp
Options	Includes all options of the current model.
Drawing templates	*.tpl files
Report templates	*.rpt files
Include model subfolders	Lists all subfolders found in the model folder. The selected folders are included in the model template. The attributes folder, which contains part and drawing properties, and the CustomComponentDialogFiles folder are included by default.

2 Create parts, reinforcement, and construction objects

When you know the basic principles of how to create and modify different types of model objects in Tekla Structures, you can start working on more detailed level with your model.

First, to get your model started, you need to create some parts (page 196). Parts are the building blocks of the physical model. You can continue working with the parts by, for example, deforming their shape (page 328), or by adding some details to the parts (page 341), such as bolts, welds, or cuts or fittings.

When you use a workshop weld or bolt to join parts together, you will learn how to work with assemblies.

For concrete parts, each concrete part is considered a separate cast unit (page 430). For construction purposes, you may need to merge several concrete parts into one cast unit. If you model cast-in-place concrete structures, you may need to check how to work with pours (page 444). And once you have created a model of concrete parts, you will need to reinforce the parts (page 469) to gain higher strength for the parts.

Additionally, you may need to use points or construction objects (page 621) as modeling aids. Points and construction objects help you to place other objects in the model.

See also

Adjust how model objects are displayed (page 642)

Change the color and transparency of model objects by using object representation (page 656)

Check the model (page 666)

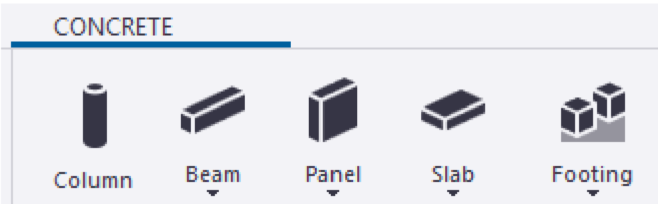
Number the model (page 712)

2.1 Create parts and modify part properties

In Tekla Structures, the term *part* refers to the basic building objects that can be modeled and detailed further. Parts are the building blocks of the physical model.

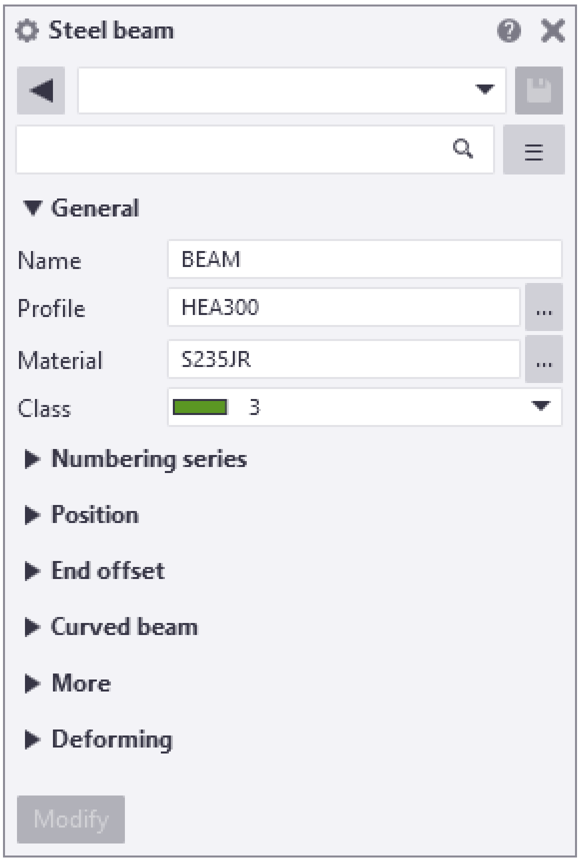
You can create steel parts and concrete parts. A special type of parts are items (page 300). Use them to model objects that would be difficult to model using basic Tekla Structures parts and commands, such as cutting.

Create steel parts by using the commands on **Steel** tab on the ribbon. Create concrete parts by using the commands on the **Concrete** tab on the ribbon.



Every part has properties that define it, such as material, profile, and location (page 310). Also, parts have user-defined attributes (UDAs) (page 323) that you can use to provide additional information about a part. You can use part properties in view filters (page 154) and selection filters (page 156) to, for example, select, modify, and hide parts. You can also include part properties and UDAs in drawing and report templates.

To view and modify the part properties, use the property pane. You can view and modify the properties of a one part type at a time, or the common properties of several, similar part types. If needed, you can copy properties from one part to another part by using the  **Copy properties** button in the property pane.



The basic steel parts are

- column (page 199)
- beam (page 201)
- polybeam (page 204)
- curved beam (page 207)
- twin profile (page 210)
- orthogonal beam (page 213)
- spiral beam (page 216)
- plate (page 220)
- bent plate (page 224)
- lofted plate (page 247)

The basic concrete parts are

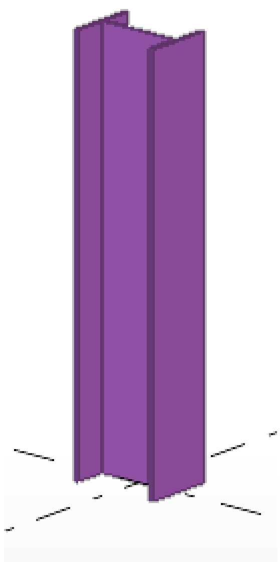
- column (page 260)
- beam (page 260)
- polybeam (page 266)
- spiral beam (page 270)

- panel (page 275)
- slab (page 278)
- lofted slab (page 282)
- pad footing (page 294)
- strip footing (page 297)

Create a steel column

1. On the **Steel** tab, click **Column** .
2. Pick a point.

   Tekla Structures creates the column using the **Steel column** properties in the property pane, and at the level defined in the properties.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Steel column** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Modify steel column properties

1. If the property pane is not open, double-click the column to open the **Steel column** properties.
2. Change the properties as needed.
3. Click **Modify**.

Steel column properties

Use the **Steel column** properties in the property pane to view and modify the properties of a steel column. To open the properties, double-click the steel column. The file name extension of a steel column property file is `*.clm`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the column. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 320) of the column.
Material	Material (page 322) of the column.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, for example, with anti-corrosive paint, hot galvanized, fire retardant coating, and so on.
Class	Use to group columns. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 713).
Assembly numbering	Assembly prefix and start number for the assembly position number (page 713).
Position	

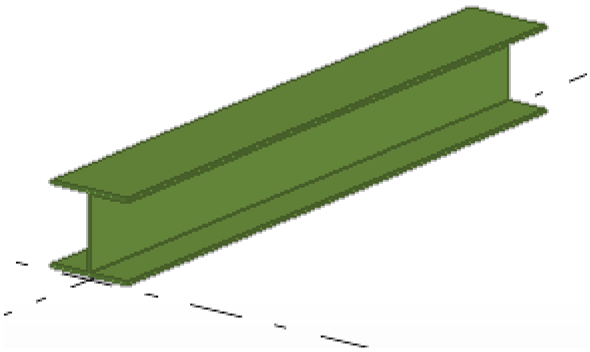
Setting	Description
Vertical	Vertical position (page 315) of the column, relative to column's reference point.
Rotation	Rotation (page 313) of the column around its axis on the work plane.
Horizontal	Horizontal position (page 317) of the column, relative to column's reference point.
Top	Position of the second end of the column in the global z direction.
Bottom	Position of the first end of the column in the global z direction.
Deforming	
Warping	Use to warp columns using deformation angles.
Cambering	Use to pre-camber (page 334) the column.
Shortening	Use to shorten the column in the model. The true length of the column is decreased in the drawing.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. 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)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the part. UDAs provide additional information about the part.

Create a steel beam

1. On the **Steel** tab, click .

2. Pick two points.

   Tekla Structures creates the beam between the points you picked, using the **Steel beam** properties in the property pane.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Steel beam** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Modify steel beam properties

1. If the property pane is not open, double-click the beam to open the **Steel beam** properties.
2. Change the properties as needed.
3. Click **Modify**.

Steel beam properties

Use the **Steel beam** properties in the property pane to view and modify the properties of a steel beam, steel polybeam, or curved beam. To open the properties, double-click the steel beam. The file name extension of a beam property file is `*.prt`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	

Setting	Description
Name	User-definable name of a beam. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 320) of the beam.
Material	Material (page 322) of the beam.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, for example, with anti-corrosive paint, hot galvanized, fire retardant coating, and so on.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 713).
Assembly numbering	Assembly prefix and start number for the assembly position number (page 713).
Position	
On plane	Beam's position on the work plane (page 311), relative to the beam's reference line.
Rotation	Rotation (page 313) of the beam around its axis on the work plane.
At depth	Position depth (page 313) of the beam. The position is always perpendicular to the work plane.
End offset	
Dx	Change the length of the beam (page 318) by moving the beam end point along the beam's reference line.
Dy	Move the beam end (page 318) perpendicular to the beam's reference line.
Dz	Move the beam end (page 318) in the z direction of the work plane.

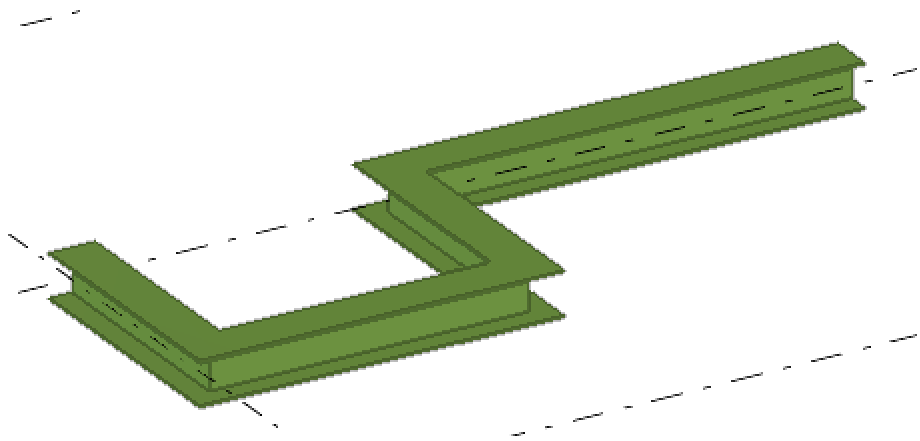
Setting	Description
Curved beam	
Plane	Plane of curvature.
Radius	Radius of the curved beam.
Number of segments	Number of segments Tekla Structures uses when drawing the curved beam.
Deforming	
Warping	Use to warp beams using deformation angles.
Cambering	Use to pre-camber (page 334) beams.
Shortening	Use to shorten beams in the model. The true length of the beam is decreased in the drawing.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. 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)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the part. UDAs provide additional information about the part.

Create a steel polybeam

A polybeam can contain straight and curved segments.

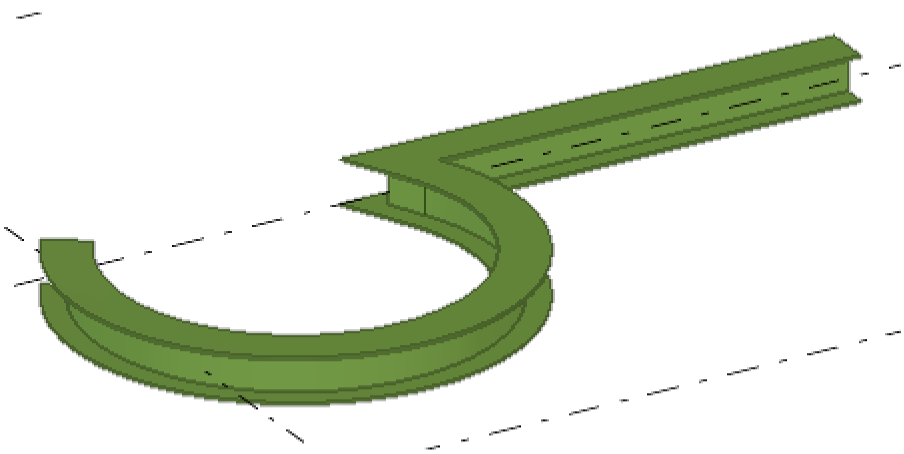
1. On the **Steel** tab, click **Beam** --> **Polybeam**.
2. Pick the points you want the beam to go through.
3. Click the middle mouse button.

Tekla Structures creates the polybeam between the points you picked, using the **Steel beam** properties in the property pane. Note that you cannot create a closed polybeam.



4. If you want to create curved segments, chamfer the corners of the polybeam.

   For example:



Modify steel polybeam properties

1. If the property pane is not open, double-click the polybeam to open the **Steel beam** properties.
2. Change the properties as needed.
3. Click **Modify**.

Steel beam properties

Use the **Steel beam** properties in the property pane to view and modify the properties of a steel beam, steel polybeam, or curved beam. To open the properties, double-click the steel beam. The file name extension of a beam property file is `*.prt`.

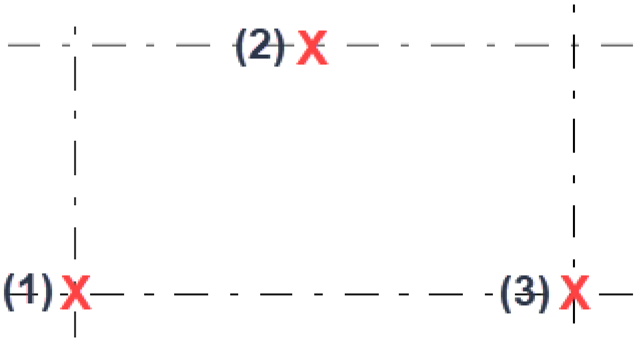
If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of a beam. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 320) of the beam.
Material	Material (page 322) of the beam.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, for example, with anti-corrosive paint, hot galvanized, fire retardant coating, and so on.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 713).
Assembly numbering	Assembly prefix and start number for the assembly position number (page 713).
Position	
On plane	Beam's position on the work plane (page 311), relative to the beam's reference line.
Rotation	Rotation (page 313) of the beam around its axis on the work plane.
At depth	Position depth (page 313) of the beam. The position is always perpendicular to the work plane.
End offset	
Dx	Change the length of the beam (page 318) by moving the beam end point along the beam's reference line.

Setting	Description
Dy	Move the beam end (page 318) perpendicular to the beam's reference line.
Dz	Move the beam end (page 318) in the z direction of the work plane.
Curved beam	
Plane	Plane of curvature.
Radius	Radius of the curved beam.
Number of segments	Number of segments Tekla Structures uses when drawing the curved beam.
Deforming	
Warping	Use to warp beams using deformation angles.
Cambering	Use to pre-camber (page 334) beams.
Shortening	Use to shorten beams in the model. The true length of the beam is decreased in the drawing.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. 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)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the part. UDAs provide additional information about the part.

Create a curved steel beam

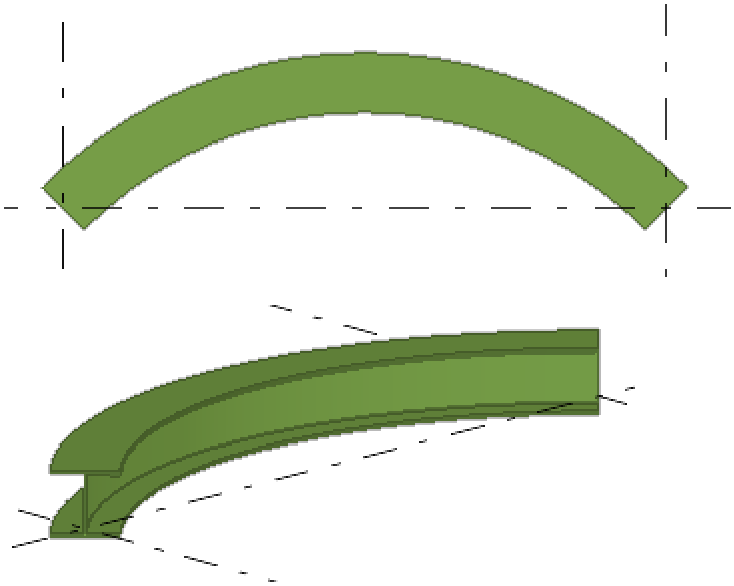
1. On the **Steel** tab, click **Beam** --> **Curved beam**.
2. Pick the start point (1).



3. Pick a point on the arc (2).
4. Pick the end point (3).

   Tekla Structures creates the beam between the points you picked, using the **Steel beam** properties in the property pane.

   The radius is defined by the points you pick.



Modify curved beam properties

1. If the property pane is not open, double-click the curved beam to open the **Steel beam** properties.
2. Change the properties as needed.
3. Click **Modify**.

Steel beam properties

Use the **Steel beam** properties in the property pane to view and modify the properties of a steel beam, steel polybeam, or curved beam. To open the properties, double-click the steel beam. The file name extension of a beam property file is `*.prt`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of a beam. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 320) of the beam.
Material	Material (page 322) of the beam.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, for example, with anti-corrosive paint, hot galvanized, fire retardant coating, and so on.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 713).
Assembly numbering	Assembly prefix and start number for the assembly position number (page 713).
Position	
On plane	Beam's position on the work plane (page 311), relative to the beam's reference line.
Rotation	Rotation (page 313) of the beam around its axis on the work plane.
At depth	Position depth (page 313) of the beam. The position is always perpendicular to the work plane.
End offset	

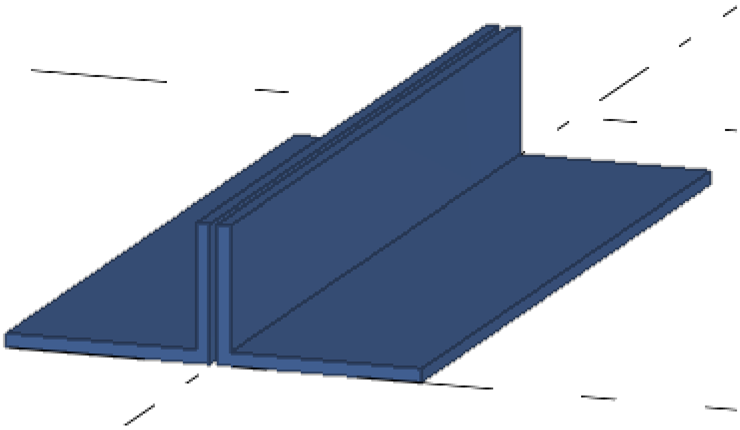
Setting	Description
Dx	Change the length of the beam (page 318) by moving the beam end point along the beam's reference line.
Dy	Move the beam end (page 318) perpendicular to the beam's reference line.
Dz	Move the beam end (page 318) in the z direction of the work plane.
Curved beam	
Plane	Plane of curvature.
Radius	Radius of the curved beam.
Number of segments	Number of segments Tekla Structures uses when drawing the curved beam.
Deforming	
Warping	Use to warp beams using deformation angles.
Cambering	Use to pre-camber (page 334) beams.
Shortening	Use to shorten beams in the model. The true length of the beam is decreased in the drawing.
IFC export	
IFC entity **Subtype (IFC4)** **User-defined type (IFC4)**	For IFC export, select the IFC entity type and subtype of the part. 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	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the part. UDAs provide additional information about the part.

Create a twin profile

A twin profile consists of two parallel and identical beams. You define the positions of both beams by selecting the twin profile type and setting the clearance between the beams in two directions.

1. On the **Steel** tab, click **Beam** --> **Twin profile**.
2. Pick two points.

   Tekla Structures creates the twin profile between the points you picked, using the **Twin profile** properties in the property pane.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Twin profile** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Modify twin profile properties

1. If the property pane is not open, double-click either of the beams to open the **Steel beam** properties.
2. Change the properties as needed.
3. Click **Modify** .

Twin profile properties

Use the **Twin profile** properties in the property pane to view and modify the properties of a twin steel profile. The file name extension of a twin profile property file is `*.dia`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	

Setting	Description
Name	User-definable name of a twin profile. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	The profile of both the beams in the twin profile.
Material	Material (page 322) of the beams.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, for example, with anti-corrosive paint, hot galvanized, fire retardant coating, and so on.
Class	Use to group twin profiles. For example, you can display parts of different classes in different colors.
Twin profile type	Define how the profiles are combined.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 713).
Assembly numbering	Assembly prefix and start number the assembly position number (page 713).
Position	
On plane	Twin profile's position on the work plane (page 311), relative to the twin profile's reference line.
Rotation	Rotation (page 313) of the twin profile around its axis on the work plane.
At depth	Position depth (page 313) of the twin profile. The position is always perpendicular to the work plane.
End offset	
Dx	Change the length of the twin profile (page 318) by moving the twin profile end point along the twin profile's reference line.
Clearance between members	

Setting	Description
Horizontal	The horizontal clearance between the profiles.
Vertical	The vertical clearance between the profiles.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. 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)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the part. UDAs provide additional information about the part.

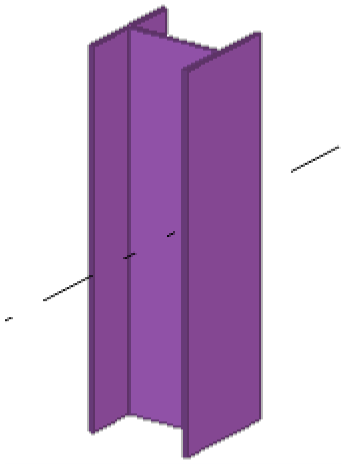
Create an orthogonal beam

Use the **Orthogonal beam** command when you want to create a steel part that is perpendicular to the current work plane. After you have created an orthogonal beam, you can modify it as if it were beam or column.

1. On the **Steel** tab, click **Beam** --> **Orthogonal beam**.

2. Pick a point.

   Tekla Structures creates the beam using the **Orthogonal beam** properties in the property pane, and at the level (page 327) defined in the properties.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button and select **Orthogonal beam** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Modify orthogonal beam properties

1. If the property pane is not open, double-click the orthogonal beam to open the properties.
2. Change the properties as needed.
3. Click **Modify**.

Orthogonal beam properties

Use the **Orthogonal beam** properties in the property pane to view and modify the properties of an orthogonal steel beam. The file name extension of an orthogonal beam property file is `*.crs`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	

Setting	Description
Name	User-definable name of the beam. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 320) of the beam.
Material	Material (page 322) of the beam.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, for example, with anti-corrosive paint, hot galvanized, fire retardant coating, and so on.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Position	
Vertical	Vertical position (page 315) of the beam, relative to beam's reference point.
Rotation	Rotation (page 313) of the beam around its axis on the work plane.
Horizontal	Horizontal position (page 317) of the beam, relative to beam's reference point.
Top	Position of the second end of the beam in the global z direction.
Bottom	Position of the first end of the beam in the global z direction.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 713).
Assembly numbering	Assembly prefix and start number the assembly position number (page 713).
IFC Export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. The
Subtype (IFC4)	
User-defined type (IFC4)	

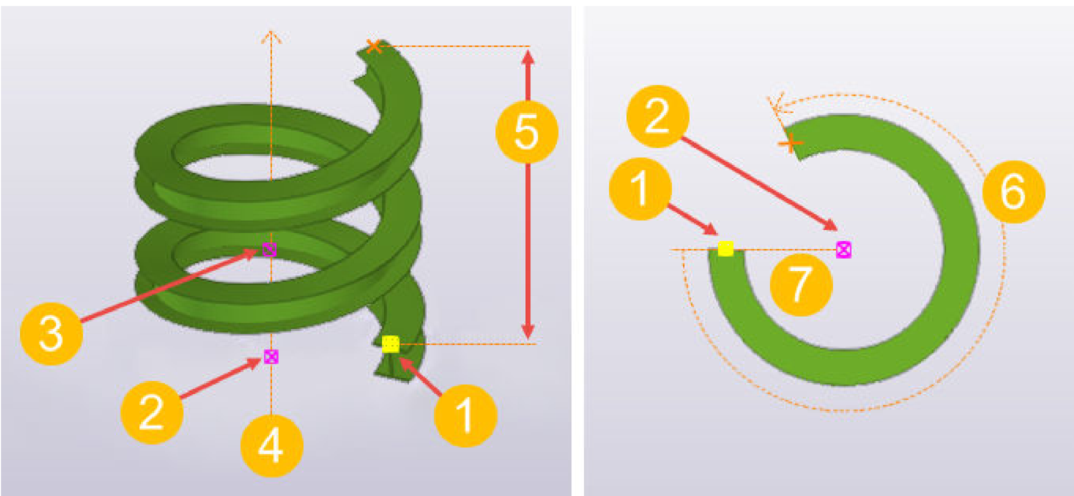
Setting	Description
	available subtypes depend on the selected IFC entity. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the part. UDAs provide additional information about the part.

Create a steel spiral beam

Use the **Create steel spiral beam** command when you want to model spiral staircases and complex architectural shapes, for example.

Basic concepts related to spiral beams

The images below illustrate some basic concepts related to the creation of spiral beams. Note that if you change the positioning, the entire geometry of the spiral beam changes.



(1) Start point (the first point picked)

(2) Center point (the second point picked)

(3) Direction of the rotation axis (the optional third point picked)

(4) Center axis

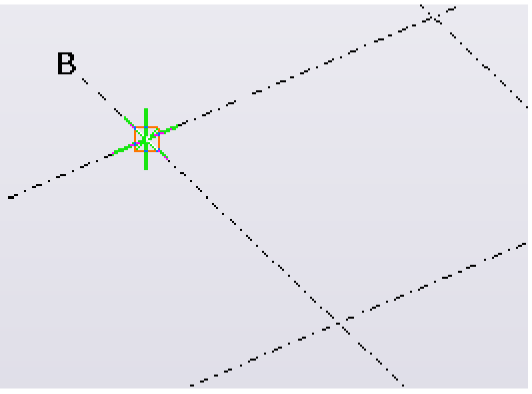
(5) Total rise: the distance from the start point to the end point, parallel to the center axis

(6) Rotation angle: the rotation angle of the spiral beam, given in degrees. Note: positive value = counter-clockwise rotation, negative value = clockwise rotation.

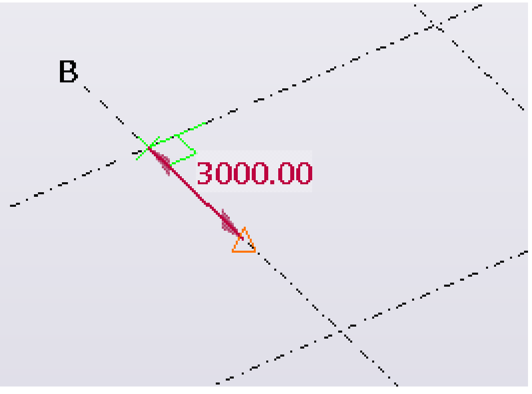
(7) Radius: the distance from the start point to the center point, perpendicular to the center axis

Create a spiral beam

1. On the **Steel** tab, click **Beam** --> **Spiral beam**.
2. Pick the start point.



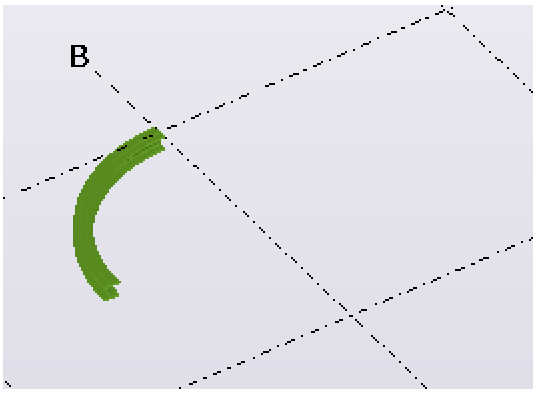
3. Pick the center point.



4. To set the rotation axis in the work plane +Z direction, click the middle mouse button to finish.

NOTE Alternatively, instead of clicking the middle mouse button, you can pick a second center axis point to define the direction of the rotation axis.

Tekla Structures creates the spiral beam. For example:



5. Click the spiral beam to select it.

   The contextual toolbar appears with the following options:

   (1) Rotation angle

   (2) Total rise

   (3) Twist angle at start

   (4) Twist angle at end

6. To add more rotation, enter a bigger value in the **Rotation angle** box.
7. To make the spiral more loose, enter a bigger value in the **Total rise** box.
8. To change the radius, move the start point or center point.

Limitations

- Spiral beam has a single, constant radius.
- Unfolding of spiral beams whose total rise is larger than 0.00 does not produce completely straight results in drawings. The amount of deviation in the part profile outlines and part length depends on several factors: the type, size, and length of the profile; the amount of total rise; and the amount of rotation angle and detailing used.
- Spiral beams are not always untwisted in unfolding. If unequal twisting is applied to the start end and end end, the unfolded drawing shows an unfolded but twisted part as a result.
- Connections and details may not work as expected with spiral beams.
- DSTV export of spiral beams may not produce a correct result.

- You cannot export spiral beams as parts in IFC export. If you are modeling cast-in-place structures with spiral beam, you can export the geometry to IFC as pour objects.

Steel spiral beam properties

Use the **Steel spiral beam** properties in the property pane to view and modify the properties of a steel spiral beam. To open the properties, double-click the spiral beam. The file name extension of a steel spiral beam property file is `*.sb`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of a beam. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 320) of the beam.
Material	Material (page 322) of the beam.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 713).
Assembly numbering	Assembly prefix and start number for the assembly position number (page 713).
Position	
On plane	Beam's position on the work plane (page 311), relative to the beam's reference line.
Rotation	Rotation (page 313) of the beam around its axis on the work plane.

Setting	Description
At depth	Position depth (page 313) of the beam. The position is always perpendicular to the work plane.
Geometry	
Rotation angle	Rotation angle of the spiral beam, given in degrees.
Total rise	Distance from the start point to the end point, parallel to the center axis.
Twist angle at start **Twist angle at end**	Twist +/- angle of the spiral beam at the start/end of the beam.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. 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)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the part. UDAs provide additional information about the part.

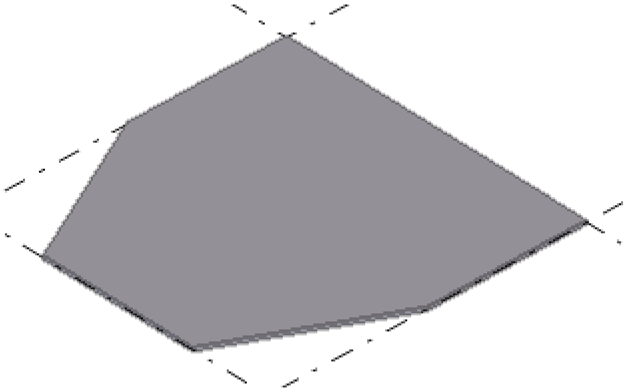
Create a contour plate

When you create a contour plate, the profile you select defines the thickness of the plate and the points you pick define the shape. The corners of the contour plate can be chamfered.

1. On the **Steel** tab, click .
2. Pick the corner points of the contour plate.

3. Click the middle mouse button.

   Tekla Structures creates the plate using the **Contour plate** properties in the property pane.

Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button and select **Contour plate** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

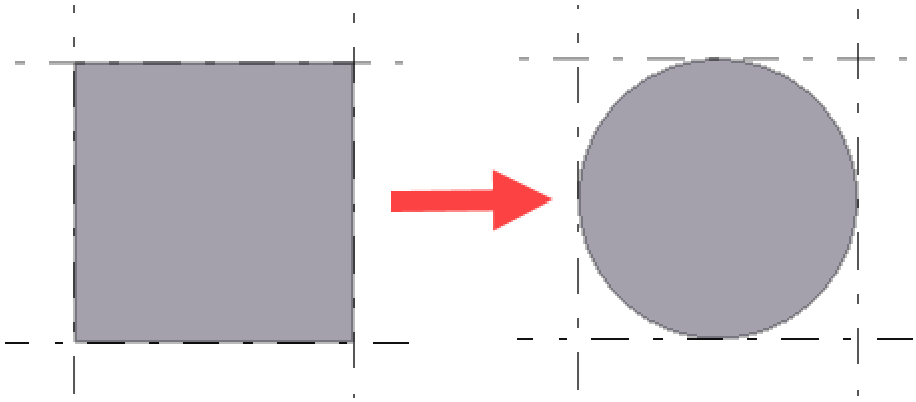
Create a round contour plate

1. Create a square contour plate with four equal sides.
2. Select the plate.
3. Double-click a handle.

   To make it easier to select the handles of the contour plate corners, ensure that the Direct modification switch is **not** active.

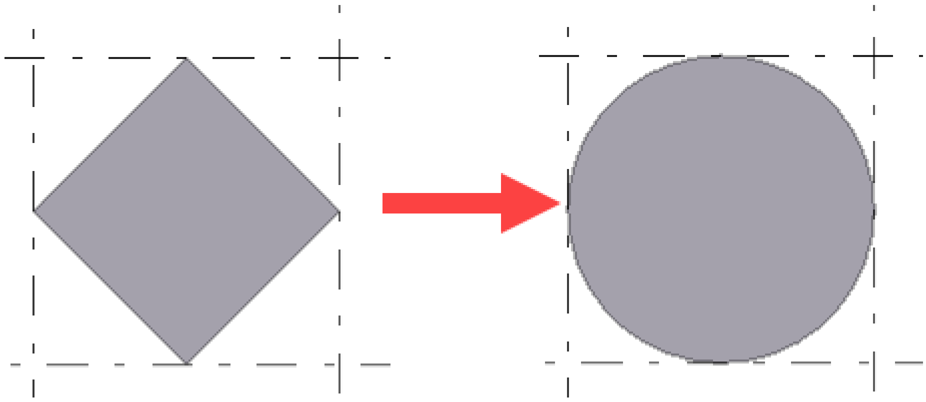
   The **Corner chamfer** properties open in the property pane.
4. In the **Type** list, select **Rounding** .
5. In the **Radius** box, enter the chamfer radius.

   The radius must be equal to half of the side of the square.
6. Click **Modify**.
7. Repeat the above steps for each corner you want to chamfer.



Alternative way of creating a round plate

1. Create a diamond-shaped plate with four equal sides.
2. To round the corners, use the **Arc point** chamfer type.



Modify contour plate properties

1. If the property pane is not open, double-click the plate to open the **Contour plate** properties.
2. Change the properties as needed.
3. Click **Modify**.

Contour plate properties

Use the **Contour plate** properties in the property pane to view and modify the properties of a contour plate. To open the properties, double-click the contour plate. The file name extension of a contour plate property file is `*.cpl`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	

Setting	Description
Name	User-definable name of the contour plate. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 320) of the contour plate.
Material	Material (page 322) of the contour plate.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, for example, with anti-corrosive paint, hot galvanized, fire retardant coating, and so on.
Class	Use to group contour plates. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 713).
Assembly numbering	Assembly prefix and start number the assembly position number (page 713).
Position	
At depth	Position depth (page 313) of the contour plate. The position is always perpendicular to the work plane.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. 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)	
More	

Setting	Description
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the part. UDAs provide additional information about the part.

Create a conical or a cylindrical bent plate

You can create cylindrical or conical bent steel plates either by selecting two parts or two part faces. The parts that you use for creating a bent plate must be contour plates, or beams whose profile is a plate (for example, PL200*20). Position the parts so that there is some room on both sides, so that Tekla Structures can create a curved section between them.

After creating a cylindrical or conical bent plate, the individual parts no longer exist in the model. The bent plate gets its properties and coordinates from the first part that you selected when creating the bent plate. The first part is the main section of the bent plate. You can later change the main section, if needed.

Limitations

- Only the side faces of the part can be used for creating a bent plate.
- Chamfered or cut faces cannot be used for creating a bent plate.
- Curved beams and deformed parts cannot be used for creating a bent plate.
- Details (such as bolts, welds, cuts, chamfers, and preparations) in simple cases are supported on the curved section of the bent plate.

In addition to the cylindrical and conical bent plates, you can also create stand-alone bent plates (page 240) which do not require any input parts.

Create a cylindrical bent plate

You can create a cylindrical bent plate by selecting two steel parts or two part faces. Cylindrical bent plate has a radius that you can modify. The bent plate properties, such as the ID, thickness, class, and material of the plate, are determined by the first part you select.

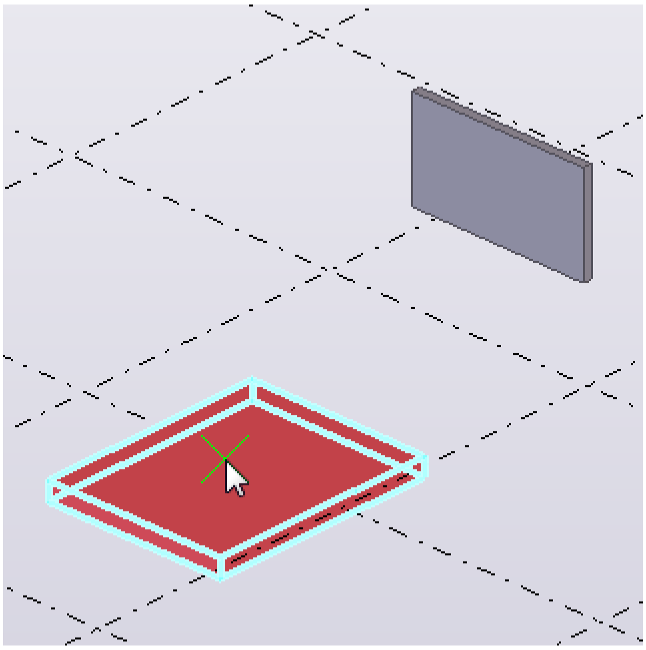
You can create cylindrical bent plates also when the selected parts are intersecting.

1. On the **Steel** tab, click **Plate** --> **Create cylindrical bent plate** .
2. On the bent plate toolbar, select whether you want to create the bent plate by selecting parts or by selecting part faces.

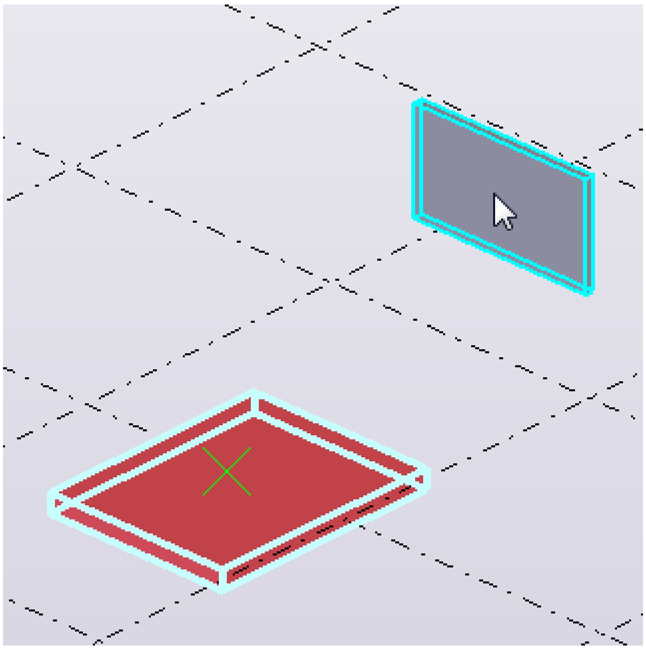
Additionally, you can enter the radius of the cylindrical bent plate. If no radius is entered, Tekla Structures creates the bent plate using the default radius.

- If you selected **By parts**:

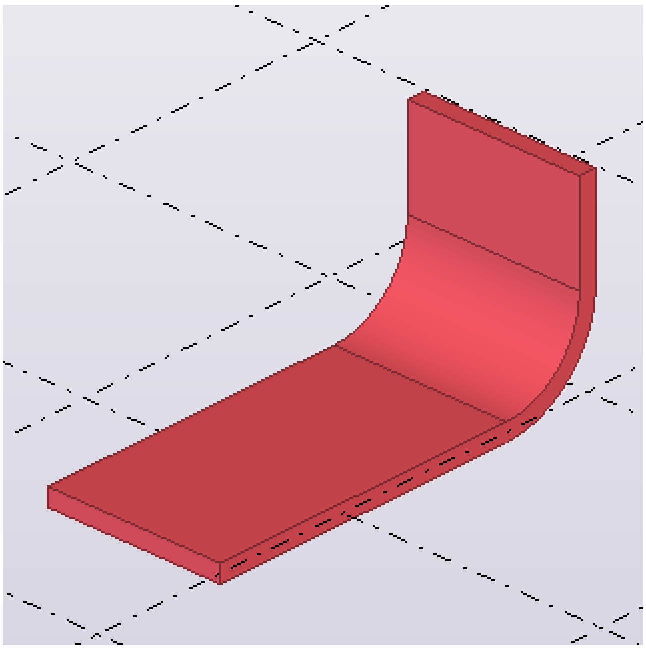
  a. Select the first part.

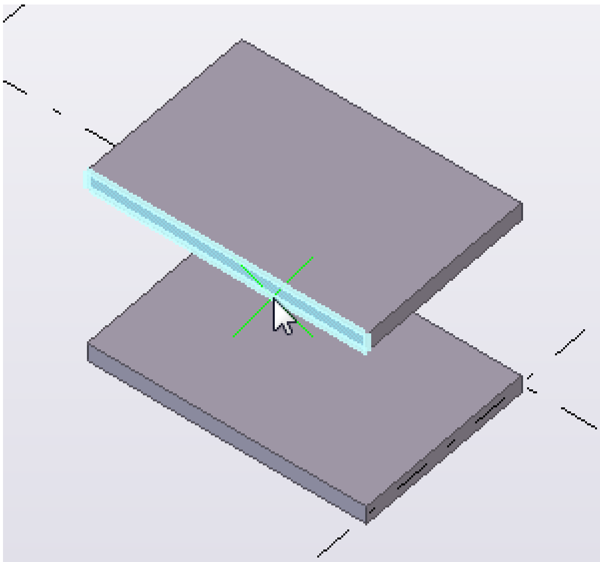
  b. Select the second part.

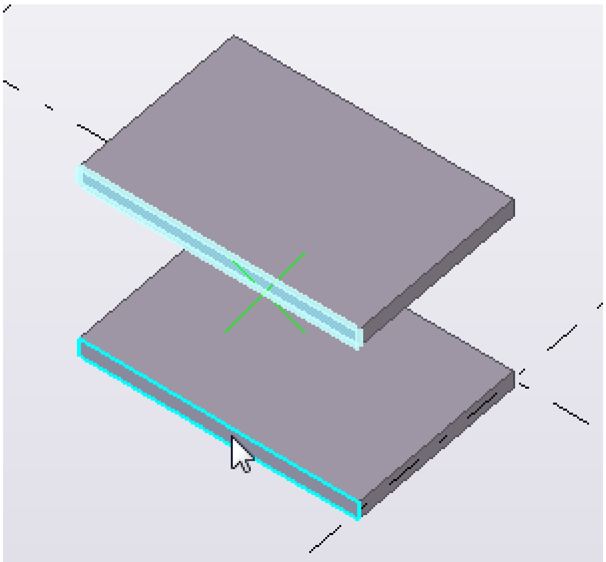
  Tekla Structures creates the cylindrical bent plate.



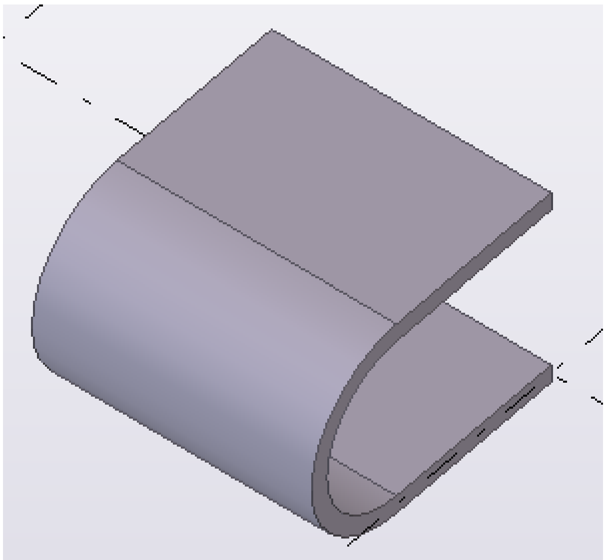
- If you selected **By faces**:
  a. Select the first part face.



b. Select the second part face.



Tekla Structures creates the cylindrical bent plate.



Create a conical bent plate

You can create a conical bent plate by selecting two steel parts or two part faces. Conical bent plate has two radiuses that you can modify. The bent plate properties, such as the ID, thickness, class, and material of the plate, are determined by the first part you select.

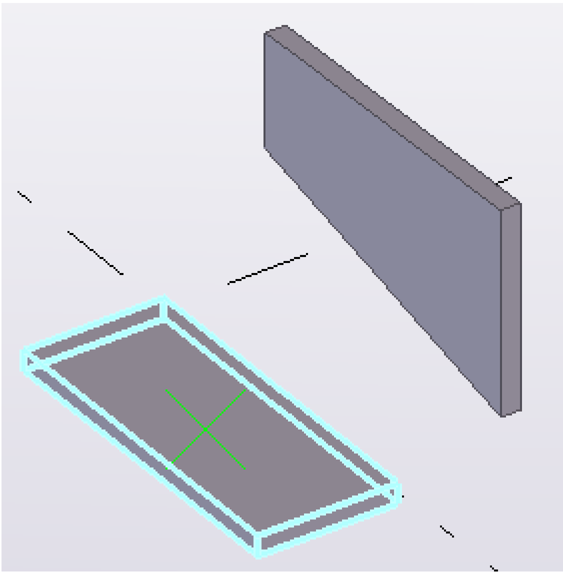
To be able to create conical bent plates, the selected parts or part faces need to be of a shape that creating a conical bent plate is possible. If the selected parts or part faces are such that creating a cylindrical bent plate is possible, a cylindrical bent plate is created. You can create different shapes of conical bent plates: inward bend, outward bend, or with 180 degrees opening angle.

1. On the **Steel** tab, click **Plate** --> **Create conical bent plate** .
2. On the bent plate toolbar, select whether you want to create the bent plate by selecting parts or by selecting part faces.

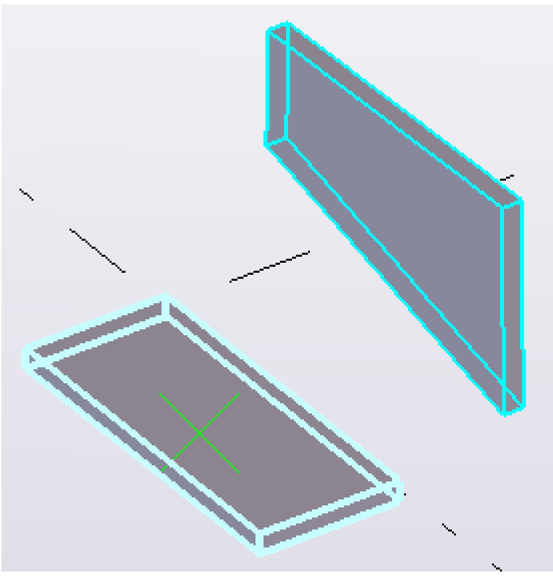
   You can enter two radiuses for the conical bent plate. If no radiuses are entered, Tekla Structures creates the bent plate using the default radiuses.

   • If you selected **By parts**:

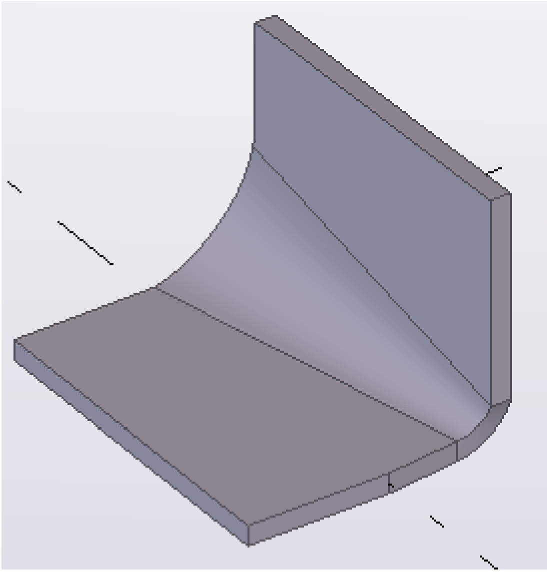
     a. Select the first part.

     b. Select the second part.

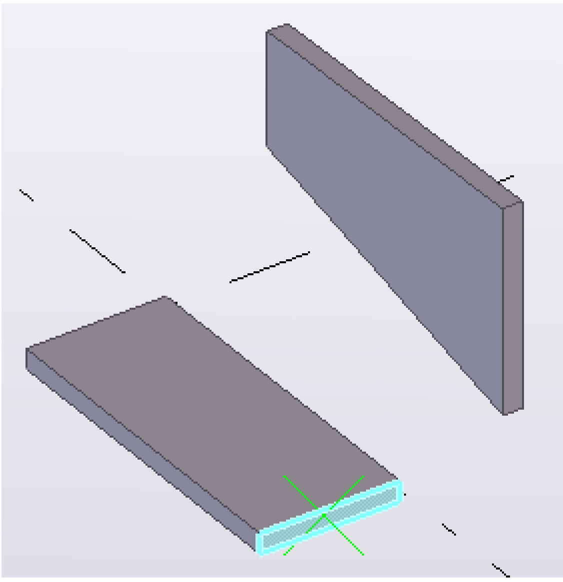
     

c. Tekla Structures creates the conical bent plate.

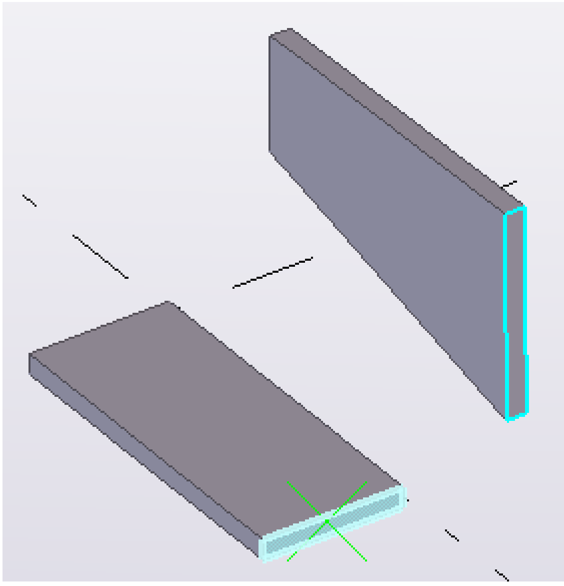


- If you selected **By faces**:

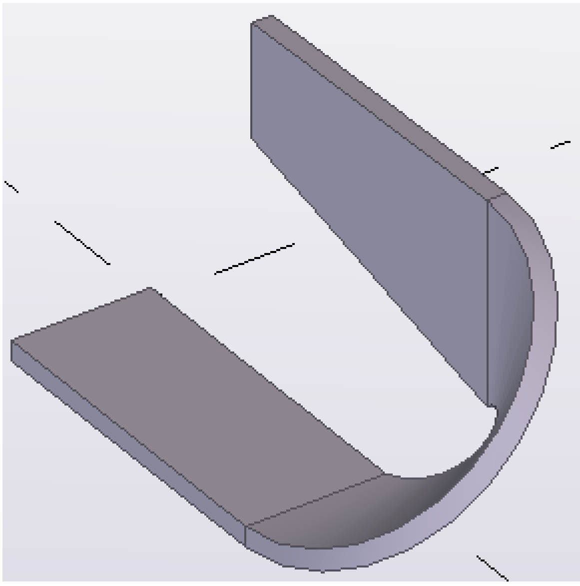
a. Select the first part face.



b. Select the second part face.



c. Tekla Structures creates the conical bent plate.

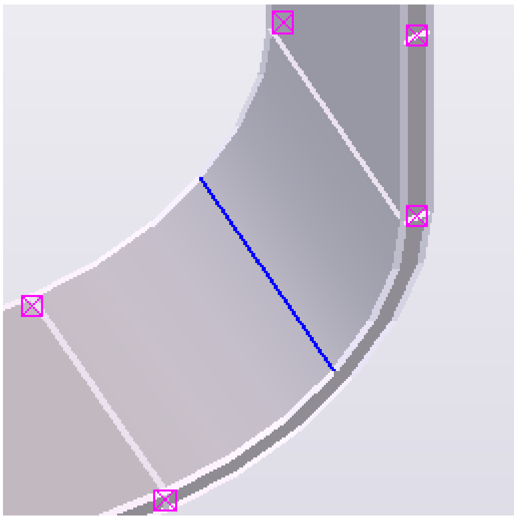


Modify the bend radius

When you create a cylindrical bent plate, you can enter a radius for the plate. For a conical bent plate you can enter two radiuses. If you do not enter any radius, Tekla Structures uses a default bend radius when creating bent plates. You can later change the bend radius to suit your needs.

1. Ensure that  **Direct modification** is switched on.
2. Select the bent plate.

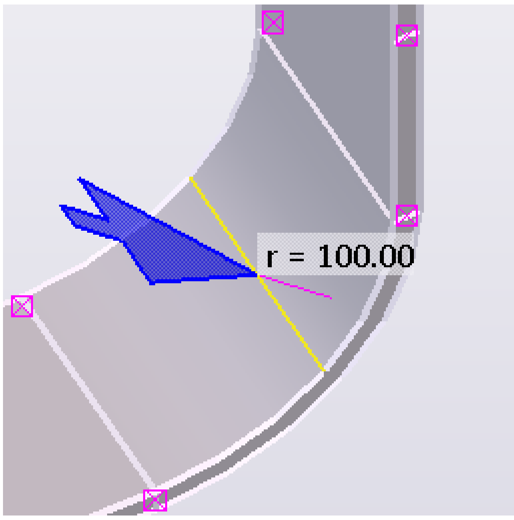
A blue line handle appears in the middle of the curved section.



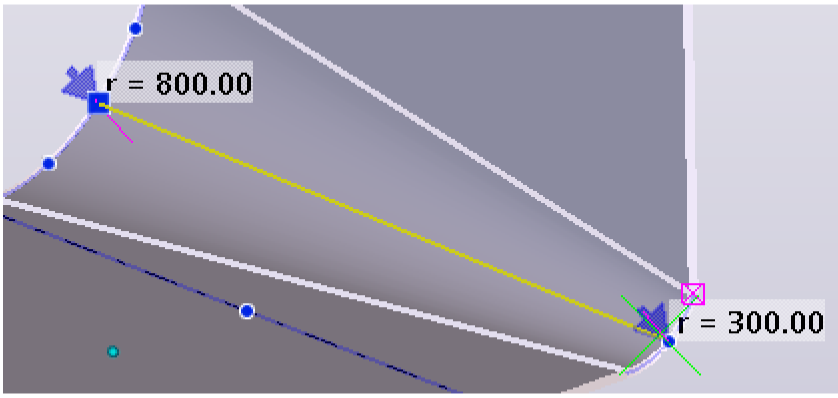
3. Select the line handle.

   Depending on the type of the bent plate, one (for cylindrical bent plate) or two (for conical bent plate) blue dimension arrows appear.

   - For cylindrical bent plate:

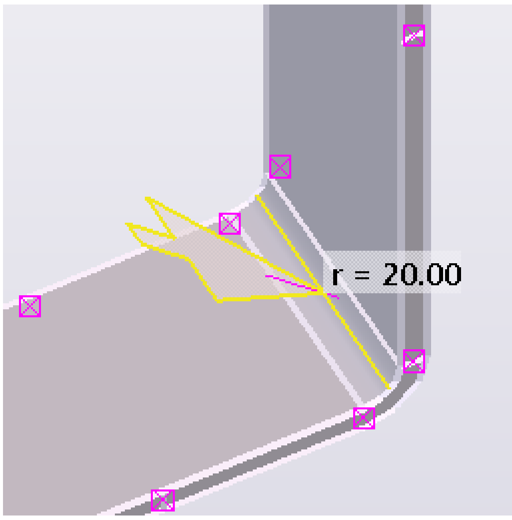


- For conical bent plate:



4. To modify the radius, do one the following:
   - Drag the arrow or arrows forward or backward along the magenta line.

     The dimension "r =" changes accordingly. When you release the arrow, the radius also changes in the model.

   - On the contextual toolbar, enter the radius or the radiuses.
   - Alternatively, you can select the arrow and type a dimension. When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog box. Click **OK** to confirm the dimension.

Modify the shape of a bent plate

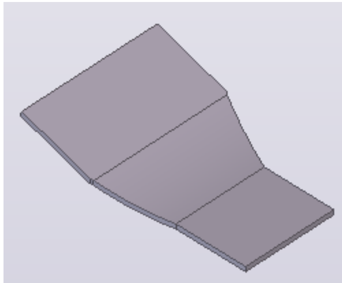
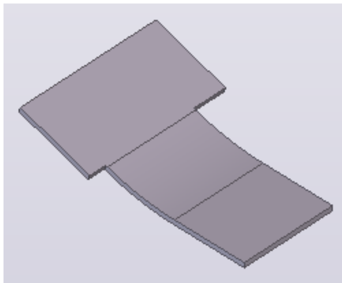
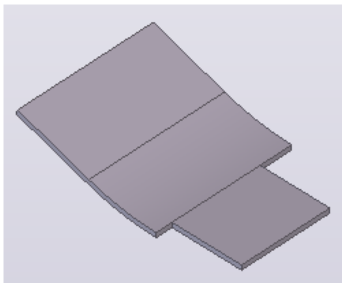
When you create a bent plate, Tekla Structures adds a curved section between the parts you select. You can modify the curved section by choosing one of the predefined options or by modifying the shape manually. You can also modify the flat sections, which are the original parts the bent plate was made of.

1. Ensure that  **Direct modification** is switched on.

2. Select the bent plate.

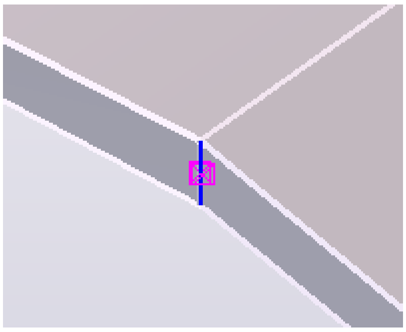
   A blue line handle appears in the middle of the curved section.
3. Select the line handle.

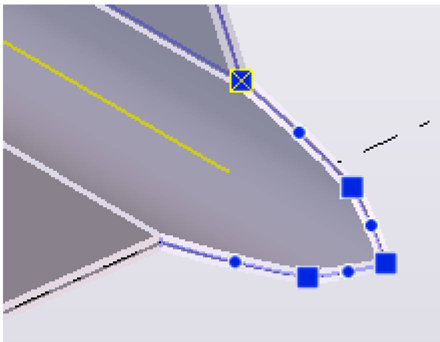
   A contextual toolbar appears.
4. On the contextual toolbar, select one of the predefined shape options:

Option	Description	Example
Tapered bend 	A gradual decrease in the width between the parts. This is the default shape.	
Narrow bend 	Constant width between the parts. The width is determined by the **narrowest** part.	
Wide bend 	Constant width between the parts. The width is determined by the **widest** part.	

5. To modify the curved section manually:
   a. Select the blue line handle.

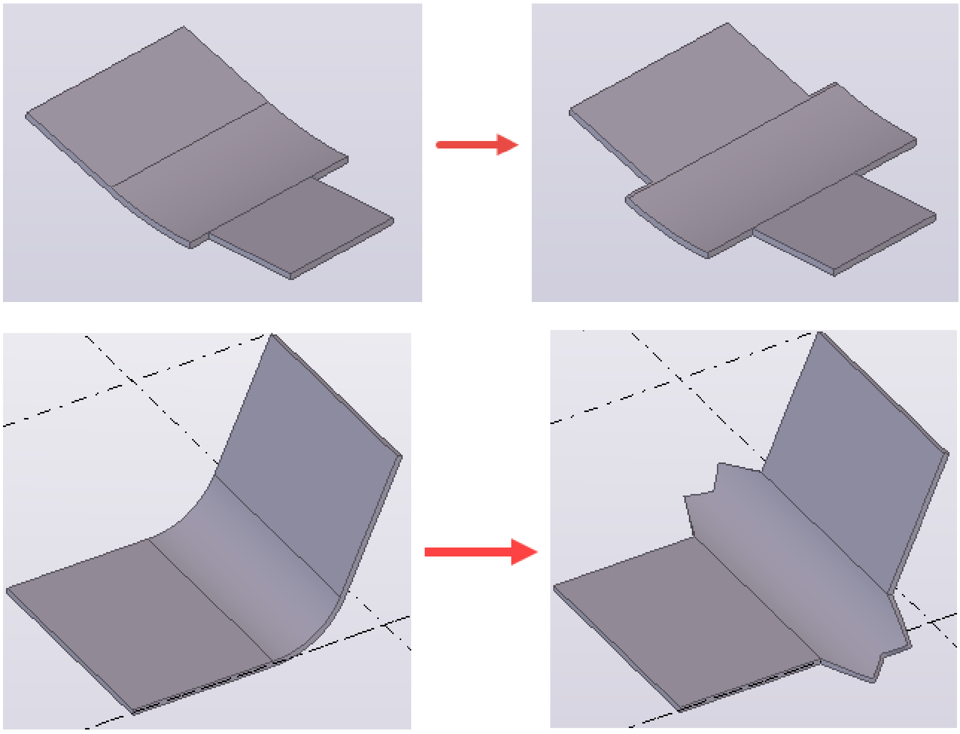
You can modify the side boundaries or the lateral boundaries of the bends both in the cylindrical and in the conical bent plate. Tekla Structures displays the boundary handles in blue:





b. Drag the handles to change the shape of the curved section.

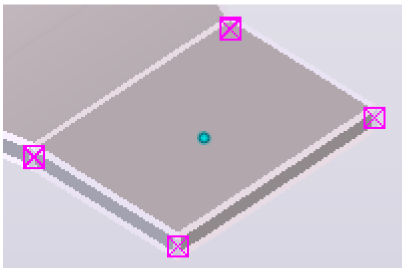
For example:



6. To modify the flat sections:

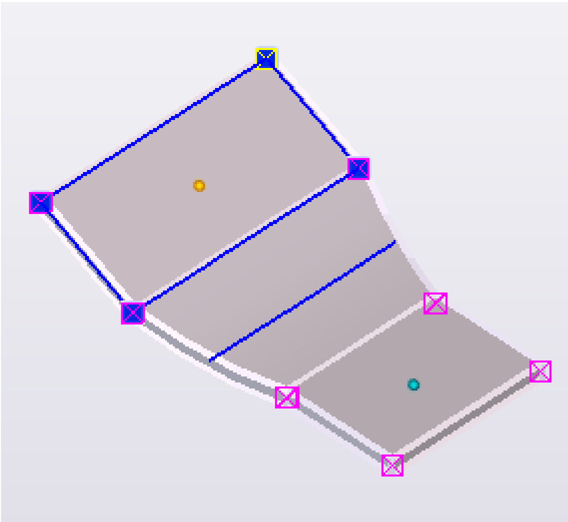
a. Select the bent plate.

Tekla Structures displays a green selection handle in the middle of each flat section:



b. Click the selection handle of the section you want to modify.

The direct modification handles of the selected section become visible:



c. Use the direct modification handles to change the shape of the flat section.

7. To modify the angle of the bent plate:

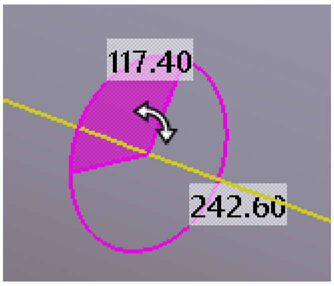
a. Click the green selection handle in the middle of the flat section whose angle you want to modify.

b. Select the line handle.

A contextual toolbar appears.

c. On the contextual toolbar, click  **Enable angle manipulator**.

The angle manipulator wheel appears in the model.



d. Modify the angle using the wheel.

   If you want to modify the angle of another flat section, click the other green selection handle.

8. To change the main section of the bent plate:

   a. Click the green selection handle of the section that you want to set as the main section.

      A contextual toolbar appears.

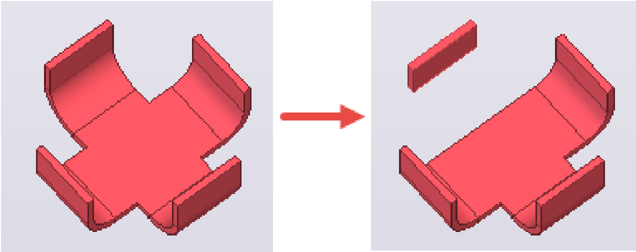
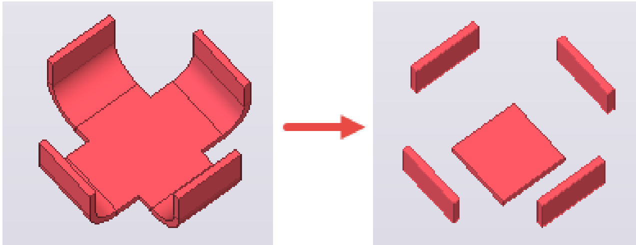
   b. On the contextual toolbar, click  **Set main section**.

      The direct modification handles become active in the new main section. The main section and the coordinate system of the bent plate change accordingly, changing the bent plate orientation in an unfolded drawing.

Remove curved sections

You can return bent plates into individual objects, and then edit and use them as any other model object. If the bent plate consists of several curved sections that are all joined to the same part, you can either remove each curved section separately or explode the entire bent plate all at once.

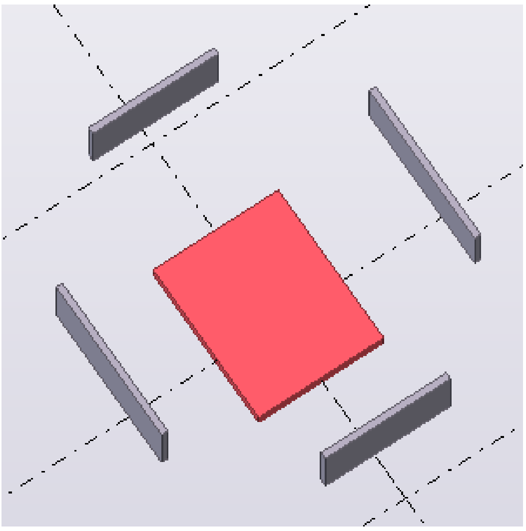
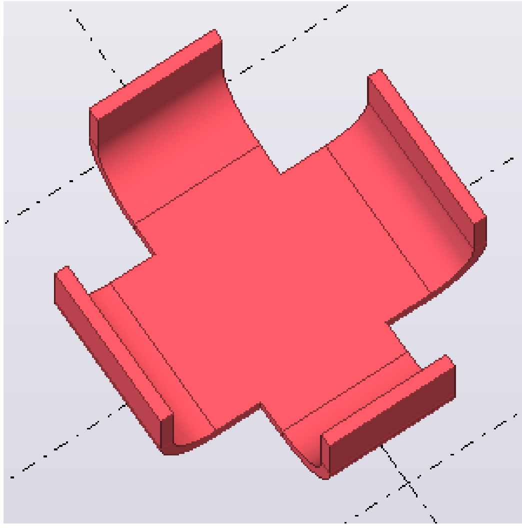
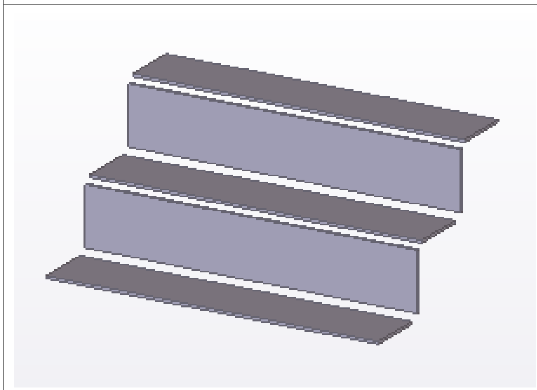
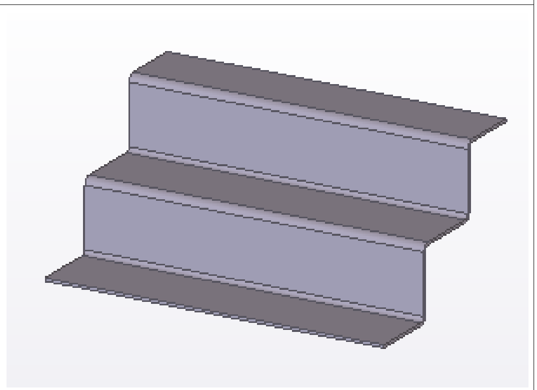
To	Do this
To remove individual curved sections	1. Ensure that  **Direct modification** is switched on. 2. Select the curved section you want to remove. A blue line handle appears. 3. Select the line handle. A contextual toolbar appears. 4. On the contextual toolbar, click  **Remove bend**.

To	Do this
	Tekla Structures removes the selected curved section. For example: 
To explode the entire bent plate	1. Select one of the curved sections. 2. Right-click and select **Explode**. Tekla Structures explodes the entire bent plate into individual objects. For example: 

If the bent plate solids that have been created using a previous version of Tekla Structures seem to be broken, you can remove or explode the bends. Then remodel the bends using the **Create cylindrical bent plate** command.

Examples

Here are some examples of bent plates that you can create:

Starting point	End result
	
	

Modify bent plate properties

1. If the property pane is not open, double-click the bent plate to open the **Bent plate** properties.
2. Change the properties as needed.
3. Click **Modify**.

Bent plate properties

Use the **Bent plate** properties in the property pane to view and modify the properties of a bent plate. To open the properties, double-click the bent plate. The file name extension of a bent plate property file is `*.bpl`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	

Setting	Description
Name	User-definable name of the plate. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 320) of the plate.
Material	Material (page 322) of the plate.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, for example, with anti-corrosive paint, hot galvanized, fire retardant coating, and so on.
Class	Use to group plates. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 713).
Assembly numbering	Assembly prefix and start number for the assembly position number (page 713).
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. 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)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the part. UDAs provide additional information about the part.

Create a stand-alone bent plate

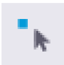
You can create stand-alone bent plates which do not require any input parts to be selected. Use stand-alone bent plates to model cylindrical and conical parts, such as hoods, hoppers, cones, and so on.

Limitations

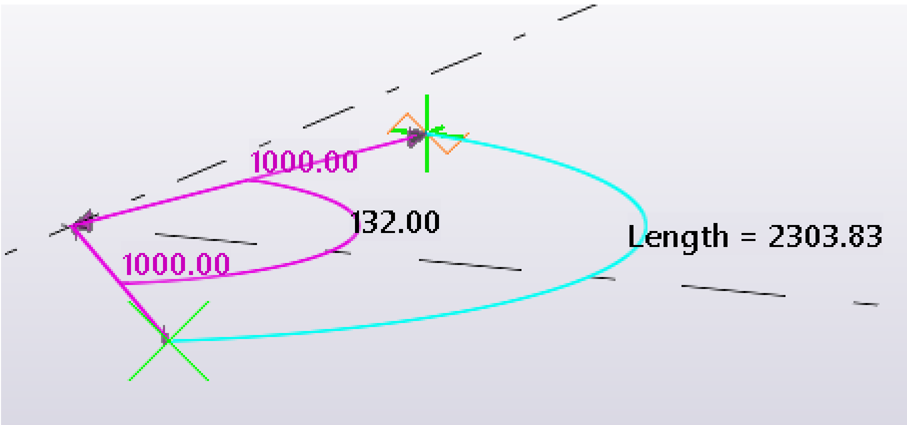
- Tekla Structures does not support full 360 degree stand-alone bent plates. However, you can create 359 degree plates instead.
- Use the local coordinate system in drawing creation.

In addition to the stand-alone bent plates, you can also create cylindrical and conical bent plates (page 224) either by selecting two parts or two part faces. The parts that you use for creating a bent plate must be contour plates, or beams whose profile is a plate.

Create a stand-alone bent plate

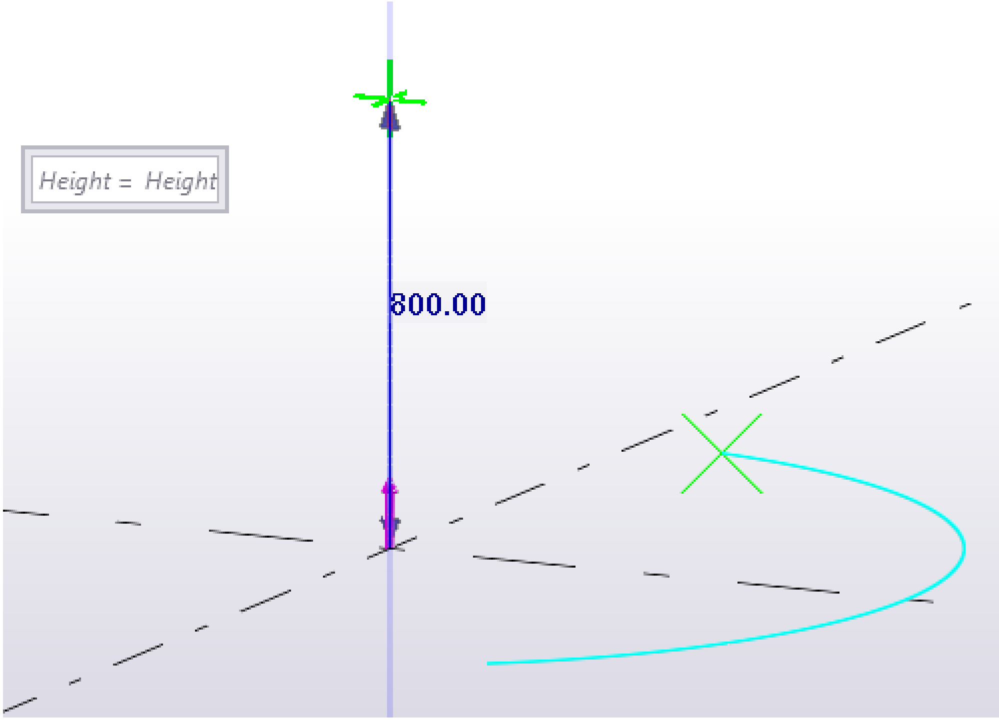
1. Ensure that  **Direct modification** is switched on.
2. On the **Steel** tab, click **Plate** --> **Create stand-alone bent plate** .
3. Define the first radius of the bend:
   a. Pick the center point.
   b. Pick the start point for the arc.
   c. Pick the end point for the arc.

   The picking order defines the up direction. For example, if you create an arc on the xy plane in the counterclockwise direction, the up direction points to the positive z axis, according to the right-hand rule (page 53).

4. Pick a point to define the height of the bend.

Alternatively, you can enter the height in the bent plate contextual toolbar.

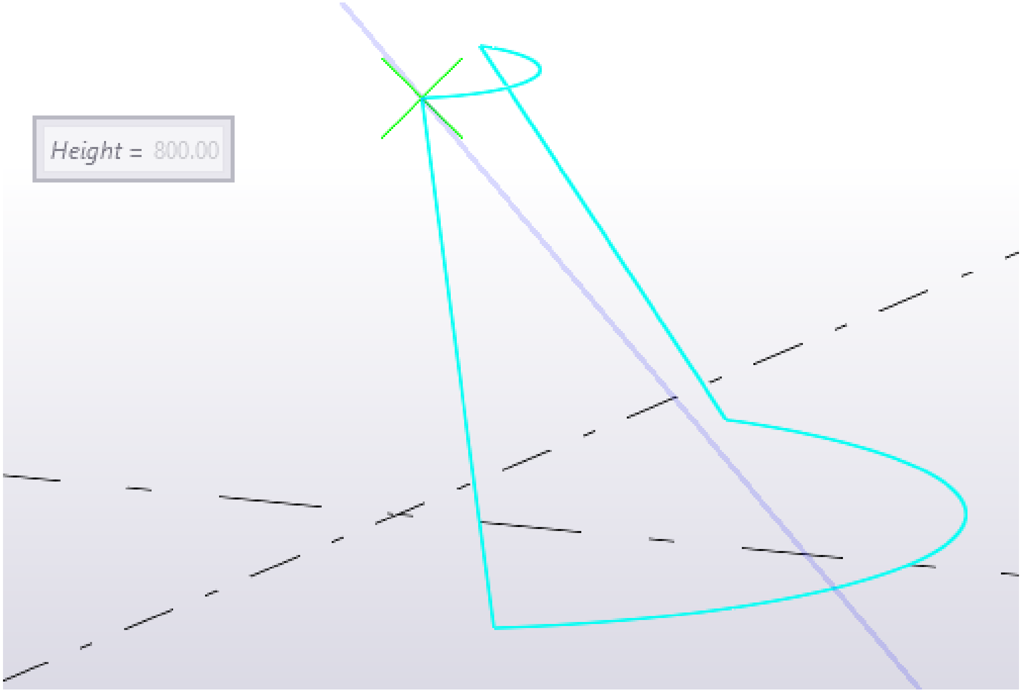


5. Define the second radius of the bend:

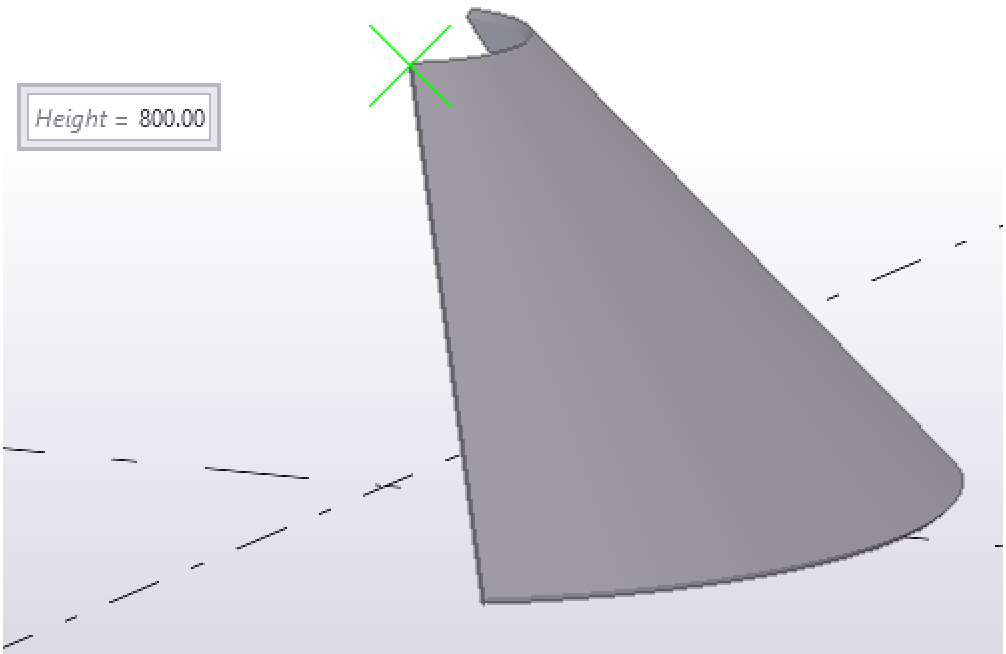
   a. Pick a point based on the preview of the plate.

   b. If you want to change the direction of the plate after you have picked a point, click the left mouse button.

      Alternatively, if you want to create a cylindrical bend, click the middle mouse button. In this case the radius 2=radius 1.

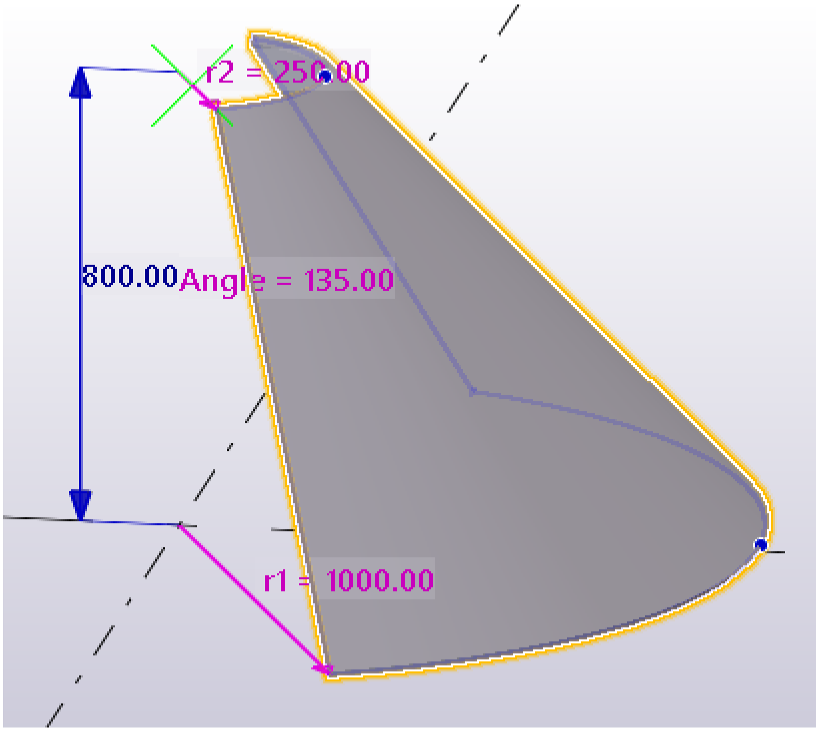


6. To complete the bent plate creation, click the middle mouse button.

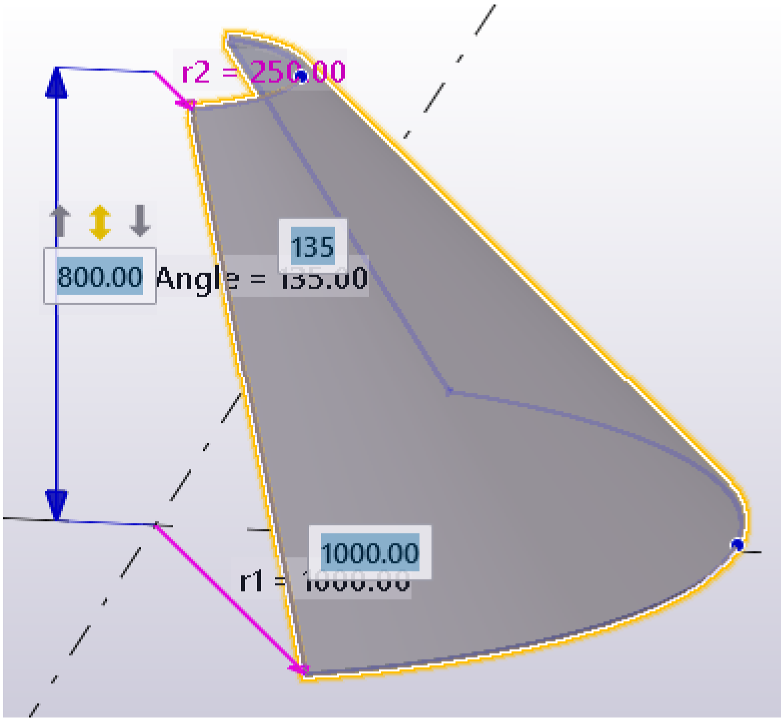


Modify the shape of a stand-alone bent plate

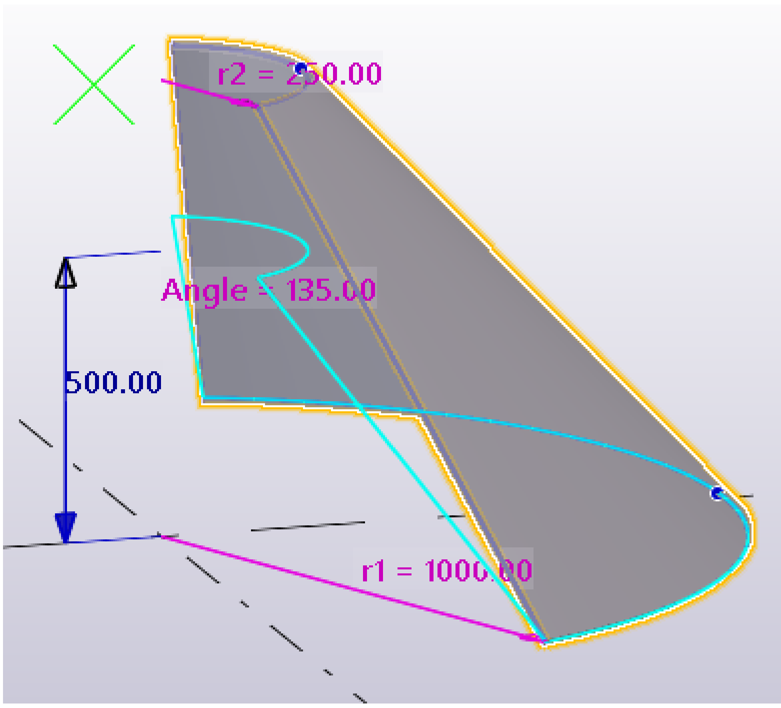
Use direct modification dimension values and handles to modify the bent plate shape.

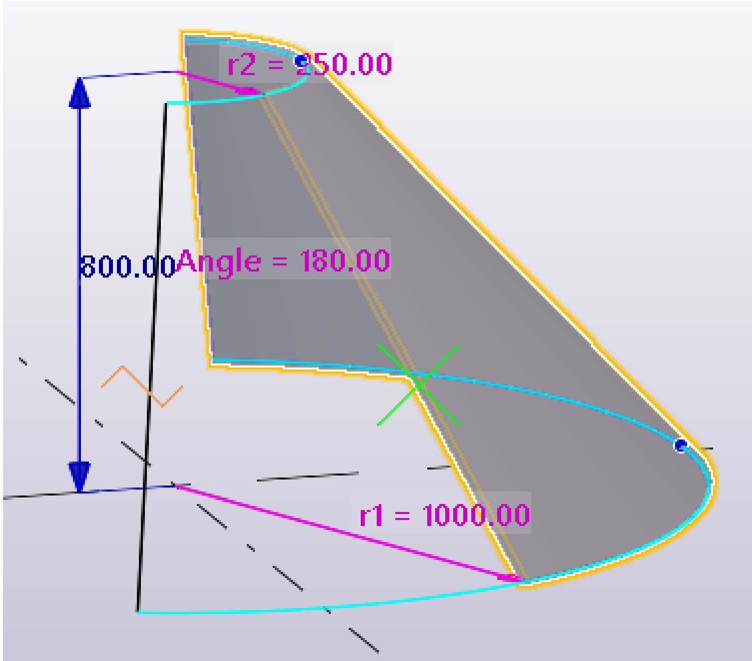


- You can change the angle, radiuses and height of the bend by entering new dimension values.

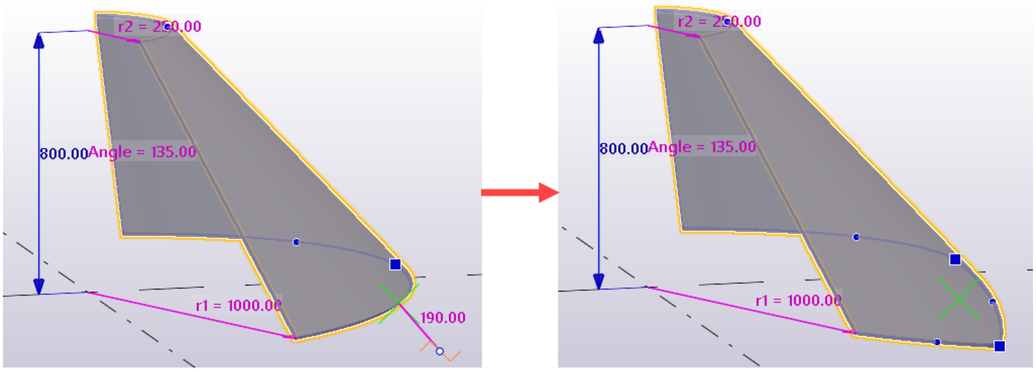


- You can drag and stretch the edges of the bent plate.





- You can add and delete intermediate points.



- You can create unfolded drawings of the stand-alone bent plates.

Modify bent plate properties

1. If the property pane is not open, double-click the bent plate to open the **Bent plate** properties.
2. Change the properties as needed.
3. Click **Modify**.

Bent plate properties

Use the **Bent plate** properties in the property pane to view and modify the properties of a bent plate. To open the properties, double-click the bent plate. The file name extension of a bent plate property file is `*.bpl`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the plate. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 320) of the plate.
Material	Material (page 322) of the plate.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, for example, with anti-corrosive paint, hot galvanized, fire retardant coating, and so on.
Class	Use to group plates. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 713).
Assembly numbering	Assembly prefix and start number for the assembly position number (page 713).
IFC export	
IFC entity **Subtype (IFC4)** **User-defined type (IFC4)**	For IFC export, select the IFC entity type and subtype of the part. 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	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the

Setting	Description
	part. UDAs provide additional information about the part.

Create a steel lofted plate

With lofted plates you can create rolled plates with varying shapes and plates with double-curved form, for example.

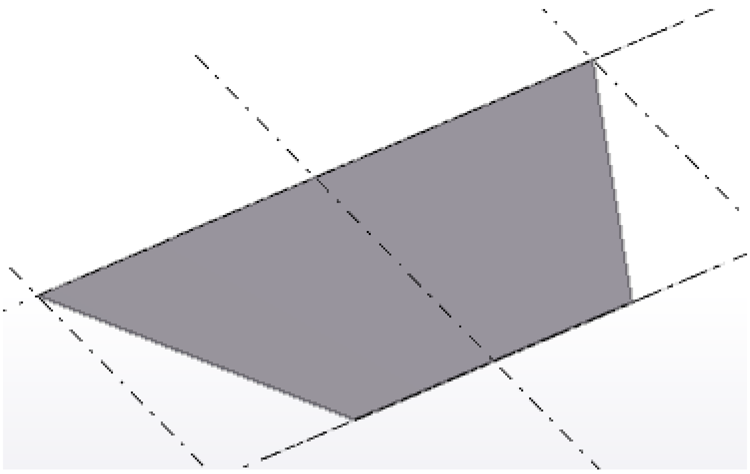
Prerequisites and examples of lofted plates

Before you can create lofted plates, you need to have construction objects (page 621) in your model. Tekla Structures creates the shape of the lofted plate according to the geometry of the used construction objects, by connecting the start point of the first construction object to the start point of the second construction object. The end points of construction objects are connected in similar way.

You can connect the following construction objects as a lofted plate:

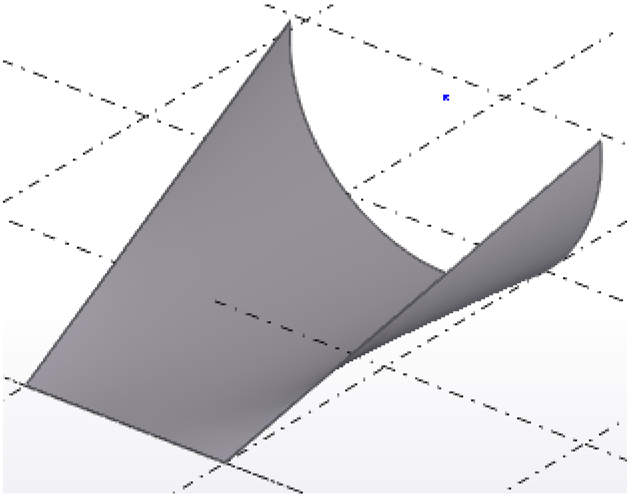
- construction line to construction line

  For example:



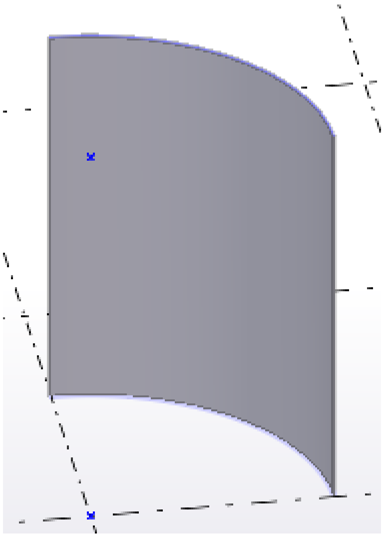
- construction line to construction arc

  For example:



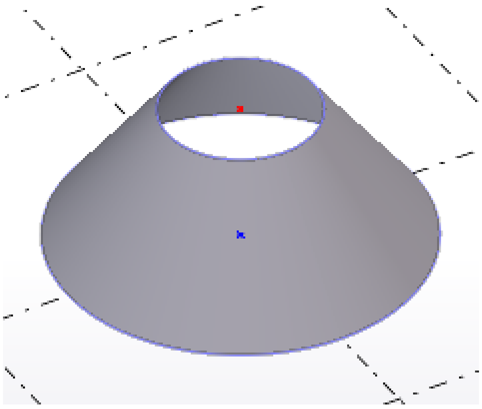
- construction arc to construction arc

  For example:

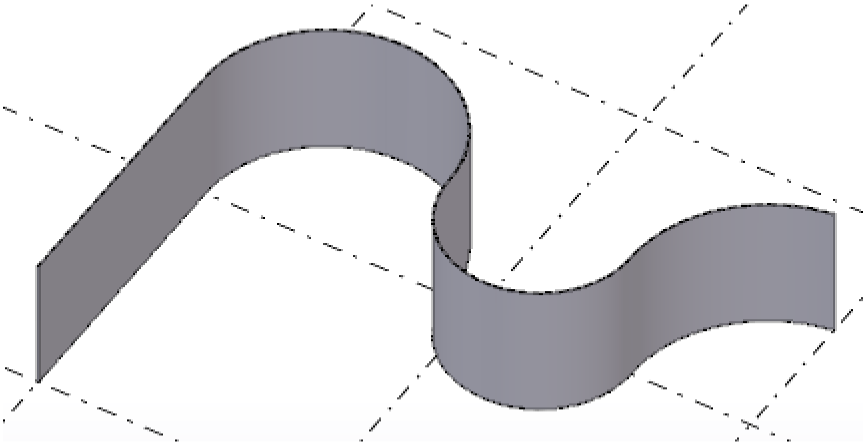


- construction circle to construction circle

  For example:



- construction polycurve to construction polycurve



Create a lofted plate

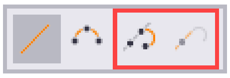
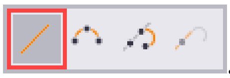
1. Create the needed construction objects in the model. The shape of the lofted plate is based on the construction objects' shape.

   You need to have

   - construction lines (page 622)
   - construction arcs (page 624)
   - construction circles (page 623)

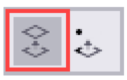
     or

   - construction polycurves (page 626)

     With polycurves, use the **Create arc by tangent** or **Create tangent line** options  on the construction polycurve toolbar. To create polycurves only with straight segments, use the **Create line** option . Note that the construction polycurves do not need to have the same number of segments, as long as both of them are tangential.

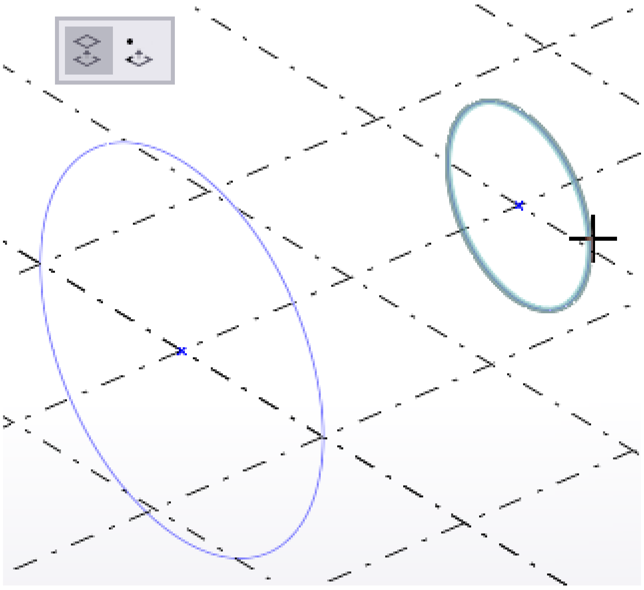
2. When you have created the needed construction objects, go to the **Steel** tab and click **Plate** --> **Create lofted plate**.

3. On the toolbar that appears, click a button to specify whether to create the lofted plate by using two construction objects, or by using a construction object and a point.

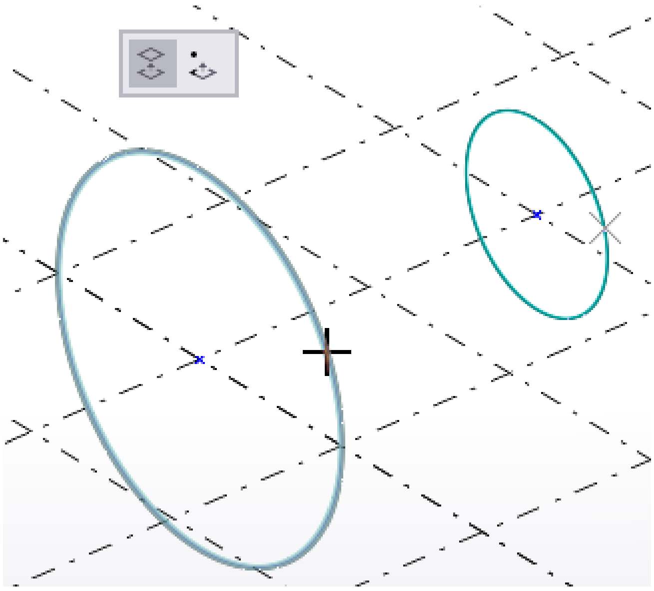
   - Use two construction objects  to create a lofted plate:

a. Select the first construction object: line, arc, circle, or polycurve.

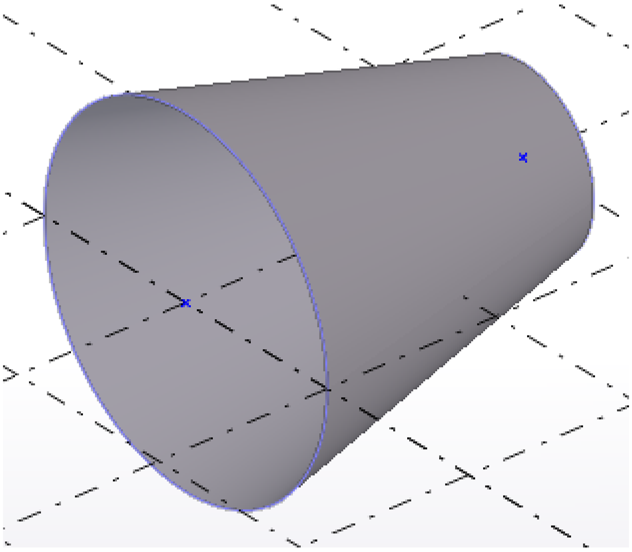
For example, if you are using two construction circles to create a lofted plate.

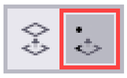


b. Select the second construction object.

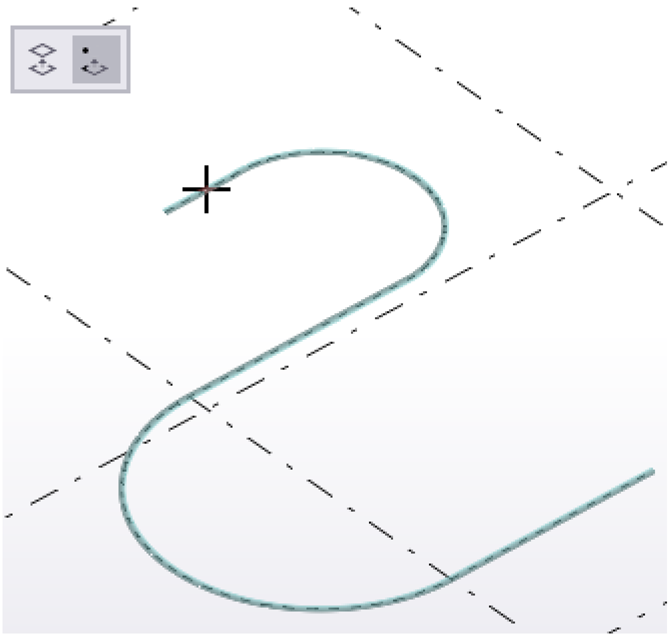


Tekla Structures creates the lofted plate between the selected construction objects, using the **Lofted plate** properties in the property pane.



- Use one construction object and a point  to create a lofted plate:
  a. Select the first construction object: line, arc, circle, or polycurve.

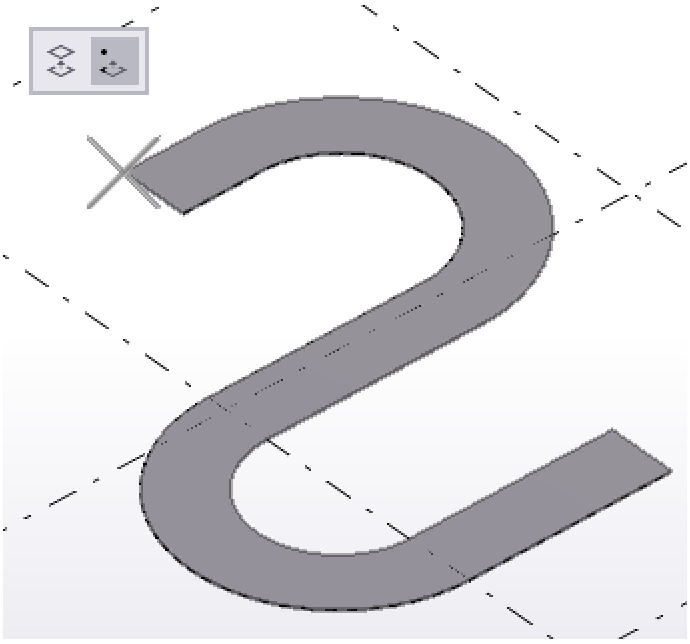
     For example, if you are using a construction polycurve and a point to create a lofted plate.

     Tekla Structures shows a preview of the part geometry. Use the preview to set the direction and size of the lofted plate.

b. Pick a point.

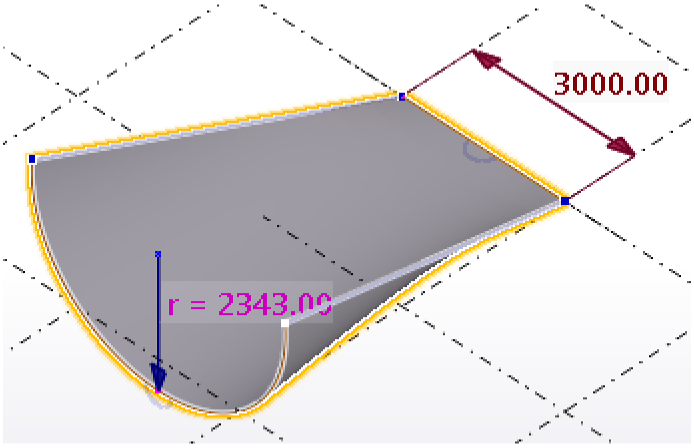
Tekla Structures creates the lofted plate based on the preview, using the **Lofted plate** properties in the property pane.

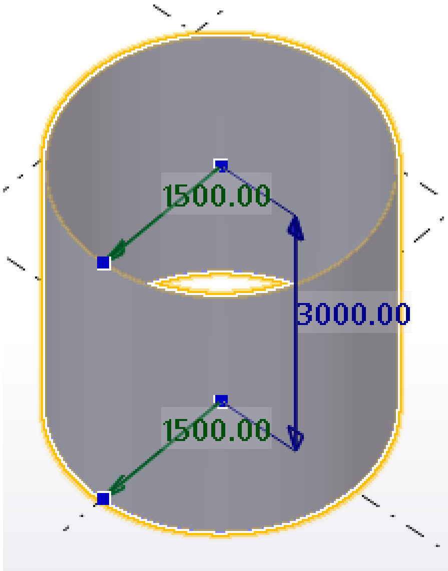


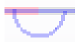
Modify the shape of a lofted plate

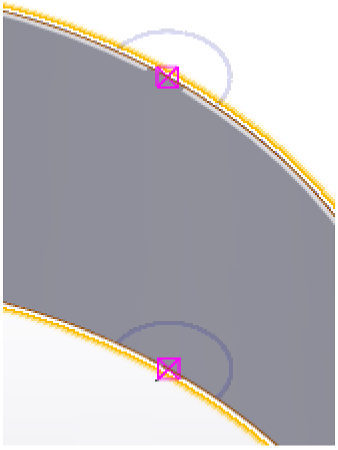
Use direct modification dimension handles and dimension values to modify the lofted plate shape.

- Modify, for example, the height and radius of the lofted plate.





- For lines and arcs: drag the arc symbol  at the midpoint of a line or an arc to modify the lofted plate shape.



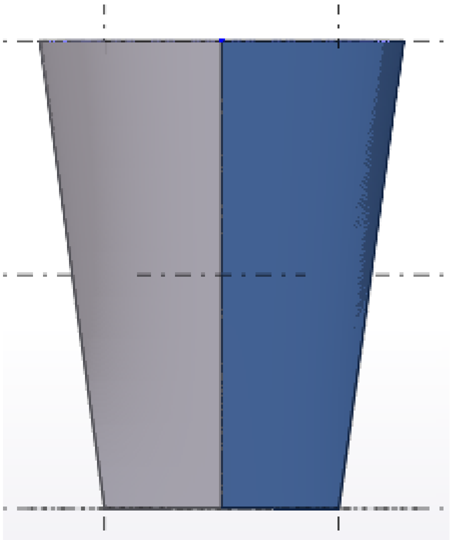
Split a lofted plate

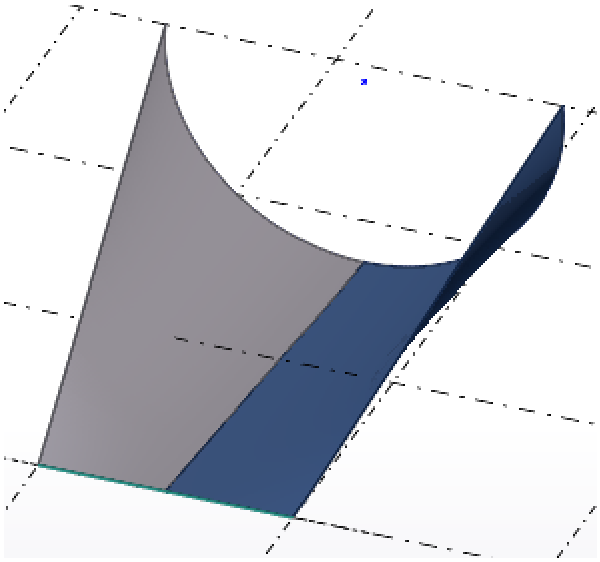
Note that you cannot split closed cylindrical or conical lofted plates.

1. On the **Edit** tab, click **Split**.
2. Select the lofted plate you want to split.
3. Pick a point for the dividing line.

   Tekla Structures splits the lofted plate.

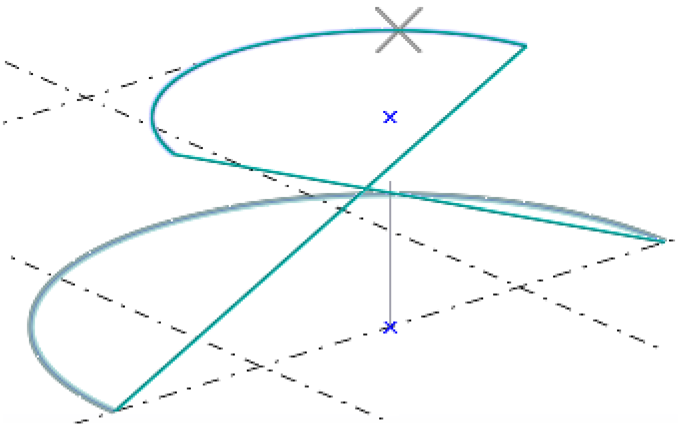
   For example:





Swap the end handle points to correct the geometry of a lofted plate

In some cases when you try to create a lofted plate, the geometry of the plate would become self-intersecting, meaning that the start and end points of the top and bottom construction object are opposite of each other. In these cases the plate is not created.



You can try to resolve the situation and create the lofted plate by changing the modeling direction of the construction lines or arcs.

1. Select the construction line or the construction arc.

   With construction lines, ensure that the **Direct modification**  switch is **not** active. You can then see the yellow and magenta object handles.

2. On the contextual toolbar, click **Swap ends**.

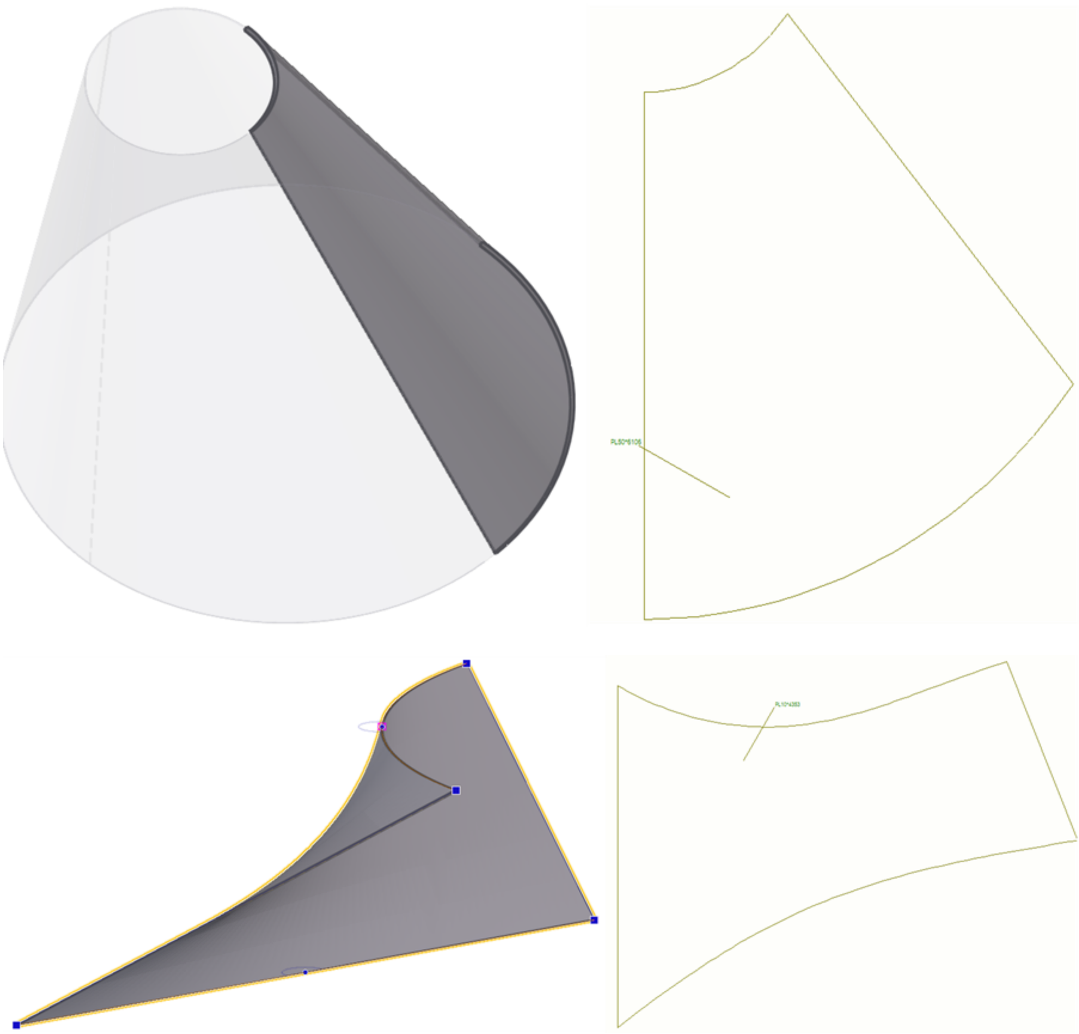
   Tekla Structures changes the modeling direction of the selected construction object, and the lofted plate can be created correctly.

With construction circles you can try to resolve the situation by moving either of the circles.

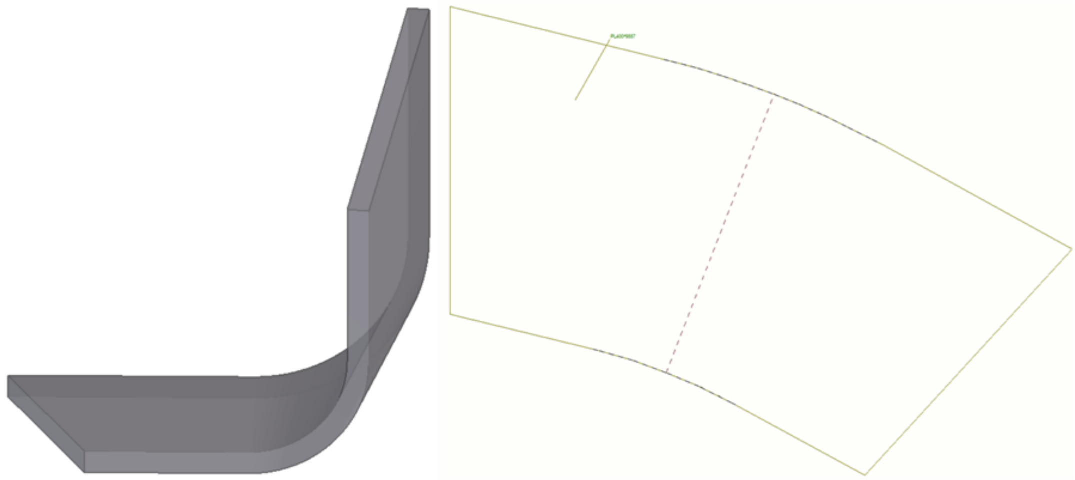
Unfold lofted plates

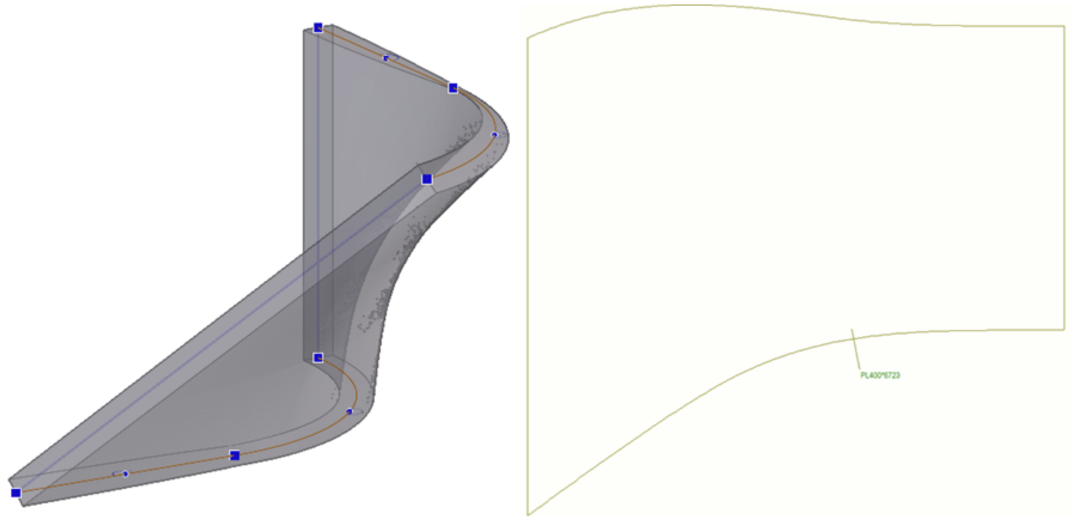
You can unfold lofted plates in single-part drawings. The unfolding works for lofted plates which have been created from single geometry to single geometry, and for tangential polycurve lofted plates.

Examples of unfolded lofted plates created from single geometry to single geometry:



Examples of tangential polycurve unfolded lofted plates:





Modify lofted plate properties

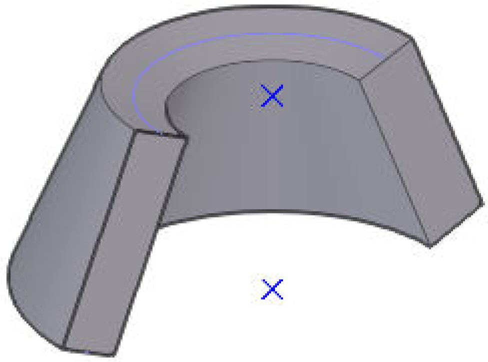
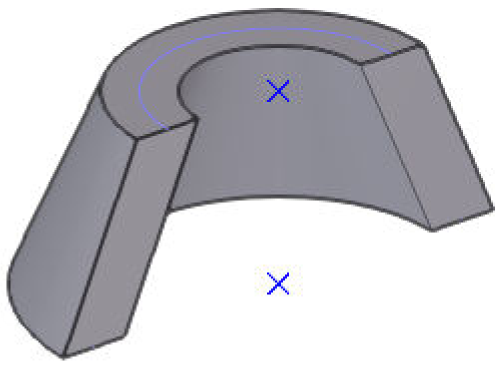
1. If the property pane is not open, double-click the lofted plate to open the **Lofted plate** properties.
2. Change the properties as needed.
3. Click **Modify**.

Lofted plate properties

Use the **Lofted plate** properties in the property pane to view and modify the properties of a steel lofted plate. To open the properties, double-click the steel lofted plate. The file name extension of a steel lofted plate property file is `*.lpl`.

If you have customized the property pane layout, the list of properties may be different.

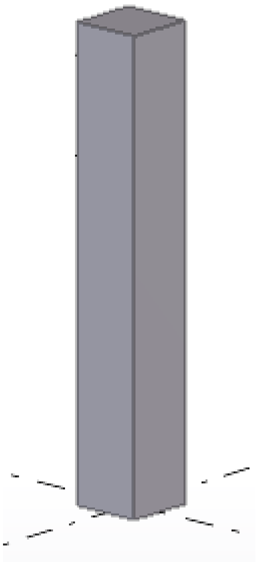
Setting	Description
General	
Name	User-definable name of the lofted plate. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 320) of the lofted plate.
Material	Material (page 322) of the lofted plate.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, e.g. with anti-corrosive paint, hot galvanized, fire retardant coating, etc.
Class	Use to group lofted plates. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 713).
Assembly numbering	Assembly prefix and start number for the assembly position number (page 713).
Face type	
Face type	Select whether the top and bottom faces of the plate are flush with the plane. **Perpendicular**:

Setting	Description
	 The top and bottom faces of the plate are non-linear.
	Bounded by curved planes:  The top and bottom faces of the plate are planar.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. 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)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the part. UDAs provide additional information about the part.

Create a concrete column

1. On the **Concrete** tab, click **Column** .
2. Pick a point.

   Tekla Structures creates the column using the **Concrete column** properties in the property pane, and at the level defined in the properties.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button and select **Concrete column** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Modify concrete column properties

1. If the property pane is not open, double-click the column to open the **Concrete column** properties.
2. Change the properties as needed.
3. Click **Modify**.

Concrete column properties

Use the **Concrete column** properties in the property pane to view and modify the properties of a concrete column. To open the properties, double-click the concrete column. The file name extension of a concrete column property file is `*.ccl`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the column. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 320) of the column.
Material	Material (page 322) of the column.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group columns. For example, you can display parts of different classes in different colors.
Position	
Vertical	Vertical position (page 315) of the column, relative to column's reference point.
Rotation	Rotation (page 313) of the column around its axis on the work plane.
Horizontal	Horizontal position (page 317) of the column, relative to column's reference point.
Top	Position of the second end of the column in the global z direction.
Bottom	Position of the first end of the column in the global z direction.
Cast unit	
Cast unit numbering	Part prefix and start number for the part position number (page 713).

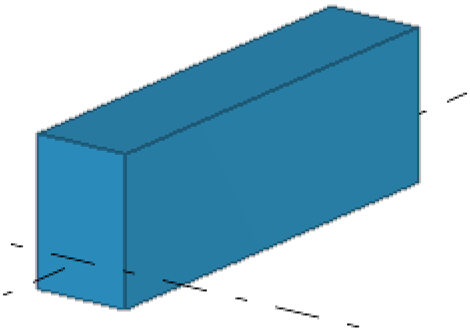
Setting	Description
Cast unit	Indicate whether the column is precast or cast-in-place.
Pour phase	Pour phase (page 450) of the cast-in-place parts. Use to separate pour objects from one another.
Deforming	
Warping	Use to warp columns using deformation angles.
Cambering	Use to pre-camber (page 334) the column.
Shortening	Use to shorten the column in the model. The true length of the column is decreased in the drawing.
Concrete covers for rebar sets	
Coordinate system	Select whether the concrete cover thickness (page 558) of the rebar sets in the part is defined in the global coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog box. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog box, define the cover thickness at each required part face.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. 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)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the

Setting	Description
	part. UDAs provide additional information about the part.

Create a concrete beam

1. On the **Concrete** tab, click .
2. Pick two points.

   Tekla Structures creates the beam between the points you picked using the **Concrete beam** properties in the property pane.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Concrete beam** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Modify concrete beam properties

1. If the property pane is not open, double-click the beam to open the **Concrete beam** properties.
2. Change the properties as needed.
3. Click **Modify**.

Concrete beam properties

Use the **Concrete beam** properties in the property pane to view and modify the properties of a concrete beam or polybeam. To open the properties, double-click the concrete beam. The file name extension of a concrete beam property file is `*.cbm`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of a beam. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 320) of the beam.
Material	Material (page 322) of the beam.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Position	
On plane	Beam's position on the work plane (page 311), relative to the beam's reference line.
Rotation	Rotation (page 313) of the beam around its axis on the work plane.
At depth	Position depth (page 313) of the beam. The position is always perpendicular to the work plane.
End offset	
Dx	Change the length of the beam (page 318) by moving the beam end point along the beam's reference line.
Dy	Move the beam end (page 318) perpendicular to the beam's reference line.
Dz	Move the beam end (page 318) in the z direction of the work plane.
Curved beam	
Plane	Plane of curvature.
Radius	Radius of the curved beam.

Setting	Description
Number of segments	Number of segments Tekla Structures uses when drawing the curved beam.
Cast unit	
Cast unit numbering	Part prefix and start number for the part position number (page 713).
Cast unit	Indicate whether the beam is precast or cast in place.
Pour phase	Pour phase (page 450) of the cast-in-place parts. Use to separate pour objects from one another.
Deforming	
Warping	Use to warp beams using deformation angles.
Cambering	Use to pre-camber (page 334) beams.
Shortening	Use to shorten beams in the model. The true length of the beam is decreased in the drawing.
Concrete covers for rebar sets	
Coordinate system	Select whether the concrete cover thickness (page 558) of the rebar sets in the part is defined in the global coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog box. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog box, define the cover thickness at each required part face.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. 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)	

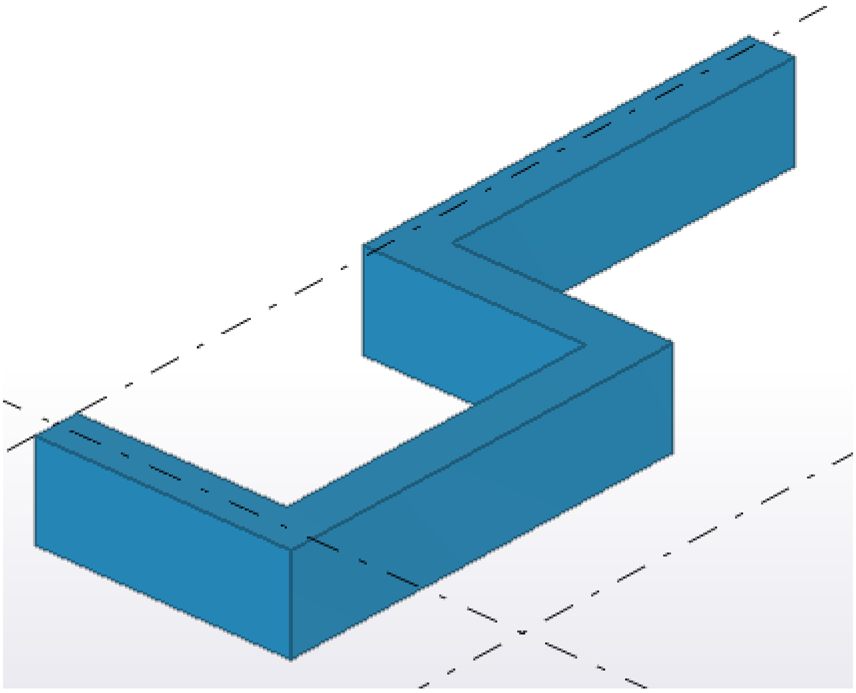
Setting	Description
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the part. UDAs provide additional information about the part.

Create a concrete polybeam

A polybeam can contain straight and curved segments.

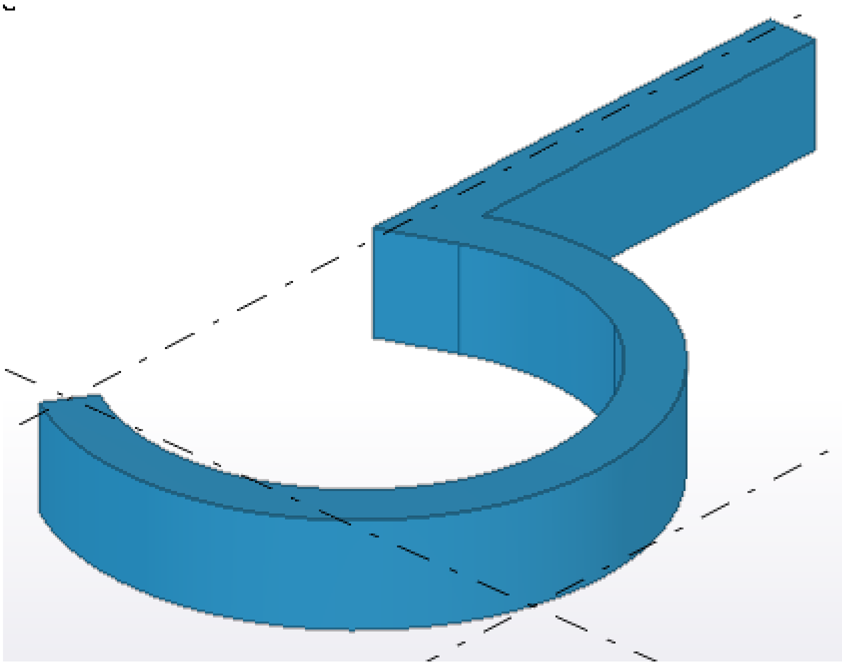
1. On the **Concrete** tab, click **Beam** --> **Polybeam**.
2. Pick the points you want the beam to go through.
3. Click the middle mouse button.

   Tekla Structures creates the beam between the points you picked, using the **Concrete beam** properties in the property pane. Note that you cannot create a closed polybeam.

4. If you want to create curved segments, chamfer the corners of the polybeam.

For example:



Modify concrete polybeam properties

1. If the property pane is not open, double-click the polybeam to open the **Concrete beam** properties.
2. Change the properties as needed.
3. Click **Modify**.

Concrete beam properties

Use the **Concrete beam** properties in the property pane to view and modify the properties of a concrete beam or polybeam. To open the properties, double-click the polybeam. The file name extension of a concrete beam property file is `*.cbm`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	

Setting	Description
Name	User-definable name of a beam. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 320) of the beam.
Material	Material (page 322) of the beam.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Position	
On plane	Beam's position on the work plane (page 311), relative to the beam's reference line.
Rotation	Rotation (page 313) of the beam around its axis on the work plane.
At depth	Position depth (page 313) of the beam. The position is always perpendicular to the work plane.
End offset	
Dx	Change the length of the beam (page 318) by moving the beam end point along the beam's reference line.
Dy	Move the beam end (page 318) perpendicular to the beam's reference line.
Dz	Move the beam end (page 318) in the z direction of the work plane.
Curved beam	
Plane	Plane of curvature.
Radius	Radius of the curved beam.
Number of segments	Number of segments Tekla Structures uses when drawing the curved beam.
Cast unit	

Setting	Description
Cast unit numbering	Part prefix and start number for the part position number (page 713).
Cast unit	Indicate whether the column is precast or cast in place.
Pour phase	Pour phase (page 450) of the cast-in-place parts. Use to separate pour objects from one another.
Deforming	
Warping	Use to warp beams using deformation angles.
Cambering	Use to pre-camber (page 334) beams.
Shortening	Use to shorten beams in the model. The true length of the beam is decreased in the drawing.
Concrete covers for rebar sets	
Coordinate system	Select whether the concrete cover thickness (page 558) of the rebar sets in the part is defined in the global coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog box. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog box, define the cover thickness at each required part face.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. 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)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined

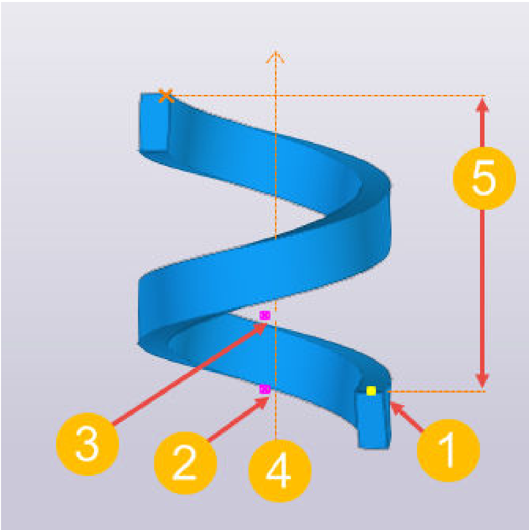
Setting	Description
	attributes (page 323) (UDAs) of the part. UDAs provide additional information about the part.

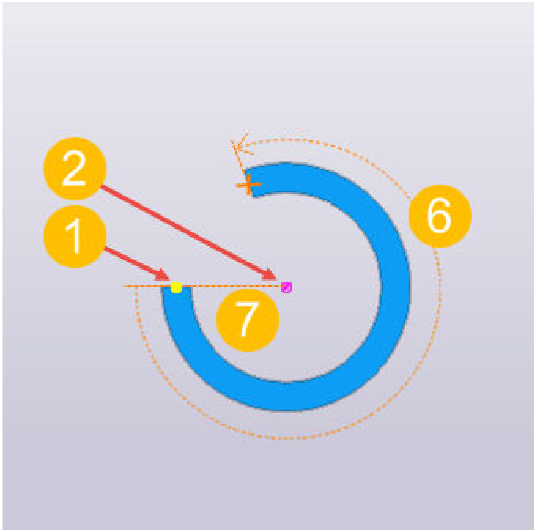
Create a concrete spiral beam

Use the **Create concrete spiral beam** command when you want to model spiral staircases, parking ramps, and complex architectural shapes, for example.

Basic concepts related to spiral beams

The images below illustrate some basic concepts related to the creation of spiral beams. Note that if you change the positioning, the entire geometry of the spiral beam changes.





(1) Start point (the first point picked)

(2) Center point (the second point picked)

(3) Direction of the rotation axis (the optional third point picked)

(4) Center axis

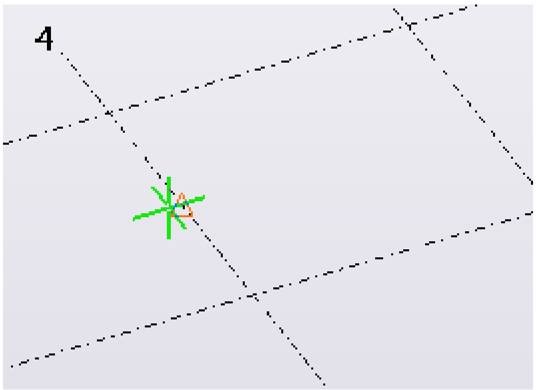
(5) Total rise: the distance from the start point to the end point, parallel to the center axis

(6) Rotation angle: the rotation angle of the spiral beam, given in degrees. Note: positive value = counter-clockwise rotation, negative value = clockwise rotation.

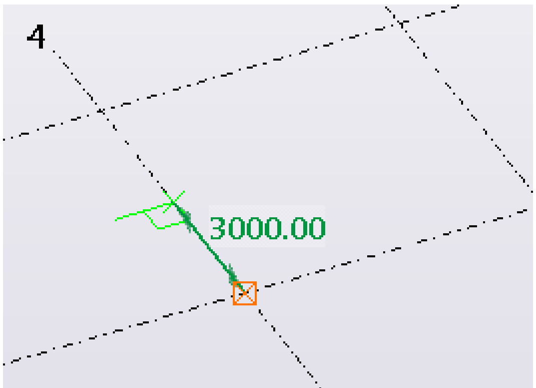
(7) Radius: the distance from the start point to the center point, perpendicular to the center axis

Create a spiral beam

1. On the **Concrete** tab, click **Beam** --> **Spiral beam**.
2. Pick the start point.



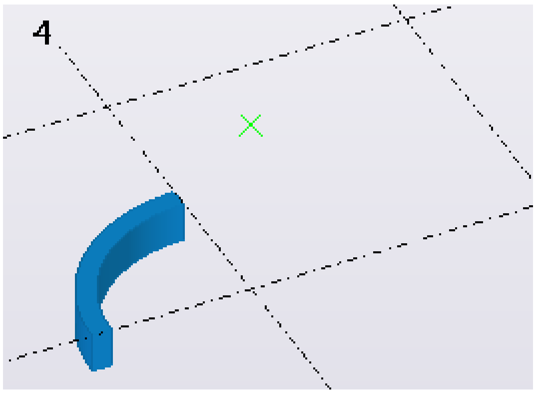
3. Pick the center point.



4. To set the rotation axis in the work plane +Z direction, click the middle mouse button to finish.

NOTE Alternatively, instead of clicking the middle mouse button, you can pick a second center axis point to define the direction of the rotation axis.

Tekla Structures creates the spiral beam. For example:



5. Click the spiral beam to select it.

   The contextual toolbar appears with the following options:

   (1) Rotation angle

   (2) Total rise

   (3) Twist angle at start

   (4) Twist angle at end

6. To add more rotation, enter a bigger value in the **Rotation angle** box.
7. To make the spiral more loose, enter a bigger value in the **Total rise** box.
8. To change the radius, move the start point or center point.

Limitations

- Spiral beam has a single, constant radius.
- Unfolding of spiral beams whose total rise is larger than 0.00 does not produce completely straight results in drawings. The amount of deviation in the part profile outlines and part length depends on several factors: the type, size, and length of the profile; the amount of total rise; and the amount of rotation angle and detailing used.
- Spiral beams are not always untwisted in unfolding. If unequal twisting is applied to the start end and end end, the unfolded drawing shows an unfolded but twisted part as a result.
- Connections and details may not work as expected with spiral beams.
- DSTV export of spiral beams may not produce a correct result.

- You cannot export spiral beams as parts in IFC export. If you are modeling cast-in-place structures with spiral beam, you can export the geometry to IFC as pour objects.

Concrete spiral beam properties

Use the **Concrete spiral beam** properties in the property pane to view and modify the properties of a concrete spiral beam. To open the properties, double-click the spiral beam. The file name extension of a concrete spiral beam property file is `*.csb`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of a beam. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 320) of the beam.
Material	Material (page 322) of the beam.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Position	
On plane	Beam's position on the work plane (page 311), relative to the beam's reference line.
Rotation	Rotation (page 313) of the beam around its axis on the work plane.
At depth	Position depth (page 313) of the beam. The position is always perpendicular to the work plane.
Geometry	
Rotation angle	Rotation angle of the spiral beam, given in degrees.

Setting	Description
Total rise	Distance from the start point to the end point, parallel to the center axis.
Twist angle at start **Twist angle at end**	Twist +/- angle of the spiral beam at the start/end of the beam.
Cast unit	
Cast unit numbering	Part prefix and start number for the part position number (page 713).
Cast unit	Indicate whether the column is precast or cast in place.
Pour phase	Pour phase (page 450) of the cast-in-place parts. Use to separate pour objects from one another.
Concrete covers for rebar sets	
Coordinate system	Select whether the concrete cover thickness (page 558) of the rebar sets in the part is defined in the global coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog box. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog box, define the cover thickness at each required part face.
IFC export	
IFC entity **Subtype (IFC4)** **User-defined type (IFC4)**	For IFC export, select the IFC entity type and subtype of the part. 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	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the

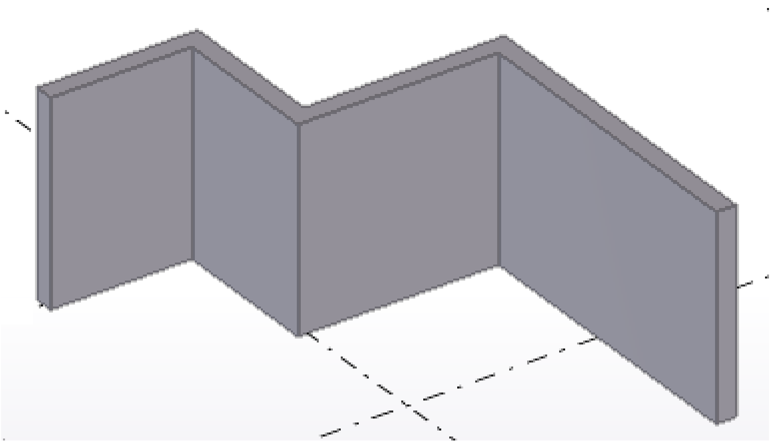
Setting	Description
	part. UDAs provide additional information about the part.

Create a concrete panel or wall

You can create a concrete panel or wall that passes through the points you pick.

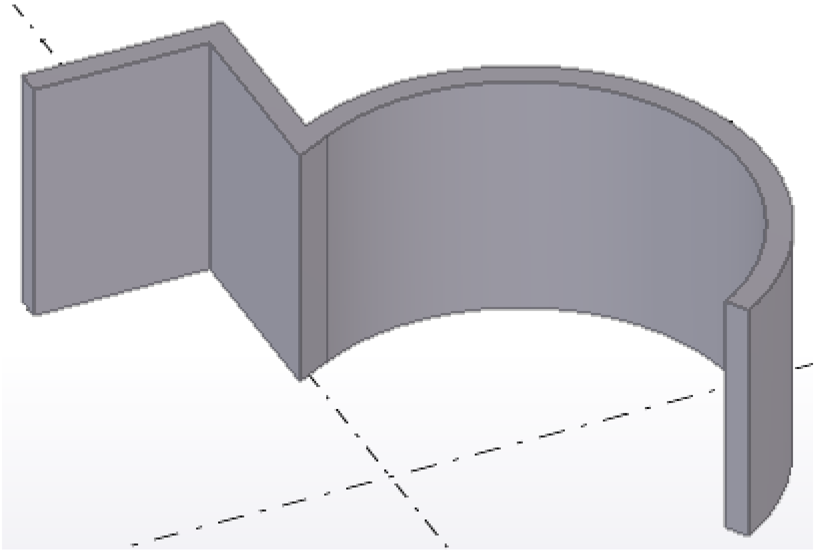
1. On the **Concrete** tab, click **Panel** .
2. Pick the points you want the panel or wall to go through.
3. Click the middle mouse button.

   Tekla Structures creates the panel or wall using the **Concrete panel** properties in the property pane. Note that you cannot create a closed panel or wall structure.

4. If you want to create curved segments, chamfer the corners of the panel or wall.

For example:



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Concrete panel** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Modify concrete panel or wall properties

1. If the property pane is not open, double-click the panel or the wall to open the **Concrete panel** properties.
2. Change the properties as needed.
3. Click **Modify**.

Concrete panel or wall properties

Use the **Concrete panel** properties in the property pane to view and modify the properties of a concrete panel or a wall. To open the properties, double-click the panel or the wall. The file name extension of a concrete panel property file is `*.cpn`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	

Setting	Description
Name	User-definable name of the panel. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 320) of the panel (thickness × height of the wall).
Material	Material (page 322) of the panel.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group panels. For example, you can display parts of different classes in different colors.
Position	
On plane	Panel's position on the work plane (page 311), relative to the panel's reference line.
Rotation	Rotation (page 313) of the panel around its axis on the work plane.
At depth	Position depth (page 313) of the panel. The position is always perpendicular to the work plane.
End offset	
Dx	Change the length of the panel (page 318) by moving the beam end point along the panel's reference line.
Dy	Move the panel end (page 318) perpendicular to the panel's reference line.
Dz	Move the panel end (page 318) in the z direction of the work plane.
Cast unit	
Cast unit numbering	Part prefix and start number for the part position number (page 713).
Cast unit	Indicate whether the panel or wall is precast or cast in place.

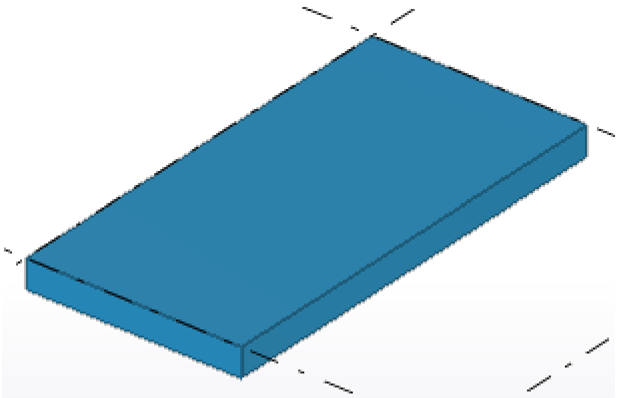
Setting	Description
Pour phase	Pour phase (page 450) of the cast-in-place parts. Use to separate pour objects from one another.
Bending	
Plane	Plane of curvature.
Radius	Radius of the curved panel.
Number of segments	Number of segments Tekla Structures uses when drawing the curved panel.
Concrete covers for rebar sets	
Coordinate system	Select whether the concrete cover thickness (page 558) of the rebar sets in the part is defined in the global coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog box. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog box, define the cover thickness at each required part face.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. 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)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the part. UDAs provide additional information about the part.

Create a concrete slab

When you create a concrete slab, the profile you select defines the thickness of the slab and the points you pick define the shape. The corners of the slab can be chamfered.

1. On the **Concrete** tab, click **Slab** .
2. Pick the corner points of the slab.
3. Click the middle mouse button.

   Tekla Structures creates the slab, using the **Concrete slab** properties in the property pane.

Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button and select **Concrete slab** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Create a round concrete slab

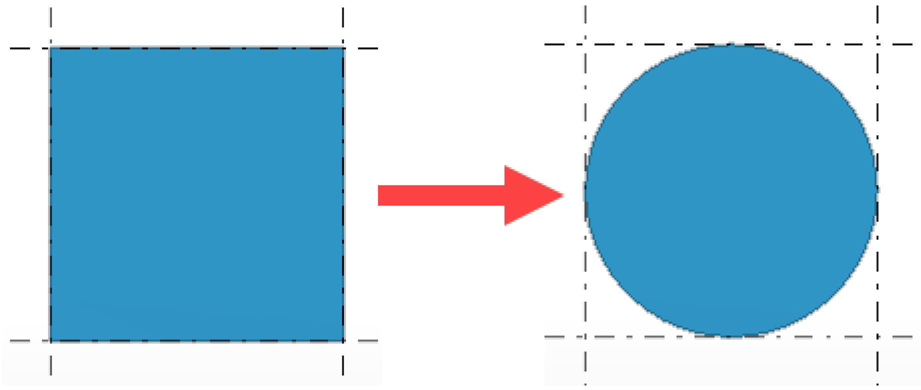
1. Create a square slab with four equal sides.
2. Select the plate.
3. Double-click a handle.

   To make it easier to select the handles of the slab corners, ensure that the Direct modification switch is **not** active.

   The **Corner chamfer** properties open in the property pane.
4. In the **Type** list, select **Rounding** .

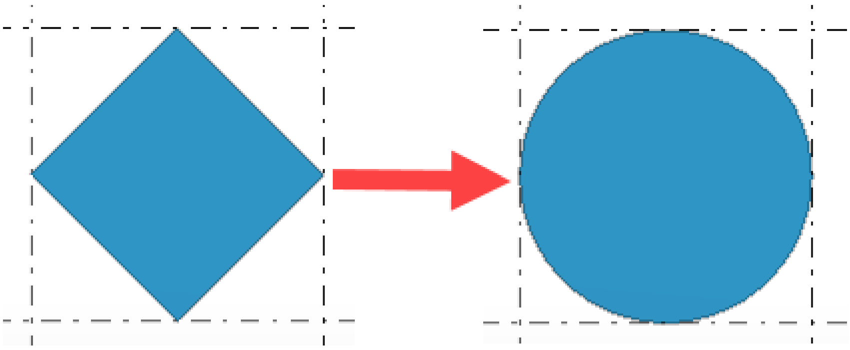
5. In the **Radius** box, enter the chamfer radius.

   The radius must be equal to half of the side of the square.
6. Click **Modify**.
7. Repeat the above steps for each corner you want to chamfer.



Alternative way of creating a round slab

1. Create a diamond-shaped slab with four equal sides.
2. To round the corners, use the **Arc point** chamfer type.



Modify concrete slab properties

1. If the property pane is not open, double-click the slab to open the **Concrete slab** properties.
2. Change the properties as needed.
3. Click **Modify**.

Concrete slab properties

Use the **Concrete slab** properties in the property pane to view and modify the properties of a concrete slab. To open the properties, double-click the concrete slab. The file name extension of a concrete slab property file is `*.csl`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the slab. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Thickness	Thickness of the slab.
Material	Material (page 322) of the slab.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group slabs. For example, you can display parts of different classes in different colors.
Position	
At depth	Position depth (page 313) of the concrete slab. The position is always perpendicular to the work plane.
Cast unit	
Cast unit numbering	Part prefix and start number for the part position number (page 713).
Cast unit	Indicate whether the slab is precast or cast in place.
Pour phase	Pour phase (page 450) of the cast-in-place parts. Use to separate pour objects from one another.
Concrete covers for rebar sets	
Coordinate system	Select whether the concrete cover thickness (page 558) of the rebar sets in the part is defined in the global

Setting	Description
	coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog box. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog box, define the cover thickness at each required part face.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. 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)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the part. UDAs provide additional information about the part.

Create a concrete lofted slab

With lofted slabs you can create curved and double-curved slabs or walls, for example.

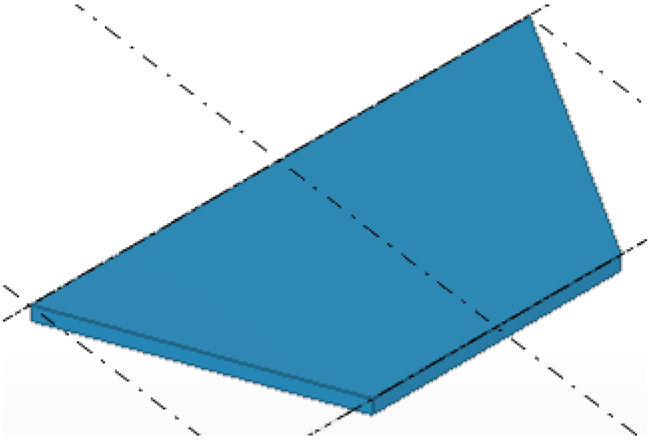
Prerequisites and examples of lofted slabs

Before you can create lofted slabs, you need to have construction objects (page 621) in your model. Tekla Structures creates the shape of the lofted slab according to the geometry of the used construction objects, by connecting the start point of the first construction object to the start point of the second construction object. The end points of construction objects are connected in similar way.

You can connect the following construction objects as a lofted slab:

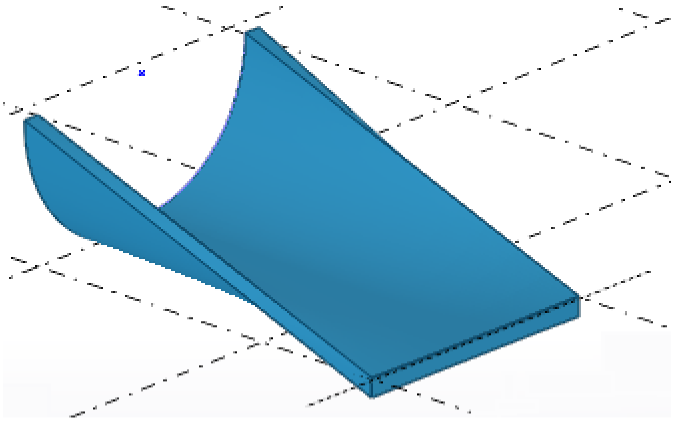
- construction line to construction line

  For example:



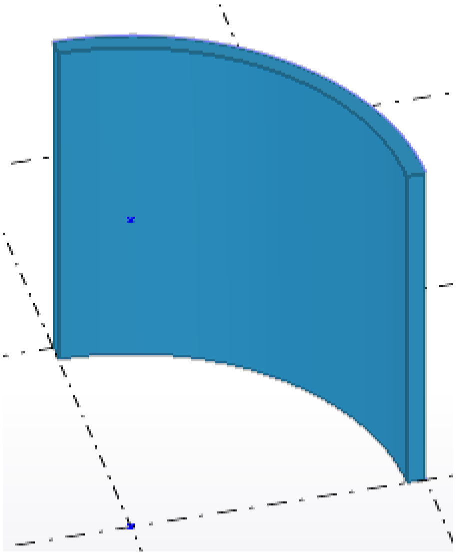
- construction line to construction arc

  For example:



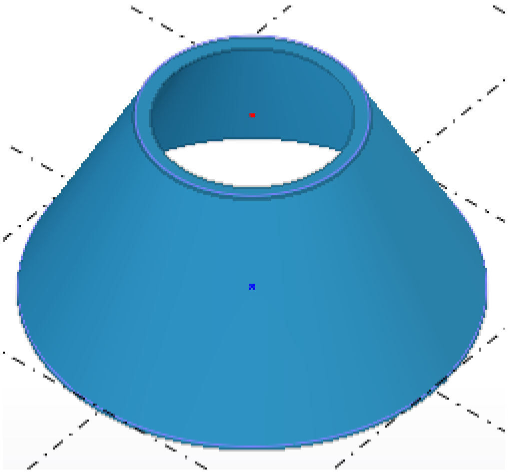
- construction arc to construction arc

  For example:

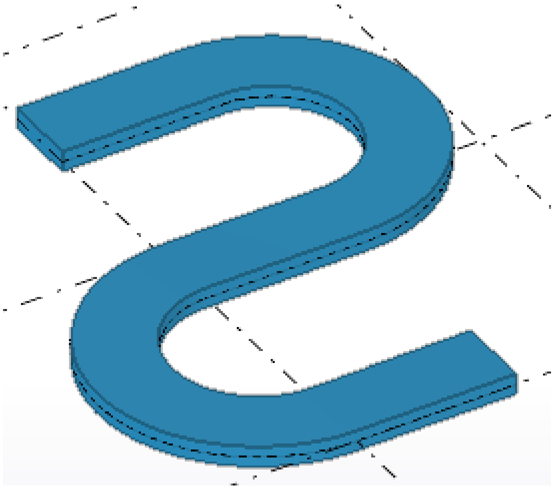


- construction circle to construction circle

  For example:



- construction polycurve to construction polycurve



Note that rebar sets do not work with lofted slabs.

Create a lofted slab

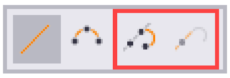
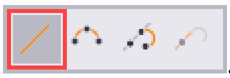
1. Create the needed construction objects in the model. The shape of the lofted slab is based on the construction objects' shape.

   You need to have

   - construction lines (page 622)
   - construction arcs (page 624)
   - construction circles (page 623)

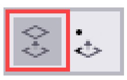
     or

   - construction polycurves (page 626)

     With polycurves, use the **Create arc by tangent** or **Create tangent line** options  on the construction polycurve toolbar. To create polycurves only with straight segments, use the **Create line** option . Note that the construction polycurves do not need to have the same number of segments, as long as both of them are tangential.

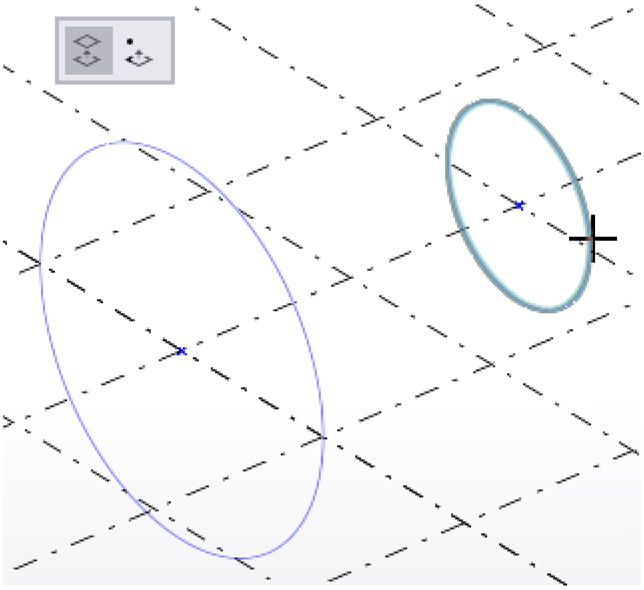
2. When you have created the needed construction objects, go to the **Concrete** tab and click **Slab** --> **Create lofted slab**.

3. On the toolbar that appears, click a button to specify whether to create the lofted slab by using two construction objects, or by using a construction object and a point.

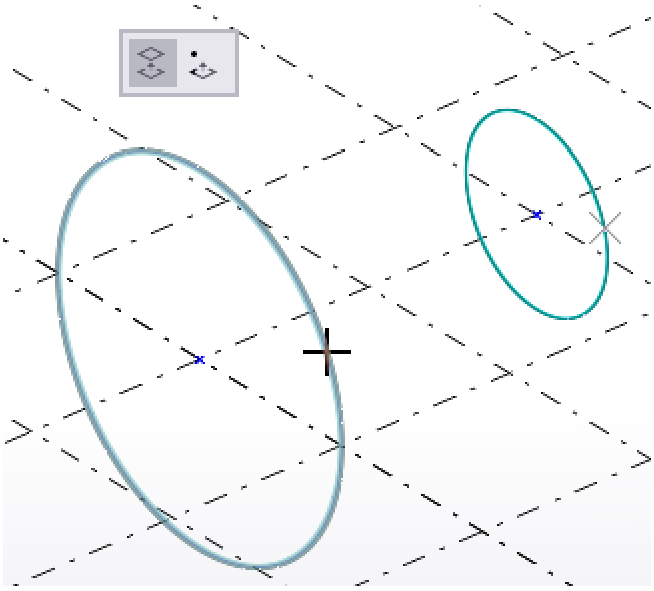
   - Use two construction objects  to create a lofted slab:

a. Select the first construction object: line, arc, circle, or polycurve.

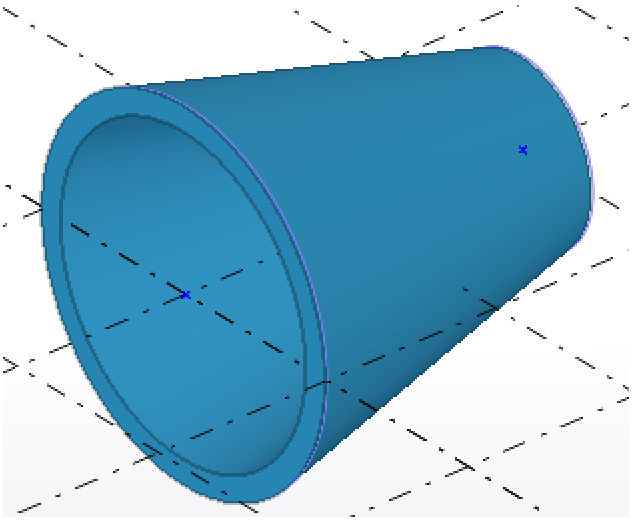
For example, if you are using two construction circles to create a lofted slab:

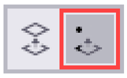


b. Select the second construction object:

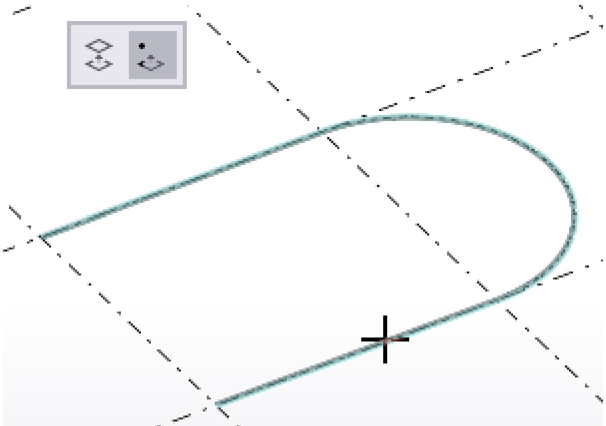


Tekla Structures creates the lofted slab between the selected construction objects, using the **Lofted slab** properties in the property pane.



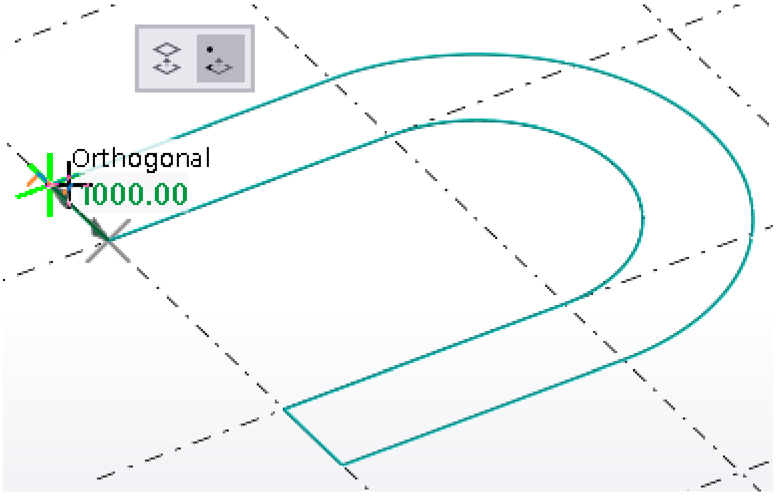
- Use one construction object and a point  to create a lofted slab:
    a. Select the first construction object: line, arc, circle, or polycurve.

       For example, if you are using a construction polycurve and a point to create a lofted slab:

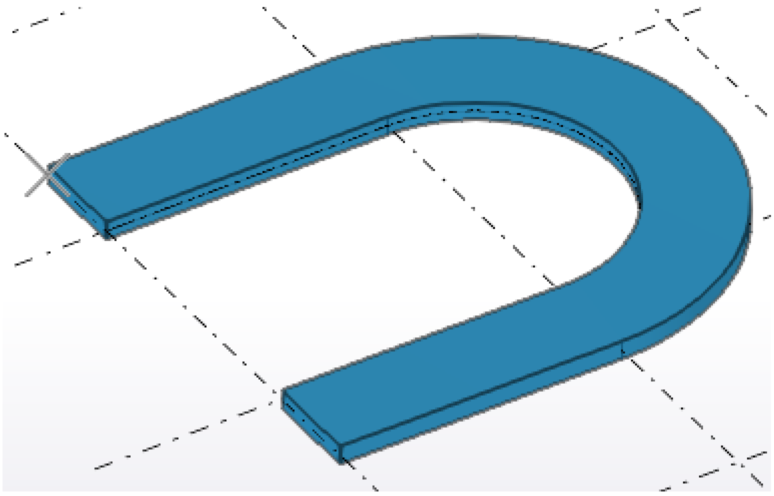
       

       Tekla Structures shows a preview of the part geometry. Use the preview to set the direction and height of the lofted slab.

b. Pick a point.



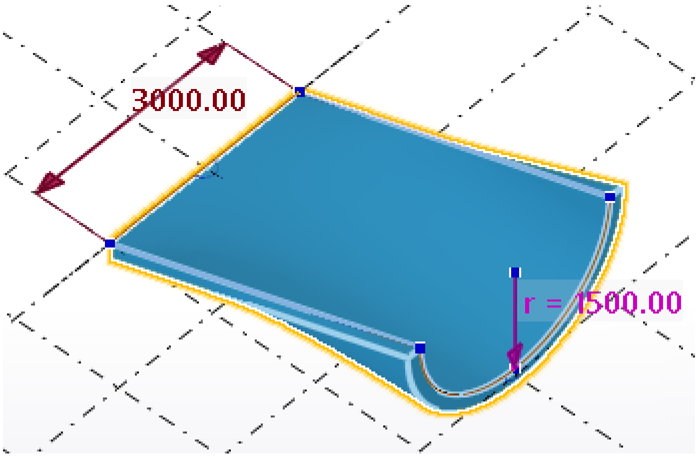
Tekla Structures creates the lofted slab based on the preview.

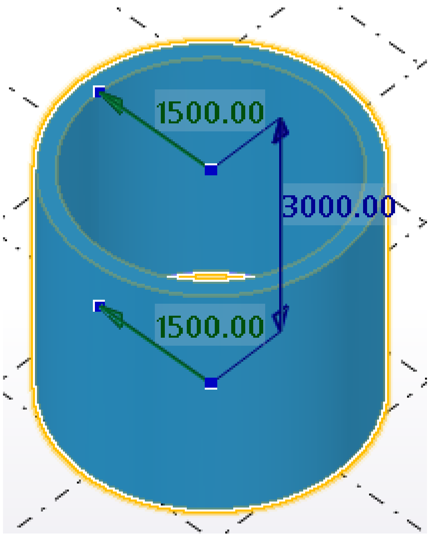


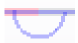
Modify the shape of a lofted slab

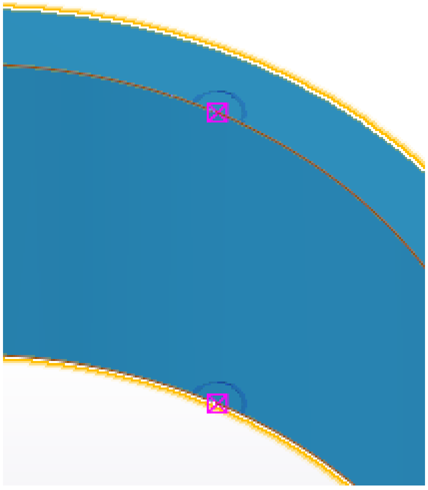
Use direct modification dimension handles and dimension values to modify the lofted slab shape.

- Modify, for example, the height and radius of the lofted slab.





- For lines and arcs: drag the arc symbol  at the midpoint of a line or an arc to modify the lofted slab shape.



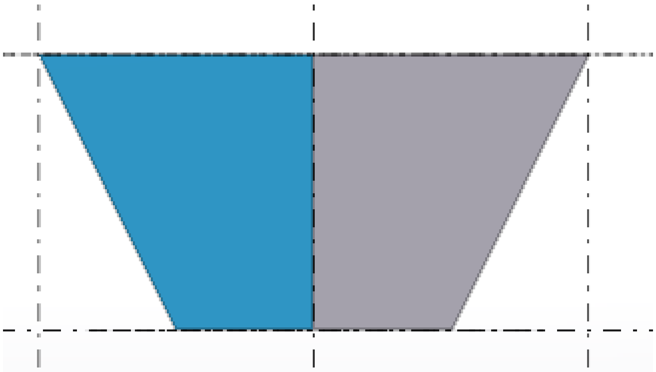
Split a lofted slab

Note that you cannot split closed cylindrical or conical lofted slabs.

1. On the **Edit** tab, click **Split**.
2. Select the lofted slab you want to split.
3. Pick a point for the dividing line.

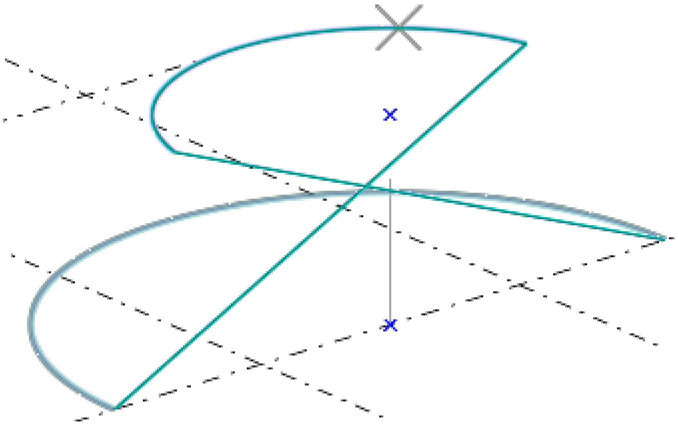
   Tekla Structures splits the lofted slab.

   For example:



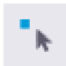
Swap the end handle points to correct the geometry of a lofted slab

In some cases when you try to create a lofted slab, the geometry of the slab would become self-intersecting, meaning that the start and end points of the top and bottom construction object are opposite of each other. In these cases the slab is not created.



You can try to resolve the situation and create the lofted slab by changing the modeling direction of the construction lines or arcs.

1. Select the construction line or the construction arc.

   With construction lines, ensure that the **Direct modification**  switch is **not** active. You can then see the yellow and magenta object handles.

2. On the contextual toolbar, click  **Swap ends**.

   Tekla Structures changes the modeling direction of the selected construction object, and the lofted slab can be created correctly.

With construction circles you can try to resolve the situation by moving either of the circles.

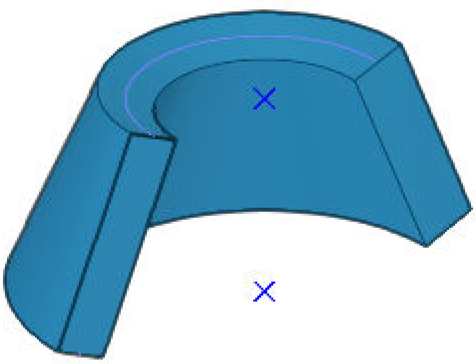
Modify concrete lofted slab properties

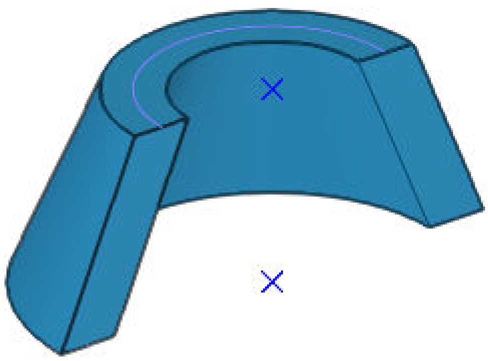
1. If the property pane is not open, double-click the lofted slab to open the **Lofted slab** properties.
2. Change the properties as needed.
3. Click **Modify**.

Lofted slab properties

Use the **Lofted slab** properties in the property pane to view and modify the properties of a concrete lofted slab. To open the properties, double-click the concrete lofted slab. The file name extension of a concrete lofted slab property file is `*.lsl`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the lofted slab. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Thickness	Thickness of the slab.
Material	Material (page 322) of the lofted slab.
Finish	Type of finish.
Class	Use to group lofted slabs. For example, you can display parts of different classes in different colors.
Face type	
Face type	Select whether the top and bottom faces of the slab are flush with the plane. **Perpendicular**:  The top and bottom faces of the slab are non-linear.

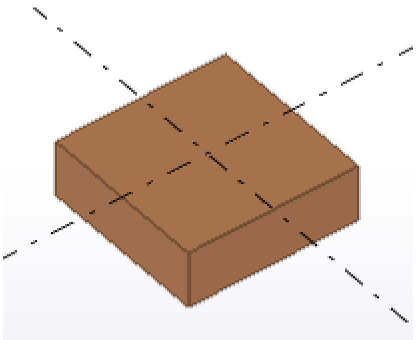
Setting	Description
	Bounded by curved planes:  The top and bottom faces of the slab are planar.
Cast unit	
Cast unit numbering	Part prefix and start number for the part position number (page 713).
Cast unit	Indicate whether the slab is precast or cast in place.
Pour phase	Pour phase (page 450) of the cast-in-place parts. Use to separate pour objects from one another.
Concrete covers for rebar sets	
Coordinate system	Select whether the concrete cover thickness (page 558) of the rebar sets in the part is defined in the global coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog box. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog box, define the cover thickness at each required part face.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. The
User-defined type (IFC4)	
More	

Setting	Description
	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	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the part. UDAs provide additional information about the part.

Create a pad footing

1. On the **Concrete** tab, click .
2. Pick a point.

   Tekla Structures creates the pad footing using the **Pad footing** properties in the property pane, and at the level (page 327) defined in the properties.

Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Pad footing** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Modify pad footing properties

1. If the property pane is not open, double-click the pad footing to open the **Pad footing** properties.
2. Change the properties as needed.

   For example, to create a circular pad footing, select a circular section for **Profile**.
3. Click **Modify**.

Pad footing properties

Use the **Pad footing** properties in the property pane to view and modify the properties of a pad footing. To open the properties, double-click the pad footing. The file name extension of a pad footing property file is `*.cpf`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the pad footing. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 320) of the pad footing.
Material	Material (page 322) of the pad footing.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group pad footings. For example, you can display parts of different classes in different colors.
Position	
Vertical	Vertical position (page 315) of the pad footing, relative to pad footing's reference point.
Rotation	Rotation (page 313) of the pad footing around its axis on the work plane.

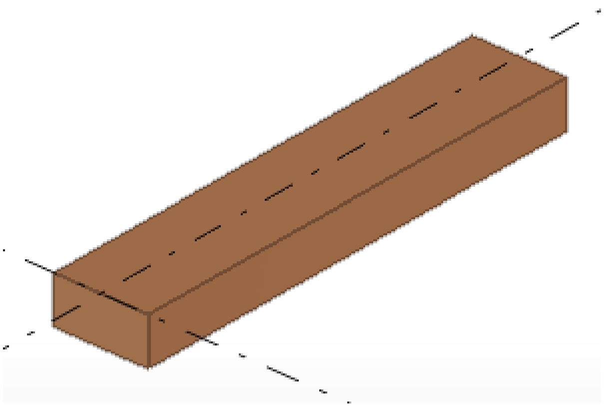
Setting	Description
Horizontal	Horizontal position (page 317) of the pad footing, relative to pad footing's reference point.
Top	Position of the top surface of the pad footing in the global z direction.
Bottom	Position of the bottom surface of the pad footing in the global z direction.
Cast unit	
Cast unit numbering	Part prefix and start number for the part position number (page 713).
Cast unit	Indicate whether the pad footing is precast or cast-in-place.
Pour phase	Pour phase (page 450) of the cast-in-place parts. Use to separate pour objects from one another.
Concrete covers for rebar sets	
Coordinate system	Select whether the concrete cover thickness (page 558) of the rebar sets in the part is defined in the global coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog box. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog box, define the cover thickness at each required part face.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. 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)	
More	

Setting	Description
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the part. UDAs provide additional information about the part.

Create a strip footing

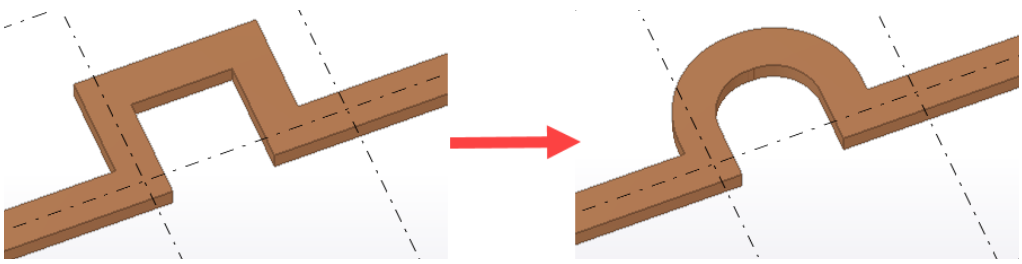
1. On the **Concrete** tab, click **Footing** --> **Strip footing**.
2. Pick the points you want the footing to go through.
3. Click the middle mouse button.

   Tekla Structures creates the strip footing between the points you picked using the **Strip footing** properties in the property pane. Note that you cannot create a closed strip footing.

4. If you want to create curved segments, chamfer the corners of the footing.

   For example:

Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.

2. In the property pane, click the **Object type list** button  and select **Strip footing** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Modify strip footing properties

1. If the property pane is not open, double-click the strip footing to open the **Strip footing** properties.
2. Change the properties as needed.
3. Click **Modify**.

Strip footing properties

Use the **Strip footing** properties in the property pane to view and modify the properties of a strip footing. To open the properties, double-click the strip footing. The file name extension of a strip footing property file is `*.csf`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the strip footing. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 320) of the strip footing.
Material	Material (page 322) of the strip footing.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group strip footings. For example, you can display parts of different classes in different colors.
Position	

Setting	Description
On plane	Strip footing's position on the work plane (page 311), relative to the footing's reference line.
Rotation	Rotation (page 313) of the strip footing around its axis on the work plane.
At depth	Position depth (page 313) of the strip footing. The position is always perpendicular to the work plane.
End offset	
Dx	Change the length of the strip footing (page 318) by moving the footing end point along the footing's reference line.
Dy	Move the strip footing end (page 318) perpendicular to the beam's reference line.
Dz	Move the strip footing (page 318) in the z direction of the work plane.
Cast unit	
Cast unit numbering	Part prefix and start number for the part position number (page 713).
Cast unit	Indicate whether the strip footing is precast or cast-in-place.
Pour phase	Pour phase (page 450) of the cast-in-place parts. Use to separate pour objects from one another.
Bending	
Plane	Plane of curvature.
Radius	Radius of the curved strip footing.
Number of segments	Number of segments Tekla Structures uses when drawing the curved strip footing.
Concrete covers for rebar sets	
Coordinate system	Select whether the concrete cover thickness (page 558) of the rebar sets in the part is defined in the global

Setting	Description
	coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog box. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog box, define the cover thickness at each required part face.
IFC export	
IFC entity **Subtype (IFC4)** **User-defined type (IFC4)**	For IFC export, select the IFC entity type and subtype of the part. 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	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the part. UDAs provide additional information about the part.

Create items

In Tekla Structures, the term *item* refers to the parts that have a 3D *shape*. Shapes are created in an external modeling software, or in Tekla Structures, and they are available in the Tekla Structures shape catalog.

Items are similar to other parts (page 196), such as beams and columns. The main difference between items and other types of parts is that a 3D shape defines the geometry of an item, whereas a part has a 2D profile that is extruded to create the length of the part.

You can use items to model objects that would otherwise be difficult to model using basic Tekla Structures parts and commands, such as cutting. You can also use items to model objects that use shapes modeled in an external software or by a manufacturer.

Every item has properties that define it, such as shape, material, and location. If you want to use item properties in view and selection filters or in drawing and report templates, you need to use the template attributes of parts and profiles. If you want to separate items from parts, use the IS_ITEM template attribute.

Limitations to items

- Items cannot be mirrored.
- Items cannot be split or combined. Splitting an imported item creates a duplicate to the splitting position.
- Items can only be cut or attached to another part if they have a solid shape.
- The gross weight value of an imported item may be different from that of an identical Tekla Structures part modeled with cuts. This is because the cuts are not taken into account when calculating the gross weight of parts.

Create an item or a concrete item

1. Ensure that **Direct modification** is switched on.

   This makes it easier to adjust the location and rotation of the item in the model.

2. Depending on the material of the item you want to create, do one of the following:

   - On the **Steel** tab, click **Item** .
   - On the **Concrete** tab, click **Item** .

TIP Alternatively, you can start the command in the property pane.

a. Ensure that you have nothing selected in the model.

b. In the property pane, click the **Object type list** button and select **Item** or **Concrete item** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

   In the property pane, you can modify the item properties and select the shape, for example.

---

3. Move the mouse pointer over object faces and edges in the model, and see how the item turns over and adjusts to the object faces.

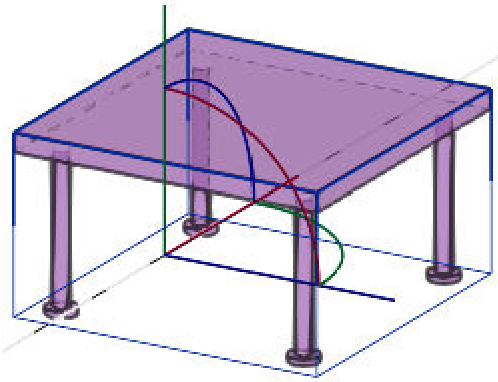
   If you are adding the item to another object (such as part or pour object), Tekla Structures also shows location dimensions to the nearest object edges.

4. Pick the first point of the item.

5. To indicate the direction of the item, pick another point.

   Tekla Structures places the item between the points you picked starting from the first point (yellow handle) towards the direction of the second point (magenta handle), using the **Item** or **Concrete item** properties in the property pane.

   Tekla Structures shows the coordinate axes, rotation handles, and location dimensions that you can use to fine-tune the location and rotation of the item. The handles are red, green, and blue, according to the local coordinate system of the item.

6. To move the item along any of its coordinate axes, drag the relevant axis handle to a new location.

7. To rotate the item around any of its coordinate axes, drag the relevant rotation handle to a new location.

   Press **Tab** to rotate the item in 90-degree steps in the direction of the selected rotation handle.

8. To move or rotate the item by specifying a distance or angle:

   a. Select an axis handle, a rotation handle, or a dimension arrowhead.

   b. Type the value by which you want the dimension to change.

      When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog box.

   c. Click **OK** to confirm the new dimension.

9. If you want to add more items to the model, click the middle mouse button and repeat steps 3–8.

10. To stop adding items, press **Esc**.

11. If needed, you can modify the item geometry (page 334) further in the **Geometry editing** mode.

Modify item or concrete item properties

1. If the property pane is not open, double-click the item or the concrete item to open the **Item** or the **Concrete item** properties.
2. Change the properties as needed.
3. Click **Modify**.

Change the shape of an item

When you create or modify an item, you can select the shape from the list that contains all the shapes available in the shape catalog.

Before you start, ensure that you have the required shape available in the shape catalog.

1. Double-click an item to open the item properties in the property pane.
2. Click the **...** button next to the **Shape** box to open the **Select shape** dialog box.
3. On the left side of the dialog box, select a group or sub-group.

   TIP The latest shapes you have used can be found in the **Recent** group, and the shapes you have marked as important are in the **Starred** group.

   ---

4. If needed, use the **Filter** box to search for a shape in the selected group.
5. Select a shape from the list.
6. Click **OK** to close the **Select shape** dialog box.
7. Click **Modify** in the property pane to apply the changes.

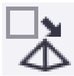
Convert a part to an item

You can change existing parts in the model to items and at the same time add corresponding shapes to the shape catalog.

When you change a part to an item, Tekla Structures deletes the original part and replaces it with the newly created item in the model. The name, material, finish, class, pour phase, and the numbering properties of the original part are saved as the corresponding item properties. Other part type specific properties and user-defined attributes are not saved. The objects that are attached to the original part, such as reinforcement and surfaces, are deleted.

1. Create the parts (page 196) that you want to change to an item.

2. If you want to include more than one part in the item, attach the parts to each other (page 330).

3. On the **Edit** tab, click  **Convert part to item**.

4. Select the part.

   Alternatively, you can first select the part, right-click, and then select **Convert part to item**.

Tekla Structures changes the part to an item and adds a new shape to the shape catalog.

The shape name is generated using the part name and part location in the format <grid location>_<elevation>_<part name>. For example:

- 1/D_+0_FOOTING
- 3/C_+0-+3600_COLUMN
- 1-2/A-B_+3600_SLAB

If there is already a shape with the same name in the shape catalog, Tekla Structures adds two underscore characters and a running number at the end of the new shape name. For example, 1/D_+0_FOOTING__1.

Item and concrete item properties

Use the **Item** and the **Concrete item** properties in the property pane to define, view, and modify the properties of an item. To open the properties, double-click the item or the concrete item.

The file name extension of an item property file is `*.ips`.

The file name extension of a concrete item property file is `*.ipc`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of an item. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.

Setting	Description
Shape	Shape of an item. To select a shape from the shape catalog, click the **...** button next to the **Shape** box. To show the item shape in reports and drawing tables, use the `PROFILE` template attribute.
Material	Material (page 322) of the item.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, e.g. with anti-corrosive paint, hot galvanized, fire retardant coating, etc.
Class	Use to group items. For example, you can display parts of different classes in different colors.
Numbering series (available for items)	
Part numbering	Part prefix and start number for the part position number (page 713).
Assembly numbering	Assembly prefix and start number for the assembly position number (page 713).
Position	
On plane	Item's position on the work plane (page 311), relative to the item's reference line.
Rotation	Rotation (page 313) of the item around its axis on the work plane.
At depth	Position depth (page 313) of the item. The position is always perpendicular to the work plane.
End offset	
Dx	Move the item along (page 318) its reference line.
Dy	Move the item perpendicular (page 318) to its reference line.
Dz	Move the item in the z direction (page 318) of the work plane.
Cast unit (available for concrete items)	

Setting	Description
Cast unit numbering	Part prefix and start number for the part position number (page 713).
Cast unit	Indicate whether the item is precast or cast-in-place.
Pour phase	Pour phase (page 450) of the cast-in-place parts. Use to separate pour objects from one another.
Concrete covers for rebar sets (available for concrete items)	
Coordinate system	Select whether the concrete cover thickness (page 558) of the rebar sets in the part is defined in the global coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog box. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog box, define the cover thickness at each required part face.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 323) (UDAs) of the part. UDAs provide additional information about the part.

2.2 Adjust part position and show part information

When you create a part, you position the part by picking points. If needed, you can adjust the position of a part in different ways after the part creation.

Part position

In part creation, the part handles and part reference line help you to define the part position. You can adjust the position of parts (page 310), such as rotation (page 313), by using the **Position** and **End offset** sections in the property pane, or by using the contextual toolbar.

Also, check the following tips that help you to create and position, for example, curved and horizontal parts:

- Create curved parts (page 325)
- Create horizontal parts (page 326)
- Create beams close to each other (page 327)
- Position columns, pad footings, and orthogonal beams (page 327)
- How to model identical areas (page 328)

Show part information

To display selected part properties in the model view, use part labels (page 324). Part labels are textual descriptions that are displayed next to the part whose properties they show.

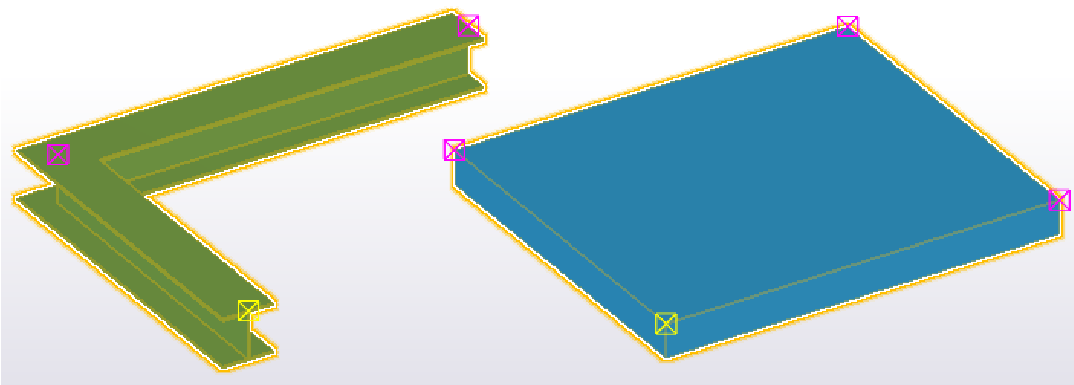
Show part handles and part reference lines in a model view

Part *handles* can be used to move a model object or to modify the model object's shape or size. Part *reference line* is line between two reference points, and the reference line has handles at the line ends.

NOTE When you create horizontal parts, such as beams, always pick points in the same direction, for example, from left to right. This ensures that Tekla Structures places and dimensions the parts in the same way in drawings, and that part marks automatically appear at the same part end.

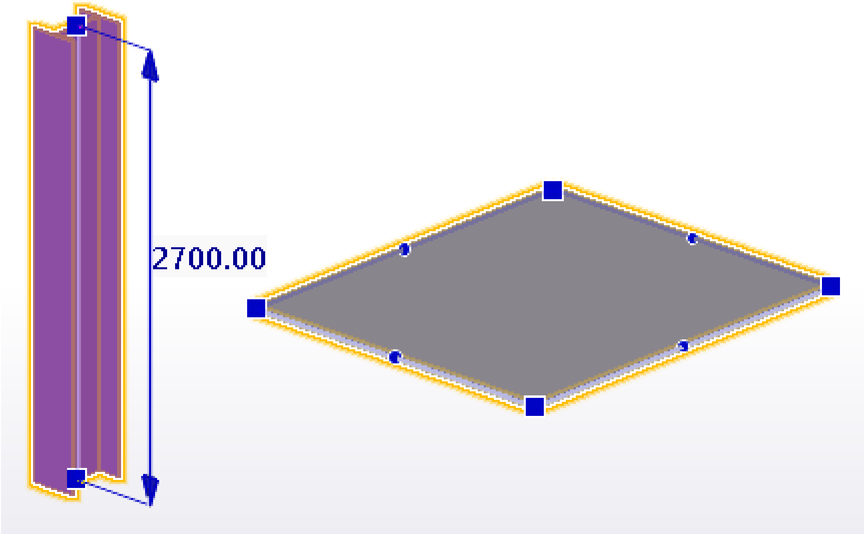
Show part handles

Tekla Structures indicates the direction of a part with handles. When you select a part, Tekla Structures shows the handles. The handle of the first end point is yellow, the rest are magenta.



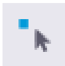
For information on how to select only the handles of a part, see Select objects (page 113).

If direct modification (page 107) is on, Tekla Structures also displays direct modification handles for the reference points, corners, segments, and segment midpoints of the selected part. These handles are blue.



Modify the length of a part using handles

If you do not want to use direct modification, you can use the part handles to modify the length of a part.

1. Ensure that the **Direct modification** switch  is **not** active.
2. Select the part.

   Tekla Structures highlights the handles of the part.
3. Click one of the handles to select it.
4. Move the handle like any other object in Tekla Structures.

   For example, right-click and select **Move**.

   If **Drag & drop** is active (page 142), just drag the handle to a new location.

WARNING Do not use cuts or fittings (page 375) to change the length of a part, for the following reasons:

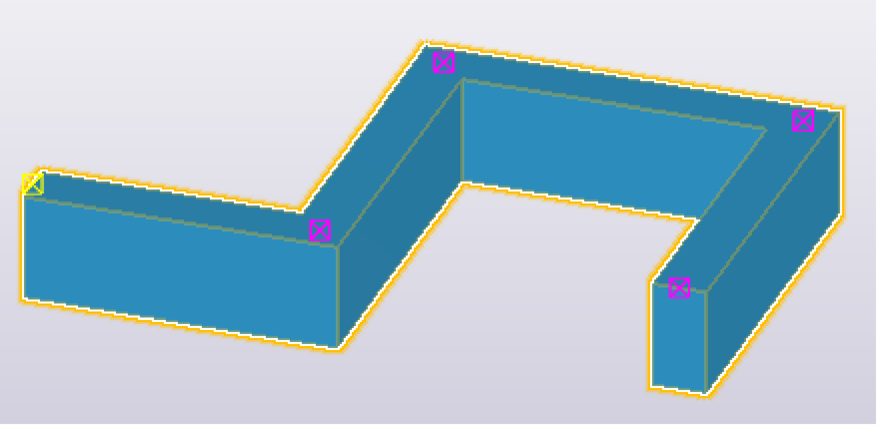
- Cuts may cause shop errors, because cuts do not always affect part length when you export information to NC files.
- Fittings may cause problems with connections and details.

Swap handles

You can change the modeling direction of a part by using the **Swap handles** macro. This changes the yellow start handle to magenta, and the other way around.

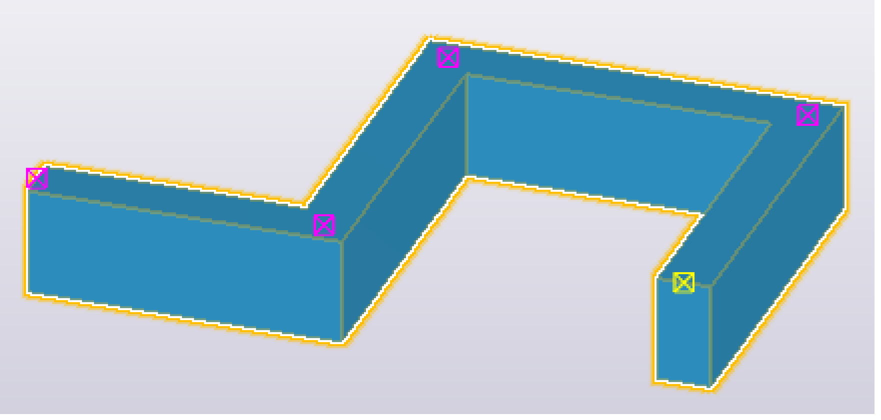
1. Select the part whose modeling direction you want to change.

   Tekla Structures shows the part handles.



2. Go to **Quick Launch**, start typing `swap handles`, and select the **Macro.Swap handles** command from the list that appears.

   Tekla Structures changes the modeling direction of the part, and swaps the start and end handles.

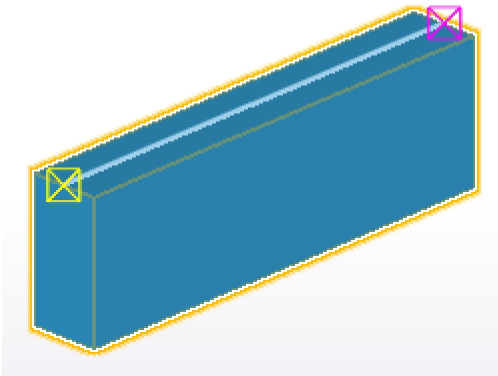


Show part reference lines in a model view

When you create a part, you position the part by picking points (page 83). These points are the part reference points. If you pick two points to position a part, the points form a part reference line, and handles appear at the line ends. By default, the part reference line is invisible in the model. It may be useful to display it when snapping to the middle points of parts, for example.

1. Double-click the view to open the **View Properties** dialog box.
2. Click **Display...** to open the **Display** dialog box.
3. On the **Advanced** tab, select the **Part reference line** check box.
4. Click **Modify**.

The part reference lines are displayed.

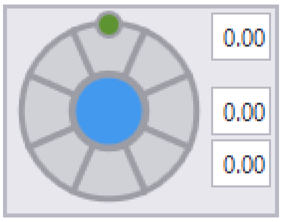
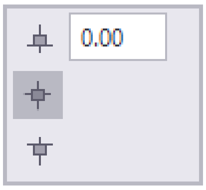


Modify the position of a part

Use the property pane and its **Position** and **End offset** sections to modify the part position. Alternatively, you can use the contextual toolbar to modify the part position.

To modify the position of a part, do one of the following:

To	Do this
Modify part position using the property pane	1. Double-click a part to open the part properties in the property pane. 2. In the **Position** or in the **End offset** section, modify the desired position settings, such as part rotation (page 313) or part vertical position (page 315). For example, you can define the part to be positioned 200 units above its handles. 3. Click **Modify**.
Modify part position using the contextual toolbar	1. Click in the contextual toolbar. 2. Modify the settings. The object moves in the model accordingly. • To change the overall position of a beam, column, panel, or footing, use the round selection dial. Click a sector in the dial to select a position.

To	Do this
	• To change the rotation angle, click and drag the green rotation angle knob. The rotation angle knob snaps to every 5 degrees. Hold down **Shift** to override this. • To change the **Angle**, **Plane offset**, or **Depth offset**, enter a value in the corresponding box.  • To change the position of a plate or slab, select an option and enter a value in the **Depth offset** box. 

Alternatively, you can use keyboard shortcuts to modify the part's position.

See also

Part position on the work plane (page 311)

Part rotation (page 313)

Part position depth (page 313)

Part vertical position (page 315)

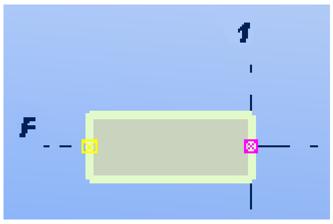
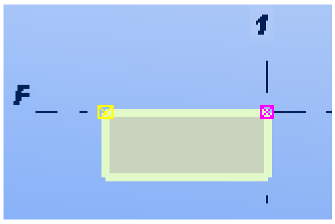
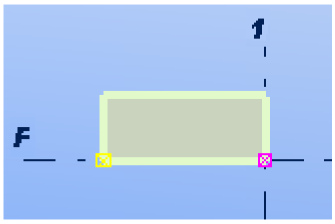
Part horizontal position (page 317)

Part end offsets (page 318)

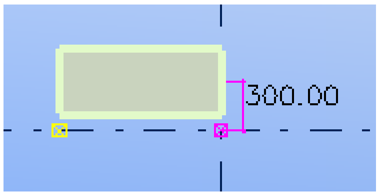
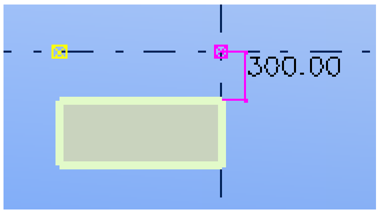
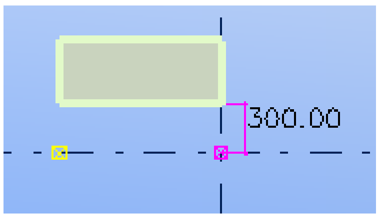
Part position on the work plane

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 (page 310) 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

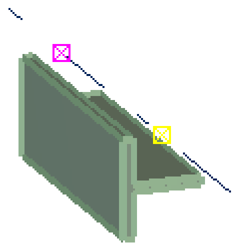
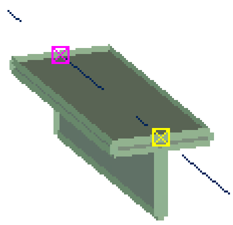
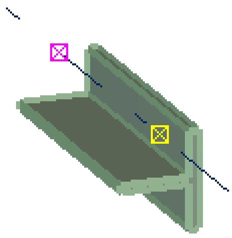
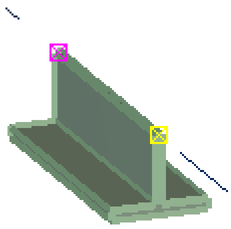
Position	Example
Middle 300	
Right 300	
Left 300	

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.

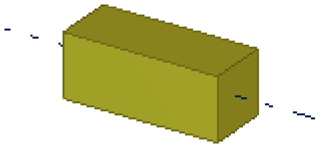
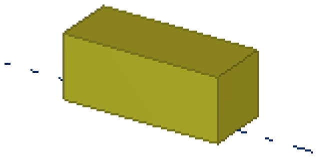
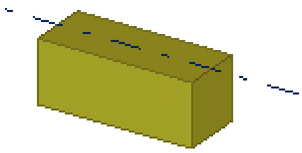
Alternatively, you can use the contextual toolbar (page 310) or keyboard shortcuts to modify the part's position.

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.	
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.	

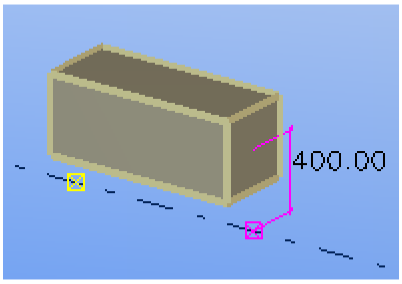
Part position depth

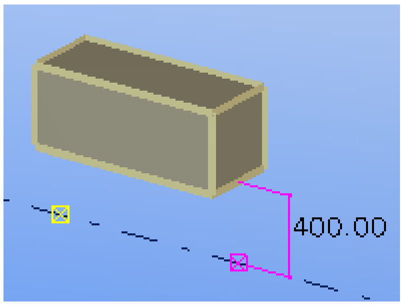
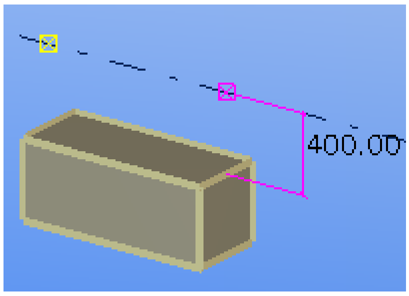
Use the **At depth** setting in the part properties to view and change the position depth of the part. The position is always relative to the part reference line between the part handles.

Alternatively, you can use the contextual toolbar (page 310) 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.	
Behind	The part is positioned underneath the reference line.	

Examples

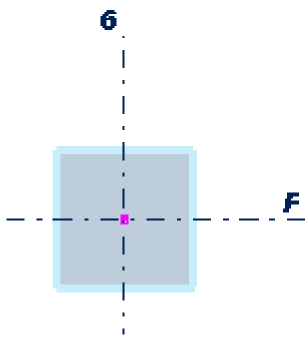
Position	Example
Middle 400	

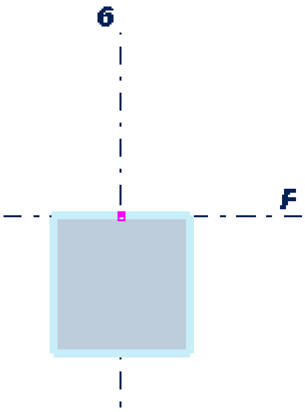
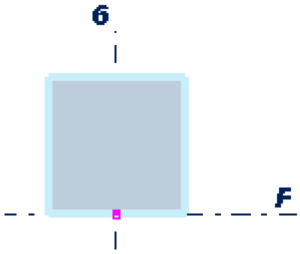
Position	Example
Front 400	
Behind 400	

Part vertical position

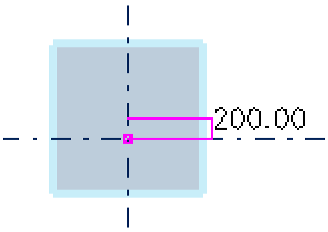
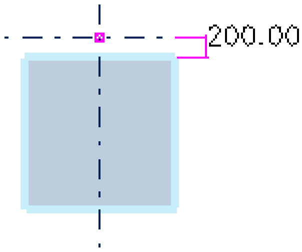
Use the **Vertical** setting in the part properties to view and change the vertical position of the part. The position is always relative to the part's reference point.

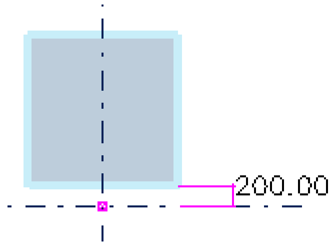
Alternatively, you can use the contextual toolbar (page 310) to modify the part's position.

Option	Description	Example
Middle	The reference point is in the middle of the part.	

Option	Description	Example
Down	The part is positioned underneath the reference point.	
Up	The part is positioned above the reference point.	

Examples

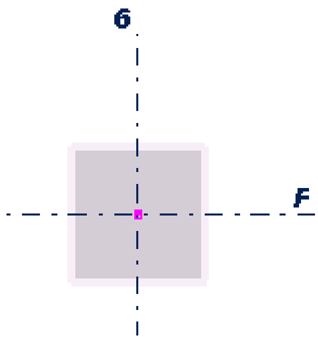
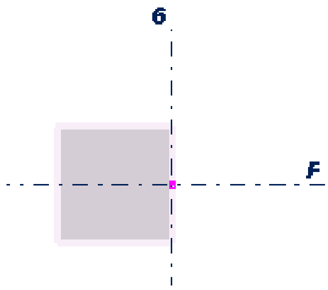
Position	Example
Middle 200	
Down 200	

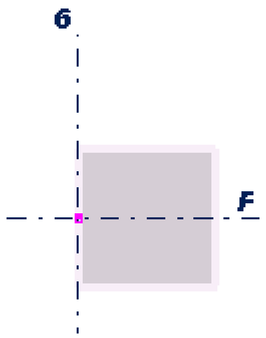
Position	Example
Up 200	

Part horizontal position

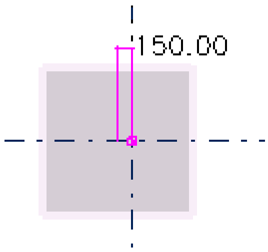
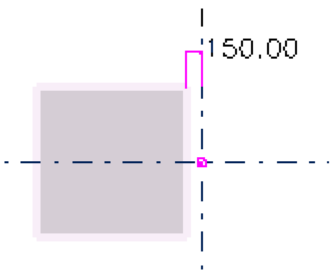
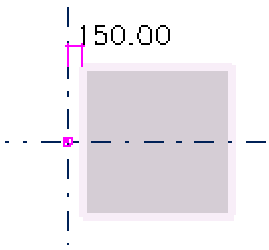
Use the **Horizontal** setting in the part properties to view and change the horizontal position of the part. The position is always relative to the part’s reference point.

Alternatively, you can use the contextual toolbar (page 310) 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	

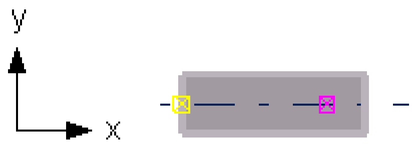
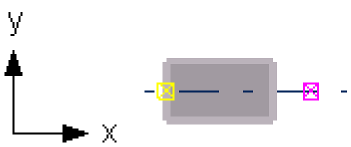
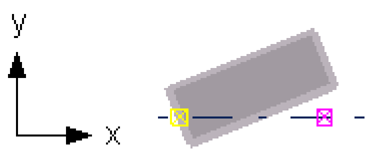
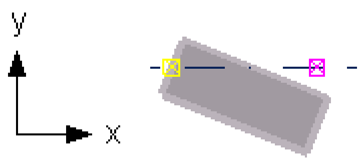
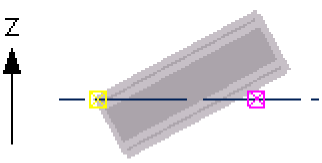
Part end offsets

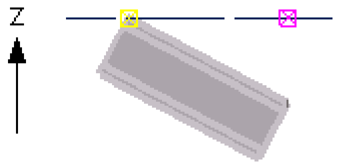
Use the **Dx**, **Dy** and **Dz** settings in the part properties to move the ends of a part, relative to its reference line. You can enter positive and negative values.

Alternatively, you can use the contextual toolbar (page 310) 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	

Select and change the profile or material of a part

Each part has a profile and a material, which are selected using the profile catalog and the material catalog.

Select and change the profile of a part

There are two types of profiles available for parts in Tekla Structures:

- *Fixed profiles*

  Fixed profiles are profiles that can be obtained pre-manufactured. The properties of the fixed profiles conform to industry standards, and you should not modify them unless you are an administrator. Fixed profiles are environment-specific.

- *Parametric profiles*

  Parametric profiles are partly user-definable: they have a predefined shape but you can change their cross section dimensions using one or more parameters. Tekla Structures calculates the cross section shape each time you open the model.

You can use the fixed or parametric profiles available in the Tekla Structures profile catalog, or you can customize the profile catalog in several ways.

Change the profile of a part

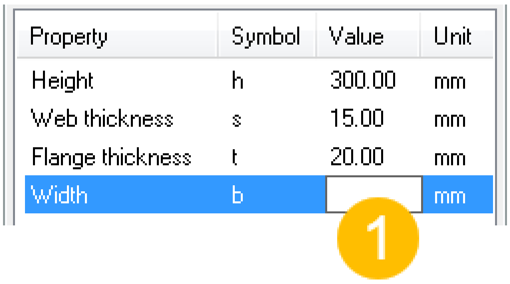
When you create or modify a part (page 196), you can select the profile of a part from a list that contains all the profiles available in the profile catalog.

1. Double-click a part to open the part properties in the property pane.
2. Click the **...** button next to the **Profile** box.

   The **Select Profile** dialog box appears.

   By default, only the profile types that are relevant to the material of the part are shown. For example, if you are changing the profile of a steel part, only the profile types that are associated with steel are shown.

3. If needed, define what profile information you want to see.
   - To display all the profiles of the profile catalog in the list, regardless of the material the profile types are associated with, select the **Show all profiles** check box.
   - To see all the properties of profiles, select the **Show details** check box.
4. Select a profile from the list.
5. If the profile is parametric, define its dimensions on the **General** tab.



   (1) Click the **Value** box and replace the existing value with a new one.
6. Click **OK** to close the **Select Profile** dialog box.
7. Click **Modify** in the property pane.

Alternatively, if you know the name of the profile, you can enter it directly in the **Profile** box in the property pane or in the contextual toolbar.

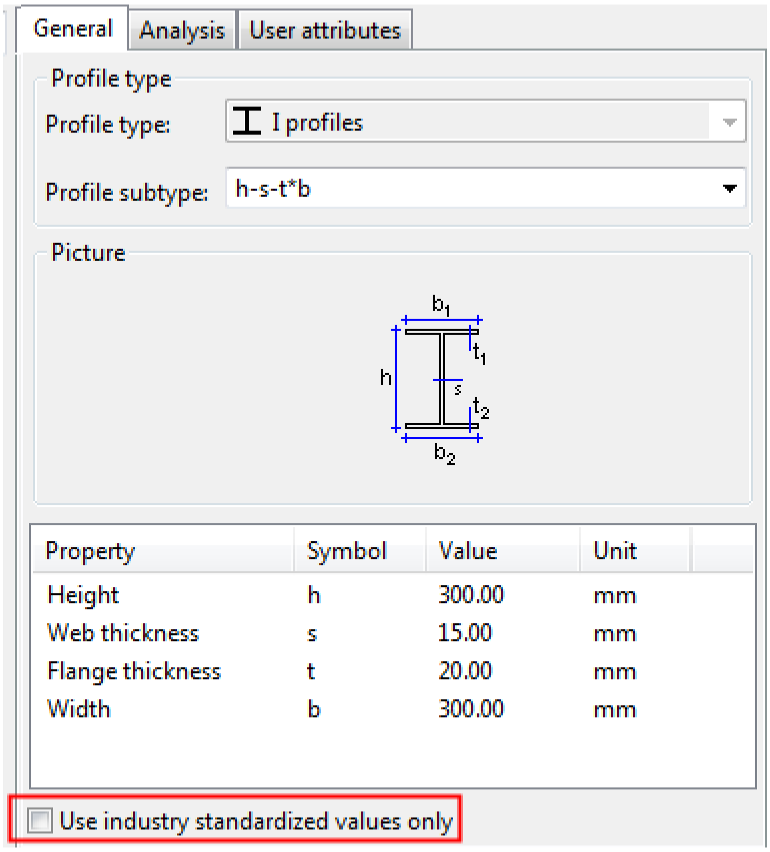
Use standardized values for profile dimensions

You can use standardized values for the dimensions of parametric profiles.

1. Double-click a part to open the part properties in the property pane.
2. Click the **...** button next to the **Profile** box.

   The **Select Profile** dialog box appears.
3. Select a parametric profile.

If standardized values have been defined for this profile, the **Use industry standardized values only** check box appears on the **General** tab under the profile properties:



4. Select the **Use industry standardized values only** check box.
5. Select the profile dimensions from a list in the **Value** column.

Select and change the material of a part

When you create or modify a part (page 196), you can select the material and grade of a part from a list that contains all the materials available in the material catalog.

1. Double-click a part to open the part properties in the property pane.
2. Click the **...** button next to the **Material** box.

   The **Select Material** dialog box appears.
3. If needed, define what material information you want to see.
   - To include aliases for material grades in the list, select the **Show aliases** check box.

     Aliases are alternative names. For example, they can be former names, or names used in different countries or standards. Tekla Structures automatically changes aliases into the standard name when you select a material grade.

- To see all the properties of materials, select the **Show details** check box.

4. Select a material from the list.
5. Click **OK** to close the **Select Material** dialog box.
6. Click **Modify** in the property pane.

Alternatively, if you know the name of the profile, you can enter it directly in the **Profile** box in the property pane or in the contextual toolbar.

TIP If needed, you can customize the material catalog.

Examples of user-defined attributes (UDAs) for parts

User-defined attributes (UDAs) provide additional information about a part. UDAs can consist of numbers, text, or lists. If needed, you can define and update user-defined attributes.

The following table gives some examples of what you can use part UDAs for:

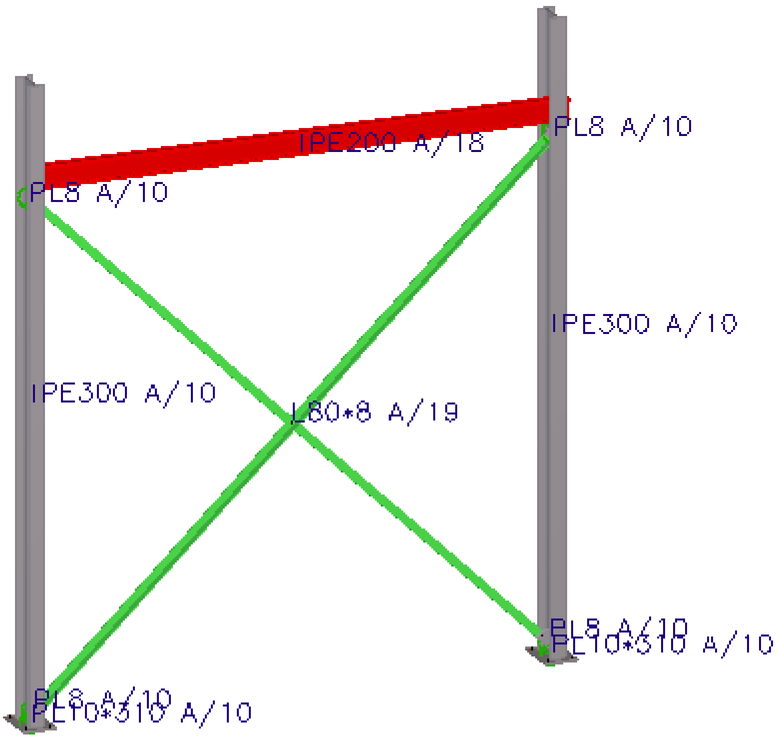
Attribute	Can be used...
Comment	In part and weld marks in Tekla Structures drawings, or in projects.
Shorten	When drawings of the parts are created, Tekla Structures decreases the true length of the part by this value. This is useful when creating assembly drawings of bracing bars that should always be under tension.
Camber	In part marks in Tekla Structures drawings.
Preliminary mark	To obtain preliminary numbers for parts in reports.
Locked	To protect objects from being accidentally changed.
Shear, Tension, and Moment	To save reaction forces for AutoDefaults. You can enter forces separately for each end of a part.
User field 1...4	User-defined fields. You can change the names of these fields and add new user-defined fields.
Connection code	When importing information on connection types into Tekla Structures. You can then use the connection codes as rules in AutoConnection and AutoDefaults. Each end of a part can have a different connection code.

Attribute	Can be used...
Moment connection	For selecting whether to show moment connection symbols in drawings or not.

Show part information by using part labels

You can display selected part properties, user-defined attributes and template attributes in a model view by using *part labels*.

Part labels are textual descriptions that are displayed next to the part they represent. You can define what information to display in the labels, such as the name, profile, and position number of the part.



1. Double-click the view to open the **View Properties** dialog box.
2. Click **Display...** to open the **Display** dialog box.
3. Go to the **Advanced** tab.
4. Next to the **Part label**, in the **Properties** list, select a property.
5. Click **Add >** to add the property to the **Part label** list. The **Part label** check box is automatically selected when you add properties.
6. To remove a property from the **Part label** list, select a property and click **Remove**. The **Part label** check box is automatically cleared if you remove all the properties.

7. If needed, define which user-defined attribute or template attribute you want to display in part labels.
    a. Select **User-defined attribute** in the **Properties** list.
    b. Click **Add >**. The **Part label** dialog box appears.
    c. Enter the user-defined attribute name exactly as it appears in the objects.inp file, or the template attribute name. For example, `PRELIM_MARK`.
    d. Click **OK**.
8. In the **Show for** list, select for which parts you want to show the part labels.
    - **All**: Part labels are shown for all parts in the view.
    - **Selected**: Part labels are only shown for the parts you select.
    - **Main part for selected**: Part labels are only shown for the main parts of the assemblies you select.
    - **Main part for all**: Part labels are shown for all main parts of all assemblies.

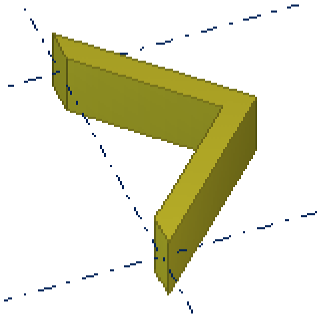
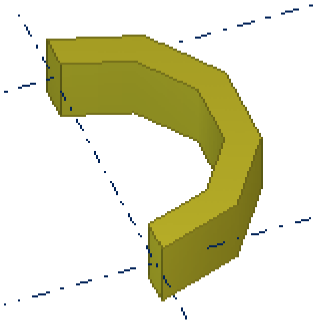
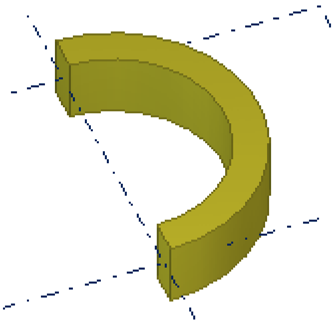
    Note that if you select the **Selected** or the **Main part for selected** option, you need to first apply the changes to the view when the view is selected. Then continue to select the objects for which you want to show the part labels.
9. Click **Modify**.

Create curved parts

You can create curved parts by defining a radius and the number of segments for a part. The number of segments determines how realistic the curved part looks: the more segments, the less angular the part appears.

1. Create a part that can be bent: a beam, panel, or strip footing.
2. Double-click the part to open the part properties.
3. Go to the **Curved beam** or **Bending** settings area, depending on the part type.
4. In the **Radius** box, enter the radius.
5. In the **Number of segments** box, enter the number of segments you want to use.
6. If needed, define the plane of curvature, which is relative to the current work plane.
7. Click **Modify** to bend the part.

Examples

Number of segments: 2	
Number of segments: 5	
Number of segments: 15	

See also

Modify the position of a part (page 310)

Create horizontal parts

When creating horizontal parts, such as beams, always pick points in the same direction. For example, pick positions from left to right, and from bottom to top (in positive x, y directions). This ensures that Tekla Structures places and dimensions the parts in the same way in drawings, and that part marks automatically appear at the same part end.

To ensure that beam rotation is correct in drawings, set part **Rotation** to **Top** in the part properties.

Create beams close to each other

When you create beams so that they are located very close to each other, Tekla Structures may consider them as a twin profile. To prevent this from happening, use the user attribute `MAX_TWIN_SEARCH_DIST` in the profile catalog.

1. On the **File** menu, click **Catalogs** --> **Profile catalog** to open the **Modify Profile Catalog** dialog box.
2. Select the desired profile in the profile tree.
3. Go to the **User attributes** tab and set the property **Twin profile detection distance** to a larger value than `0`, for example to `0.1`.
4. Click **OK**.
5. Create the beams using the profile.

See also

Create a steel beam (page 201)

Create a twin profile (page 210)

Position columns, pad footings, and orthogonal beams

For parts that you create by picking only one point (such as columns), you can define the top and bottom level of the part in the global z direction. The part is created at the defined level, **not** at the level you picked in the model. This can be useful when creating multi-story structures, as you can define exact levels for each part you create.

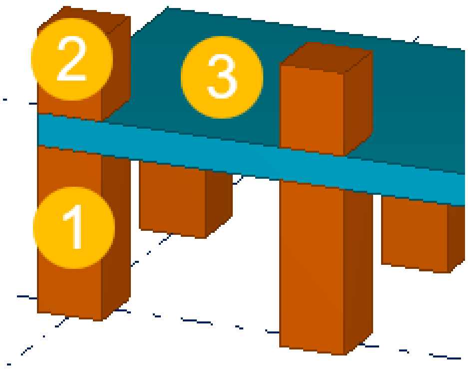
To define the top and bottom levels of a part:

1. Create a part that requires you to pick only one point.

   For example, a column.
2. Double-click the part to open the part properties.
3. Go to the **Position** settings area.
4. Modify the top and bottom levels of the part.
   - **Top**: Use to define the top level of the part.
   - **Bottom**: Use to define the bottom level of the part.
5. Click **Modify**.

Example

In this example, the concrete columns form a two-story structure. To position the upper columns correctly, you must modify their bottom level position.



(1) Top level = 1000, Bottom level = 0

(2) Top level = 1700, Bottom level = 1200

(3) Slab thickness = 200

See also

Modify the position of a part (page 310)

How to model identical areas

Most structures contain identical areas, from simple frames to entire floors. Save time by modeling these areas once, then copying them throughout the model. For example, create a column with a base plate and cap plate, then copy the column to all the locations where it occurs in the model.

You can use this technique to create and reproduce any identical area. Depending on the project, you may even be able to add connections before copying the area of the building.

TIP For a project that has several identical floors, try modeling an entire floor, then copying it to several levels.

See also

Copy and move objects (page 125)

2.3 Modify parts

This section explains how to modify different part properties, such as shape, position and length of a part. It also explains how to split and combine parts, and how to use the deforming options to warp and camber parts.

Click the links below to find out more:

Split parts

Use splitting to split a part into two. You can use splitting with straight parts, polybeams and curved beams without offsets, and normal and tapered reinforcing bar groups. You can also split contour plates and concrete slabs.

Split a straight or curved part or polybeam

1. On the **Edit** tab, click **Split**.
2. Select the part you want to split.
3. Pick a point for the splitting line.
4. If you split a polybeam, check that the following are correct:
   - The position and orientation settings of the split polybeams
   - The components related to the split polybeams

Split a plate or slab

1. On the **Edit** tab, click **Split**.
2. Select the contour plate or concrete slab you want to split.

   Note that bent plates, lofted plates, or lofted slabs cannot be split.
3. Pick the points you want the splitting line to go through.

   The first picked point and the last picked point need to be on an edge of the plate or slab.

NOTE Areas with chamfers, cuts, or openings cannot be split. If you try to pick points in these areas, the preview of the splitting line turns red, indicating that splitting is not possible.

Combine parts

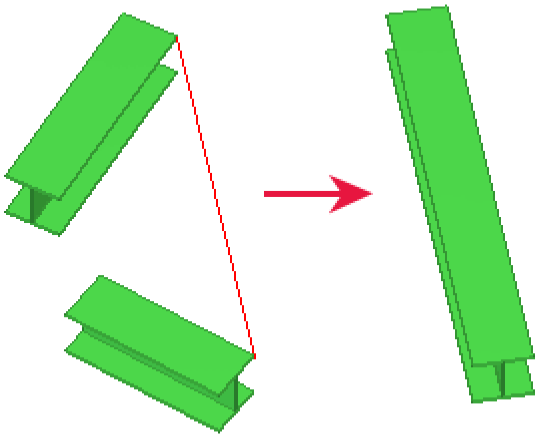
You can combine two parts into one. This can be useful when you want to model complex parts (such as folded plates) that are otherwise difficult to model, or when you want to model prefabricated parts that are delivered to the workshop already attached to profiles.

1. On the **Edit** tab, click **Combine**.
2. Select the first part.

   The properties of the first selected part will be used for the combined part.
3. Select the second part.

   The parts are combined into one.

   If the center lines of the parts are not in line with each other, Tekla Structures combines them by taking the largest distance between the start and end points from both parts. For example:



Limitations

- Combining does not work for contour plates, polybeams or slabs.
- When you combine parts, Tekla Structures retains the attached objects and connections. Tekla Structures does not recreate connections in the part that was selected first.

See also

Attach parts to each other (page 330)

Attach parts to each other

You can attach one or more parts to another part, or detach or explode the attached parts using the **Added material** commands.

When you modify the properties of attached parts, note that some of the part properties are taken from the main part. These properties are not shown in the properties of the attached part. You can inquire the properties of the whole part and the properties of each attached part separately. The attached parts are taken into account when calculating area, volume, and weight:

- **Weight (Gross)** compares the weight with fittings and without fittings, and shows the biggest weight result without cuts and with attached parts.
- **Weight (Net)** shows the weight with cuts and attached parts based on the geometry volume of the modeled part.
- **Weight** shows the net weight.

Limitations

- Connections must be added to the part to which other parts have been attached. You cannot add connections to an attached part.
- All reinforcement components may not work correctly with parts that have been attached to each other using the **Added material** commands. The geometry of the parts is not always kept suitable for adding a component. For example, the reference points of the attached part may be lost and therefore the orientation information needed for adding the reinforcement is not known anymore.

Attach a part to another part

1. Double-click the view to open the view properties, click the **Display...** button, and ensure that the **Cuts and added material** option is selected in the display settings.
2. On the **Edit** tab, click **Added material** --> **Attach to part** .
3. Select the part to attach to.
4. Select the part you want to attach.

   You can attach more than one part at a time.
5. Click the middle mouse button to attach the part.

Detach an attached part

1. Double-click the view to open the view properties, click the **Display...** button, and ensure that the **Cuts and added material** option is selected in the display settings.
2. On the **Edit** tab, click **Added material** --> **Detach from part** .
3. Select the attached part you want to detach.

   You can detach more than one part from several different parts at a time. Select the parts either by clicking them or by using area selection.

4. Click the middle mouse button to detach the part.

   The detached part keeps the color it had as an attached part.

Explode attached parts

You can explode a part that has attached parts.

1. Double-click the view to open the view properties, click the **Display...** button, and ensure that the **Cuts and added material** option is selected in the display settings.
2. On the **Edit** tab, click **Added material** --> **Explode part** .
3. Select the part you want to explode.
4. Click the middle mouse button to explode the part.

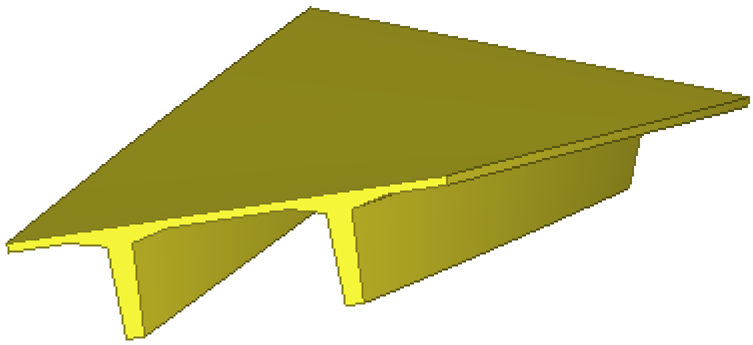
Warp a part

You can warp steel and concrete beams and columns, and concrete slabs. The warping functionality is available only in the **Full**, **Precast Concrete Detailing**, and **Steel Detailing** configurations.

Warp a beam or a column using deformation angles

1. Double-click a beam or a column to open the properties.
2. Go to the **Deforming** section.
3. In the **Warping Start** box, enter the angle of the beam at its start point, relative to the part handles.
4. In the **Warping End** box, enter the angle of the beam at its end point, relative to the part handles.

   For example, to warp the beam 10 degrees at the end point, enter `0` in the **Start** angle box and `10` in the **End** angle box.
5. Click **Modify** to warp the beam.



Warp a concrete slab by moving chamfers

Before you start, create a concrete slab by using the **Slab** command on the **Concrete** tab.

1. Double-click a chamfer to open the **Corner chamfer** properties.
2. Modify the chamfer properties.

   Do not modify the chamfers so that the slab faces are no longer planar.

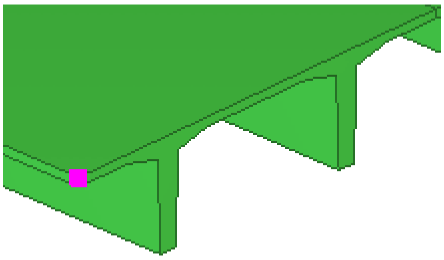
   - To move the upper corner of the chamfer, modify the **Dz1** value.
   - To move the lower corner of the chamfer, modify the **Dz2** value.
3. Click **Modify** to warp the slab.

Warp a Floor Bay (66) slab

Before you start, create a concrete slab by using the Modeling of floor bay (66) component.

1. Ensure that the **Select components** selection switch is on.
2. Select the chamfer you want to move.

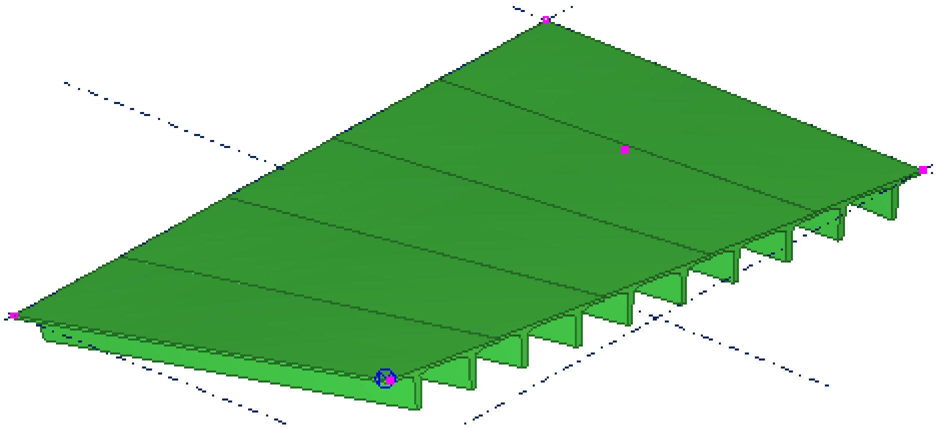
   For example, select the corner point of a slab component to warp that end of the slab:

3. Right-click and select **Move special** --> **Linear...** .
4. In the **Move - Linear** dialog box, enter a value in the appropriate direction box.

   For example, enter `100` in the **dZ** box to lift that corner up 100 mm.
5. Click **Move**.

Tekla Structures moves the point in the direction you selected, which warps the slabs.



6. Right-click and select **Interrupt**.
7. Ensure that the **Select objects in components** selection switch is on.
8. To see the warping angle of a single slab, double-click a slab to open the **Concrete beam** properties, and go to the **Deforming** section.

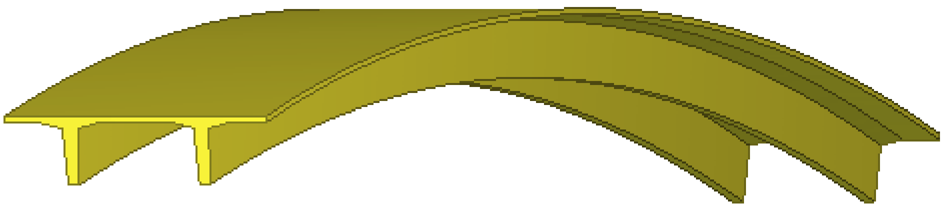
   The **Warping** start and end values show the warping angle at the start point and at the end point of the part.

Camber a part

You can use cambering to pre-camber parts, in other words, to curve long heavy sections that will settle on site and become flat. Use cambering to show the natural camber of a prestressed part in a model. Cambering affects the position of cuts, skews, and embeds in the model.

1. Double-click the part to open the part properties.
2. Go to the **Deforming** section.
3. In the **Cambering** box, define the degree of camber.
4. Click **Modify**.

   Tekla Structures cambers the part in the local z direction.



Modify items

You can fine-tune the geometry of the model by modifying items. This is how you can create sloping slabs, for example.

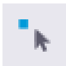
TIP You can create item shapes using existing geometry and parts in the model.

When you modify an item's geometry, you can move item vertexes (corner points), edges, and faces. However, the faces always need to stay planar, so you cannot move all vertexes or edges freely. You may first need to add new edges to split existing faces. These edges act as hinges or turning joints between the faces.

When you have modified an item's geometry and want to save the changes, you can choose to update the item's current shape, or to create a new shape. Tekla Structures also stores temporary shapes in the shape catalog in case you need to undo actions. The unused temporary shapes will be deleted from the shape catalog when you save the model.

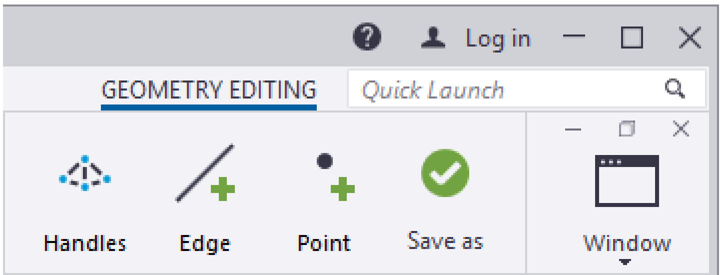
Start geometry editing

Before you start, create items (page 300).

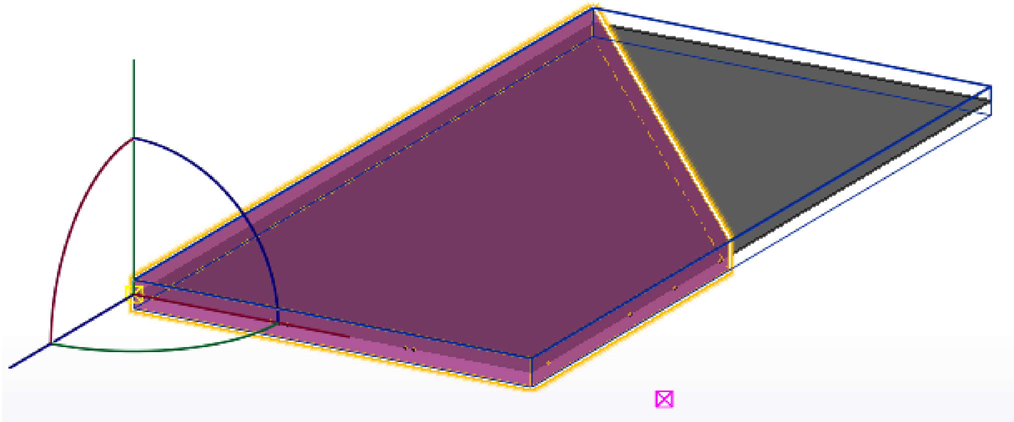
1. Ensure that  **Direct modification** is switched on.
2. Select the item whose geometry you want to modify.

   You can only modify items that have a solid shape.

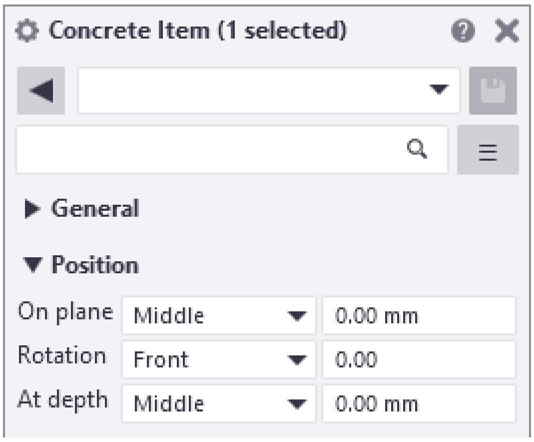
   The **Geometry editing** tab appears at the right end of the ribbon:

   Tekla Structures shows the direct modification axis and rotation handles for the selected item. For example:



3. Modify the position settings in the item properties (page 304) so that both **On plane** and **At depth** are set to **Middle** with offset 0.



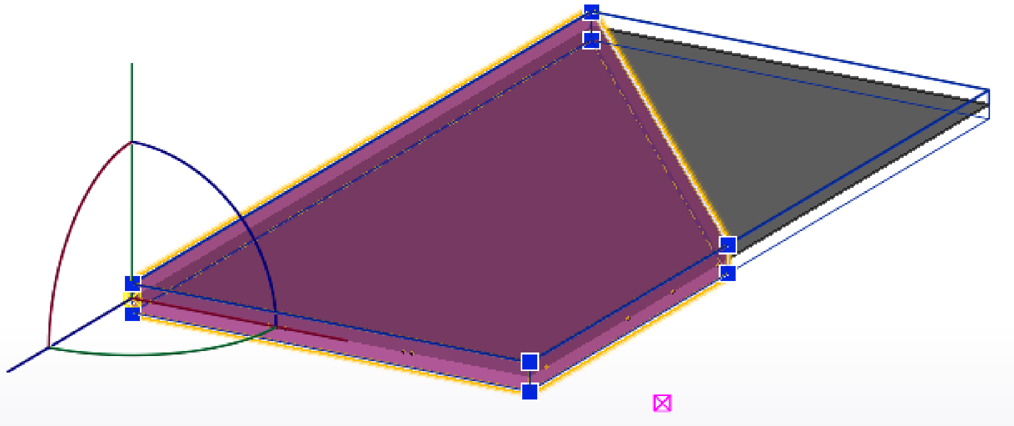
   This ensures that the later geometry modifications are accurate.

4. If you need to adjust the item location in the model, drag the direct modification axis handles or location dimension arrowheads.

Modify the geometry of an item

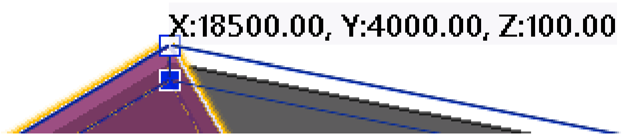
Before you start, ensure that you have the **Geometry editing** tab open and the correct item selected.

1. Click  **Handles** to display handles for the selected item. For example:



2. Move the mouse pointer over the selected item to highlight its faces, edges, and vertexes.
3. To check the coordinates of a vertex, select the vertex.

   Tekla Structures shows the absolute, work plane based coordinates in the model view. For example:



4. To move a face, edge, or vertex, drag it to a new location.

   Alternatively, you can use numeric snapping (page 87) and the **Enter a Numeric Location** dialog box.

   To move a selected vertex to an exact location, you must enter all three coordinates (x, y, and z) of the new location, for example `400,200,0`.

   Note that you can use a special character (page 91) in front of the coordinates to temporarily override the default (relative) snapping mode, for example `!` for global coordinates. With relative and absolute coordinates, you can also use coordinate axis prefixes `x`, `y`, and `z` to move a vertex or dragged edge in the prefixed directions only, for example `@z500` or `$y6000,z-500`.
5. To delete an edge or vertex, select it and press **Delete**.

   You can only delete edges whose both adjacent faces are in the same plane, and vertexes whose all adjacent faces are in the same plane.

   If the adjacent faces of an edge or vertex are not in the same plane, you can drag the edge or vertex on top of another one to combine them.
6. If you need to undo actions, press **Ctrl+Z**.

Add an edge to an item

You can add edges to the selected item, and so split existing faces. Create the new edges so that they do not cross over any existing edges.

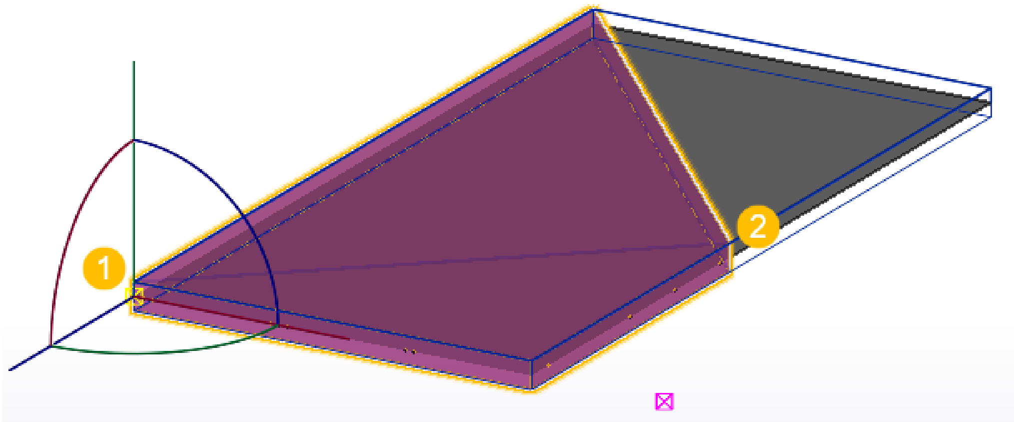
Before you start, ensure that you have the **Geometry editing** tab open and the correct item selected.

1. On the **Geometry editing** tab, click  **Edge**.
2. Pick the start point of the edge.

   The start point must be located on an existing edge or vertex.
3. Pick the end point of the edge.

   The end point must be located on a vertex or on the same face as the start point, but on another edge. If you try to pick an invalid point, Tekla Structures shows a preview of the edge in red.

   Tekla Structures creates the edge between the points you picked. For example:

   (1) Start point

   (2) End point
4. If you want to add more edges, repeat steps 2 and 3.
5. To stop adding edges, press **Esc**.

Add a vertex to an item

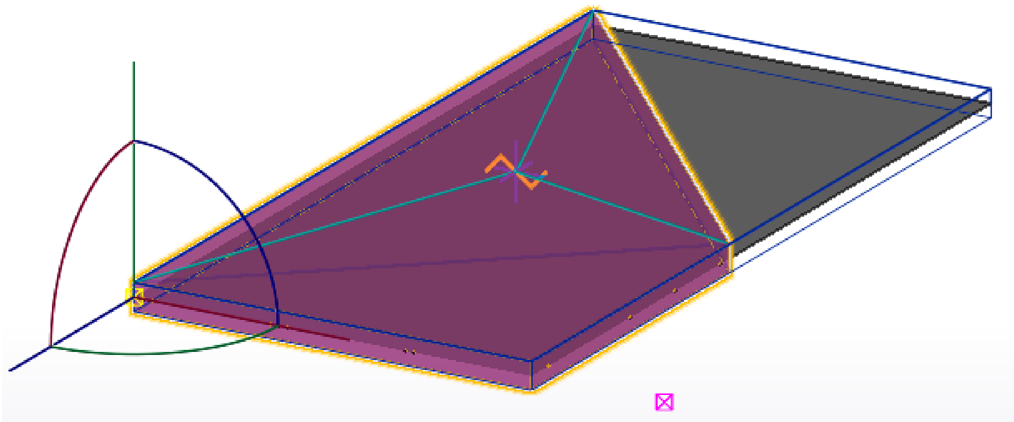
You can add vertexes and related edges to the faces of the selected item. You can use the vertexes as drainage points when you create sloping slabs and plan drainage, for example.

Note that you cannot add vertexes to faces that have openings or recesses if these openings are part of the item **shape**. However, if only the item in the model has been cut, and not its shape, you can add vertexes to these faces.

Before you start, ensure that you have the **Geometry editing** tab open and the correct item selected.

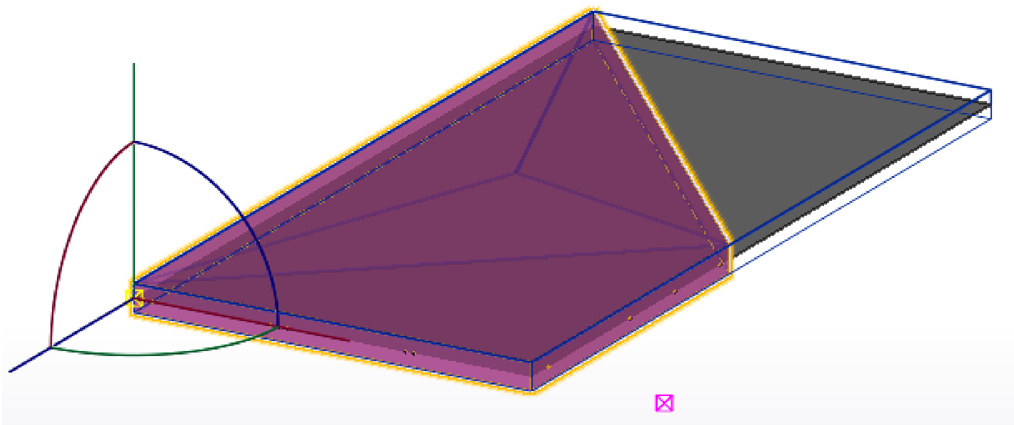
1. On the **Geometry editing** tab, click  **Point**.
2. Move the mouse pointer over the faces of the selected item.

   In convex-polygonal areas, where all internal angles at face corners are less than 180 degrees, Tekla Structures shows a preview of the edges that start from each face corner and join at the current location of the mouse pointer. For example:

   If the item faces have concave-polygonal areas, where at least one internal angle is larger than 180 degrees, Tekla Structures automatically shows and adds edges that separate the concave-polygonal areas from the convex-polygonal areas.
3. Pick a point on the desired face.

Tekla Structures creates the vertex and related edges that join the new vertex to all existing vertexes of the same face. For example:



4. If you want to add more vertexes, pick more points.
5. To stop adding vertexes, press **Esc**.

Save a modified item and shape

When you save modified items in the model, you can choose to update the selected item's current shape, or to create a new shape for later use.

Before you start, ensure that you have the **Geometry editing** tab open and the correct item selected.

1. On the **Geometry editing** tab, click  **Save as**.

   The **Save as** dialog box opens.
2. Select one of the following options:
   - **Update current shape to use this geometry**

     to update the current shape, the modified item, and all the items using that shape in the model.

     If all items are not updated in the model, save and re-open the model.
   - **Create a new shape in the shape catalog with the name**

     to create a new shape in the shape catalog, and to update the modified item in the model to use the new shape.
3. If you chose to create a new shape, enter a name for the shape.

4. Click **Save**.

   Tekla Structures updates the items in the model and saves the shape in the shape catalog.

The temporary shapes that were created during the editing process will be deleted from the shape catalog when you save the model if the shapes are not used by any item.

2.4 Add details to parts

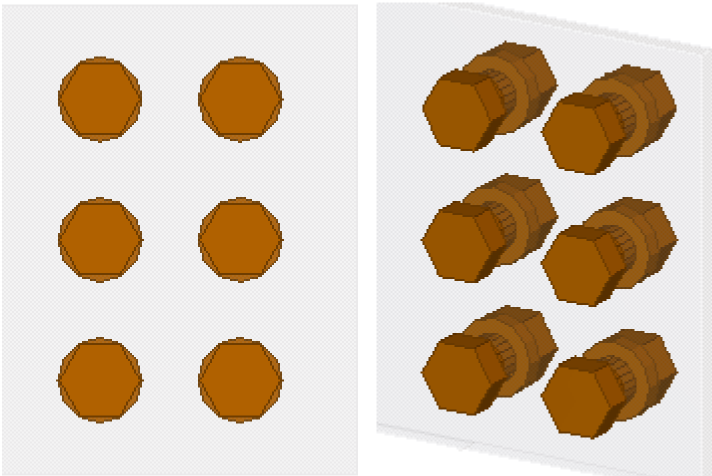
This section explains how to create and view details using Tekla Structures. It also gives you some techniques for fine-tuning the part shape.

You can create and view details, for example, the following ways:

- create bolts (page 341), studs (page 353) and holes (page 353)
- create welds (page 359) and define how they appear in the model (page 371)
- create fittings (page 375), cuts (page 389) and part chamfers (page 397) to refine the shape of a part, and to fit beams and columns (page 376) and also contour plates, slabs, and panels (page 380) to each other
- add surface treatment to parts (page 402) and surfaces to part faces (page 415)
- modify adaptivity (page 416) of the details
- show all the detailing that belongs to a part (page 416)

Create bolts

To create bolts, you can either create a single bolt group or apply a component that automatically creates bolt groups.



Tekla Structures uses the same command for creating bolts, studs (page 353) and holes (page 353). If you want to create only holes, do not use any bolt elements (such as bolts, washers, and nuts).

If you want to create bolts without creating holes, use the **No hole** option for **Special hole type** in the **Bolt** properties.

You can create different marks for bolts and holes in drawings.

Create a bolt group

1. On the **Steel** tab, hold down **Shift** and click **Bolt** .

   The **Bolt** properties open.

2. Modify the **Bolt** properties, if needed.

   For example, the **Bolt group** settings affect the end result: the shape of the bolt group, the number of bolts in the group, and the spacing of the bolts. See the **Bolt group shape** section below for examples.

   To define the hole properties, use the **Holes** settings in the **Bolt** properties.

3. Select the main part, to which the secondary parts will be bolted.
4. Select the secondary parts.
5. Click the middle mouse button to finish selecting parts.
6. Pick a point to indicate the bolt group origin.
7. Pick a second point to indicate the direction of the bolt group x axis.

   NOTE Tekla Structures determines the location of the bolt group using the following values: the bolt group x axis and the work plane. Dimensions are relative to the bolt group origin, which is the first point picked. Tekla Structures sets the x direction of the bolt group using the second point picked. It is important that the points you pick to create the bolt group are close enough to the parts you want to connect.

Create a single bolt

1. On the **Steel** tab, hold down **Shift** and click **Bolt** to open the **Bolt** properties.
2. Under **Bolt group**, select **Array** from the **Shape** list.
3. In the **Bolt dist X** and **Bolt dist Y** boxes, enter `0`.
4. Create the bolt the same way you would create a bolt group:
   a. Select the main part, to which the secondary parts will be bolted.
   b. Select the secondary parts.
   c. Click the middle mouse button to finish selecting parts.
   d. Pick a point to indicate the bolt origin.
   e. Pick a second point to indicate the direction of the x axis.

Create bolts using the Auto bolt component

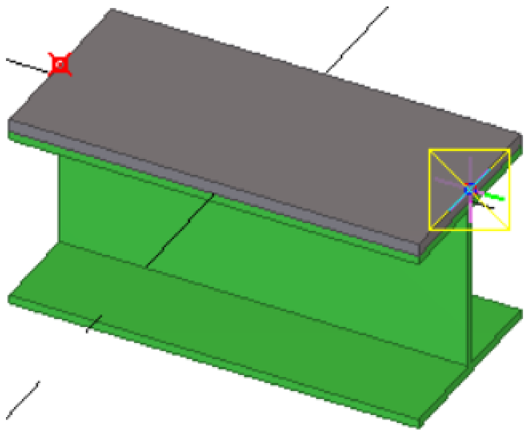
Use the **Auto bolt** component to bolt parts and nearby parts, shim plates, splice plates, or other plates. **Auto bolt** follows the part rotation and finds the best rotation so that you do not need to set the work plane. With **Auto bolt** one bolt group can span many parts, for example, manage a splice as a single group.

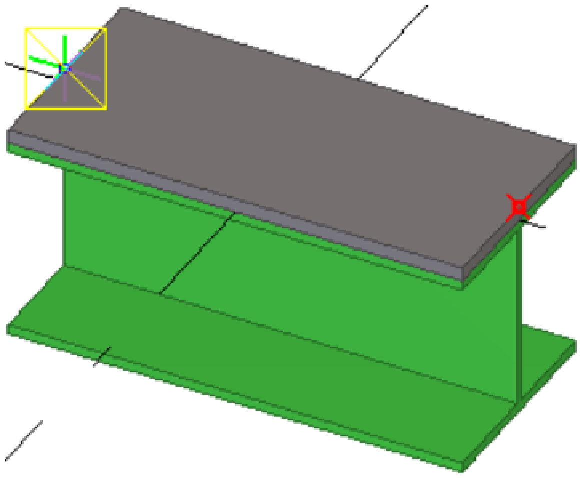
1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
2. Start typing `auto bolt` in the search box.
3. Double-click **Auto bolt** in the catalog to open the **Auto bolt** dialog box.
4. Define the bolt properties.
5. If needed, you can show cut length as temporary lines to view where the bolts should be placed even if they are not created.
   - Select in the list at the bottom of the dialog box to not show the temporary lines.
   - Select in the list at the bottom of the dialog box to show the temporary lines.

   To delete the temporary lines, right-click the view and select **Redraw view**.

6. Click **Apply**.
7. Select the main part.

   Auto bolt uses this part to identify the best rotation. This part will be the main part of the assembly.
8. Select the secondary part.
9. Click the middle mouse button.
10. Pick the first and the second position to define the bolt group direction.

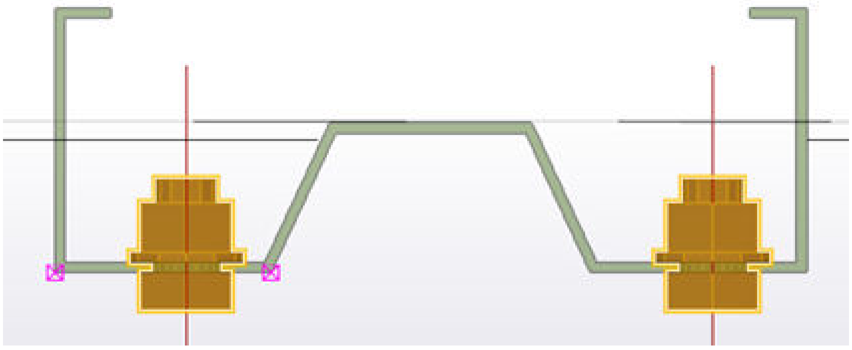




The positions you pick define a line that is used to find all parallel part faces and the optimal plane and locations for the bolts. The plane is chosen as follows:

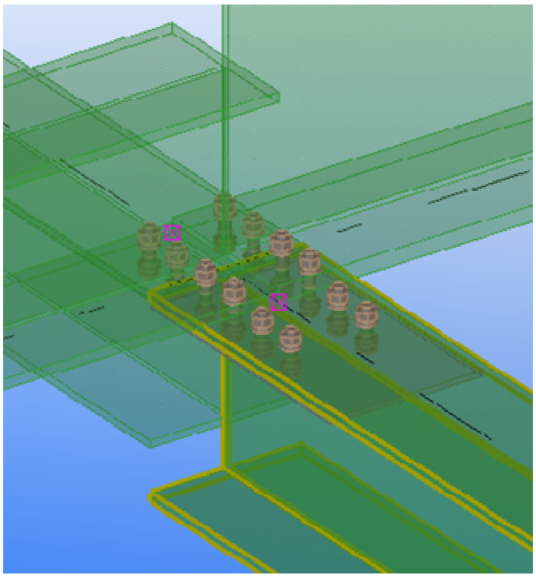
   a. The face with the longest projection of the line
   b. If there are two faces with equally long projections, the face which is closer to the line
   c. If there are two equally close faces, the larger face

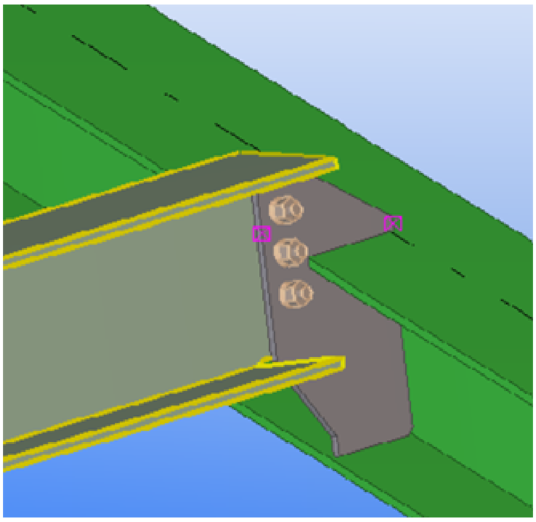
For example, with special part profiles, selecting points close to the wanted face ensures that the bolts are placed correctly:

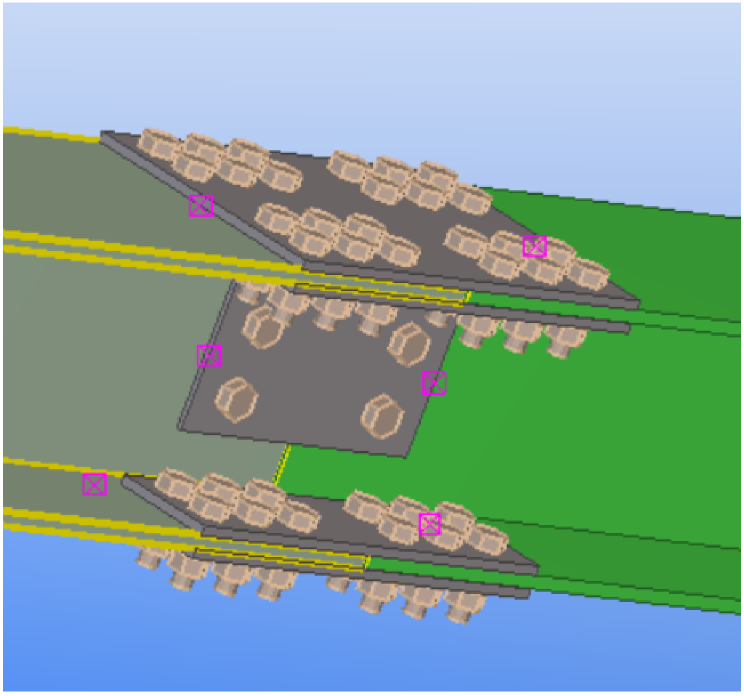


Examples

Examples of parts bolted using the **Auto bolt** component are shown below. The main parts and the selected points are highlighted.







Create a bolt group by exploding a component

An alternative way to create bolts is to first apply a component that includes bolt groups, and then explode the component.

1. Apply a component that includes bolt groups.

   For example, connect two beams, or a beam to a column, using a bolted end plate.

2. Explode (page 831) the component.

   a. Select the component to explode.

b. Right-click and select **Explode component**.

   Tekla Structures separates the objects in the component.

3. Modify the bolt group.
   a. Select the bolt group and double-click it to open the properties.
   b. Modify the properties.
   c. Click **Modify**.

Change or add bolted parts

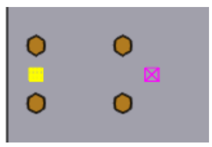
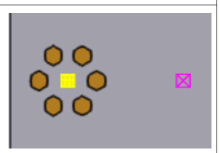
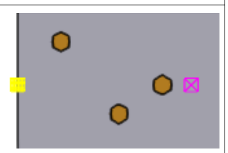
You can change the parts a bolt group connects to.

1. On the **Steel** tab, click **Bolted parts**.
2. Select the bolt group.
3. Reselect the main and secondary parts.
4. Click the middle mouse button to finish selecting the parts.

Bolt group shape

Use the **Bolt group** settings in the **Bolt** properties to select the shape of a bolt group and to determine how many bolts the bolt group contains.

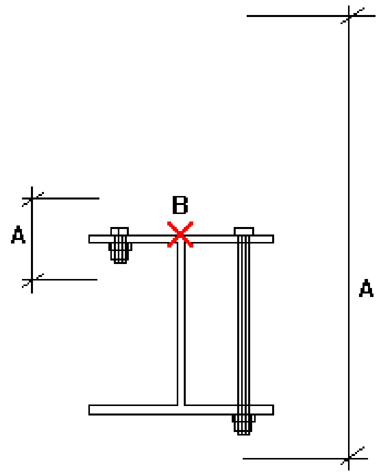
In the images below, the yellow handle indicates the origin of the bolt group, and the magenta handle indicates the x direction of the bolt group.

Shape	Other bolt group settings	Example values	Result
Array	**Bolt dist X** Spacing between bolts, in the x direction of the bolt group.	150	
	Bolt dist Y Spacing between bolts, in the y direction of the bolt group.	100	
Circle	**Number of bolts**	6	
	Diameter of the bolt group	100	
List	**Bolt dist X** x coordinate of each bolt, from the bolt group point of origin.	75 175 250	
	Bolt dist Y y coordinate of each bolt, from the bolt group point of origin.	75 -50 0	

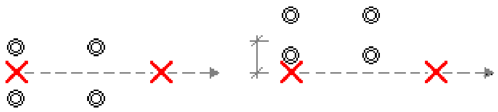
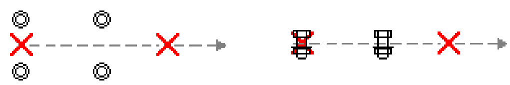
Bolt properties

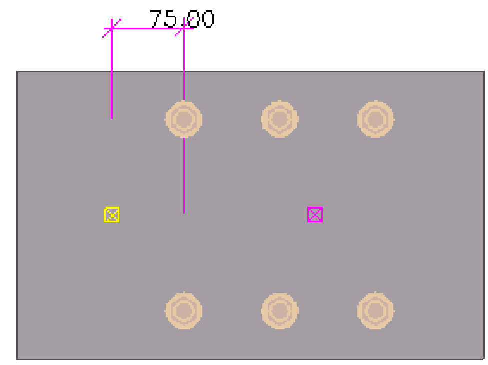
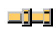
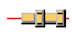
Use the **Bolt** properties to view or modify the properties of a bolt group. Use the **Holes** section to define the properties of bolt holes. The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals** .

Setting	Description
Bolt	
Size	Bolt diameter.
Standard	Bolt assembly standard/grade.
Bolt type	Define whether the bolts are assembled on-site or in the shop.
Connect as	Indicate whether you are bolting a secondary part or a sub-assembly.
Thread in material	Indicate if the thread of the bolt can be inside the bolted parts. Tekla Structures does not use this value when calculating the length of full-threaded bolts.
Cut length	Indicate which parts the bolt connects. The value defines the area Tekla Structures should search for parts that belong to the bolt group. Using cut length you can determine whether the bolt will go through one flange or two. Tekla Structures searches for parts using half the cut length value, in both directions from the bolt group plane. In the illustration below, A is the cut length and B is the bolt origin. Tekla Structures calculates the search area as A/2 in both directions from point B.

Setting	Description
	 Tekla Structures warns you if the cut length is too small (i.e. the bolt group contains no parts) and makes the bolt length 100 mm. If there are large gaps between the connected parts, the gap is added to the length of the bolt. Tekla Structures calculates bolt length using the total distance between the first and last surfaces. **NOTE:** If you want to force a bolt to be a certain length, enter a negative value for cut length (e.g. -150). **NOTE:** If holes or blind holes cannot be created, increase the cut length.
Extra length	Additional bolt length. Increases the material thickness that Tekla Structures uses when calculating bolt length. For example, you might need extra bolt length to allow for painting. You can also build additional lengths into bolt assemblies.
Assembly	Select whether to create washers and nuts with the bolt. If you want to create only holes (page 353) without any bolts, clear all the check boxes.
Bolt group	

Setting	Description
Shape	Bolt group shape. You have the following options: • **Array** for rectangular • **Circle** for circular • **List** for any shape
Bolt dist X	Bolt spacing or coordinate, depending on the bolt group shape (**Array** or **List**).
Bolt dist Y	Bolt spacing or coordinate, depending on the bolt group shape (**Array** or **List**).
Number of bolts	Number of bolts in a circular bolt group.
Diameter	Diameter of a circular bolt group.
Holes	
Tolerance	Tolerance = Hole diameter - Bolt diameter
Plain hole type	Select **Through** to create holes that are open throughout the part. Select **Blind** to create partial-depth holes that do not extend completely through parts.
Hole depth	Depth of a blind hole measured from the bolt/hole reference points (yellow and magenta handles). Note that you may also need to adjust the **Cut length** value.
Parts with special holes	If you want to create oversized, slotted, or tapped holes, or if you want to omit holes, select the desired check boxes to indicate which plies of the connection get special holes.
Use the same settings for all special holes	Select this check box to create similar special holes in each of the connected parts. The properties of the special hole that is closest to the bolt head will be used for all special holes in the bolt group. If you clear this check box, you can define special hole properties separately for each of the parts.

Setting	Description
Special hole type	Oversized, slotted, tapped, or no holes. This option becomes active when you select one or more **Special hole** check boxes next to **Parts with special holes**.
Oversized	Allowance of an oversized hole.
Slotted hole X	x allowance of a slotted hole. Zero for a round hole. To create slotted holes with an offset from the bolt center in the x direction, enter an offset value in the second box (**Slot offset**).
Slotted hole Y	y allowance of a slotted hole. Zero for a round hole. To create slotted holes with an offset from the bolt center in the y direction, enter an offset value in the second box (**Slot offset**).
Core hole size	Size of a predrilling hole.
Position	
On plane	Move the bolt group perpendicular to the bolt group x axis. 
Rotation	Define how far the bolt group is rotated around the x axis, relative to the current work plane. For example, you can use this box to indicate on which side of the connected parts you want the bolt head to be. 
At depth	Move the bolt group perpendicular to the current work plane.
Offset from	
Dx, Dy, Dz	Offsets that move the bolt group by moving the bolt group x axis. Use to change the position of the bolt group.

Setting	Description
	The start point values **Dx**, **Dy** and **Dz** move the first end of the bolt group, relative to the bolt group x axis. The end point values move the second end of the bolt group. • A positive **Dx** value moves the start point towards the end point. • **Dy** moves the end point perpendicular to the bolt group x axis on the current work plane. • **Dz** moves the end point perpendicular to the current work plane. An example bolt group with the **Dx** start point set to 75: 
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (UDAs) of the bolt. UDAs provide more information about the bolts.
Show cut length as temporary lines	This option is available in the **Auto bolt** component. Shows where the bolts should be placed even if they are not created. • Select  to not show the temporary lines. • Select  to show the temporary lines.

Create studs

A stud is special type of bolt that is welded to steel parts to transfer loads between steel and concrete.

Tekla Structures uses the same command for creating bolts (page 341), studs and holes (page 353). When you create studs, select the stud assembly standard in the **Bolt** properties. You can create a group of studs or a single stud.

You can also create studs by using the **Shear Stud (1010)** component.

1. Ensure that the needed studs are added to the bolt catalog and bolt assembly catalog.
2. On the **Steel** tab, hold down **Shift** and click **Bolt**  to open the **Bolt** properties.
3. In the **Standard** list, select the bolt assembly standard for the studs.
4. Under **Bolt group**, do one of the following:
   - To create a group of studs, define **Shape** and the related properties as desired.
   - To create a single stud, select **Array** from the **Shape** list and type `0` in the **Bolt dist X** and **Bolt dist Y** boxes.
5. Modify the other properties as desired.
6. Select the main part.
7. Click the middle mouse button to finish selecting parts.
8. Pick a point to indicate the origin of the stud or stud group.
9. Pick a second point to indicate the direction of the stud group x axis.

Create bolt holes

Tekla Structures uses the same command for creating bolts, studs and holes. Before creating holes, you need to change some of the properties in the **Bolt** properties. If you want to create only holes, do not use any bolt elements (such as bolts, washers, and nuts).

You can create the following types of holes:

- Round
- Oversized
- Slotted
- Tapped

When several parts are connected together with a bolt group or single bolt, you can create different holes in each of the parts, or you can use the same properties for the special holes in all the parts.

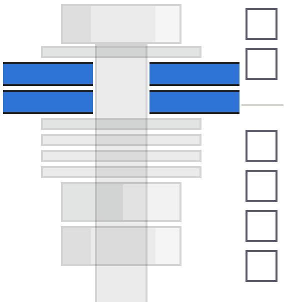
You can also select whether the holes you create are open throughout the parts, or partial-depth or blind holes that do not extend completely through the parts.

The hole properties are available in the **Holes** section in the **Bolt** properties (page 348).

Create round holes

You can create a group of round holes, or a single round hole. Tekla Structures calculates the diameter of a round hole as the sum of **Size** and **Tolerance**.

1. On the **Steel** tab, hold down **Shift** and click  **Bolt** to open the **Bolt** properties.
2. If you do not want to create any bolts, clear all the **Assembly** check boxes.

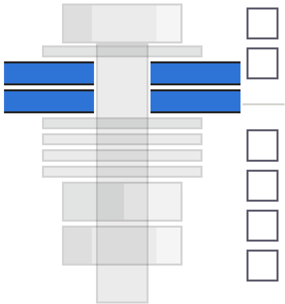
3. If needed, modify the hole properties.

   For example, to create partial-depth holes, set **Plain hole type** to **Blind** and enter a value in the **Hole depth** box.
4. Create the holes the same way you would create a bolt group (page 342):
   a. Select the main part, to which the secondary parts will be bolted.
   b. Select the secondary parts.
   c. Click the middle mouse button to finish selecting parts.
   d. Pick a point to indicate the hole group origin.
   e. Pick a second point to indicate the direction of the hole group x axis.

Create oversized holes

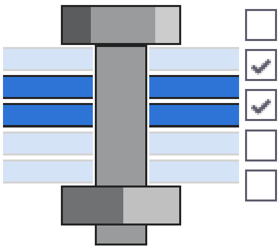
You can create a group of oversized holes.

1. On the **Steel** tab, hold down **Shift** and click  **Bolt** to open the **Bolt** properties.
2. If you do not want to create any bolts, clear all the **Assembly** check boxes.



3. Next to **Parts with special holes**, indicate which plies of the connection get oversized holes by selecting the desired **Special hole** check boxes.

   For example:



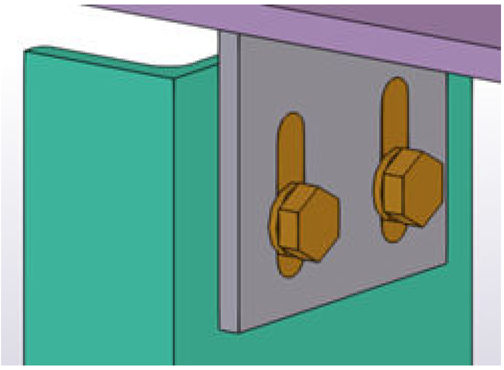
4. If you want to create similar holes in each of the parts with oversized holes, select the **Use the same settings for all special holes** check box.

   The holes will be created using the properties of the holes in the first part from the bolt head.

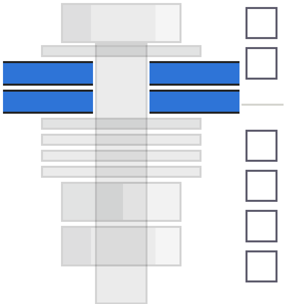
   If you clear this check box, you can define hole properties separately for each of the parts with oversized holes.
5. In the **Special hole type** list, select **Oversized**.
6. In the **Oversized** box, enter the allowance for the oversized hole.
7. If needed, modify the other hole and bolt properties.
8. Create the holes the same way you would create a bolt group (page 342):
   a. Select the main part, to which the secondary parts will be bolted.
   b. Select the secondary parts.
   c. Click the middle mouse button to finish selecting parts.
   d. Pick a point to indicate the hole group origin.
   e. Pick a second point to indicate the direction of the hole group x axis.

Create slotted holes

You can create a group of slotted holes. You can also create slotted holes with offsets from the bolts.

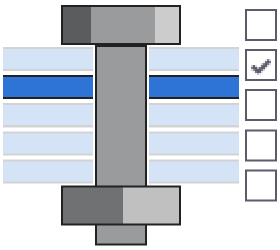


1. On the **Steel** tab, hold down **Shift** and click  **Bolt** to open the **Bolt** properties.
2. If you do not want to create any bolts, clear all the **Assembly** check boxes.

3. Next to **Parts with special holes**, indicate which parts should be slotted by selecting the desired **Special hole** check boxes.

   Tekla Structures counts the pieces of steel from the head of the bolt down. For example, if you select the second check box from the head of the bolt, Tekla Structures slots the second piece of steel from the head of the bolt.

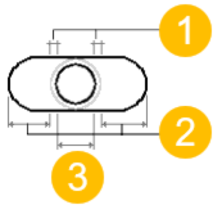
   

4. If you want to create similar holes in each of the slotted parts, select the **Use the same settings for all special holes** check box.

The holes will be created using the properties of the holes in the first slotted part from the bolt head.

If you clear this check box, you can define hole properties separately for each of the slotted parts.

5. In the **Special hole type** list, select **Slotted**.
6. Enter the allowance for the slotted hole in the x and y directions of the hole group using the **Slotted hole X** and **Slotted hole Y** boxes.



(1) Tolerance

(2) Slotted hole X or Y

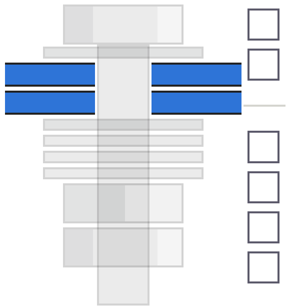
(3) Bolt size

7. If you want to offset the slotted holes from the bolts so that the bolts are not in the middle of the holes, enter an offset value in the **Slot offset** box next to **Slotted hole X** and **Slotted hole Y**.
8. If needed, modify the other hole and bolt properties.
9. Create the holes the same way you would create a bolt group (page 342):
   a. Select the main part, to which the secondary parts will be bolted.
   b. Select the secondary parts.
   c. Click the middle mouse button to finish selecting parts.
   d. Pick a point to indicate the hole group origin.
   e. Pick a second point to indicate the direction of the hole group x axis.

Create tapped holes

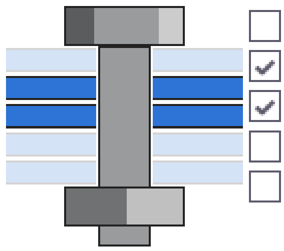
You can create tapped holes that have threads drilled in them.

1. On the **Steel** tab, hold down **Shift** and click  **Bolt** to open the **Bolt** properties.
2. If you do not want to create any bolts, clear all the **Assembly** check boxes.



3. Next to **Parts with special holes**, indicate which plies of the connection get tapped holes by selecting the desired **Special hole** check boxes.

   For example:



4. If you want to create similar holes in each of the tapped parts, select the **Use the same settings for all special holes** check box.

   The holes will be created using the properties of the holes in the first tapped part from the bolt head.

   If you clear this check box, you can define hole properties separately for each of the tapped parts.

5. In the **Special hole type** list, select **Tapped**.
6. In **Core hole size**, enter the predrilling hole size.
7. If needed, modify the other hole and bolt properties.
8. Create the holes the same way you would create a bolt group (page 342):
   a. Select the main part, to which the secondary parts will be bolted.
   b. Select the secondary parts.
   c. Click the middle mouse button to finish selecting parts.
   d. Pick a point to indicate the hole group origin.
   e. Pick a second point to indicate the direction of the hole group x axis.

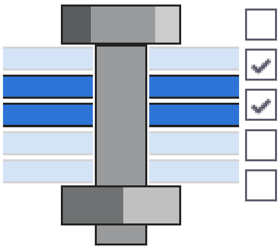
Create different holes with one bolt group

When several parts are connected together with a bolt group or single bolt, you can create different holes in each of the parts. For example, the holes in one part can be slotted, and the holes in the other parts can be either special holes or regular round holes.

Up to five parts can get special bolt holes with one bolt. The rest of the connected parts (the sixth and so on) will get regular round holes.

1. On the **Steel** tab, hold down **Shift** and click  **Bolt** to open the **Bolt** properties.
2. Next to **Parts with special holes**, indicate which parts get special holes by selecting the desired **Special hole** check boxes.

   For example:

3. Clear the **Use the same settings for all special holes** check box.
4. For each of the parts with special holes, select an option in the **Special hole type** list.
5. Depending on the type of the special holes in each of the parts, enter the needed hole dimensions and other properties.
6. Create the holes the same way you would create a bolt group (page 342):
   a. Select the main part, to which the secondary parts will be bolted.
   b. Select the secondary parts.
   c. Click the middle mouse button to finish selecting parts.
   d. Pick a point to indicate the hole group origin.
   e. Pick a second point to indicate the direction of the hole group x axis.

Create welds

You can either create a weld manually, or use a component that automatically creates welds.

By default, Tekla Structures places the welds to the arrow side using the **Above line** properties according to the ISO standard. You can change this to **Below line** to comply with the AISC standard with the advanced option XS_AISC_WELD_MARK.

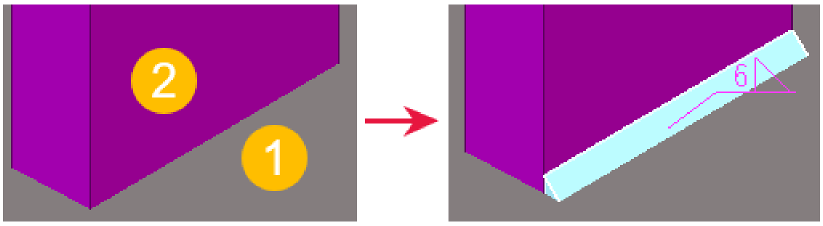
Create a weld between parts

Weld two parts together using the weld position defined in the **Weld** properties. The length of the weld depends on the length of the connection between the welded parts.

1. On the **Steel** tab, click **Weld** --> **Create weld between parts** .
2. Select the part to weld to.

   If you are creating a workshop weld, this is the main part of the assembly.
3. Select the part to be welded.

   If you are creating a workshop weld, this is the secondary part of the assembly.

   (1) Main part

   (2) Secondary part

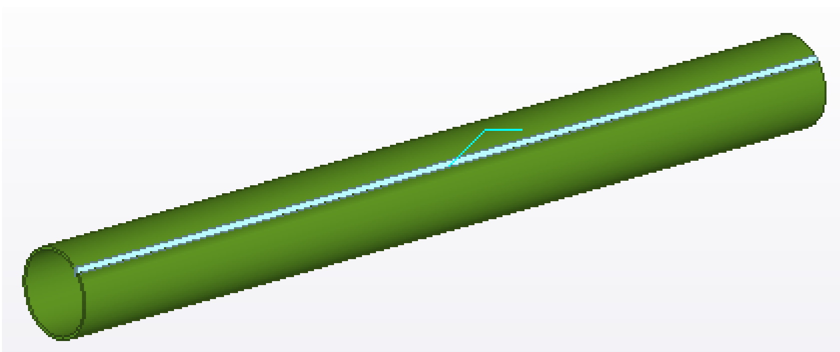
Create a weld to a part

You can create a weld to a single part, without connecting any other parts.

1. On the **Steel** tab, click **Weld** --> **Create weld to part** .
2. Select the part that you want to weld.
3. Pick the starting and end point, or alternatively, pick the points you want the weld to go through.
4. Click the middle mouse button to create the weld.

Example

Use the **Create weld to part** command to weld seams in tubular sections:



TIP	To model tubular sections with visible seams, use the SPD profile.

Create a polygon weld

Create polygon welds when you want to define the exact position of the weld by picking the points you want the weld to traverse.

If you want to create double-sided polygon welds, define both the **Above line** and **Below line** properties.

1. On the **Steel** tab, click **Weld** --> **Create polygon weld** .
2. Select the part to weld to.

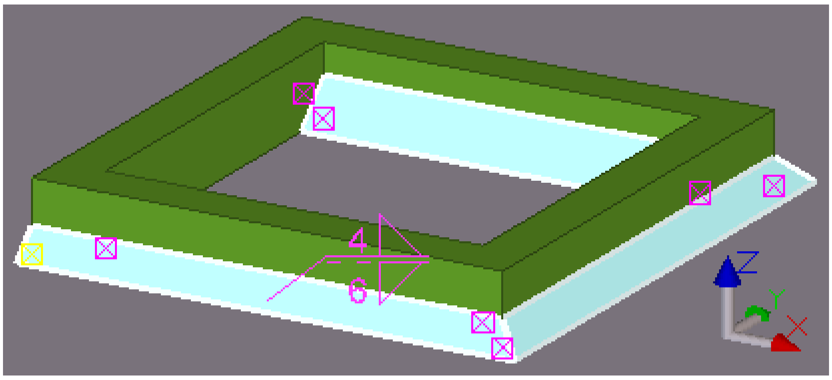
   If you are creating a workshop weld, this is the main part of the assembly.
3. Select the part to be welded.

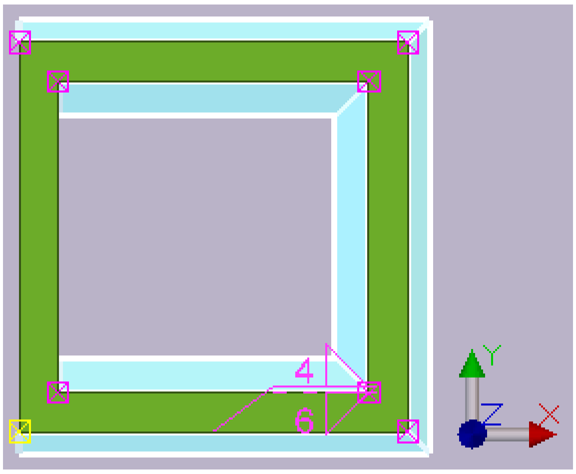
   If you are creating a workshop weld, this is the secondary part of the assembly.
4. Pick the starting and end point, or alternatively, pick the points you want the weld to go through.

   To create a double-sided polygon weld, pick the polygon points on one side of the part to be welded. Tekla Structures automatically searches for the corresponding points on the other side of the part.
5. Click the middle mouse button to create the weld.
6. If needed, modify the weld by dragging the handles.

Example

This example shows a double-sided polygon weld along three (outer and inner) edges of a rectangular hollow profile:



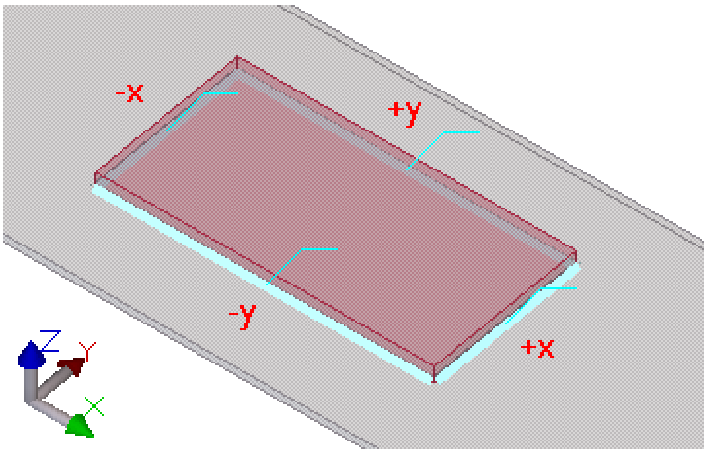


Weld properties

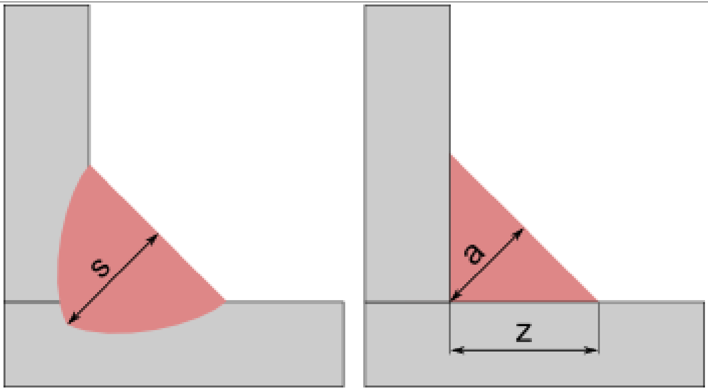
Use the **Weld** properties to view or modify the properties of a weld. The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals** .

NOTE Some of the properties are only displayed in reports, not in drawings.

Setting	Description
Common attributes	
Edge/Around	Indicates whether only one edge or the entire perimeter of a face should be welded. **Edge**:  **Around**: 
Workshop/Site	Indicates where the weld should be made. This setting affects assemblies and drawings. **Workshop**: **Site**: 
Position	Not available for polygon welds. Defines the position of a weld relative to the work plane. The type and position of the parts to be welded affect the position of the weld. The options for weld position are: • **+ x** • **- x**

Setting	Description
	• **+ y** • **- y** • **+ z** • **- z** In most cases, Tekla Structures creates the weld on the face or side of the part that faces in the selected direction (x, y, or z). Also the following factors may affect the position of the weld: • perpendicularity of the part edge towards the selected direction (x, y or z) • length of the part edge • distance of the part edge in the selected direction (x, y or z) The following image shows welds in different positions: 
Shape	The shape of the weld can be: •  (Regular, continuous weld) •  (Intermittent weld) •  (Staggered, intermittent weld)
Connect as	See Use welds to create assemblies (page 425).
Placement	Defines how the weld is placed in relation to the assembly parts. The options are:

Setting	Description
	• **Auto** The weld placement adapts to the typical situation of the weld type. Square-, V-, and U-groove welds are located in the middle of the main and secondary parts. Single-bevel and J-groove welds are located on the secondary part side. This is the default option. • **Main part** The weld is located fully on the main part side. Does not affect V- or U-groove welds. • **Secondary part** The weld is located fully on the secondary part side. Does not affect V- or U-groove welds.
Preparation	Defines which assembly parts, if any, are automatically prepared for welding. The options are: • **None** Parts are not prepared for welding. This is the default option. • **Auto** Parts are prepared for welding according to the weld type. • **Main part** The main part is prepared for welding. • **Secondary part** The secondary part is prepared for welding.
Weld	
Prefix	The weld size prefix. Shown in drawings, but only if the weld size is also defined. The standard ISO 2553 prefixes are: • **a** - Design throat thickness • **s** - Penetration throat thickness • **z** - Leg length

Setting	Description
	 Note that if the last character of the prefix is `s`, Tekla Structures creates the solid weld object according to the right-hand image so that `a` equals weld size.
Type	See the list of weld types (page 367) below.
Size	The size of the weld. If you enter a negative weld size, Tekla Structures creates the weld, but does not display it in drawings. If you enter a zero weld size, Tekla Structures creates the weld, and in drawings displays the weld reference line and the weld type symbol without the weld size. For compound weld types and , you can enter two size values.
Angle	The angle of weld preparation, bevels, or groove. Enter a positive value for bevel and groove welds. 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 This setting does not affect solid weld objects.
Finish	Tekla Structures displays the finish symbol above the weld type symbol in drawings. The options are: • **G** (Grind) • **M** (Machine)

Setting	Description
	• **C** (Chip) • (Flush finished weld) • (Smooth blended weld face) This setting does not affect solid weld objects.
Root face	Root face thickness is the height of the narrowest part inside the root opening. Root face values do not appear in drawings, but you can use the `WELD_ROOT_FACE_THICKNESS` template attribute in reports to show the root face dimension in the weld list.
Effective throat	The weld size used in weld strength calculation.
Root opening	The space between the welded parts. Enter a positive value for square-groove welds.
# of incr.	The amount of increments in an intermittent weld. Only use with the ISO standard.
Length	Defines the length value that is shown in the weld mark. For intermittent welds, defines the length of an increment. Does not affect continuous solid weld objects.
Pitch	If the advanced option XS_AISC_WELD_MARK is set to `TRUE`, the center-to-center spacing of weld increments in an intermittent weld. If the advanced option XS_AISC_WELD_MARK is set to `FALSE`, the space between the weld increments in an intermittent weld. Tekla Structures uses the `–` character by default 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 to `@`.
	Use these buttons to copy and to link together the **Above line** and **Below line** property values. Click the and buttons to copy values between the **Below line** and **Above line** columns. Click the button to switch the linking on or off.

Setting	Description
	The middle button is yellow when the values are linked. This means that if you change a value in either of the columns, also the corresponding value in the other column changes.
Tail information	
NDT inspection level	Defines the non-destructive testing and inspection level.
Electrode classification	Defines the weld electrode classification.
Electrode strength	Defines the electrode strength.
Electrode coefficient	Defines the electrode strength coefficient.
Process type	Defines the process type.
Reference text	Additional information to appear in the weld mark. For example, information about the weld specification or process. 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. Note that the special characters are shown in weld marks in model views only if the Arial font supports the special characters.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (UDAs) of the weld. UDAs provide additional information about the weld.

List of weld types

Use the weld properties to define the weld type. Some weld types also automatically prepare the parts to be welded. The following table shows the available weld types:

Number	Type	Name	Optional automatic weld preparation	Supported solid weld object
0		None	No	No
10		Fillet weld	No	Yes

Number	Type	Name	Optional automatic weld preparation	Supported solid weld object
3		Bevel-groove (single-V butt weld)	Yes	Yes
4		Bevel-groove (single-bevel butt weld)	Yes	Yes
2		Square-groove (square butt weld)	Yes	Yes
5		Single-V butt weld with broad root face	Yes	Yes
6		Single-bevel butt weld with broad root face	Yes	Yes
7		U-groove weld (single-U butt weld)	Yes	Yes
8		J-groove weld (single-J butt weld)	Yes	Yes
16		Flare V-groove weld	No	No
15		Flare-bevel-groove weld	No	No
1		Edge-flange weld	No	No
17		Corner-flange weld	No	No
11		Plug weld	No	No
9		Bevel backing weld	No	No
12		Spot weld	No	No
13		Seam weld	No	No
14		Slot weld	No	No

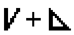
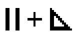
Number	Type	Name	Optional automatic weld preparation	Supported solid weld object
18		Partial penetration weld (single-bevel butt + fillet)	No	Yes
19		Partial penetration weld (square groove + fillet)	No	Yes
20		Melt-through weld	No	No
21		Steep-flanked single-V butt weld	Yes	Yes
22		Steep-flanked single-bevel butt weld	Yes	Yes
23		Edge weld	No	No
24		Surfacing weld	No	No
25		Fold joint	No	No
26		Inclined joint	No	No

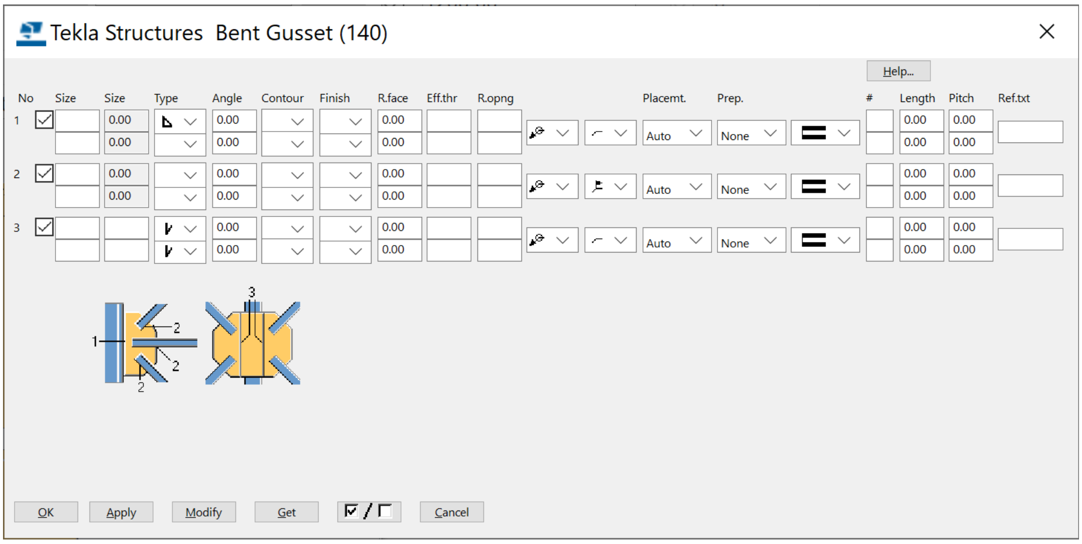
Welds in components

You can define the properties of the welds used in components. Tekla Structures displays the appropriate weld dialog box when you click the **Welds** button in the component properties dialog box.

The example image shows each weld definition using a number for the **Bent gusset (140)** connection. For each weld definition, use the upper row to define the above-line properties of the weld, and the lower row for the below-line properties.

You can select the weld type from the **Type** list.

For compound welds  and , you can enter two size values.



See also

[Create welds (page 359)]

Weld preparation

When parts are prepared for welding, their edges can be beveled to produce a groove for the weld. You can define the angle of bevels and grooves.

You can either prepare a part for welding manually, or apply a component that does it automatically, or you can use the **Preparation** options in the **Weld** properties or in the component weld properties.

NOTE When you use components' weld preparation options, the supported weld types (page 367) are placed correctly in the model. If you use cuts for preparing the part edges, the welds may not be placed correctly.

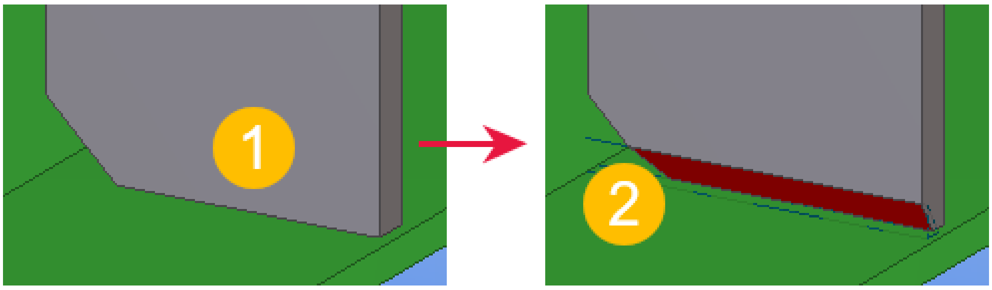
Prepare a part for welding with a polygon

You can manually prepare a part for welding by cutting it with a polygonal shape.

Before you start, ensure that the work plane (page 53) is on the plane you are cutting on.

1. On the **Steel** tab, click **Weld** --> **Prepare part for welding with polygon** .
2. Select the part that you want to cut.
3. Pick positions to outline the polygon to be used for cutting.

   Extend the polygon outside the part, so that it is clear that the part edge should be cut away.
4. Click the middle mouse button to close the polygon and to cut the part.



(1) The part to be cut

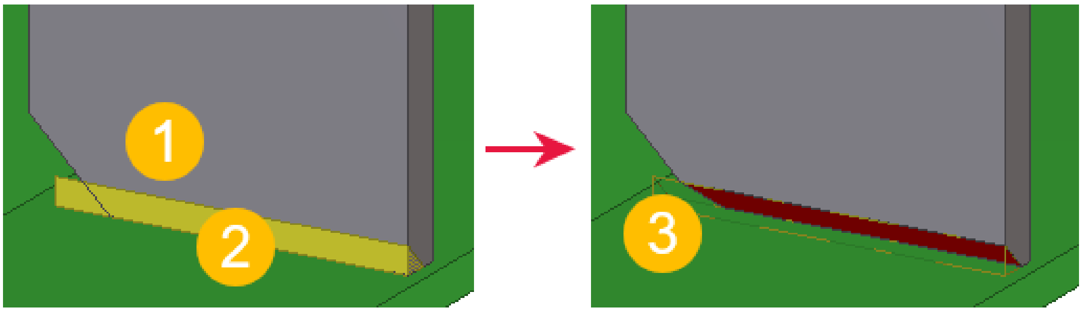
(2) Cuts are displayed using dash-and-dot lines

Prepare a part for welding with another part

You can manually prepare a part for welding by cutting it with another part. The cutting part will subsequently be deleted.

Before you start, create a cutting part and position it through the part you want to cut.

1. On the **Steel** tab, click **Weld** --> **Prepare part for welding with another part** .
2. Select the part that you want to cut.
3. Select the cutting part.



(1) The part to be cut

(2) The cutting part

(3) Cuts are displayed using dash-and-dot lines

Set the visibility and appearance of welds

Modify the display settings to define how welds appear in the model.

1. Double-click the view to open the **View Properties** dialog box.
2. Click **Display...** to open the **Display** dialog box.
3. Ensure that the **Welds** check box is selected.
4. Select a representation option for welds:

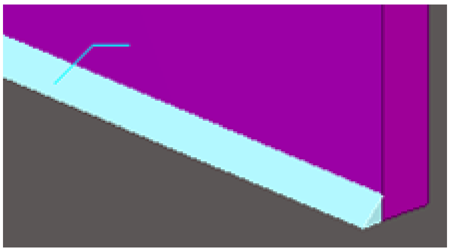
- **Fast**

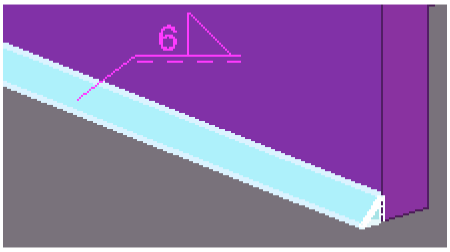
  Use this option to show the weld symbols only.

- **Exact**

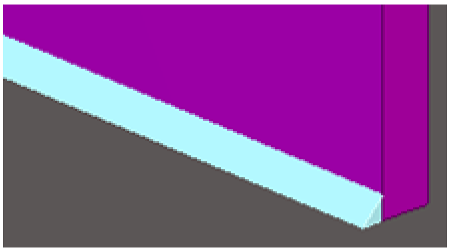
  Use this option to show welds as solid objects with weld symbols, and to display weld marks when you select welds.

- **Exact - no weld mark**

  Use this option to show welds as solid objects without weld symbols. Weld marks will not be displayed when you select welds.

5. Ensure that the view is being selected.
6. Click **Modify** to apply the changes.

NOTE If the representation option is **Exact** and you still cannot see the weld object in the model, check that the following properties have been defined for the weld in question:

- **Size**
- **Type**
- **Angle**
- **Root opening**

See also

Create welds (page 359)

Adjust the display settings (page 647)

Change a weld to a polygon weld

You can change existing welds to polygon welds if the existing welds have been created by using the **Create weld between parts** command or by a component. The new polygon welds will traverse the same points as the original welds did.

When converting double-sided weld segments to a polygon weld, Tekla Structures may not be able to create the polygon weld. If the welds to be converted consist of more than one polygon, or if there is a different number of weld segments on the sides of the welded part, Tekla Structures does not create the double-sided polygon weld but separate single-sided polygon welds.

1. Select the weld you want to change.

   To select multiple welds, hold down the **Ctrl** or **Shift** key.

2. On the **Steel** tab, click **Weld** --> **Convert to polygon weld** .

See also

Create welds (page 359)

Split a polygon weld

You can split a double-sided polygon weld into two single-sided polygon welds.

1. Select the double-sided polygon weld that you want to split.
2. Right-click and select **Split**.

See also

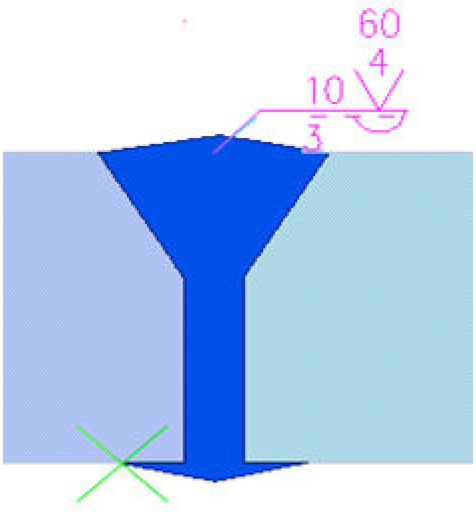
Create welds (page 359)

Change a weld to a polygon weld (page 373)

Create user-defined cross sections for welds

You can define special cross sections for model welds. This is useful when you need weld cross sections that are not predefined in Tekla Structures.

For example, you can create bevel backing welds:



To find the welds in the model that have a user-defined cross section, set **Category** to **Weld** and **Property** to **User-defined cross section** in the selection or view filter, or in the color and transparency settings.

Define a user-defined cross section for a weld

1. Select the weld you want to modify.
2. Right-click and select **Define Cross Section**.
3. In the weld cross section editor view:
    a. Pick points to indicate the weld cross section corners.
    b. Click the middle mouse button to finish picking.

Remove a user-defined cross section from a weld

You can remove user-defined cross sections from model welds and revert to the previous standard cross sections.

1. Select a weld that has a user-defined cross section.
2. Right-click and select **Remove Cross Section**.

    Tekla Structures removes the user-defined cross section and uses the previous standard cross section and properties for the weld.

Limitations

- User-defined weld cross sections are reported using only the above-line properties.

- User-defined weld cross sections do not create automatic weld preparation.

Fit part ends

The **Fit part end** command adjusts the end of a beam or a column by creating *a fitting plane, perpendicular to the view plane* that passes through the two points you pick. Use the **Fit part end** command to cut or to extend the part end, as needed. Do not use the command to make considerable extensions to part length.

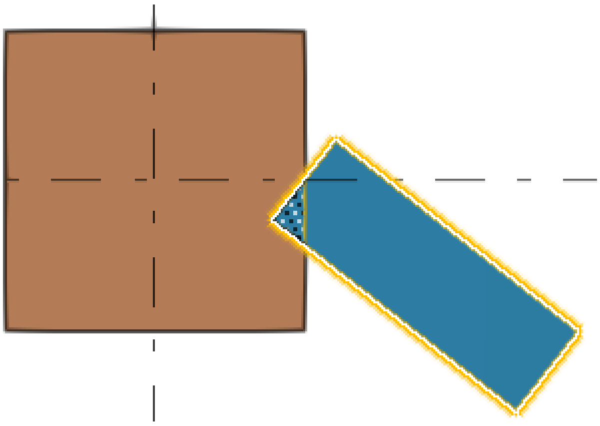
You can use the **Fit part end** command to adjust only the end of a beam or a column, not the sides. The command affects the nearest end of the part.

TIP To make it easier to pick the points, ensure that you are working in a view with a suitable view plane.

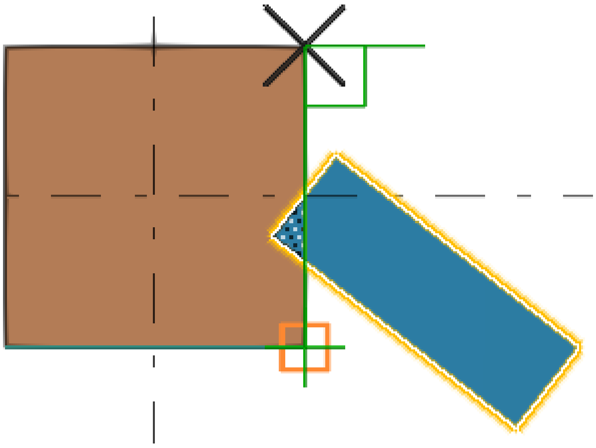
- In many cases you can create a view with a suitable view plane by using the **New view** --> **Default part views** command. The command creates views which have view planes along the main axes (x,y,z) of the selected part.
- If you need the view plane on some other angle which is not along the part axis, you can first define the work plane and then use the **New view** --> **On work plane** command.

---

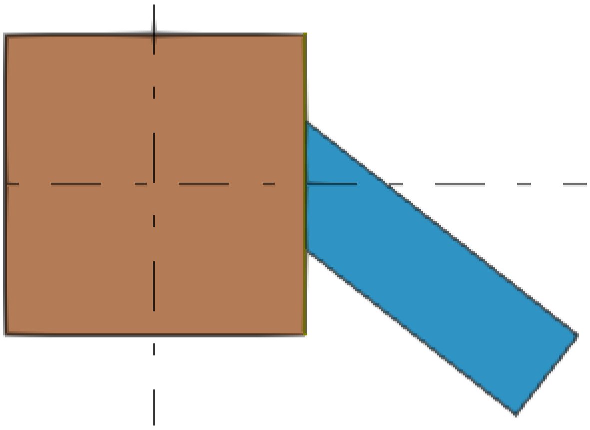
1. On the **Edit** tab, click **Fit part end**.
2. Select the part that you want to adjust with a fitting.



3. Pick the first point of the fitting line.
4. Pick the second point of the fitting line.



The end of the part is adjusted.



If you apply a second fitting to the same part end, Tekla Structures will ignore the first fitting, and use the second one. You can apply one fitting and one or more line cuts (page 391) to the part end.

Note that when creating NC files in DSTV format, fittings affect the length of the beam in the NC file. The overall length of a beam will be the fitted net length of the beam.

If you want have more options on how to create and adjust fittings, use the Fit beams and columns (page 376) or the Fit objects (page 380) component.

See also

Fit beams and columns (page 376)

Fit objects (page 380)

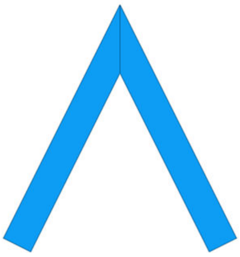
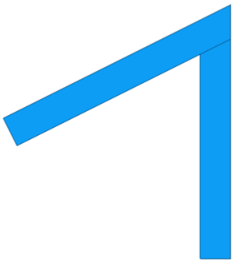
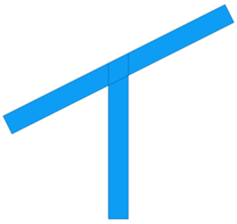
Cut objects with a polygon, line, or part (page 389)

Fit beams and columns

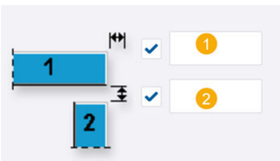
You can use the **Fit beams and columns** connection to connect two beams, two columns, or a beam and a column with fittings and cuts only.

The fittings and cuts that connect the parts can be placed in several ways. You can define that a gap is created between the parts, and use welds to connect them.

The table below shows examples of the connections.

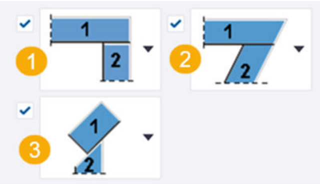
Example	Description
	Mitre connection
	Fit both profiles
	One of the two profiles is not cut

1. Select the **Fit beams and columns** component in the **Applications & components** catalog.
2. To change the default properties, open the **Fit beams and columns** component dialog box and modify the properties on the **Settings** tab.
3. Define the **gap size** between the main part and the secondary part. If you do not enter any value, a gap is not created.



	Description
1	Cut offset of the main part
2	Gap between the main part and the secondary part

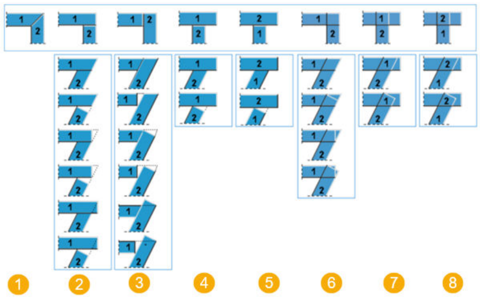
4. Select the **fitting type**.



The type you select in list **(1)** defines which fitting methods are available in list **(2)**. For the fitting type options, see the image below.

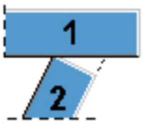
(3) When both the main part and the secondary part are skewed, select whether the fitting is created in the direction of the web or the flange.

The fitting type options are as follows:



	Description
1	Mitre
2	Secondary part is fitted to the main part, both parts are fitted.
3	Main part is fitted to the secondary part, both parts are fitted.
4	Secondary part is fitted to the main part, the main part is not fitted.
5	Main part is fitted to the secondary part, the secondary part is not fitted.
6	Secondary part is fitted with an overlap on the main part, both parts are fitted.
7	Secondary part is fitted with an overlap on the main part, the main part is not fitted.
8	Main part is fitted with an overlap on the secondary part, the secondary part is not fitted.

5. Define the **maximum gap size** between the main part and the secondary part. To do this, select the following fitting type:

   Select one of the following:

   - **Use default**

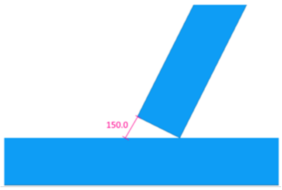
     The secondary part is fitted perpendicularly.

   - **Method by threshold**

     Define the gap threshold distance. If the distance between the main part and the secondary part is shorter than the value you define, the secondary part is fitted perpendicularly.

     In the example image below, the distance is `150.0`, while the value you have defined is `200.0`.

     If the distance is greater than the threshold value you define, the fitting on the secondary part is aligned with the main part.

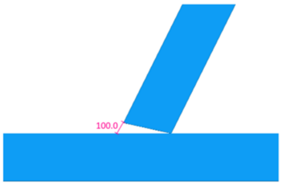
     In the example image below, the defined distance is `0`.

   - **Define gap**

     The value you define is used as the distance to fit the secondary part.

     In the example image below, the defined distance is `100.0`.



6. Select whether the **Main part (1)** and **Secondary part (2)** are fitted, or cut with a line cut.
7. In **Connect parts**, select how the parts are connected:
   - **No**: Parts are not connected.
   - **Cast unit**: The secondary part is added to the main part cast unit.
   - **Sub assembly**: The secondary part is added to the main part cast unit as a sub-assembly.
   - **By welding**: The secondary part is welded to the main part.

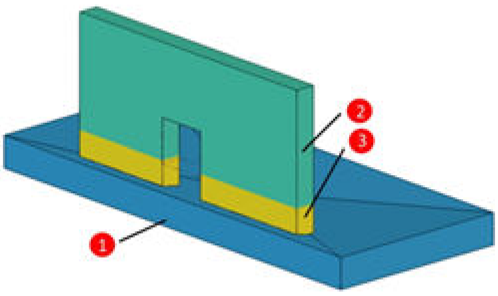
     When you select **By welding**, you can define the weld properties on the **Welding** tab.
8. On the **Welding** tab, define the weld properties.

   For detailed information on the weld properties, see the **Weld properties** section in Create welds (page 359).
9. Click **OK**.
10. In the model, select the main part (beam or column).
11. Select the secondary part (beam or column).

    The connection is automatically created when you select the secondary part.

Fit objects

You can use the **Fit objects** component to fit contour plates, slabs, beams, columns, and panels in any order.



(1) Main part

(2) Secondary part

(3) Additional material

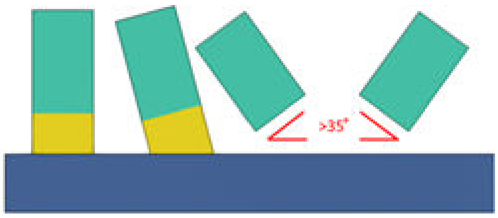
For example, in concrete constructions, you can use the **Fit objects** component to fit walls to sloping slabs and keep them connected. The walls are fitted to follow the slab geometry, and they adapt to any changes made in the slab.

You can also use the **Fit objects** component to fill the gap between inclined slabs, such as ramps and horizontal floor slabs. Filling these gaps exactly without small gaps or overlaps creates steady pours of these cast-in-place parts.

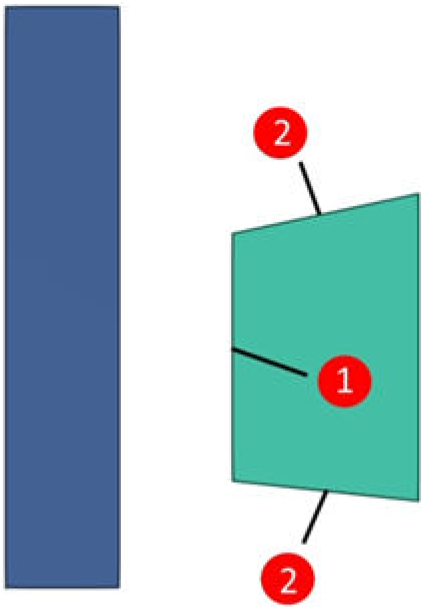
Connection logic

The **Fit objects** component fits the secondary part to the main part by creating cuts where necessary and filling the gaps with additional parts.

The component only fits secondary part faces that create an angle between -35° and 35° degrees with the main part. The example image below shows that additional material is not created for secondary parts with an angle more than 35°.



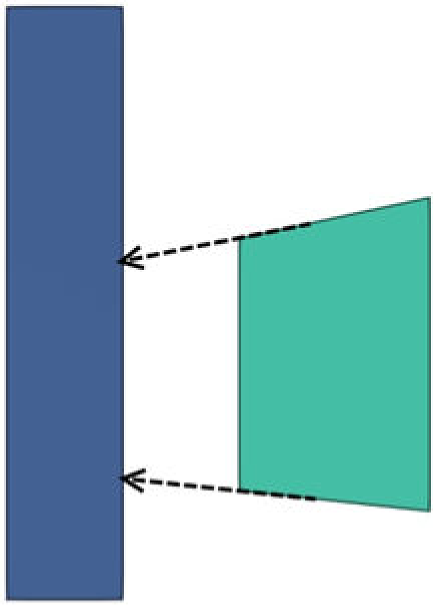
Once the main part and secondary part faces to be fitted have been identified, the secondary part face is fitted in the direction of its adjacent faces.



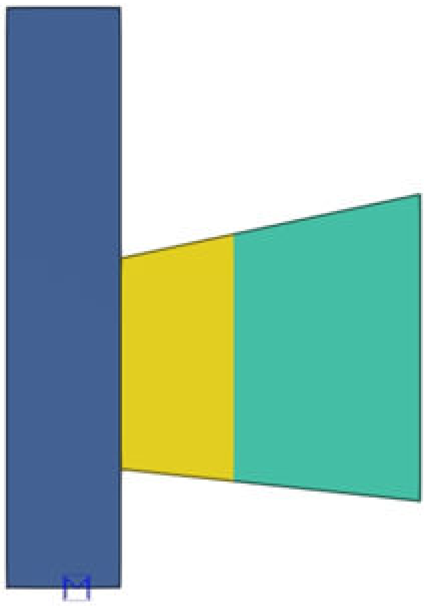
(1) Face to be fitted

(2) Adjacent faces

Planes are extended from the adjacent faces until they intersect with the main part.



Additional material is created to fill the space between the extended planes, the main part, and the secondary part.



Limitations

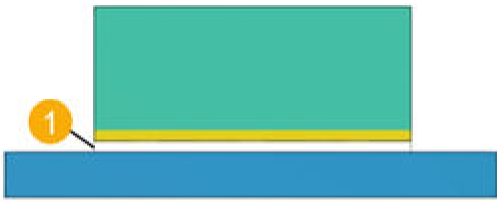
- Curved panels, polywalls, and polybeams are not supported.
- **Fit to main part** setting: With the **Face projection** option, only the closest face of the main part is projected. Multiple faces cannot be projected.

- Additional material is not always created from the same object type as the secondary part. For example, for walls, the additional material is created as a slab.
- Parts created with sketched profiles are not supported as secondary parts.

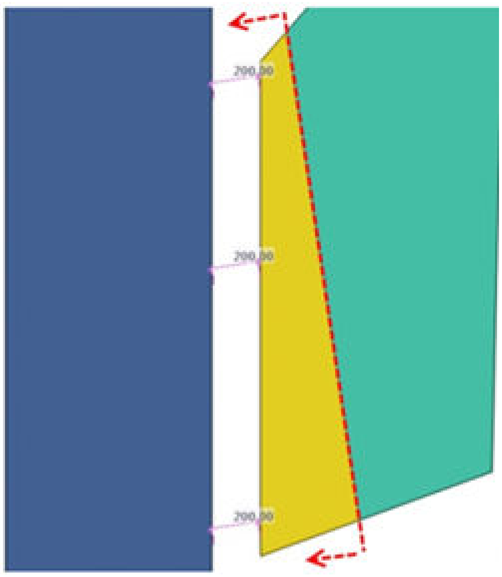
Fit objects in the model

We recommend that you first define the properties that you want to use when fitting the objects. When you have defined the properties, select the parts that you want to fit, and create the fitting in the model.

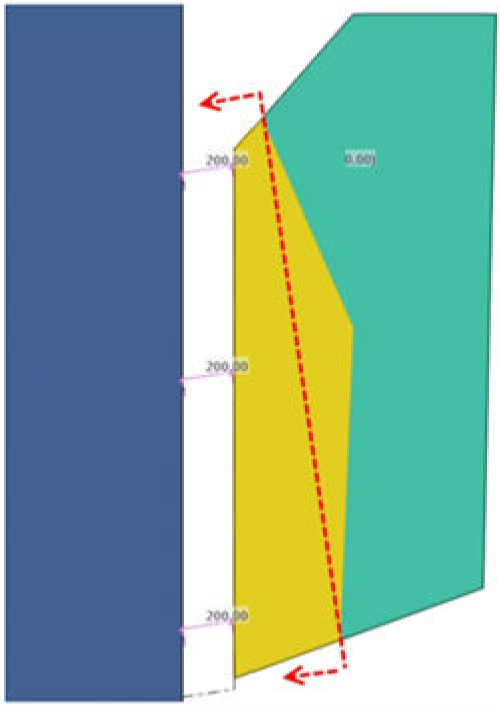
1. Select the **Fit objects** component in the **Applications & components** catalog.
2. To change the default properties, open the **Fit objects** dialog box and modify the properties.
3. **Create gap**: Define the gap distance between the main part and the additional material or the secondary part. To create a fitting that is level with the main part, leave the value as `0.0`.

   The gap distance is measured in the direction perpendicular to the secondary part face, shown with the dashed line in the image below. After the gap direction is established, the same gap is applied along the main part face. In the example, the gap distance is `200 mm`.



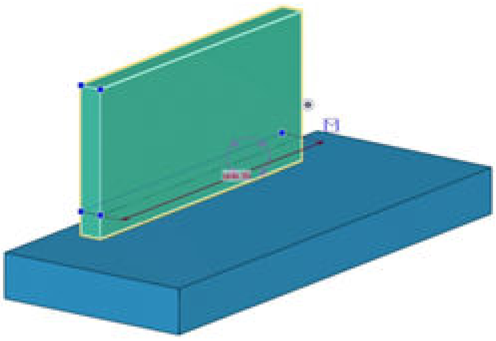
When more than one face is fitted, the gap direction is perpendicular to an imaginary line between the outer faces of the object. To use the same example, when there is only one face between the outer faces, the direction of the gap distance remains the same.



4. **Create additional material**: Select what kind of additional material is created.

   - **As added material**

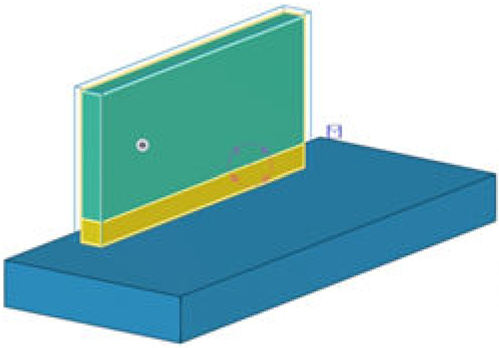
     Tekla Structures automatically attaches the added material (page 330) to the secondary part to form one single combined part.

     This is a useful option if you are creating quantity reports based on parts, because quantities such as `HEIGHT` take into account the added material and attempt to take the full height of the combined part.

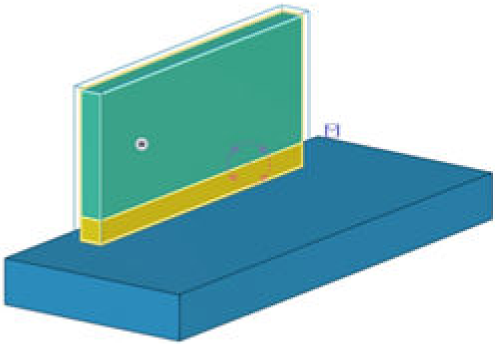
- **As parts added to assembly/cast unit**

  Additional material is added to the cast unit (concrete) or assembly (other material) of the fitted part.

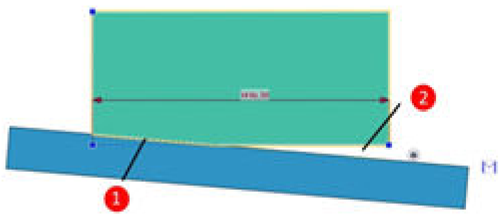
- **As loose parts**

  Additional material is created as a separate individual part.

- **No**

  Additional material is not added to fill the gap between the part to be fitted and the main part. Note that if the part to be fitted overlaps with the main part, cuts are used to remove the overlapping material.

  (1) Wall fitted only by cutting material

  (2) Additional material not added

5. **Create to**: Select whether the part is fitted to the nearest face or to all faces within a certain distance.

   - **Nearest face** only fits the nearest face to the main part.

     Fit objects considers each secondary part face one at a time to determine which secondary part face is closest to the main part. For each face, **Fit objects** checks the following:

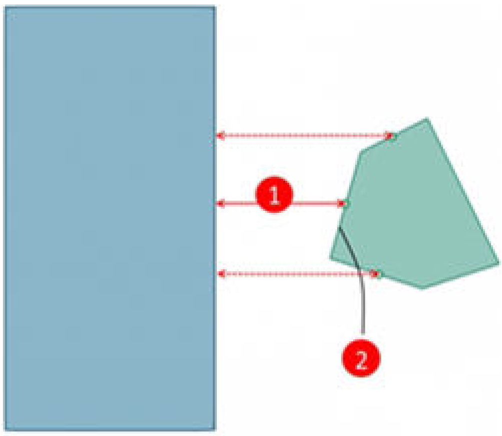
     a. Identifies the center point of the secondary part face.

     b. Creates a line from the center point of the secondary part face towards the main part, perpendicular to the main part face, until the line intersects with the main part.

     c. Measures the line.

     d. Repeats steps **a - c** for all secondary parts and the main part.

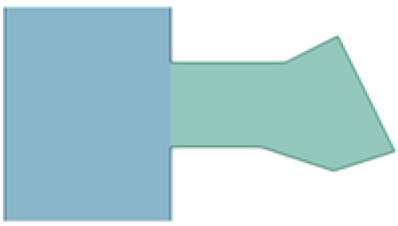
     The shortest line indicates the two closest faces. The face corresponding to the secondary part is the **Nearest face**.

     (1) Shortest distance

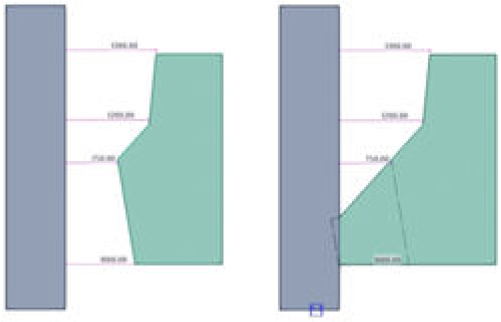
     (2) Face to be fitted (= nearest face)

     The result is shown in the image below:

   - **Faces closer than** fits all the faces that are closer than the specified distance.

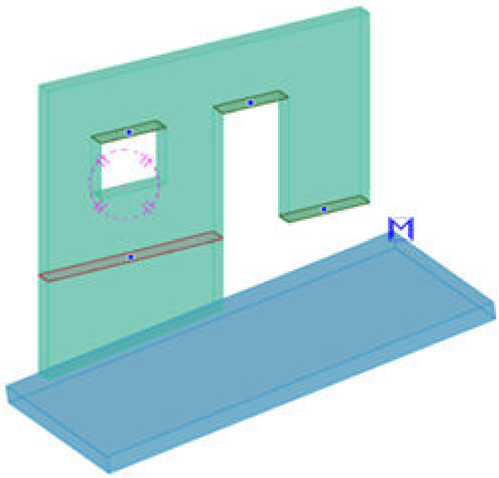
     The distance is measured similarly as in **Nearest face**, except that all points of a face need to be below the defined distance for that face to be fitted.



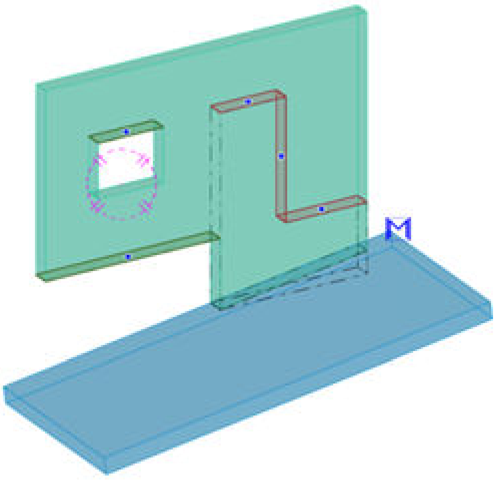
In the example above, the used limit distance is `1000 mm`. The only face that has all its vertexes closer than `1000 mm` is the face near the bottom, and it is therefore the only face fitted.

- Use **Direct modification** to manually select which secondary part faces are to be fitted.

  In the model, select the component. The green and red direct modification handles that appear on the secondary part faces indicate the faces that can be fitted.

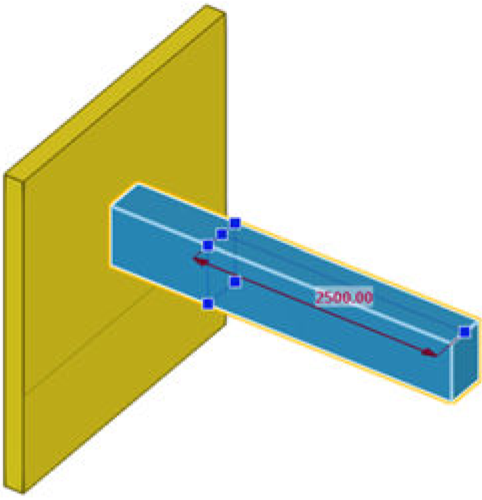
  Red and green faces indicate the faces that have been and that have not been fitted, respectively. Click a green face to fit that face. Click a red face to stop that face from getting fitted, as shown in the example image below.



6. **Use fit part end if possible**: Add fittings to parts in the same way as with the Fit part end command (page 375).

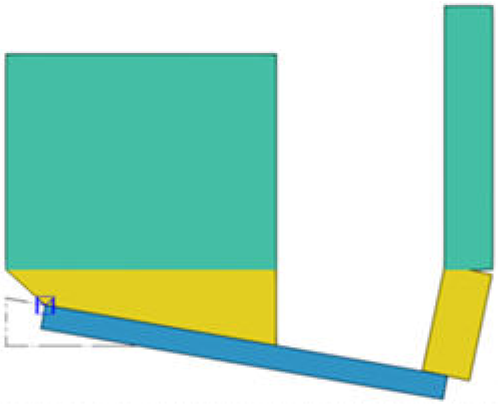
   This option only applies to those objects that can be fitted with the **Fit part end** command: beams, columns, and footings. Slabs, plates, and walls cannot be fitted in this way, so this option does not affect them.

   - **Yes**: If possible, parts are fitted by using the **Fit part end** command.

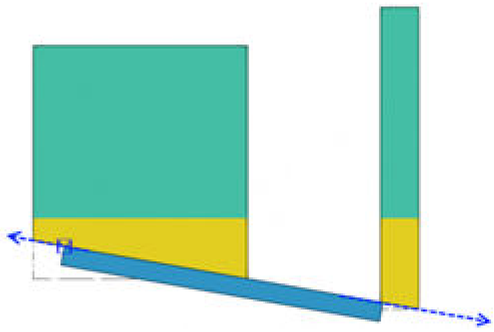


   - **No**: The **Fit part end** command is not used, and parts are fitted using the option you select in **Create additional material**.

7. **Fit to main part**: Define how the main part is used to fit the secondary part.
   - **Actual face**: The secondary parts are fitted exactly to the geometry of the main part. The image below shows an example of **Actual face** applied to a wall and a column.



   - **Face projection**: The secondary parts are fitted to the plane that is coplanar with the main part face. The image below shows an example of **Face projection** applied to a wall and a column.



8. Click **Modify** to save the properties that you have defined.
9. Select the main part (part to be fitted to).
10. Select the secondary part (part to be fitted).

    The part is fitted when you select the secondary part.

Cut objects with a polygon, line, or part

Use polygon cuts, lines cuts, or part cuts to shape a part in a model.

If you need to modify the length of a part, do not use cuts but move the part handles (page 307). In addition to cuts, you can use the **Fit part end** command to adjust the part end.

NOTE Do not use cuts to cut a part into two. Numbering, material lists and drawings will consider the part as one, not as two separate parts.

Do not use cuts to shorten steel parts because cuts are not taken into account in LENGTH_GROSS, which is often used in steel material lists.

Cut objects with a polygon

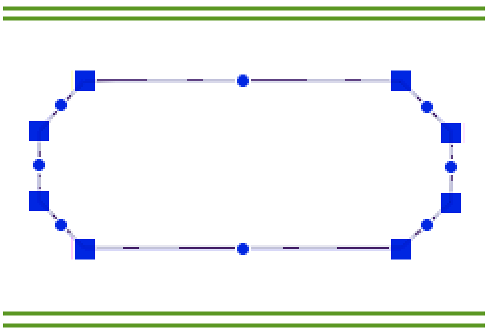
Use a polygon cut to cut a part, or a rebar set, *with a closed polygon*, perpendicular to the plane that is defined by the points you pick. Tekla Structures displays the cut using dash-and-dot lines.

The depth of the cut is calculated automatically based on the dimensions of the object to be cut. The **Polygon cut** command automatically extends the cut slightly (3 mm) outside of the part face in depth direction.

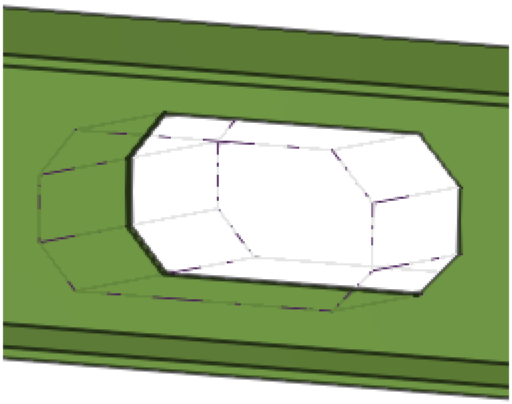
Tekla Structures uses the parametric profile BL to create polygonal cuts.

TIP To make it easier to pick the points for a polygon cut in a 3D view on a correct plane, use the **New view** --> **Default part views** command to create a suitable view. The command creates views that have view planes along the main axes (x,y,z) of the selected part.

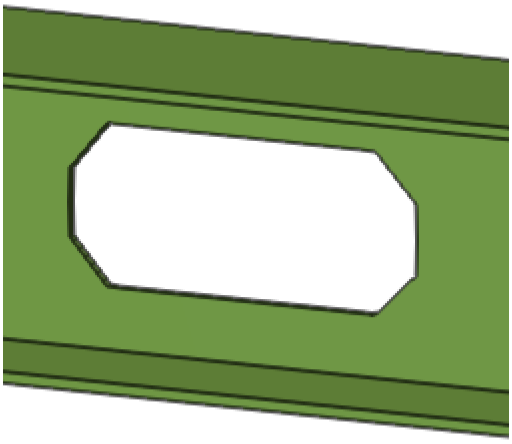
1. On the **Edit** tab, click **Polygon cut**.
2. Select the object that you want to cut.
3. Pick points to outline the polygon to be used for cutting.

   When defining the polygon, do not place the cut edges exactly at the same location as the edge of the part to be cut, as it can be unclear whether the edge should be cut away or not.
4. Click the middle mouse button to close the polygon and to cut the object.



You can hide the cuts (page 395), if needed.



If you need to modify the cut shape, use direct modification (page 107) or handles (page 307) to add or remove points, to add or change chamfers (page 397), or move an edge.

Cut objects with a line

Use line cuts to shape a beam or a column, or to cut a plate, a slab, an item, or a rebar set. A line cut creates *a cutting plane, perpendicular to the view plane* that passes through the two points you pick. You can use the line cut to cut the end of a part, or the side of a part, for example to make a part narrower. Tekla Structures displays the cut using dash-and-dot lines.

You can have several line cuts to one part end.

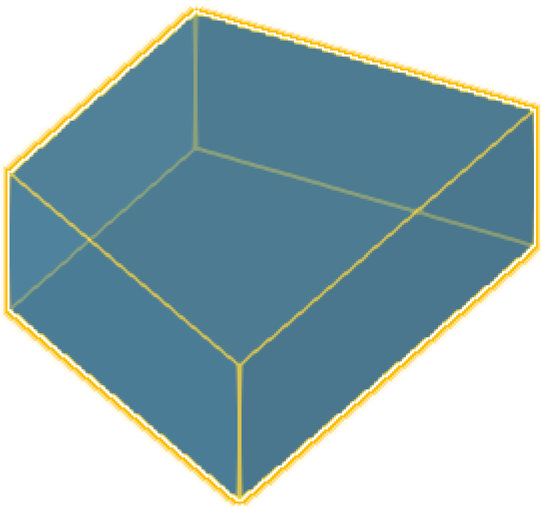
TIP To make it easier to pick the points for the cutting line, ensure that you are working in a view with a suitable view plane.

- In many cases you can create a view with a suitable view plane by using the **New view** --> **Default part views** command. The command creates views that have view planes along the main axes (x,y,z) of the selected part.

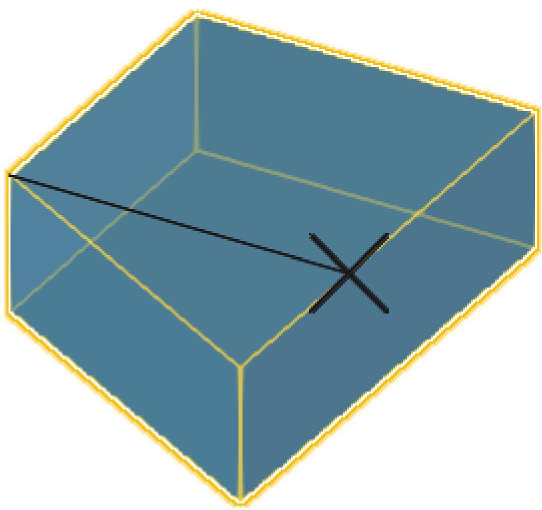
- If you need the view plane at some other angle, not along the part axis, you can first define the work plane and then use the **New view** --> **On work plane** command. Alternatively, you can use the **3D cut** component from the **Applications & components** catalog.

---

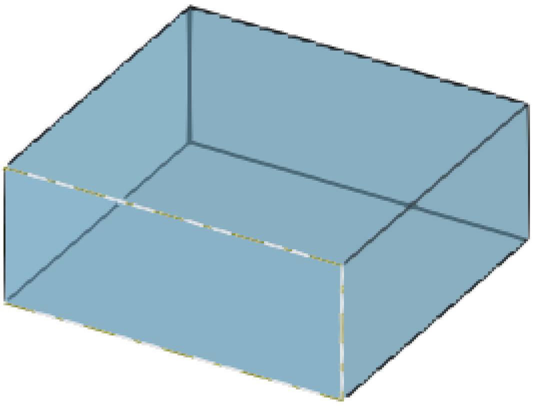
1. On the **Edit** tab, click **Line cut**.
2. Select the object that you want to cut.



3. Pick the first point of the cutting line.
4. Pick the second point of the cutting line.



5. Pick the side that you want to remove.



You can hide the cuts (page 395), if needed.

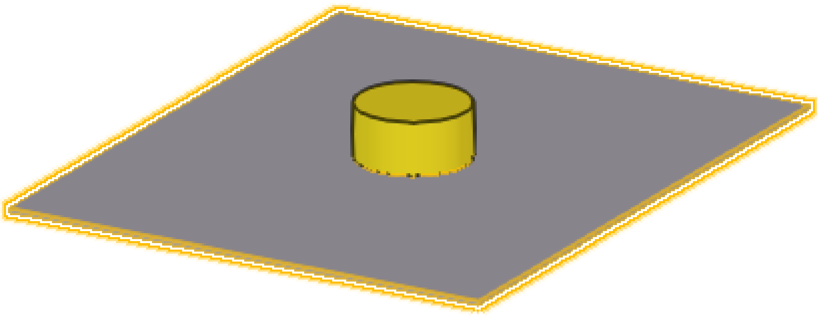
Cut objects with a part

You can cut a part, or a rebar set, *using another part*. Note that you can cut parts that already have cuts. That can be useful, for example, when you want to create more sophisticated cut shapes. Tekla Structures displays the cut using dash-and-dot lines.

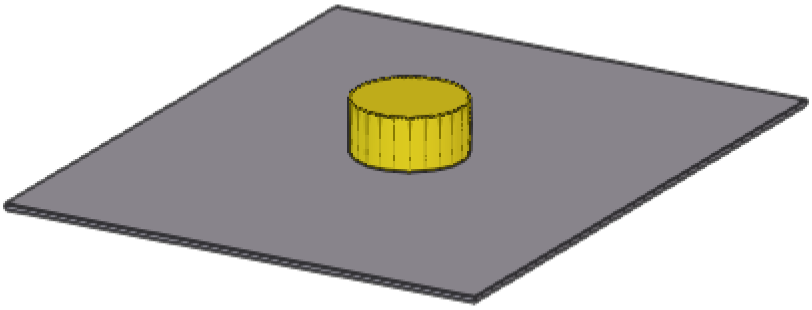
1. If you do not have a part to cut with, create a cutting part and position it through the part you want to cut.

   Do not place the cutting part so that the cut edges or vertices are exactly at the same location as the edges or vertices of the part to be cut. It may be unclear what should be cut away.

2. On the **Edit** tab, click **Part cut**.
3. Select the object that you want to cut.



4. Select the cutting part.

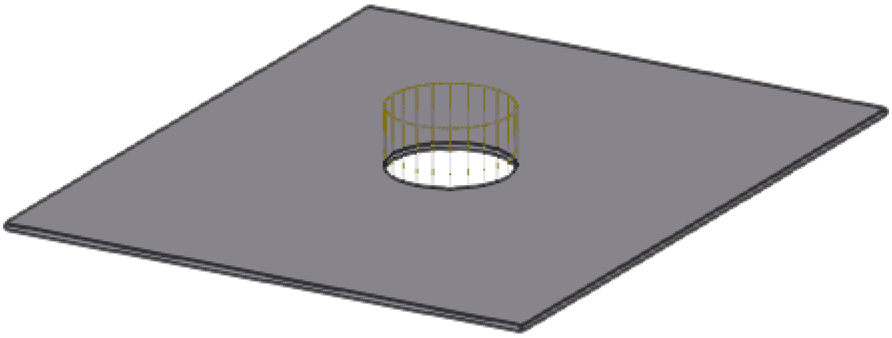


Using the cutting part, Tekla Structures creates the cut to the selected object. The part cut does not affect other objects.

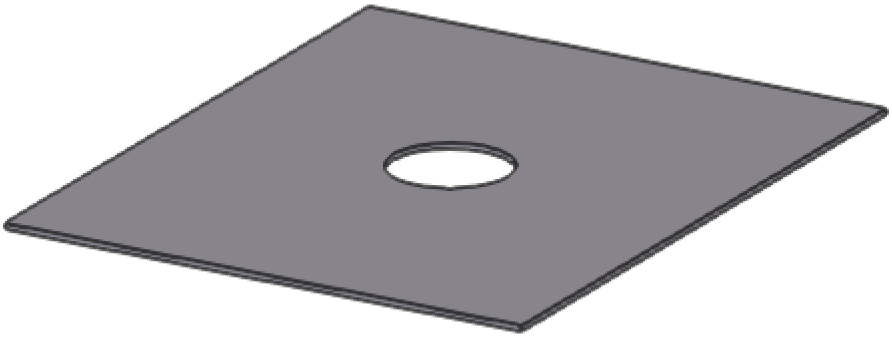
5. Delete the cutting part.

   a. Ensure that the  **Select cuts and added materials** selection switch is off.

   b. Select the cutting part and press **Delete**.

   You can hide the cuts (page 395), if needed.

Tips on how to cut efficiently

- **Avoid part faces**

  Avoid placing cut edges, cut vertices, or other cuts cutting the same part exactly at the same location as the edges or vertices of the part to be cut. Extend the cut outside of the part or other cut at least 0.3 mm. This practice helps you to avoid solid errors.

- **Use polygon cuts**

  Whenever possible, use polygon cuts. The **Polygon cut** command automatically extends the cut slightly outside of the part face in depth direction.

- **Use edge chamfers**

  Whenever possible, use edge chamfers (page 398) instead of small cuts, especially in components.

- **Tips for flange cuts of steel profiles**

  When cutting a flange, if the cutting part cuts very slightly the web as well (at least 0.3 mm), the cut is more likely to succeed. For example, if you are cutting a beam that has roundings, it may be useful to cut even further onto the web in depth direction than just the flange thickness.

- **Tips for round tube cuts**

  Use the Round tube (23) component for round tube cuts. The component automatically rotates the cutting part until a successful cut position is found. If the component fails, rotate the cutting part slightly until you find a successful cut position.

- **If a cut causes a solid error**

  If a cut causes a solid error, Tekla Structures is unable to render part faces and the part becomes transparent, only some edge lines being visible.

  An error notification is printed in the session history log stating which part and which cut caused the failure.

  To locate the failure in the model, click a row that contains a GUID identifier in the session history log. Tekla Structures selects the corresponding part and cut in the model.

  To fix the error, move the problematic cut slightly (0.3 mm) to a different direction. If the cut is intersecting with other cuts, you can try to move also the other cuts.

Hide cuts in a model view

1. Double-click the view background to open the **View Properties** dialog box.
2. Click **Display...** to open the **Display** dialog box.

3. Ensure that the **Cuts and added material** option is **not** selected in the display settings.
4. Click **Modify**.
5. If you want to temporarily display the hidden cuts in a model view:
   a. Select the part.
   b. Click **Display detailing** on the contextual toolbar.

   All the cuts of the selected part are displayed. To hide them again, redraw the view (page 46).

Polygon cut properties

Use the **Polygon cut** properties in the property pane to view and modify the properties of a polygon cut.

Note that the polygon cut properties are available in the property pane only after a polygon cut has been created and selected. You cannot access or modify the cut properties before the cut is created.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	Name of the polygon cut.
Profile	Profile of the polygon cut, by default parametric profile `BL`.
Material	Material of the polygon cut, by default `ANTIMATERIAL`. The cut material cannot be changed.
Class	Use to group polygon cuts.
Position	
At depth	Position of the polygon cut in depth direction.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (UDAs) of the cut. UDAs can be used to store more information for each cut, for example, to control the dimension creation in drawings.

Part cut properties

Part cut uses the properties of the cutting part. For example, if the cutting part is a steel beam, part cut uses the **Steel beam cut** properties. The default part cut properties depend on the used cutting part.

If the cutting part is a hollow section, the cut is created with a similar, but non-hollow section, so that the inside of the cut is removed as well.

Note that the part cut properties are available in the property pane only after a part cut has been created and selected. You cannot access or modify the cut properties before the cut is created.

Chamfer part corners and edges

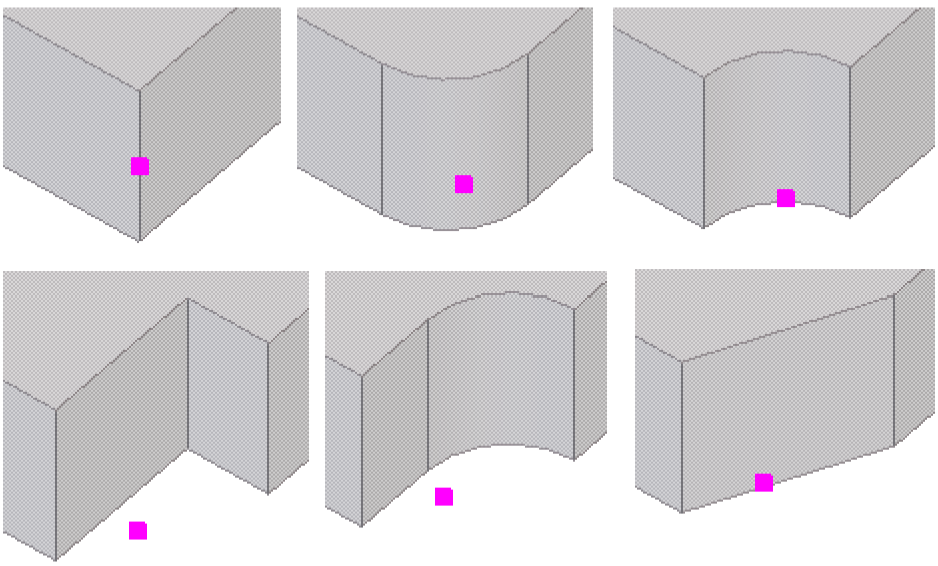
Chamfers are modeling details that can be used to refine the shape of parts for aesthetic, practical, and manufacturing reasons. In Tekla Structures, you can chamfer part corners and part edges.

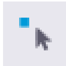
Limitations:

- Only the following parts have corner chamfers: contour plates, concrete slabs, strip footings, steel and concrete polybeams, and concrete panels.
- The end points of a part do not have corner chamfers. The handles that you select must be at corner points or between two segments of a part.

Chamfer part corners

When Tekla Structures creates a part, by default it has a rectangular chamfer at each corner, which does not change the geometry of the part. You can modify the default chamfers.



TIP To make it easier to select the handles of part corners, ensure that the **Direct modification** switch  is **not** active.

1. Select the part.
2. Double-click the handle of a part corner.

   The **Corner chamfer** properties open.
3. Modify the chamfer properties.
4. Select the handles of the part corners you want to modify.
5. Click **Modify**.

Chamfer part edges

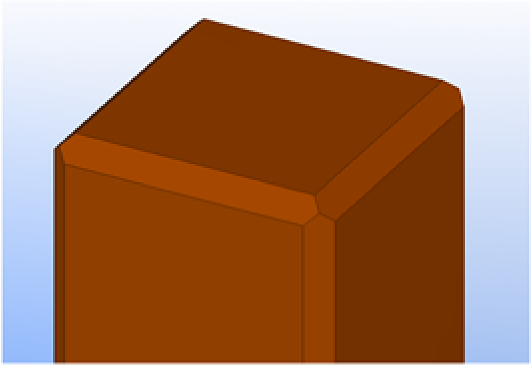
1. Double-click the view to open the **View Properties** dialog, click the **Display...** button, and ensure that the **Cuts and added material** option is **not** selected in the display settings.
2. On the **Edit** tab, click **Chamfer edge**.

   Alternatively, you can start the command in the **Object type list** in the property pane.
3. Select the part you want to chamfer.
4. Pick a point where you want the chamfer to start on the part edge.
5. Pick a second point where you want the chamfer to end on the part edge.

   Tekla Structures displays the chamfer in light blue.
6. If needed, you can modify the chamfer.
   a. Double-click the chamfer to open the **Edge chamfer** properties.
   b. Change the chamfer properties.
   c. Click **Modify**.

   TIP Alternatively, use the contextual toolbar to modify the edge chamfer.
7. Right-click the view and select **Redraw view**.

Tekla Structures removes the chamfered edge.



Corner chamfer properties

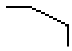
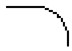
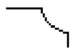
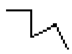
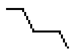
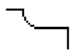
Use the **Corner chamfer** properties in the property pane to view and modify the properties of a corner chamfer. To open the properties, double-click the handle of chamfered corner.

The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals** .

Setting	Description
Shape	
Type	Shape of the chamfer. See **Corner chamfer types and dimensions** for more information.
X / Distance X / Radius **Y / Distance Y / Radius**	Dimensions of the chamfer. The dimension depends on the chamfer type.
Dz1 **Dz2**	Only used for contour plates and concrete slabs. Moves the top or bottom surface of the part corner in the part’s local z direction. Use these options, for example, to give plates varying thicknesses.

Corner chamfer types and dimensions

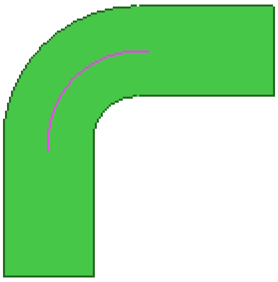
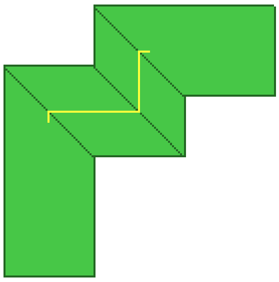
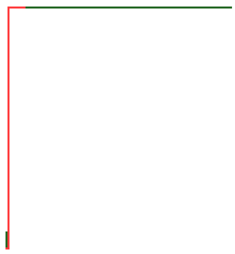
The table below describes the available corner chamfer types and dimensions. The chamfer type numbers can be used in sketches and custom components. Straight chamfers can have different dimensions in two directions. Curved chamfers only use one dimension.

Number	Type	Symbol	Dimensions
0	**None**		x: not used y: not used
1	**Line**		x:the distance in the x coordinate direction from the corner y: the distance in the y coordinate direction from the corner
2	**Rounding**		x: the radius y: not used
3	**Arc**		x: the radius y: not used
4	**Arc point**		x: not used y: not used
5	**Square**		The chamfer is perpendicular to the edges. x: the distance in the x coordinate direction from the corner y: the distance in the y coordinate direction from the corner
6	**Square parallel**		The chamfer is parallel to the opposite edge. x: the distance in the x coordinate direction from the corner y: the distance in the y coordinate direction from the corner
7	**Line and arc**		x (if smaller than y): the arc radius x (if bigger than y): the distance in the x coordinate direction from the corner y (if smaller than x): the arc radius y (if bigger than x): the distance in the y coordinate direction from the corner

Status of corner chamfers in polybeams

To see the corner chamfer lines of polybeams, set the advanced option XS_DRAW_CHAMFERS_HANDLES to `CHAMFERS` or to `CHAMFERS_AND_HANDLES`.

Tekla Structures shows the status of polybeam chamfers using the following colors:

Color	Description	Example
Magenta	Correct chamfer	
Yellow	Correct chamfer that cannot be unfolded	
Red	Incorrect chamfer	

Edge chamfer properties

Use the **Edge chamfer** properties in the property pane to view and modify the properties of an edge chamfer. To open the properties, double-click the edge chamfer when the chamfer if visible in the model. The file name extension of an edge chamfer property file is `*.cha`.

The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals** .

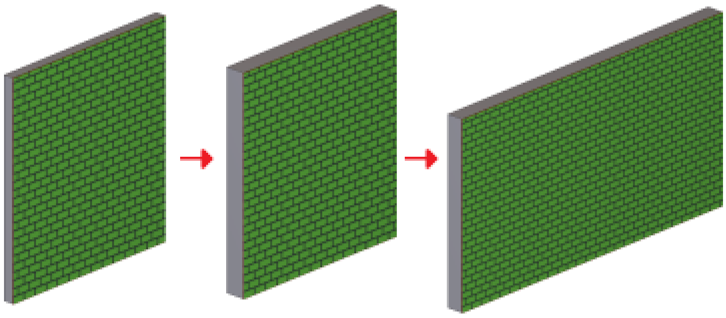
Setting	Description	More information
Shape on edge		
Type	Shape of the chamfer.	
Distance X	Defines how far away from the chamfered	

Setting	Description	More information
	edge the chamfer will end in the x direction.	
Distance Y	Defines how far away from the chamfered edge the chamfer will end in the y direction.	
Shape at end		
First end type	The shape and position of the first end point.	The options are: • **Full**: The end point is positioned at the end of the part (moving along the nearest edge), and the shape is straight. • **Straight**: The end point is positioned at the point you pick, and the shape is straight. • **Bevelled**: The end point is positioned at the point you pick, and the shape is angled.
Second end type	The shape and position of the second end point.	
Distance	The distance between the (picked) end point and the bevelled points.	
General		
Name	Name of the chamfer.	

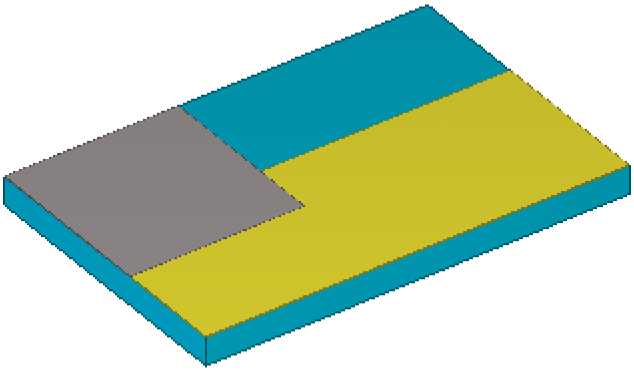
Add surface treatment to parts

Use the surface treatment tools to add surface treatment to parts. Surface treatment for concrete parts include flat finishes, surface mixes, and tiles. Surface treatment for steel parts include fire-proofing and unpainted areas, for example.

When you modify the shape or size of a part, Tekla Structures automatically modifies the surface treatment to fit the part (page 416).



When you create overlapping surface treatments, the smaller surface treatment overrides the larger one. The overlapping area is recognized in reports: only the topmost (visible) surface treatment is calculated.



Add surface treatment to an entire part face

1. On the **Edit** tab, click **Surfaces** --> **Surface treatment to part face** .
2. Pick the origin of the surface treatment.
3. Pick a point to indicate the direction of the surface treatment.
4. Select the part to apply the surface treatment to.
   a. Move the mouse cursor over a part. Tekla Structures highlights the faces that you can select.
   b. Select the part face.

Add surface treatment to a selected area on a part face

1. On the **Edit** tab, click **Surfaces** --> **Surface treatment to selected area** .
2. Pick the origin of the surface treatment.
3. Pick a point to indicate the direction of the surface treatment.

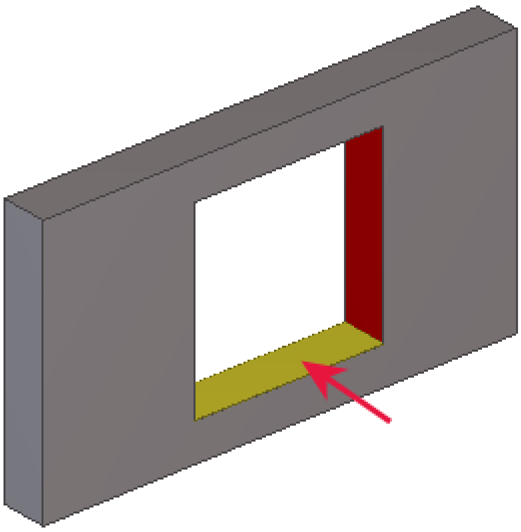
4. Select an area of the part face to apply the surface treatment to.
   a. Move the mouse pointer over a part. Tekla Structures highlights the faces that you can select.
   b. Select the part face.
   c. Pick three or more points on the part face to define a polygonal area.

Add surface treatment to all faces of a part

1. On the **Edit** tab, click **Surfaces** --> **Surface treatment to all faces of part** .
2. Select the part to apply the surface treatment to.

Add surface treatment to cut faces

1. On the **Edit** tab, click **Surfaces** and then either **Surface treatment to part face** or **Surface treatment to selected area**.
2. Pick the origin of the surface treatment.
3. Pick the direction.
4. Select the cut face to apply the surface treatment to:



5. If you are using the **Surface treatment to selected area** command, pick the points to define the area of the surface treatment.

Surface treatment on chamfered parts

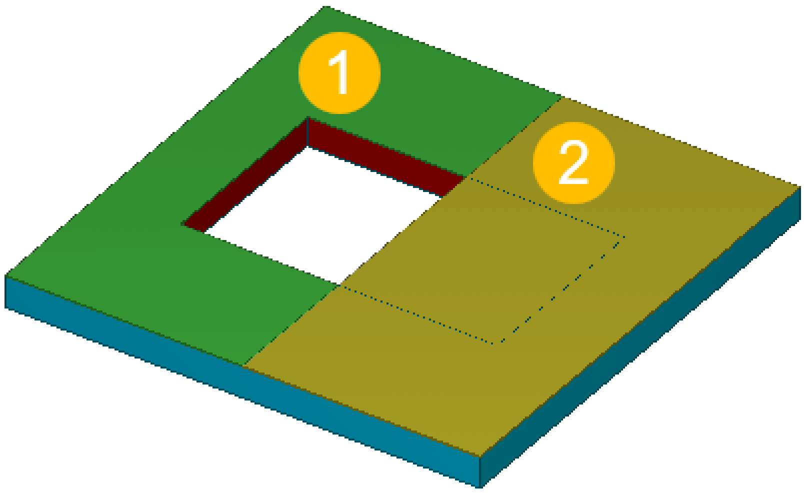
Take these things into account when adding surface treatment to chamfered parts:

- Surface treatment does not work on sketched profiles with chamfers.

- Add surface treatment before chamfering the part. If surface treatment is applied to a chamfered part, the surface treatment chamfer cannot be modified later on.
- The chamfers for the main part and surface treatment are separate. Modifying the main part chamfer does not affect the surface treatment chamfer.
- The orientation of unsymmetrical chamfers depends on the face where it was created (such as top, bottom, left, or right). To change the orientation of an unsymmetrical chamfer, you must swap the chamfer's x and y values.

Surface treatment on parts with openings and recesses

To force Tekla Structures to consider openings and recesses in parts when adding surface treatment, set the **Cut by father part cuts** to **Yes** in the **Surface treatment** properties.



(1) The green surface treatment has the **Cut by father part cuts** set to **Yes**.

(2) The tiled surface treatment is not cut by the cut in the part: **Cut by father part cuts** is set to **No**.

NOTE If you use the **Surface treatment to all faces of part** command and set the **Cut by father part cuts** to **Yes**, Tekla Structures automatically adds surface treatment also to the cut faces.

Modify surface treatment properties

1. If the property pane is not open, double-click the surface treatment to open the **Surface treatment** properties.
2. Change the properties as needed.

3. Click **Modify**.

Note that if you modify the **Pattern** properties, you need to first click **Modify** in the property pane and then redraw the view to make the changes visible.

Surface treatment properties

Use the **Surface treatment** properties in the property pane to view and modify the properties of a surface treatment. To open the properties, double-click the surface treatment. The file name extension of a surface treatment property file is `*.srf`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the surface treatment. The name can contain a maximum of 61 characters.
Type	Select the type of surface treatment.
Subtype	Select the subtype of a specific surface treatment.
Material	Depending on the type of the surface treatment, select the material of the surface treatment.
Color	Depending on the type of the surface treatment, set the color of the surface treatment.
Thickness	Depending on the type of the surface treatment, enter the thickness of the surface treatment.
Cut by father part cuts	To force Tekla Structures to consider openings and recesses in parts when adding surface treatment, set this to **Yes**.
Position	
In depth	Select the location of the surface treatment, and set the **Depth offset** value.
Pattern (for tiled surface treatment)	
Pattern	If you have set **Tile surface** as the **Type** of the surface treatment, select

Setting	Description
	the pattern of the tiled surface treatment.
Tile width **Tile height**	Define the tile width and height.
Mortar height **Mortar width**	Define the mortar height and width.
Tile color **Mortar color**	If needed, use the color picker to pick the tile and the mortar color.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (UDAs) of the surface treatment. UDAs provide more information about the surface treatment.

Define new surface treatment subtypes

You can add new options to the **Subtype** list in the **Surface treatment** properties by editing the `product_finishes.dat` file.

NOTE This section is for advanced users.

1. Copy the `product_finishes.dat` file to your firm, project, or model folder. The file is located under the `\ProgramData\Trimble\Tekla Structures\<version>\environments` folder. The exact file location may vary depending on the folder structure of your environment files.
2. Open the copied file using any text editor.

   The first section of the file explains the available types of surface treatment. The surface treatment types are hard-coded, so do not modify this section:

   ```
   // Product finishes
   // -------------------------
   //
   // Type          : Type of surfacing
   //                   1 = concrete finish
   //                   2 = special mix
   //                   3 = tile surface
   //                   4 = steel finish
   ```
3. Go to the sections that define the options for each type of surface treatment:

```
// ==========================================
// *** Concrete Finish
// ==========================================
// WET FINISH
// ----------
1             MF        "Magnesium Float"
1             SMF       "Smooth Magnesium Float"
1             WT        "Wet Trowel"
```

4. Add rows to define new options.

   a. Define the surface treatment type. For example, 1 for concrete finish.

   b. Define a code for the surface treatment option. For example, MF for Magnesium Float.

   c. Define the full name of the surface treatment option. For example, Magnesium Float. Remember to enclose the name in double quotes " ".

5. Save the file.

See also

Add surface treatment to parts (page 402)

Tiled surface treatment

Tekla Structures includes complex tile and brick surface treatment options, such as basketweave and herringbone patterns. Tiled surface treatment options are based on repeating tile patterns that are stored in XML format.

The tile pattern surface treatment are available in **Surface treatment** properties if you have set the **Type** to **Tile surface**.

NOTE	This section is for advanced users.

Define a new tile pattern

1. Copy the TilePatternCatalog.xml file to your firm, project, or model folder. The file is located under the \ProgramData\Trimble\Tekla Structures\<version>\environments folder. The exact file location may vary depending on the folder structure of your environment files.

2. Open the copied file using any text editor.

3. Add a new <TilePattern> element to the file.

The `<TilePattern>` element must have `<HOffset>` and `<VOffset>` elements and at least one `<Tile>` element. Other elements are optional.

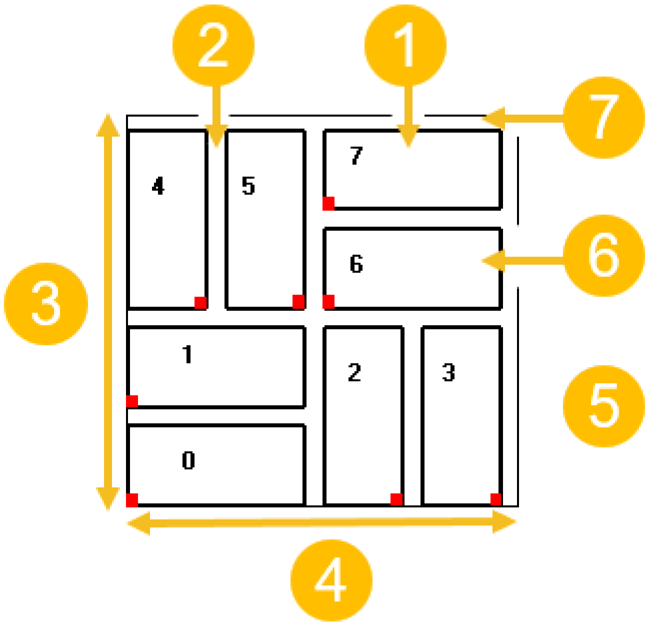
TIP You may find it easier to copy one of the existing elements, and then modify it to suit your needs.

4. Repeat adding `<TilePattern>` elements for all the patterns you want to define.
5. Save the `TilePatternCatalog.xml` file.

Example tile pattern definition

This example explains how the **Basketweave** tile pattern is defined in the `TilePatternCatalog.xml` file.

The **Basketweave** pattern block is made up of eight tiles:



(1) Tile width

(2) Mortar width

(3) `VOffset`

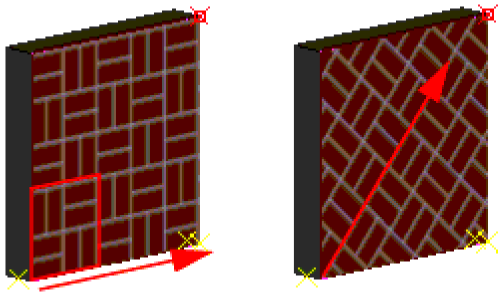
(4) `HOffset`

(5) Red marks indicate `TileOrigin`. Angle value for vertical tiles is 90.

(6) Tile height

(7) Mortar height

The pattern is repeated in the x and y direction of the surface treatment, starting from the origin of the surface treatment. You can run the pattern in different x directions:



In the `TilePatternCatalog.xml` file, the pattern is defined as follows:

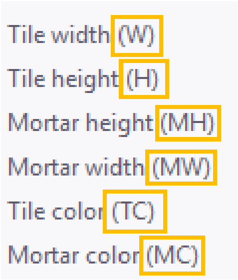
```
<TilePattern Name="Basketweave">
        <Parameter Name="W" DefaultValue="220">
                <Label> _Tile_Width </Label>
        </Parameter>
        <Parameter Name="H" DefaultValue="100">
                <Label> _Tile_Height </Label>
        </Parameter>
        <Parameter Name="TH" DefaultValue="100">
                <Label> _Tile_Thickness </Label>
        </Parameter>
        <Parameter Name="MH" DefaultValue="20">
                <Label> _Mortar_Height </Label>
        </Parameter>
        <Parameter Name="MW" DefaultValue="20">
                <Label> _Mortar_Width </Label>
        </Parameter>
        <HOffset>
                <Vector2D X="W+2*H+3*MW" Y="0" />
        </HOffset>
        <VOffset>
                <Vector2D X="0" Y="W+2*H+3*MH" />
        </VOffset>
        <Tile Angle="0" Width="W" Height="H" Thickness="TH">
                <TileOrigin>
                        <Vector2D X="0" Y="0" />
                </TileOrigin>
```

(1) Name of the pattern

(2) Size of the pattern block in the x direction, after which the pattern repeats

(3) Size of the pattern block in the y direction, after which the pattern repeats

The definition file uses the same symbols as the **Surface treatment** properties:



Tile pattern definitions

The predefined tile patterns that are available in the **Surface treatment** properties are stored in the following files:

File	Description
`TilePatternCatalog.xml`	• Contains the tile pattern definitions. • Located under the `\ProgramData\Trimble\Tekla Structures\<version>\environments` folder.
`TilePatternCatalog.dtd`	• Document Type Declaration (DTD) file that defines the elements allowed in the `TilePatternCatalog.xml` file. • Located in the same folder as the `TilePatternCatalog.xml` file.
Thumbnail images	• The images that appear in the **Pattern** section in the **Surface treatment** properties. • Located in the `..\ProgramData\Trimble\Tekla Structures\<version>\Bitmaps` folder. • File names identify the pattern types. For example, `herringbone.bmp` illustrates the herringbone pattern type.

Tile pattern elements

The `TilePatternCatalog.xml` file can contain the following elements:

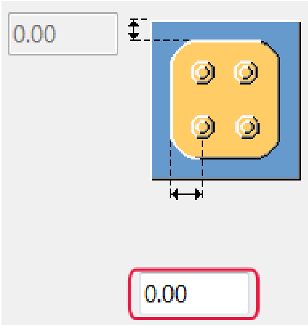
Element	Description
`TilePatternCatalog`	The container for tile patterns. Required.
`TilePattern`	Tile pattern element. Required. This element can contain the following elements listed in this table.
`HOffset`	Horizontal offset of the tile pattern. Required.
`VOffset`	Vertical offset of the tile pattern. Required.
`Tile`	The individual tiles used in a tile pattern. At least one required.
`Color`	Color of the tile or mortar, defined by the RGB values (0–255). Optional.
`Parameter`	Creates an attribute for any element in the `TilePattern`. Optional.
`Label`	The label that identifies a parameter in the **Surface treatment** properties. Optional.
`TileOrigin`	The origin of an individual tile, defined from the origin of the pattern. Optional.

Create an unpainted area using the No paint area component

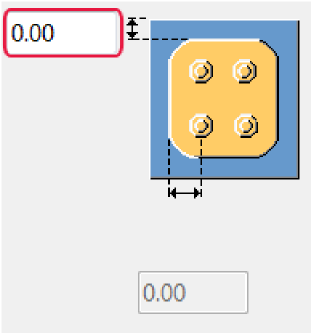
You can create an unpainted area between bolted steel parts using the **No paint area** component.

1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
2. Search for **No paint area**.
3. On the **General** tab:
    a. Click the **Load bolt standards** button to display the available bolt standards, and select the relevant standards.
    b. Select the clearance location from the **Create for** list.

- Define the hole tolerance.



- Define the contact area offset.



c. In the **Allowable gap** box, enter the maximum distance that can exist between two plates so that the surface treatment can be created.

4. On the **Surfacing attributes** tab:

   a. Select one of the following **Surfacing attributes**.

      - `standard` surface treatment properties file
      - A custom surface treatment properties file

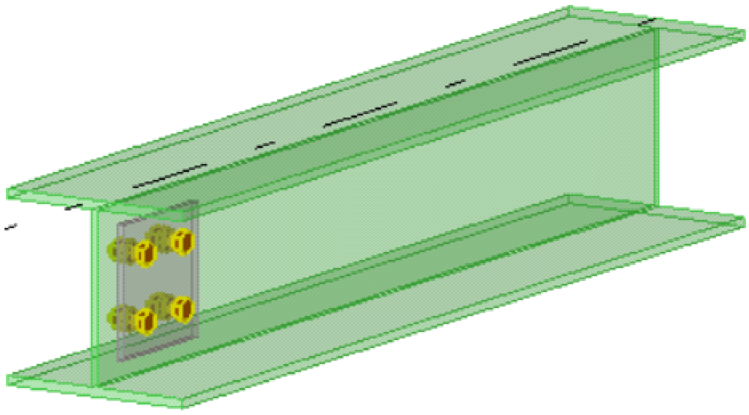
        You can create your own property files in the **Surface treatment** properties. The **Type** has to be **Steel finish** and the **Subtype** has to be **UP - Unpainted**.

      - ...

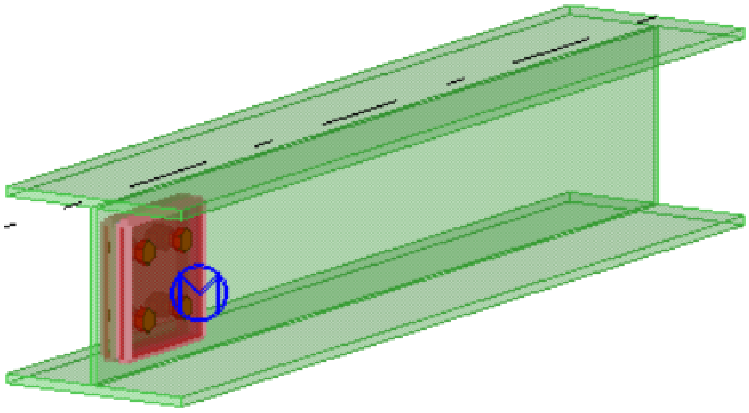
        Define the custom attributes and the position for the surface treatment.

5. Click **OK**.

6. Select a bolt group in the model.



The unpainted area is created between the bolted parts.



See also

Add surface treatment to parts (page 402)

Add surfaces to part faces and pour object faces

You can add surfaces to the faces of parts and pour objects in the model. You can add surfaces to any face geometry, for example curved faces. You can use surfaces for calculating surface areas, such as formwork areas, or for defining face-specific concrete covers for rebar sets.

Each surface is added to a single flat or curved face. Surfaces do not extend over fillet edges between flat and curved faces.

Surfaces are associated with the objects they are attached to. Surfaces cannot exist as standalone objects. A surface can be associated either with a cast-in-place part or a pour object, but not with both. Surfaces associated with parts or pours can be shown in reports as belonging to those objects.

If you modify the geometry of the associated part, the surface will adapt to the changes. If you copy, move, or delete the associated part, the surface will follow. If a part and one or more of its surfaces are selected and copied or moved, all the surfaces of the part will follow.

If a surface is added to a pour object, it will not automatically adapt to changes that only affect the pour object, such as inserting pour breaks. The surface will not be copied if the parts that form the pour object are copied.

You can also copy and move surfaces separately from the objects they are attached to, but only within the same object type, i.e. from a part to another part, or from a pour object to another pour object. The source object face and the target object face need to be similar enough and facing in the same direction. The point picked when the source surface was created must also land on the target face.

Limitations:

- Surfaces do not recognize faces created by displaying objects with high accuracy, such as curved profile fillets.
- Surfaces do not have handles, so their geometry cannot be modified separately from the associated object.
- Surfaces are not shown in drawings.

Add a surface to a face

1. On the **Edit** tab, click **Surfaces** --> **Add surface to face**.
2. Depending on whether you want to create the surface on a part or on a pour object, use a part view or a pour view (page 447).

   To switch between part and pour views, click **Pour view** on the **Concrete** tab.
3. Select the part face or pour object face to which you want to add the surface.

   Tekla Structures adds the surface using the **Surface** properties in the property pane.

   If you modify the properties, Tekla Structures uses the new properties the next time you create an object of the same type.

Modify surface properties

1. If the property pane is not open, double-click the surface to open the **Surface** properties.
2. Change the properties as needed.

   For example, you can define the surface type and whether you want the surface to be cut by the holes in the part or pour object.

   If you want to use the surface for defining a specific concrete cover thickness (page 558) for rebar sets at this part face or pour object face, set

Type to **Concrete cover** and then enter the value in the **Concrete cover** box in the **Rebar set** section.

3. Click **Modify**.

Modify the adaptivity of reinforcement, surface treatment, or edge chamfers in parts

Reinforcement, surface treatment and edge chamfers adapt to the parts they are linked to. For example, reinforcement, surface treatment and edge chamfers automatically adapt to changes in part geometry and size. You can modify the adaptivity settings either for the entire model or for each model object separately. If you modify the adaptivity of individual model objects, these modifications override the default settings that you may have defined for the entire model.

The options are:

- **Off**: adaptivity is not defined
- **Relative**: handles retain their relative distances to the nearest part faces in relation to the part's overall size
- **Fixed**: handles retain their absolute distances to the nearest part faces

Define default adaptivity settings

You can define default adaptivity settings that affect the entire model.

1. On the **File** menu, click **Settings** --> **Options** , and go to the **General** settings.
2. Under **Default adaptivity**, select one of the options.
3. Click **OK** to save the changes.

Modify the adaptivity of an individual model object

You can modify the adaptivity settings for each reinforcement or surface treatment separately. These modifications override the default settings that you may have defined for the entire model.

1. In the model, select the reinforcement (page 561) or surface treatment (page 402) whose adaptivity settings you want to change.
2. Right-click, select **Adaptivity**, and then select one of the options.

Display detailing of a part

In some cases it is useful to see in the model all the objects that are connected to a part, such as components, welds, fittings, reinforcement, and surfaces. You can then examine, for example, whether parts are welded correctly.

1. Select a part.
2. Click **Display detailing** on the contextual toolbar.

   Alternatively, you can use **Quick Launch**.

Tekla Structures displays all bolts, welds, cuts, fittings, and other details belonging to the part, even if you had defined them as hidden in the display settings (page 647). For concrete parts, Tekla Structures displays also reinforcement, surface treatment, and surfaces.

See also

Add details to parts (page 341)

Adjust how model objects are displayed (page 642)

2.5 Work with assemblies

In Tekla Structures, assembly is a structure that consists of one or several parts or items and possibly other objects, such as bolts, welds and reinforcement. When a part or an item is modeled, Tekla Structures automatically creates an assembly for it. This means that also a loose part or an item has an assembly. When you model more parts in Tekla Structures, you may need to merge several parts into one assembly or join several assemblies together.

Building up assemblies

There are different methods on how to form assemblies in the model and achieve the desired assembly structure. The structure can be flat where the assemblies are on one level, or there can be sub-assemblies on several assembly levels in the assembly structure.

Try out the different ways of composing assembly structures to find out which ones suit your purposes. Note that the different methods to form the assemblies may affect your drawings and reports. We recommend you to test the different outputs early on in the modeling phase to ensure that all the relevant assembly information is included in the needed deliverables, such as production drawings.

If needed, you can later add parts and assemblies to existing assemblies, or otherwise modify the assembly structure (page 426) to produce the desired outcome.

Additionally, consider the direction how you form the assembly: it might be useful to build the assembly from bottom to top, so that you first create sub-assemblies and then form a higher-level assembly (page 419) that includes the sub-assemblies.

NOTE Concrete cast units (page 430) are technically considered as assemblies, and they have similar commands available as steel assemblies. In some cases, the assembly commands need to be used for cast units as well.

Hierarchical assemblies

A sub-assembly is an assembly that is added to another assembly to form a hierarchical nested assembly (page 419). A hierarchical assembly consists of the highest level (main) assembly, and one or more levels of sub-assemblies under it. Hierarchical assemblies support the manufacturing process of large assemblies, by enabling the combining of assemblies from smaller sub-assemblies into a larger one.

Note that the *assembly hierarchy* affects the drawing and report outputs. Report and drawing templates need to be made with similar hierarchy as the assembly hierarchy in the model in order to output the data from the correct assembly levels.

Assembly types and assembly hierarchy

Before you start to build up the assemblies in the model, consider the needed assembly structure, how it needs to be composed, and how is affects the drawings and reports. There are several methods how the assembly structure can be formed, depending on the needed structure.

Method	Assembly type	More information
Bolt or weld parts to an existing assembly as secondary parts.	Basic assembly	Use bolts and welds to create and connect assemblies and sub-assemblies (page 424)
Add parts to an existing assembly a secondary parts.	Basic assembly	Add parts to an existing assembly (page 427)
Bolt or weld assemblies to an existing assembly as sub-assemblies.	Nested assembly	Use bolts and welds to create and connect assemblies and sub-assemblies (page 424)
Add parts or assemblies to an existing assembly as sub-assemblies.	Nested assembly	Create nested assemblies (page 427)
Join existing assemblies together without adding any loose parts.	Nested assembly	Join existing assemblies (page 427)

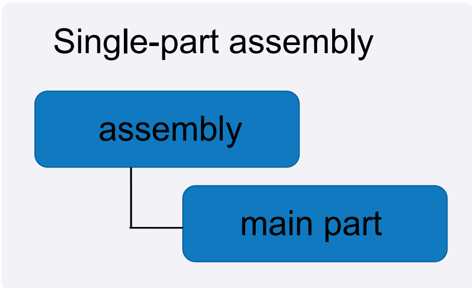
TIP To easily check the structure and hierarchy of an assembly, use the Assembly tree viewer tool from Tekla Warehouse.

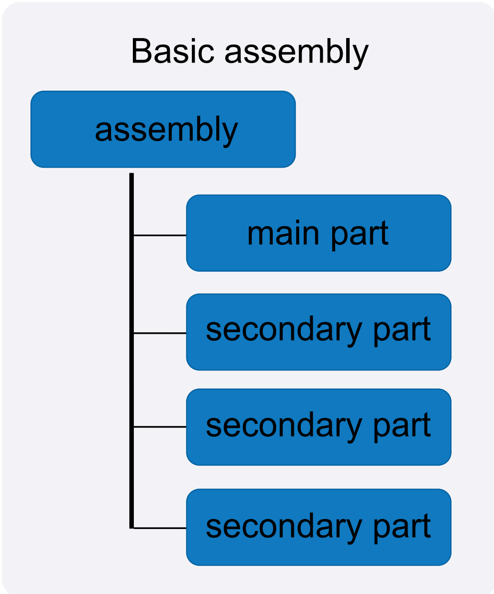
Assembly types

Single-part assembly and basic assembly

A *single-part assembly* consists of one single part. When a part is modeled, Tekla Structures automatically creates an assembly for it.

A *basic assembly* consists of a main part with secondary parts, and the assembly is on one assembly level.





Tekla Structures creates basic assemblies when you:

- Use workshop bolts or workshop welds to manually connect parts together.
- Apply a component that automatically creates workshop welds or workshop bolts.
- Use the **Add to assembly** command to add parts to an existing assembly as secondary parts.

The main part in an assembly has other parts welded or bolted to it, or added with the **Add to assembly** command. By default, the main part is not welded or bolted to any other parts. You can change the main part if needed.

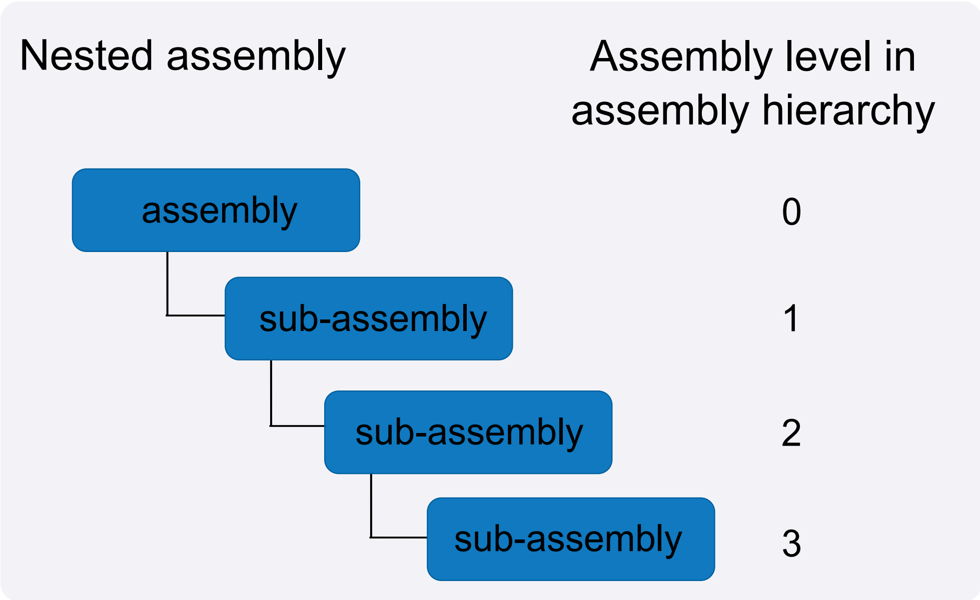
Nested assembly

A *nested assembly* consists of a parent assembly and sub-assemblies which are on the lower assembly levels in the assembly hierarchy.

Tekla Structures creates a nested assembly when you:

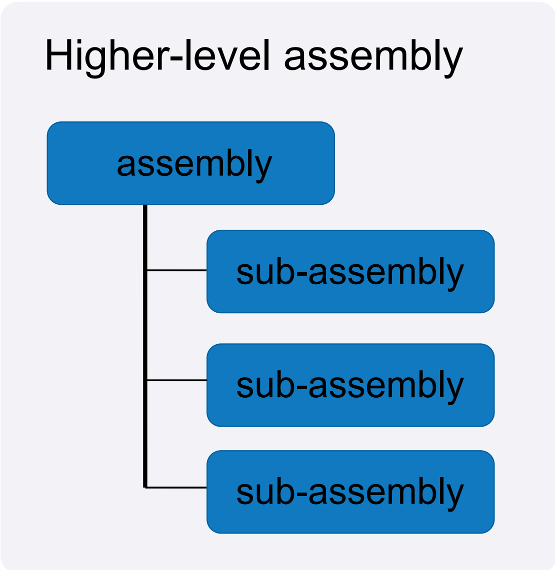
- Use bolts or welds to connect assemblies to an existing assembly as sub-assemblies.

- Use the **Add as sub-assembly** command to add parts or assemblies to an existing assembly as sub-assemblies.



Another type of a nested assembly is created when you join existing assemblies as sub-assemblies. The sub-assemblies are on the same assembly level.

Use the **Join as sub-assemblies** command to join existing assemblies together. You can add assemblies as sub-assemblies, but not any loose parts (parts or items).



NOTE Sub-assemblies in a nested assembly retain their own assembly information and main part. You can define the properties separately for the sub-assemblies and the parent assembly.

To select assemblies in different assembly hierarchy levels, place the mouse pointer on any part in the assembly, press **Shift** and scroll up or down with the middle mouse button.

How to check the assembly hierarchy in a nested assembly

You can work on any level of a *nested assembly*, from single parts and bolts, through the sub-assemblies, up to the highest level of the nested assembly.

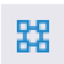
The active selection switch defines on which level you start and toward which direction you move in the assembly hierarchy. The status bar shows the level of the assembly hierarchy you are on.

The assembly on the highest hierarchy level gets the number 0, and the assemblies or objects inside an assembly lower in the hierarchy get the number 1, 2, and so on. You can continue to level 9.

- When the  **Select assemblies** selection switch is active:

  Start from the assemblies on the highest level, move to their sub-assemblies, and finally select single parts, bolts, and so on.

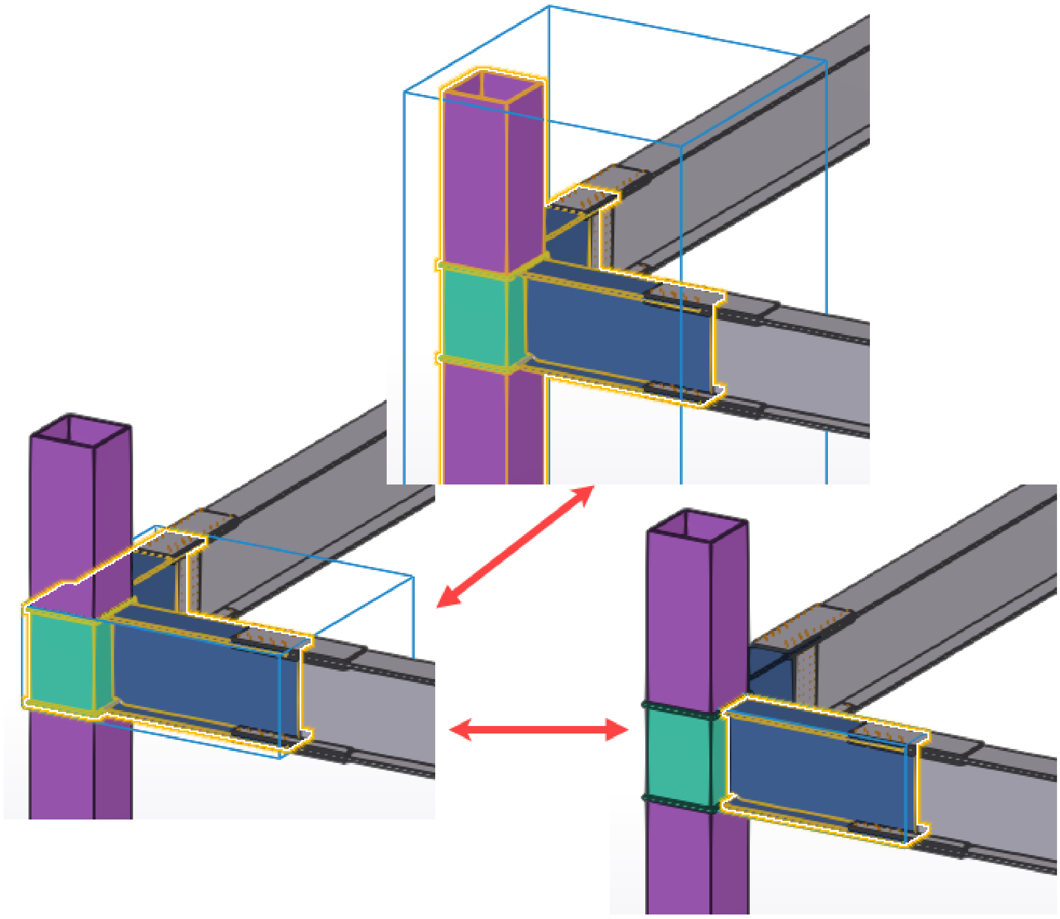
  To shift to a specific level in an assembly, place the mouse pointer on any part in the assembly, press **Shift** and scroll up with the middle mouse button to highlight the objects on the lower levels in assembly hierarchy, level by level.

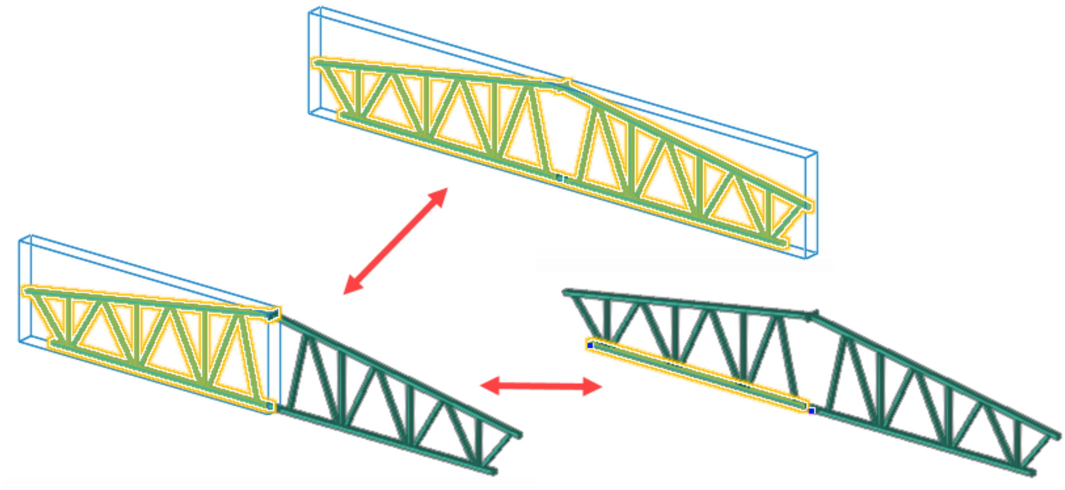
- When the  **Select objects in assemblies** selection switch is active:

  Start from single objects and move to higher levels of nested assemblies.

  To shift to a specific level in an assembly, place the mouse pointer on any part in the assembly, press **Shift** and scroll up with the middle mouse button to highlight the objects on the higher levels in assembly hierarchy.

The blue highlighting box indicates the assembly that you can select.





The assembly hierarchy in nested assemblies affects drawings and reports. Report and drawing templates need to be made with similar hierarchy as the assembly hierarchy in the model to output the data from the correct assembly levels.

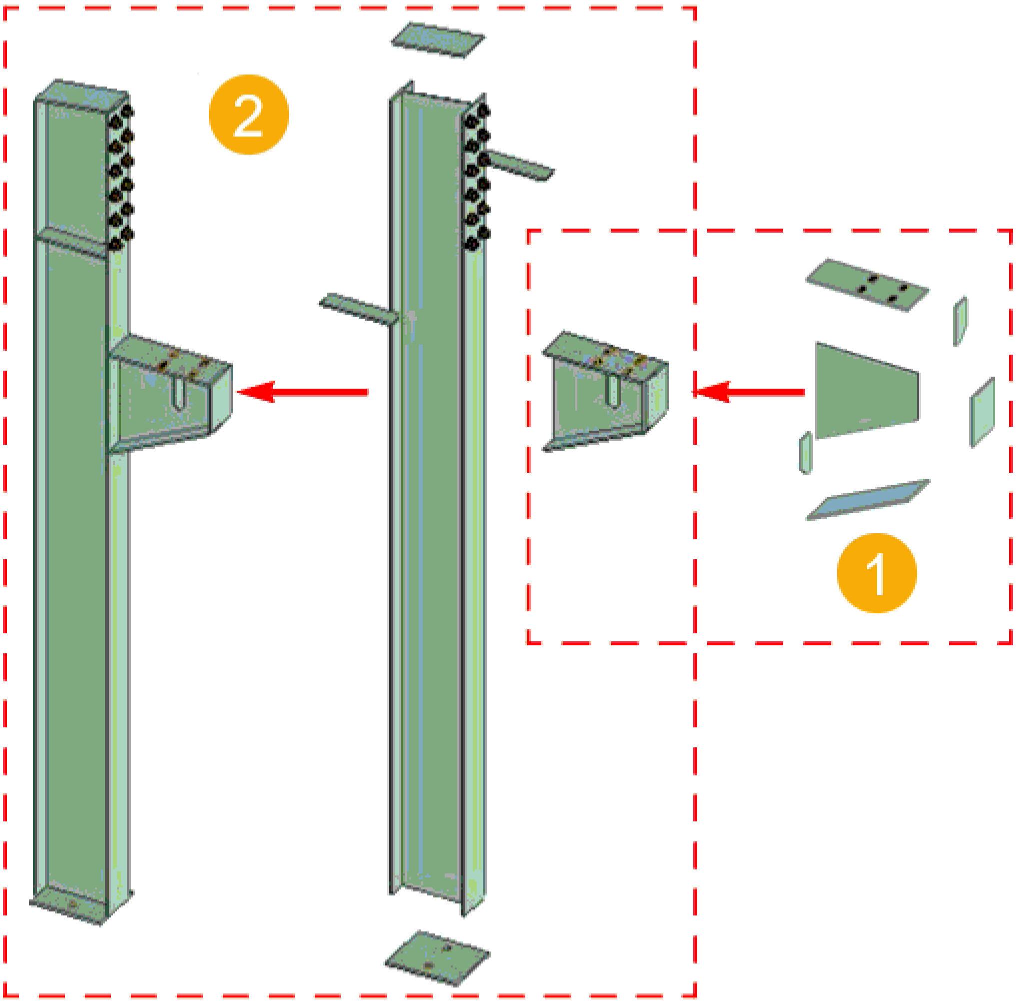
You can create separate drawings and reports of the sub-assemblies and the nested assembly, and still produce dimensions, marks, fabrication information, and so on for all the assembly levels.

For more information how to work with the assembly hierarchy, watch the Working with multi-level assemblies video.

Assembly examples

Column corbel

A column corbel is fabricated in one workshop, and then attached to the column in another workshop. Model the corbel as a sub-assembly of the column. Then create an assembly drawing for each workshop: one assembly drawing showing how the corbel is welded together, another assembly drawing showing how the corbel and the other parts are welded to the column.



(1) Drawing 1, Workshop 1

(2) Drawing 2, Workshop 2

Complex truss

Model the halves of a complex truss as assemblies. Create assembly drawings for the workshop to fabricate the truss halves. Then create another assembly drawing showing how the halves should be joined on site.

Built-up profile

In a frame of built-up columns and beams, each built-up profile can be a sub-assembly. You can create an assembly drawing showing the entire frame, and separate drawings showing how the columns and beams are constructed.

Use bolts and welds to create and connect assemblies and sub-assemblies

You can use bolt and weld properties in the property pane to create assemblies, or to connect sub-assemblies to an existing assembly.

Use bolts to create and connect assemblies

Assemblies and their main parts are automatically defined when you create workshop bolts. Additionally, you can connect more parts to assemblies by using bolts, or create nested assemblies by connecting sub-assemblies to an existing assembly.

To control how Tekla Structures creates the assembly, use the **Connect as** and **Bolt type** lists in the **Bolt** properties in the property pane

The order in which you select parts when creating the bolt determines the main and secondary parts of the assembly, or the assembly hierarchy.

Assembly type	Connect as	Bolt type	Result
Basic assembly	**As secondary part**	**Workshop**	Basic assembly with the part you are bolting as a secondary part. The first part you select usually becomes the main part in the assembly.
Nested assembly	**As sub-assembly**	**Workshop** or **Site**	Nested assembly with the assembly

Assembly type	Connect as	Bolt type	Result
			you are bolting as a sub-assembly. The first part you select determines the assembly to which you are bolting.
	As secondary part	**Site**	No assembly created.

Bolt secondary parts to an assembly

1. Open the **Bolt** properties in the property pane.
2. In the **Connect as** list, select **As secondary part**.
3. Select the main part to which the secondary parts will be bolted.
4. Select the secondary parts.
5. Click the middle mouse button to finish selecting parts.
6. Pick a point to indicate the bolt group origin.
7. Pick a second point to indicate the bolt group x direction.

Bolt sub-assemblies to an assembly

1. Open the **Bolt** properties in the property pane.
2. In the **Connect as** list, select **As sub-assembly**.
3. Select a part in the assembly to bolt to.
4. Select a part in the sub-assembly to be bolted.
5. Click the middle mouse button to finish selecting parts.
6. Pick a point to indicate the bolt group origin.
7. Pick a second point to indicate the bolt group x direction.

Use welds to create and connect assemblies

Assemblies and their main parts are automatically defined when you create workshop welds. Additionally, you can connect more parts to assemblies by using welds, or create nested assemblies by connecting sub-assemblies to an existing assembly.

To control how Tekla Structures creates the assembly, use the **Connect as** and **Workshop/Site** lists in the **Weld** properties in the property pane

The order in which you select parts when creating the weld determines the main and secondary parts of the assembly, or the assembly hierarchy. The first part you select becomes the main part of the assembly. Tekla Structures dimensions secondary parts relative to the main part in assembly drawings.

Assembly type	Connect as	Bolt type	Result
Basic assembly	**As secondary part**	**Workshop**:	Basic assembly with the part you are welding as a secondary part. The first part you select usually becomes the main part in the assembly.
Nested assembly	**As sub-assembly**	**Workshop**: or **Site**:	Nested assembly with the assembly you are welding as a sub-assembly. The first part you select determines the assembly to which you are welding.
	As secondary part	**Site**:	No assembly created.

Weld secondary parts to an assembly

1. Open the **Weld** properties in the property pane.
2. In the **Connect as** list, select **As secondary part**.
3. Select the part to weld to.
4. Select the parts to be welded.

Weld sub-assemblies to an assembly

1. Open the **Weld** properties in the property pane.
2. In the **Connect as** list, select **As sub-assembly**.
3. Select a part in the assembly to weld to.
4. Select a part in the sub-assembly to be welded.

Modify the assembly structure

You can modify the assembly structure by adding more parts or assemblies, or by changing the main part of an assembly, for example.

Concrete cast units (page 430) are technically considered as assemblies, and they have similar command available as steel assemblies. In some cases, the assembly commands need to be used for cast units as well.

NOTE The active selection switch controls whether you can select assemblies or parts when you use the different assembly commands.

Add parts to an existing assembly

You can add secondary parts to a basic assembly, or to any level of a nested assembly.

1. Ensure that the **Select objects in assemblies** or **Select objects in components** selection switch is active.
2. Select the parts you want to add.
3. Right-click and select **Assembly** --> **Add to assembly**.
4. Select the assembly to which you want to add the parts.

Create nested assemblies

You can add parts and assemblies as sub-assemblies in another assembly on a lower level in the assembly hierarchy.

1. Ensure that the **Select assemblies** selection switch is active.
2. Select the parts or assemblies you want to add to another assembly.

   They will become sub-assemblies in the nested assembly.
3. Right-click and select **Assembly** --> **Add as sub-assembly**.
4. Select the assembly to which you want to add the parts or assemblies.

Alternatively, you can use the **Assembly** --> **Add as sub-assembly** command on the ribbon.

Join existing assemblies

You can join existing assemblies as sub-assemblies without adding any loose parts. The joined sub-assemblies are on the same assembly level. For example, you can model two halves of a complex truss as assemblies, and then join the halves together to form a higher-level assembly that contains the halves as sub-assemblies. The assembly with the largest volume becomes the main assembly.

1. Ensure that the **Select assemblies** selection switch is active.

2. Select the assemblies you want to join.
3. Right-click and select **Assembly** --> **Join as sub-assemblies**.

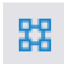
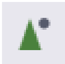
Alternatively, you can use the **Assembly** --> **Join as sub-assemblies** command on the ribbon.

NOTE This command adds a new assembly hierarchy level on top, and the resulting new assembly contains only sub-assemblies.

If needed, you can change the main assembly. Use the **Assembly** --> **Set as main object** command on the ribbon.

Create a sub-assembly of assembly parts

You can create a sub-assembly of parts that are already in an assembly.

1. Ensure that the **Select objects in assemblies**  or **Select objects in components**  selection switch is active.
2. Select the parts you want to include in the sub-assembly.

   Note that you need to select several parts to create the sub-assembly.
3. Right-click and select **Make into Sub-Assembly**.

Check and highlight objects in an assembly

Use the **Inquire** tool to check which objects belong to a particular assembly, cast unit, or rebar assembly.

1. On the ribbon, click the down arrow next to , and then select **Assembly objects** to inquire the assembly objects.
2. Select an assembly, cast unit, or rebar assembly. Ensure that you are on the intended assembly hierarchy level.

   Tekla Structures highlights the other objects that belong to the same assembly, cast unit, or rebar assembly. The following colors are used:

Object type	Highlight color
Steel part - main part Main reinforcement object in rebar assembly	orange
Steel part - secondary part Secondary reinforcement object in rebar assembly	yellow
Concrete - main part	magenta

Object type	Highlight color
Concrete - secondary part	cyan
Reinforcement in cast unit	blue

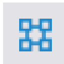
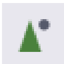
Change the main part in an assembly

The *main part* in an assembly has other parts welded or bolted to it. By default, the main part is not welded or bolted to any other parts. You can change the main part in an assembly.

1. If needed, check what is currently the main part of the assembly.

   Use the **Inquire** tool to check which objects belong to a particular assembly.

   To easily check the structure and hierarchy of an assembly, use the Assembly tree viewer tool from Tekla Warehouse.

2. Ensure that the **Select objects in assemblies**  or **Select objects in components**  selection switch is active.
3. Select the new main part.
4. Right-click and select **Assembly** --> **Set as new main part of assembly**.

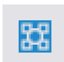
   Tekla Structures changes the main part.

Alternatively, you can use the **Assembly** --> **Set as main object** command on the ribbon.

Use this command also to change the main part in a concrete cast unit.

Change the main assembly

If you have joined existing assemblies together, the assembly with the largest volume has become the main assembly. You can change the main assembly within the sub-assemblies.

1. Ensure that the **Select assemblies**  selection switch is active.
2. Place the mouse pointer on any part in the assembly, press **Shift** and scroll with the middle mouse button in the assembly hierarchy to see the sub-assemblies.
3. Select the new main assembly.

4. Right-click and select **Set as new main sub-assembly**.

   Tekla Structures changes the main sub-assembly. When the assembly is inquired, Tekla Structures highlights the main sub-assembly's main part in orange.

Alternatively, you can use the **Assembly** --> **Set as main object** command on the ribbon.

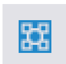
Remove objects from assemblies

If you want to disassemble your assembly, you can remove objects from an assembly, or ungroup the objects belonging to an assembly.

Remove a part or a sub-assembly from an assembly

1. Select the part or sub-assembly you want to remove.
2. Right-click and select **Assembly** --> **Remove from assembly**.
3. To remove another part or sub-assembly, restart the command.

Explode an assembly or a sub-assembly

1. Ensure that the  **Select assemblies** selection switch is active.
2. Select the assembly or sub-assembly you want to explode.
3. Do one of the following:
   - To explode the entire assembly, right-click and select **Assembly** --> **Explode**.
   - To only explode the sub-assembly, right-click and select **Assembly** --> **Explode Sub-Assembly**.

When you explode a nested assembly, always start from the highest assembly hierarchy level. Tekla Structures breaks the assembly hierarchy level by level and ungroups the objects that belong to the assembly. You need to use the **Explode** command several times to break a nested assembly back to single parts.

You can also use the **Explode Sub-Assembly** to explode sub-assemblies to single parts without breaking the entire assembly hierarchy.

2.6 Create cast units

In Tekla Structures, each modeled concrete part is considered as its own *cast unit*. When you model concrete parts in Tekla Structures, you may need to merge several concrete parts into one cast unit. For example, a single cast unit could consist of a column with corbels. The cast unit can then have the production drawings and other deliverables.

Concrete cast units are technically considered as assemblies, and they have similar commands available as steel assemblies (page 417). In some cases, the assembly commands need to be used for cast units as well.

Create a cast unit

You need to specify which parts form the cast unit. Cast units can include reinforcement, as well as concrete parts.

1. On the **Concrete** tab, click **Cast unit** --> **Create cast unit** .
2. Select the objects you want to include in the cast unit.
3. Click the middle mouse button to create the cast unit.

Check and highlight objects in a cast unit

Use the **Inquire** tool to check which objects belong to a particular cast unit or assembly.

1. On the ribbon, click the down arrow next to , and then select **Assembly objects**.
2. Select a part that belongs to a cast unit or an assembly.

   Tekla Structures highlights the other parts that belong to the same cast unit or assembly. The following colors are used:

Object type	Highlight color
Concrete - main part	magenta
Concrete - secondary part	cyan
Reinforcement	blue
Steel part - main part	orange
Steel part - secondary part	yellow

Change the cast unit main part

The *main part* in a concrete cast unit is the one with the largest volume of concrete. You can change the main part in a cast unit.

1. If needed, check what is currently the main part of the cast unit.

   Use the **Inquire** tool to check which objects belong to a particular cast unit.

2. Ensure that the **Select objects in assemblies** selection switch is active.
3. Select the new main part.
4. Right-click and select **Set as new main part of assembly**.

Define the cast unit type of a part

You must define the cast unit type of concrete parts. Tekla Structures checks the cast unit type of the main part each time you create or modify a cast unit. It is important to use the correct cast unit type, because some functionalities, for example numbering, are based on the cast unit type.

1. Double-click a concrete part to open the part properties in the property pane.
2. Go to the **Cast unit** section.
3. In the **Cast unit** list, select one of the following:
   - **Cast in place**

     Cast units that are constructed fully in their final location.
   - **Precast**

     Cast units that are built at another location and transported to their final location for placement in the full structure.
4. Click **Modify** to save the changes.

Some best practices when you are working with cast units

- Always check that all the concrete parts and related accessories are connected to the cast unit.
- Cast units always have a main part. The main part in a cast unit is usually the part with the largest volume of concrete. You can change the main part. Always ensure that the main part is the intended one as the cast unit gets many of its properties from the main part. Keep the possible sub-assembly hierarchy clear.
- Cast units get their position numbers (page 712) according to the numbering settings. Identical cast units have the same position number and they are included in the same drawing. In addition to the position numbers, you can assign other identifiers to separate the cast units with the same drawing and position number.
- The cast unit drawings contain all the hierarchical content included in the cast unit. Always check that all of the concrete parts, cuts, reinforcement, and surface treatment are connected to the correct parts and that embeds

are added as sub-assemblies to the cast unit. This ensures that all the objects are included in the drawings.

- To indicate the casting direction (page 434) of a precast concrete part, you can define which part face you want to face upwards in the casting form. Therefore, this face will be the front face view in the fabrication drawing. Casting direction affects the numbering of parts. By modeling beams and interior walls from left to right, bottom to top, the default front face is towards the viewer, and any fixed part marks are oriented upright. When using wall layout for exterior walls, model in the clockwise direction.
- If you copy fully detailed cast units in a model, use the **Copy special** --> **Copy all content to another object** command. With this command, all objects in a cast unit are copied to other similar cast units without individually selecting each object to copy. This means that, for example, secondary parts, sub-assemblies, components, and cuts and fittings are copied with the cast unit.

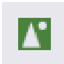
Additionally, you can

- inquire (page 686) the cast unit’s general properties, such as volume and weight which are automatically calculated
- inquire the cast unit’s own properties, such as position number
- inquire the cast unit’s main part properties
- list all parts, rebars and sub-assemblies of the cast unit.

Add objects to cast units

You can use different methods to add objects to cast units. The available methods depend on the objects' material and on the hierarchy you want to create in the cast unit.

- The default way of working is to use the **Add to cast unit** command. Using this command you can add concrete parts that need to be in the same casting unit. The parts retain their geometry and you can use them as input when modeling components, shown in drawings.
- Use the **Add as sub-assembly** command to add embeds. Embeds are always added as sub-assemblies on a lower level in the cast unit hierarchy. Sub-assemblies can consist of one or multiple parts. In both cases the parts have to be added to a cast unit as sub-assemblies. Sub-assemblies in a cast unit retain their own assembly information and their main part. If in a precast cast unit the precast element consists of numerous shells or layers, the secondary cast units need to be added as sub-assemblies to the main cast unit.
- Use the **Attach to part** command to attach one or more concrete parts to another part in the cast unit. Additionally, you can use the command to merge parts if you need them to be reported or drawn as one part.

To	Do this	Available for
Add an object as a secondary part	1. On the **Concrete** tab, click **Cast unit** --> **Add to cast unit**. 2. Select the object you want to add. 3. Select an object in the cast unit.	Concrete, timber, miscellaneous materials
Add an object as a sub-assembly	1. If you are adding a custom part, ensure that the **Select components (custom objects)**  selection switch is active. 2. On the **Steel** tab, click **Assembly** --> **Add as sub-assembly**. 3. Select the object you want to add. 4. Select the cast unit to which you want to add the object.	Steel, concrete, timber, miscellaneous materials

Remove objects from cast units

If you want to disassemble your cast unit, you can remove objects from a cast unit, or ungroup the objects belonging to a cast unit.

Remove objects from a cast unit

1. On the **Concrete** tab, click **Cast unit** --> **Remove from cast unit** .
2. Select the object you want to remove from the cast unit.

   You need to restart the command to remove another object.

Explode a cast unit

You can ungroup the objects belonging to a cast unit.

1. On the **Concrete** tab, click **Cast unit** --> **Explode** .
2. Select an object in the cast unit you want to explode.

Casting direction

To indicate the casting direction of a concrete part, you can define which part face you want to face upwards in the casting form. The top-in-form face is displayed in the front view of a drawing.

Casting direction for concrete and non-concrete parts

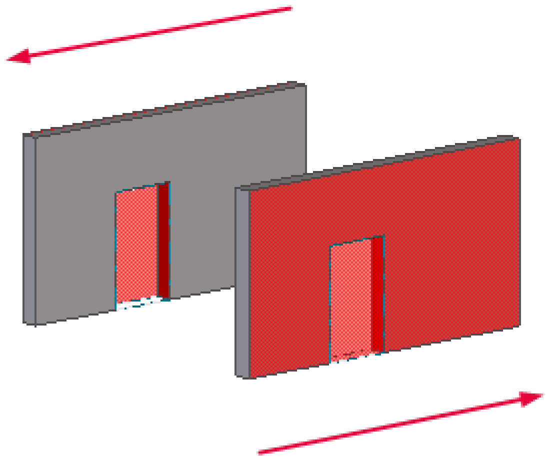
To enable this feature for non-concrete parts as well, and to indicate in the model the part face that is shown in the drawing main (front) view, use the advanced option XS_SET_FIXEDMAINVIEW_UDA_TO_AFFECT_NUMBERING.

The casting direction affects the numbering of parts. If you define the casting direction for parts that differ only by their modeling direction, they get different position numbers. This is because the modeling direction affects the top-in-form face of the parts. By default, the casting direction of the parts is undefined, which means the modeling direction does not affect numbering.

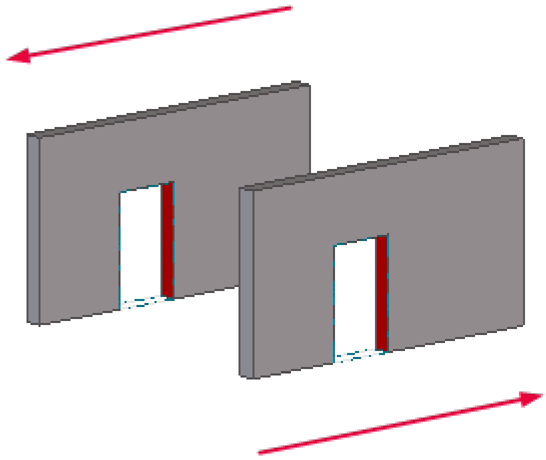
NOTE In drawings, use the **Fixed** coordinate system to show the top-in-form face in the front view.

Example

In the following example, each cast unit gets a **different** position number, because the top-in-form setting and the orientation of the panels is different. The red arrow indicates the modeling direction.



In the following example, the cast units get the **same** position number, because their top-in-form setting has not been defined. The red arrow indicates the modeling direction.



Define the casting direction of a part

You can define the casting direction for concrete parts.

1. Set the rendering of parts to **Rendered** by doing one of the following:
   - On the **View** tab, click **Rendering** --> **Parts rendered**.
   - Press **Ctrl + 4**.
2. On the **Concrete** tab, click **Cast unit** --> **Set top-in-form face**.
3. Move the mouse pointer over the selected concrete part.
4. Select the part face that will face upwards in the form.

TIP Alternatively, you can do this in the user-defined attributes of the part.

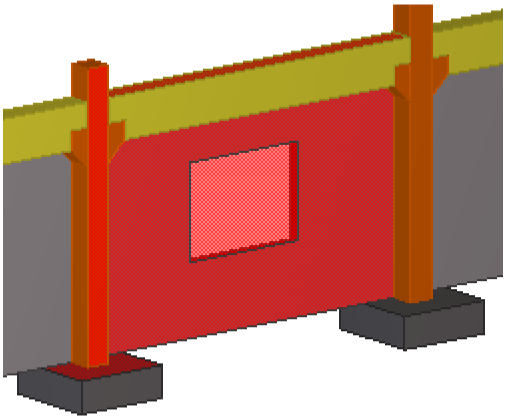
- Concrete parts: Select an option for the **Top in form face** user-defined attribute.
- Non-concrete parts: With XS_SET_FIXEDMAINVIEW_UDA_TO_AFFECT_NUMBERING set to `STEEL`, `TIMBER`, and/or `MISC`, select an option for the **Fixed drawing main view** user-defined attribute.

Show the top-in-form face

You can display the top-in-form face of a concrete part in a model view.

1. On the **Concrete** tab, click **Cast unit** --> **Show top-in-form face**.
2. Click the concrete part whose top-in-form face you want to show.

Tekla Structures highlights the top-in-form face in red:



TIP To hide the top-in-form face again, right-click the view and select **Update window**.

Note that if you have not specifically set the top-in-form value, the **Show top-in-form face** command highlights the default face, which is the front face of the part.

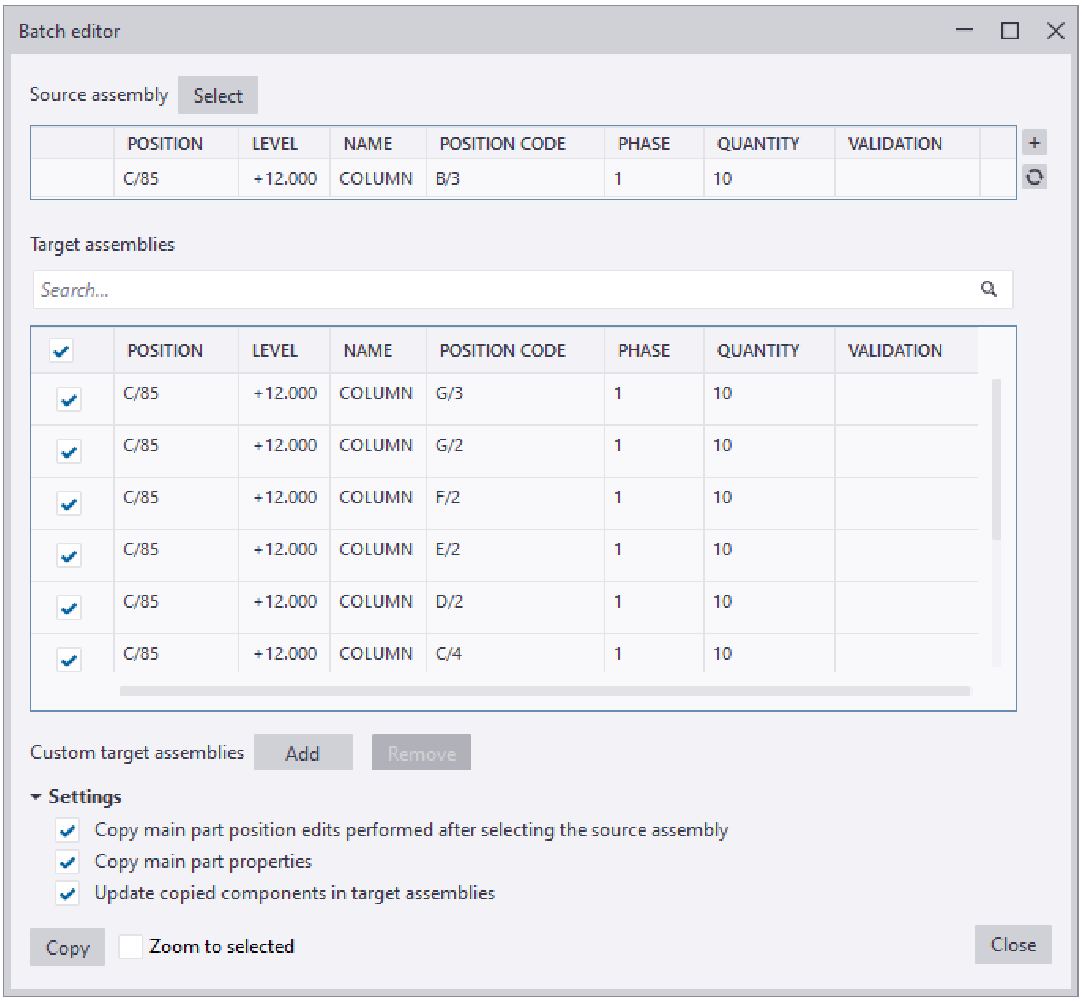
2.7 Batch editing of assemblies or cast units

Using the **Batch editor** tool you can quickly copy modifications done to an assembly or a cast unit to other, identical assemblies or cast units that have the same position number, or to very similar but differently numbered assemblies or cast units.

Batch editor reduces the amount of repetitive work in situations where you need to edit very similar assemblies or cast units by modifying the object geometry or by changing the part properties.

Unlike the **Copy special** --> **To another object** or **Copy special** --> **All content to another object** commands which replace the duplicate objects in the target assemblies or cast units, **Batch editor** detects matching objects in the targets and modifies their geometry and properties without replacing the objects.

This is important especially in the late stages of a project when the detailing of all objects is done and the documentation has been created or issued. For example, replacing of objects and components might damage not only the target but nearby assemblies or cast units which would not require any modifications, and the changes would have a negative impact on the drawings that are ready for issuing.



With **Batch editor** you can

- Select a source assembly or cast unit in the model, and based on the source, generate a list of target assemblies or cast units to which you can copy the edits done in the source. **Batch editor** only detects assemblies or cast units which have the same position number as the source.

  If needed, you can manually add very similar but differently numbered assemblies or cast units to the list of targets.

- Use the list to copy all edits done in the source assembly or cast unit to the specified target assemblies or cast units in one go.

  Batch editor attempts to avoid replacing of modified objects. **Batch editor** detects matching objects in the targets assemblies or cast units and modifies their geometry and properties.

  Both the main and secondary part geometry and property edits, as well as modifications in component properties can be copied. Note that to copy the main part geometry edits, the **Copy main part position edits performed after selecting the source assembly** option needs to be **on**.

If there are secondary parts in the target assemblies or cast units which do not exist in the source anymore, **Batch editor** deletes the excess secondary parts in the targets assemblies or cast units when you copy the edits.

- Use the list of target assemblies or cast units for
  - viewing the differences between the source and the target assemblies or cast units before copying the edits.
  - browsing the list of target assemblies or cast units and fixing possible errors before running numbering.

  Batch editor uses the **Validation** attribute to report detected differences in the number of objects between the selected source and the listed target assemblies or cast units, and to compare the assembly or cast unit weight and volume. The **Validation** attribute also reports any missing or additional objects in the target assemblies or cast units.

Limitations and recommendations in using Batch editor

- **Batch editor** works only on the assembly or on the cast unit level.

  Assembly or cast unit secondary parts cannot be selected as a source or target. For example, you cannot copy edits from one part to another, or from one sub-assembly to another.
- **Batch editor** does not copy the following:
  - values of unique user-defined attributes
  - sub-assembly properties
  - assembly numbering information
- **Batch editor** does not support the following object types:
  - Pour units and pour objects
  - Load object types in analysis and design
- We recommend that you do not use **Batch editor** if the source assembly or cast unit has been split.
- We recommend that you do not use **Batch editor** with mirrored assemblies or cast units.

  Rebar groups and components are not always adapted correctly to the main part. This applies especially to cases where new objects are added to the mirrored target assemblies or cast units.

- We recommend that you do not use **Batch editor** with assemblies or cast units that look similar but are modeled using different methods.

  This applies to, for example, assemblies or cast units that have different orientation of secondary object, or assemblies or cast units that have been modeled using different object types as the main part.
- Rebar groups and some macros do not adapt to the geometry of the main part with different dimensions or different shape.
- The main part geometry edits that are copied to rotated or mirrored assemblies or cast units may not work as expected.
- **Batch editor** overrides the overall dimensions of the parts that the **Wall layout** tool creates. Therefore, we recommend to use **Batch editor** for copying edits to precast walls that have identical dimensions.

Edit similar assemblies or cast units with Batch editor

1. On the **Edit** tab, click **Batch editor**.

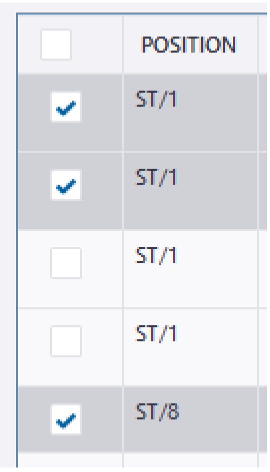
   The **Batch editor** dialog box opens.
2. Click the **Select** button and select a source assembly or a cast unit in the model.

   Batch editor finds all assemblies or cast units with the same position number. The assemblies or cast units are listed in the **Target assemblies** list.
3. To add target assemblies or cast units manually, select them in the model and click the **Add** button.

   To remove target assemblies or cast units from the list, select them in the list and click the **Remove** button.

   Note that if you delete the source in the model, the source list becomes empty. Similarly, if you delete the targets in the model, the list of targets becomes empty.
4. In the **Target assemblies** list, use the check boxes to select the assemblies or cast units to which you want to copy the modifications from the source assembly or cast unit.

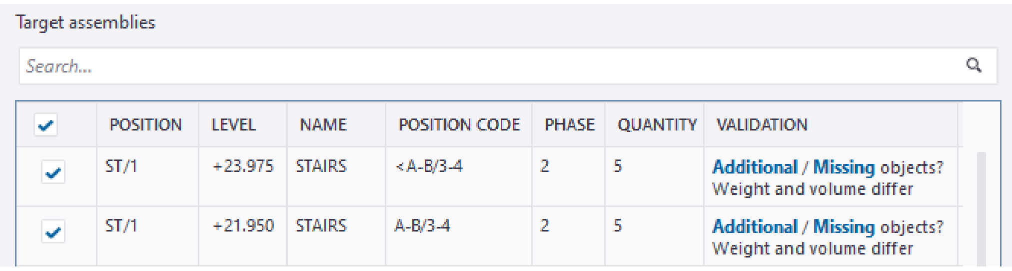
To select several check boxes in one go, select the targets, and hold down the **Shift** key and click the left mouse button.



5. To view the differences between the source and the target assemblies or cast units, or to fix errors, use the **Validation** attribute in the **Target assemblies** list.

   Batch editor uses the **Validation** attribute to report detected differences in the number of objects between the selected source and the listed target assemblies or cast units, and to compare the assembly or cast unit weight and volume. The **Validation** attribute also reports any missing or additional objects in the target assemblies or cast units.

   You can check the missing and additional objects in the model. Click the **Additional** or **Missing** objects link to check the highlighted the objects in the model.



6. To copy the modification from the source assembly or the cast unit to the targets, click the **Copy** button.

   Batch editor analyzes the detected differences between the selected source and the targets, and makes all the selected target assemblies or cast units identical to the source.

7. To review the copying results in the model, select the target objects in the list.

   Select the **Zoom to selected** check box to automatically zoom to the object selected in the list.

8. Number the modified assemblies or cast units to validate that all modifications have been copied properly.

Settings in Batch editor

Use the **Settings** to define how the edits are copied.

- **Copy main part position edits performed after selecting the source assembly**

  When the option is **on**, **Batch editor** copies the geometrical modifications done to the assembly or cast unit main part, such as dragging of end point handles of the main part, or changing the main part location with the **Move** or **Move special** commands.

  NOTE **Batch editor** recognizes only the edits that you have done after selecting the source assembly or cast unit, and switching the option on. If you first modify the main part geometry and only then switch the option on, the main part edits are not copied.

  If the option is **off**, only the secondary part and component edits are copied.

- **Copy main part properties**

  When the option is **on**, **Batch editor** copies the assembly or cast unit main part property modifications to the selected targets. Use this option, for example, when you want to copy the edits to targets with different main part profile dimensions.

  Note that user-defined attributes are copied but unique user-defined attributes are not copied.

- **Update copied components in target assemblies**

  When the option is **on**, **Batch editor** modifies the components in the target assemblies or cast units according to the properties set in the component properties dialog box. All manual changes, such as modifying the geometry of component objects, are lost.

  If the option is **off**, manual changes in the component, for example, modifying the geometry of component objects or the component object properties, are copied to the target assemblies or cast units.

Work with property columns in Batch editor

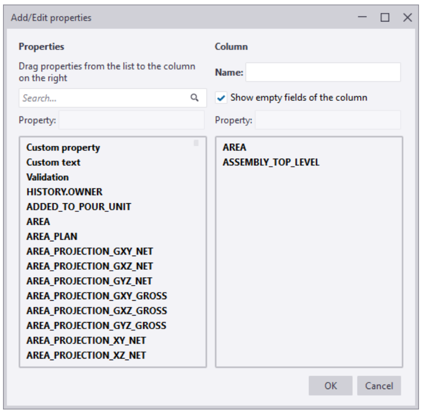
You can organize the list of target assemblies or cast units and the property columns in **Batch editor**. Add, edit, or remove property columns to show the needed properties about the target assemblies or cast units.

<table>
<tr><th>To</th><th>Do this</th></tr>
<tr><td>Add more property columns in the Target assemblies list</td><td>
1. Click the + button in the upper right corner of the Batch editor.

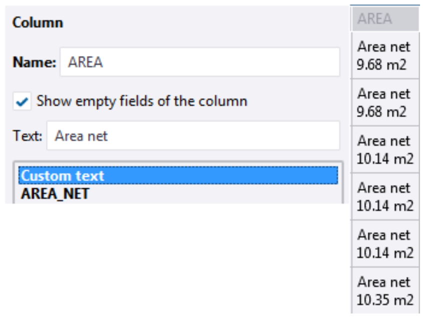
Alternatively, you can right-click the property column header and select Edit.

2. In the Add/Edit properties dialog box, do the following:

• Select the required property from the list on the left and drag it to the list on the right. Use the Search box for searching properties. You can add several properties to the same column.



• If you want to add a custom text in a column cell, select Custom text and type the required text in the displayed Text box. Then press Enter to
</td></tr>
</table>

To	Do this
	add the custom text in the list on the right.  • To show the property cells even though there is no value, select the **Show empty fields of the column** option. 3. Enter the name for the property column and click **OK**. The new property columns are added to the **Target assemblies** view.
Change the order of the property columns	Drag the property column header to a new location in the list of targets. 
Change the sort order of a property column	Click the column header. The arrow symbol next to the column header indicates if the sort order is ascending ▲ or descending ▼.
Resize a property column	Drag the edge between this and the following column header.
Refresh the **Target assemblies** list	Click the refresh icon in the upper right corner of the **Batch editor**.

2.8 Manage pours

With the pour management functionality of Tekla Structures, you can view the geometry of cast-in-place concrete structures, show them as parts or as pour objects, plan pours and pour breaks, and report pour information, such as concrete volumes and formwork areas. You can define pours, pour units, pour objects, and pour breaks for concrete parts whose cast unit type is **Cast in place**.

In Tekla Structures, a *pour object* is a building object that consists of one or more cast-in-place concrete parts, or parts of them. The cast-in-place concrete parts are merged into one pour object if they have the same material grade and they touch each other. They also need to be in the same *pour phase* to be merged. Pour objects are visible in *pour views*.

A *pour unit* is an entity for cast-in-place concrete and it consists of a pour object and all related reinforcement, embeds, and other objects that need to be in place before concrete can be poured on the building site.

A *pour* is a group of pour objects that is poured at one go.

With a *pour break* you can split a pour object into smaller pour objects.

NOTE Pour management is mainly targeted at contractors for quantity take-off, planning, and on-site activities. By default, pour management is disabled in new models in most roles. You can enable pour management (page 445) in the current model using the advanced option XS_ENABLE_POUR_MANAGEMENT.

See also

Enable pour management (page 445)

View cast-in-place concrete structures (page 447)

Define the pour phase of a part (page 450)

Pour objects (page 451)

Pour units (page 453)

Pour breaks (page 458)

Troubleshoot pours (page 464)

Example: Create concrete geometry and work with pours (page 467)

Enable pour management

By default, pour management is disabled in new models in most roles. You can enable pour management in the current model in the **Advanced Options** dialog box.

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 all pour-related modeling work.

1. On the **File** menu, click **Settings** --> **Advanced options** to open the **Advanced options** dialog box.
2. Under **Concrete detailing**, set XS_ENABLE_POUR_MANAGEMENT to `TRUE`.
3. Click **OK**.
4. Save and re-open the model for the change to take effect.

   The commands that show and create pour objects and pour breaks are now available in the model and in drawings.

See also

Disable pour management temporarily (page 446)

Disable pour management temporarily

You can temporarily disable pour management. This may be needed if pour management seems to significantly slow down your model, for example, when the pours and pour objects are very large and require splitting into smaller ones.

When pour management is temporarily disabled, the existing pour objects and pour breaks are still present in the model, but any modifications to the model geometry that would normally update pour objects and pour breaks automatically will not do that. Any information related to pours will be outdated and inaccurate, for example, in reports, and the pour breaks will not be adaptive. They will be updated automatically when you re-enable pour management.

To disable or re-enable pour management:

1. Go to **Quick Launch**, start typing `pours and pour breaks`, and select the **Toggle pours and pour breaks** command from the list that appears.
2. Click **Yes** in the confirmation dialog box.

NOTE If you are working in a Tekla Model Sharing model, remember to re-enable pour management before you write out. Similarly, if you are working in the multi-user mode, re-enable pour management before you save the model. In this way, information related to pours stays up to date for all users of the model.

TIP If you have problems opening a large model with pour objects that contain many parts, you may need to disable pour management before you open the model.

You can do this by modifying the `xs_user.[user name]` file located in the model folder. Set `PAPB` to `0` to disable pours, and then save the file.

When needed, remember to re-enable pour management.

See also

Enable pour management (page 445)

View cast-in-place concrete structures

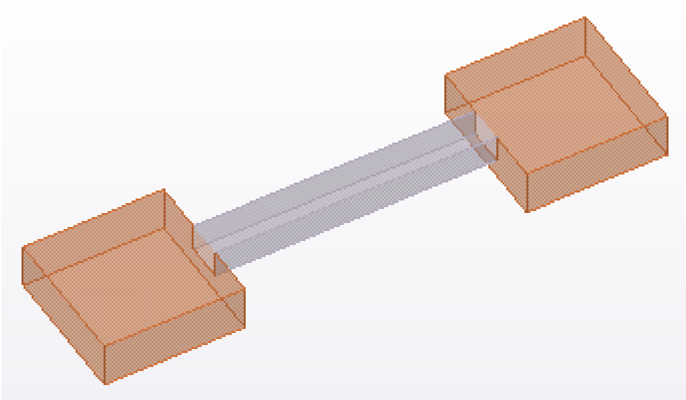
When pour management is enabled, you can view cast-in-place concrete structures in model views either as parts or as pour objects.

Depending on your needs, you can switch between the different representation options for cast-in-place concrete structures. For example, working in a part view is useful when you want to reinforce individual parts or change their geometry. A pour view is useful when you want to find out the volume of concrete to be poured or check which objects belong to a pour unit, or when you want to reinforce continuous structures that span multiple parts.

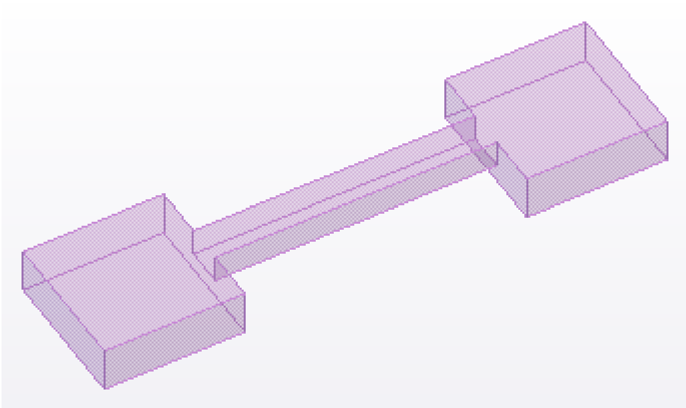
Set the appearance of cast-in-place concrete structures

You can define how cast-in-place concrete structures appear in a model view.

1. Ensure that pour management is enabled (page 445).
2. Double-click the view to open the **View Properties** dialog box.
3. Click **Display** to open the **Display** dialog box.
4. Ensure that the **Parts** check box is selected.
5. In the **Cast in place** list, select either:
    - **Parts**

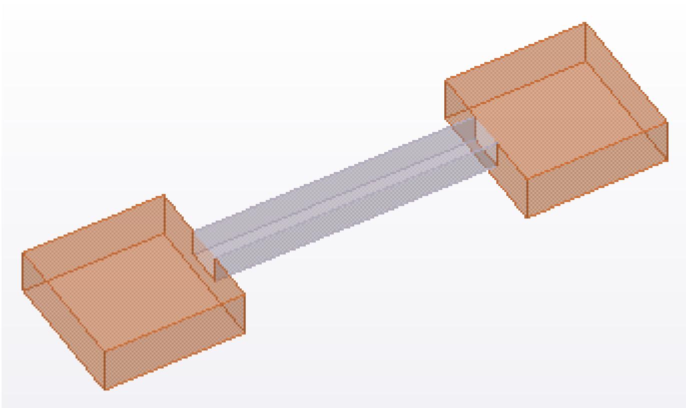
    - **Pours**



6. If you selected **Parts** for the cast-in-place concrete structures, in the **Cast in place parts** list, select either:

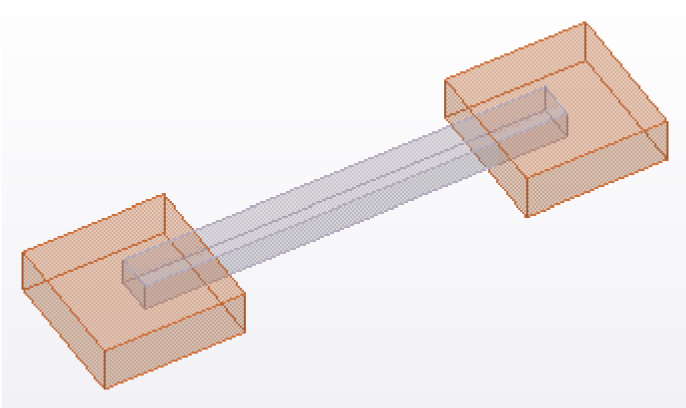
   - **Merged**

     Tekla Structures shows concrete parts as merged in the model if their cast unit type is **Cast in place**, if they have the same material grade and pour phase (page 450) number, and if they touch or overlap one another. When these criteria are met, Tekla Structures removes the outlines of the individual parts within each continuous concrete structure.

   - **Separated**

     Tekla Structures shows concrete parts as individual parts and separated by their outlines.



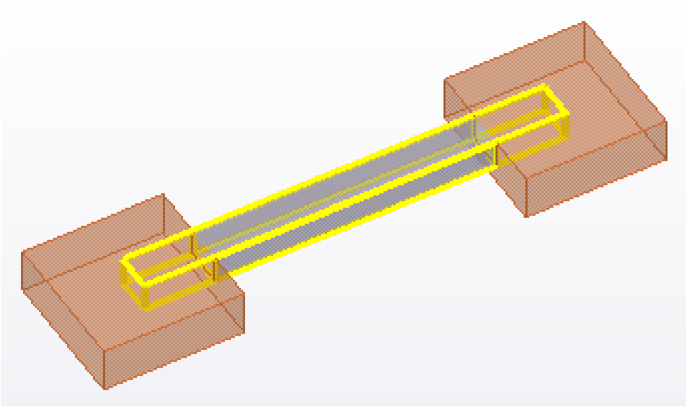
7. Ensure that the view is selected.
8. Click **Modify** to save the changes.

TIP To quickly change the representation of the active view from **Parts** to **Pours**, and the other way round, click  **Pour view** on the **Concrete** tab.

You can also create two views, a pour view and a part view, and keep them both open side by side on your screen.

Part view vs. pour view

Continuous concrete structures cannot be selected or highlighted in part views. When you hold the mouse pointer over a concrete structure in a part view, Tekla Structures highlights the original parts belonging to it. You can select a part and modify it if needed:



Duplicates and overlapping parts are counted only once in the volume calculations for pour objects. Note that single part and cast-unit volumes are still calculated the same way as before, which means that the sum of single part and cast-unit volumes may be higher than the volume of pour objects that are defined from exactly the same part geometry.

When you reinforce a concrete structure, you need to reinforce the individual concrete parts within it in part views, or you can reinforce pour objects by using **Reinforcing Bar Shape Catalog** or rebar sets in pour views. Therefore, you can reinforce a part of a continuous concrete structure independently from the whole continuous concrete structure. All reinforcement are visible both in part views and in pour views.

Define the pour phase of a part

Use the pour phase property to separate pour objects from one another. By defining pour phases, you can prevent cast-in-place concrete parts from merging even if they have the same material grade and they touch or overlap one another.

NOTE Pay attention to pour phases when you create cast-in-place concrete parts. For example, use pour phase 0 for horizontal structures, like beams and slabs, and pour phase 1 for vertical structures, like columns and walls, to separate them to different pour objects. In this way you can make sure that the number of parts included in each single pour object is reasonable, and that your models do not slow down due to too large pour objects.

To modify the pour phase of a part:

1. Double-click a concrete part to open the part properties in the property pane.
2. Under **Cast unit**:
   a. In the **Cast unit type** list, ensure that the cast unit type is set to **Cast in place**.
   b. In the **Pour phase** box, enter a pour phase.

   By default, the pour phase is 0. If you are unable to change the value, the cast unit type that you set in step 2a is incorrect.
3. Click **Modify**.

NOTE When you define pour phases, make sure that parts in different pour phases do not overlap. If you use parts (not pour objects) to report geometric information, overlapping volumes of different pour phases are not merged but counted twice in calculations, and you may end up with incorrect volume, area, or weight information.

See also

View cast-in-place concrete structures (page 447)

Pour objects

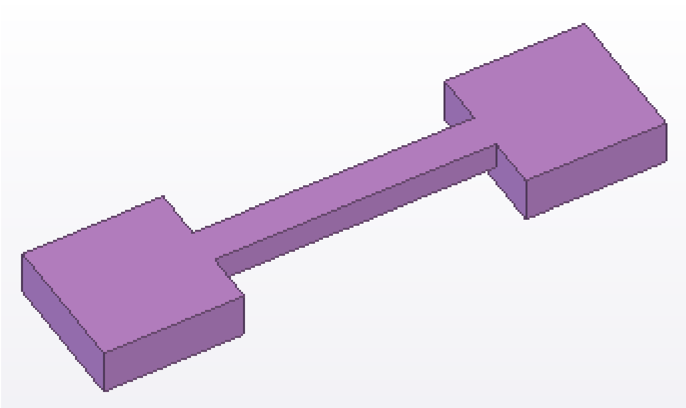
When pour management is enabled (page 445), each concrete part whose cast unit type is **Cast in place** automatically forms a pour object.

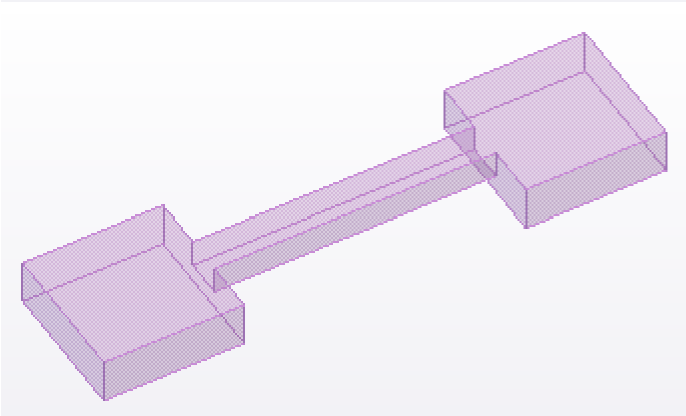
Tekla Structures automatically merges multiple cast-in-place concrete parts to a pour object if they have the same material grade and pour phase (page 450) number, and if they touch or overlap one another.

By creating pour breaks (page 458) you can split pour objects into smaller pour objects.

NOTE Make sure that the number of parts included in each single pour object is reasonable. Too large number of parts and part surfaces in a pour object slows down the model.

Pour objects are visible in pour views (page 447). All pour objects are displayed using the same color, regardless of the individual parts' color within a concrete structure. You can change the default color by using the advanced option XS_POUR_OBJECT_COLOR in **File** --> **Settings** --> **Advanced options** --> **Concrete detailing** .





You can also use different color and transparency settings (page 452) to show pour object groups, for example, by pour number.

TIP You can group pour objects using **Organizer** or **Task manager**.

Limitations

The following commands are not available for pour objects: **Copy**, **Move**, **Delete**, **Split**, and **Combine**. This is because pour object geometry is defined by parts. If you want to change the geometry of pour objects, you have to modify the parts instead of pour objects, or you can create pour breaks.

See also

Modify the properties of a pour object (page 453)

Reinforce pour objects using Rebar shape catalog (page 504)

Create a rebar set (page 470)

Change the color and transparency of pour objects

By default, all pour objects are displayed using the same color in pour views, regardless of the individual parts' color. You can customize the pour object color and transparency in model views by defining object groups and then selecting specific color and transparency settings for each group.

TIP To change the default color of pour objects, use the advanced option XS_POUR_OBJECT_COLOR in **File** --> **Settings** --> **Advanced options** --> **Concrete detailing** .

1. On the **View** tab, click **Representation**.

   The **Object Representation** dialog box is displayed.

2. Create a new object group for the pour objects whose color and transparency you wish to change.

   a. In the **Object Representation** dialog box, click **Object group...**.

   b. In the **Object Group - Representation** dialog box, click **Add row**.

   c. To direct the settings to pour objects instead of parts, select the following options for the row:

      - **Category** = **Object**
      - **Property** = **Object type**
      - **Condition** = **Equals**
      - **Value** = **Pour object**

   d. If needed, add any additional filtering criteria.

      For example, to filter pour objects by a certain user-defined attribute, add a row with **Pour object** as **Category**, and define the **Property**, **Condition**, and **Value** as desired.

e. Enter a unique name in the box next to the **Save as** button.

f. Click **Save as** to save the object group.

g. Click **Close**.

3. Repeat step 2 if you wish to create more object groups.
4. In the **Object Representation** dialog box, select an object group from the **Object group** list.
5. In the **Color** list, choose a color for the object group.
6. In the **Transparency** list, set the transparency of the object group.
7. Click **Modify**.

   The object group’s color and transparency changes in the model.

See also

Change the color and transparency of model objects by using object representation (page 656)

Modify the properties of a pour object

Pour objects have properties and user-defined attributes which you can view, define, and modify.

For example, you can enter a **Pour number** that you can use to define the sequence of pours, and a **Pour type** that you can use to describe each pour object.

1. Ensure that you are using a pour view. If not, click **Pour view** on the **Concrete** tab to show the pour objects.
2. Ensure that the  **Select objects in assemblies** selection switch is active.
3. Double-click the pour object whose properties you want to modify.
4. In the property pane, enter or modify the pour object properties.
5. Click **Modify**.

See also

Pour objects (page 451)

Pour units

When pour management is enabled, you can create pour units that combine together pour objects and other objects. A *pour unit* is an entity for cast-in-place concrete and it consists of a pour object and all related reinforcement, embeds, and other objects that need to be in place before concrete can be poured on the building site.

For each pour object (page 451) in the model, there is a corresponding pour unit to which the pour object belongs. You can automatically add other objects to pour units by using the **Calculate pour units** command. You can also modify the pour units manually.

The following model objects can be added to pour units:

- Reinforcement, such as single reinforcing bars, bar groups, reinforcement meshes and strands
- Assemblies (for example, rebar assemblies and embeds)
- Sub-assemblies (for example, embeds in cast-in-place cast units)
- Bolts (for example, anchor bolts and shear studs)
- Surfaces added to the pour object
- Precast cast units

Precast cast units can only be added manually, not by using the **Calculate pour units** command.

Note that some model objects such as parts and welds cannot be directly added to a pour unit. Instead, these objects are indirectly linked to the pour unit through the assemblies and cast units they belong to.

One model object can be included in only one pour unit at a time.

Calculate pour units

You can have Tekla Structures detect which objects form pour units and automatically add the objects to pour units.

1. Ensure that pour management is enabled (page 445).
2. On the **Concrete** tab, click **Calculate pour units**.

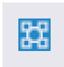
   Tekla Structures adds the objects (page 457) to pour units.

You can check the pour units in a pour view, or using the **Inquire** tool, **Organizer**, or reports.

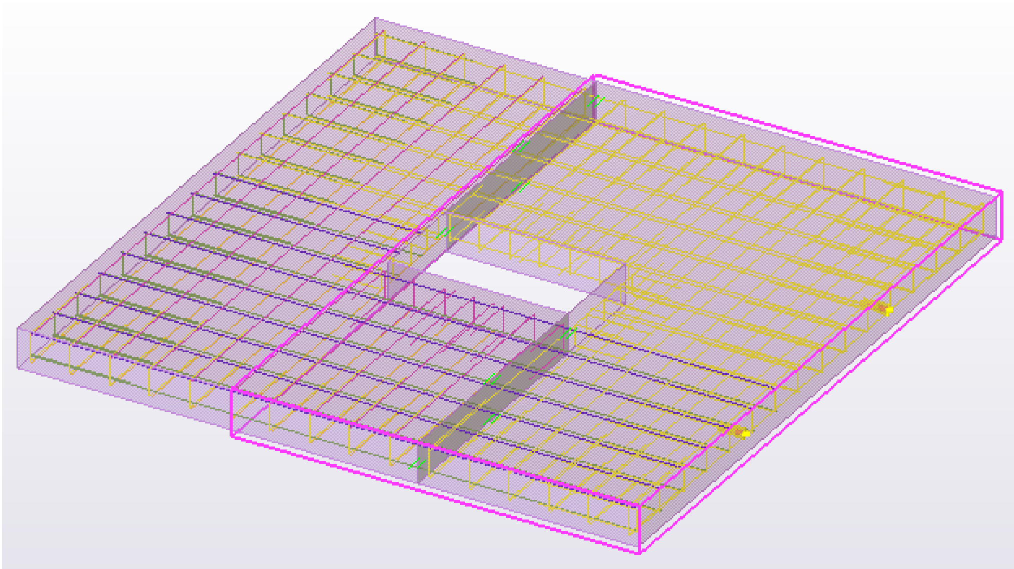
If you want to modify the pour units, you can manually add and remove objects. The manual additions will be kept even if you use the **Calculate pour units** command again, but the objects you manually remove from pour units will be added again.

Check and inquire objects in a pour unit

You can visually check which objects are included in a pour unit. You can also use the **Inquire** tool to get information about the pour unit and the objects in it.

1. Ensure that you are using a pour view (page 447). If not, on the **Concrete** tab, click **Pour view** to show the pour objects.
2. Ensure that the  **Select assemblies** selection switch is active.
3. Click a pour object to select its pour unit.

   A magenta box indicates the pour unit.

4. Right-click and select **Inquire** .

   Tekla Structures lists the objects in the pour unit and shows their properties in the **Inquire object** dialog box.

Add objects to a pour unit

In addition to using the **Calculate pour units** command, you can manually add objects to pour units.

1. Ensure that you are using a pour view (page 447). If not, on the **Concrete** tab, click **Pour view** to show the pour objects.
2. Select the objects that you want to add to a pour unit.

   You can add reinforcement, assemblies, precast cast units, and bolts.

   If you have other objects selected, they will not be added.
3. Right-click and select **Pour** --> **Add to pour unit**.

   Alternatively, you can go to **Quick Launch** and use the **Add selected objects to pour unit** command. You can also assign a keyboard shortcut to this command.
4. Click a pour object to add the objects to the corresponding pour unit.

Tekla Structures adds all the objects that can be added to the pour unit. The unallowed objects are not added.

TIP If you do not have objects selected, you can first start the **Add to pour unit** command by using **Quick Launch** or a customized keyboard shortcut, and then select the object that you want to add to the pour unit.

Remove objects from a pour unit

After using the **Calculate pour units** command, you can manually remove objects from pour units.

1. Select the object that you want to remove from a pour unit.
2. Right-click and select **Pour** --> **Remove from pour unit**.

   Alternatively, you can go to **Quick Launch** and use the **Remove selected objects from pour unit** command. You can also assign a keyboard shortcut to this command.

The removed object can then be added to another pour unit either manually with the **Pour** --> **Add to pour unit** command, or automatically with the **Calculate pour units** command.

TIP If you do not have objects selected, you can first start the **Remove from pour unit** command by using **Quick Launch** or a customized keyboard shortcut, and then select the object that you want to remove from the pour unit.

Reset pour unit relations

In some cases you may need to reset all or some of the pour unit contents and relations defined using the **Calculate pour units** command and/or the **Add to pour unit** command.

To do this:

1. Go to **Quick Launch**.
2. Search for and select the appropriate command of the following:
   - **Reset all pour unit relations**
   - **Reset all manually assigned pour unit relations**
   - **Reset all pour unit relations except manual assignments**
3. In the confirmation dialog box, click **Yes** to reset pour unit relations.

   Note that if you use the **Reset all pour unit relations** command, and then click **No** in the confirmation dialog box to cancel the resetting of the manual assignments, the automatic relations are still reset. If you want to recreate the automatic relations, use the **Calculate pour units** command again.

Modify the properties of a pour unit

You can modify the pour unit properties in the same way as pour object properties, but using a different selection switch.

1. Ensure that you are using a pour view (page 447). If not, click **Pour view** on the **Concrete** tab to show the pour objects.
2. Ensure that the  **Select assemblies** selection switch is active.
3. Double-click the pour unit whose properties you want to modify.
4. In the property pane, enter or modify the pour unit properties.

   For example, you can define the pour unit name and user-defined attributes.
5. Click **Modify**.

How Tekla Structures automatically adds objects to pour units

When you use the **Calculate pour units** command, Tekla Structures automatically adds objects to pour units.

Each object that collides with a pour object, meaning that the object at least partly overlaps the pour object, is added to the same pour unit to which the pour object belongs.

If any object in an assembly collides with a pour object, the entire assembly is added to the pour unit.

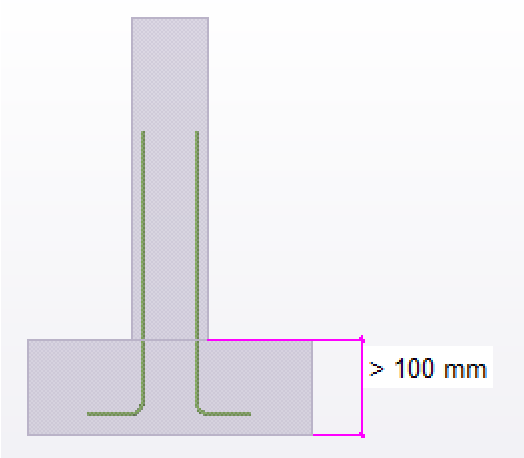
Only reinforcement objects that belong to cast-in-place parts are added to pour units.

If any object in a reinforcing bar group or strand group collides with a pour object, the entire group is added to the pour unit. On the other hand, individual reinforcing bars in a rebar set can be added to different pour units.

Objects colliding with more than one pour object

If an object collides with more than one pour object, the object will be associated with the pour object whose bounding box has the lowest global z coordinate.

For example, the reinforcing bars colliding with a footing pour object and a column pour object are associated with the footing pour object because its bottom face has a lower global z coordinate than the column pour object.



If the lowest global z coordinates of the pour object bounding boxes are the same or differ less than 100 mm, the object will be associated with one of the pour objects according to these rules:

1. If the object's center of gravity is within only one of the colliding pour object bounding boxes, the object will be associated with that pour object.
2. If the object's center of gravity is within more than one pour object bounding box, or completely outside any pour object bounding box, the object will be associated with the pour object whose center of gravity is closest to the object's center of gravity.

If there are changes in pour units

Whenever there are changes in a pour object or pour unit, all associations to that pour unit are reset. Similarly, if there are changes in an object associated with the pour unit, this association is reset. The next time you use the **Calculate pour units** command, only the unresolved associations are calculated.

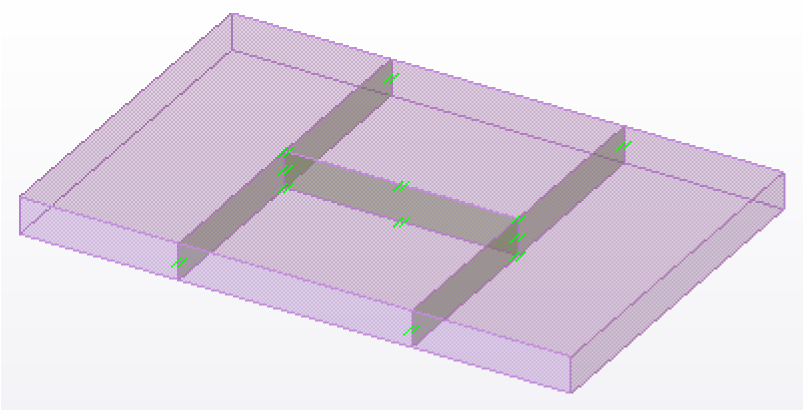
See also

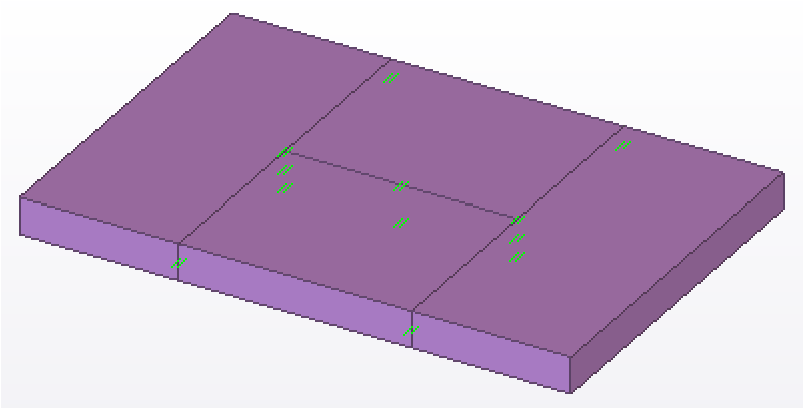
Pour units (page 453)

Pour breaks

When pour management is enabled, you can use pour breaks to split pour objects into smaller pour objects.

Pour breaks are visible and you can work with them both in pour views and in part views (page 447). Pour breaks are displayed as a thin plane or line, depending on the part rendering option (page 642) you use.





WARNING If you move or copy a part, pour breaks do not follow the part. Pour breaks remain at their original locations, and adapt to any cast-in-place concrete parts (page 459) they still touch.

If a pour break does not split a pour object completely into two, the pour break is displayed in red by default. This means that it is invalid and needs to be remodeled.

See also

Set the visibility of pour breaks (page 460)

Create a pour break (page 461)

Modify a pour break (page 463)

Pour break adaptivity

Pour breaks are adaptive to changes in cast-in-place concrete parts and pour objects. This means that if you change the geometry or location of a cast-in-place concrete part or pour object, its pour breaks change accordingly.

If you delete a cast-in-place concrete part, its pour breaks disappear as well.

If you modify a cast-in-place concrete structure in any of the following ways, its pour breaks adapt:

- Change the profile or dimensions of a part
- Add or remove cuts or fittings
- Change chamfer shape or dimensions
- Add or remove parts of the cast-in-place concrete structure by:
    - Changing the cast unit type of a part from **Precast** to **Cast in place** or vice versa
    - Changing the pour phase of a part
    - Changing the concrete grade of a part
    - Moving, copying, or deleting parts

If you move a cast-in-place concrete part outside its pour breaks, the pour breaks disappear. If you move a part so that it still hits one or more pour breaks, the pour breaks that are inside the part stay in their original locations and adapt to the part in the new location.

If you copy or move a pour break and it hits a cast-in-place concrete part in the destination location, the pour break adapts to the part. Also the pour breaks that you copy from another model adapt to the parts in the model they are copied to.

If a pour break is dependent on another pour break that is split or deleted, the dependent pour break is deleted as well. If a pour break is dependent on another pour break that is moved, the dependent pour break adapts inside the pour object, as long as the pour break plane can touch the moved pour break.

If a pour break is split so that it becomes partial, the pour break is deleted. A partial pour break can only split a cast-in-place part or pour object in conjunction with other pour breaks.

Set the visibility of pour breaks

You can show pour breaks in model views.

Before you start, ensure that pour management is enabled (page 445).

1. Double-click a model view to open the **View Properties** dialog box.
2. Click **Display...** to open the **Display** dialog box.
3. Select the **Pour break** check box.
4. Click **Modify**.

See also

[Pour breaks (page 458)]

Create a pour break

You can add pour breaks to pour objects or concrete parts whose cast unit type is **Cast in place**.

You can create pour breaks by picking one, two, or more points in the model.

When you create a pour break that traverses more than two points, the pour break will be limited to the pour object it splits and perpendicular to the current work plane. If you need to create an inclined or horizontal pour break using multiple points, shift the work plane (page 53) first.

TIP Use the  **Snap to nearest points (points on line)** snap switch (page 84) to start or end pour breaks on part or pour object edges.

Use the  **Snap to any position** snap switch (page 84) to pick intermediate points for pour breaks.

To create a pour break, do any of the following:

To	Do this
Create a pour break, perpendicular to a part face, using one point	1. On the **Concrete** tab, click **Pour break** --> **Single point** . 2. Pick the location for the pour break.
Create a pour break that splits all cast-in-place concrete parts and pour objects located between two points	1. On the **Concrete** tab, click **Pour break** --> **Two points**. 2. Pick two points to define the location of the pour break.
Create a pour break using multiple points	1. If needed, shift the work plane. 2. On the **Concrete** tab, click **Pour break** --> **Multiple points** . 3. Pick the points you want the pour break to go through.
Create a pour break defined by the opposite corners of a rectangle	1. If needed, shift the work plane. 2. On the **Concrete** tab, click **Pour break** --> **Multiple points** .

To	Do this
	3. Hover over , and then click on the toolbar that appears. 4. Pick two opposite corner points of the pour break.
Create a pour break defined by the center and one corner of a rectangle	1. If needed, shift the work plane. 2. On the **Concrete** tab, click **Pour break** --> **Multiple points** . 3. Hover over , and then click on the toolbar that appears. 4. Pick the center point of the pour break. 5. Pick one corner point of the pour break.
Create a pour break defined by three corners of a rectangle	1. If needed, shift the work plane. 2. On the **Concrete** tab, click **Pour break** --> **Multiple points** . 3. Hover over , and then click on the toolbar that appears. 4. Pick three corner points of the pour break.
Create a pour break defined by one side midpoint and two corners of a rectangle	1. If needed, shift the work plane. 2. On the **Concrete** tab, click **Pour break** --> **Multiple points** . 3. Hover over , and then click on the toolbar that appears. 4. Pick one side midpoint of the pour break. 5. Pick two corner points of the pour break.

If the pour break you are creating does not split a pour object or cast-in-place part completely into two, Tekla Structures does not add the pour break to the model. You may need to use another **Pour break** command to create a valid pour break, for example **Multiple points** instead of **Single point**.

See also

Modify a pour break (page 463)

Pour breaks (page 458)

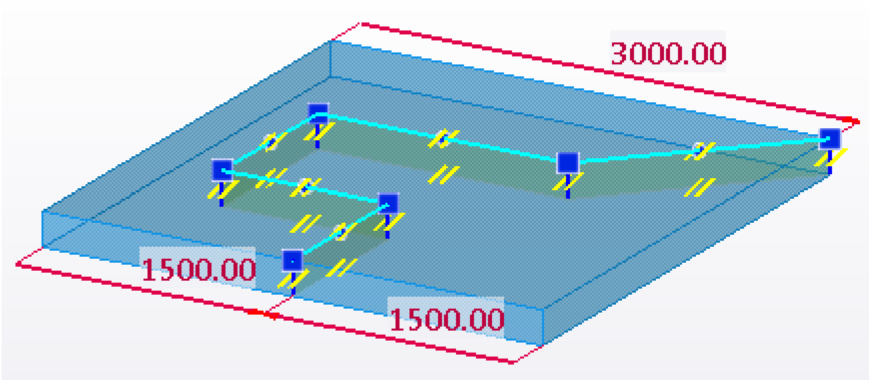
Modify a pour break

You can modify, copy, move, and remove existing pour breaks.

Before you start:

- Ensure that the  **Direct modification** switch is active.
- Ensure that the  **Select pour breaks** selection switch is active.
- Select the pour break.

  Tekla Structures displays the handles and dimensions that you can use to modify the pour break.

To modify a pour break:

To	Do this
Change the shape or location of the pour break	Drag a corner point or an end point to a new location.
Change a location dimension	Drag a dimension arrowhead to a new location, or: 1. Select the dimension arrowhead which you want to move. 2. Using the keyboard, enter the value with which you want the dimension to change.

To	Do this
	To start with the negative sign (-), use the numeric keypad. To enter an absolute value for the dimension, first enter $, then the value. 3. Press **Enter**, or click **OK** in the **Enter a Numeric Location** dialog box.
Add an intermediate point to the pour break	Drag a midpoint handle to a new location.
Remove an intermediate point from the pour break	1. Select an intermediate corner point. 2. Press **Delete**.
Modify the pour break properties	1. Double-click the pour break to open the property pane. 2. Modify the properties. 3. Click **Modify**.
Copy the pour break	Copy (page 127) the pour break like any other object in Tekla Structures. For example, right-click and select **Copy**.
Move the pour break	Move (page 140) the pour break like any other object in Tekla Structures. For example, right-click and select **Move**. You may need to move the pour break if you have moved the part, because the pour break does not follow the part.
Remove the pour break	Press **Delete**.

See also

Pour breaks (page 458)

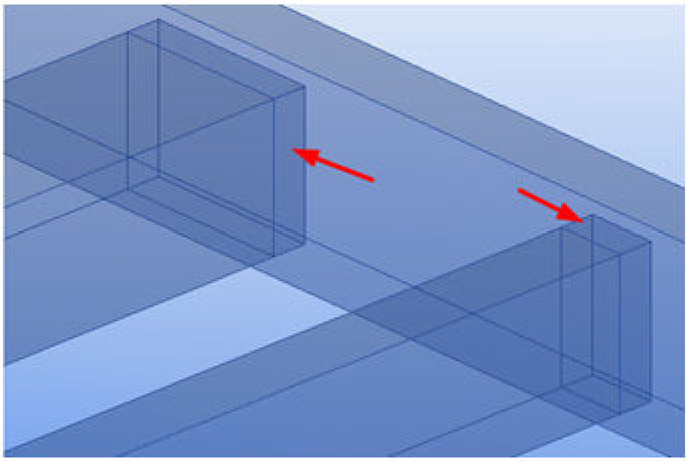
Resize and reshape model objects (page 107)

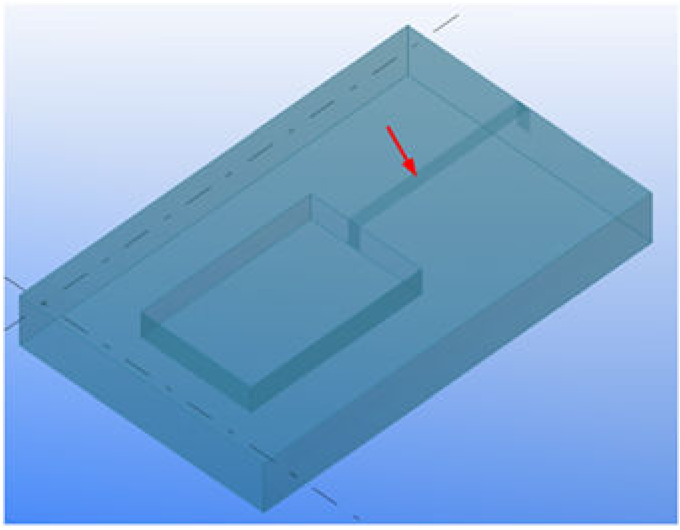
Troubleshoot pours

When you work with cast-in-place concrete parts, it is important that you regularly check the resulting pour objects, and try to get rid of the errors related to them, before you start detailing, or creating drawings and reports. The errors in solid pour objects may lead to inaccuracies in volume and other quantity calculations, and to incorrect representation and hatching in drawings.

While you model, use the following methods to check the model for pour-related errors:

- Check if there are `Solid error` rows in the session history log file (page 707).
- Ensure that the cast-in-place concrete parts and pour objects look continuous in model views. They should not have part outlines or shadow lines inside them, like in the following images:





If you notice errors or overlapping volumes or faces, try remodeling some of the parts.

You can also try out the following tips to avoid pour-related errors:

- Make sure that the number of parts included in a single pour object is reasonable.
- Sometimes modeling the parts in a different order may fix errors in pour objects.

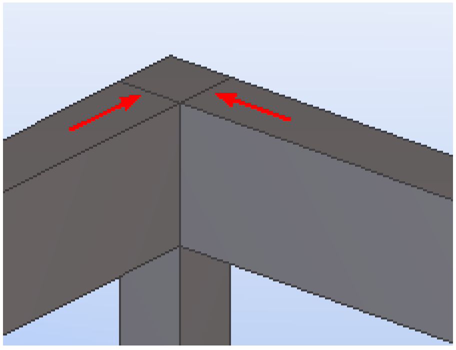
- To control which lines are visible in drawings, use the advanced options XS_DRAW_CAST_PHASE_INTERNAL_LINES and XS_DRAW_CAST_UNIT_INTERNAL_LINES.

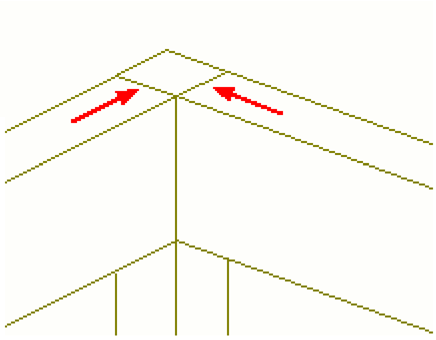
  This may help because the cast-in-place concrete parts that have errors are treated in the same way in drawings as the precast concrete parts.

If remodeling the parts does not fix errors in solid pour objects, then overlap the parts as little as possible to ensure that the volume and quantity calculations are close to the correct values.

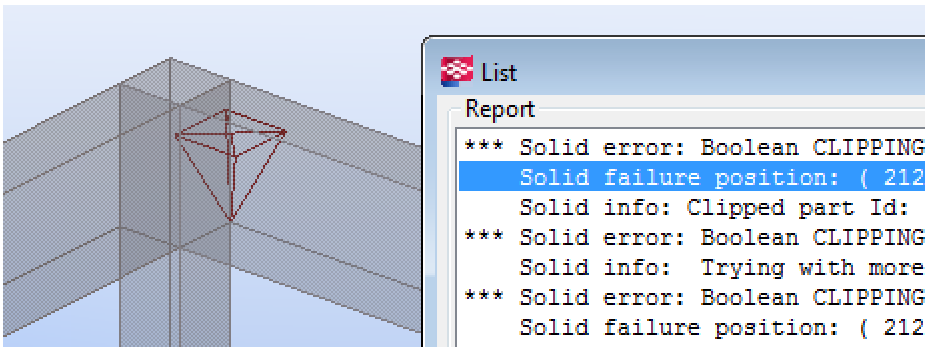
Example: Identify and fix a pour error

This is how an error related to a solid pour object can be indicated in a model view and in a drawing. The pour object is not shown as continuous, and there are extra lines between the parts in the pour object:

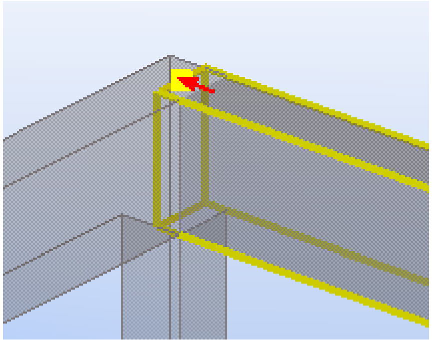




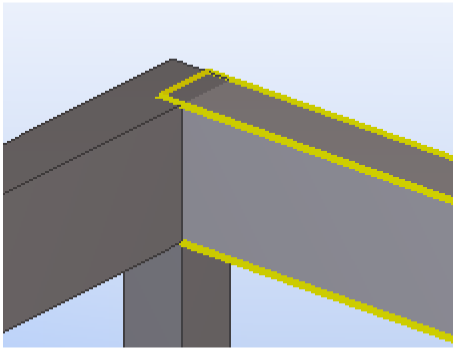
Checking the session history log file (page 707) and clicking a `Solid failure position` row helps you to locate the error in the model (press **Ctrl+2** to see through the parts):

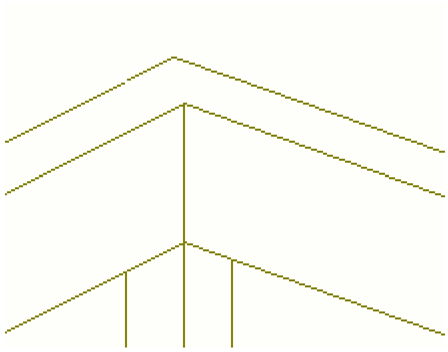


Try moving a beam end so that it is no longer on the same surface as the column side:



This is how the model and the drawing look like after you have fixed the model:





Also the volume of the pour object is now correct, for example, in reports. The overlapping volume of the beam and the column is only counted once.

See also

View solid errors (page 707)

Manage pours (page 444)

Example: Create concrete geometry and work with pours

The guidelines in this example help you to efficiently model cast-in-place concrete geometries, and to define, visualize, sequence, and report pours and pour breaks.

Before you start, ensure that you have pour management enabled. See Enable pour management (page 445).

1. If possible, use an existing engineering or architectural model or drawing as a basis when you create concrete structures in Tekla Structures.

   Import the existing model or drawing as a reference model to your Tekla Structures model.

   See Insert a reference model and Reference models and compatible formats.

2. If you are using an IFC model as a reference model:

a. Convert the concrete structures you need from the IFC model to native Tekla Structures objects.

   See Convert IFC objects into native Tekla Structures objects and Example: Convert IFC objects into Tekla Structures objects at one go.

b. Check the conversion results.

c. If needed, modify the converted objects.

   For example, you may need to change the profile, material, or cast unit type of the converted objects.

   TIP Use **Organizer** for checking and selecting objects.

3. If you are using a different reference model type, or if there are structures that cannot be converted from an IFC model, model the needed concrete structures as cast-in-place concrete parts in Tekla Structures.

   You can model by tracing over the reference model.

   See Create parts and modify part properties (page 196).

4. For each cast-in-place concrete part, define a pour phase number to divide your Tekla Structures model into pour objects.

   For example, use the default pour phase 0 for horizontal structures, like beams and slabs, and the default pour phase 1 for vertical structures, like columns and walls, to separate them to different pour objects.

   See Define the pour phase of a part (page 450).

   TIP Use selection filters or **Organizer** to efficiently select multiple parts and to modify them all at the same time.

5. View and check the pour objects in a pour view.

   See View cast-in-place concrete structures (page 447) and Pour objects (page 451).

6. If needed, modify the pour phases or create pour breaks to fine-tune the pour objects.

   For example, create pour breaks to split large slabs into smaller pour objects.

   See Create a pour break (page 461) and Pour breaks (page 458).

7. Once you are ready with the concrete geometry and pour objects, you can define pour sequences by entering pour numbers for pour objects, or by using the **Organizer** categories.

   See Modify the properties of a pour object (page 453) and Categories in Organizer.

8. Calculate the pour units, and modify them by adding and removing objects if needed.

   See Pour units (page 453).

9. You can also define other properties for pour objects and pour units, for example, concrete mixtures, or dates or status of workflow.

   See Modify the properties of a pour unit (page 457) and Categories in Organizer.

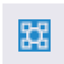
10. Use **Organizer** to categorize pours. Then you can select them by their sequence and report pour-specific information, such as pour volumes and formwork areas.

    See View object properties in Organizer and Example: Organize the model into location and custom categories, and view quantities.

11. If you wish, use **Task manager** to include pour objects and pour units in tasks and to schedule pours. You can then visualize pour status information based on planned and actual dates by using **Project status visualization**.

    See Create a task in Task manager and Project status visualization.

12. Create general arrangement drawings for pour units.

    Select a pour unit using the  **Select assemblies** switch, create a 3D view of the pour unit, and then create a GA drawing using the 3D view.

    This way you can automatically include in the drawing all reinforcement, embeds, and other objects that need to be shown with the pour object.

    See Pours in drawings.

2.9 Create reinforcement

Once you have created a model of concrete parts, you will need to reinforce the parts to gain higher strength for the parts.

In Tekla Structures, you can use different methods to create reinforcement. In many cases, you may need to use a combination of several reinforcement tools to get the desired results.

The most automated method is to use the various reinforcement components Tekla Structures contains. We recommend that you use reinforcement components to create reinforcement whenever possible. The components are adaptive, attached to a concrete part, and updated automatically if the dimensions of the reinforced part change.

Rebar sets are another flexible and versatile method to create reinforcement. Also the rebar sets are adaptive to concrete geometry, and easy to modify using direct modification.

In addition to these methods, you can manually create:

- single reinforcing bars (page 496)
- reinforcing bar groups (page 496)

  For more automated reinforcing bar group creation you can use Rebar shape catalog (page 498), which contains predefined reinforcement shapes.
- reinforcement meshes
- prestressed strands (page 517)
- reinforcement splices (page 519)

When needed, you can also combine several reinforcement objects into rebar assemblies (page 614), for example, to create reinforcement cages.

Create a rebar set

Rebar sets are reinforcing bars that you can modify by using direct modification and the rebar set guidelines, leg faces, and local modifiers. You can create rebar sets when you want to flexibly reinforce various areas in concrete parts or pour objects.

You have several options for creating rebar sets. Longitudinal and crossing rebar sets, and rebar sets created according to concrete part or pour object faces, are attached and adaptive to a concrete part or pour object. Using the **Point input** command you can create rebar sets even outside concrete objects. You can also use Rebar shape placing tool to create rebar sets.

NOTE When you work with rebar sets, ensure that the **Direct modification** switch is active.

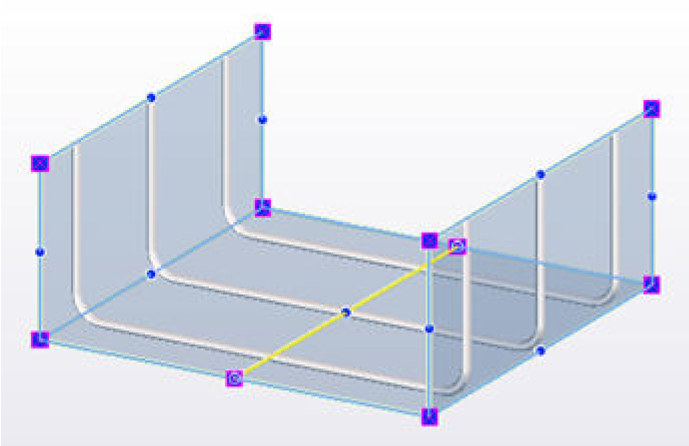
Basic concepts related to rebar sets

The leg faces (page 525) of a rebar set are planes that define where the reinforcing bar legs are created. Tekla Structures creates leg faces at the reinforced faces of concrete parts or pour objects, or according to the points you pick when you create rebar sets.

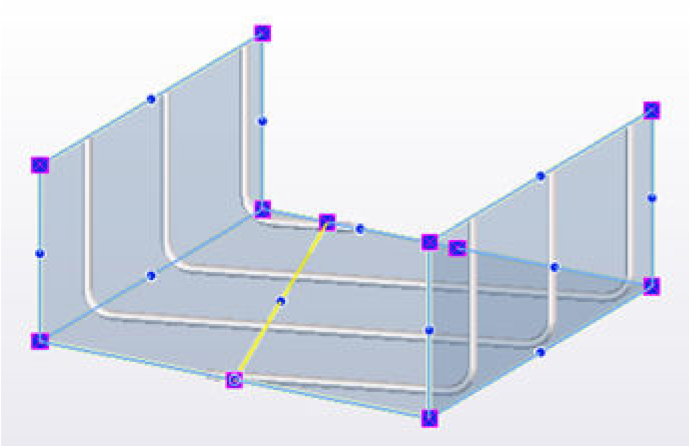
The rebar set *leg surfaces* are similar to leg faces, but they can be curved. Tekla Structures creates leg surfaces when you create rebar sets using the **By guidelines** command.

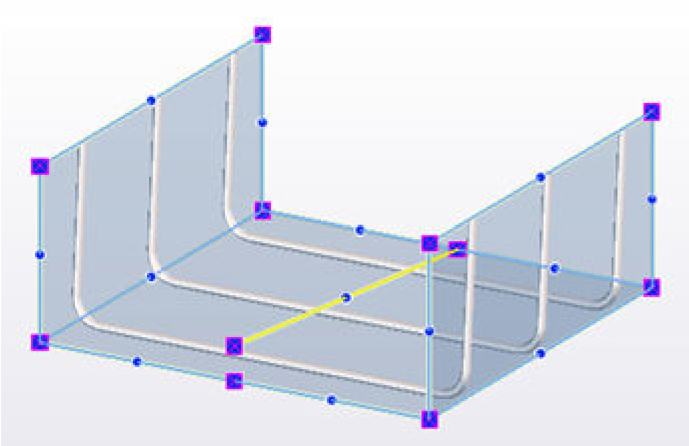
Each rebar set has at least one *guideline* that defines the distribution direction of the bars. The spacing of the bars is also measured along the guideline. The guideline can be a line, or a polyline that may have corner chamfers.

In the example below, the leg faces are shown in gray, and the guideline is highlighted in yellow:



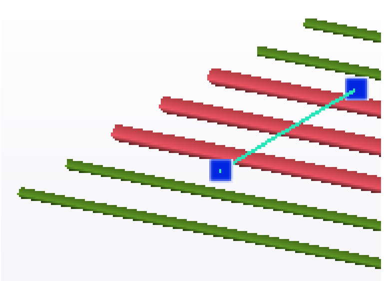
The position of the guideline affects the bar creation. If you move or lift an end of the guideline, the bars turn respectively. For example:

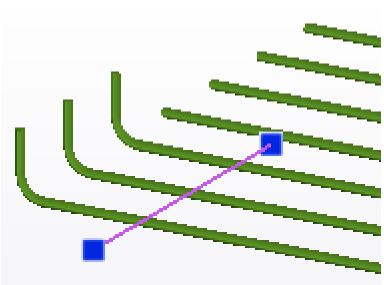


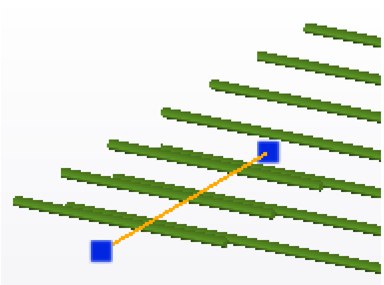


If needed, you can create up to two *secondary guidelines*, and use them to define different spacings along the rebar set. You can also use secondary guidelines when you create longitudinal bars for curved structures (page 489). Tekla Structures automatically creates three guidelines for longitudinal rebar sets in curved beams, polybeams, strip footings, and wall panels. The rebar sets that have leg surfaces can only have one secondary guideline.

If you need to modify a rebar set only at certain locations, you can create local *property modifiers*, *end detail modifiers*, and *splitters*.





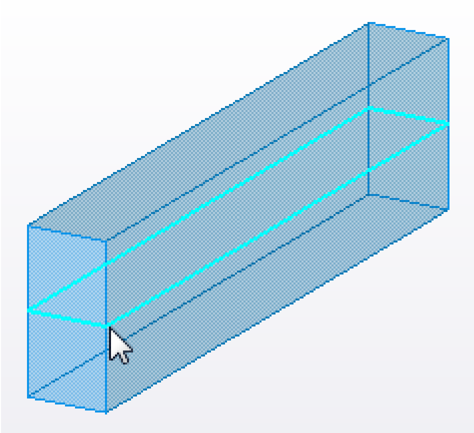


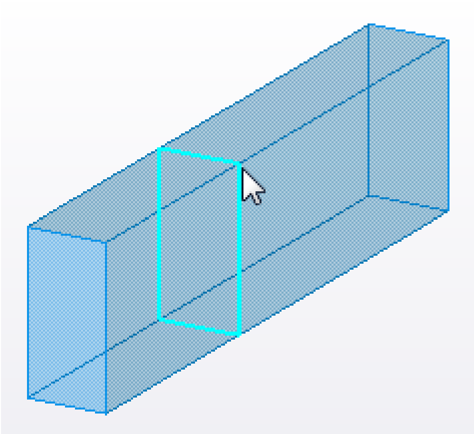
Create longitudinal rebars

You can create rebar sets that consist of longitudinal reinforcing bars in a concrete part or pour object.

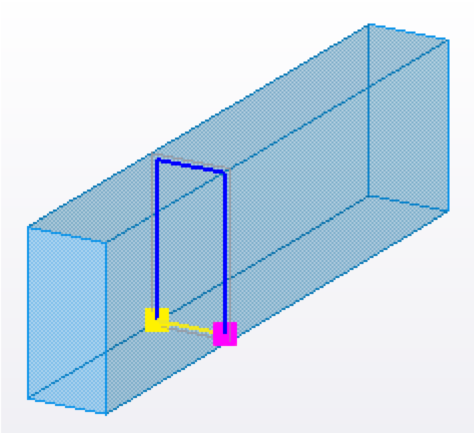
1. Depending on the concrete object you want to reinforce, use a part view or a pour view (page 447).
2. On the **Rebar** tab, click **Longitudinal**.
3. Move the mouse pointer over the edges of a concrete part or pour object.

Tekla Structures highlights the cross sections that you can select.





4. Select the cross section that you want to reinforce.



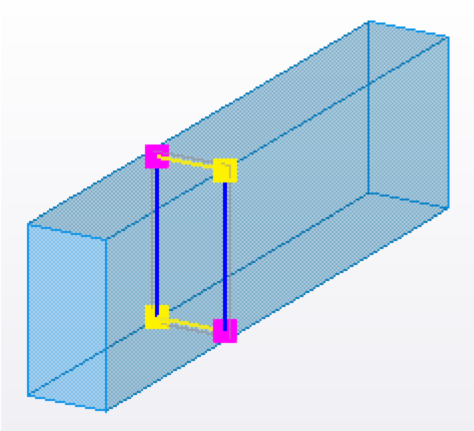
5. If needed, modify the cross section size or shape for the bars.

   To do this, click on the contextual toolbar, and then drag the cross section handles.

6. In the selected cross section, select the faces that you want to reinforce.

By default, only one of the faces is selected. To select multiple faces, hold down **Shift** or **Ctrl**.

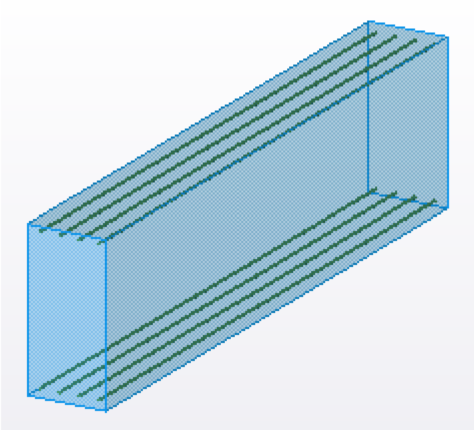
Tekla Structures highlights the selected faces in yellow.



7. To extend or shorten individual faces, click on the contextual toolbar. Then drag the yellow and magenta end handles.

8. To finish, click the middle mouse button, or click **Create the rebar set** on the contextual toolbar.

Tekla Structures creates a rebar set at each selected face, and the bars perpendicular to the selected cross section.



Create crossing rebars

You can create a rebar set that consists of crossing reinforcing bars in a concrete part or pour object.

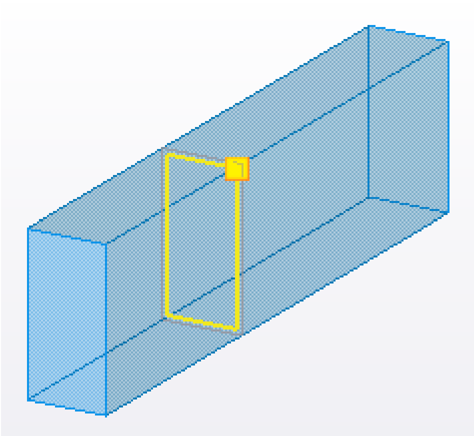
1. Depending on the concrete object you want to reinforce, use a part view or a pour view (page 447).

2. On the **Rebar** tab, click **Crossing**.

3. Move the mouse pointer over the edges of a concrete part or pour object.

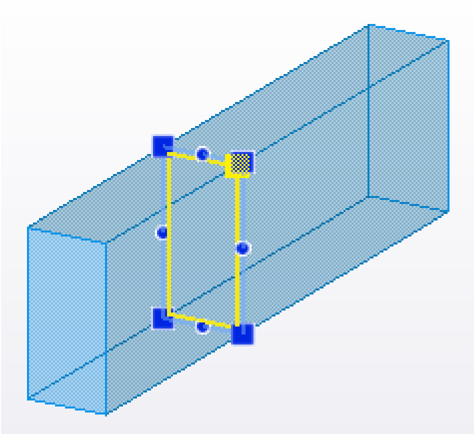
   Tekla Structures highlights the cross sections that you can select.

4. Select the cross section that you want to reinforce.

5. If needed, modify the shape of the bars.

   To change the cross section size for the bars, click on the contextual toolbar and then drag the cross section handles.

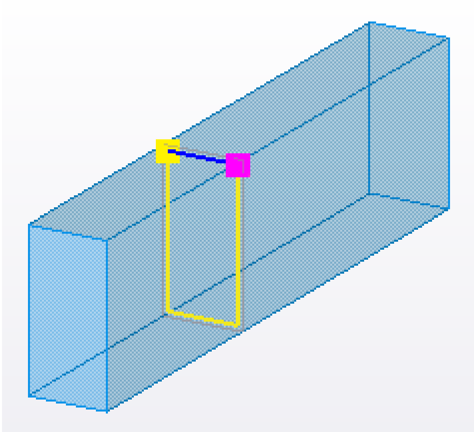
   

6. With active on the contextual toolbar, in the selected cross section, select the bar legs that you want to create.

   By default, all legs are selected and Tekla Structures creates a leg for each object face.

   - To unselect a selected leg, hold down **Ctrl** and click the leg.
   - To unselect all legs, click on the contextual toolbar.
   - To select more than one leg, select the first leg, and then hold down **Ctrl** or **Shift** when selecting the rest of the legs.
   - To select all legs, click on the contextual toolbar.

Tekla Structures highlights the selected legs in yellow, and creates a continuous bar shape of the legs.



7. If you want to rotate the bar shape, for example, to move stirrup hooks to another corner, press **Tab** for counterclockwise or **Shift+Tab** for clockwise direction.

8. If you need to extend or shorten individual legs, click on the contextual toolbar and then drag the bar end handles.

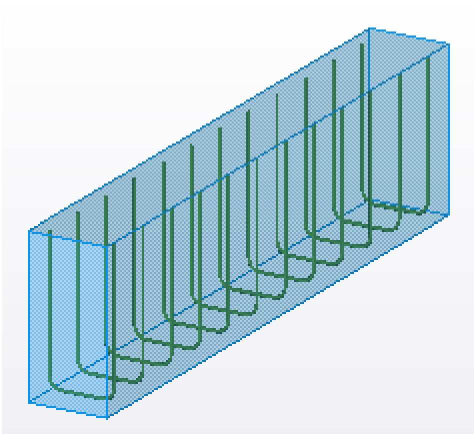
   This way you can also create overlapping bar shapes, or extend bar ends outside the concrete object.

9. If you want to modify the distribution length of the bars, adjust the length of the rebar set guideline.

   Click on the contextual toolbar and then drag the guideline end handles .

10. To finish, click the middle mouse button, or click **Create the rebar set** on the contextual toolbar.

Tekla Structures creates the bars parallel to the selected cross section, and distributes the bars along the length of the guideline.



Create rebars by face

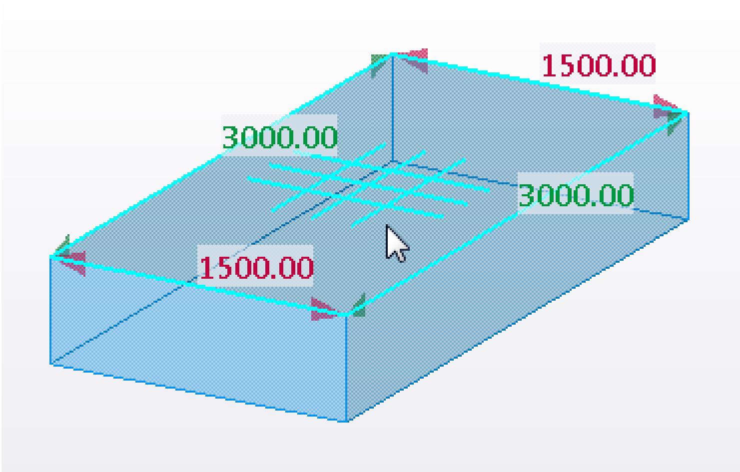
You can create rebar sets according to a face of a concrete part or pour object.

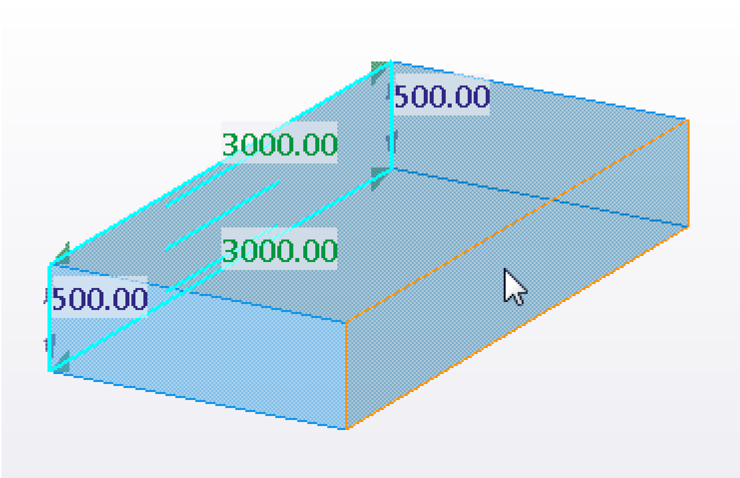
1. Depending on the concrete object you want to reinforce, use a part view or a pour view (page 447).
2. On the **Rebar** tab, click **By face**.
3. Define the faces and areas of the concrete object you want to reinforce, and the direction of the bars using the following options on the contextual toolbar:

Click this button	To do this
	Create bars at the near face of the concrete object.
	Create bars at the far face of the concrete object.
	Create bars parallel to the longest object face edge. With the option, bars are created so that they are parallel to the edge that is closest to the mouse pointer.
	Create bars perpendicular to the longest object face edge. With the option, bars are created so that they are perpendicular to the edge that is closest to the mouse pointer.
	Create bars in two directions: one set of bars parallel to the longest object face edge, and the other set of bars perpendicular to the edge. With the option, one set of bars is parallel and the other set of bars perpendicular to the edge that is closest to the mouse pointer.
	Create bars for an entire object face.
	Create bars for a rectangular area on an object face.
	Create bars for a polygonal area on an object face.

4. Depending on the area you selected to reinforce, do one of the following:
   - To reinforce an entire object face:
     a. Move the mouse pointer over the faces of a concrete part or pour object.

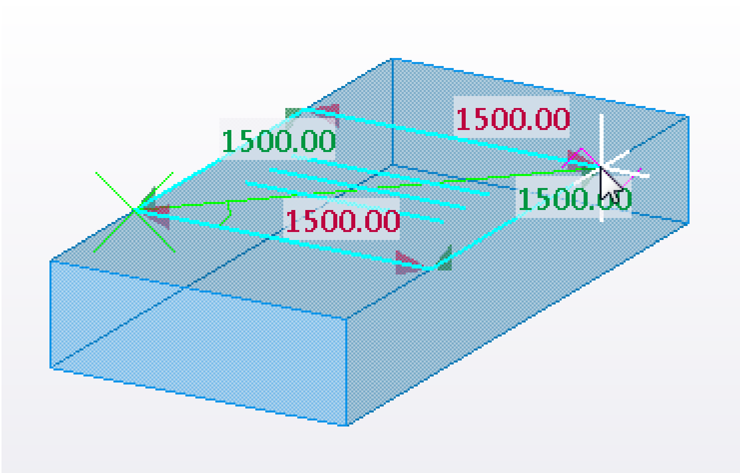
Tekla Structures shows the object face dimensions, and a symbol that indicates the direction of the bars.



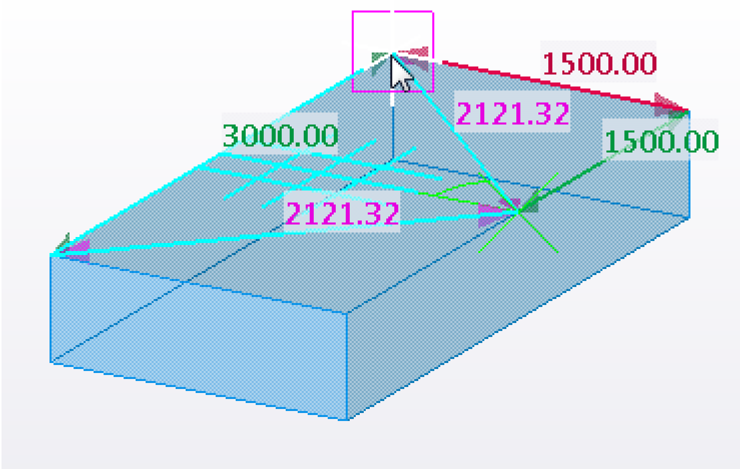


b. Select the object face.

- To reinforce a rectangular area, pick two opposite corners of the area.

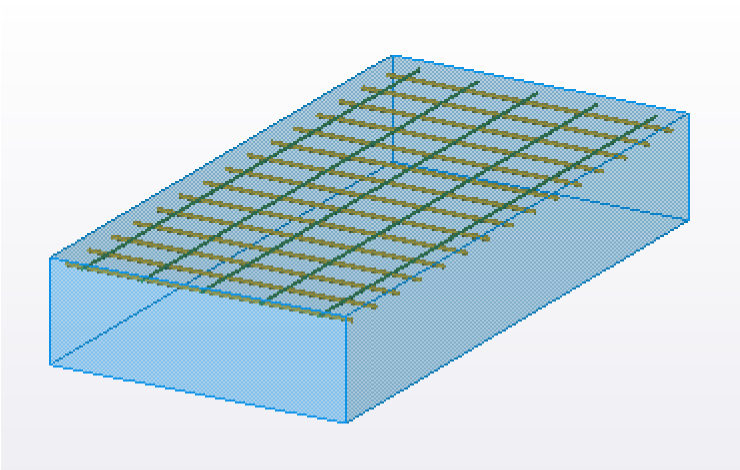


- To reinforce a polygonal area, pick the polygon corners.



Then click the middle mouse button.

Tekla Structures creates the bars according to the options you selected. If you selected to create bars in two directions, Tekla Structures creates two rebar sets: one with bars parallel to the longest object face edge, and one with bars perpendicular to it.

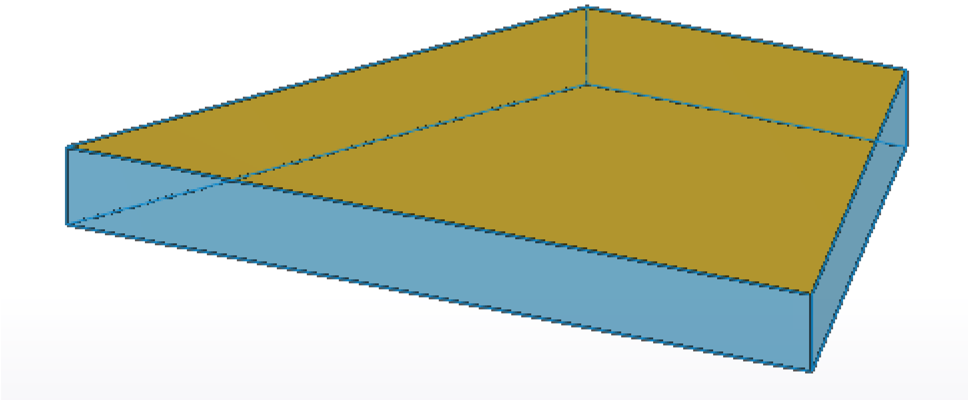


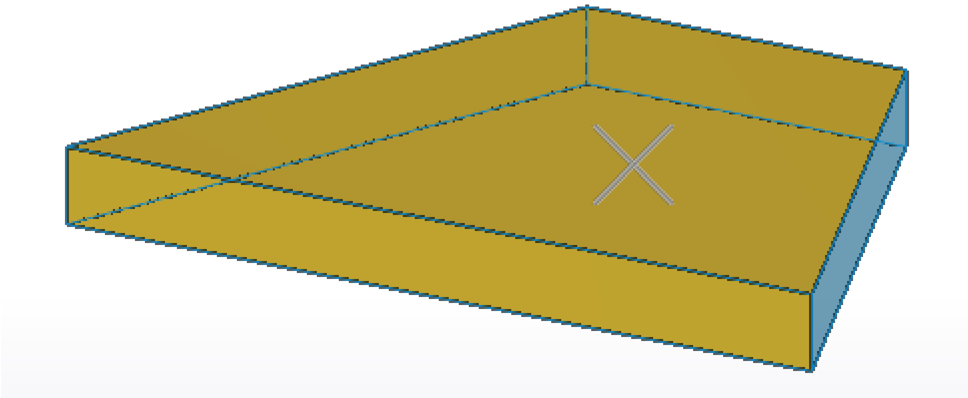
Create rebars by guidelines

You can create a rebar set at one or more faces of a concrete part, according to one or two guidelines you define by picking points.

1. On the **Rebar** tab, click **By guidelines**.
2. Select the part faces that you want to reinforce.

Tekla Structures highlights the selected faces in yellow.

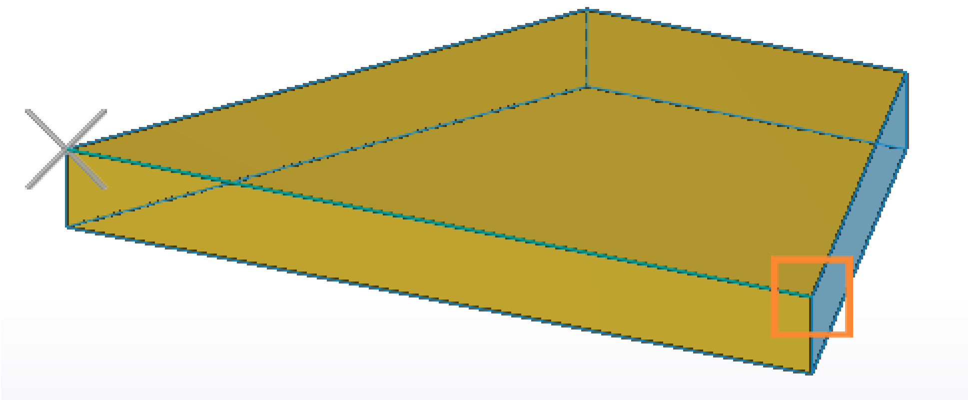


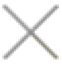
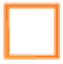


You can select as many faces as you like.

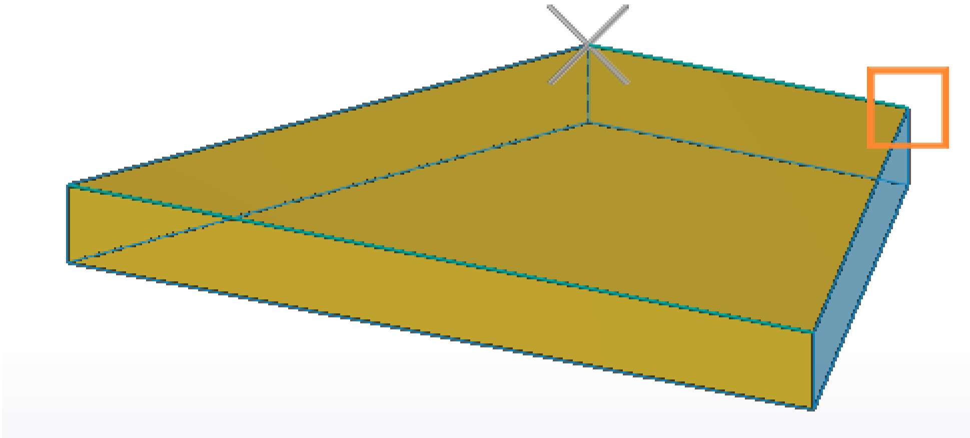
If you need to deselect a selected face, click the face again.

3. Click the middle mouse button to finish selecting faces.
4. Pick points to define the location for the primary guideline, and then click the middle mouse button.



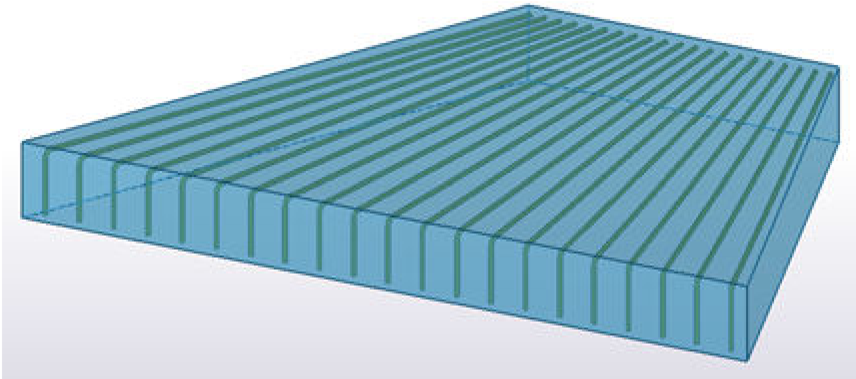
 = start point,  = end point

5. If you want to create a secondary guideline, pick points to define its location.



6. Click the middle mouse button to create the rebar set and the guidelines.

Tekla Structures creates a rebar set that extends over the faces you selected, and distributes the bars along the guidelines.



Create rebars by point input

You can create a set of reinforcing bars so that you define the shape of the bars by picking points in the model.

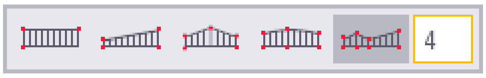
1. On the **Rebar** tab, click **More** --> **Point input**.
2. On the contextual toolbar, select an option to define the rebar set type and the number of cross sections in the rebar set.

   The options are:

   - **Normal**
   - **Tapered**

- **Tapered ridge**
- **Tapered curved**
- **Tapered N**

If you select the **Tapered N** option, enter the number of cross sections.

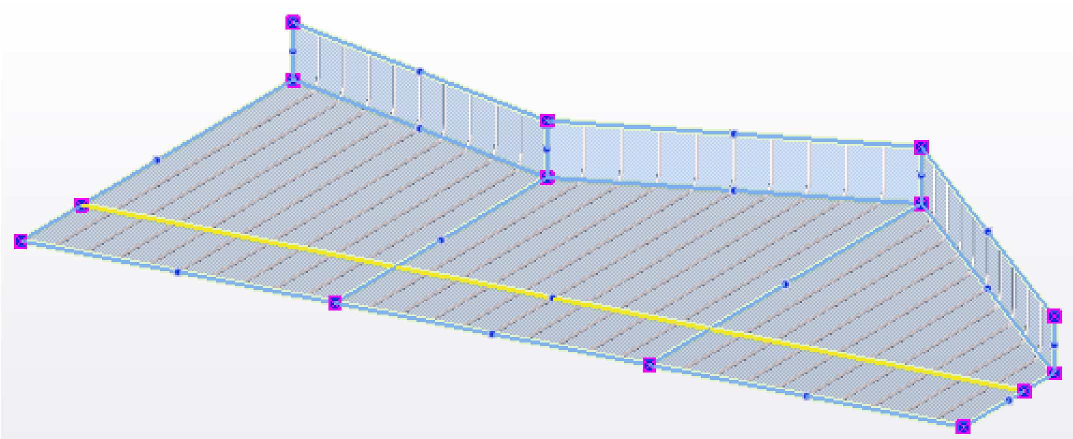


3. Pick points to define the shape of the bar at the first cross section.

   You can use different snapping (page 83) methods, such as **Ortho** and temporary reference points.
4. Click the middle mouse button to finish picking.
5. Depending on the rebar set type and the number of cross sections, do one of the following:

   - For a  **Normal** rebar set, pick the start point and the end point of the rebar set.
   - At the second and subsequent cross sections for other types of rebar set, pick points to define the bar shape. Then click the middle mouse button to finish picking at each cross section.

Tekla Structures creates a rebar set with leg faces between each cross section.



Rebar set properties

Use the contextual toolbar or the property pane to view and modify the properties of rebar sets. The file name extension of the property file is `.rst`.

See also Rebar set properties (page 1016) and Modify a rebar set (page 521).

Limitations

- Roundings in bent bar corners are not taken into account in automatic clash avoidance when Tekla Structures creates rebar sets and arranges them to layers.
- You cannot create rebar sets in deformed parts.

Create a rebar set using Rebar shape placing tool

You can create a rebar set by selecting a predefined bar shape from **Rebar shape placing tool**. The predefined shapes in **Rebar shape placing tool** are based on the shapes that have been defined in **Rebar shape manager** and saved in the `RebarShapeRules.xml` file.

Use **Rebar shape placing tool** to reinforce parts and pour objects. The rebar sets may extend across single or multiple objects.

Rebar shape placing tool does not work with round, spiral, or 3D bar shapes, or in tapered variable cross sections.

Create rebar sets

1. On the **Rebar** tab, click **More** --> **Rebar placing**.

   The **Rebar shape placing tool** dialog box opens.

2. If you want to create bars that extend across several parts or pour objects, for example dowel bars, select **Multiple objects** from the list at the bottom of the dialog box.

3. If you want to create several rebar sets in the same cross section, select the **Keep cross section** check box.

4. Select one of the predefined bar shapes from the tree view on the left.

   If the shape that you need is not available, or if you want to remove the shapes that you do not need, you can reorganize the tree view (page 488).

5. Define the bar dimensions.

   The dimensions you can define vary depending on the selected bar shape.

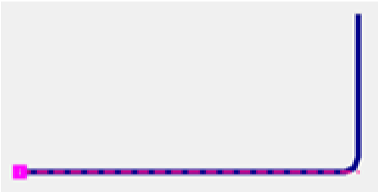
   The hook properties are visible only if you have set the advanced option XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION to `FALSE` in **File menu** --> **Settings** --> **Advanced Options** --> **Concrete Detailing**.

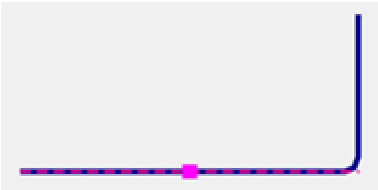
   - To set a **Length of leg** value, click a leg in the preview of the shape.

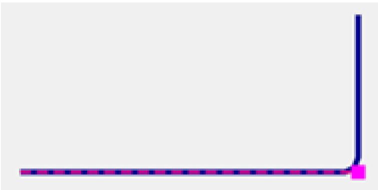
     If you do not enter a **Length of leg** value, the leg length is calculated automatically according to the dimensions of the concrete structure.

   - To set a **Bending angle** value for a bend that is not 90 degrees, click one of the legs next to the bend.

6. Set the rebar set reference point to start, middle, or end by double-clicking the different legs or hooks in the preview of the shape.







When you are placing the rebar set in the model, you can move the rebar set preview to a new location by dragging the reference point.

7. Modify the other bar properties as needed.

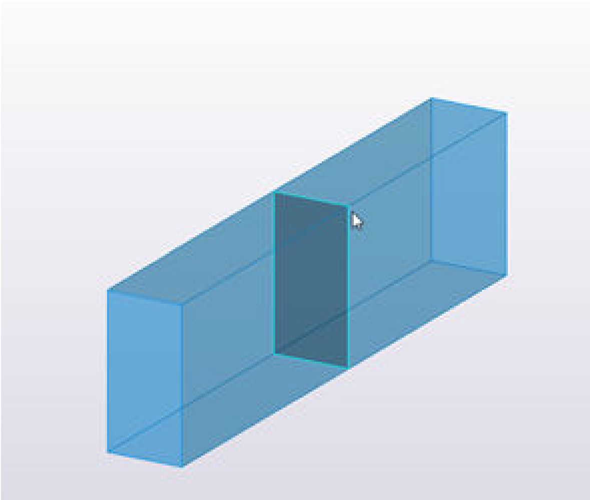
   For example, you can use **Layer order number** to arrange bars to layers when two or more rebar sets overlap.

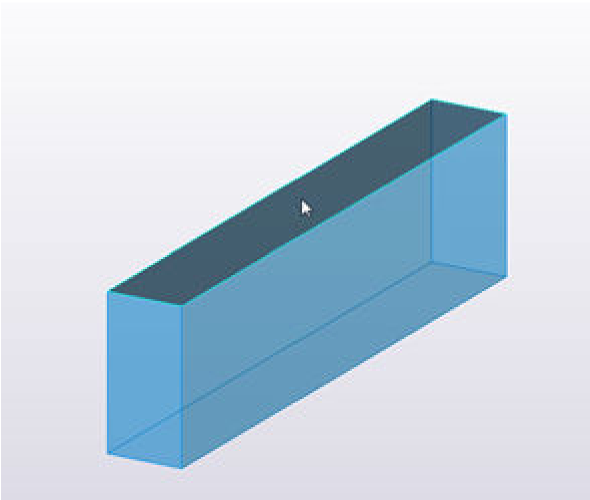
8. On the **Spacing** tab, define the spacing properties of the rebar set.

9. To place the rebar set in the model, move the mouse pointer over the edges and faces of a concrete structure.

   Depending on the concrete structure you want to reinforce, use a part view or a pour view (page 447).

   Tekla Structures highlights the cross sections and faces that you can select. For example:

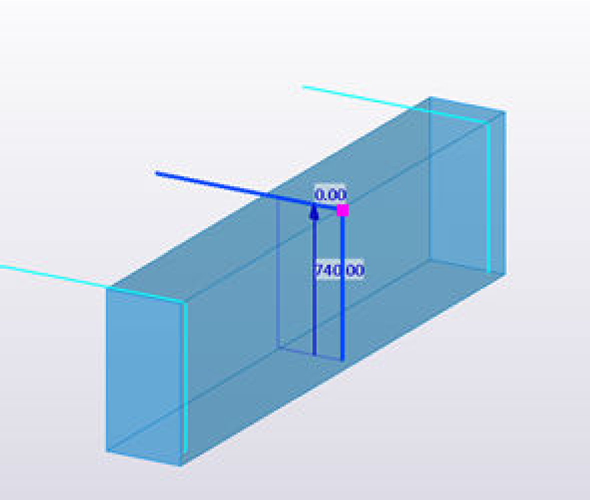




10. Select the cross section or face that you want to reinforce.

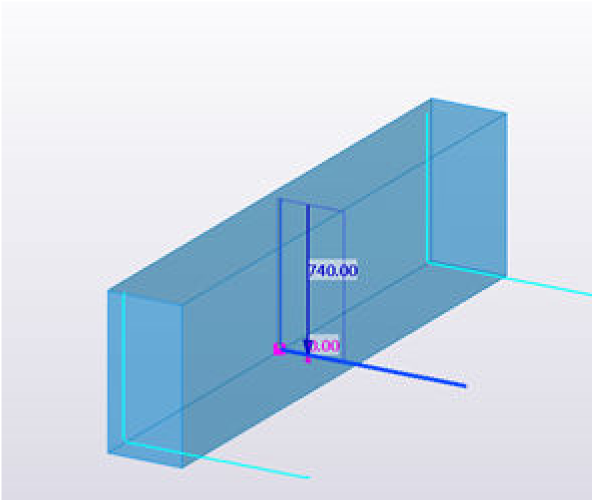
    If you have **Multiple objects** selected, click each cross section or face to select them. Click the middle mouse button to finish selecting.

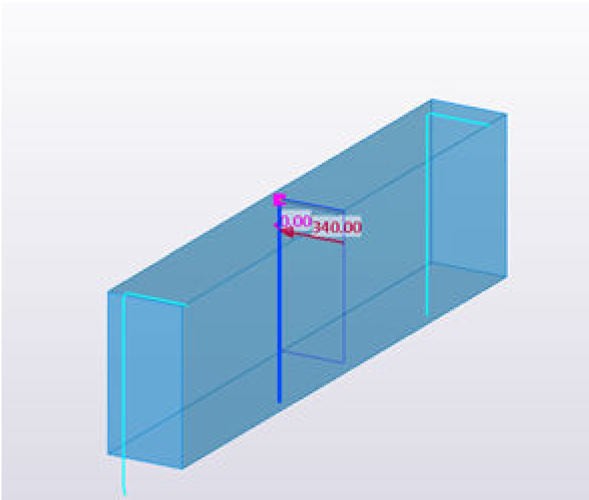
    Tekla Structures shows a preview of the bar shape in the model, and the first and last bars in the rebar set in cyan.



11. To move the rebar set to the desired location in the selected cross section or face, do any of the following:

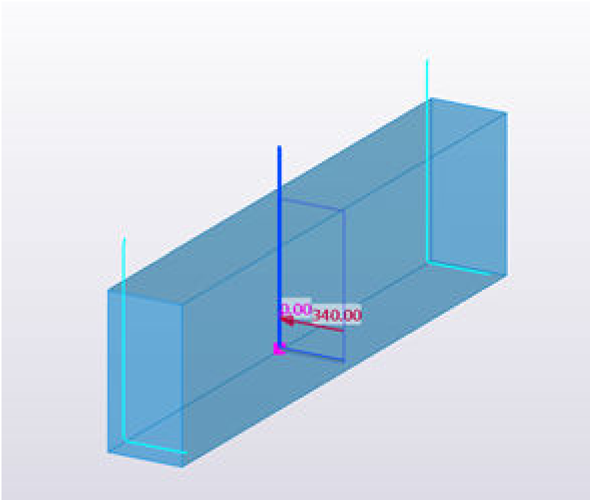
    - Click a blue line segment to place the rebar set reference point on that line segment. For example:



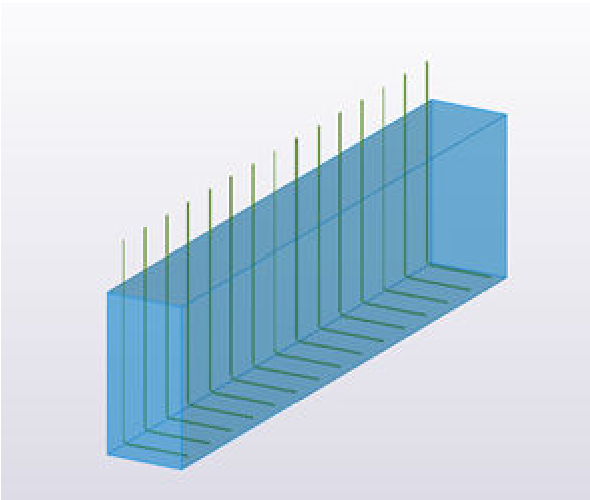


If needed, you can still change the reference point location in the **Rebar shape placing tool** dialog box by double-clicking the desired location in the preview.

- Drag the magenta reference point handle to a new location on the blue line.
- To turn the bar shape, click on the contextual toolbar.



12. Click the middle mouse button to create the rebar set.



13. If you have the **Keep cross section** check box selected, repeat steps 4–12 to create more rebar sets in the same cross section.

TIP If the **Rebar shape placing tool** dialog box is already open but the command is not active, click the **Select cross section** button to start creating rebar sets again.

Add and remove rebar shapes

You can modify the tree view in **Rebar shape placing tool** by adding frequently used bar shapes to the tree, or removing the bar shapes that you do not need.

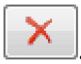
1. On the **Rebar** tab, click **More** --> **Rebar placing**.

   The **Rebar shape placing tool** dialog box opens.

2. Click **Organize catalog**.

3. To create a new category folder, click .
4. Drag and drop the selected shapes to the folder.

   If multiple shapes have the same shape code and you drag them to the categories, the shape codes get a suffix **(1)**, **(2)**, and so on. You can rename the shapes by clicking the name twice and entering a new name or suffix, for example, **(a)**, **(b)**.

   When the shapes are listed in a report, they all get the same shape code.
5. Change the name of the folder in a similar way, if needed.
6. To remove a shape from a category, select the shape and click .
7. Click **OK**.

Examples: Rebar sets in curved structures

You can reinforce curved concrete structures using rebar sets.

The curved concrete structures can include curved beams (page 263), polybeams (page 266) that have **Arc point** chamfers, and flat spiral beams (page 270) whose total rise is zero. You can also reinforce strip footings and wall panels the same way as you reinforce beams and polybeams.

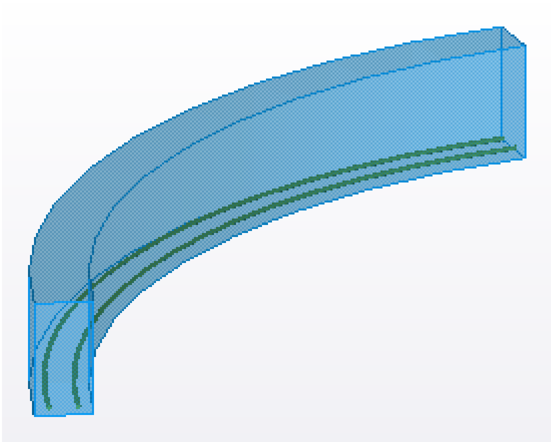
For more information about creating rebar sets, see also Create a rebar set (page 470).

Create longitudinal bars for a curved beam

In this example, we will create longitudinal bottom bars for a curved concrete beam.

1. Create a curved concrete beam.
   a. On the **Concrete** tab, click **Beam**.
   b. Pick two points.
   c. Double-click the beam to modify its properties.
   d. Set the radius and number of segments, and then click **Modify**.
2. Create longitudinal bars at the bottom face of the beam.
   a. On the **Rebar** tab, click **Longitudinal**.
   b. Move the mouse pointer over the edges of the beam, and select the cross section that you want to reinforce.
   c. Click the middle mouse button to create the rebar set.

Tekla Structures creates the curved longitudinal bars according to the beam geometry. For example:



The rebar set has three guidelines: one at each end of the beam and one in the midpoint of the beam.

Create crossing bars for a curved polybeam

In this example, we will create crossing U-shaped stirrups for a curved concrete polybeam.

1. Create a concrete polybeam with curved segments.

    a. On the **Concrete** tab, click **Beam** --> **Polybeam**.

    b. Pick at least three points you want the beam to go through, and then click the middle mouse button.

    c. Select the polybeam.

    d. Select the handle in a polybeam corner, and then select the **Arc point** chamfer type on the contextual toolbar.

2. Create crossing bars that follow the bottom and side faces of the beam.

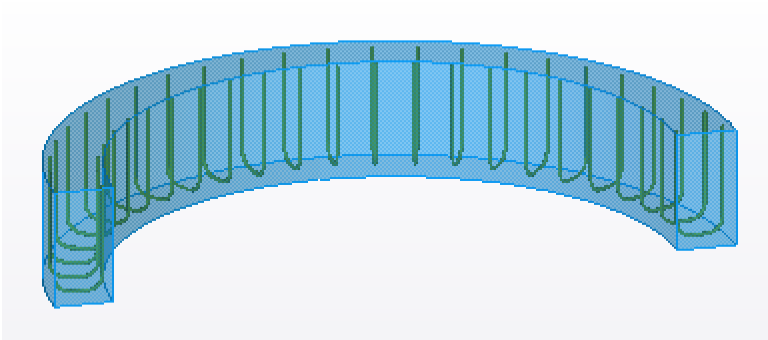
    a. On the **Rebar** tab, click **Crossing**.

    b. Move the mouse pointer over the edges of the beam, and select the cross section that you want to reinforce.

    c. In the selected cross section, hold down **Ctrl** and click the top bar leg to unselect it.

    d. Click the middle mouse button to create the rebar set.

Tekla Structures creates the crossing bars radially according to the beam geometry. For example:



The rebar set guideline is a polyline with three points, and the midpoint has the **Arc point** chamfer.

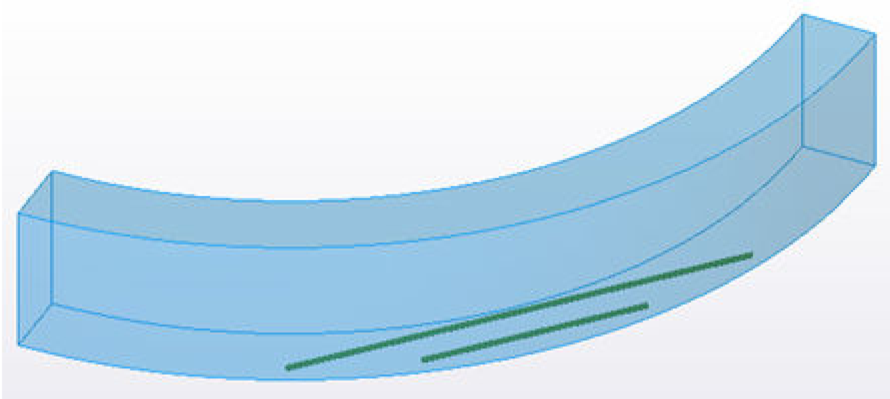
Create reinforcing bars for a spiral beam

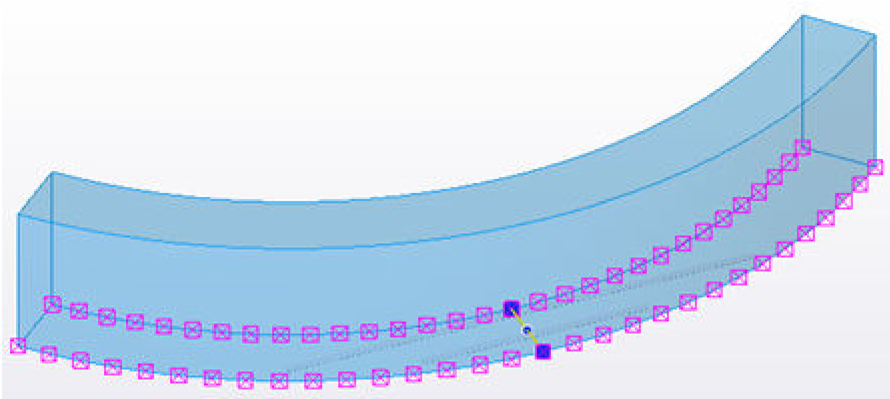
In this example, we will reinforce a spiral beam whose total rise is zero. We will create longitudinal bottom bars and crossing U-shaped stirrups.

You can use the following manual method also for more complex concrete objects that have been imported and that cannot be reinforced automatically.

1. Create a flat spiral concrete beam.
    a. On the **Concrete** tab, click **Beam** --> **Spiral beam**.
    b. Pick the start point of the beam.
    c. Pick a point to indicate the beam's center of curvature.
    d. Click the middle mouse button.
    e. Ensure that **Total rise** is 0.
2. Create longitudinal bars at the bottom face of the beam.
    a. On the **Rebar** tab, click **Longitudinal**.
    b. Move the mouse pointer over the edges of the beam, and select the cross section that you want to reinforce.
    c. Click the middle mouse button to create the rebar set.

Tekla Structures creates a longitudinal rebar set with one guideline.





d. Press **Esc** to interrupt the command.

3. Modify the longitudinal rebar set by creating more guidelines.

   a. Select the rebar set.

   b. On the contextual toolbar, click **Add secondary guideline**.

   c. Ensure that you are in the single point picking mode (shown on the contextual toolbar).

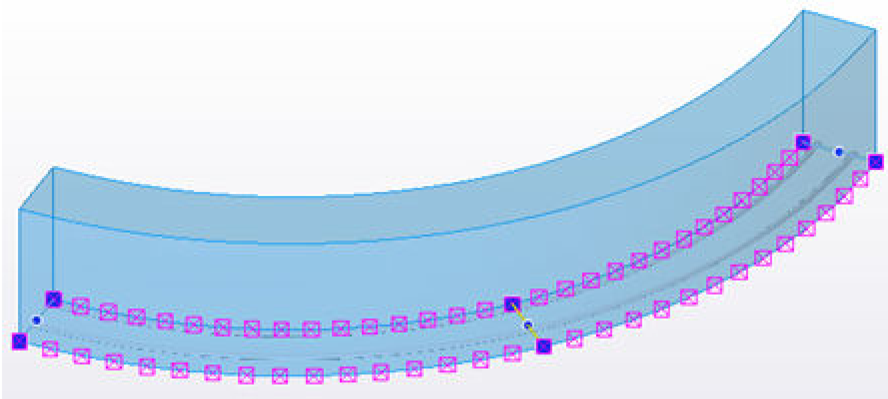
   d. Pick the start point for a secondary guideline.

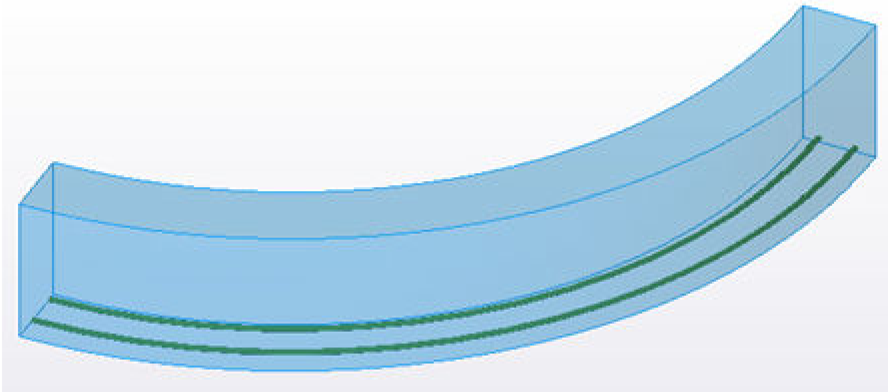
   e. Pick the start point for another secondary guideline.

   f. Press **Esc** to finish creating secondary guidelines.

   g. If needed, move the guidelines to desired locations by dragging them or their end point handles.

For example, you can move the primary guideline to the midpoint of the beam, one of the secondary guidelines to the start of the beam, and the other secondary guideline to the end of the beam.





h. Modify the geometry (page 523) and properties (page 1019) of the guidelines as needed.

   The bars are constructed according to the locations and spacing settings of these three guidelines.

4. Create crossing bars that follow the bottom and side faces of the beam.

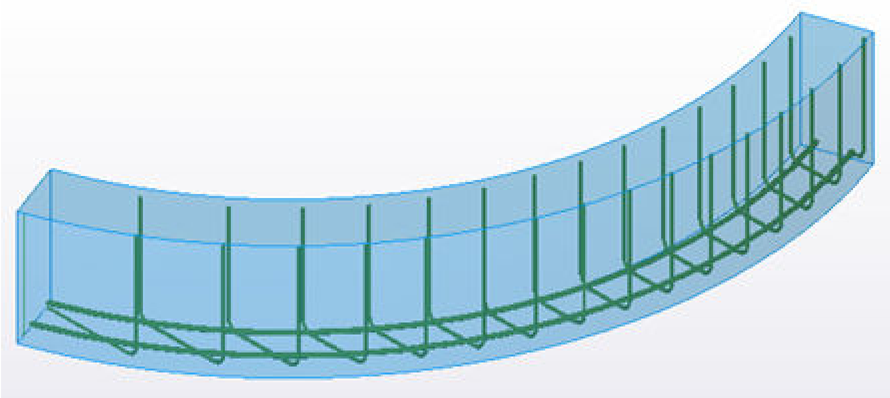
   a. On the **Rebar** tab, click **Crossing**.

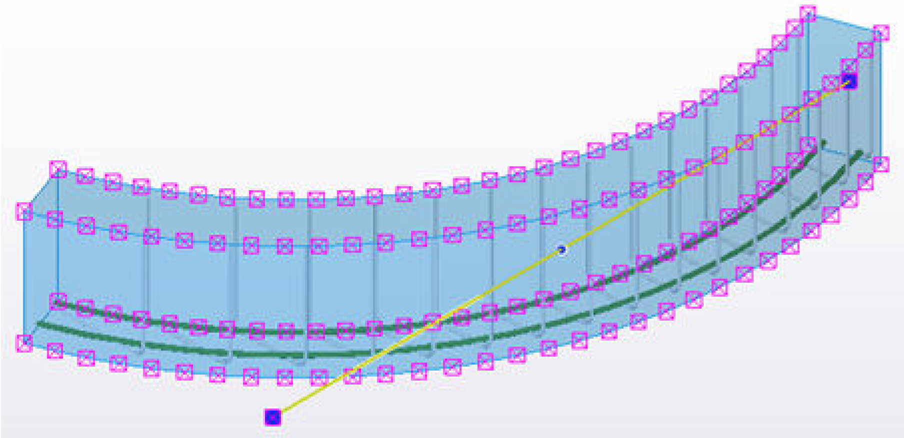
   b. Move the mouse pointer over the edges of the beam, and select the cross section that you want to reinforce.

   c. In the selected cross section, hold down **Ctrl** and click the top bar leg to unselect it.

   d. Click the middle mouse button to create the rebar set.

Tekla Structures creates a crossing rebar set with one guideline.



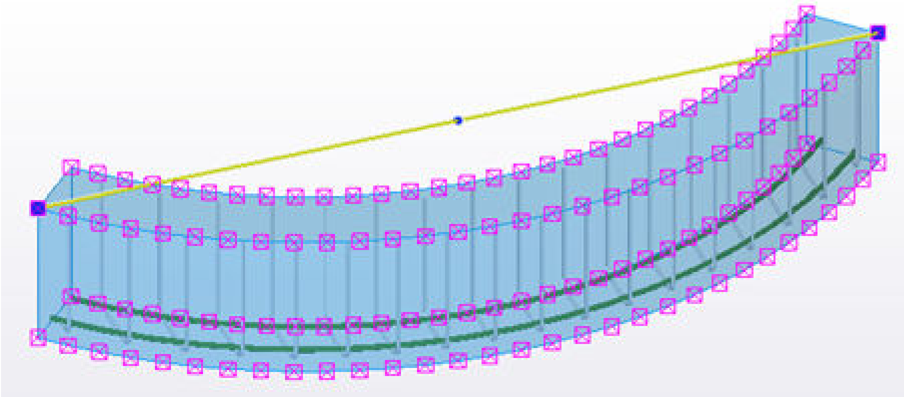


e. Press **Esc** to interrupt the command.

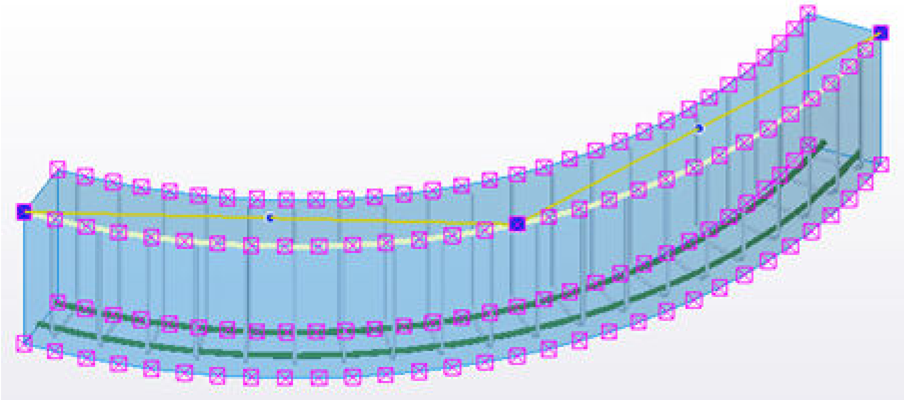
5. Modify the crossing rebar set by modifying the guideline.

   a. Select the rebar set to highlight the guideline.

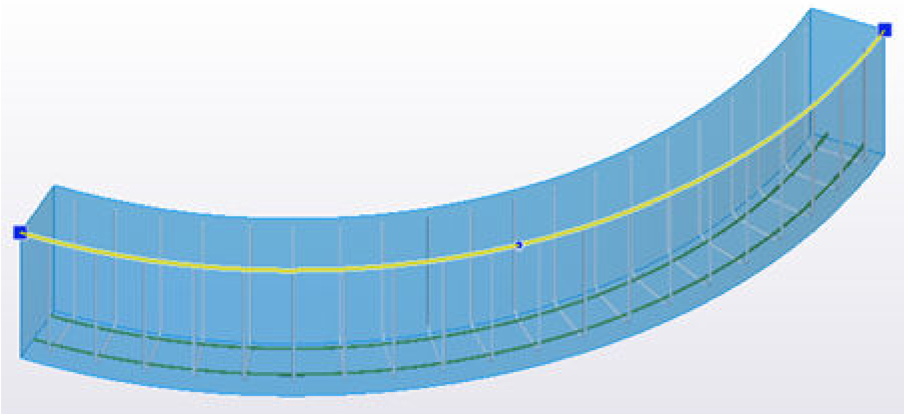
   b. Drag the guideline end points  to the beam ends.



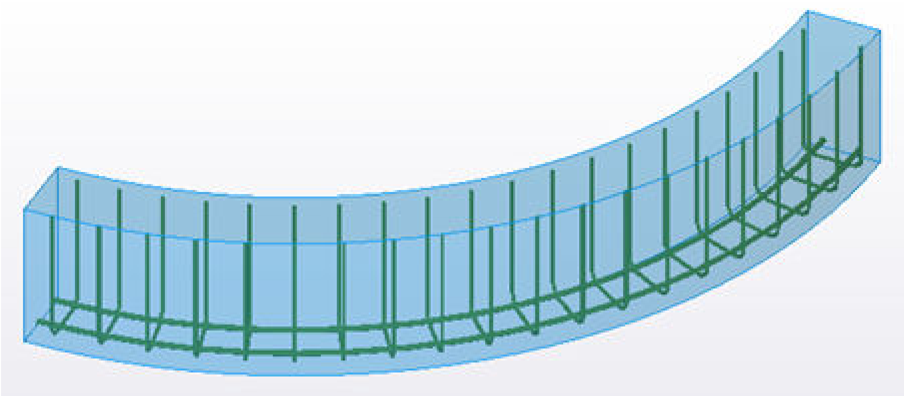
c. Drag the guideline midpoint to the beam midpoint. Ensure that the new guideline corner has the **Arc point** chamfer.



Alternatively, you can click on the contextual toolbar to make the guideline follow the leg face edges that are located between the guideline end points.



Tekla Structures arranges the crossing bars radially along the beam.



Limitations

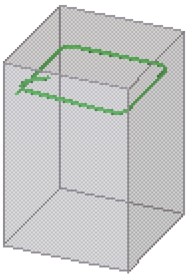
- If curved longitudinal bars have too small start and/or end offset values, the bars closest to leg face edges may be divided into small bar segments. To avoid this, increase the offset values.

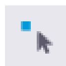
Create a single reinforcing bar

1. On the **Rebar** tab, click  **Single bar**.

   If you need to modify the properties before you create the reinforcement, hold down **Shift** and click the **Single bar** command to open the **Single rebar** properties.
2. Select the part to reinforce.
3. Pick the bar start point.
4. Pick the other bar reference points to set the bar shape.
5. Click the middle mouse button to finish picking.

   Tekla Structures attaches the bar to this part.

   
6. If you want to modify the reinforcement, do one of the following:

   - Use direct modification (page 544). Ensure that the  **Direct modification** switch is active.
   - Double-click the reinforcement to open the **Single rebar** properties and modify the properties (page 1009).

See also

Create a reinforcing bar group using Rebar shape catalog (page 498)

Create a reinforcing bar group (page 496)

Create a reinforcing bar group

A reinforcing bar group includes several identical, or very similar, reinforcing bars. Tekla Structures always treats these bars as a group, modifies them in the same way, deletes them all at the same time, and so on. You first define the shape of a single bar, then the direction in which Tekla Structures distributes the bars.

NOTE If you do not want to manually define the bar shape, use Rebar shape catalog (page 498) and its predefined reinforcement shapes instead.

---

1. On the **Rebar** tab, click .

   If you need to modify the properties before you create the reinforcement, hold down **Shift** and click the command button to open the **Rebar group** properties.

2. Select the part to reinforce.

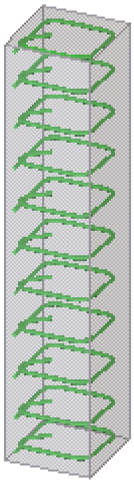
   Tekla Structures attaches the bar group to this part.

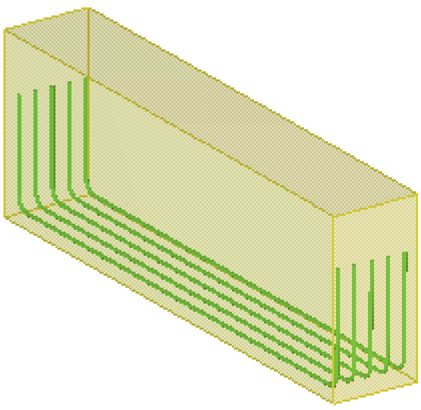
3. Pick the bar start point.
4. Pick the other bar reference points.

   These points define the plane of the first bar and the shape of a single bar in the group.

5. Click the middle mouse button to finish picking.
6. Pick the start point of the bar group.
7. Pick the end point of the bar group.

The start and end points indicate the distribution length and direction of the bars. Usually the distribution length of the bars is perpendicular to the plane so that the cover thickness on the sides can be defined.





8. If you want to modify the reinforcement, do one of the following:

    - Use direct modification (page 544). Ensure that the  **Direct modification** switch is active.
    - Double-click the reinforcement to open the **Rebar group** properties and modify the properties (page 1009).

See also

Create a curved reinforcing bar group (page 505)

Create a circular reinforcing bar group (page 507)

Create a tapered or spiral reinforcing bar group (page 509)

Create a reinforcing bar group using Rebar shape catalog

A reinforcing bar group includes several identical, or very similar, reinforcing bars. You can create a reinforcing bar group by selecting a predefined reinforcement shape from **Rebar shape catalog**. The predefined shapes in **Rebar shape catalog** are based on the shapes that have been defined in **Rebar shape manager** and saved in the `RebarShapeRules.xml` file.

Rebar shape catalog does not work with tapered reinforcing bar groups (page 509) or with 3D bar shapes.

NOTE If you do not want to use the predefined shapes but want to manually define the bar shape, use the Bar group (page 496) command instead.

---

1. On the **Rebar** tab, click **Bar group** --> **Rebar shape catalog**.

   The **Rebar Shape Catalog** dialog box opens.

2. Select one of the predefined shapes from the tree view on the left.

   You can add frequently used shapes to the tree view (page 501), or delete the shapes that you do not need.

   If you select an existing reinforcement in the model and click the **Get** button, the properties of that reinforcement are displayed in the **Rebar Shape Catalog** dialog box.

3. If needed, modify the bar properties.

   - To set a **Length of leg** value, click a leg in the preview of the shape.

     If you do not enter a **Length of leg** value, the leg length is calculated automatically according to the dimensions of the concrete part.

   - To set a **Bending angle** value for a bend that is not 90 degrees, click one of the legs next to the bend.

   - For circular, polygonal and spiral reinforcement, you can enter **Circle diameter** and **Overlap distance** values.

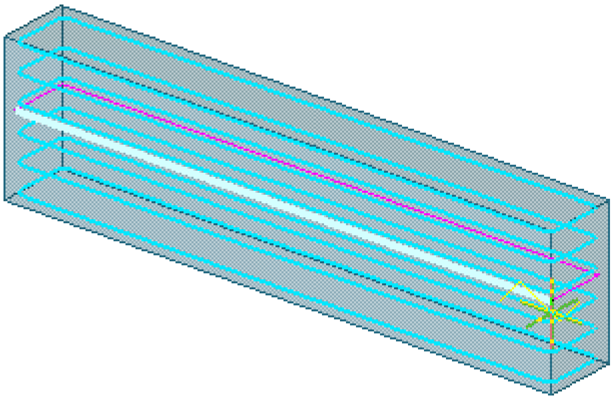
   The hook properties are visible only if you have set the advanced option XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION to `FALSE` in **File menu** --> **Settings** --> **Advanced Options** --> **Concrete Detailing**.

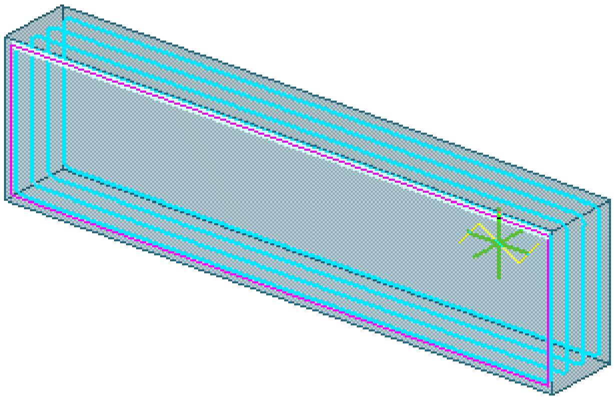
4. If needed, set the reference point of the reinforcement (page 501) to start, middle, or end by double-clicking the different legs or hooks in the preview of the shape.

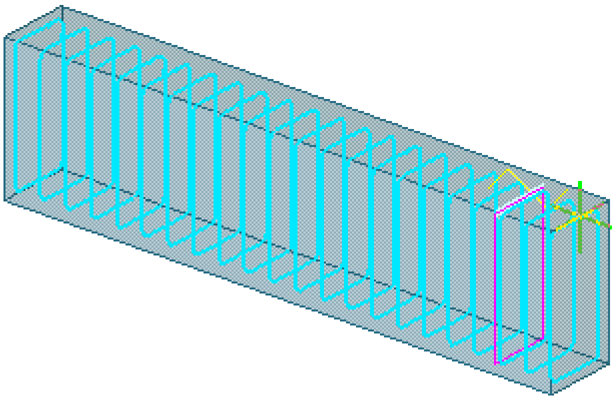
5. Click **OK**.

6. In the model, place the mouse pointer over a part face or edge.

A preview showing the placing and dimensions of the reinforcement is displayed.







7. Based on the preview, select a placing for the reinforcing bar group and click the left mouse button.

   Tekla Structures creates the reinforcement.

8. If you want to modify the reinforcement, do one of the following:

- Use direct modification (page 544). Ensure that the **Direct modification** switch is active.
- Double-click the reinforcement to open the reinforcing bar group properties, and modify the properties (page 1009).

See also

Create a reinforcing bar group (page 496)

Create a rebar set using Rebar shape placing tool (page 483)

Add more reinforcement shapes to the tree view in Rebar shape catalog

You can modify the tree view in **Rebar shape catalog** by adding frequently used shapes to the tree, or deleting the shapes that you do not need.

1. On the **Rebar** tab, click **Bar group** --> **Rebar shape catalog**.

   The **Rebar Shape Catalog** dialog box opens.

2. Click **Organize catalog**.
3. Create a new category folder by clicking .
4. Drag and drop the selected shapes to the folder.

   If multiple shapes have the same shape code and you drag them to the categories, the shape codes get a suffix **(1)**, **(2)**, and so on. You can rename the shapes as you wish by clicking the name twice and entering a new name or suffix, for example, **(a)**, **(b)**.

   When the shapes are listed in a report, they all get the same shape code.

5. Change the name of the folder in a similar way, if needed.
6. To remove a shape from a category, select the shape and click .
7. Click **OK**.

See also

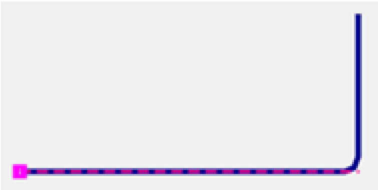
Create a reinforcing bar group using Rebar shape catalog (page 498)

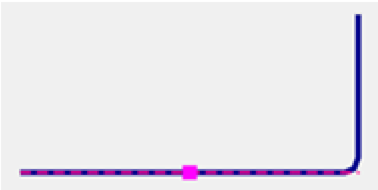
Set the reinforcement reference point in Rebar shape catalog

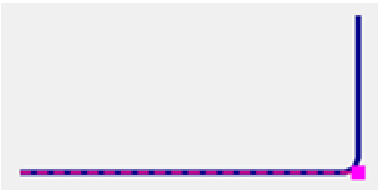
When you use **Rebar shape catalog** and select a shape, you can set the reference point to the start, middle or end of the reinforcing bar leg. When you create the reinforcement in the model, you can move the reinforcement to a new location by dragging the reference point. This is useful, for example, when the reinforcing bar legs are of certain length and you want to aim the reference point, for example, to the middle of a part edge. You can also move the reference point of circular reinforcement shapes.

1. On the **Rebar** tab, click **Bar group** --> **Rebar shape catalog**.

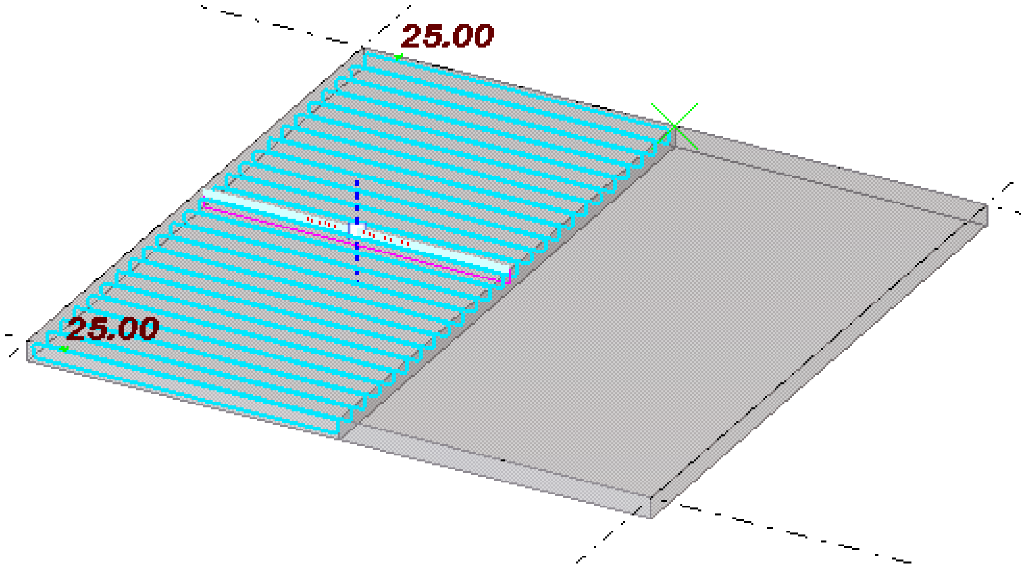
   The **Rebar Shape Catalog** dialog box opens.
2. Select a reinforcement shape.
3. Set the reference point to the desired location (start, middle, end) by double-clicking the position in the preview of the shape.

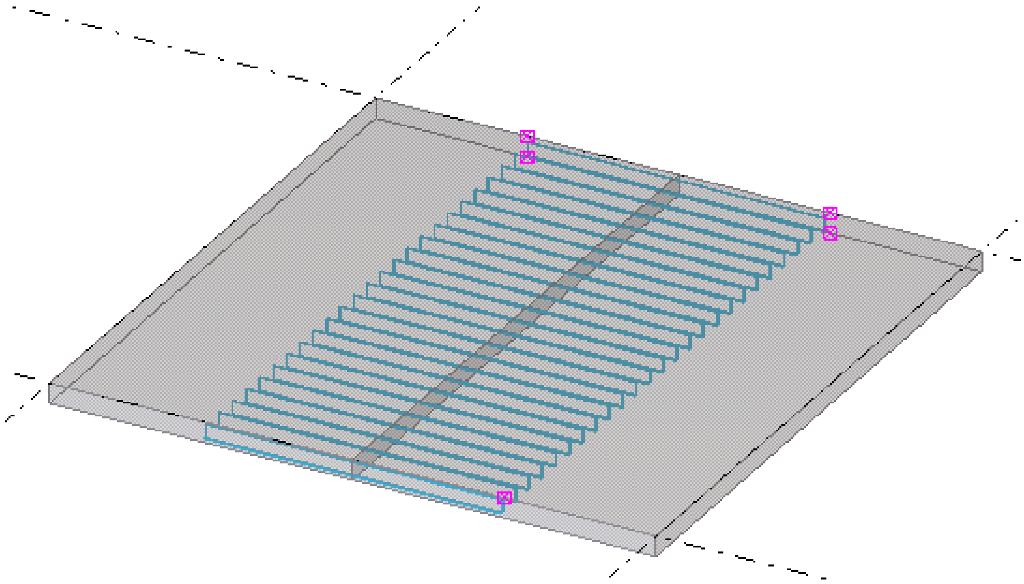
   

   
4. If needed, modify the bar properties.
5. Click **Apply** or **OK**.
6. In the model, place the mouse pointer over a part face or edge.
7. Based on the preview, select the desired placing and hold down the **Alt** key and click the left mouse button.

The reference point is displayed.



8. Move the reinforcement to a new location by dragging the reference point.
9. Click the middle mouse button to create the reinforcement.



NOTE For circular reinforcement you can set the reference point to the center line as follows:

a. Place the mouse pointer over a column edge to have the reinforcement oriented correctly.

b. Hold down the **Alt** key and click the left mouse button.

c. Drag the reference point and hold down **Shift** key to snap to the center of column.

d. Click the middle mouse button to create the reinforcement.

See also

Create a reinforcing bar group using Rebar shape catalog (page 498)

Reinforce pour objects using Rebar shape catalog

You can reinforce pour objects in pour views using **Rebar shape catalog**.

NOTE Rebar sets (page 470) and **Rebar shape catalog** are the methods to reinforce pour objects in pour views. If you want to use other reinforcement commands, such as Bar group (page 496), or reinforcement components, you need to reinforce single parts in part views. All reinforcement are visible both in part views and in pour views.

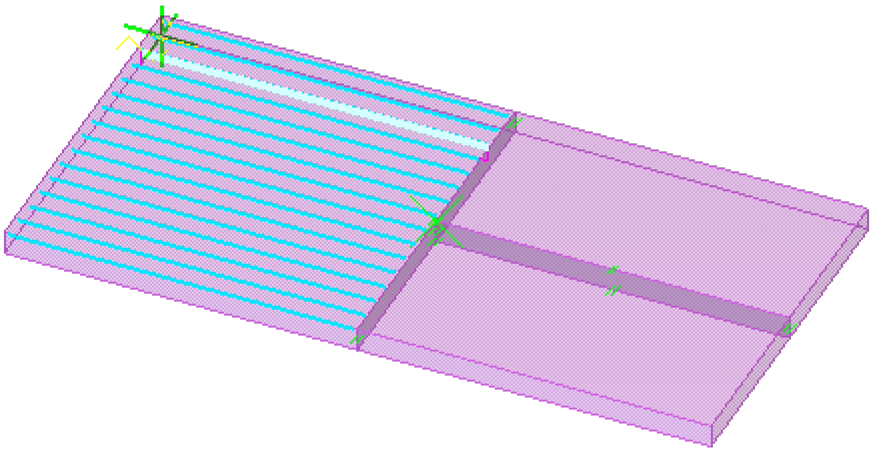
When you reinforce pour objects using **Rebar shape catalog**:

- The reinforcement is attached to the reinforced part, not to the pour object.
- The reinforcement geometry is defined in accordance with the pour object geometry even though the reinforcement is attached to a part. For example, pour breaks can limit the length of reinforcing bars.
- In reports the reinforcement information is listed according to the part, not to the pour object.

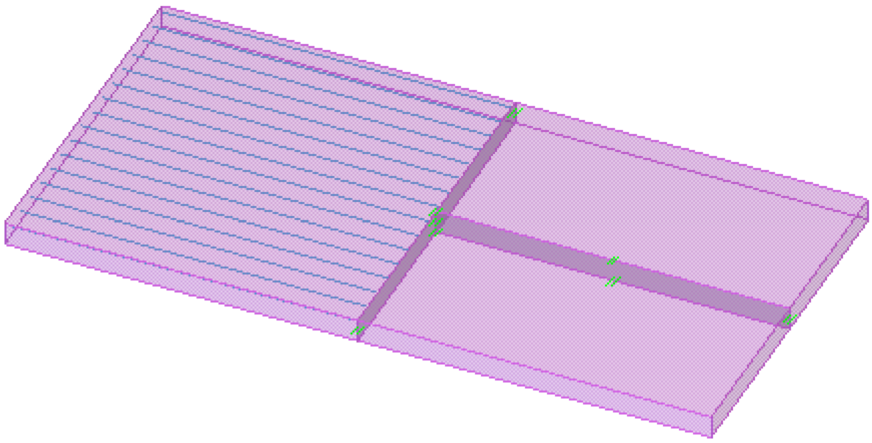
Before you start, create concrete parts whose cast unit type is **Cast in place**. Tekla Structures automatically forms pour objects of them.

1. Ensure that you are using a pour view. If not, click **Pour view** on the **Concrete** tab.
2. If needed, create pour breaks by selecting any of the **Pour break** commands on the **Concrete** tab:
    - **Single point**
    - **Two points**
    - **Multiple points**
3. To insert a reinforcement to a pour object, on the **Rebar** tab, click **Bar group** --> **Rebar shape catalog**.

    The **Rebar Shape Catalog** dialog box opens.
4. Select a shape from the tree view on the left and modify the properties, if needed.
5. Click **OK**.
6. In the model, place the mouse pointer over a face or an edge of a pour object.



7. Based on the preview, select a placing for the reinforcement and click the left mouse button to create the reinforcement.



See also

Create a reinforcing bar group using Rebar shape catalog (page 498)

Manage pours (page 444)

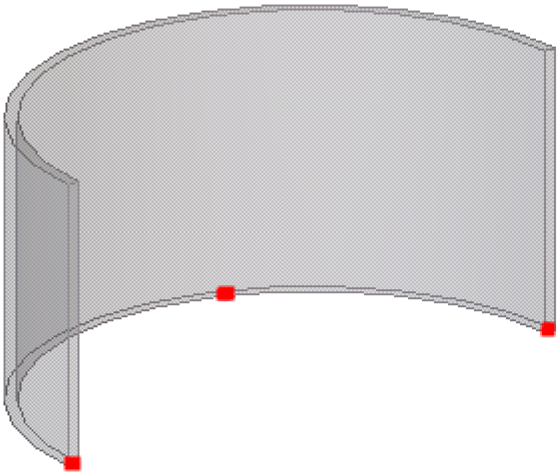
Create a curved reinforcing bar group

You can reinforce curved segments in a concrete beam or a curved wall.

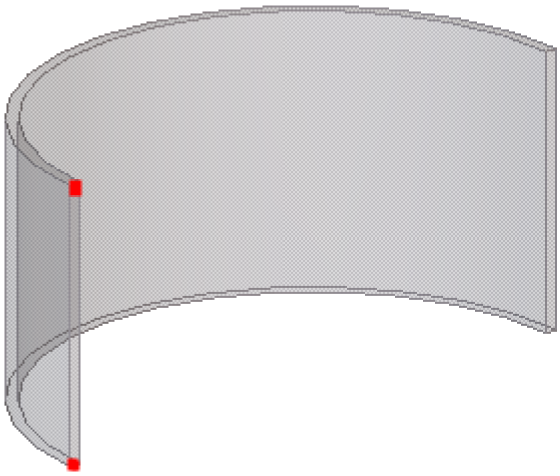
1. On the **Rebar** tab, click **Bar group** --> **Curved group**.

   If you need to modify the properties before you create the reinforcement, hold down **Shift** and click the **Curved group** command to open the **Curved bar** properties.
2. Select the part to reinforce.

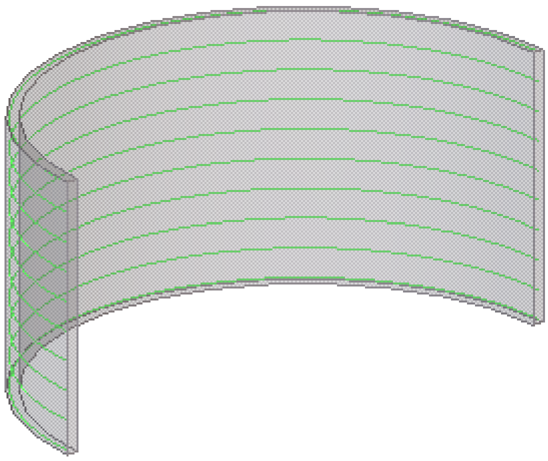
   Tekla Structures attaches the bar group to this part.
3. Pick three points on an arc to define the curve.



4. Pick two points to indicate the distribution direction of the bars.



Tekla Structures creates a group of curved reinforcing bars.



5. If you want to change the curved reinforcing bar group properties:
   a. Double-click the curved reinforcing bar group to open the **Curved bar** properties.
   b. Modify the properties (page 1009).
   c. Click **Modify**.

See also

Create a reinforcing bar group using Rebar shape catalog (page 498)

Create a reinforcing bar group (page 496)

Create a circular reinforcing bar group (page 507)

Create a tapered or spiral reinforcing bar group (page 509)

Modify reinforcement (page 520)

Create a circular reinforcing bar group

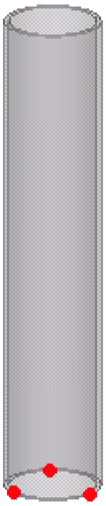
You can reinforce round circular columns.

1. On the **Rebar** tab, click **Bar group** --> **Circular group**.

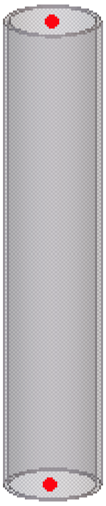
   If you need to modify the properties before you create the reinforcement, hold down **Shift** and click the **Circular group** command to open the **Circular rebar** properties. For example, to create a circular spiral bar, set **Rebar group type** to **Spiral**.
2. Select the part to reinforce.

   Tekla Structures attaches the bar group to this part.
3. Pick three points on the outer contour of the concrete part to define the circular bars.

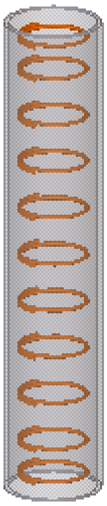
The radius is automatically calculated from these three points.



4. Pick two points to indicate the distribution direction of the bars.



Tekla Structures creates a group of circular reinforcing bars.



NOTE If you want to modify the splice length of the round stirrups, enter negative values in the **Start** and **End** boxes in the **Circular rebar** properties.

5. If you want to change the circular reinforcing bar group properties:
   a. Double-click the circular reinforcing bar group to open the **Circular rebar** properties.
   b. Modify the properties (page 1009).
   c. Click **Modify**.

See also

Create a reinforcing bar group using Rebar shape catalog (page 498)

Create a reinforcing bar group (page 496)

Create a curved reinforcing bar group (page 505)

Create a tapered or spiral reinforcing bar group (page 509)

Modify reinforcement (page 520)

Create a tapered or spiral reinforcing bar group

For example for rectangular concrete parts with variable cross sections, you can create tapered reinforcing bar groups by specifying the bar shape at each cross section. For concrete parts with a uniform cross section, you can also create spiral reinforcing bar groups, and it is sufficient to pick two points to define the distribution area of the bar group. If the part shape is not rectangular, an alternative bar shape can be defined.

Use the **Rebar group type** list in the reinforcing bar group properties to select or change the bar group type.

To create a circular spiral bar group (page 507), use the **Bar group** --> **Circular group** ribbon command and the **Circular rebar** properties instead.

1. On the **Rebar** tab, hold down **Shift** and click .

   The **Rebar group** properties open.

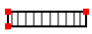
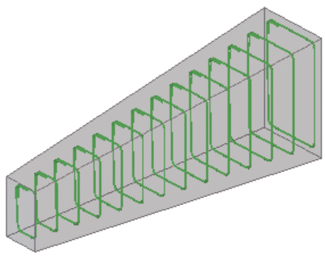
2. In the **General** section, select a tapered or spiral option from the **Rebar group type** list.
3. If needed, enter or modify the other bar properties.
4. Select the part to reinforce.

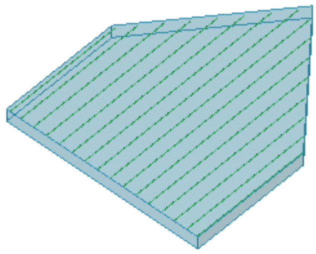
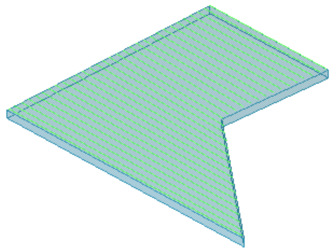
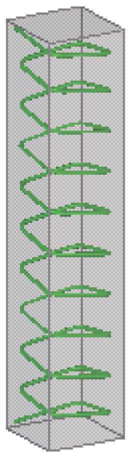
   Tekla Structures attaches the bar group to the part.

5. Pick points to define the shape of the bar at the first cross section.
6. Click the middle mouse button to finish picking.
7. Depending on the bar group type and the number of cross sections, do one of the following:
   - At the second and each subsequent cross section for a tapered bar group, pick points to define the bar shape. Then click the middle mouse button to finish picking at the cross section.
   - For a spiral bar group, pick two points to define the distribution area of the bar group.

Tekla Structures creates the reinforcement.

Reinforcing bar group types

Option	Description	Example
 Normal	Not tapered. Pick two points to define the distribution area of the bar group.	
 Tapere d	One bar dimension changes linearly in the group.	

Option	Description	Example
Tapered ridge	One bar dimension changes linearly in the group. The dimension is longest in the middle of the group.	
Tapered curved	One bar dimension changes along a curve. The dimension is longest in the middle of the group.	
Tapered N	One bar dimension changes linearly between N cross sections. Enter the number of cross sections in the **Number of cross sections** box.	
Spiral	The reinforcing bars rise in a polygonal or circular shape along the longitudinal axis of the part.	

See also

Create a reinforcing bar group using Rebar shape catalog (page 498)

Create a reinforcing bar group (page 496)

Reinforcing bar and bar group properties (page 1009)

Modify a single reinforcing bar, bar group, or mesh (page 544)

Create a reinforcement mesh

You can create a reinforcement mesh that consists of two perpendicular bar groups. Tekla Structures treats mesh bars as one unit but distinguishes the main and crossing bars.

The reinforcement mesh can be rectangular, polygonal, or bent. You can also create a customized reinforcement mesh.

NOTE You cannot change the mesh type once the mesh has been created.

Create a rectangular reinforcement mesh

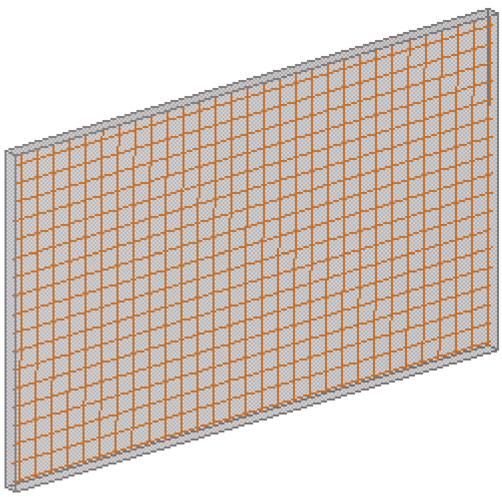
1. On the **Rebar** tab, hold down **Shift** and click  **Mesh**.

   The **Rebar mesh** properties open in the property pane.
2. In the **Mesh type** list, select **Rectangle**.

   NOTE You cannot change the mesh type once the mesh has been created.
3. Select the part to reinforce.

   Tekla Structures attaches the mesh to this part.
4. Pick the start point of the mesh.
5. Pick a point to indicate the direction of the longitudinal bars.
6. Click the middle mouse button to finish picking.

   Tekla Structures creates the mesh parallel to the work plane, to the left of the points you picked.

   
7. If you want to modify the reinforcement mesh, do one of the following:

- Use direct modification (page 544). Ensure that the **Direct modification** switch is active.
- Double-click the reinforcement to open the **Rebar mesh** properties and modify the properties (page 1011).

Create a polygonal reinforcement mesh

1. On the **Rebar** tab, hold down **Shift** and click **Mesh**.

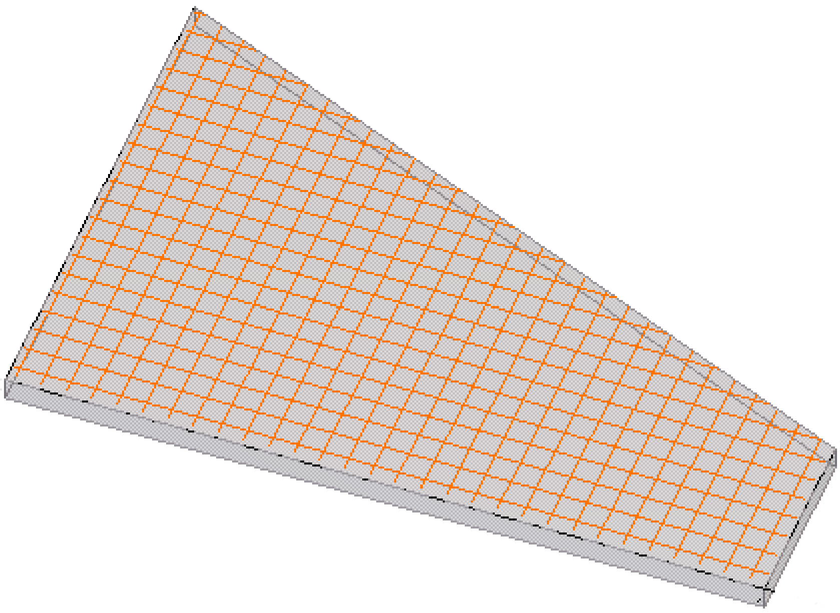
   The **Rebar mesh** properties open in the property pane.
2. In the **Mesh type** list, select **Polygon**.

   NOTE You cannot change the mesh type once the mesh has been created.

3. Select the part to reinforce.

   Tekla Structures attaches the mesh to this part.
4. Pick the start point of the mesh.
5. Pick the corner points of the mesh.
6. Click the middle mouse button to finish picking.
7. Pick a point to indicate the direction of the longitudinal bars.

   Tekla Structures creates the mesh.

8. If you want to modify the reinforcement, do one of the following:

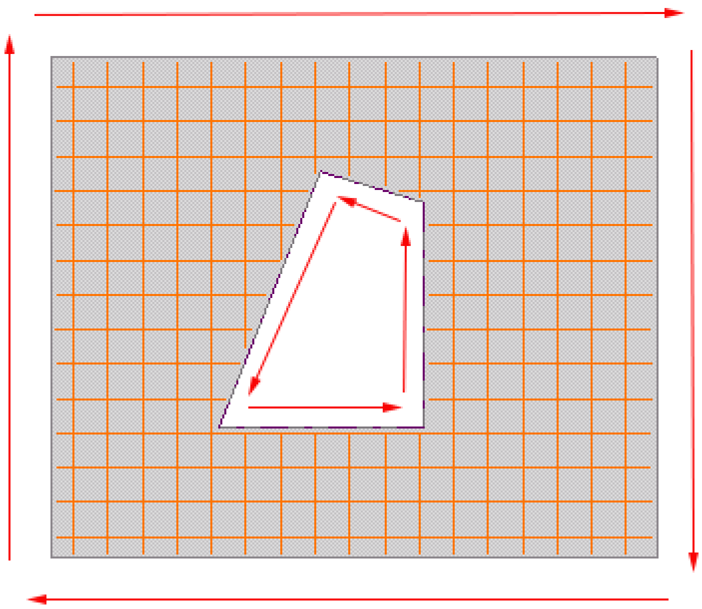
- Use direct modification (page 544). Ensure that the **Direct modification** switch is active.
- Double-click the reinforcement to open the **Rebar mesh** properties and modify the properties (page 1011).

Reinforcement mesh with holes

If you want to reinforce a part that has holes, you need to pick the corner points of holes when you create the reinforcement.

1. Select the part to reinforce.
2. Pick the start point of the mesh.
3. Pick the corner points of the mesh.
4. Pick the corner points of the hole.

   Note that you need to pick the corner points of the hole in the opposite direction than the corner points in the mesh.



5. Click the middle mouse button to finish the picking.
6. Pick a point to indicate the direction of the longitudinal bars.

Create a bent reinforcement mesh

1. On the **Rebar** tab, hold down **Shift** and click **Mesh**.

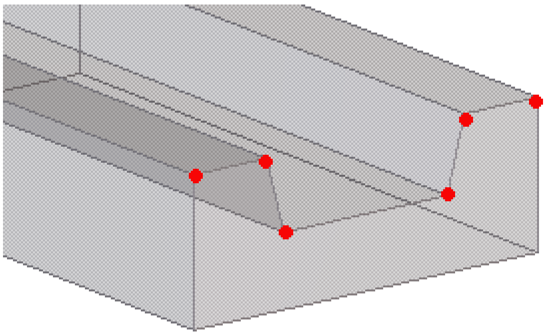
   The **Rebar mesh** properties open in the property pane.

2. In the **Mesh type** list, select **Bent**.

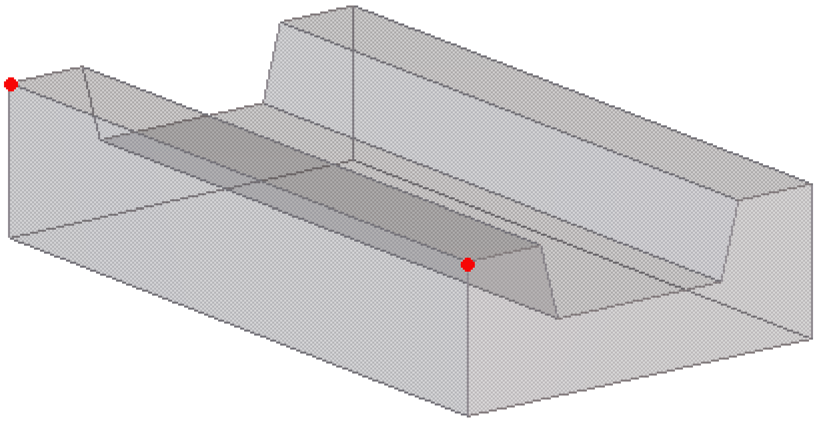
NOTE You cannot change the mesh type once the mesh has been created.

3. Enter the bending radius.
4. Select the part to reinforce.

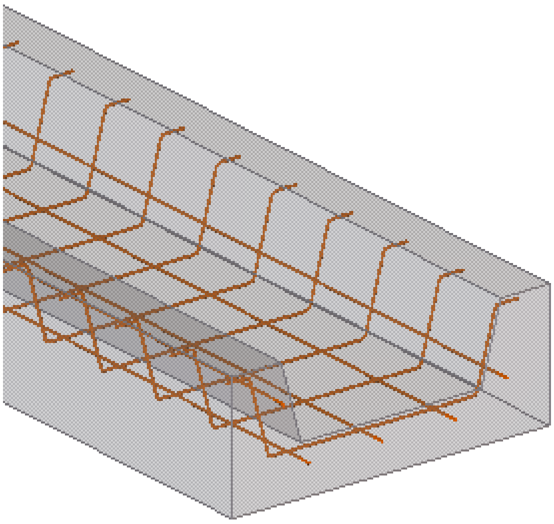
   Tekla Structures attaches the mesh to this part.
5. Pick three or more points to indicate the bending shape of the crossing bars.



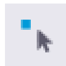
6. Click the middle mouse button to finish picking.
7. Pick two points to indicate the length and direction of the longitudinal bars.



Tekla Structures creates the mesh.



8. If you want to modify the reinforcement mesh, do one of the following:

   - Use direct modification (page 544). Ensure that the  **Direct modification** switch is active.
   - Double-click the reinforcement to open the **Rebar mesh** properties and modify the properties (page 1011).

Create a customized reinforcement mesh

You can create a customized reinforcement mesh that consists of two perpendicular bar groups.

1. On the **Rebar** tab, hold down **Shift** and click  **Mesh**.

   The **Rebar mesh** properties open in the property pane.
2. In the **Layout** section, select the **Custom Mesh** option.
3. Enter an identifier for the mesh in the **Mesh** box.

   The default identifier is **Custom Mesh**.
4. Modify the other mesh properties (page 1013) as needed.
5. Select the part to reinforce.

   Tekla Structures attaches the mesh to this part.
6. Pick two points to indicate the direction of the longitudinal bars.
7. If you want to define the mesh plane, pick one more point.
8. Click the middle mouse button to finish picking.

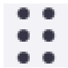
9. If needed, you can save customized properties as property files, and load these properties later on when you create new meshes.

Create a reinforcement strand pattern

You can create prestressed straight or deflected strands for concrete parts.

NOTE To allow positioning of the strands, first create points to the part you are creating the strands for. On the **Edit** tab, click **Points** and select **On plane** to open the **Point Array** dialog box. Define the point coordinates.

---

1. On the **Rebar** tab, click  **Strand**.

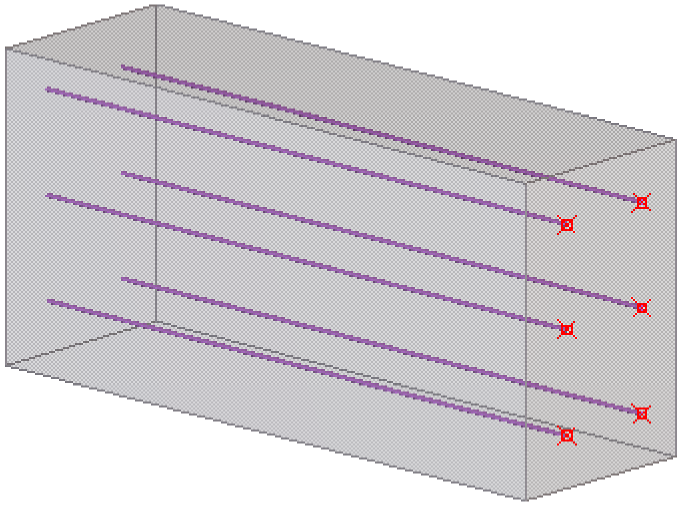
   If you need to modify the properties before you create the reinforcement, hold down **Shift** and click the **Strand** command to open the **Strand pattern** properties.

2. Select the part you are creating strands for.
3. Pick each of the points that you are using to position the strands (for example, at the end of a part).

   The points you pick define the first cross section.

4. Click the middle mouse button to finish picking.
5. Pick points to position the strands.
   - If you create a single cross section, pick two points to define the length of the strands.
   - If you create two or more cross sections, for each cross section, pick points to indicate the strand positions. Pick the strand positions in the same order as for the first cross section.
6. Click the middle mouse button to finish picking.

Tekla Structures creates the strands.



7. If you want to change the strand properties:

   a. Double-click the strand pattern to open the **Strand pattern** properties.

   b. Modify the properties (page 1032).

   c. Click **Modify**.

See also

Debond reinforcement strands (page 518)

Debond reinforcement strands

1. Double-click the strand pattern in which you want to debond strands.

   The **Strand pattern** properties opens.

2. Click the **Debonding** button to open the debonding properties.

3. On the **Debonding** tab, click the **Add** button to create a new row in the table.

4. Enter the strand numbers in the **Debonded strands** field.

   The strand number is the selection order number of the strand.

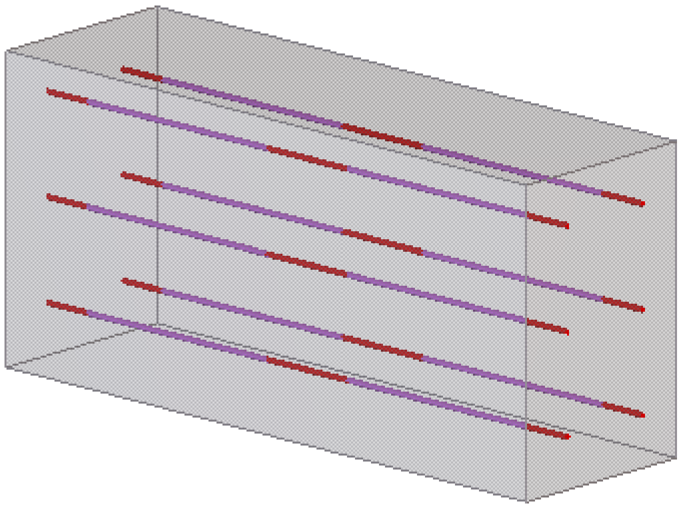
   - To set the same values for all the strands, enter all the strand numbers, separated by a space. For example, `1 2 3 4`.
   - To set separate values for each strand, click **Add** to add a new row, then enter the strand number in the **Debonded strands** field.

5. Define the debonded lengths.

   To set symmetrical lengths, select the **End lengths = start lengths** check box and only enter values in the **From start** or **Middle to start** fields.

6. Click **Modify**.

   Tekla Structures displays the debonded section of the strand in red.



See also

Create a reinforcement strand pattern (page 517)

Reinforcement strand properties (page 1032)

Create a reinforcement splice

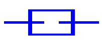
You can join reinforcing bars or reinforcing bar groups together with reinforcement splices. There can be a gap between the bars or groups.

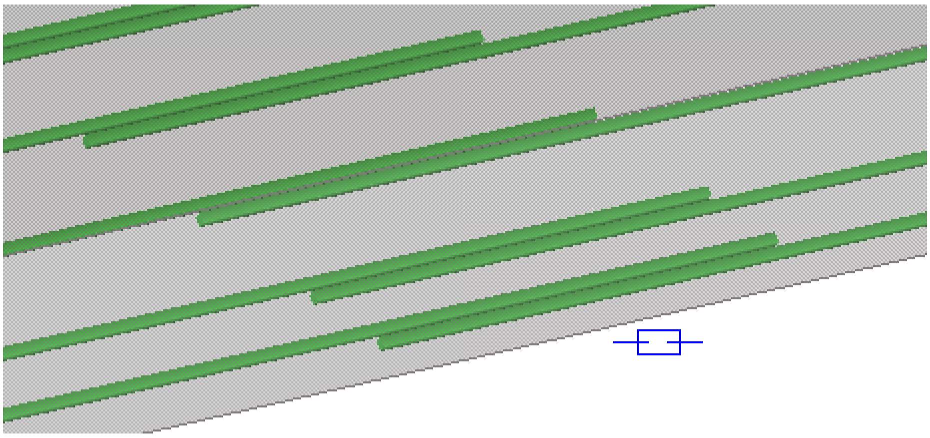
1. On the **Rebar** tab, click **Edit** --> **Rebar splice**.

   If you need to modify the properties before you create the reinforcement, hold down **Shift** and click the **Rebar splice** command to open the **Rebar splice** properties.

2. Select the first reinforcing bar or bar group.

3. Select the second reinforcing bar or bar group.

   Tekla Structures creates the splice. The splices have blue splice symbols  in the model.



4. If you want to change the splice properties:
   a. Double-click the splice to open the **Rebar splice** properties.
   b. Modify the properties.
   c. Click **Modify**.

Splice properties

Use the **Rebar splice** properties to view and modify the properties of splices. The file name extension of a saved splice property file is `.rsp`.

Option	Description
Joint type	Splice type. **Lap left** creates the lap to the direction of the first reinforcing bar or bar group selected, **Lap right** to the direction of the second. **Lap both** centers the lap between the bars or bar groups.
Lap length	Length of the lap joint.
Offset	Offset of the splice center point from the point where the bars originally met.
Bar positions	Select whether the lapping bars are on top of each other or parallel to each other.

See also

Create a reinforcing bar group using Rebar shape catalog (page 498)

Create a reinforcing bar group (page 496)

Split and splice a reinforcement (page 564)

2.10 Modify reinforcement

Once you have added reinforcement to your model, you can modify the shape of the reinforcement, for example. Tekla Structures includes several methods for the modification.

Rebar sets

When you modify rebar sets, you can use direct modification on the rebar set guidelines (page 521), leg faces (page 525), and modifiers (page 533).

Single reinforcing bars, bar groups, and meshes

When you modify single reinforcing bars, bar groups, or meshes, you can use:

- direct modification (page 544)
- handles (page 554)
- grouping (page 551)
- combining (page 552)
- splitting (page 553)

See also

Use adaptivity to modify a reinforcement (page 561)

Attach a reinforcement object to a concrete part (page 562)

Split and splice a reinforcement (page 564)

Assign running numbers to reinforcement (page 566)

Classify reinforcement to layers (page 567)

How to calculate the reinforcing bar length (page 570)

How to calculate the reinforcing bar leg length (page 573)

Modify a rebar set

You can modify rebar sets by changing the rebar set properties, by using the rebar set guidelines or leg faces, or by creating local rebar set modifiers. The guidelines, leg faces, and modifiers all have direct modification handles.

NOTE When you work with rebar sets, ensure that the  **Direct modification** switch is active.

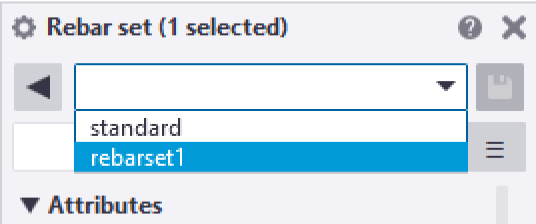
In case you open an existing model using a new version of Tekla Structures, always update the existing rebar sets first: on the **Rebar** tab, click **More** --> **Regenerate**.

See also Modify a rebar set using leg faces and leg surfaces (page 525) and Modify a rebar set locally using modifiers (page 533).

Modify the properties of a rebar set

You can change the properties of a rebar set in the property pane or on the contextual toolbar.

1. Double-click the rebar set that you want to modify.
2. If you want to use previously saved properties from a file, select the property file from the topmost list in the property pane:



3. Modify the rebar set properties (page 1016) in the property pane.
4. Click **Modify** to save the changes.
5. To save the properties for later use, enter a name for the property file in the topmost box in the property pane, and then click .

TIP Alternatively, you can modify the rebar set properties on the contextual toolbar.

Change the layer order of a rebar set

You can adjust the order of the bar layers when two or more rebar sets overlap.

By default, the layer order is based on the creation order of the rebar sets. Tekla Structures automatically places the bars that are created first closest to the concrete surface, and the bars created last will be the furthest.

1. Select a rebar set.
2. On the contextual toolbar, adjust the layer order by using the following buttons:

   - Click  to move the bars to the outermost layer.
   - Click  to move the bars one layer outwards.
   - Click  to move the bars one layer inwards.
   - Click  to move the bars to the innermost layer.

   Alternatively, you can enter a number in the **Layer number** box in the property pane, and then click **Modify** to save the changes.

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 set the same layer number for several rebar sets, the bars will be placed on the same layer, and the bars may collide.

3. If needed, fine-tune the layer order at any individual leg face or leg surface (page 525) separately.

   These modifications override the default settings and the layer order settings of the entire rebar set.

Modify a rebar set using guidelines

The guidelines of a rebar set define the distribution direction of the bars. The spacing of the bars is also measured along the guidelines. You can modify the rebar set guidelines by using direct modification.

See also Resize and reshape model objects (page 107), Distribute bars in a rebar set (page 541), and Create a secondary guideline (page 537).

To show or hide (page 540) the guidelines when you select rebar sets in the model, go to the **Rebar** tab and click **Visibility** --> **Guidelines**. Alternatively, you can use the keyboard shortcut **Alt+2** or the advanced option XS_REBARSET_SHOW_GUIDELINES.

To modify a guideline, select a rebar set and do any of the following:

- To move a guideline, drag the line handle.
- To move a guideline point, drag the point handle .
- To add a new point to the start or end of a guideline:
  1. Select the start or end point of the guideline .
  2. Click  **Add new point** on the contextual toolbar.
  3. Pick a location for the new start or end point.
- To add an intermediate point to a guideline, drag a midpoint handle .
- To remove a point from a guideline, select the point and press **Delete**.
- To modify chamfers at intermediate corner points of a guideline:
  1. Select a corner point.
  2. Define the chamfer type and dimensions (page 397) on the contextual toolbar.
- To change the direction (page 539) of a guideline, select the guideline and click  **Swap ends** on the contextual toolbar.

- To make a guideline follow the leg face edges that are located between the guideline end points:
  1. Move the guideline end points to the leg face edges.
  2. If there are cuts at the edges, add intermediate points to the guideline, and drag the handles to the corners of the cuts.
  3. Double-click the guideline to open its properties in the property pane. Select **Yes** in the **Follow edges** list, and then click **Modify**.

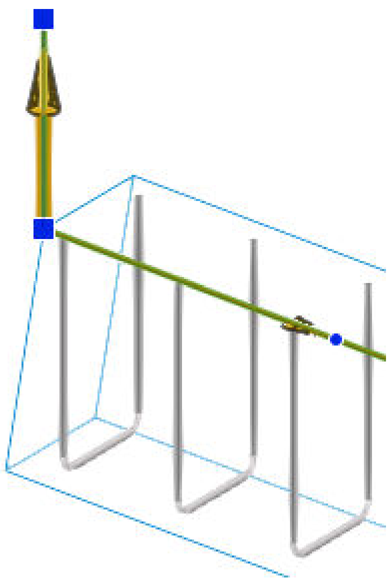
     Alternatively, select the guideline and click on the contextual toolbar.

Change the bar plane orientation of a rebar set

By default, Tekla Structures creates rebar set bars at planes that are perpendicular to the guidelines. In slanted or complex concrete structures, you may want to align the bar planes with the global z direction of the model, or some other way.

1. Select a rebar set.
2. On the contextual toolbar, click **Bar plane orientation**.

   Tekla Structures shows a yellow arrow with two blue direct modification handles, at the start point of the primary guideline. The bars become aligned with the global z direction.



3. To adjust the bar plane orientation some other way, drag the handles.

   For example, you can let the top handle snap to a certain location, such as a part corner, in the model.

4. If you want to revert to the original bar plane orientation, and to hide the yellow arrow, click  again on the contextual toolbar.

Modify a rebar set using leg faces and leg surfaces

In addition to modifying an entire rebar set, you can make changes to any individual leg face or leg surface. Leg surfaces are especially useful when you reinforce curved or complex concrete structures.

NOTE If a leg face or leg surface is shown in red in the model, it means that the geometry of some bars in the rebar set is not valid. Check and fix the properties of the rebar set (page 1016) (for example, the bending radius) or the leg face (page 1020) or leg surface (page 1021).

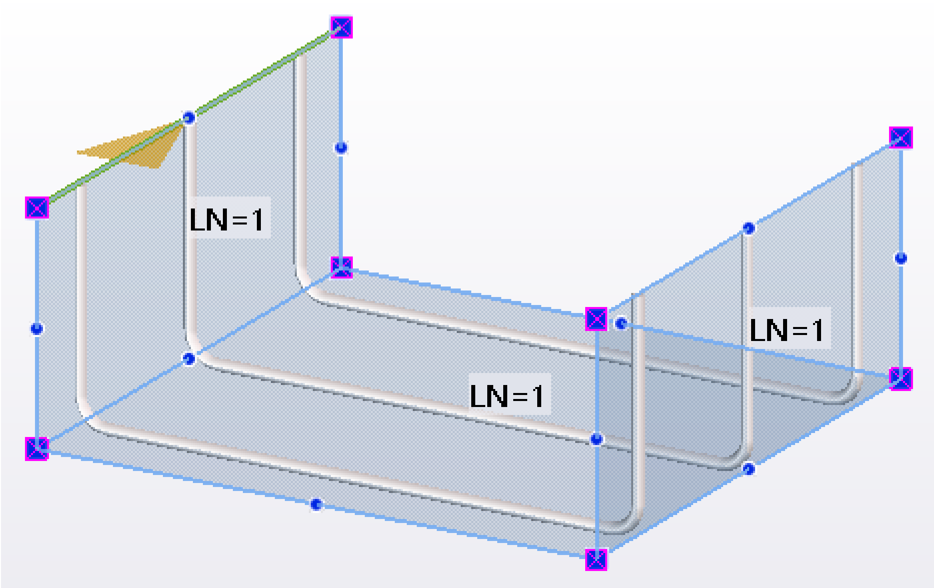
Show the leg faces and leg surfaces

To modify rebar sets by using leg faces, you first need to make the leg faces visible.

Tekla Structures also shows the leg surfaces. They automatically adapt to the geometry of the parts they are associated with. You can only modify leg surfaces by using their properties (page 1021) or contextual toolbar, not by using direct modification.

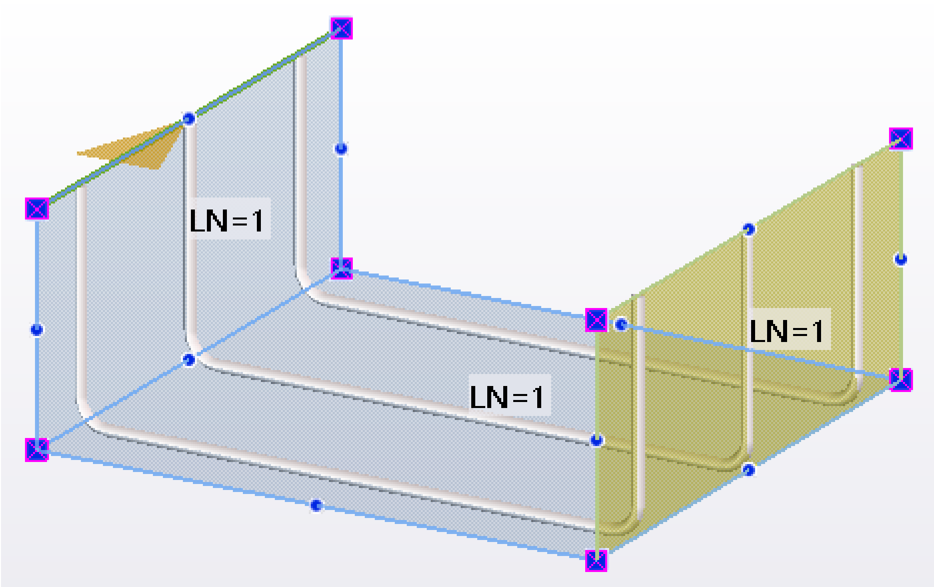
1. Ensure that the  **Direct modification** switch is active.
2. On the **Rebar** tab, click **Visibility** --> **Leg faces**.
3. Select a rebar set.

Tekla Structures shows the leg faces and leg surfaces. Tekla Structures also shows the bar layer numbers of the rebar set on each leg face or leg surface, for example LN=1.



4. Move the mouse pointer over a leg face or leg surface and click to select it.

   Tekla Structures highlights the leg face or leg surface in yellow.



Alternatively, you can use the keyboard shortcut **Alt+1** or set the advanced option XS_REBARSET_SHOW_LEGFACES to `TRUE`.

Modify the leg faces

You can use any of the following methods when you modify rebar set leg faces.

1. To move a leg face, drag it to a new location.

   The connected leg face planes remain intact.

   If you want the connected leg faces to follow the dragged leg face, hold down **Alt** when dragging. The size of the dragged leg face remains the same, but the connected leg face planes may change.

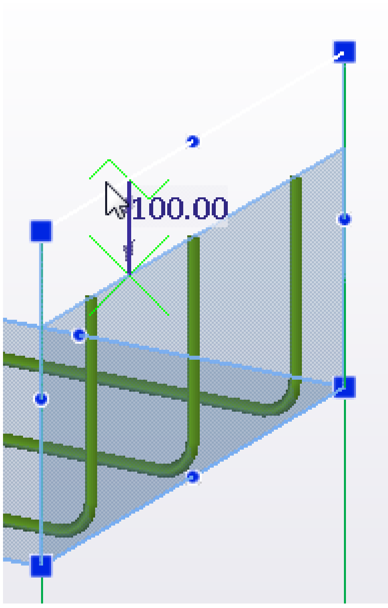
   If you want to detach the leg face from the connected leg faces, hold down **Shift** when dragging.

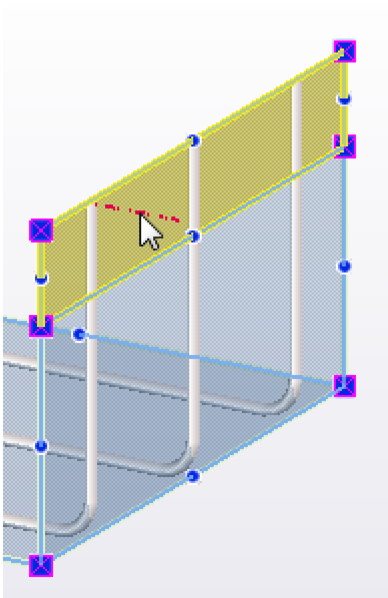
2. To move an edge of a leg face, drag the edge to a new location.

   The connected leg faces follow, if possible.

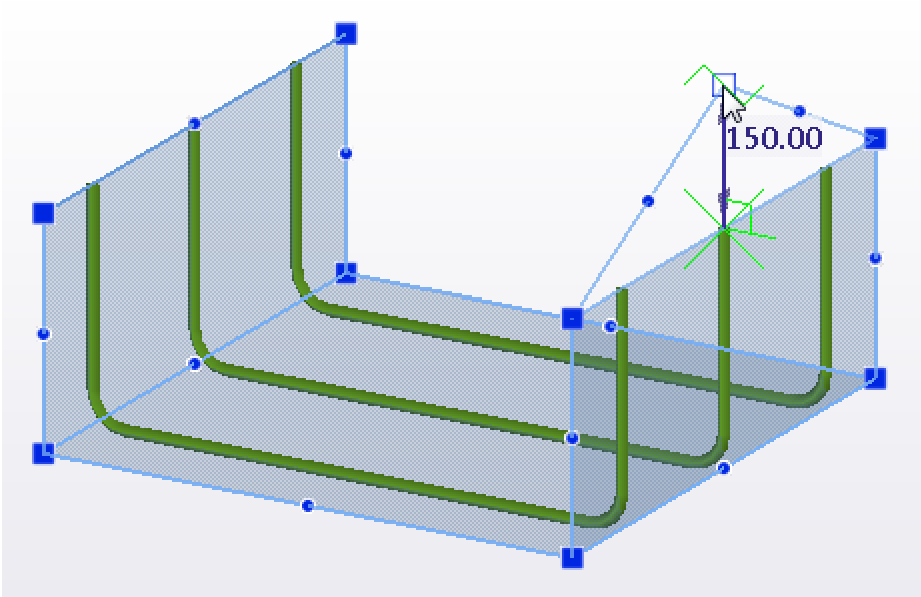
3. To create a parallel copy of a leg face, hold down **Ctrl** and drag the leg face.

4. To create a new, connected leg face, hold down **Ctrl** and drag an edge of the leg face.



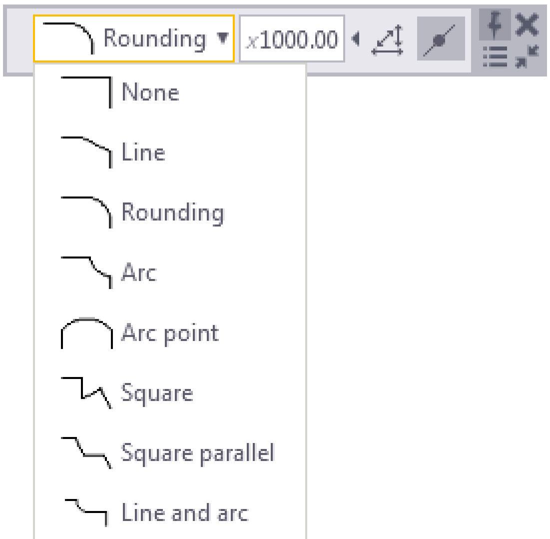


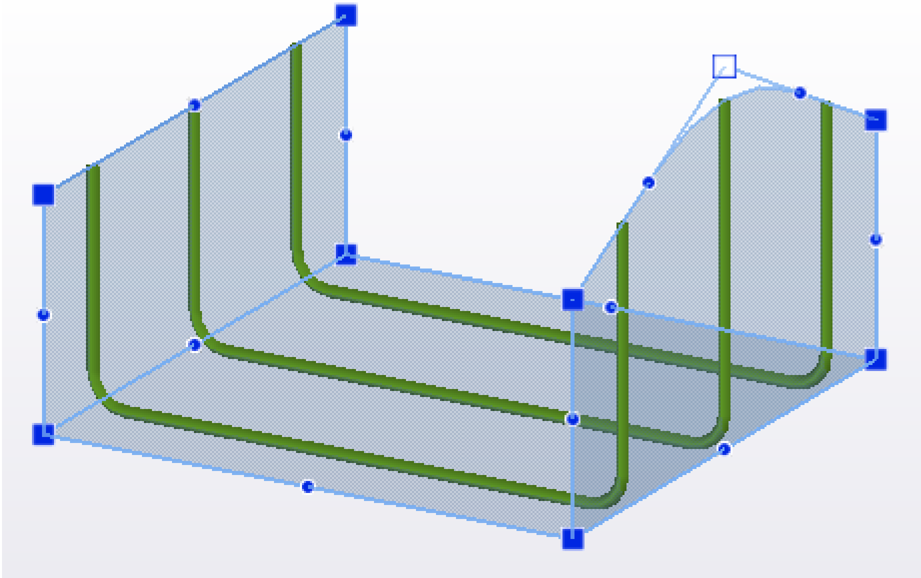
5. To add a new leg face at a part face or pour object face, ensure that the rebar set is selected, click **Add leg face** on the contextual tab on the ribbon, and then select the part face or pour object face.
6. To create a new leg face on the basis of its corner points, switch **Picking mode** to , click **Add leg face** on the contextual tab on the ribbon, and then pick points to indicate the leg face corners. Click the middle mouse button to finish picking.
7. To add a new corner point to a leg face, drag a midpoint handle.



8. To remove a corner point from a leg face, select the point and press **Delete**.

9. To modify a corner chamfer of a leg face, select the corner point, and then select the chamfer type (page 397) and enter the chamfer dimensions on the contextual toolbar.

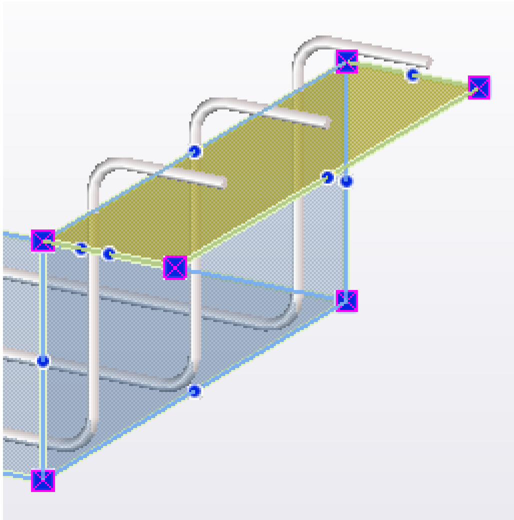


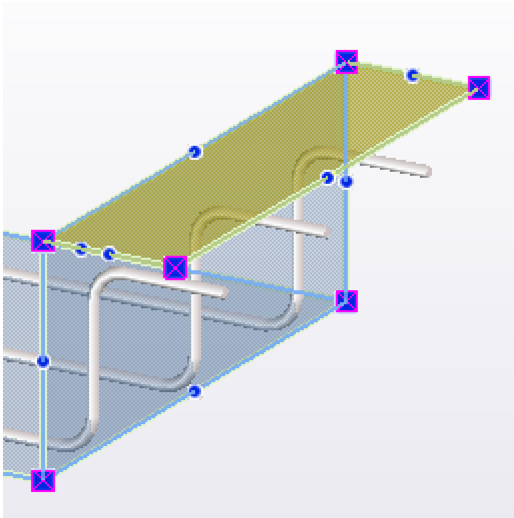


10. To remove a leg face, select it and press **Delete**.
11. To define an additional offset between a leg face and the bars, select the leg face and enter a value for **Additional offset** on the contextual toolbar, for example, 

.

    A negative value moves the bars outside the concrete.
12. To flip bars over to the other side of a leg face, select the leg face and click  **Flip bar side** on the contextual toolbar.





Note that after flipping, Tekla Structures searches for concrete on the other side of the leg face to create a concrete cover and apply the concrete cover settings. If there is no concrete, the concrete cover thickness will be zero.

13. To change the order of the bar layers at an individual leg face, select the leg face and adjust the layer order by using the following buttons on the contextual toolbar:

    - Click  to move the bars to the outermost layer.
    - Click  to move the bars one layer outwards.
    - Click  to move the bars one layer inwards.
    - Click  to move the bars to the innermost layer.

Alternatively, you can enter a number in the **Layer number** box in the property pane, and then click **Modify** to save the changes.

The smaller the layer number, the closer to the concrete surface the bar layer is. You can use both positive and negative numbers.

These modifications override the layer order settings of the entire rebar set (page 521).

TIP You can modify leg face properties (page 1020) also in the property pane.

Modify the leg surfaces

Use these methods to modify rebar set leg surfaces.

1. To add a new leg surface at a part face, ensure that the rebar set is selected, click **Add leg surface** on the contextual tab on the ribbon, and then select the part face.

   If you try to add a leg surface to a part face that already has a leg surface that belongs to the same rebar set, Tekla Structures does not add the new, duplicate leg surface.

2. To remove a leg surface, select it and press **Delete**.

3. To change the order of the bar layers at an individual leg surface, select the leg surface and adjust the layer order by using the following buttons on the contextual toolbar:

   - Click to move the bars to the outermost layer.
   - Click to move the bars one layer outwards.
   - Click to move the bars one layer inwards.
   - Click to move the bars to the innermost layer.

   Alternatively, you can enter a number in the **Layer number** box.

   The smaller the layer number, the closer to the concrete surface the bar layer is. You can use both positive and negative numbers.

   These modifications override the layer order settings of the entire rebar set (page 521).

4. To define an additional offset between a leg surface and the bars, select the leg surface and enter a value for **Additional offset** on the contextual toolbar.

   A negative value moves the bars outside the concrete.

5. To define whether the bars at a leg surface are cut by the openings in the concrete, select the leg surface and click the **Create holes** button on the contextual toolbar.

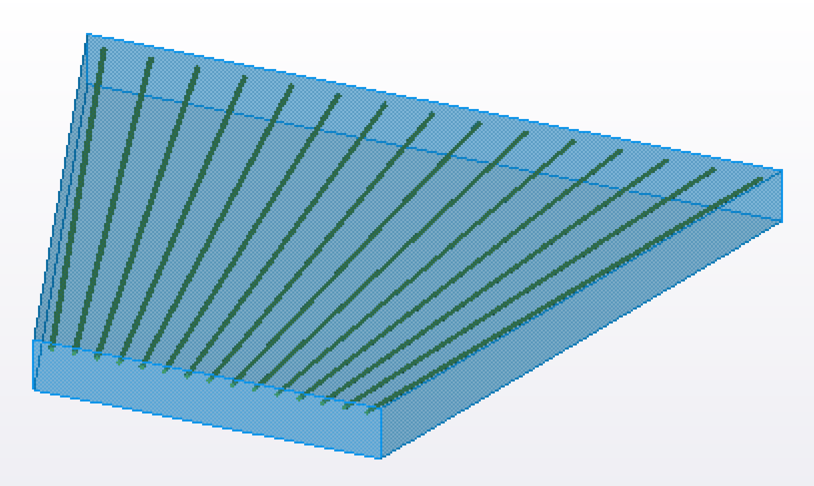
TIP You can modify leg surface properties (page 1021) also in the property pane. After modifying the properties of the selected leg surfaces, click **Modify** to save the changes.

Modify a rebar set locally using modifiers

You can use *modifiers* to modify a rebar set at certain locations only.

For example, you can create a local *property modifier* to only change the properties of certain bars in a rebar set, or you can create hooks or threading by adding an *end detail modifier*, or you can split rebar set bars into stock lengths by using *splitters*.

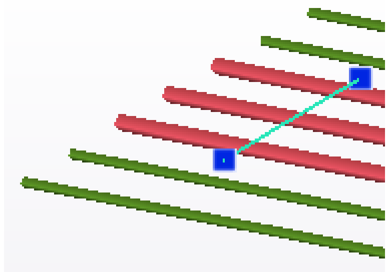
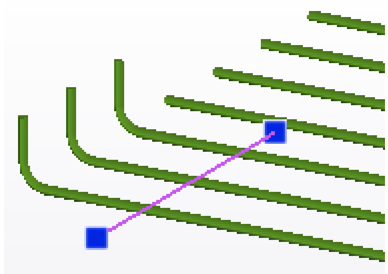
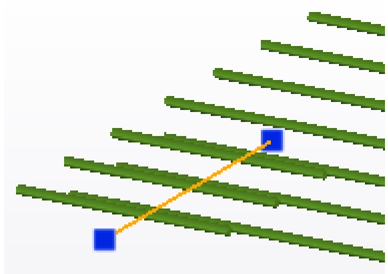
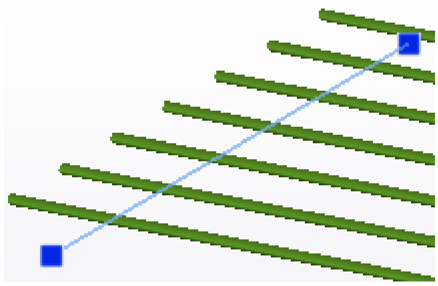
You can also create secondary guidelines for the rebar set. With a secondary guideline you can define a different spacing at the start and end of the rebar set bars, for example.

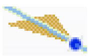


Modifiers are lines, or polylines that may have corner chamfers. Modifiers are projected to rebar set leg faces. Each modifier then only affects the rebar set bars that its projection touches.

NOTE When you work with rebar sets, ensure that the **Direct modification** switch is active.

To select entire rebar sets, or bar groups or individual bars within rebar sets in the model, you can use the three reinforcement selection switches . Tekla Structures then shows the existing modifiers that affect the selected rebar set bars, and the direct modification handles of the modifiers. Different modifiers have different colors as follows:

Modifier	Color	Example
Property modifier	Light green	
End detail modifier	Magenta	
Splitter	Orange	
Secondary guideline	Light blue	

An arrowhead symbol  close to the midpoint of each modifier indicates the direction (page 539) of the modifier, pointing from the start towards the end of the modifier.

When you select a modifier, Tekla Structures indicates the rebar set bars that are affected by the modifier, and shows the other, unaffected bars as semi-transparent.

You can modify the modifiers by using direct modification, or by changing their properties in the property pane or on the contextual toolbar. When you change the modifier properties, the properties of the rebar set bars change at the location defined by the modifier.

When you delete a modifier, the rebar set reverts to the state it had without the modifier.

Create a property modifier

Property modifiers are shown in light green.

1. Using the reinforcement selection switches , select the rebar set bars for which you want to create the modifier.

2. On the **Rebar set** contextual tab on the ribbon, click **Property modifier**.

3. Define how you want to place the modifier in the model.

   Click the **Picking mode** button on the contextual tab to cycle through the picking modes and to select the picking mode.

   The button indicates that you can pick one point, and the button indicates that you can pick multiple points. The button indicates that the modifier will be created for the selected bars only.

4. Depending on the selected picking mode, do one of the following:

   - Pick one point to create a single line modifier for the selected rebar set, bar group, or bars.
   - Pick two points to define the end points of a single line modifier. Then click the middle mouse button.
   - Pick multiple points to create a polyline modifier. Then click the middle mouse button to finish picking.

5. To end the command, press **Esc**.

6. To apply local modifications to the rebar set bars at the modifier location:

   a. Select the modifier.

   b. If you need to change the modifier geometry, drag the direct modification (page 107) handles.

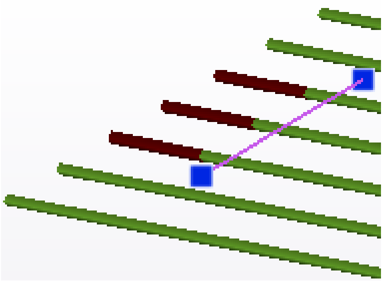
   c. Change the property modifier properties (page 1022) on the contextual toolbar or in the property pane.

   d. If you used the property pane, click **Modify** to save the changes.

Create an end detail modifier

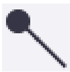
End detail modifiers affect the nearest end of the bar, so by moving the modifier you can change the bar end that it affects. End detail modifiers are shown in magenta.

Tekla Structures displays the threaded bar ends in dark red.



1. Using the reinforcement selection switches , select the rebar set bars for which you want to create the modifier.
2. On the **Rebar set** contextual tab on the ribbon, click  **End detail**.
3. Define how you want to place the modifier in the model.

   Click the **Picking mode** button on the contextual tab to cycle through the picking modes and to select the picking mode.

   The  button indicates that you can pick one point, and the  button indicates that you can pick multiple points. The  button indicates that the modifier will be created for the selected bars only.
4. Depending on the selected picking mode, do one of the following:
   - Pick one point to create a single line modifier for the selected rebar set, bar group, or bars.
   - Pick two points to define the end points of a single line modifier. Then click the middle mouse button.
   - Pick multiple points to create a polyline modifier. Then click the middle mouse button to finish picking.
5. To end the command, press **Esc**.
6. To apply local modifications to the rebar set bars at the modifier location:

   a. Select the modifier.

   b. If you need to change the modifier geometry, drag the direct modification (page 107) handles.

   c. Change the end detail modifier properties (page 1026) on the contextual toolbar or in the property pane.

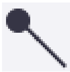
d. If you used the property pane, click **Modify** to save the changes.

Create a splitter

Splitters split bars and create lap or crank splices. Splitters are shown in orange.

1. Using the reinforcement selection switches , select the rebar set bars for which you want to create the modifier.
2. On the **Rebar set** contextual tab on the ribbon, click  **Splitter**.
3. Define how you want to place the splitter in the model.

   Click the **Picking mode** button on the contextual tab to cycle through the picking modes and to select the picking mode.

   The  button indicates that you can pick one point, and the  button indicates that you can pick multiple points. The  button indicates that the splitter will be created for the selected bars only.
4. Depending on the selected picking mode, do one of the following:
   - Pick one point to create a single line splitter for the selected rebar set, bar group, or bars.
   - Pick two points to define the end points of a single line splitter. Then click the middle mouse button.
   - Pick multiple points to create a polyline splitter. Then click the middle mouse button to finish picking.

   TIP If you want to create the splitter at a rounded distance from a bar end, and the dimension shown is measured from the other bar end, hold down **Shift** when you are placing the splitter in the model to switch the measuring point to the other bar end.

5. To end the command, press **Esc**.
6. To apply local modifications to the rebar set bars at the splitter location:

   a. Select the splitter.

   b. If you need to change the splitter geometry, drag the direct modification (page 107) handles.

   c. Change the splitter properties (page 1029) on the contextual toolbar or in the property pane.

   d. If you used the property pane, click **Modify** to save the changes.

Create a secondary guideline

You can create up to two secondary guidelines for a rebar set. Secondary guidelines are shown in light blue.

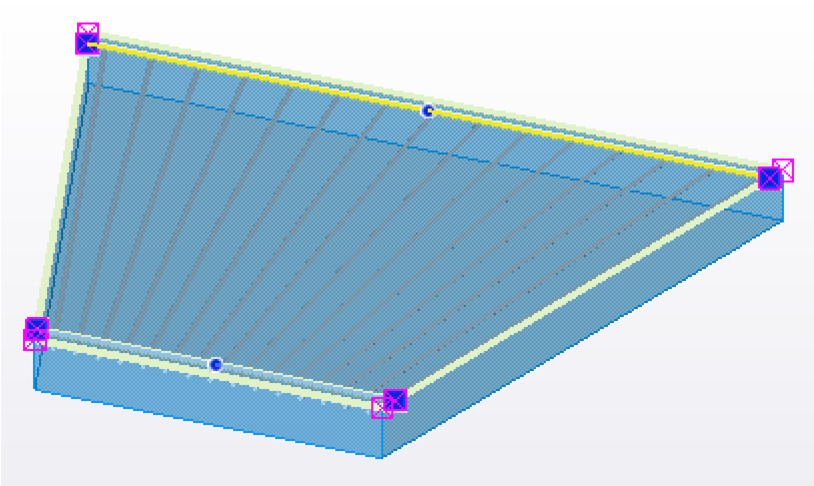
Note that if a rebar set was created using the **By guidelines** command, or if it otherwise has leg surfaces, you can only create one secondary guideline.

1. Select a rebar set.
2. On the **Rebar set** contextual tab on the ribbon, click **Secondary guideline**.
3. Define how you want to place the guideline in the model.

   The button on the contextual tab indicates that you can pick one point, and the button indicates that you can pick multiple points. Click the button to change the picking mode.
4. Depending on the picking mode, do one of the following:
   - Pick one point to create a single line guideline.
   - Pick two points to define the end points of a single line guideline. Then click the middle mouse button.
   - Pick multiple points to create a polyline guideline. Then click the middle mouse button to finish picking.
5. To create another secondary guideline, repeat steps 3 and 4.

   If there are already two secondary guidelines in a rebar set, the tooltip of the button changes to **Maximum number of secondary guidelines reached** and you cannot create more guidelines.
6. Press **Esc** to stop creating secondary guidelines.
7. If needed, select a secondary guideline and modify its geometry (page 107) and properties (page 1019).

For example, you can adjust the secondary guideline's length or spacing values.



See also Distribute bars in a rebar set (page 541).

8. To set a secondary guideline as the primary guideline, select the secondary guideline and click **Set as primary** on the contextual toolbar.

TIP Alternatively, you can create secondary guidelines the same way as you copy other modifiers (page 539): hold down **Ctrl** and drag the primary guideline.

Create a modifier by copying

You can copy rebar set modifiers.

1. Select a rebar set, bar group, or bar to show its modifiers.

   If you cannot see modifiers, ensure that the required modifier types are set visible (page 540).

2. Select the modifier that you want to copy.
3. Hold down **Ctrl** and drag the modifier to the desired location.

   Tekla Structures creates a new modifier when you release the mouse button.

4. Select the modifier to modify its geometry (page 107) and properties as needed.

Change the direction of a modifier

You can change the direction of rebar set modifiers, splitters, and guidelines.

1. Select a rebar set, bar group, or bar to show its modifiers.

   If you cannot see modifiers, ensure that the required modifier types are set visible (page 540).

2. Select the modifier whose direction you want to change.

3. On the contextual toolbar, click **Swap ends**.

   The arrowhead symbol close to the midpoint of the modifier changes direction, indicating the changed modifier direction.

Make a modifier follow edges

You can define that a rebar set modifier, splitter, or guideline attempts to follow the leg face edges that are located between the modifier end points. This is useful when you reinforce and detail curved concrete structures, for example.

1. Move the modifier end points to the leg face edges.

2. If there are cuts at the edges, add intermediate points to the modifier, and drag the handles to the corners of the cuts.

3. Double-click the modifier to open its properties in the property pane. Select **Yes** in the **Follow edges** list, and then click **Modify**.

   Alternatively, select the modifier and click on the contextual toolbar.

Show or hide rebar set modifiers

If you have many rebar set modifiers in your model, it may be useful to only show some of them when you select rebar set bars, and hide the ones that are not currently needed. You can show and hide modifiers according to their type.

For example, you can show the end detail modifiers only, and hide all property modifiers and splitters.

You can also show or hide primary and secondary guidelines.

1. Go to the **Rebar** tab and click **Visibility**.

2. Do any of the following:

   • Click **Guidelines** to switch the guidelines on or off.

   • Click **Property modifiers** to switch the property modifiers on or off.

   • Click **Splitters** to switch the splitters on or off.

   • Click **End detail modifiers** to switch the end detail modifiers on or off.

Alternatively, you can use the following keyboard shortcuts or advanced options:

• **Alt+2**, XS_REBARSET_SHOW_GUIDELINES

• **Alt+3**, XS_REBARSET_SHOW_PROPERTY_MODIFIERS

• **Alt+4**, XS_REBARSET_SHOW_SPLITTERS

- **Alt+5**, XS_REBARSET_SHOW_END_DETAIL_MODIFIERS

To show or hide the rebar set modifiers that have been created by using components, use the advanced option XS_REBARSET_SHOW_MODIFIERS_CREATED_BY_COMPONENTS. By default, this advanced option is set to `FALSE` and these modifiers are hidden when you select rebar set bars.

How to cut rebar sets

You can cut rebar sets automatically by the existing cuts in the concrete parts or manually by using the cutting commands on the **Edit** tab. You can modify cuts in rebar sets the same way as you modify cuts in parts in the model, by using direct modification.

You can use the following commands to create cuts:

- line cut (page 391)
- polygon cut (page 390)
- part cut (page 393)

Concrete cover settings are also applied to cuts, even on cut edges that are parallel to reinforcing bars.

Cut a rebar set using a cut in a concrete part

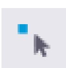
When you create rebar sets for concrete parts using the **Longitudinal**, **Crossing**, **By face**, and **By guidelines** commands, Tekla Structures automatically cuts the new rebar sets using the existing cuts in the concrete parts. If you add a new cut to a concrete part with a rebar set, the rebar set is not automatically cut. If you want to cut the rebar set as well, use the **Part cut** command, and use the new cut as the cutting part.

1. On the **Edit** tab, click **Part cut**.
2. Select the rebar set you want to cut.
3. Select the cut in the concrete part.

   Tekla Structures cuts the rebar set.

Modify a cut in a rebar set

You can modify cuts in rebar sets using direct modification. For example, you can make a cut in a rebar set of a different size or shape from a cut in a concrete part.

1. Ensure that the  **Direct modification** switch is active.
2. Select the cut in the rebar set.
3. Modify the cut using direct modification (page 107).

Distribute bars in a rebar set

Rebar sets may have different spacing values between the bars. The spacing of the bars is measured along the rebar set guidelines. You can modify the spacing values in model views or in the property pane, or by using property modifiers. You can also add, move, and delete single bars.

NOTE When you work with rebar sets, ensure that the **Direct modification** switch is active.

Modify the spacing properties

You can modify the spacing properties in model views and using the contextual toolbar, or in the property pane.

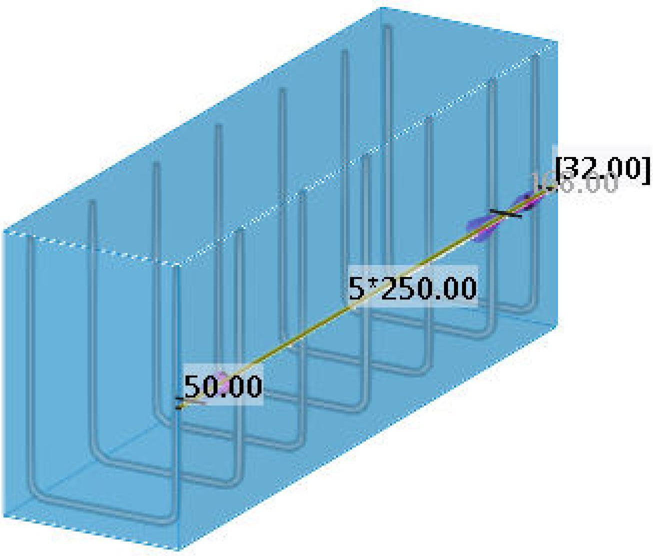
1. Select a rebar set.
2. If you want to modify spacing properties using a secondary guideline or property modifier, select the secondary guideline or property modifier.

   To modify secondary guideline spacings independently from the primary guideline, click on the contextual toolbar, or set **Inherit from primary** to **No** in the secondary guideline properties (page 1019).

   For the property modifier (page 1022), set **Modify distribution** to **Yes**.

3. On the contextual toolbar, click **Modify the spacings**.

   Tekla Structures shows the spacing values and the start and end offsets in the model.



For rebar sets and property modifiers, the values are shown on the primary guideline, for secondary guidelines on the secondary guideline.

When spacing modification is enabled, you cannot modify the geometry of the guidelines.

4. To specify how the bars are spaced, select an option for **Creation method** on the contextual toolbar.

5. To modify a spacing value, an offset, or the number of bars, click the value in a model view, enter a new value, and then press **Enter**.

   When **Creation method** is set to **By exact spacings**, you can list multiple spacing values (separated with space characters) and/or use multiplication to repeat the same spacing, for example, `100 200 5*300`.

Alternatively, you can modify the spacing properties in the property pane.

Add, move, and delete single bars

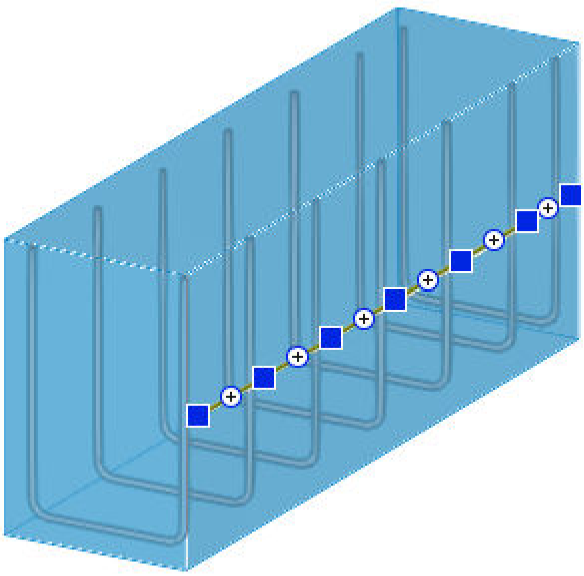
1. Select a rebar set.

2. If you want to move bars along a secondary guideline, select the secondary guideline.

   Click on the contextual toolbar, or set **Inherit from primary** to **No** in the secondary guideline properties.

3. On the contextual toolbar, click **Move, add, delete reinforcement**.

   Tekla Structures displays a handle for each bar on the guideline, and symbols between the bars.



4. Do any of the following:

- To add a bar between two existing bars, click .
- To move a bar, select the handle of the bar and drag it to a new location.

  You can also use the keyboard to enter a numeric location (page 87). Then press **Enter** to confirm.
- To delete a bar, select the handle of the bar and press **Delete**.

You can also use the **Exclude** list in the rebar set (page 1016), secondary guideline (page 1019), or property modifier (page 1022) properties to leave out the first and/or last rebar set bars.

Modify a single reinforcing bar, bar group, or mesh

You can modify a reinforcement by using direct modification. You can either modify the reinforcement simply by dragging handles, or select a command from the contextual toolbar.

NOTE Direct modification does not work for the following reinforcement types:

- Circular (page 507), curved (page 505), or 3D reinforcing bars
- Reinforcement strand patterns (page 517)

If you have created the reinforcement using a component, you need to explode the component before using direct modification.

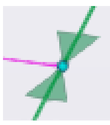
Before you start:

- Ensure that the **Direct modification** switch is active.
- Select the reinforcement.

  Tekla Structures displays the handles that you can use to modify the reinforcement, and a toolbar icon . Click the icon to open the toolbar and to select the appropriate command. The available commands depend on the type of the reinforcement you are modifying.

To modify single reinforcing bars, reinforcing bar groups, or reinforcement meshes:

To	Do this	Command available for
Change the cover thickness of a reinforcing bar	Drag a line handle to the desired location. 	Reinforcing bars, reinforcing bar groups, reinforcement meshes
Add polygon points to a reinforcing bar	Drag a midpoint handle to the desired location.	Reinforcing bars, reinforcing bar groups, polygonal and bent reinforcement meshes
Add points to the start or end of a reinforcing bar	1. Click the start or end reference point of the reinforcing bar . 2. Click the **Add new point** button  on the toolbar. 3. Pick a location for the new start or end point.	Reinforcing bars, reinforcing bar groups

To	Do this	Command available for
Remove points from a reinforcing bar	1. Select one or more reference points. 2. Press **Delete**.	Reinforcing bars, reinforcing bar groups, polygonal and bent reinforcement meshes
Add hooks	1. Click the start or end point of the reinforcing bar . A toolbar for hook properties appears. 2. Select the desired shape for the hook. 3. If you selected **Custom hook**, enter the angle, radius, and length for the hook and click .	Reinforcing bars, reinforcing bar groups
Change the bending radius of a reinforcing bar	1. Click the **Change bending radius** button on the toolbar. 2. Enter a value in the box next to the **Change bending radius** button and press **Enter**.	Reinforcing bars, reinforcing bar groups
Change the diameter of a reinforcing bar	1. Click the **Change diameter** button on the toolbar. 2. Select a value from the list next to the **Change diameter** button.	Reinforcing bars, reinforcing bar groups, reinforcement meshes
Modify the spacings by adjusting the range	1. Click the **Modify spacings** button on the toolbar. 2. Drag a handle  to the desired location.	Reinforcing bar groups, reinforcement meshes
Modify the spacings by splitting the range in two	1. Click the **Modify spacings** button on the toolbar.	Reinforcing bar groups, reinforcement meshes

To	Do this	Command available for
	2. Drag a midpoint handle to the desired location and release the handle. Tekla Structures creates a new reinforcing bar and the range is split in two. The spacing in the two new ranges is as close as possible to the original spacing. 3. If needed, change the number of spaces or the spacing value. Click the midpoint handle and enter the desired values in the boxes on the toolbar and press **Enter**.	
Move, add, or remove reinforcement	1. Click the **Move, add, delete reinforcement** button on the toolbar. Tekla Structures displays the line handles for each reinforcing bar. 2. Do one of the following: • To move a reinforcing bar, highlight it and drag it to the desired location. • To add a reinforcing bar between two reinforcing bars, click . • To delete reinforcing bars, select them and press **Delete**.	Reinforcing bar groups, reinforcement meshes

See also

Use handles to modify a reinforcement (page 554)

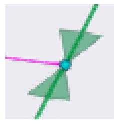
Use adaptivity to modify a reinforcement (page 561)

Check the validity of reinforcement geometry (page 564)

Distribute bars in a reinforcing bar group

You can select how reinforcing bars are distributed in a bar group by modifying the bar spacings.

To modify the spacings of bars in a reinforcing bar group, do one of the following:

To	Do this
Modify the spacings using direct modification (page 544)	1. Ensure that the **Direct modification** switch  is active. 2. Select a reinforcing bar group. 3. On the contextual toolbar, click the **Modify the spacings** button . 4. Drag a handle  to the desired location.
Modify the spacings using the **Rebar group** properties	1. Select a reinforcing bar group. 2. Double-click the reinforcement to open the **Rebar group** properties. 3. In the **Distribution** section, select a spacing option from the **Creation method** list. 4. Enter the required values. 5. Click **Modify**.

Spacing options in the **Creation method** list:

Option	Description	Example
Equal distribution by number of reinforcing bars	Enter the number of reinforcing bars. Tekla Structures divides the available distance by the number of bars. Enter the number of bars in the **Number of reinforcing bars** box.	
Equal distribution by target spacing value	Enter a spacing value. Tekla Structures aims the spacing value as close as possible to the value in the **Target spacing value** box.	

Option	Description	Example
By exact spacing value with flexible first	Enter the spacing value in the **Exact spacing value** box. Creates fixed, regular spaces between the bars. The first space adjusts to even out the bar distribution. If the first space is less than 10% of the exact spacing value, Tekla Structures removes one bar.	
By exact spacing value with flexible last space	Enter the spacing value in the **Exact spacing value** box. Creates fixed, regular spaces between the bars. The last space adjusts to even out bar distribution.	
By exact spacing value with flexible middle space	Enter the spacing value in the **Exact spacing value** box. Creates fixed, regular spaces between the bars. The middle space adjusts to even out bar distribution. If there are an odd number of bars (two middle spaces), the other middle space adjusts to even out bar distribution.	
By exact spacing value with flexible first and last space	Enter the spacing value in the **Exact spacing value** box. Creates fixed, regular spaces between the bars. Both the first and last spaces adjust to even out bar distribution.	
By exact spacings	Enter the spacing values manually in the **Exact spacing values** box. Use multiplication to repeat spacings, for example, `5*200` to create five spaces of `200`.	

See also

Create a reinforcing bar group (page 496)

Create a reinforcing bar group using Rebar shape catalog (page 498)

Modify a single reinforcing bar, bar group, or mesh (page 544)

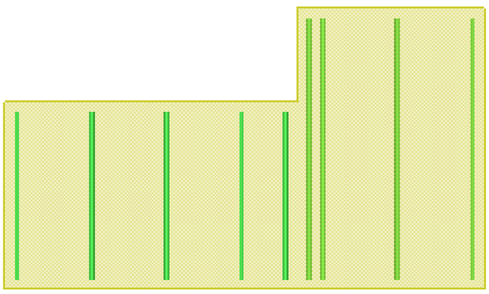
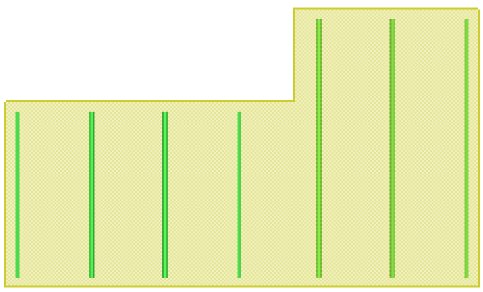
Delete bars from a reinforcing bar group

You may occasionally need to delete or exclude specific reinforcing bars. For example, when several reinforced areas intersect, causing reinforcing bars to overlap, or when you want to start bar distribution at a specific distance from the end of a part.

To delete reinforcing bars from a group, do one of the following:

To	Do this
Delete bars using direct modification (page 544)	1. Ensure that the **Direct modification** switch is active. 2. Select a reinforcing bar group. 3. On the contextual toolbar, click the **Move, add, delete reinforcement** button. 4. Select the bars to be deleted and press **Delete**.
Delete bars using the **Rebar group** properties	1. Select a reinforcing bar group. 2. Double-click the reinforcement to open the **Rebar group** properties. 3. In the **Distribution** section, select an option from the **Exclude** list. 4. Click **Modify**.

Examples when you have used the **Exclude** options:

Before excluding bars	After excluding bars
Two reinforcing bar groups have been added to a concrete beam: • one bar group with flexible last space • one bar group with flexible first space 	Two reinforcing bar groups without the excluded bars: • one bar group with the last bar excluded • one bar group with the first bar excluded 

See also

Create a reinforcing bar group (page 496)

Create a reinforcing bar group using Rebar shape catalog (page 498)

Modify a single reinforcing bar, bar group, or mesh (page 544)

Ungroup a reinforcement

You can ungroup reinforcing bar groups and reinforcement meshes. Only reinforcement where each reinforcing bar is in one plane can be ungrouped.

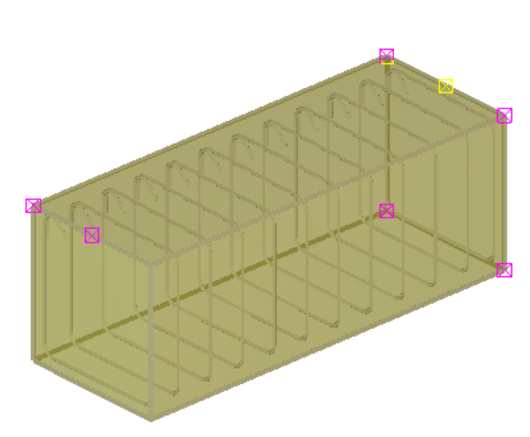
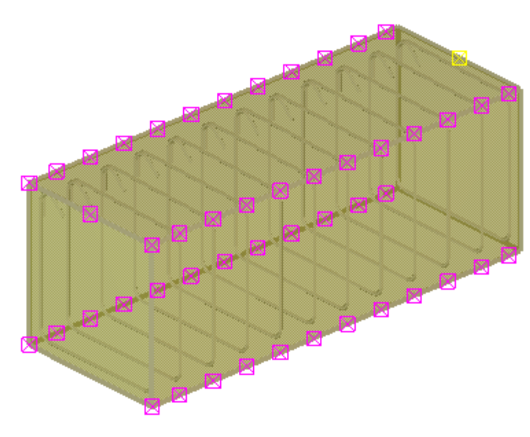
NOTE You cannot ungroup circular (page 507) or curved (page 505) reinforcing bar groups.

---

1. On the **Rebar** tab, click **Edit** --> **Group**.
2. Select one of the reinforcing bars in a reinforcing bar group or in a reinforcement mesh.

   The reinforcing bar group is replaced with single reinforcing bars. The single bars get the same properties and offsets as the group.

   If you ungroup a reinforcement mesh, the offsets for single bars are zero.

Before ungrouping	After ungrouping
	

See also

Modify reinforcement (page 520)

Create a reinforcing bar group using Rebar shape catalog (page 498)

Create a reinforcing bar group (page 496)

Create a reinforcement mesh (page 511)

Group reinforcement

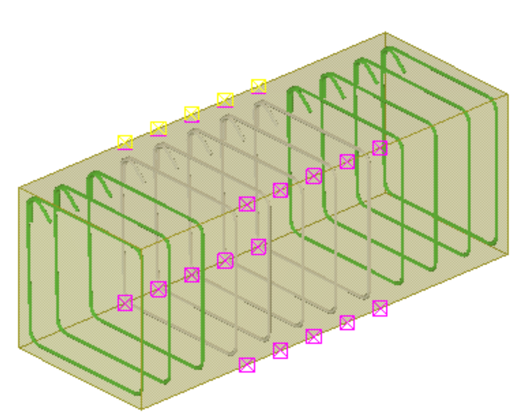
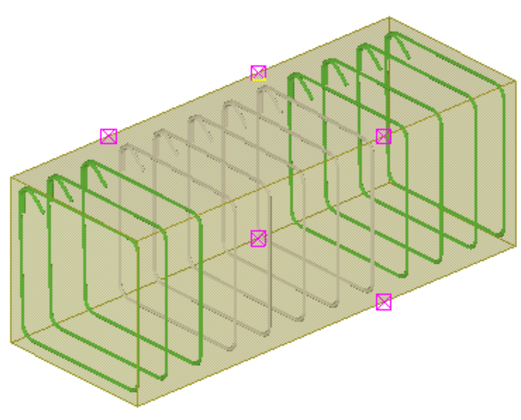
You can group single reinforcing bars and reinforcing bar groups. Only reinforcement where each reinforcing bar is in one plane can be grouped. All groups are created with exact spacings. Single reinforcing bars need to have the same bending shape.

NOTE You cannot create circular (page 507) or curved (page 505) reinforcing bar groups by grouping.

1. On the **Rebar** tab, click **Edit** --> **Group**.
2. Select all the reinforcing bars or reinforcing bar groups that you want to group.
3. Click the middle mouse button.
4. Select one reinforcing bar or reinforcing bar group to copy the properties from.

   The new group gets the same properties as the selected reinforcing bar.

   NOTE The reinforcing bar or reinforcing bar group that you copy the properties from is also added to the group. This means, for example, that you cannot copy properties from a separate reinforcing bar group which you do not want to include in your new reinforcing bar group.

Before grouping	After grouping
	

See also

Modify reinforcement (page 520)

Create a reinforcing bar group using Rebar shape catalog (page 498)

Create a reinforcing bar group (page 496)

Create a single reinforcing bar (page 496)

Combine two reinforcing bars or reinforcing bar groups into one

You can combine two single reinforcing bars or reinforcing bar groups into one. Reinforcing bars can be combined if their end points are connected, or the bars are parallel and close to each other. However, in certain cases it is possible to combine bars or groups that are not connected nor parallel. The combined reinforcement gets the same properties as the bar selected first.

NOTE You cannot combine **Tapered N** reinforcing bar groups.

1. On the **Edit** tab, click **Combine**.
2. Select the first single bar or bar group to be combined.
3. Select the second single bar or bar group to be combined.

   Tekla Structures combines the reinforcing bar groups or bars into one.

See also

Create a reinforcing bar group using Rebar shape catalog (page 498)

Create a reinforcing bar group (page 496)

Create a single reinforcing bar (page 496)

Modify reinforcement (page 520)

Split a reinforcing bar group

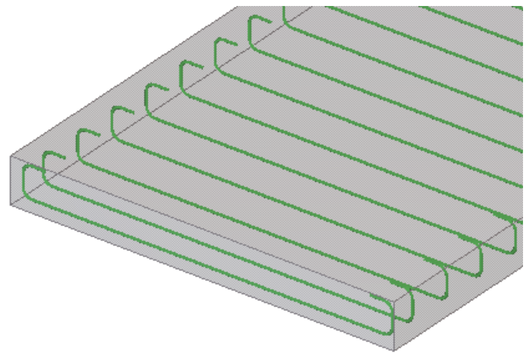
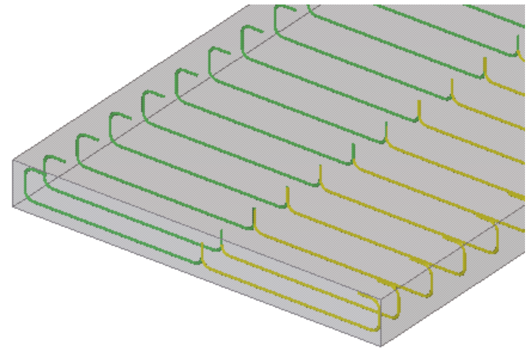
You can split normal and tapered reinforcing bar groups into two groups. You can also split single reinforcing bars into two.

1. On the **Edit** tab, click **Split**.
2. Select a reinforcing bar group.
3. Pick two points to indicate where to split the group.

   Tekla Structures splits the reinforcing bar group.

   NOTE You cannot split reinforcing bar groups diagonally.

   Once split, each new reinforcing bar group retains the properties of the original group. For example, if the bars in the original group had hooks at both ends, bars in the new groups also have hooks at both ends. Modify the properties of the new groups, if needed.

Before splitting	After splitting
	

See also

Create a reinforcing bar group using Rebar shape catalog (page 498)

Create a reinforcing bar group (page 496)

Create a single reinforcing bar (page 496)

Modify a single reinforcing bar, bar group, or mesh (page 544)

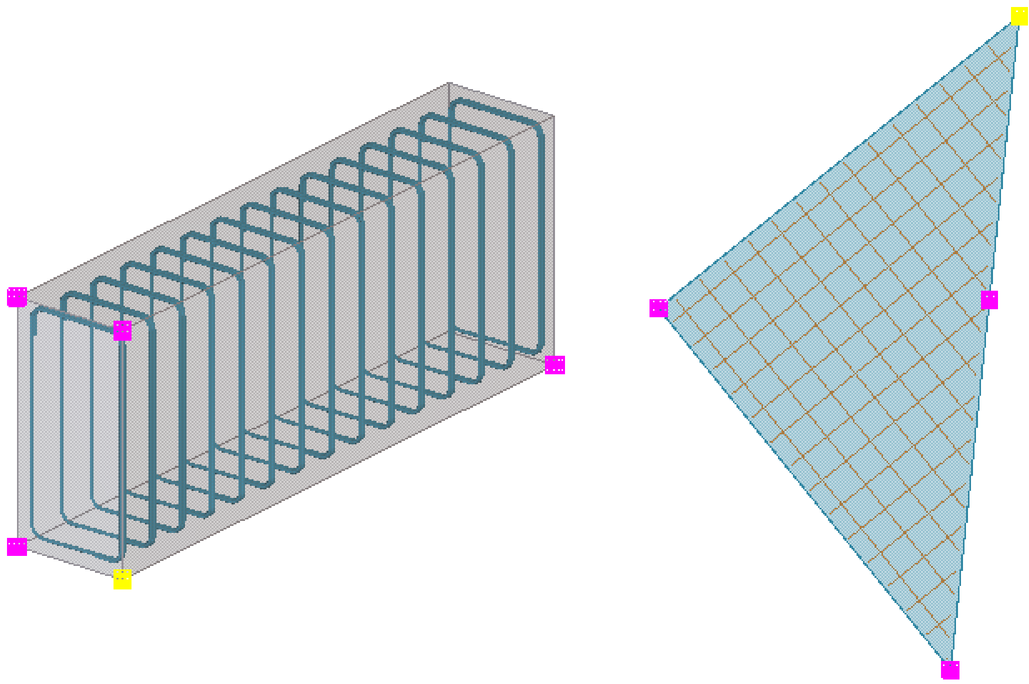
Use handles to modify a reinforcement

If you do not want to use direct modification to modify a reinforcement, you can use, for example, the reinforcement handles to modify the reinforcement.

Tekla Structures uses handles to indicate:

- The ends and corners of a reinforcing bar.
- The distribution length of a bar group.
- The corners and main bar direction of a mesh.

When you select a reinforcement, Tekla Structures highlights the handles. The handle of the first end point is yellow, the rest are magenta.



1. Select the reinforcement.

   Tekla Structures highlights the handles.
2. Click one of the handles to select it.
3. Move the handle like any other object in Tekla Structures.

   For example, if **Drag and drop** is active, just drag the handle to a new position.

NOTE If you want to use the reinforcement handles, ensure that the **Direct modification** switch  is not active. If the switch is active and direct modification (page 544) is on, Tekla Structures displays direct modification handles for the reference points, ends, legs, and leg midpoints of the selected reinforcement. These handles are blue.

See also

Check the validity of reinforcement geometry (page 564)

Add hooks to reinforcing bars

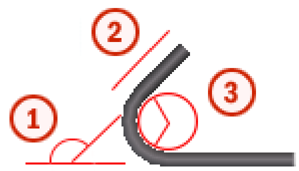
You can add hooks to the ends of reinforcing bars for anchoring purposes.

NOTE Hooks are intended to be used only for anchoring purposes. Do not use hooks as a method to model other reinforcing bar geometry, because it may cause problems with visibility in drawings, in adaptivity, and in bar bending shape recognition.

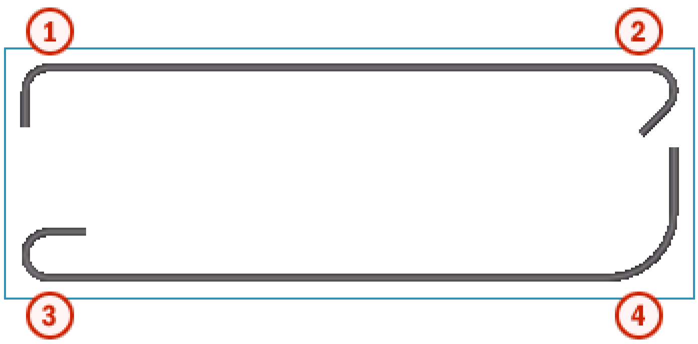
To add hooks to reinforcing bars, do one of the following:

To	Do this
Add hooks using direct modification (page 544)	1. Ensure that the **Direct modification** switch is active. 2. Select a single reinforcing bar or a reinforcing bar group. 3. Click the start or end point of the reinforcing bar . A toolbar for hook properties appears. 4. Select the desired shape for the hook. 5. If you select **Custom hook**, enter the angle, radius, and length for the hook. Click .
Add hooks using the **Single rebar** or **Rebar group** properties	1. Select a single reinforcing bar or a reinforcing bar group. 2. Double-click the reinforcement to open its properties. 3. In the **Hooks** section, select a hook type for the bar start and/or end from the **Hook type** list. 4. If you select **Custom hook**, enter the angle, radius and length for the hook. 5. Click **Modify**.
Add hooks to rebar sets using end detail modifiers	See Modify a rebar set locally using modifiers (page 533).

For custom hooks you need to enter the hook information:

Option	Description	
Angle	Enter a value between -180 and +180 degrees.	 1. Angle 2. Length 3. Radius
Radius	Enter the internal bending radius of the hook. Use the same radius for the hook and for the reinforcing bar. If the hook and the reinforcing bar have different radiuses, Tekla Structures does not recognize the shape of the bar.	
Length	Enter the length of the straight part. If the length is set to zero, no hooks are created.	

Hook examples



	Description
1	Standard 90-degree hook
2	Standard 135-degree hook
3	Standard 180-degree hook
4	Custom hook

If you select a standard hook, the **Angle**, **Radius**, and **Length** use predefined dimensions.

The `rebar_database.inp` file contains the predefined minimum bending radius and minimum hook length for all standard hooks.

See also

Create a reinforcing bar group (page 496)

Create a reinforcing bar group using Rebar shape catalog (page 498)

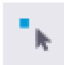
Modify a single reinforcing bar, bar group, or mesh (page 544)

Define the reinforcement cover thickness

Reinforcing bars need a concrete cover to protect them against harmful elements, such as the weather and fire. When you create single bars, Tekla Structures uses the thickness of concrete cover to determine the position of the bar.

To define the reinforcement cover thickness, do one of the following:

<table>
<tr><th>To</th><th>Do this</th></tr>
<tr><td>Change the cover thickness using direct modification (page 544)</td><td>

Ensure that the  Direct modification switch is active.
Select a single reinforcing bar, a reinforcing bar group, or a mesh.
Drag a line handle to the desired location.

</td></tr>
<tr><td>Change the cover thickness using the Single rebar, Rebar group, or Rebar mesh properties</td><td>

Select a single reinforcing bar, a reinforcing bar group, or a mesh.
Double-click the reinforcement to open its properties.
Define the reinforcing bar cover thickness in the Cover thickness section.

The cover thickness can be defined in three directions:

On plane, that is, the distance from part’s bottom, top, and side faces to the bar.

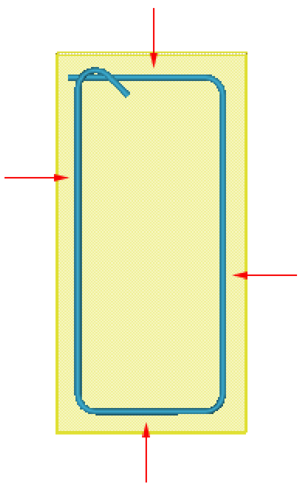
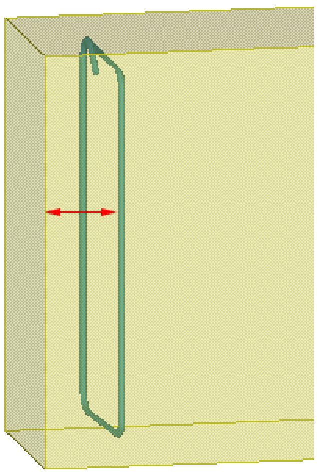
You can enter several values. Enter the values in the order you pick the points to create the bar. If you enter less values than there are bar legs, Tekla Structures uses the last value for the remaining legs.
From plane, that is, the distance from the end face of the part to the bar.

If the reinforcing bar is outside the part, enter a negative value in the On plane and/or the From plane boxes.

</td></tr>
</table>

To	Do this
	• In the longitudinal direction of the bar, that is, start and end. To define the length of an ultimate leg of a bar, use the **Leg length** option and the **Snap to nearest points** switch. Then pick anywhere on a part edge or line to indicate the direction for the bar leg. 4. Click **Modify**.
Change the default cover thickness of the rebar sets in a model	1. On the **File** menu, click **Settings** --> **Options** to open the **Options** dialog box. 2. Go to the **Rebar set** settings and to the **Covers and locations** tab. 3. Modify the settings and click **OK**. You can define the default cover thickness values in the global and/or parts' local coordinate systems and at different part faces. In each concrete part's properties, you can then select whether to use the global or local cover thickness values. 4. To apply the changes to all or selected existing rebar sets in the model, go to the **Rebar** tab on the ribbon and click **More** --> **Regenerate**.
Change the cover thickness of the rebar sets in an individual concrete part or for a part type	1. Double-click a concrete part to access its properties. 2. Go to the **Concrete covers for rebar sets** section. Alternatively, you can click **User-defined attributes** and go to the **Rebar set** tab. 3. Select the coordinate system: **Global** or part's **Local**. If you select the empty option, Tekla Structures uses the global cover thickness values from the **Options** dialog box. 4. To override the default values from the **Options** dialog box, define the cover thickness at each required part face. • With the **Global** coordinate system, you can enter the top, bottom, and side face values. • With the **Local** coordinate system, you can enter the top, bottom, front, back, start, and end face values.

To	Do this
	5. Click **Modify** to update the selected part's properties. 6. To save the cover thickness values for later use for this part type, update the standard file or create a property file.
Change the concrete cover thickness of the rebar sets at a concrete part face or pour object face	1. Add a surface (page 414) to the object face at which you want to change the concrete cover. 2. Double-click the surface to modify its properties in the property pane. 3. In the **Type** list, select **Concrete cover**. 4. In the **Rebar set** section, enter the concrete cover thickness in the **Concrete cover** box. 5. Click **Modify** to apply the changes. **NOTE:** If you add surfaces to pour object faces, update the rebar sets each time you reopen the model. On the **Rebar** tab on the ribbon, click **More** --> **Regenerate**.

On plane	From plane	Start and end
		

See also

Create a reinforcing bar group (page 496)

Create a reinforcing bar group using Rebar shape catalog (page 498)

Create a rebar set (page 470)

Modify reinforcement (page 520)

Select the definition for a reinforcement

When you create (page 469) or modify (page 520) reinforcement, such as a reinforcing bar group, rebar set, or reinforcement component, you can select a definition for the bars from the rebar catalog. Selecting the definition automatically sets some basic reinforcement properties, such as grade, size, and bending radius.

1. Open the properties of the reinforcement.
2. In the property pane or component dialog box, click the **...** button next to the **Size** box.

   The **Select rebar** dialog box appears.
3. If needed, organize the rebar catalog view.

   For example, you can filter rebar definitions, or group and sort them differently.
4. Select a rebar definition from the list.
5. Click **OK** to close the **Select rebar** dialog box.
6. Click **Modify** in the reinforcement properties to apply the changes.

Use adaptivity to modify a reinforcement

Reinforcement follows the shape of the part also when the reinforcement handles are located on the face or edge of the part.

The following types of adaptivity are available:

- Fixed adaptivity: handles retain their absolute distances to the nearest part faces.
- Relative adaptivity: handles retain their relative distances to the nearest part faces in relation to the part's overall size.

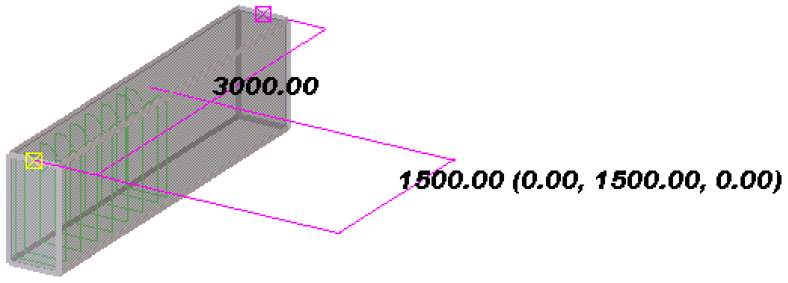
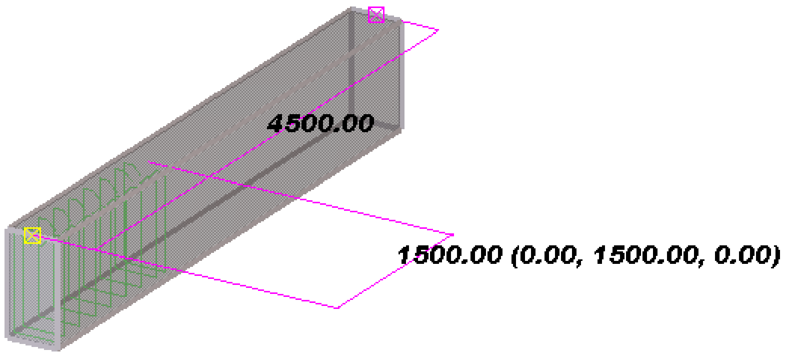
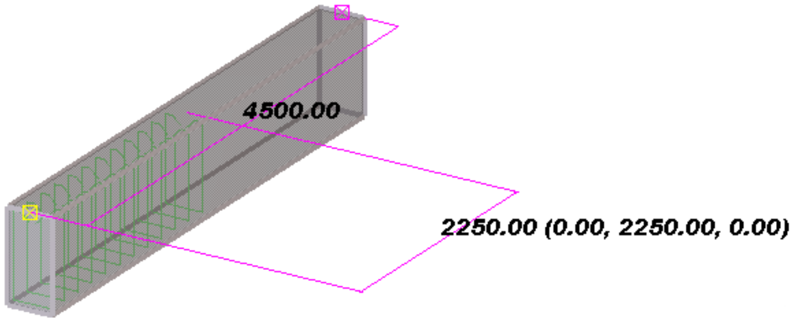
1. Select a reinforcement.
2. Right-click and select **Adaptivity** and then one of the adaptivity options from the pop-up menu.

   When a part is modified, Tekla Structures handles the reinforcement according to the adaptivity selection.

   TIP To modify the general adaptivity settings, click **File menu** --> **Settings** --> **Options** --> **General**.

   You can also modify the adaptivity settings for each part separately. These modifications override the general settings in the **Options** dialog box.

Adaptivity examples

Reinforcing bars in their original position	
Fixed adaptivity	
Relative adaptivity	

See also

Check the validity of reinforcement geometry (page 564)

Modify a single reinforcing bar, bar group, or mesh (page 544)

Attach a reinforcement object to a concrete part

When you create a reinforcement object, such as a bar group, rebar set, or mesh, Tekla Structures attaches it to the concrete part that you have selected to reinforce. Later in some cases, you might need to attach the reinforcement object to another concrete part manually. The attached reinforcement objects follow the part if the part is moved, copied, or deleted.

NOTE Only attached reinforcement objects are shown in cast unit drawings and reports with the part.

1. If you want to manually attach only certain single bars in a rebar set to a part, create a property modifier (page 535) for the bars that you want to attach.
2. Select the reinforcement object that you want to attach.

   To attach single rebar set bars, first select the rebar set, then the property modifier that affects the bars that you want to attach.
3. Right-click and select **Attach to part**.
4. Select the part to which you want to attach the reinforcement object.

   The reinforcement object is attached to the part.

Note that if you have manually attached single bars in rebar sets to parts in Tekla Structures 2022 SP2 or older versions, using property modifiers for manual attachment does not work with these bars. To revert all manually attached bars in rebar sets to automatic attachment (page 568), select all the rebar sets in the model and use the **Detach from part** command.

Detach a reinforcement object from a concrete part

If needed, you can detach a reinforcement object from a concrete part.

1. Select the reinforcement object that you want to detach.

   To detach single rebar set bars, first select the rebar set, then the property modifier that affects the bars that you want to detach.
2. Right-click and select **Detach from part**.

   The reinforcement object is detached from the part.

Note that if you detach a rebar set or a property modifier, the bars in the rebar set are reattached automatically if they are inside a concrete part.

TIP Alternatively, you can use the ribbon when you manually attach or detach reinforcement objects.

1. On the **Rebar** tab, click **More** --> **Attach to part** or **More** --> **Detach from part**.
2. Select the reinforcement object that you want to attach or detach.
3. If you are attaching the reinforcement object, select the part to which you want to attach the reinforcement object.

See also

How Tekla Structures automatically attaches rebar set bars to concrete parts (page 568)

Modify reinforcement (page 520)

Check the validity of reinforcement geometry

Reinforcement creation or modification can result in invalid reinforcement geometry. For example, too big bending radius can cause invalid reinforcement geometry. If a model contains a reinforcement with invalid geometry, the reinforcement is not shown in the drawings. The reinforcement becomes visible and the drawings are updated when the geometry is corrected.

If a rebar set leg face or leg surface (page 525) is shown in red in the model, it means that the geometry of some bars in the rebar set is not valid. Check and fix the properties of the rebar set (page 1016) (for example, the bending radius) or the leg face (page 1020) or leg surface (page 1021).

NOTE Reinforcement geometry validity check does not work with circular (page 507) or curved (page 505) reinforcing bar groups.

---

1. On the **File** menu, click **Diagnose & Repair** and in the **Model** area, click **Diagnose model**.
2. Check the results.

   If there are inconsistencies in the geometry, Tekla Structures displays a warning message, and draws a thin line between the reinforcement handles to show the invalid geometry.

   You can correct the reinforcement geometry by selecting the line and modifying the reinforcement properties.

See also

Modify a single reinforcing bar, bar group, or mesh (page 544)

Modify a rebar set (page 521)

Split and splice a reinforcement

You can split long reinforcing bars and bar groups that exceed the stock length, and create splices in split locations.

Use the **Automatic splicing tool** macro to split and splice reinforcement that exceeds the stock length. You can first check the length of the reinforcing bars in the model according to the manufacturer. You can then define the portion of the reinforcement to be split and spliced in the same cross section, and the location, symmetry, type, and length of the splices.

NOTE **Automatic splicing tool** does not work with rebar sets (page 470). To split bars in rebar sets, use splitters instead.

If the reinforcement that is going to be split belongs to a rebar assembly (page 614), the newly split bars are also added to the rebar assembly.

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 **Automatic splicing tool** to start the macro.
4. In the **Automatic Splicing Tool** dialog box:
    a. 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 define the length information in the `AutomaticSplicingTool_Manufacturers.dat` file. You can copy the default file from `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\system`, edit it, and save it to your project or firm folder.

    b. For the bar grades and sizes that are not listed in the `AutomaticSplicingTool_Manufacturers.dat` file, use the **Maximum length for unspecified stock** box to define the maximum reinforcing bar length after which the bars are split and spliced.

    c. To check if the length of the reinforcing bars exceeds the maximum length, use one of the buttons next to **Perform check to**:

       - To check all reinforcement in the model, click **All**.
       - To check specific reinforcement, first select the reinforcement or cast units in the model, and then click **Selected**.

       Tekla Structures lists the reinforcing bars that are longer than the maximum length under **Longer rebars** on the right side of the dialog box.

       When you select a row in the **Longer rebars** list, Tekla Structures highlights the corresponding reinforcement in the model.

    d. Define which proportion of the reinforcement can be spliced in the same cross section.

    e. Define the symmetry that is applied when the reinforcing bars are spliced.

    f. Define the offset of the splice center point.

g. Define the minimum longitudinal distance between two parallel bar splices.

h. Select the splice type.

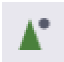
   You can create lap splices, coupler splices, or welded splices.

i. For lap splices, define the default lap length as a distance or in relation to the nominal bar diameter.

   This value will be used if there is no lap length defined for a bar grade and size in the `AutomaticSplicingTool_Manufacturers.dat` file.

j. For lap slices, define whether the lapping bars are on top of each other or parallel to each other.

k. To split and splice reinforcement, use one of the buttons next to **Perform split and splice to**:

   - To splice all reinforcement in the model, click **All**.
   - To splice specific reinforcement, first select the reinforcement in the **Longer rebars** list, or in the model using the **Select objects in components** switch , and then click **Selected**.

See also

Create a reinforcement splice (page 519)

Assign running numbers to reinforcement

You can assign running numbers to reinforcement in cast units. You can then use the running numbers in addition to or instead of the position numbers in reinforcement marks and tables in drawings, and in reports.

Use the **Rebar sequence numbering** macro to assign cast unit specific running numbers (1, 2, 3...) to the reinforcement in the model. Running numbers are unique inside each cast unit. The macro does the following:

- Updates the position numbers of the modified model objects using the **Number modified objects** command in **Drawings & reports** --> **Perform numbering** .
- Assigns running numbers to the reinforcing bars, reinforcing bar groups, and reinforcement meshes in the model.
- Saves a running number as the user-defined attribute **Rebar sequence number** (`REBAR_SEQ_NO`) of each bar, group, or mesh.

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 **Rebar sequence numbering** to start the macro.
4. To show the running numbers in drawings and reports, use the user-defined attribute `REBAR_SEQ_NO`.

See also

Number reinforcement (page 724)

Classify reinforcement to layers

To be able to show in drawings what is the order of different reinforcement layers near a surface of a concrete part, you need to classify reinforcement in the model. You can do this by using the **Rebar classification** macro.

Rebar classification classifies the reinforcing bars and reinforcement meshes by their order of depth in concrete slabs and panels. The reinforcing bars and meshes get an attribute indicating the layer where they are placed inside the concrete part.

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 **Rebar classification** to start the macro.
4. In the **Rebar classification** dialog box:
    a. Enter the prefixes that you want to use for the reinforcement layers near the top, bottom, front, and back surfaces of the concrete parts.
    b. Select whether you want to classify **All objects** or **Selected objects**.

       If you select **Selected objects**, select the reinforcement or the concrete parts containing the reinforcement that you want to classify.
    c. Click **Preview** to view the properties of the reinforcement in each layer.

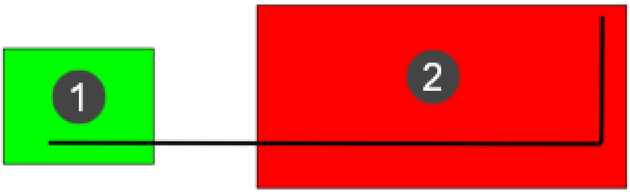
       The layers are named using the relevant surface prefix and numbered starting from the surface.
    d. If you do not want to classify a reinforcement, select it from the list and click **Delete item**.
    e. To save the classification attributes of the reinforcement, do one of the following:
        - Click **Modify** to also keep the **Rebar classification** dialog box open.
        - Click **OK** to also close the **Rebar classification** dialog box.
5. In a drawing, run the **Rebar layering marker** macro to create layer-specific markers for reinforcement.

How Tekla Structures automatically attaches rebar set bars to concrete parts

When you create or modify a rebar set, Tekla Structures automatically attaches each bar in the rebar set to a concrete part. This concrete part is the *parent part* of the rebar set bar.

Depending on the cast unit type, bottom level, and volume of the concrete parts, and the bar length, location, and orientation, and the number of bars in the rebar set, Tekla Structures searches for and chooses the parent part for each of the bars as follows:

- Each bar that is at least partly inside only one concrete part is attached to that part.
- Each bar that is inside both precast and cast-in-place concrete parts is attached to a precast part.

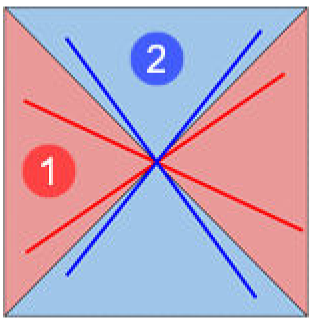


(1) = precast part, **(2)** = cast-in-place part

In this and the following images, the parent part is shown in green.

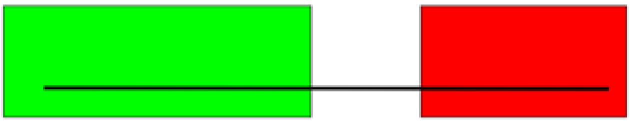
Then, for each bar that is inside two or more concrete parts, either precast parts or cast-in-place parts, but not both:

- A bar is considered horizontal if the angle between the horizontal plane and the bar plane that contains the vertices of the bar is less than 45 degrees, or for a straight bar, if the angle of the bar direction to the horizontal plane is less than 45 degrees. Otherwise the bar is vertical.



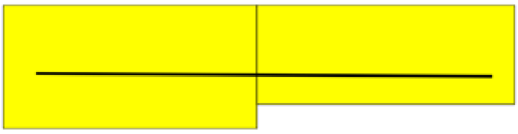
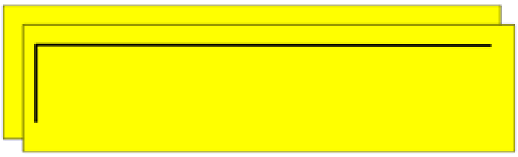
(1) = horizontal bars, **(2)** = vertical bars

- If the bar is placed horizontally, it is attached to the part that contains the longest bar length.



- If the bar is placed vertically and if the bar is fully inside one part and partially inside other parts, it is attached to the part that contains the whole bar.
- If the bar is placed vertically and partly inside several parts, it is attached to the part whose bottom face has the lowest global z coordinate.

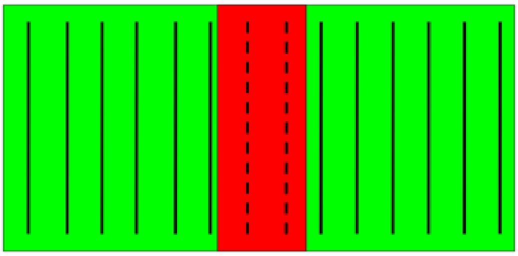
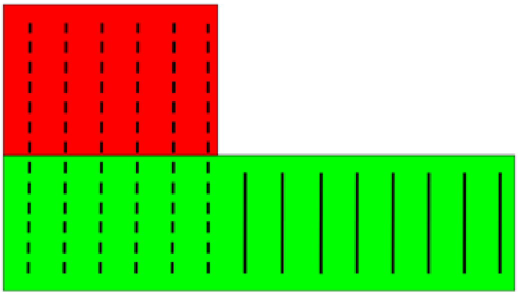
Sometimes several parts might match one of the above criteria and could be the parent part of a rebar set bar. For example:

	A horizontal bar is equally in two parts.
	Two parts contain some of a vertical bar and their bottom levels are at equal height.
	Two parts contain all of a vertical bar but are at different heights.

In cases like these, Tekla Structures chooses the parent part for each bar from among the matching parts as follows:

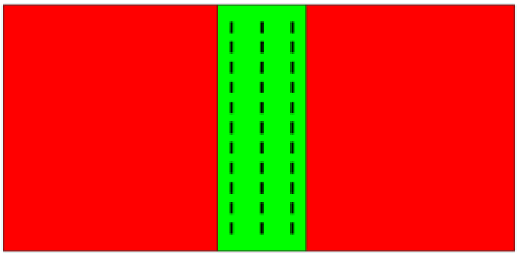
- If one of the parts is the more common parent part in the rebar set than the other parts, Tekla Structures attaches the bar to that more common parent part.

  For example:

	In this plan or elevation view, two parts overlap and the dashed bars are entirely in both parts. The dashed bars are attached to the green part because it contains more bars of the rebar set than the red part.
	In this plan view, the dashed bars are in two parts. The bar planes are vertical and the parts are at equal height. The dashed bars are attached to the green part because the most bars of the rebar set are attached to it.

- If there is no more common parent part for a rebar set, Tekla Structures attaches the bar to the part that has the smallest volume.

  For example:

	In this plan view, two parts overlap and a rebar set with the dashed bars is entirely in both parts. The bars are attached to the green part because it has the smaller volume.

  If the volumes of the parts are equal for several parts, Tekla Structures chooses the part with the smallest ID.

If you need to override the automatic attachment of rebar set bars to concrete parts, you can manually attach (page 562) rebar sets and bars in them.

How to calculate the reinforcing bar length

You have three options how to calculate the reinforcing bar length in Tekla Structures:

- Along center line, which is the default method
- As a sum of leg lengths
- Using a formula

Along center line

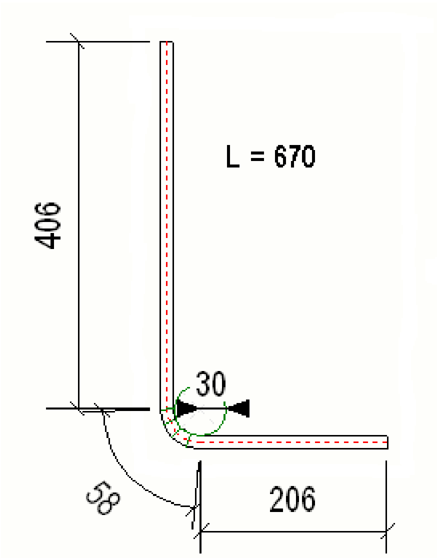
The center line length calculation is used by default when XS_USE_USER_DEFINED_REBAR_LENGTH_AND_WEIGHT is set to `FALSE` in **File menu** --> **Settings** --> **Advanced options**.

Center line length calculation uses the actual reinforcing bar diameter by default.

In the example below, the center line length is calculated as follows: `450 – (30 + 14) + 2*3.14*(30+14/2)*1/4 + 250 - (30 + 14) = 670.1`

where

- `30` = bending radius
- `14` = actual diameter (12 is nominal)

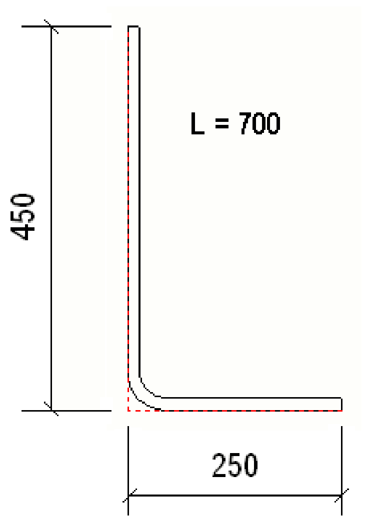


Sum of leg lengths (SLL)

The sum of leg length calculation is based on the dimensions of the straight legs and it does not take the bending radius into account.

This calculation is used when XS_USE_USER_DEFINED_REBAR_LENGTH_AND_WEIGHT and XS_USE_USER_DEFINED_REBARSHAPERULES are set to `TRUE` in **File menu** --> **Settings** --> **Advanced options**.

In the example below, the reinforcing bar length is `450 + 250 = 700`



If the length value is shown as zero in reports and inquiries, you need to define the length in **Rebar Shape Manager** for each shape.

To define the length in **Rebar Shape Manager**:

1. In **Bending schedule fields**, right-click in the **L** cell and select **SLL (Sum of leg lengths)** from the pop-up menu.
2. Click **Update**.
3. Click **Save**.

Using a formula

You can also use a formula in **Rebar Shape Manager** to calculate the reinforcing bar total length.

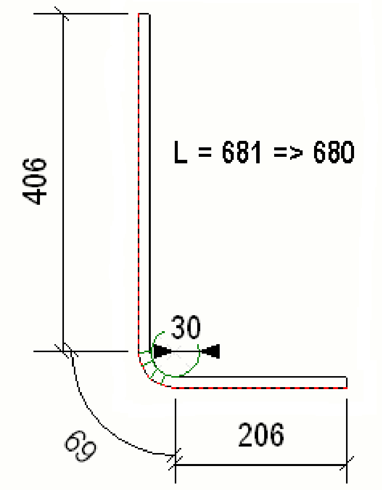
You need to set XS_USE_USER_DEFINED_REBAR_LENGTH_AND_WEIGHT and XS_USE_USER_DEFINED_REBARSHAPERULES to `TRUE` **File menu** --> **Settings** --> **Advanced options**.

For example, to take the bending radius into account and to calculate the length along the reinforcing bar outer surface, do the following:

1. In **Bending schedule fields**, right-click in the **L** cell and select **(formula)** from the pop-up menu.
2. Enter the following formula for the length calculation: `S1 + S2 + 2*3.14*(RS + DIA)*1/4`

where

- `S1` = straight leg length 1 (`406`)
- `S2` = straight leg length 2 (`206`)
- `RS` = rounding radius (`30`)
- `DIA` = actual diameter (`14`)



Accuracy

The accuracy of reinforcing bar length is defined in the `rebar_config.inp` file. The values can vary in each environment.

For example, the values shown below are from a `rebar_config.inp` file. In the default environment the file is located in `..\ProgramData\Trimble\Tekla Structures\<version>\Environments\default\system\` folder.

The following settings define the accuracy and rounding for the leg lengths:

- `ScheduleDimensionRoundingAccuracy=1.0`
- `ScheduleDimensionRoundingDirection="DOWN"`

The following settings define the accuracy and rounding for the total reinforcing bar length:

- `ScheduleTotalLengthRoundingAccuracy=10.0`
- `ScheduleTotalLengthRoundingDirection="DOWN"`

Note that also XS_USE_ONLY_NOMINAL_REBAR_DIAMETER affects the reinforcing bar length calculation.

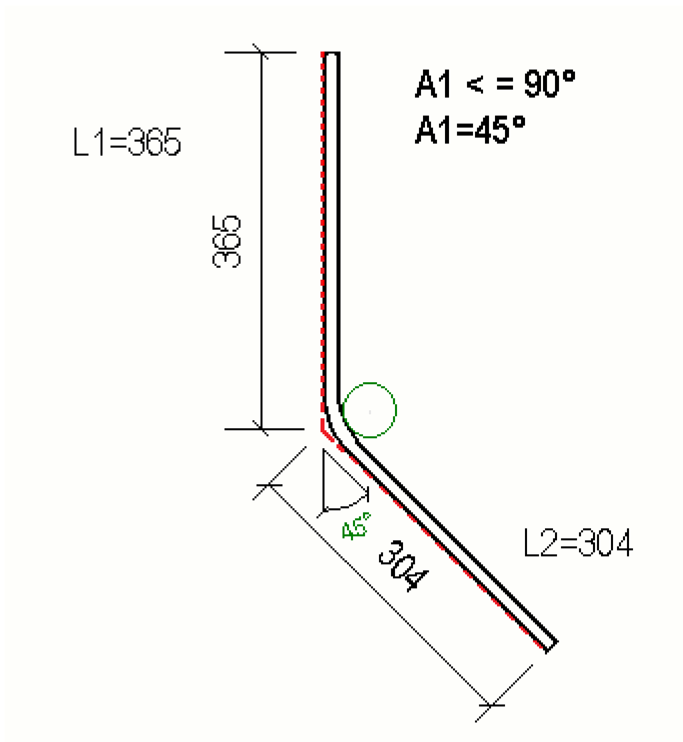
See also

Rebar shape manager in reinforcement shape recognition (page 576)

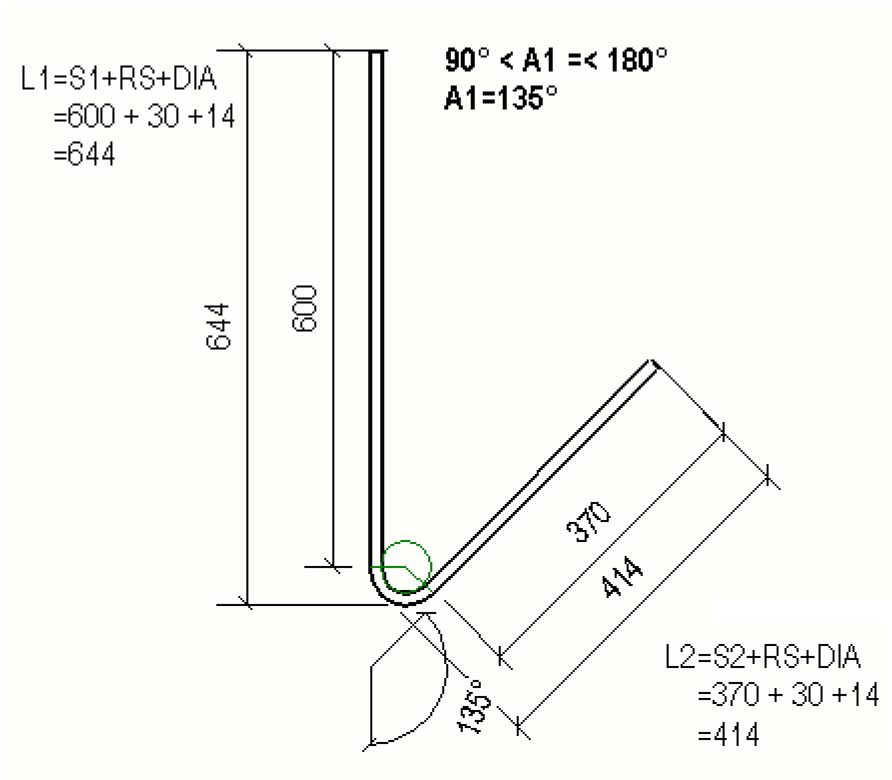
How to calculate the reinforcing bar leg length

How the reinforcing bar leg length is calculated depends on the angle between the bar legs.

- When the angle is <= 90°, the length is measured to the extension of a leg along the outer edge



- When the angle is > 90 ° and <= 180 °, the tangential length is used



The leg lengths are calculated using **Rebar Shape Manager**, where

- S1 = straight portion of a bar for the first segment
- S2 = straight portion of a bar for the second segment
- A1 = bending angle measured between the extension of the first leg and the second leg. The angle is 0° if the second segment continues to the same direction as the first segment (the bar is straight)
- L1 = leg length for the first reinforcing bar segment
- L2 = leg length for the second reinforcing bar segment
- RS = bending radius
- DIA = actual diameter of the reinforcing bar

See also

Rebar shape manager in reinforcement shape recognition (page 576)

Reinforcing bar and bar group properties (page 1009)

Reinforcement shape recognition

Tekla Structures recognizes different reinforcing bar bending shapes and assigns shape codes to them. Tekla Structures then uses the shape and dimension information in bending schedules, pull-out pictures, templates and reports.

Tekla Structures includes two methods for shape recognition.

User-defined bending shape definitions.	These definitions are created with Rebar shape manager (page 576) and saved in the RebarShapeRules.xml file. The file is located in the environment folder under ..\ProgramData\Trimble\Tekla Structures\<version>\Environments\. The exact file location may vary depending on the folder structure of your environment files.
Tekla Structures internal, hard-coded bending type definitions. Only used if the advanced option XS_USE_USER_DEFINED_REBARSHAPERULES is set to FALSE.	These internal bending types (page 588) of reinforcing bars are mapped to area-specific reinforcing bar bending type codes in the rebar_schedule_config.inp file. The file is located in ..\ProgramData\Trimble\Tekla Structures\<version>

	`\environments\common\system` folder.

See also

Reinforcement in templates (page 612)

Rebar shape manager in reinforcement shape recognition

You can define your own bar bending shapes and assign shape codes with **Rebar shape manager**, and thus increase the amount of recognized bar shapes. User-defined bending shapes are useful when Tekla Structures does not recognize the bending shape and assigns the `UNKNOWN` bending type to the shape.

Rebar shape manager is meant for users who need to customize the bending shapes according to company or project requirements.

With **Rebar shape manager** you can:

- Customize the existing bending shapes and create new bending shapes (page 576).
- Establish your own rules (page 580) for defining the bending shapes.
- Compare selected bars (page 585) in the model to the existing bending shapes.
- Customize your own dimension mappings that are used in templates and reports (page 586).
- Import and export user-defined bending shapes.
- Use user-defined bending shapes in bending schedules and pull-out pictures.

NOTE **Rebar shape manager** is a tool for recognizing reinforcing bar shapes. You cannot control the reinforcing bar creation properties, such as cover thickness, reinforcing bar grade, or size, with this tool.

See also

Tips for reinforcement shape recognition in Rebar shape manager (page 587)

Define reinforcing bar bending shapes in Rebar shape manager

With **Rebar shape manager** you can establish your own rules for defining the bending shapes. When you define your own reinforcing bar bending shapes and shape codes, an `.xml` file called `RebarShapeRules.xml` is created in the current model folder.

In addition, Tekla Structures installation contains by default another `.xml` file called `RebarShapeRules.xml`. This file contains the most typical bending shapes in your environment, and it is located in the environment folder under ..\ProgramData\Trimble\Tekla Structures\<version>\Environments\. The exact file location may vary depending on the folder structure of your environment files.

When you define new shapes, the shapes in the default RebarShapeRules.xml rule file can be appended to your own shapes. Tekla Structures reads valid RebarShapeRules.xml rule files in the model, project, firm, and system folders in that order. When the shape codes and report field values are applied, Tekla Structures uses the first matching shape in a RebarShapeRules.xml file which is found first based on the search order. If a shape is unknown according to the definitions in the first RebarShapeRules.xml file, only then the next rule file (or files) will be used. All the found bending shapes are displayed in **Rebar shape manager**.

1. Select reinforcing bars in the model.
2. On the **File** menu, click **Editors** --> **Rebar shape manager**.

   Rebar shape manager opens and lists the selected reinforcing bars in the **Model rebars** list.

   Alternatively, you can first open **Rebar shape manager** and then select reinforcing bars in the model. Click **Get selected** to add the reinforcing bars to the **Model rebars** list.

   - The **Model rebars** list shows the ID and the shape code of the selected reinforcing bars.
   - The **Shape catalog** list shows the shapes that exist in the default RebarShapeRules.xml rule file.
   - The **Tolerances** tab shows the tolerances used when the bending shape rules are compared.
3. Select one unknown shape from the **Model rebars** list.

   Tekla Structures shows a preview of the shape. The blue numbers in the preview are related to the straight bar legs in the shape, and the green numbers to the arc legs.

   Under the preview, Tekla Structures shows a description of the recognized bar geometry, and the number of bendings and arcs. If you move the mouse pointer over the text, a tooltip shows which tolerance values and other properties are in use for the selected bar. If the bar geometry has been simplified in the recognition process, **(Simplified)** is shown next to the description text, and the tooltip shows the simplification result.
4. If needed, you can compare selected bars (page 585) in the model to the existing bending shapes in the rule files.
5. To define the needed information for a bending shape, do the following:

To define	Do this
Tolerances	Enter tolerance values for the following measurements: • **Dimension** (leg lengths and other distances)

To define	Do this
	• **Angle** (bending and twist angles) • **Radius** (bending radiuses) Note that the tolerance values are stored along with the rule file `RebarShapeRules.xml`, and thus the tolerances are specific to each rule file. The following tolerances are also used in rebar shape recognition, and they can be adjusted using the `XS_REBAR_RECOGNITION_...` advanced options, the rebar catalog (`rebar_database.inp`), and the **Shape recognition** user-defined attributes of individual reinforcement objects: • Extra point shortening • Curve tolerance • Max curve radius requiring bending Additionally, you can use the **Recognize as straight bar** user-defined attribute to have Tekla Structures recognize individual reinforcement objects as straight bars even though they are bent in the model.
Shape code	Enter a shape code for an unknown shape. Note that multiple reinforcement shapes that are variants of the same shape can have the same **Shape code** but different **Bending shape rules**.
Bending shape rules	If the bending shape rules that **Rebar shape manager** defines automatically are not sufficient to distinguish certain bending shapes, you can manually add (page 580) new bending shape rules. Add or delete a bending shape rule by clicking the **Add** and **Delete** buttons on the right. Use the **Reset** button to restore the original values.
Check hooks	Select the check box if you want to define different shape codes or bending schedule fields for two bars that otherwise have exactly the same geometry, but one bar has hooks and the other one does not. If you select the check box, hooks are considered as hooks. If you clear the check box, hooks are considered as normal legs. Note that the **Check hooks** option works independently from the advanced option , and it allows bars that have different hooks to have different shape code or schedule fields regardless of the value of the advanced option.

To define	Do this
Update	Update the existing shape code definition of the selected reinforcing bar. You can update the definition if you have modified the shape code, bending shape rules or the contents of bending schedule fields.
Bending schedule fields	Define the content (page 586) for a bending schedule. Right-click a field to select a bending shape property or to enter a formula. The names of the **Bending schedule fields** (**A**, **B**, and so on) are used in templates and reports. To make sure that old reports also work correctly, we recommend that you use the same `DIM_XX` fields as in the `rebar_schedule_config.inp` file.
Schedule fields...	Click the **Schedule fields** button to add, remove or change the order of the available schedule fields. If needed, you can reset the schedule fields to original defaults. If you change the set of available schedule fields and update an existing shape, the old schedule fields which do not exist anymore will be cleared. Therefore we recommend that you do not to remove any of the default schedule fields unless you are sure they have not been used in any of the existing shapes. You can change the names of the existing schedule fields or, if you add new fields, give names to them. To use the fields in templates and reports, use the `DIM_XX` or `ANG_xx` fields, and replace `xx` with the schedule field name.

6. When you have finished defining the new shape, click **Add** to add the bending shape definition to the `RebarShapeRules.xml` file.

   To enable the **Add** button you need to change the bending shape rule, enter a shape code or select the **Check hooks** check box.

7. Click **Save** to save the `RebarShapeRules.xml` file.

   By default, the file is located in the current model folder.

   When you create, for example, a bending schedule, Tekla Structures uses the updated bending shape information, recognizes the added bending shape and assigns a correct shape code to it.

   NOTE **Rebar shape manager** is able to recognize bending shapes regardless of the modeling direction of the bars. This means that the modeling direction has no effect on the shape definition and the shape code.

When defining the bending shapes, the start or the end of modeling direction is always sorted based first on bending angles, then on twist angles, and finally on leg lengths. However, bending radius is not taken into account in the sorting. This means that radius 1 may not always be less than radius 2 or vice versa.

After the sorting, the bars that are not recognized with their original input point order are automatically examined also in the reversed input point order. This eliminates the need to create duplicate definitions for certain bar shapes.

See also

Rebar shape manager in reinforcement shape recognition (page 576)

Add new bending shape rules manually in Rebar shape manager

In some cases, the bending shape rules that are defined in **Rebar shape manager** are not sufficient to distinguish certain bending shapes. If needed, you can manually add new bending shape rules for reinforcing bars in **Rebar shape manager**.

1. In **Rebar shape manager**, click **Add** next to the **Bending shape rules** list.
2. In the **New bending rule** dialog box, select the options from the lists to define the new rule.

   The content of the lists depends on the shape and the bending of the reinforcing bar.
3. Click **OK** to add the new rule to the **Bending shape rules** list.

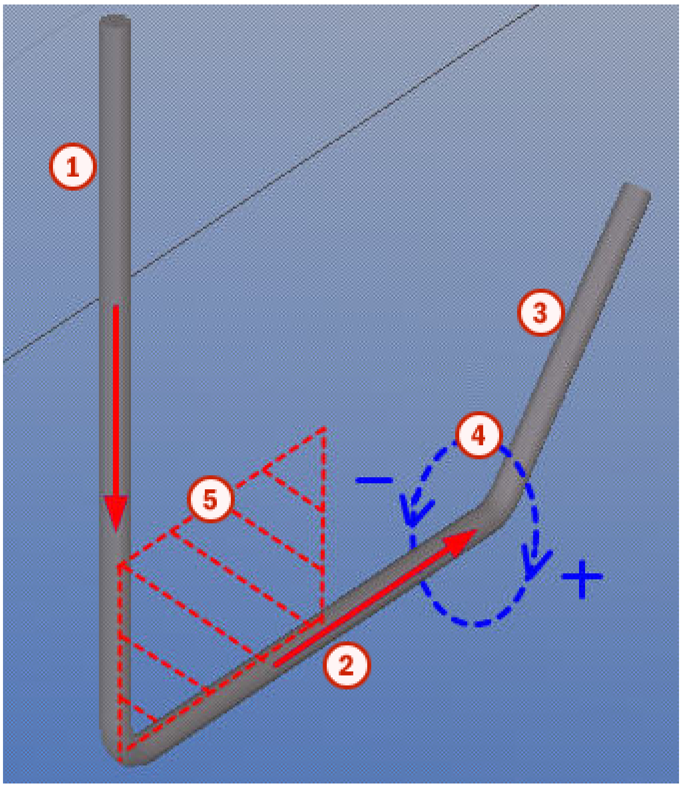
   The **OK** button is enabled only when the rule is valid.

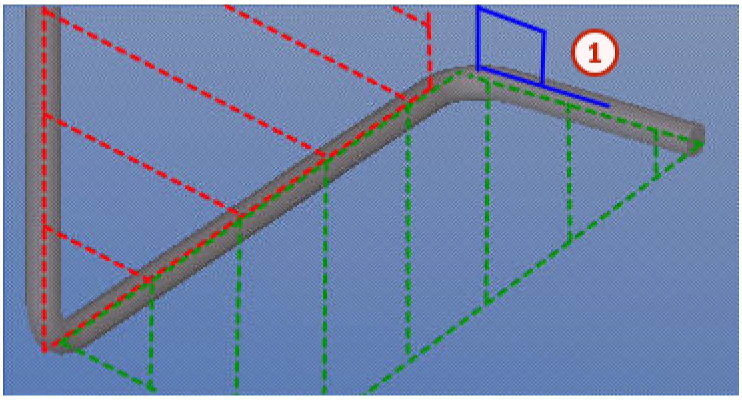
Bending shape rule settings

All the rule options are available in the **New bending rule** dialog box, even though only certain selections are valid, depending on the type of the conditions used. The left and right condition of a rule need to be of the same type. The values in the parentheses are the values that were used to create the bar shape.

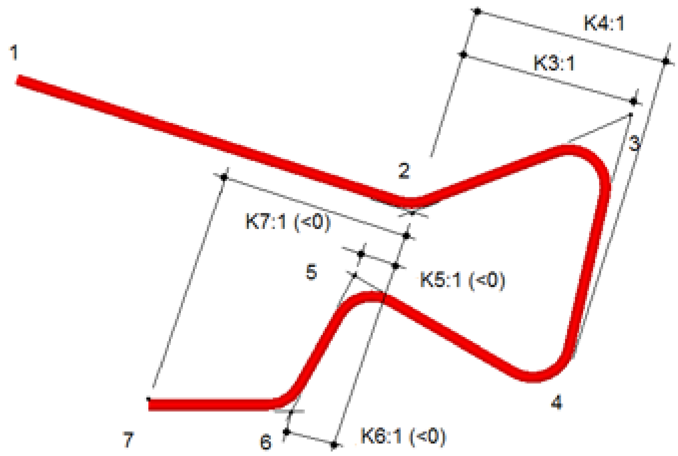
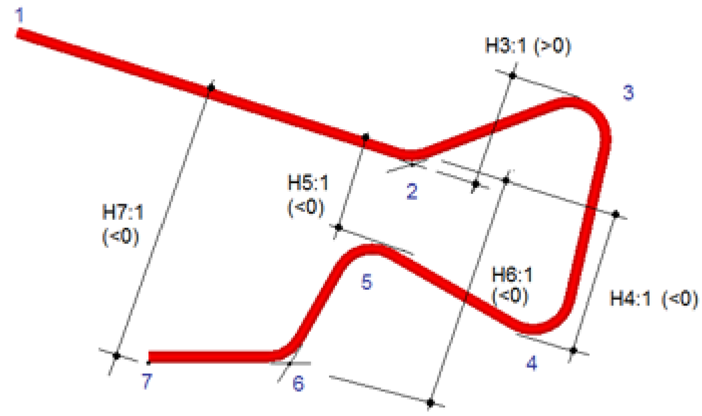
Use the **New bending rule** dialog box in **Rebar shape manager** to manually define rules for reinforcing bar bending shapes.

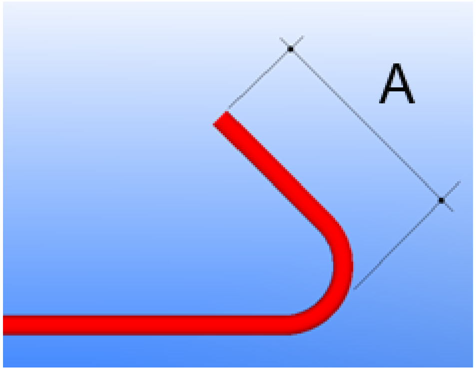
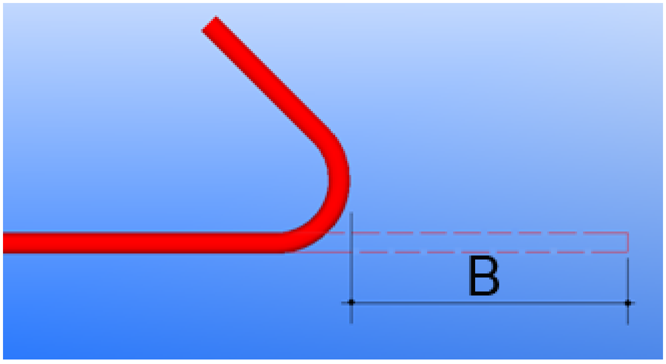
Option	Description
Angle (A)	Bending angle between the legs. Bending angle is always between 0 and +180 degrees. The angle cannot be negative.

Option	Description
Twist angle (T)	Rotation angle of a plane that has been created by two bars of consecutive lengths. The plane is rotated around the axis of the last bar creating the plane. For bars where all the legs lie in the same plane, the twist angle is either 0 degrees or +180 degrees. If the bar twists out of the plane, i.e. the bar is in 3D, the twist angle is between -180 and +180 degrees.  1. Leg 1 2. Leg 2 3. Leg 3 4. Twist angle direction 5. Plane created by legs 1 and 2

Option	Description
Twist angle example	The twist angle between two planes is +90 degrees. The planes are created by legs 1 and 2, and legs 2 and 3.  1. Twist angle: +90 degrees
Radius (R, RX)	Bending radius of the bending. `(RX) Radius *` is the value of the bending radius when all the bendings have equal radius. Otherwise the value is zero (0). `Radius * = Radius 1` ensures that all the bendings have been created using the same radius.
Bending length (BL)	Center line length of the bending.
Straight length (S)	Straight length between the start and the end of adjacent bendings. The rule is generated only when there is no straight part, for example, `Straight length 2 = 0`.
Leg length (L)	Length of the leg.
Leg (V)	Leg direction as a vector value.
Bar diameter (DIA, DIAX)	Diameter of the reinforcing bar.
Nominal diameter (NDIA, NDIAX)	Nominal diameter of the reinforcing bar.
Center line length (CLL)	Leg length according to the center line.
Sum of leg lengths (SLL)	Sum of all leg lengths.
Reversed	Reversed reinforcing bar. You can use **Reversed** to have additional bending shape rules and/or formulas for the schedule fields.

Option	Description
	When used in a rule, you can have separate definitions in shape code and/or schedule fields for reinforcing bars that have different modeling order of the points. When used as a part of a formula, you can eliminate the automatic normalization of the modeling order of the points. For example, a formula `if (REVERSED) then L2 else L3 endif` forces the content of the field to show the desired leg length depending on the order of the points or legs.
Arc inner radius (RI)	Inner radius of the arc.
Arc outer radius (RO)	Outer radius of the arc.
Arc angle (AA)	Angle of the arc.
Arc length (AL)	Length of the arc.
Arc twist angle (AT)	Rotation angle of the arc.
Arc width (AW)	Width of the arc.
Arc height (AH)	Height of the arc.
Curve width (CW)	Extreme width of the curved bar.
Curved height (CH)	Extreme height of the curved bar.
RFACTOR	Relative radius.
LFACTOR	Relative length.
Spiral rounds (SR)	Rounds of the spiral bar.
Spiral pitch (SP)	Pitch of the spiral bar.
Spiral length (SL)	Distance between the reference points of the spiral bar.
Spiral total length (STL)	Total length of the spiral bar when the bar is installed on site.
Standard radius (RS)	Standard minimum bending radius. The bending radius depends on the size and the grade of the bar.
Weight per length (WPL)	Weight per leg length.
Leg distance from leg (D)	Similar to **Point/arc distance off from leg (H)**. The difference is that **Point/arc distance off from leg (H)**

Option	Description
	considers the bending radius, whereas **Leg distance from leg (D)** is measured from the sharp corner. When the legs are parallel, both **Leg distance from leg (D)** and **Point/arc distance off from leg (H)** give the same result.
Point/arc distance along leg (K)	Distance parallel to a leg from outer edge to outer edge, or tangential to the bending. The distances are positive or negative depending on the leg direction. Example: 
Point/arc distance off from leg (H)	Distance perpendicular to a leg from outer edge to outer edge, or tangential to the bending. The distances are positive or negative depending on the leg direction. Example: 

Option	Description
Start hook (SH) **End hook (EH)** **Hook angle (SHA, EHA)** **Hook radius (SHR, EHR)** **Hook straight length (SHS, EHS)** **Hook length, method A (SHLA, EHLA)** **Hook length, method B (SHLB, EHLB)**	Start and end hook properties. Use method A or B for the hook length calculation:  
Constant angle	Constant value of the angle. Enter the value in the rightmost box.
Constant radius	Constant value of the radius. Enter the value in the rightmost box.
MAXCURVE	Maximum radius of an arc that requires bending. When an arc's bending radius exceeds **MAXCURVE**, the arc is treated as a straight leg.
Custom properties, template attributes, user-defined attributes	The custom properties, template attributes, and user-defined attributes defined in the `RebarShapeManager.CustomProperties.dat` file appear at the end of the list and can be used like any other option.

See also

Define content for templates and reports in Rebar shape manager (page 586)

Rebar shape manager in reinforcement shape recognition (page 576)

Compare reinforcing bars to bending shapes in Rebar shape manager

In **Rebar shape manager**, you can compare selected reinforcing bars in the model to the existing bending shapes in the `RebarShapeRules.xml` files.

Comparing is useful, for example, when you want to check if a bar with a special shape complies with any of the bending shapes or shape rules, or when you want to check why a bar is not recognized as you would expect. You can then modify either the bar shape in the model or the bending shape rules as needed.

1. Select reinforcing bars in the model.
2. On the **File** menu, click **Editors** --> **Rebar shape manager**.

   Rebar shape manager opens and lists the selected reinforcing bars in the **Model rebars** list.

   Alternatively, you can first open **Rebar shape manager** and then select reinforcing bars in the model. Click **Get selected** to add the reinforcing bars to the **Model rebars** list.
3. In the **Model rebars** list, select the bar shape that you want to compare.
4. In the **Shape catalog** list, select an existing bending shape.
5. Under the **Shape catalog** list, click **Compare to selected**.

In the **Bending shape rules** list, you can see the passed rules in green and the failed rules in red. An additional warning dialog box is shown if there are other reasons for mismatches between the shapes.

See also

Rebar shape manager in reinforcement shape recognition (page 576)

Define reinforcing bar bending shapes in Rebar shape manager (page 576)

Add new bending shape rules manually in Rebar shape manager (page 580)

Define content for templates and reports in Rebar shape manager

Use the **Bending schedule fields** in **Rebar shape manager** to define the content for templates and reports. Each of the **Bending schedule fields** cells can contain a shape property or a formula.

When you right-click in a **Bending schedule fields** cell, you can:

- Select a shape property from the list. The content of the list depends on the reinforcing bar geometry.
- Select the **(empty)** option to clear the content of the current cell.
- Select the **(formula)** option to enter a formula. The variables in the formula can either be the shape properties visible in the pop-up menu, or direct references to other non-empty bending schedule field cells.

  You can use the following functions in the formulas, in the same way as in custom components:

- Mathematical functions
- Statistical functions
- Trigonometric functions

  When you map angles and trigonometric functions in the **Schedule field formula** dialog box, enter the functions (sin, cos, tan) in small letters, for example `sin(A1)`. Capital letters are not recognized, and a blank space will appear in reports.

If you have angles in a formula, the formula needs to be in radians. For example, if you want to subtract 180 degrees from angle A1, enter `A1-PI` in capital letters. If you enter `A1-180` or `A1-pi`, the formula does not work.

Bending schedule fields cell shows the result of a valid formula. If the formula is not valid, a question mark and text describing the error is shown.

NOTE Use the fields **S**, **T**, **U** or **V** to report angles. If you do not use these fields, you need to override the default unit settings in the **Template Editor**.

Example

Formula is `L1+L3+L5-2*DIA`

- `L1`, `L3` and `L5` are the leg lengths measured from outer edge to outer edge
- `H1` is the total width
- to achieve `H1`: `L1+L3+L5 minus 2*bar diameter`

See also

Add new bending shape rules manually in Rebar shape manager (page 580)

Rebar shape manager in reinforcement shape recognition (page 576)

Tips for reinforcement shape recognition in Rebar shape manager

The reinforcing bar shape recognition is based on each shape’s bending shape rules. The shapes and their rules are listed in the `RebarShapeRules.xml` file, located in the environment folder under `..\ProgramData\Trimble\Tekla Structures\<version>\Environments\` by default. (The exact file location may vary depending on the folder structure of your environment files.) Sometimes a shape matches the rules of two shapes, and Tekla Structures does not recognize the reinforcing bar shape as expected.

NOTE The most convenient way to ensure that a shape is recognized correctly is to modify the shape definition by adding more rules (page 580) to the shape in **Rebar shape manager**.

However, if necessary, you can manually modify the `RebarShapeRules.xml` file and thus affect the shape recognition. When Tekla Structures recognizes the shape, the order of the shapes in the `RebarShapeRules.xml` is important:

- The first shape that matches the rules is the one that Tekla Structures recognizes as the shape. If you need to change the order of the shapes to change how Tekla Structures recognizes the shape, you can do it by manually modifying the `RebarShapeRules.xml` file. If you modify the file and change the order of the shapes, make sure that the structure of the file stays valid.
- You can divide the shape definitions in several `RebarShapeRules.xml` files and different folders. Tekla Structures searches the `RebarShapeRules.xml` file in the model, project, firm, and system folders in that order. Tekla Structures uses the first matching shape in the first `RebarShapeRules.xml` file which is found based on the search order.

See also

Rebar shape manager in reinforcement shape recognition (page 576)

Hard-coded bending type identifiers in reinforcement shape recognition

Tekla Structures recognizes different reinforcing bar bending shapes and assigns bending type identifiers to them.

NOTE This bending shape recognition method and the following bending types are only used if the advanced option XS_USE_USER_DEFINED_REBARSHAPERULES is set to `FALSE`.

We recommend having `XS_USE_USER_DEFINED_REBARSHAPERULES` set to `TRUE`, which is the default value, and using Rebar shape manager (page 576) instead in bending shape recognition.

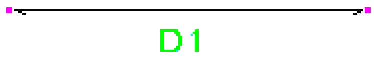
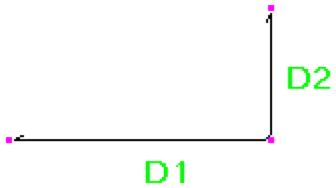
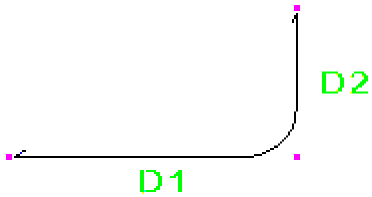
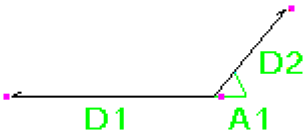
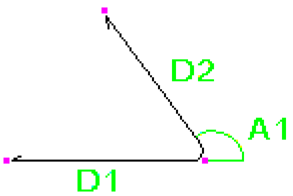
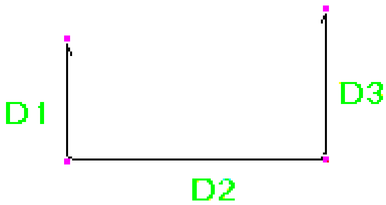
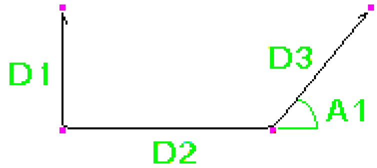
The bending type identifiers in the table below are internal, hard-coded types of Tekla Structures. The leg dimensions (`D1`, `D2`, and so on) and bending angles (`A1`, `A2`, and so on) of reinforcing bars are Tekla Structures internal dimensions and angles. You can map the Tekla Structures internal bending types, for example, to country- or project-specific bending types, and the Tekla Structures internal dimensions and angles to specific template attributes. You can do this in the `rebar_schedule_config.inp` file.

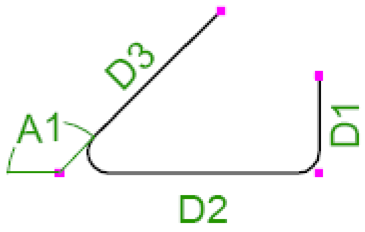
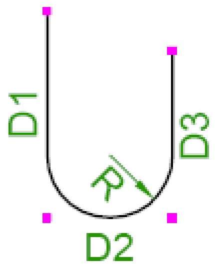
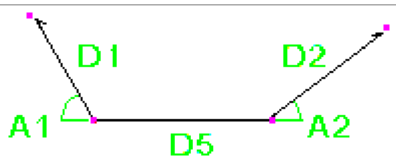
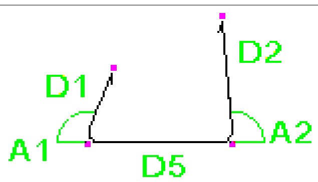
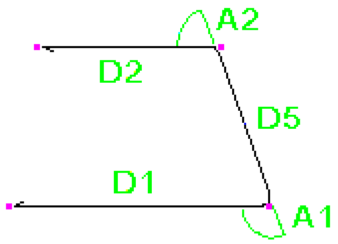
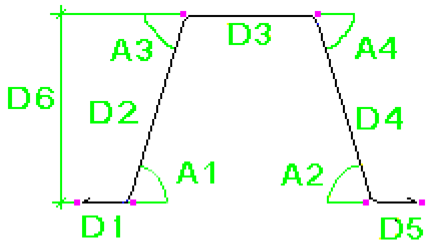
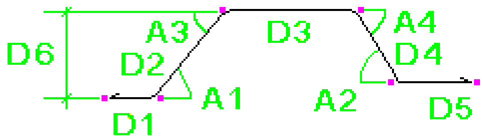
Reinforcing bar bending dimensions are calculated so that the leg dimensions (`D1`, `D2`, and so on) follow the outer edge, or the edge extension, of the reinforcing bar. The total length is calculated according to the center line of the reinforcing bar.

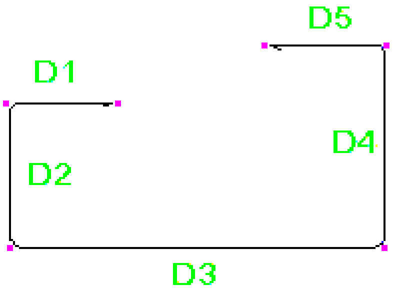
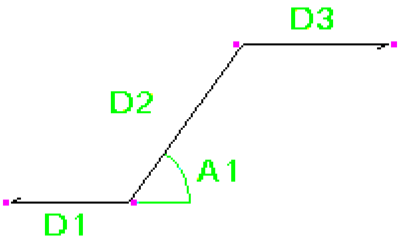
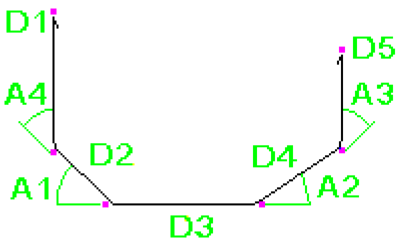
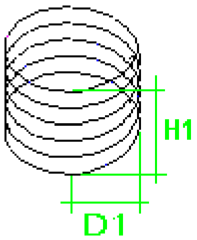
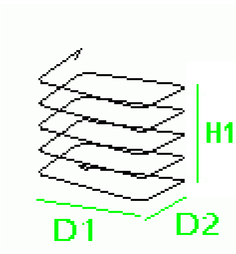
If Tekla Structures does not recognize the shape of a reinforcing bar, it assigns the `UNKNOWN` bending type to it.

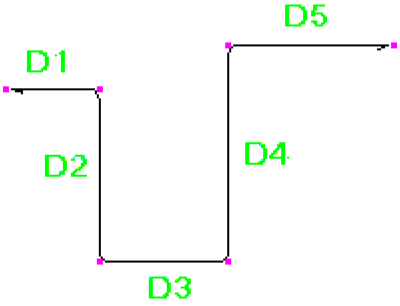
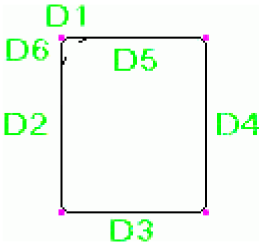
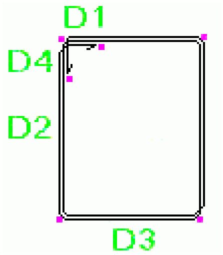
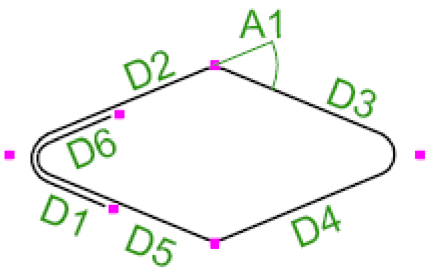
The magenta points in the images in the table below represent the points you pick in the model when you create reinforcing bars.

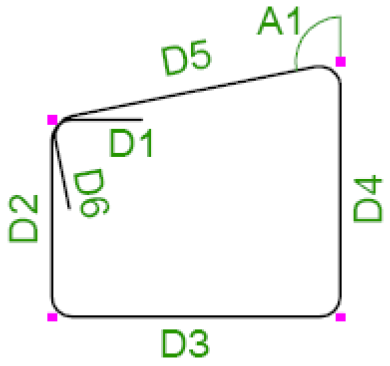
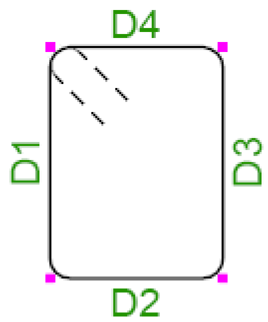
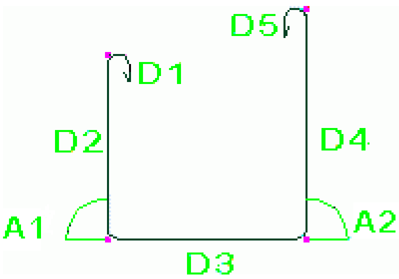
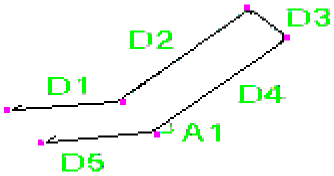
NOTE If you want to customize the hard-coded bending shapes or define new bending shapes, use Rebar shape manager (page 576).

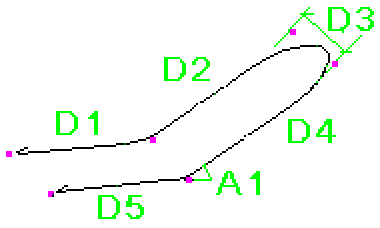
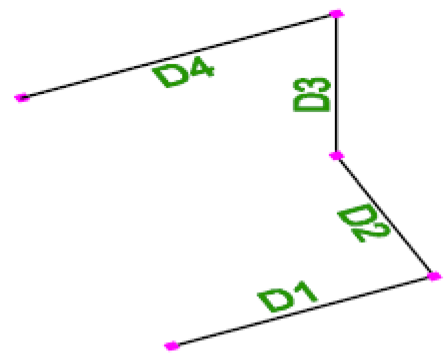
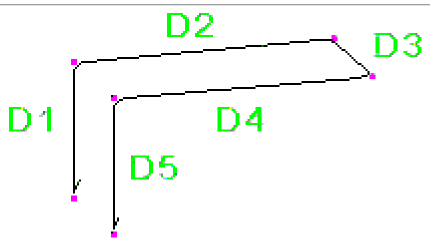
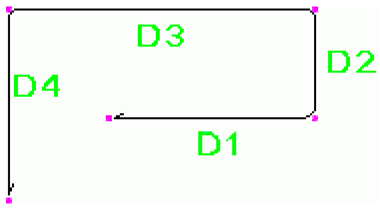
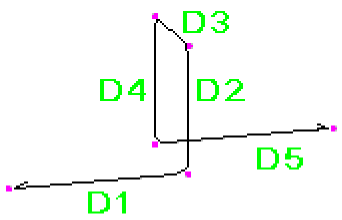
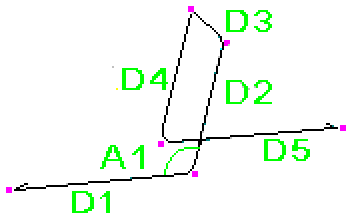
Bending type identifier	Bending shape
1	
2_1	 Requires standard bending radius.
2_2	 Non-standard bending radius.
3_1	
3_2	
4	
4_2	

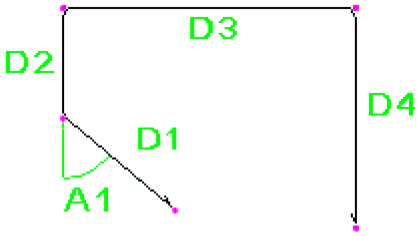
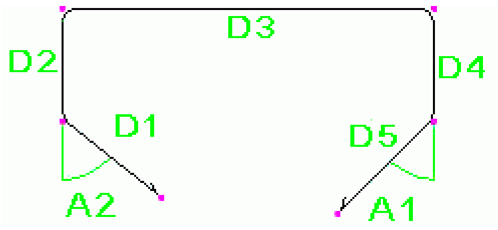
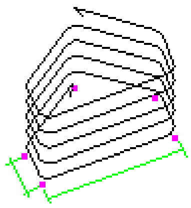
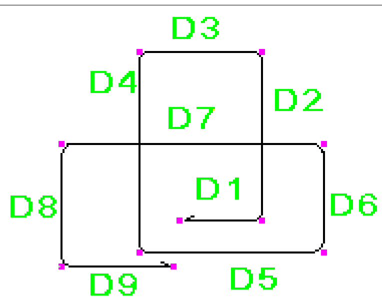
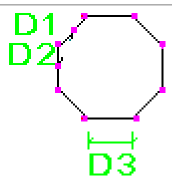
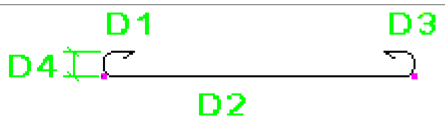
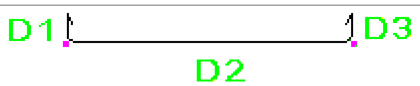
<table>
<tr><th>Bending type identifier</th><th>Bending shape</th></tr>
<tr><td>4_3</td><td></td></tr>
<tr><td>4_4</td><td></td></tr>
<tr><td>5_1</td><td></td></tr>
<tr><td>5_2</td><td></td></tr>
<tr><td>5_3</td><td></td></tr>
<tr><td>6_1</td><td></td></tr>
<tr><td>6_2</td><td></td></tr>
</table>

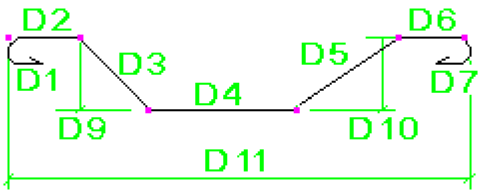
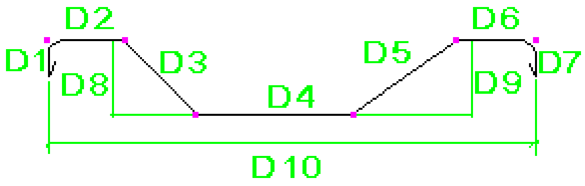
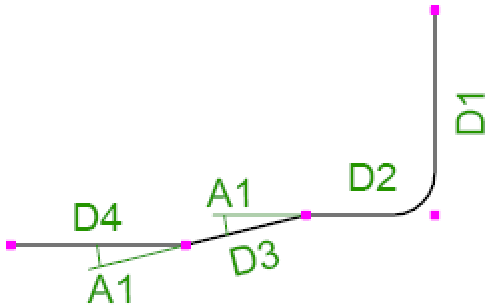
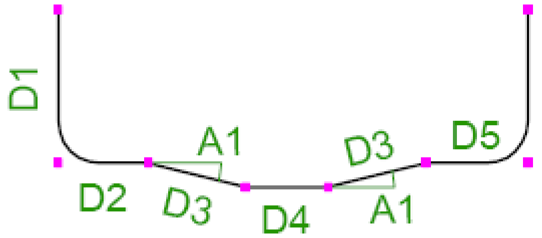
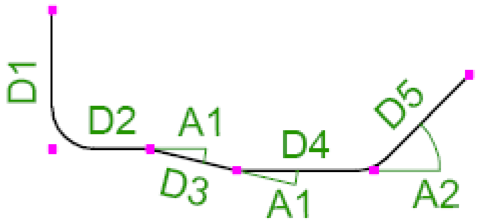
Bending type identifier	Bending shape
7	
8	
9	 Requires 180 degree hook.
10	
11	 D1 = Radius from center of circle to the center line of reinforcing bar.
12	

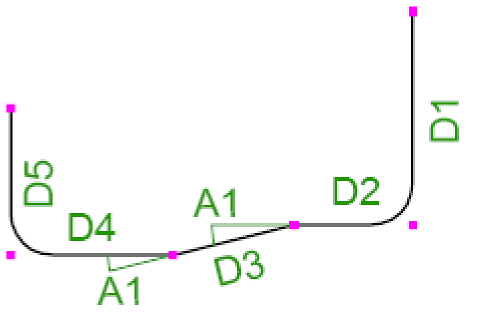
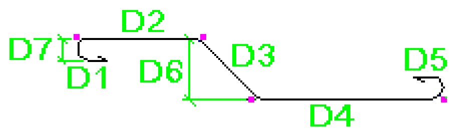
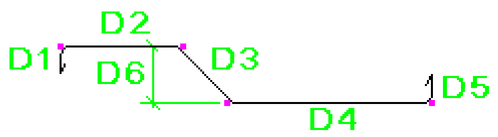
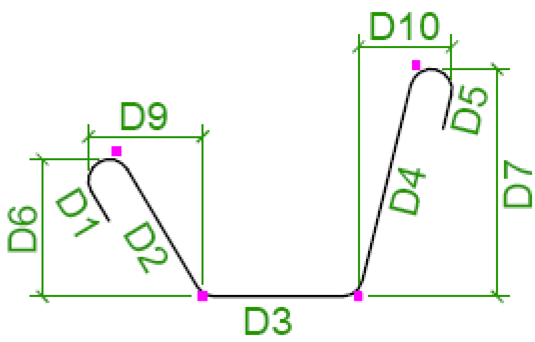
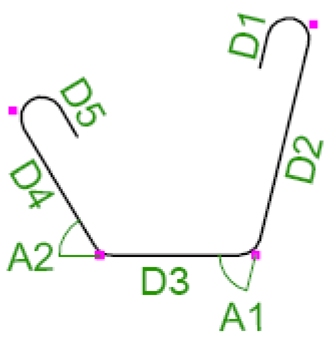
Bending type identifier	Bending shape
13	 Can also be modeled using hooks at both ends (i.e. model D1 and D5 using 90 degree hooks).
14	 Requires 90 degree hooks at both ends.
14_2	
14_3	

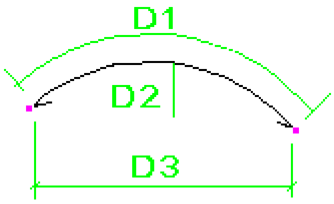
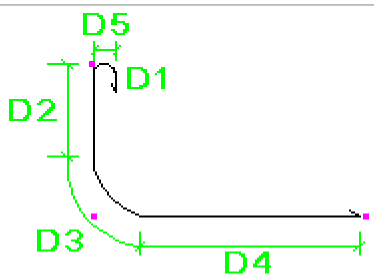
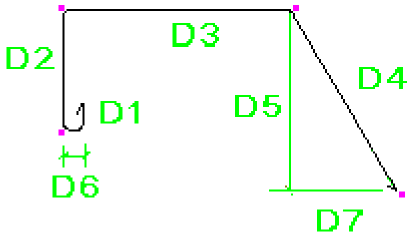
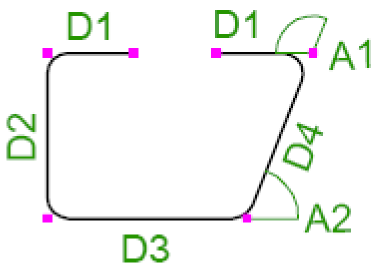
Bending type identifier	Bending shape
14_4	 Requires 90 degree hooks at both ends.
14_5	 Recognized when the start point and end point are in the same location and no hooks are used. If `XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION` is set to `FALSE`, reinforcing bars with hooks (types 14 and 48) are recognized as 14_5.
15	 Requires hooks at both ends.
16_1	

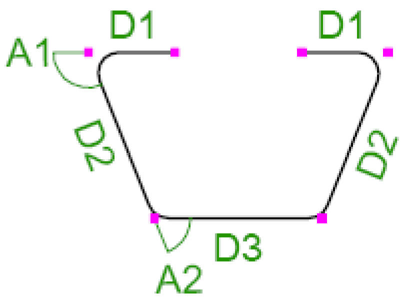
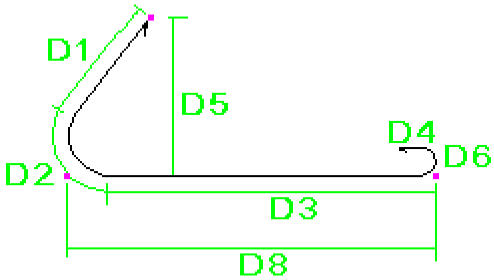
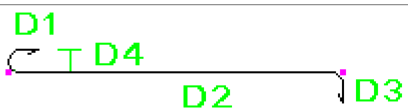
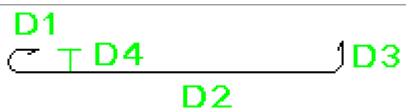
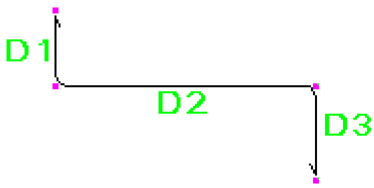
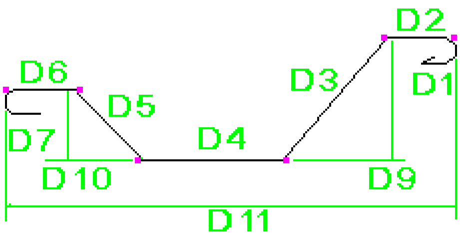
Bending type identifier	Bending shape
16_2	
17	
18	
19	
20_1	
20_2	

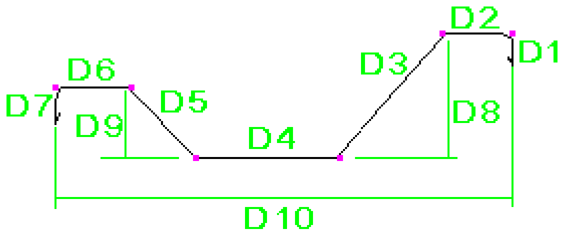
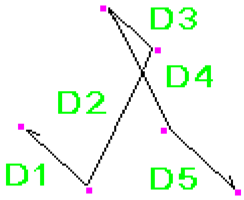
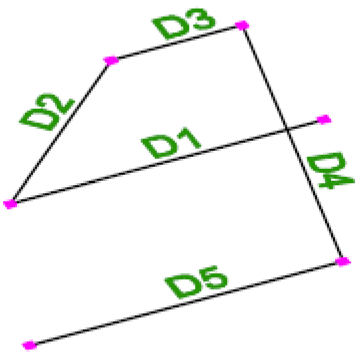
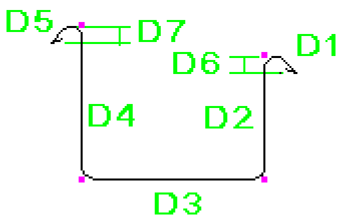
Bending type identifier	Bending shape
21	
22	
23	
24	
25	
26	 Requires 180 degree hooks at both ends.
27	 Requires 90 degree hooks at both ends.

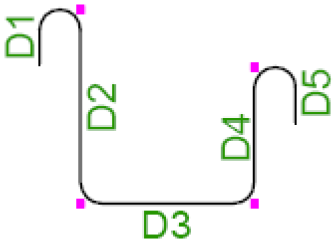
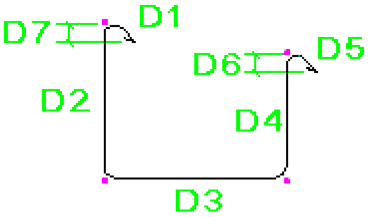
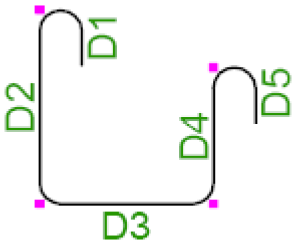
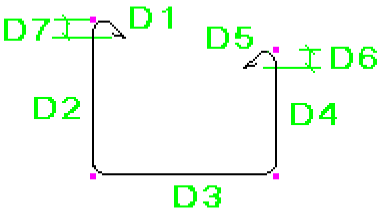
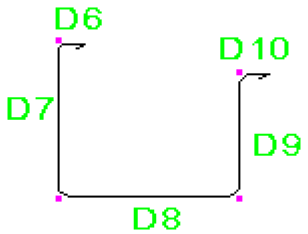
Bending type identifier	Bending shape
28	 Requires 180 degree hooks at both ends.
29	 Requires 90 degree hooks at both ends.
29_2	
29_3	
29_4	

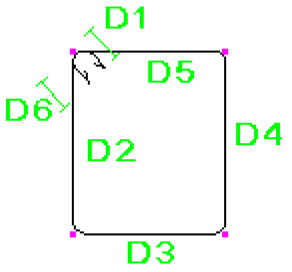
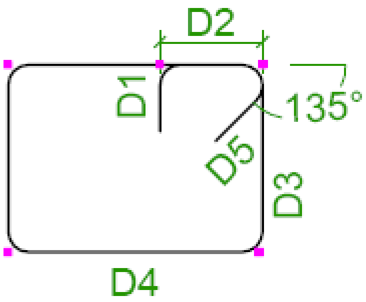
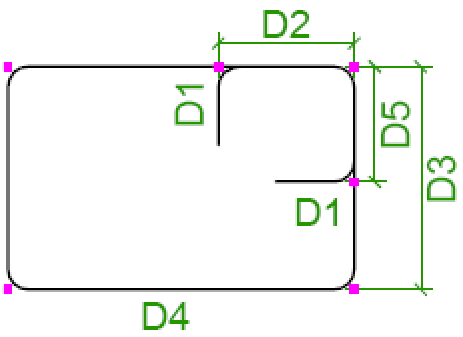
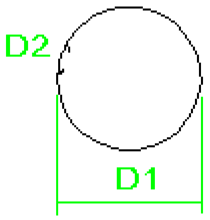
Bending type identifier	Bending shape
29_5	
30	 Requires 180 degree hooks at both ends.
31	 Requires 90 degree hooks at both ends.
32	 Requires 180 degree hooks at both ends.
32_2	

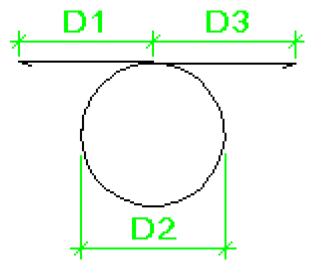
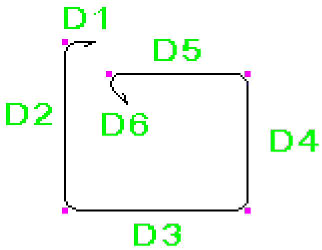
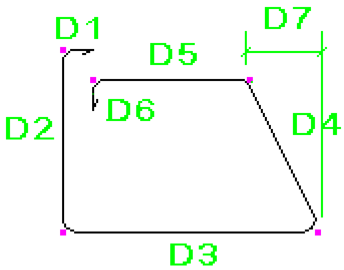
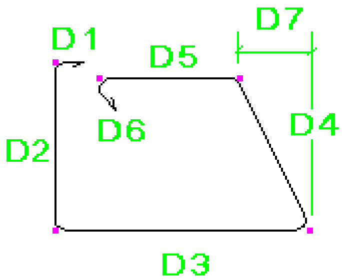
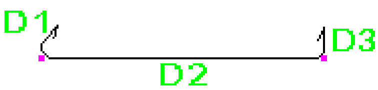
Bending type identifier	Bending shape
33	 Requires 90 degree hooks at both ends.
34	
35	 Requires 180 degree hook.
36	 Requires 180 degree hook.
36_2	 Can also be modeled using hooks at both ends.

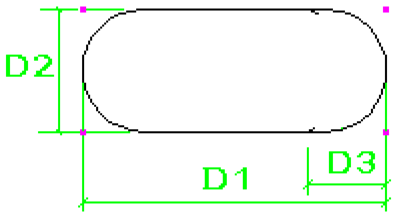
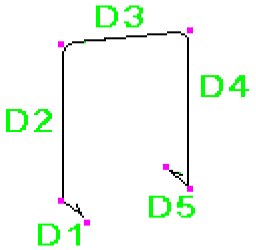
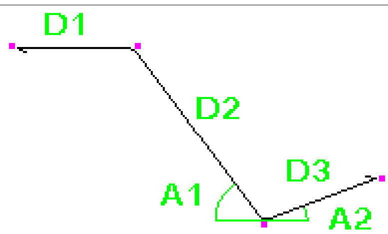
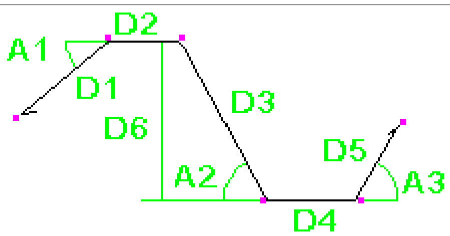
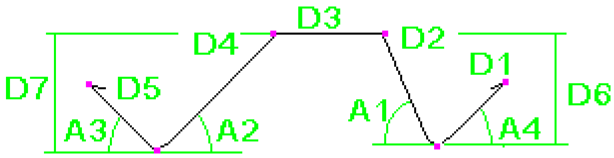
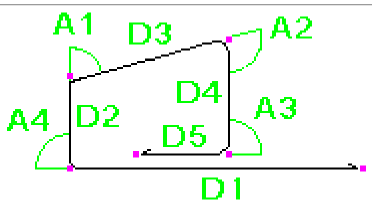
Bending type identifier	Bending shape
36_3	 Can also be modeled using hooks at both ends.
37	 Requires 180 degree hook.
38	 Requires 180 degree hook at one end and 90 degree hook at the other end.
38_2	
39	
40	 Requires 180 degree hooks at both ends.

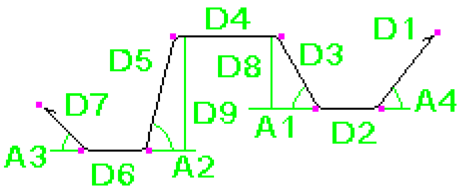
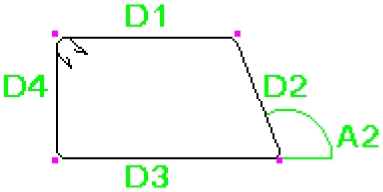
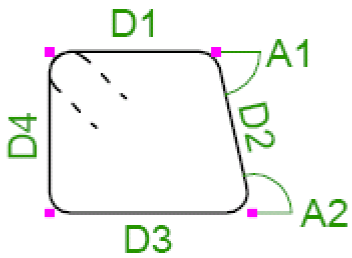
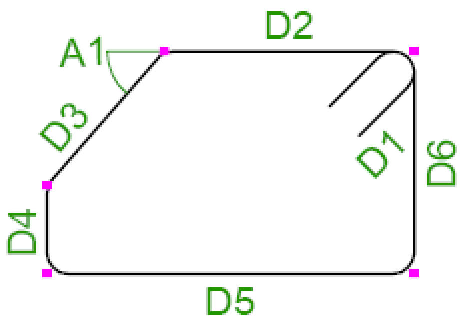
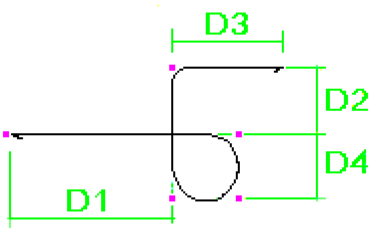
Bending type identifier	Bending shape
41	 Requires 90 degree hooks at both ends.
42	 Requires 180 degree hooks at both ends.
43	
43_2	
44	 Requires hooks at both ends.

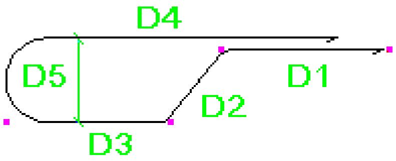
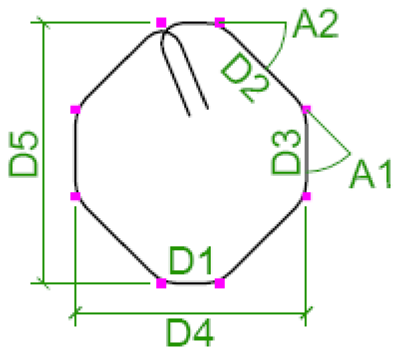
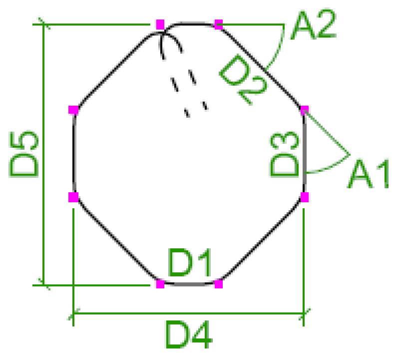
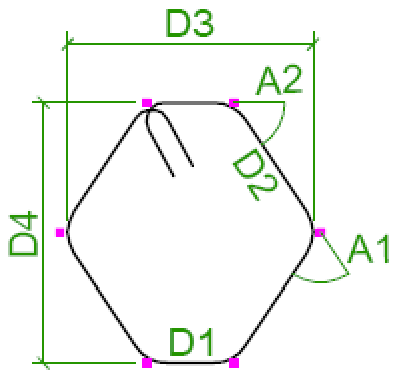
Bending type identifier	Bending shape
44_2	 Requires 180 degree hooks at both ends.
45	 Requires hooks at both ends.
45_2	 Requires 180 hooks at both ends.
46	 Requires hooks at both ends.
47	 Requires 90 degree hooks at both ends.

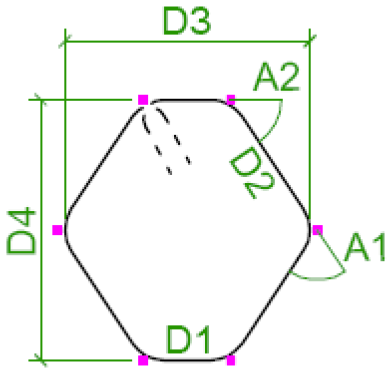
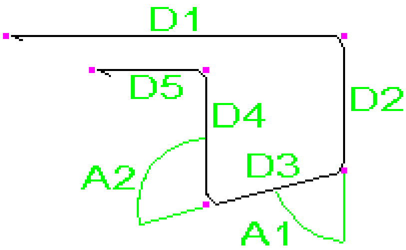
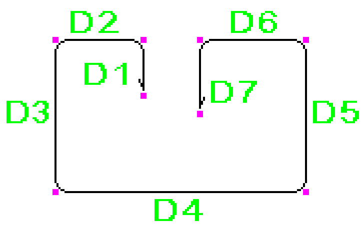
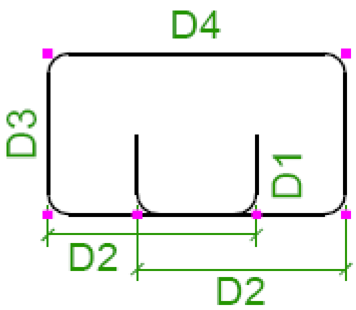
Bending type identifier	Bending shape
48	 Requires hooks at both ends.
48_2	 Requires hooks at both ends.
48_3	
49	 D1 = Reinforcing bar center line diameter.

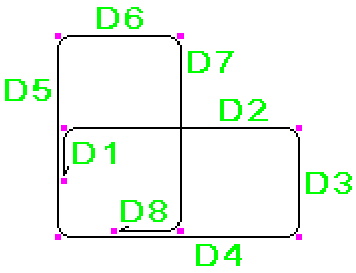
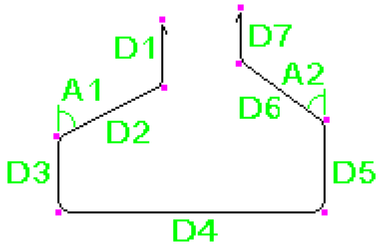
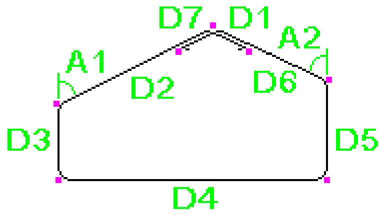
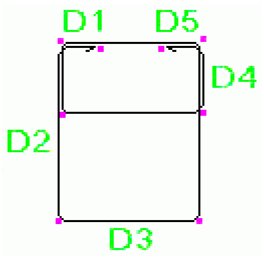
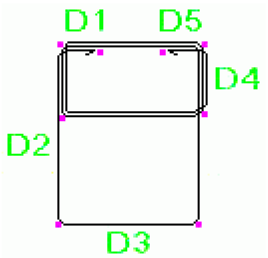
Bending type identifier	Bending shape
49_2	
50	 Requires hooks at both ends.
51	 Requires 90 degree hooks at both ends.
52	 Requires hooks at both ends.
53	 Requires hooks at both ends.

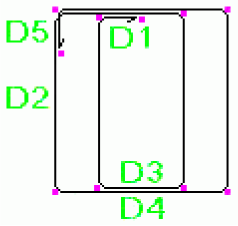
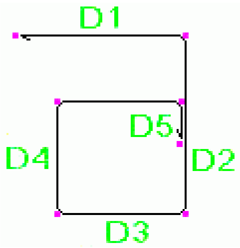
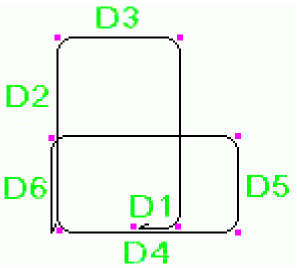
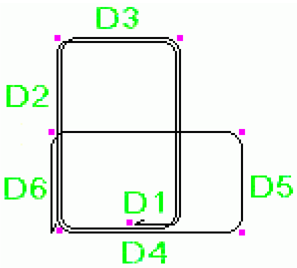
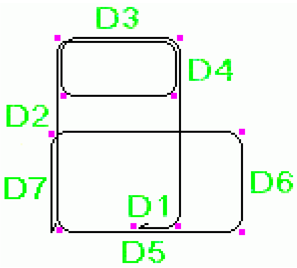
Bending type identifier	Bending shape
54	 Requires hooks at both ends.
55	
56	
57	
58	
59	

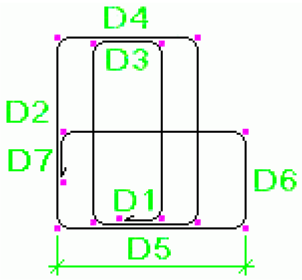
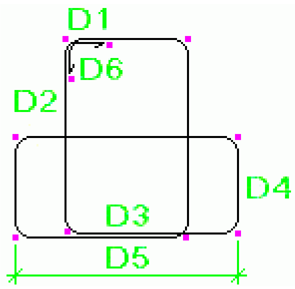
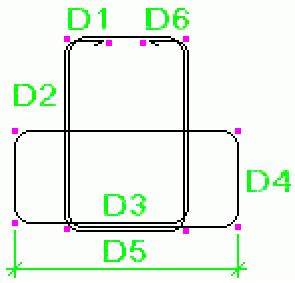
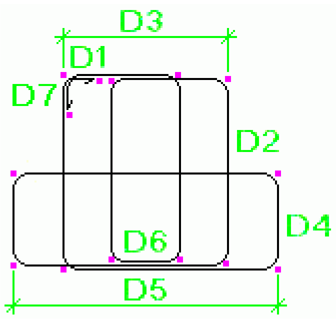
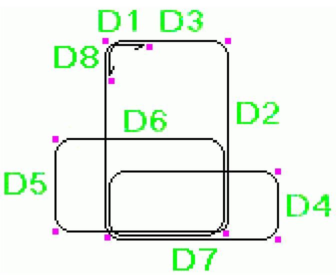
Bending type identifier	Bending shape
60	
61	 Requires hooks at both ends.
61_2	 Recognized if `XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION` is set to `FALSE`.
61_3	 Requires hooks at both ends.
62	 Requires hook.

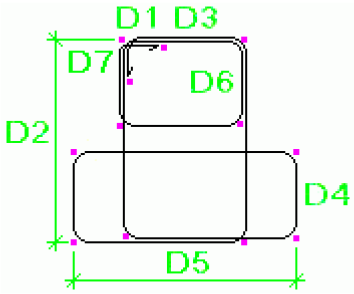
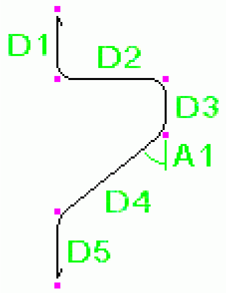
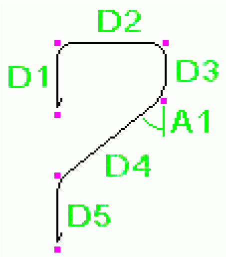
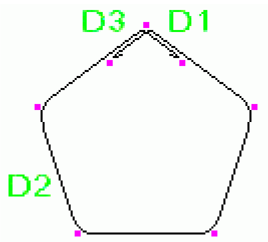
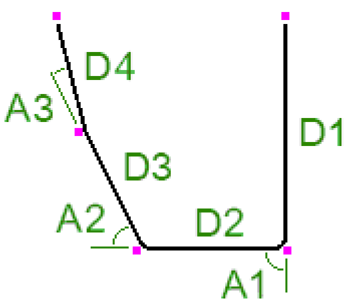
Bending type identifier	Bending shape
63	 Requires hook.
64	 Requires hooks at both ends.
64_2	 Recognized if `XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION` is set to `FALSE`.
65	 Requires hooks at both ends.

Bending type identifier	Bending shape
65_2	 Recognized if `XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION` is set to `FALSE`.
66	
67	
67_2	

Bending type identifier	Bending shape
68	
69_1	
69_2	
70_1	
70_2	

Bending type identifier	Bending shape
71	
72	
73_1	
73_2	
73_3	

Bending type identifier	Bending shape
74	
75_1	
75_2	
76	
77	

Bending type identifier	Bending shape
78	
79_1	
79_2	
80	
`UNKNOWN`	For example: 

See also

Reinforcement in templates (page 612)

Reinforcement shape recognition (page 575)

Reinforcement in templates

Sometimes you need to localize reinforcing bar bending types or to create templates for reinforcing bar bending schedules.

NOTE If you want to customize the hard-coded bending shapes or define new bending shapes, use **Rebar shape manager**. See Define reinforcing bar bending shapes in Rebar shape manager (page 576).

Reinforcement templates

You can show dimensions, bending angles, and bending types of reinforcing bars in drawings and reports by including reinforcement-specific attributes, such as `DIM_A`, `ANG_S`, `SHAPE`, and `SHAPE_INTERNAL`, in template fields. For more information on creating templates, see Template Editor User's Guide.

Mapping dimensions

Use the `rebar_schedule_config.inp` file in the `..\ProgramData\Trimble\Tekla Structures\<version>\environments\<environment>\system` folder to map

- Tekla Structures internal reinforcing bar dimensions and angles with specific template attributes
- Tekla Structures internal reinforcing bar bending types with specific bending types

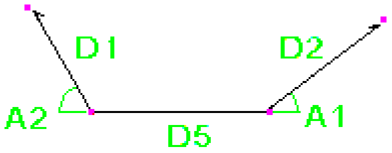
These mappings are environment-specific by default. You can modify them to suit your company or project needs.

You can use equations, functions, and `if` statements to calculate the dimensions and angles you need to show.

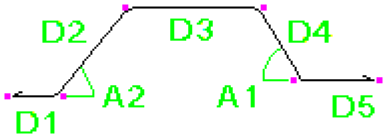
Use any standard text editor (for example, Microsoft Notepad) to edit the `rebar_schedule_config.inp` file.

Examples

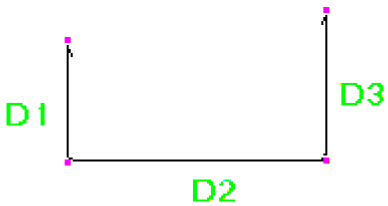
The following example of the `rebar_schedule_config.inp` file maps the internal bending type `5_1` to the bending type identifier `E`, and the leg dimensions and bending angles to specific template attributes.

rebar_schedule_config.inp	
BEND_TYPE_5_1[1]="E" BEND_TYPE_5_1[2]="DIM_A=D1" BEND_TYPE_5_1[3]="DIM_B=D5" BEND_TYPE_5_1[4]="DIM_C=D2" BEND_TYPE_5_1[5]="DIM_TD=TD" BEND_TYPE_5_1[6]="ANG_U=A1" BEND_TYPE_5_1[7]="ANG_V=A2"	

With this mapping, the internal bending type `6_2` becomes `XY`, and the template attributes `DIM_B` and `DIM_C` will show the horizontal and vertical dimensions of the second leg `D2`, and `DIM_E` and `DIM_F` the horizontal and vertical dimensions of the fourth leg `D4`.

rebar_schedule_config.inp	
BEND_TYPE_6_2[1]="XY" BEND_TYPE_6_2[2]="DIM_A=D1" BEND_TYPE_6_2[3]="DIM_B=D2*COS(A2*PI/180)" BEND_TYPE_6_2[4]="DIM_C=D2*SIN(A2*PI/180)" BEND_TYPE_6_2[5]="DIM_D=D3" BEND_TYPE_6_2[6]="DIM_E=D4*COS(A1*PI/180)" BEND_TYPE_6_2[7]="DIM_F=D4*SIN(A1*PI/180)" BEND_TYPE_6_2[8]="DIM_G=D5" BEND_TYPE_6_2[9]="DIM_TD=TD"	

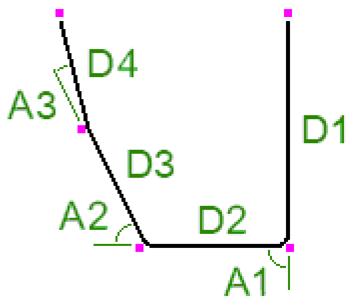
The following example maps the internal bending type `4` to the bending type identifier A **if** the dimensions `D1` and `D3` are the same. Otherwise it maps `4` to `B`.

rebar_schedule_config.inp	
BEND_TYPE_4[1]=if (D1==D3) then ("A") else ("B") endif BEND_TYPE_4[2]="DIM_A=D1" BEND_TYPE_4[3]="DIM_B=D2" BEND_TYPE_4[4]="DIM_C=D3" BEND_TYPE_4[5]="DIM_TD=TD"	

If Tekla Structures does not recognize a reinforcing bar bending shape, it uses the internal bending type `UNKNOWN` for it. In the `rebar_schedule_config.inp` file you can also define how unknown bending types appear in drawings and reports. For example, you may just want to use the bending type identifier `???`, and list all leg dimensions and bending angles.

rebar_schedule_config.inp

```
BEND_TYPE_UNKNOWN[1]="???"
BEND_TYPE_UNKNOWN[2]="DIM_A=D1"
BEND_TYPE_UNKNOWN[3]="DIM_B=D2"
BEND_TYPE_UNKNOWN[4]="DIM_C=D3"
BEND_TYPE_UNKNOWN[5]="DIM_D=D4"
BEND_TYPE_UNKNOWN[6]="DIM_E=D5"
BEND_TYPE_UNKNOWN[7]="DIM_F=D6"
BEND_TYPE_UNKNOWN[8]="ANG_S=A1"
BEND_TYPE_UNKNOWN[9]="ANG_T=A2"
BEND_TYPE_UNKNOWN[10]="ANG_U=A3"
BEND_TYPE_UNKNOWN[11]="ANG_V=A4"
BEND_TYPE_UNKNOWN[12]="DIM_TD=TD"
```




```
==========================================================================================================
No. Grade    Size  Mark  Length  Type   A     B     C     D     E     F     S     T     U     V     TD
----------------------------------------------------------------------------------------------------------

1   A615-40  #4    R/5   1930    ???    740   420   430   380               90    65    15          76
```

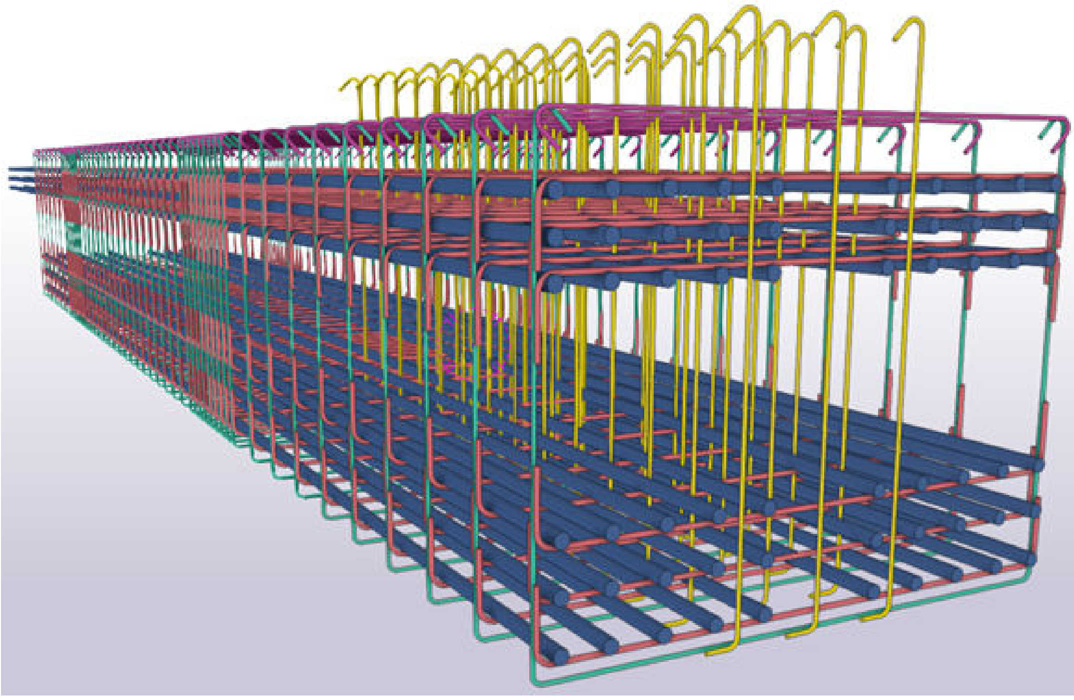
See also

[Hard-coded bending type identifiers in reinforcement shape recognition (page 588)]

[Reinforcement shape recognition (page 575)]

2.11 Work with rebar assemblies

A *rebar assembly* is a prefabricated reinforcement entity or cage that usually consists of several reinforcement objects, such as single reinforcing bars, different bar groups and rebar sets, reinforcement meshes, and possibly strands. Rebar assemblies can also contain assemblies and parts as sub-assemblies.



The structure of a rebar assembly can be flat where all objects are on one assembly hierarchy level, or there can be sub-assemblies on several assembly levels in the assembly structure.

Note that the assembly hierarchy affects the drawing and report outputs. Report and drawing templates need to be made with similar hierarchy as the assembly hierarchy in the model in order to output the data from correct assembly levels. We recommend you to test the different outputs early on in the modeling phase to ensure that all the relevant assembly information is included in the needed deliverables, such as production drawings.

When you work with rebar assemblies, use the same selection switches and the same methods to move in the assembly hierarchy as with other nested assemblies (page 421).

Create a rebar assembly

You can create basic, one-level rebar assemblies or nested, hierarchical rebar assemblies. Start by creating basic rebar assemblies. If needed, you can later add more objects or sub-assemblies, or form hierarchies by adding these basic rebar assemblies as sub-assemblies to higher-level rebar assemblies.

A reinforcement object can only have one relation, either to a concrete part or to a rebar assembly. When you add a reinforcement object to a rebar assembly, it will be detached from the part. If a reinforcement object already belongs to a rebar assembly, it cannot be attached to a part.

Note that individual rebar set bars or rebar set bar groups cannot be added to rebar assemblies, only entire rebar sets can be added.

Create a basic rebar assembly

You can create a basic rebar assembly with various reinforcement objects on the same assembly hierarchy level.

The reinforcement object with the highest weight will become the main object of the rebar assembly.

1. Create the reinforcement objects that you want to include in the rebar assembly.
2. On the **Rebar** tab, click **Assembly** --> **Create rebar assembly**.
3. Select the objects that you want to include in the rebar assembly.

   You can select reinforcing bars, bar groups, rebar sets, reinforcement meshes, and strands.
4. Click the middle mouse button to finish selecting objects and to create the rebar assembly.

TIP Alternatively, you can first select the objects, then start the command, and finally click the middle mouse button. This may help in controlling that you have a correct set of objects selected for the rebar assembly.

Note that if you select components and want to include the component objects in the rebar assembly, the components need to be exploded first. Tekla Structures warns you about this and explodes the components when you click **Yes** in the warning dialog box.

Note that if you select parts and assemblies, they are added as sub-assemblies, and the rebar assembly becomes a nested assembly (page 618).

To modify the rebar assembly structure, for example, by adding or removing objects and hierarchy levels, see Modify a rebar assembly (page 617) and Remove objects from a rebar assembly (page 620).

Rebar assembly properties

Use the **Rebar assembly** property pane to view and modify the properties of rebar assemblies. The file name extension of a rebar assembly property file is `.ras`.

Property	Description
Assembly numbering	Numbering prefix and start number (= numbering series) of the rebar assembly.
Name	User-definable name of the rebar assembly. By default, the name of the rebar assembly main object is shown in brackets [ ].

Property	Description
Rebar assembly type	Select the type for the rebar assembly. The options are (empty), **Cage**, **Mesh**, **Bent mesh**, **Braced girder**, **Embed**, and **Roll mat**.
IFC entity	For IFC export, select the IFC entity type and subtype of the rebar assembly. 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	User-defined attributes provide additional information about the rebar assembly, for example, workflow information. User-defined attributes can consist of numbers, text, lists, or dates. 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.

Modify a rebar assembly

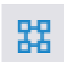
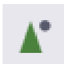
You can modify the rebar assembly structure by adding more objects, or by changing the main object of a rebar assembly, for example.

NOTE The active selection switch controls whether you can select reinforcement objects or assemblies when you use the different rebar assembly commands.

Add objects to an existing rebar assembly

You can add more objects to a basic rebar assembly, or to any level of a nested rebar assembly.

Note that you can only add parts to rebar assemblies as sub-assemblies. See the 'Create a nested rebar assembly' instructions below.

1. Ensure that the  **Select objects in assemblies** or  **Select objects in components** selection switch is active.
2. Select the objects that you want to add.
3. Right-click and select **Assembly** --> **Add to assembly**.
4. Select the rebar assembly to which you want to add the objects.

Create a nested rebar assembly

You can add parts, assemblies, and rebar assemblies as sub-assemblies in a rebar assembly on a lower level in the assembly hierarchy.

1. Ensure that the **Select assemblies** selection switch is active.
2. Select the parts and assemblies that you want to add to the rebar assembly.

   They will become sub-assemblies in the nested rebar assembly.
3. Right-click and select **Assembly** --> **Add as sub-assembly**.
4. Select the rebar assembly to which you want to add the parts and assemblies.

Alternatively, you can use the **Rebar** --> **Assembly** --> **Add as sub-assembly** command on the ribbon.

Check and highlight objects in a rebar assembly

Use the **Inquire** tool to check which objects belong to a particular rebar assembly.

1. On the ribbon, click the down arrow next to , and then select **Assembly objects** to inquire the rebar assembly objects.
2. Select a rebar assembly. Ensure that you are on the intended assembly hierarchy level.

   Tekla Structures highlights all the objects that belong to the same rebar assembly. The main reinforcement object is highlighted in orange and the other rebar assembly objects in yellow.

Change the main object of a rebar assembly

By default, the main object in a rebar assembly is the reinforcement object with the highest weight. You can change the main object in a rebar assembly.

1. If needed, check what is currently the main object of the rebar assembly.

   Use the **Inquire** tool to check which objects belong to a particular rebar assembly.
2. Ensure that the **Select objects in assemblies** or **Select objects in components** selection switch is active.
3. Select the new main object.

4. Right-click and select **Assembly** --> **Set as new main object of assembly**.

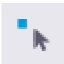
   Tekla Structures changes the main object.

Alternatively, you can use the **Rebar** --> **Assembly** --> **Set as main object** command on the ribbon.

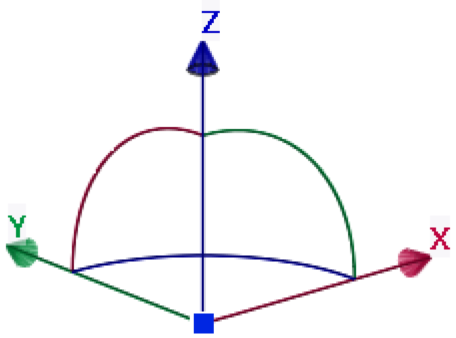
Note that if you remove the main object from a rebar assembly, the remaining reinforcement object with the highest weight is automatically set as the new main object.

Change the coordinate system of a rebar assembly

By default, the main reinforcement object of a rebar assembly sets the local coordinate system of the rebar assembly. The coordinate system then defines the orientation, length, width, and height of the rebar assembly for drawings and reports. If a rebar assembly is not shown the way you want in drawings and reports, even though you changed the main object, you can adjust the coordinate system of the rebar assembly.

1. Ensure that the  **Direct modification** switch is active.
2. Select a rebar assembly.
3. On the contextual toolbar, click  **Define or remove coordinate system**.

   Tekla Structures shows a blue direct modification handle at the origin, and the red, green, and blue axis handles and rotation handles that you can use to adjust the coordinate system of the rebar assembly.



4. To move the coordinate system to a new location, drag the  handle that is located at the origin.
5. To move the coordinate system along any of the coordinate axes, drag the relevant axis handle to a new location.

6. To rotate the coordinate system around any of the coordinate axes, drag the relevant rotation handle to a new location.

   Press **Tab** to rotate the coordinate system in 90-degree steps in the direction of the selected rotation handle.

7. To move or rotate the coordinate system by specifying a distance or angle:

   a. Select an axis handle or a rotation handle.

   b. Type the value by which you want the handle to move.

      When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog box.

   c. Click **OK** or press **Enter** to confirm the value.

8. To revert to the original coordinate system, click again on the contextual toolbar.

Remove objects from a rebar assembly

If you want to disassemble a rebar assembly, you can remove objects from it, or ungroup the objects belonging to the rebar assembly.

Remove an object from a rebar assembly

You can remove reinforcement objects or sub-assemblies from a rebar assembly one by one.

1. Select the object that you want to remove.
2. Right-click and select **Assembly** --> **Remove from assembly**.

Alternatively, you can use the **Rebar** --> **Assembly** --> **Remove from assembly** command on the ribbon.

To remove another object, restart the command.

Explode a rebar assembly or a sub-assembly

You can explode a rebar assembly back to single reinforcement objects, or just ungroup the objects belonging to a sub-assembly.

When you explode a nested rebar assembly, always start from the highest assembly hierarchy level. Tekla Structures breaks the assembly hierarchy level by level and ungroups the objects that belong to the highest level of the rebar assembly.

You can also explode sub-assemblies to single reinforcement objects without breaking the entire assembly hierarchy. The ungrouped sub-assembly objects remain within the higher-level, parent rebar assembly.

1. Ensure that the **Select assemblies** selection switch is active.

2. Select the rebar assembly or sub-assembly that you want to explode.

   Ensure that you are on the intended assembly hierarchy level.

3. Do one of the following:

   - To explode a basic rebar assembly, or the highest level of a nested rebar assembly, right-click and select **Assembly** --> **Explode**.

     Use the **Explode** command several times to break a hierarchical nested rebar assembly back to single reinforcement objects.

   - To explode a sub-assembly, right-click and select **Assembly** --> **Explode Sub-Assembly**.

Alternatively, you use the **Explode assembly** and **Explode sub-assembly** commands on the ribbon, under **Rebar** --> **Assembly**.

2.12 Create construction objects and points

Points and construction objects help you to place other objects in the model.

When you want to place objects to a position where no lines or objects intersect in the model, you can create construction lines (page 622), planes (page 622), circles (page 623), arcs (page 624), and polycurves (page 626). For example, you can easily pick (page 83) the points at intersections of construction lines and circles. The snap priority of construction objects is the same as with the other lines.

Use the display settings (page 647) to control if points and construction objects are visible in model views. Points and construction objects remain in the model when you update or redraw views and windows. They do not appear in drawings.

Use the following selection switches to control if you can select points and construction objects in the model:

, , 

You can also create magnetic construction lines or planes to bind and move groups of objects (page 859). For example, rather than binding lots of handles and chamfers to part faces, simply create a construction plane that goes through all the handles and chamfers. Then make this plane magnetic and bind the plane to the appropriate face. When you move the plane, the attached handles and chamfers move with it.

See also

Create a construction line (page 622)

Create a construction plane (page 622)

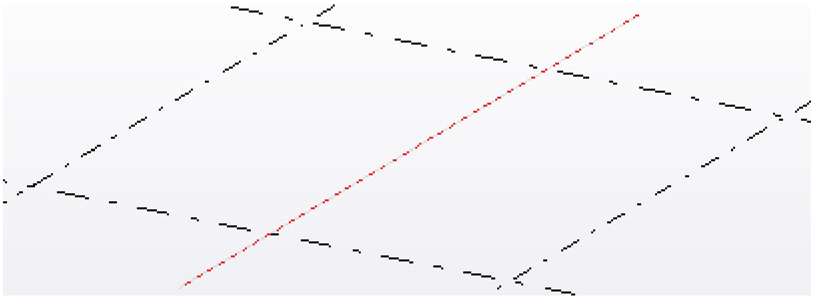
Create a construction circle (page 623)

[Create a construction arc (page 624)](#)
Create a construction polycurve (page 626)
Copy a construction object with offset (page 627)
Modify a construction object (page 628)
Create points (page 631)

Create a construction line

1. On the **Edit** tab, click **Construction object** --> **Line** .
2. Pick the start point of the construction line.
3. Pick the end point of the construction line.

   Tekla Structures creates the line.

4. To end the command, press **Esc**.
5. To modify the construction line properties, double-click the line in the model.

   The line properties are shown in the property pane.

   a. If you want to make the line magnetic, select **Yes** in the **Magnetic** list.
   b. Select a color for the line.
   c. Define how far the line extends beyond the picked points.
   d. Select a line type for the line.
   e. Click **Modify** .

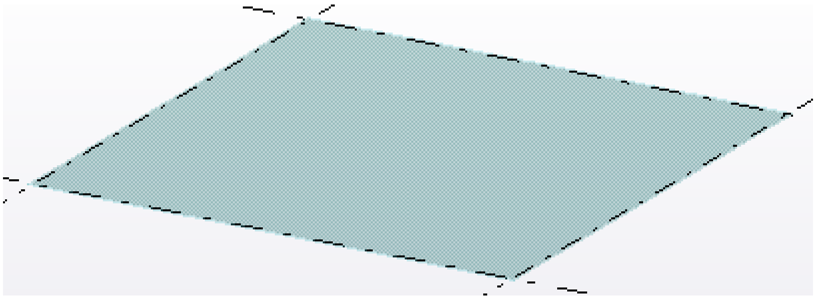
See also

Create construction objects and points (page 621)
Copy a construction object with offset (page 627)
Modify a construction object (page 628)

Create a construction plane

1. On the **Edit** tab, click **Construction object** --> **Plane** .
2. Pick three points.
3. Click the middle mouse button.

   Tekla Structures draws the plane.

4. To end the command, press **Esc**.
5. To modify the construction plane properties, double-click the plane in the model.

   The plane properties are shown in the property pane.

   a. Enter a name for the plane.

   b. If you want to make the construction plane magnetic, select **Yes** in the **Magnetic** list.

   c. Click **Modify**.

See also

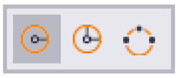
Create construction objects and points (page 621)

Modify a construction object (page 628)

Create a construction circle

You can create construction circles that are parallel to the view plane by picking two points, or you can create construction circles by picking three points in the 3D space in the model.

1. On the **Edit** tab, click **Construction object** --> **Circle** .
2. On the contextual toolbar that appears, click a button to specify which set of points you want to pick:

- Click , and then pick the center point and a point to define the radius of the circle.

  Tekla Structures creates the circle parallel to the view plane.

- Click , and then pick three points: the center point, a point to define the radius, and a point to define the plane of the circle.

- Click , and then pick three points along the arc of the circle.

Tekla Structures creates the circle using the points you picked and using the current properties. Tekla Structures also indicates the center point of the circle with an X in the model.

3. To end the command, press **Esc**.
4. To modify the construction circle properties, double-click the circle in the model.

   The circle properties are shown in the property pane.

   a. Select a color for the circle.

   b. Select a line type for the circle.

   c. Click **Modify**.

See also

Create construction objects and points (page 621)

Copy a construction object with offset (page 627)

Modify a construction object (page 628)

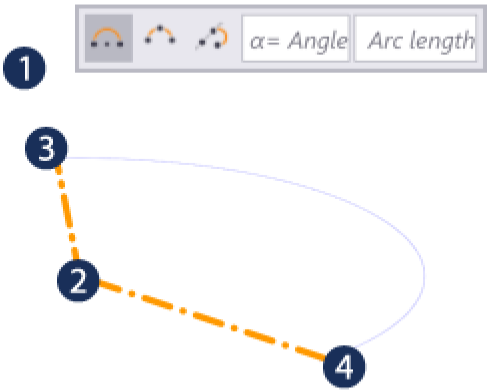
Create a construction arc

You can create construction arcs by picking three points in the 3D space in the model.

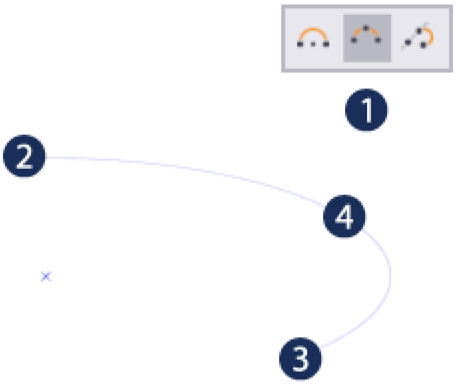
1. On the **Edit** tab, click **Construction object** --> **Arc** .
2. On the contextual toolbar that appears, click a button to specify which set of points you want to pick:

   - Click , and then pick three points: the center point, start point, and end point of the arc.

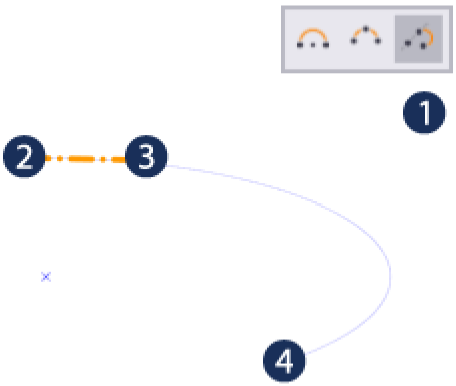
     You can also define the angle or the length of the arc.



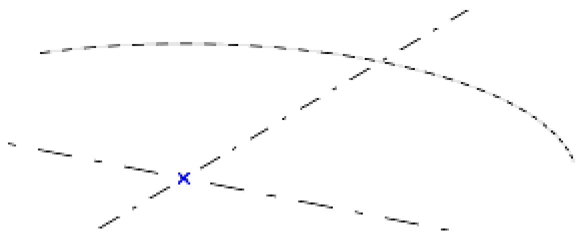
- Click , and then pick the start point, end point, and a point along the arc.



- Click , and then pick a point to define a tangent, and two points along the arc.



Tekla Structures creates the arc using the points you picked and using the current properties. Tekla Structures also indicates the center point of the arc with an X in the model.



3. To end the command, press **Esc**.
4. To modify the construction arc properties, double-click the arc in the model.

   The arc properties are shown in the property pane.

   a. Select a color for the arc.
   b. Select a line type for the arc.
   c. Click **Modify**.

See also

Create construction objects and points (page 621)

Copy a construction object with offset (page 627)

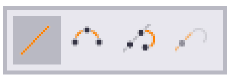
Modify a construction object (page 628)

Create a construction polycurve

You can create 3D construction polycurves that pass through the points you pick and that can have straight and curved segments.

1. On the **Edit** tab, click **Construction object** --> **Polycurve** .
2. On the contextual toolbar that appears, click a button to specify which set of points you want to pick to create a polycurve segment.

   You can switch between these picking modes each time you complete a segment.

   - For a straight segment, click  and then pick the start point and the end point of the segment.
   - For a curved segment, click  and then pick three points along the segment.

- For a curved tangential segment, click  and then pick a point on the tangent line, the start point, and the end point of the segment.
- For a straight segment that is tangential to the previous segment, click  and then pick a point on the tangent line.

Tekla Structures creates a polycurve segment.

3. Repeat step 2 for each polycurve segment that you want to create, but skip picking the first point of the segment as it is the same as the last point of the previous segment.
4. Click the middle mouse button to finish picking.

   Tekla Structures creates the polycurve through the points you picked, using the polycurve's current properties.
5. To end the command, press **Esc**.
6. To modify the construction polycurve properties, double-click the polycurve in the model.

   The polycurve properties are shown in the property pane.

   a. Select a color for the polycurve.

   b. Select a line type for the polycurve.

   c. Click **Modify** .

See also

Create construction objects and points (page 621)

Copy a construction object with offset (page 627)

Modify a construction object (page 628)

Copy a construction object with offset

You can copy construction lines, circles, arcs, and polycurves in the direction that you indicate, and using the offset values that you specify. For example, you can create new circles and arcs centered in the same location as the original circle or arc, and adjust the radiuses using the offset values.

1. On the **Edit** tab, click **Construction object** --> **Copy with offset** .
2. Select the construction object that you want to copy.

   You can copy lines (page 622), circles (page 623), arcs (page 624), and polycurves (page 626).
3. Enter the offset values in the box that appears, and then press **Enter**.

If you only enter one offset value, Tekla Structures creates one copy of the object.

To create several copies, enter multiple offset values. For example, `500 1000 1500`, or `4*800`.

4. Click in the direction where you want to copy the object.

Tekla Structures copies the selected object in the direction you indicated.

For example, if you selected a line, Tekla Structures makes a new copy of the line in the specified location. If you selected a circle or arc, Tekla Structures creates a new object that is centered in the same location as the original object, and adjusts the radius using the offset value that you specified.

See also

Create construction objects and points (page 621)

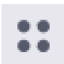
Modify a construction object (page 628)

Modify a construction object

You can modify construction points, lines, circles, arcs, polycurves, and planes using direct modification.

Before you start:

- To show (or hide) construction objects in model views, adjust their visibility (page 647) in the **Display** settings.
- Ensure that the  **Direct modification** switch is active.
- Select the construction object. Use the following selection switches:

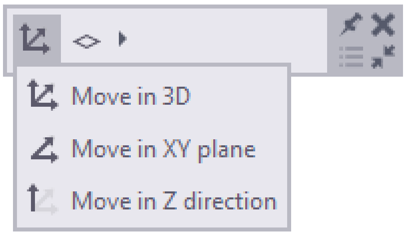
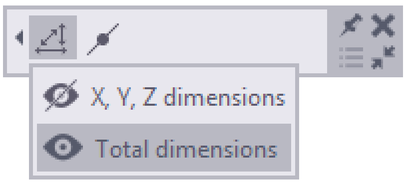
  , , 

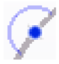
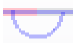
  Tekla Structures displays the handles and dimensions that you can use to modify the construction object.

  When you select a handle and move the mouse pointer over , Tekla Structures displays a toolbar with more modification options. The available options depend on the type of the construction object you are modifying.

To modify a construction object, do any of the following:

To	Do this	Available for
Set a reference point to move in one, two, or any direction	1. Select the handle in the reference point.	Construction points, lines, circle center points, planes

To	Do this	Available for
	2. To define in which directions the handle can move, select an option from the list on the toolbar:  You can also press **Tab** to cycle through the options. 3. To move the handle parallel to a certain plane only, click and select the plane.	
Move a point, a point on a line, circle, arc or polycurve, or a plane corner	Drag the handle in the reference point to a new location.	All construction objects
Move a circle or arc	Drag the handle in the center point to a new location.	Construction circles, arcs
Move a line or a plane edge	Drag the line handle to a new location.	Construction lines, planes
Move a plane	Drag the plane to a new location.	Construction planes
Show or hide diagonal dimensions	1. Select a handle. 2. On the toolbar, click . 3. Click the eye button to show or hide orthogonal and total dimensions:  	Construction lines, planes
Change a dimension	Drag a dimension arrowhead to a new location, or: 1. Select the dimension arrowhead which you want to move.	Construction lines, circles, planes Arcs (numeric input only)

To	Do this	Available for
	To change the dimension at both ends, select both arrowheads. To change the radius of a circle or arc, select the outer arrowhead. 2. Using the keyboard, enter the value with which you want the dimension to change. To start with the negative sign (-), use the numeric keypad. To enter an absolute value for the dimension, first enter $, then the value. 3. Press **Enter**, or click **OK** in the **Enter a Numeric Location** dialog box.	
Chamfer a polycurve corner	1. Select a corner handle. 2. On the toolbar: • Click  to create a rounded chamfer, and then enter the chamfer radius. • Click  to create a straight chamfer, and then enter the chamfer dimensions X and Y. 3. Press **Enter** to confirm the chamfer dimensions.	Polycurves
Change an arc to a line Make a curved segment straight	Select the arc or segment midpoint handle (with an arc symbol)  and press **Delete**.	Arcs, curved polycurve segments
Change a line to an arc Make a straight segment curved	Drag the arc symbol  at the midpoint of the line or segment.	Lines, straight polycurve segments
Add a corner point and an intermediate segment to a polycurve	Drag a segment midpoint handle to a new location.	Polycurves
Remove a corner point and the two	Select the corner point handle and press **Delete**.	Polycurves

To	Do this	Available for
connected segments		
Remove the last polycurve segment	Select the end point handle and press **Delete**.	Polycurves
Change the radius of an arc and keep the end point locations	Click the radius dimension, enter a new value, and press **Enter**.	Arcs
Change the radius of a polycurve segment and keep the end point locations	Drag the arc symbol at the segment midpoint handle.	Curved polycurve segments
Change the angle or length of an arc	Drag the start or end point to a new location.	Arcs
Copy a construction object using offset	See Copy a construction object with offset (page 627).	Lines, circles, arcs, polycurves
Change the modeling direction of the selected construction object	On the contextual toolbar, click **Swap ends**. This may be needed when you create lofted plates (page 247) or lofted slabs (page 285) by using construction objects and if the geometry of the plate or slab would become self-intersecting.	Lines, arcs

See also

Create construction objects and points (page 621)

Create points (page 631)

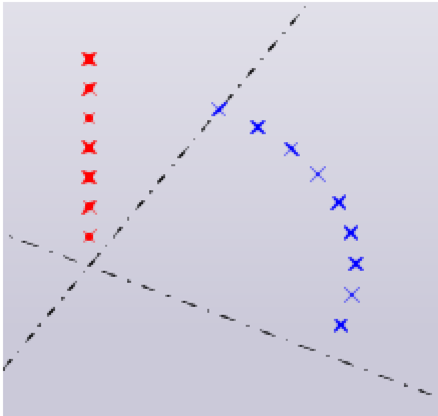
Resize and reshape model objects (page 107)

Create points

You can create points to make it easier to place model objects at positions where no lines or objects intersect.

There are many ways to create points in Tekla Structures. Which method is the most convenient at each time depends on what you have already created in the model and which locations you can easily pick.

When you create points, Tekla Structures always places them according to the work plane coordinate system. Points located on the view plane are by default blue and points outside the view plane are by default red. You can change the color of points in the point properties.



Use the display settings (page 647) to control if points are visible in model views, and to adjust the point size if needed.

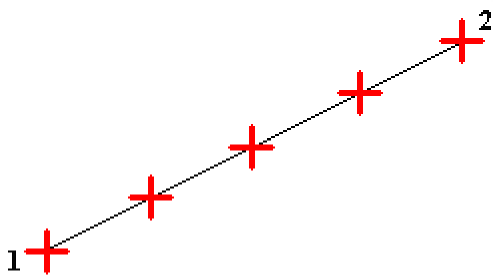
Create points on a line

You can create points at equal intervals along a line that is defined by two points.

1. On the **Edit** tab, click **Points** --> **On line** .

   The **Divided Line Points** dialog box appears.

2. Define the number of points to be created.
3. Click **OK**.
4. Pick the start point of the line (1).
5. Pick the end point of the line (2).



Create points on a plane

You can create several equally spaced points in the desired area in the model. The points are created in relation to the picked origin position.

A point array consists of several points in a rectangular xy(z) pattern relative to the current work plane. The x, y, and z coordinates of the points define the array pattern. The x and y coordinates are relative distances between the points on the work plane. The z coordinates are absolute distances perpendicular to the work plane.

1. On the **Edit** tab, click **Points** --> **On plane** .

   The **Point Array** dialog box appears.

2. Define the array point coordinates.

   Use positive or negative values to define the direction of the array.

   Use a zero at the beginning of the row to represent a point in the array origin. Separate multiple values with spaces. For example, `0 4000 4000`, or `0 5*1000`.

3. Pick the origin of the array in the view.

   Alternatively, you can define the origin in the **Point Array** dialog box.

4. Click **Create**.

Create points parallel to two points

You can create offset points that are parallel to a line between two points you have picked.

1. On the **Edit** tab, click **Points** --> **Parallel to two points** .

   The **Point Input** dialog box appears.

2. Define the distances at which the points are created.

   If you want to create multiple pairs of offset points, enter multiple values separated with spaces.

3. Click **OK**.

4. Pick the start point of the line (1).

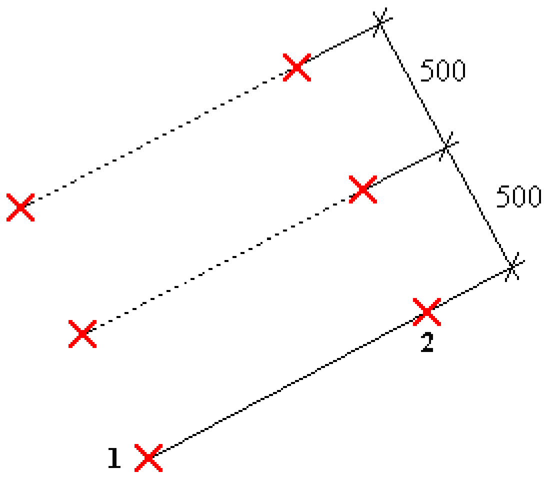
5. Pick the end point of the line (2).

   The picking order of the start point and the end point defines the offset direction of the new points.

   When you look from the start point to the end point, Tekla Structures creates the new points to the left of the picked points. If you enter negative values to the **Point Input** dialog box, Tekla Structures creates points to the right of the picked points.

When you pick points, Tekla Structures uses arrows to indicate the offset direction.

For example, if you enter `500 500` to the **Point Input** dialog box, the first pair of new points is created at a 500 mm distance from the picked points, and the second pair of points is created at a 500 mm distance from the first pair of points.



Create points along the extension line of two points

1. On the **Edit** tab, click **Points** --> **Along extension of two points** .

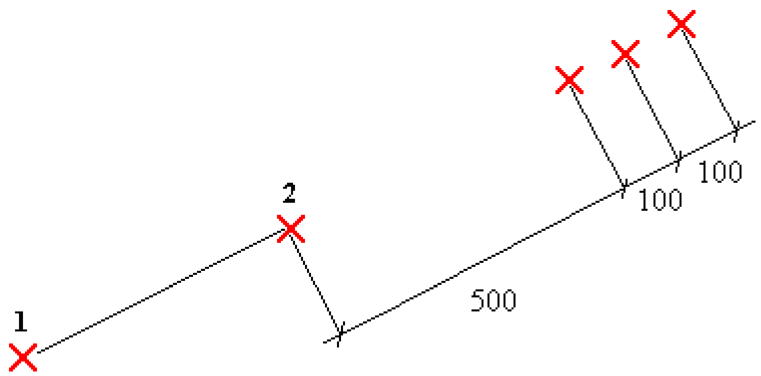
   The **Point Input** dialog box opens.

2. Define the distances at which the points are created.

   Separate multiple values with spaces.

3. Click **OK**.
4. Pick the start point of the line (1).
5. Pick the end point of the line (2).

   For example, if you enter `500 100 100` to the **Point Input** dialog box, the first point is created at 500 mm distance from the end point of the line, and the second and the third point are each created at 100 mm distance from the previous point.

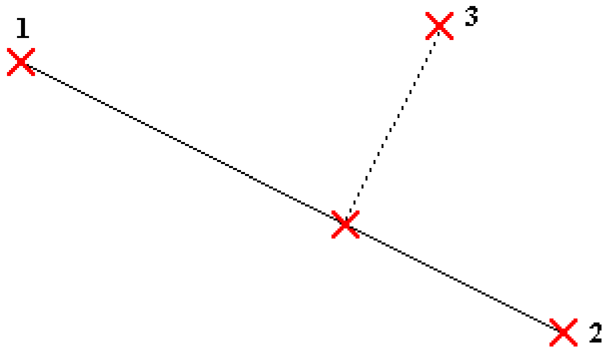


TIP Enter a negative value to the **Point Input** dialog box to create a point between the start point and the end point.

Create projected points on a line

You can project a point onto a selected line or its extension.

1. On the **Edit** tab, click **Points** --> **Projected points on line** .
2. Pick the first point on the line (1).
3. Pick the second point on the line (2).
4. Pick the point to be projected (3).



Create points along an arc using center and arc points

You can create points along an arc.

1. On the **Edit** tab, click **Points** --> **Along arc using center and arc points** .

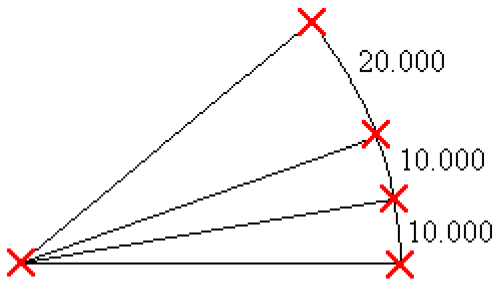
   The **Arc Points** dialog box appears.
2. Select either **Angles** or **Distances** and enter the angles or distances between the points along the arc.

   Give the angle values in degrees.

   Separate multiple angle and distance values with spaces.

3. Click **OK**.
4. Pick the center point.
5. Pick the start point of the arc.

   Tekla Structures creates the arc points counterclockwise from the start point.



Create points along an arc using three arc points

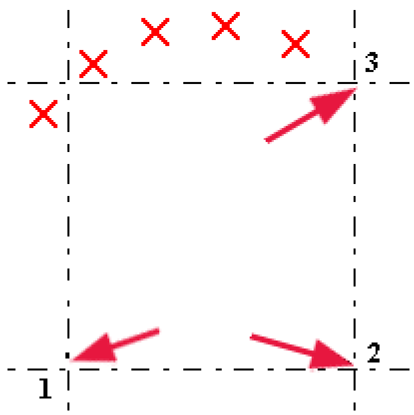
You can create points as an extension of an arc.

1. On the **Edit** tab, click **Points** --> **Along arc using three arc points** .

   The **Arc Points** dialog box appears.
2. Select either **Angles** or **Distances** and enter the angles or distances between the points along the arc.

   Give the angle values in degrees.

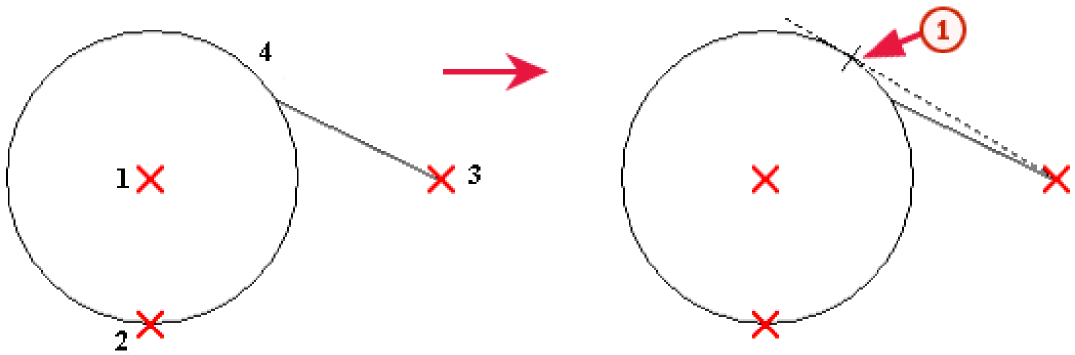
   Separate multiple angle and distance values with spaces.
3. Click **OK**.
4. Pick three points along the arc (1-3).



Create points tangent to a circle

You can create a point along the tangent of a circle.

1. On the **Edit** tab, click **Points** --> **Tangent to circle** .
2. Pick the center point of the circle (1).
3. Pick a point on the circle to define the radius (2).
4. Pick the end point of the tangent (3).
5. Pick a side to indicate the side on which Tekla Structures creates the tangent point (4).



(1): Tangent point

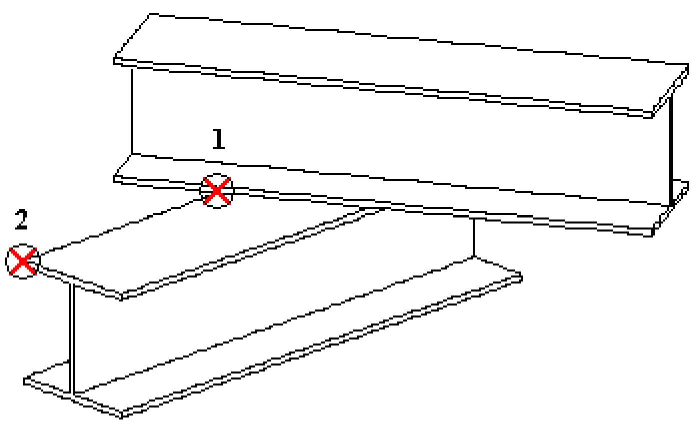
Create points at any position

You can create a point at the position you pick.

NOTE Snap switches (page 84) determine the positions you can pick.

You can also use temporary reference points and numeric snapping to create a point, for example, to a certain distance from an existing corner or a point.

1. On the **Edit** tab, click **Points** --> **At any position** .
2. Pick the intersection of two part edges (1), or the corner of a part (2).



Create bolt points

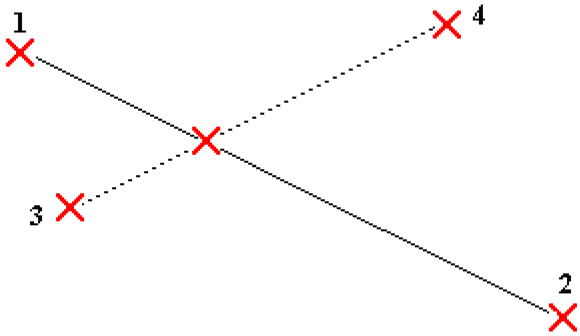
You can create points on the view plane at the center points of single bolts and bolt group bolts.

1. On the **Edit** tab, click **Points** --> **Bolt points** .
2. Select a bolt or a bolt group.

Create points at the intersection of two lines

You can create a point at the intersection of two lines in the view plane. The lines are treated as infinite length. The extensions of the lines must intersect at some point.

1. On the **Edit** tab, click **Points** --> **At intersection of two lines** .
2. Pick the start point of the first line (1).
3. Pick the end point of the first line (2).
4. Pick the start point of the second line (3).
5. Pick the end point of the second line (4).



Create points at the intersection of a plane and a line

You can create a point where a line intersects with a plane.

1. On the **Edit** tab, click **Points** --> **At intersection of plane and line** .
2. Pick three points to define the plane.
3. Pick the first point of the line.
4. Pick the second point of the line.

Create points at the intersection of a part and a line

You can create points where a line intersects with the surface of a part.

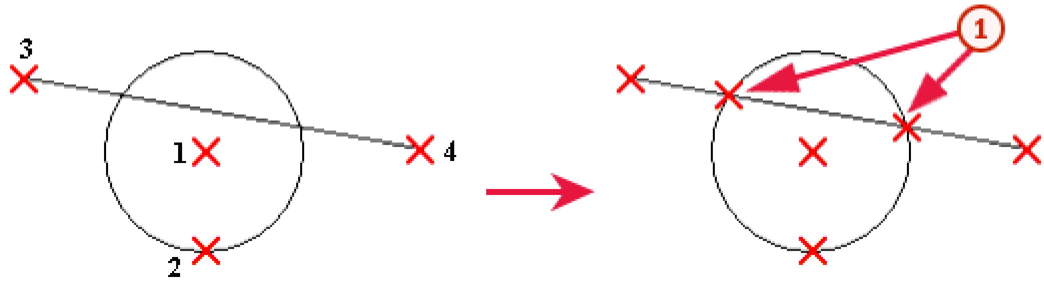
1. On the **Edit** tab, click **Points** --> **At intersection of part and line** .
2. Select the part.

3. Pick the first point on the line.
4. Pick the second point on the line.

Create points at the intersection of a circle and a line

You can create points where a circle and a line intersect.

1. On the **Edit** tab, click **Points** --> **At intersection of circle and line** .
2. Pick the center point of the circle (1).
3. Pick a point on the circle to define the radius (2).
4. Pick the first point on the line (3).
5. Pick the second point on the line (4).



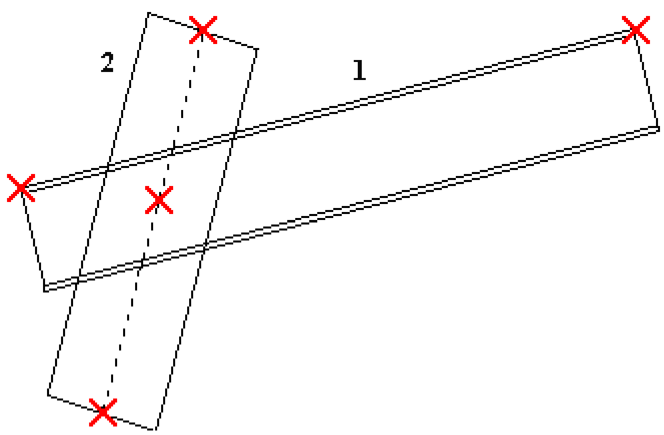
(1): New points

Create points at the intersection of two part axes

You can create points where the axes of two parts intersect, and project the points onto the axis of the part that you select first.

1. On the **Edit** tab, click **Points** --> **At intersection of two part axes** .
2. Select the first part (1).
3. Select the second part (2).

   Tekla Structures projects the point onto the axis of the first part.



Import points

NOTE This is for advanced users.

You can import points to specific locations in an open Tekla Structures model using the **Point Creation Import (8)** component. You need to specify the point coordinates in a text file. In some cases this file is generated by another software package.

1. Create a point import file.

   a. Create a text file that consists of single lines for each point.

      Use commas or tabs as delimiters for the three point coordinates on a line. For example:

      `100,500,1000`

      `300,700,1500`

   b. Save the file.

   NOTE During the import process, Tekla Structures ignores all lines in the import file which do not consist of valid values delimited by tabs or commas.

2. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
3. Enter `point` in the **Search...** box.
4. Click **Point Creation Import (8)**.
5. Enter the ASCII file name.

   Include the full path and the file name extension (for example `.txt`). If you do not specify the path, Tekla Structures looks for the file in the current model folder.
6. Define the origin of the imported points by entering the coordinates.
7. Click **Create**.

Point properties

Use the **Point** properties to view and modify the properties of a point.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	

Setting	Description
Color	Change the color of the points. Note that if you change the point color, the changed color is not applied the next time you create points. The points are created using the default point color.
Location	
X **Y** **Z**	The local (work plane) and global x, y, and z coordinates of a point. Indicates the correct location of the point.

TIP You can change the point size in Adjust the display settings (page 647).

3 Adjust how model objects are displayed

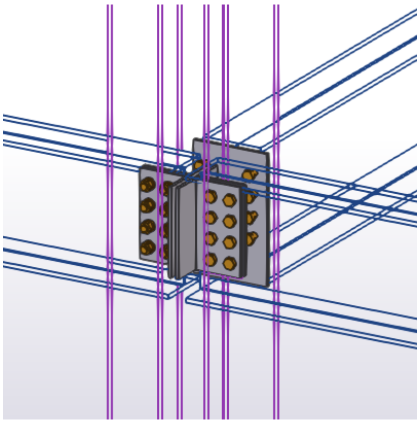
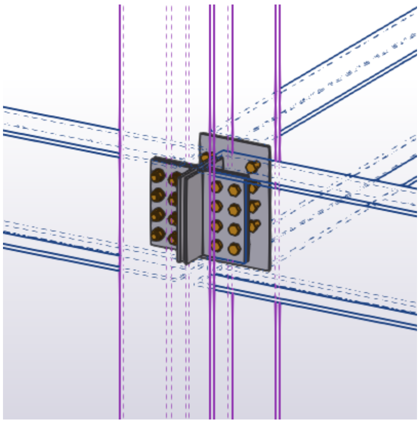
You can use different methods to adjust how model objects are displayed:

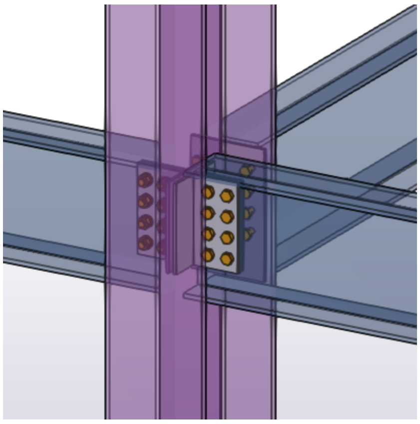
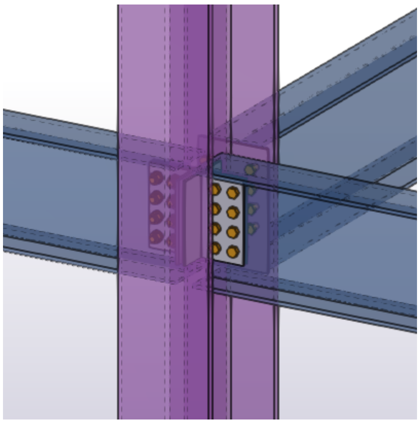
- To change the rendering of parts and components and select whether the object surfaces or the object outlines are displayed in the views, see Change the rendering of parts, components and reference models (page 642).
- To change the general display settings, Adjust the display settings.
- You can temporarily change the part representation to show parts with exact lines or with high accuracy as explained in Change the part representation to show parts with exact lines or with high accuracy.
- You can temporarily hide selected objects or show only the selected objects as explained in Temporarily hide model objects or show only selected model objects (page 652).
- Detailed instructions for changing model object color and transparency can be found in Change the color and transparency of model objects by using object representation.
- You can group model objects using various criteria (for example, profile) to handle them as a single unit when you define display settings, see Use object groups in object representation and in filters (page 662).

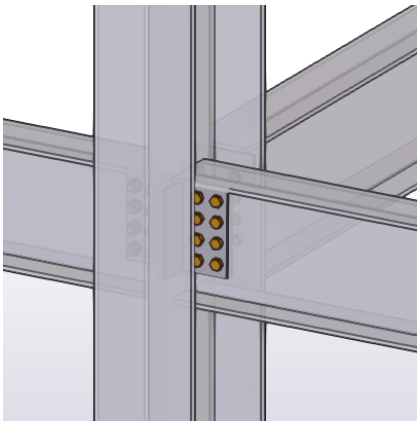
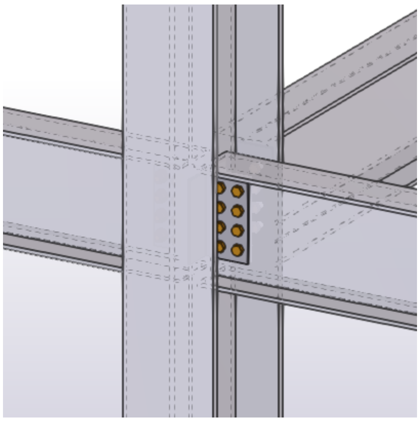
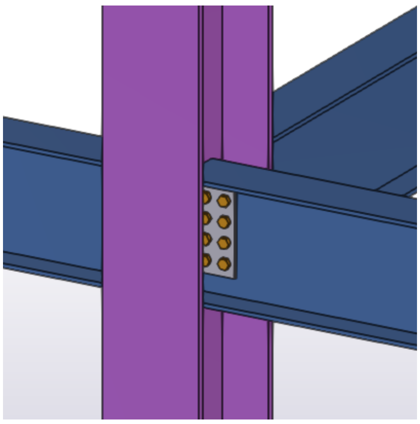
3.1 Change the rendering of parts, components and reference models

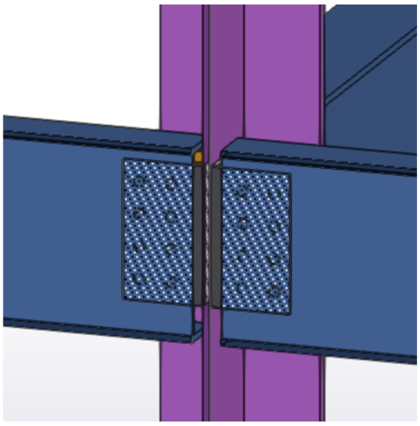
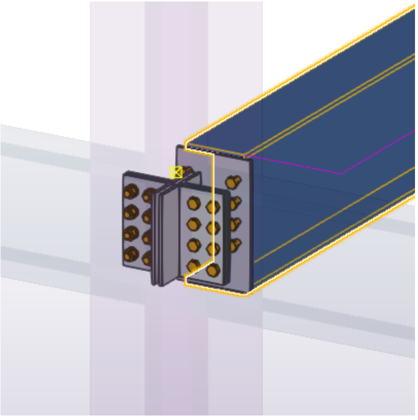
You can easily change the rendering of parts, components and reference models in model views.

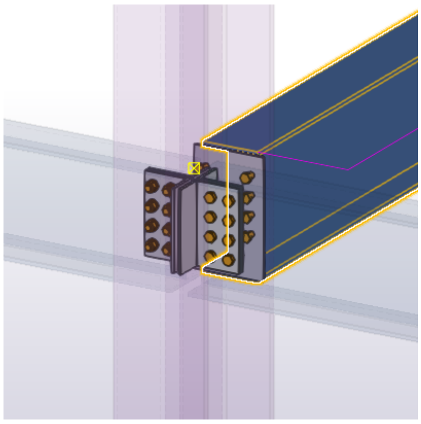
1. On the **View** tab, click **Rendering**.
2. Select one of the rendering options for parts, components, or reference models:

<table>
<tr><th>Option</th><th>Description</th><th>Example</th></tr>
<tr><td rowspan="2">Parts wireframe/ Components wireframe/ References wireframe</td><td>Object outlines are displayed, surfaces are not. Objects are transparent.</td><td>In this example, component objects are displayed as rendered.
</td></tr>
<tr><td>In DirectX rendered views, the part edge lines hidden behind another part are visualized with dashed lines if you have switched the Dashed line for hidden line option on in File menu --> Settings --> Switches.</td><td>In this example, component objects are displayed as rendered.
</td></tr>
</table>

Option	Description	Example
Parts shaded wireframe/ Components shaded wireframe/ References shaded wireframe	Object outlines are displayed. Objects are transparent, and their surfaces are rendered.	In this example, component objects are displayed as rendered. 
	In DirectX rendered views, the part edge lines hidden behind another part are visualized with dashed lines if you have switched the **Dashed line for hidden line** option **on** in **File menu** --> **Settings** --> **Switches**.	In this example, component objects are displayed as rendered. 

Option	Description	Example
Parts grayscale/ Components grayscale/ References grayscale	Objects are shown in grayscale.	In this example, component objects are displayed as rendered. 
	In DirectX rendered views, the part edge lines hidden behind another part are visualized with dashed lines if you have switched the **Dashed line for hidden line** option **on** in **File menu** --> **Settings** --> **Switches**.	In this example, component objects are displayed as rendered. 
Parts rendered/ Components rendered/	Object surfaces are displayed. Objects are not transparent.	

Option	Description	Example
References rendered	In DirectX rendered views, any duplicate objects or overlapping parts on the same plane are visualized with a hatch if you have switched the **Hatching of overlapping surfaces** option **on** in **File menu** --> **Settings** --> **Switches**.	In this example, the overlapping surfaces are visualized with a hatch. 
Show only selected part/Show only selected component/ Show only selected reference	Selected objects are displayed. Other objects are almost completely transparent. This option is useful, for example, when viewing clash check results in a large model.	In this example, component objects are displayed as rendered. 

Option	Description	Example
	In DirectX rendered views, the part edge lines hidden behind another part are visualized with dashed lines if you have switched the **Dashed line for hidden line** options **on** in **File menu** --> **Settings** --> **Switches**.	In this example, component objects are displayed as rendered. 

TIP Alternatively, you can use the keyboard shortcuts **Ctrl+1...5** for parts and **Shift +1...5** for components to switch between the rendering options. **Ctrl+Shift+1...5** works for references.

See also

Adjust the display settings (page 647)

Change the rendering mode for model views (page 72)

3.2 Adjust the display settings

Adjust the display settings to define how parts and other model objects appear in model views.

Set the visibility and representation of model objects in the Display settings

You can define the visibility and representation separately for different types of model objects.

1. Double-click the view to open the **View Properties** dialog box.

   Alternatively, on the **View** tab, click **View properties**.

2. Click the **Display...** button to open the **Display** dialog box.

3. Select or clear check boxes to specify which objects are visible in the view. You can specify separately the object visibility for the objects in model and for the objects in components.

4. Select a representation option for parts, bolts, holes, welds, construction planes and reinforcing bars.

   You have the following options:

   - **Fast**
   - **Exact**
   - **Reference line** (only for parts)
   - **Exact slotted holes** (only for holes)
   - **Exact - no weld mark** (only for welds)

5. If you are working with cast-in-place (page 447) concrete structures, and the pour management functionality is enabled (page 445):

   a. In the **Cast in place** list, select whether you want to show the structures as **Parts** or as **Pours**.

   b. If you selected **Parts** for the cast-in-place concrete structures, select whether you want to show the parts as **Merged** or as **Separated**.

6. Ensure that the view is selected.

7. Click **Modify** to apply the changes.

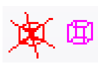
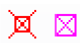
NOTE To quickly modify the visibility of objects in model and in components, use the contextual toolbar. Click the view, and use the eye icon on the contextual toolbar to set the visibility.

Display settings

Note that some of these settings may affect system performance.

Option	Description
Settings	
Parts	Defines how parts are displayed. **Fast** uses a rapid drawing technique that displays internal hidden edges, but skips cuts. The setting does not automatically affect already modeled parts. When you switch this setting on, the fast representation mode will be applied only to newly created parts and to parts that are displayed with the **Show with exact lines** command. **Exact** displays the cuts, but hides the internal hidden lines of parts.

Option	Description
	Reference line shows parts as sticks (page 309). This option increases display speed significantly, when viewing the entire model, or large parts of it. Cast-in-place concrete structures can be displayed as **Pours**, or as **Parts** that can be **Merged** or **Separated**. For more information, see View cast-in-place concrete structures (page 447).
Bolts	Defines how bolts are displayed. **Fast** displays the axis and a cross to represent the bolt head. This is the recommended representation mode for bolts, because it increases display speed significantly and consumes less system memory. **Exact** shows bolts, washers, and nuts as solid objects.
Holes	Defines how holes are displayed. **Fast** only displays the circle in the first plane. When using this option, Tekla Structures always displays fast holes on the first part (counting from the head of the bolt). If there are slotted holes in any of the parts, a slotted hole is displayed on the first part, even if the hole in that part is not slotted. The new slotted hole has the same size and rotation as the first slotted hole (counting from the head of the bolt). Holes that are outside a part are always displayed as fast holes. **Exact** shows holes as solid objects. **Exact slotted holes** only displays slotted holes in exact mode and ordinary holes in fast mode.
Welds	Defines how welds are displayed. **Fast** displays a symbol for welds. **Exact** shows welds as solid objects and displays the weld symbols. When

Option	Description
	you select welds, the weld marks are displayed. **Exact - no weld mark** shows welds as solid objects but does not display the weld symbols, nor the weld marks when you select welds. For more information, see Set the visibility and appearance of welds (page 371).
Construction planes	Defines how construction planes are displayed.
Reinforcing bars	Defines how reinforcement objects are displayed. **Fast** displays the shape of reinforcement meshes using an outline polygon and a diagonal line. Single reinforcing bars and bar groups are displayed as solid objects. **Exact** shows reinforcing bars, bar groups, and reinforcement meshes as solid objects.
Advanced	
Part label	See Show part information by using part labels (page 324).
Point size	Defines the size and appearance of points in views. Also affects the size and appearance of the handles, together with XS_HANDLE_SCALE. **In model** increases the point size on the screen when you zoom in. Shows points and handles as 3D cubes:  **In view** does not increase the point size. Shows points and handles as flat 2D objects: 

3.3 Change the part representation to show parts with exact lines or with high accuracy

Even if you have set the part representation to **Fast** in the display settings, you can temporarily view the parts using different representation options.

Show parts with exact lines

Use the **Show part with exact lines** command to temporarily display a part with exact lines even if you are using the **Fast** representation option for parts.

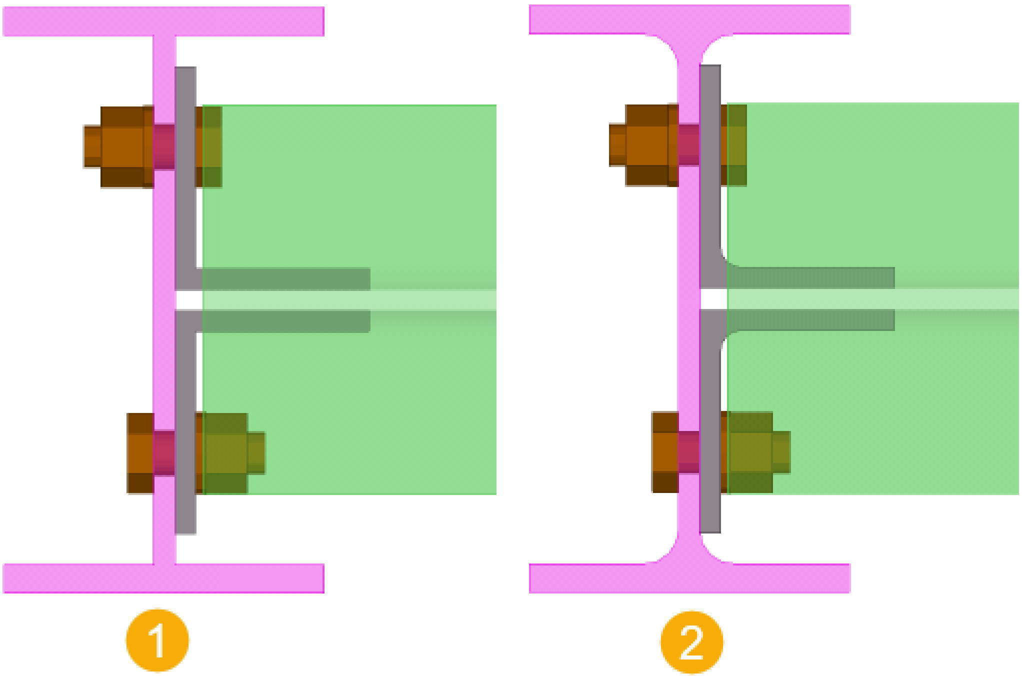
1. Select the part.
2. Go to **Quick Launch**, start typing `show part with exact lines`, and select the **Show part with exact lines** command from the list that appears.
3. Click the view in which you want to display exact lines.
4. To clear the exact lines effect, on the **View** tab, click .

Show parts with high accuracy

You can temporarily display parts with the highest possible level of accuracy. This can be useful, for example, when checking a large model, because the entire model can still be displayed in the **Fast** or **Exact** representation mode but the individual part can be shown in more detail.

1. Select the parts.
2. Right-click, and then hold down the **Shift** key while selecting **Show with exact lines**.

   Tekla Structures displays the selected parts with the highest possible level of accuracy.
3. To clear the high accuracy effect, right-click and select **Show with exact lines**.



(1) Normal display mode

(2) High accuracy mode

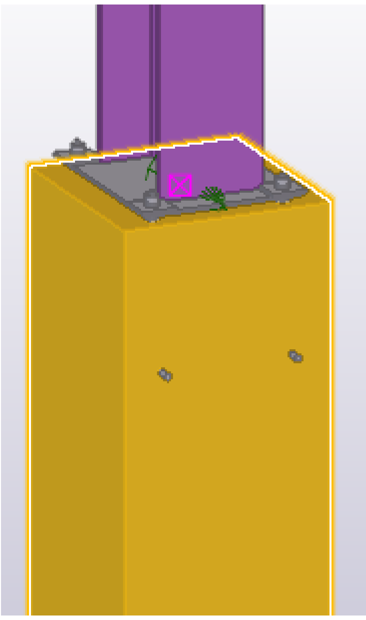
3.4 Temporarily hide model objects or show only selected model objects

You can temporarily show, hide, or show hidden model objects and assemblies in model views.

Hide parts or other objects in a model view

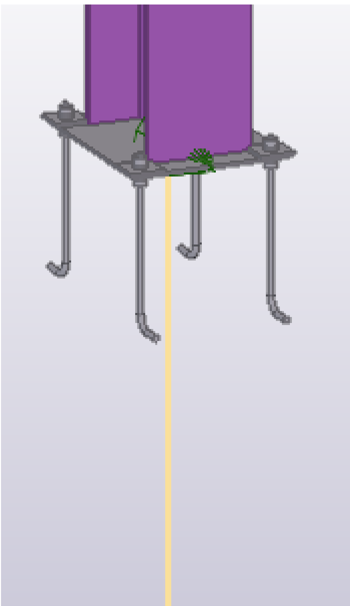
You can quickly hide selected parts or other objects in a model view. This can be useful, for example, when you want to temporarily hide some parts to see the parts behind them.

1. Select the parts or objects you want to hide.



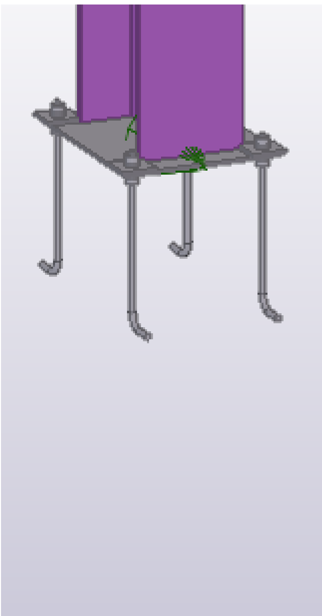
2. Right-click and select **Hide**.

   Tekla Structures shows the hidden parts as part reference lines.



   To completely hide the selected parts, hold down the **Shift** key when selecting the command.

   The selected parts become invisible.



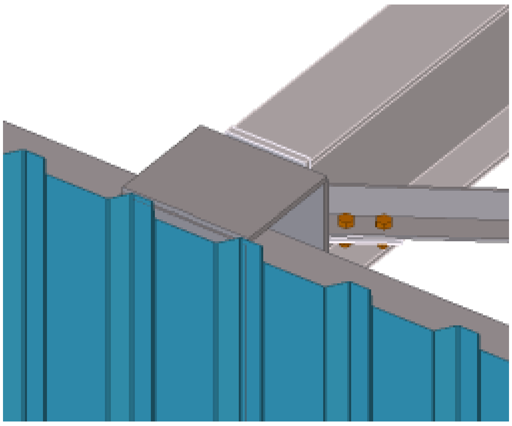
3. To make the hidden parts or objects visible again, click  on the **View** tab.

   Alternatively, right-click in the view and select **Redraw view**.

Show only selected parts or other objects in a model view

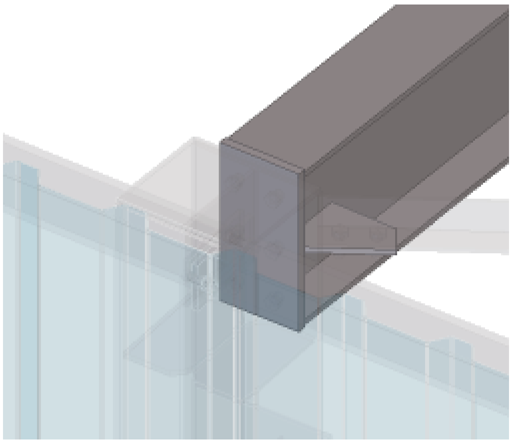
As an alternative to hiding single parts or other objects in a model view, you can define which parts you want to keep visible. All of the other, unselected parts will be hidden.

1. Select the parts or objects that you want to keep visible.



2. Right-click and select **Show only selected**.

Tekla Structures shows the unselected parts almost as transparent.



TIP To completely hide the unselected parts, hold down the **Shift** key when selecting the command.

To show the unselected parts as part reference lines, hold down the **Ctrl** key when selecting the command.

3. To make the unselected parts or objects visible again, click  on the **View** tab.

   Alternatively, right-click in the view and select **Redraw view**.

Temporarily show assembly and component objects in a model view

You can temporarily show the contents of an assembly or a component even if some of the assembly or component objects are not visible in a model view.

To	Do this
Show the contents of an assembly	1. Right-click an assembly, or a part in the assembly. 2. Select **Assembly** --> **Show assembly** . For a concrete part, select **Show assembly**. Tekla Structures shows an orange box around the assembly, and displays all parts, bolts, welds, and other details (not cuts or fittings) belonging to the assembly, even if you had defined

To	Do this
	them as hidden in the display settings (page 647). For concrete parts, Tekla Structures displays reinforcement and surface treatment (not surfaces), even if you had defined them as hidden in the display settings.
Show the contents of a component	1. On the **View** tab, click **Rendering** --> **Show component content** . 2. Select a component. Tekla Structures displays all bolts, welds, and other details belonging to the component, even if you had defined them as hidden in the display settings (page 647).
Reapply the display settings (page 647) and make assembly or component objects hidden again	On the **View** tab, click .

3.5 Change the color and transparency of model objects by using object representation

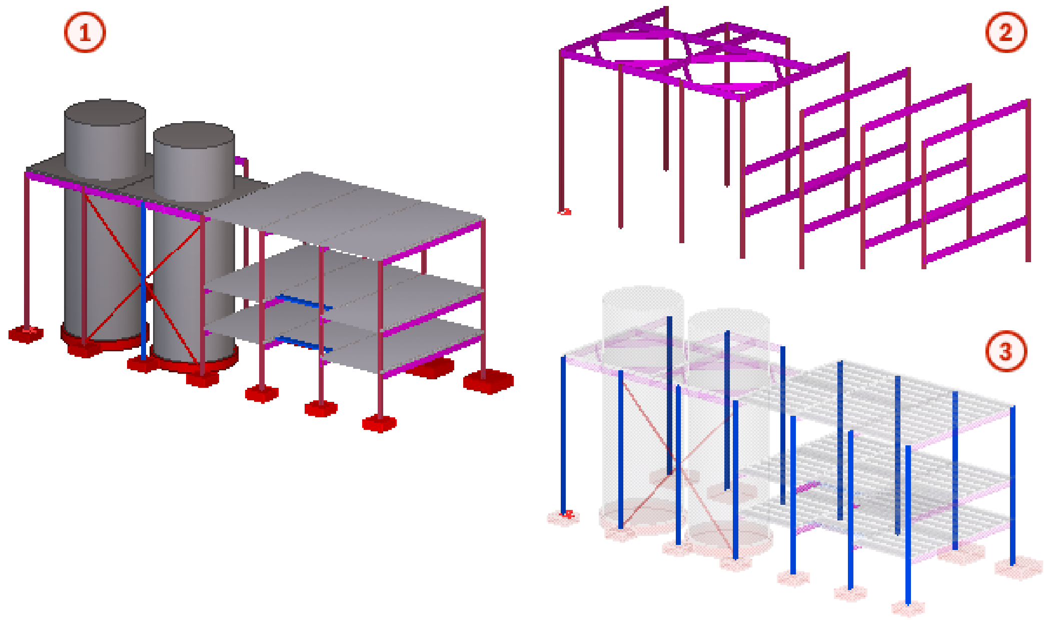
Use *object representation* to set the color and transparency of model objects in model views and to create customized presentations of the model by using defined *object groups*.

With object representation settings and object groups you can define that, for example, only some model objects are visible or that some model objects are of certain color.

To open the object representation settings, on the **View** tab, click **Representation**. The **Object Representation** dialog box opens.

Note that changing the settings in the **Object Representation** box affects all model views.

The following images show the same model with different transparency settings:



1. Standard color and transparency settings
2. Only parts whose profile name starts with IPE* or HEA* are visible
3. Parts whose user-defined attribute **Planned erection date** is set to a certain date are displayed in blue, while all other parts are 90% transparent

Use the Class property to change the color of parts and reinforcement

Use the object representation settings **Color by class** and **Visible** to enable the changing of the color of parts and reinforcement in the part and reinforcement properties.

1. To open the object representation settings, on the **View** tab, click **Representation**. The **Object Representation** dialog box opens.
2. In the dialog box, ensure that
   - **Object group** is set to **All**
   - **Color** is set to **Color by class**
   - **Transparency** is set to **Visible**

   With these settings, all parts and reinforcement are colored according to their **Class** property in the part and reinforcement properties.

   The possible class numbers range between 0 and 14, and result in different colors as follows:

Class numbers above 14 produce the same colors as 1...14. For example, class numbers 2, 16, 30, 44, and so on, all result in red.

3. To change the part or reinforcement color in the part or reinforcement properties:

   a. In the property pane, in the **Class** box, select a new class.

   b. Click **Modify**.

   Alternatively, you can change the **Class** property on the contextual toolbar.

NOTE You can also use class numbers to define the default color of pour objects and pour breaks.

Define color and transparency settings for object groups

1. On the **View** tab, click **Representation** to open the **Object Representation** dialog box.

   The settings **Color by class** and **Visible** are on by default.

2. To define the color and visibility of a selected object group, click **Add row**.
3. Select an object group (page 662) from the **Object group** list.
4. Use the **Color** list to define the color for the objects in the object group.
5. Use the **Transparency** list to define the transparency for the objects in the object group.
6. Repeat steps 3–5 for each row you add.

7. Use the **Move up** and **Move down** buttons to change the order of the rows.

   The reading order of the rows is from bottom to upwards. If an object belongs to several groups, the upmost row defines the color and transparency setting defined for the objects.

8. Enter a unique name in the box next to the **Save as** button.
9. Click **Save as** to save the settings.

NOTE If your setting does not contain the group **All**, Tekla Structures adds that row to the bottom of the list when you click **Modify**, **Apply** or **OK**.

Color settings in object representation

Option	Description
As is	The current color is used. If the object belongs to one of the object groups defined in the following rows, its color is defined by the settings that the object group in question has on that row.
Colors	Select color from the list.
Color by class	All parts are colored according to their **Class** property.
Color by lot	Parts belonging to different lots or phases get different colors according to the lot or phase number: 
Color by phase	
Color by analysis type	Displays parts according to the member analysis class.

Option	Description
Color by analysis utility check	Displays parts according to the utilization ratio in analysis.
Color by attribute	Displays parts in different colors according to the values of a user-defined attribute.

Transparency settings in object representation

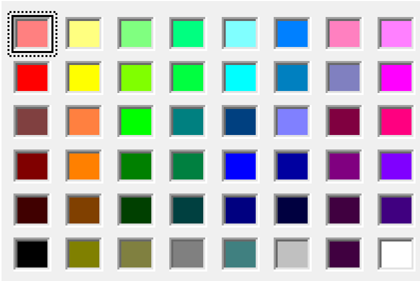
Option	Description
As is	The current visibility. If the object belongs to any object group whose visibility and color settings have been defined, the settings will be read from that object group.
Visible	Object is shown in the views.
50% transparent	Object is transparent in the views.
70% transparent	
90% transparent	
Hidden	Object is not shown in the views.

Define your own colors for object groups in object representation

If the color properties are not sufficient, you can define your own colors.

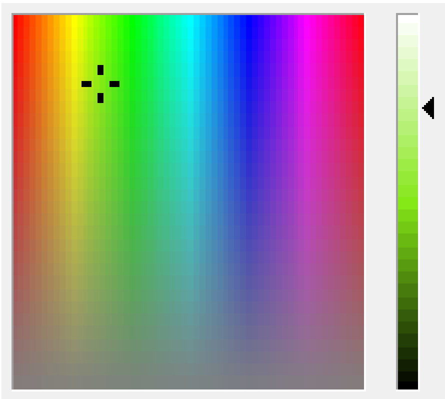
1. On the **View** tab, click **Representation** to open the **Object Representation** dialog box.
2. Select an object group from the **Object group** list.
3. In the **Color** list, select **Choose color...**.
4. Do one of the following:

- Click a color in the **Basic colors** palette.



- Click **Define Custom Colors** and create a color of your own:

  a. Click a color in the color window.

  b. Define the color depth by using the color bar on the right, or enter the exact RGB values.

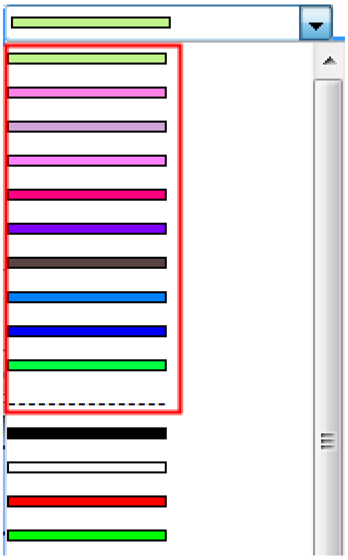
  c. Click **Add to Custom Colors**.

  d. Click the color in the **Custom colors** palette to select it.

5. Click **OK**.
6. Click **Save** to save the changes.

When you open the **Object Representation** dialog box the next time, the **Color** list shows a maximum of 10 colors that you last defined. Customized colors are shown above the dashed line:



Information about the colors that you define for the object groups is stored in the `used_custom_colors.clr` file, which is located in the current model's `\attributes` folder.

Information about the colors that you add in the **Custom colors** palette is stored in the `xs_user.xxx` file in the model folder (`xxx` is your user name).

Copy object representation settings to another model

1. Select the settings you want to copy.

   The settings you have created are located in the current model’s `\attributes` folder, and they have the file name extension `.rep`.

2. Select where you want to copy the settings.

   - To make the settings available in another model, copy them to the `\attributes` folder of the destination model.

   - To make the settings available in all models, copy them to the project or firm folder, defined by the advanced option or .

3. Restart Tekla Structures.

If you want to delete the settings in the `\attributes` folder, select the `.rep` file and press **Delete**.

3.6 Use object groups in object representation and in filters

In *object groups* the objects are grouped by a set of rules and conditions. Use the object groups to control the transparency and coloring of model objects in the model views, and in filters to define which objects are displayed or can be selected.

Where object groups are used

Object groups are sets of rules with which you can group objects by selected properties and conditions.

Use object groups in

- object representations to control the transparency and coloring of model objects in all views
- model view filters (page 154) to define which objects are displayed in the selected view
- selection filters (page 156) to define which objects can be selected
- Organizer to filter the content of categories
- Project status visualization to review the status of model objects in a specific time frame

Object groups for different purposes use different object group files. Object group files are saved in the current model's `attributes` folder.

Object group	File name extension
Object group - Representation	`.PObjGrp`
Object group - View filter	`.VObjGrp`
Object group - Select filter	`.SObjGrp`
Object group - Organizer	`.OrgObjGrp`

Create an object group for object representation

1. On the **View** tab, click **Representation** to open the **Object Representation** dialog box.
2. Click **Object group...** to open the **Object Group - Representation** dialog box.
3. Select an existing object group from the **Save/Load** list to create a modified version or click **New filter** to start without existing settings.
4. Click **Add row** or continue to modify the settings on an existing row.

5. Select options from the **Category**, **Property**, and **Condition** lists.

   You can use the same object properties (page 164) and techniques (page 161) as in filtering.

6. In the **Value** list, enter a value or select one from the model.

   The values can be complete strings, such as the profile name `UC310*97`. You can also use incomplete strings together with wildcards (page 181). For example, the value `UC*` will match with all parts whose profile name begins with the characters `UC*`. Empty values are matched to empty object properties.

   If you use multiple values, separate the strings with blank spaces (for example, `12 5`). If a value consists of multiple strings, enclose the entire value in quotation marks (for example, `"custom panel"`), or use a question mark (for example, `custom?panel`) to replace the space.

7. Use the **And/Or** options and brackets (page 161) to define how multiple rows work together.

8. To temporarily disable rules without deleting them, you can deselect the check boxes in the first column of the row. Select the check box to re-enable the rule.

9. Enter a unique name in the box next to the **Save as** button.

10. Click **Save as** to save the object group.

Object group - Representation settings

When you create object groups in the **Object Group - Representation** dialog box, you can use the same object properties and techniques as in filtering. The following table introduces the options shortly, for more detailed information see Object properties in filtering (page 164) and Filtering techniques (page 161).

Column	Description
Check boxes	Select the check box to include the row in the rule. By default each new row is disabled
Parentheses	Use parentheses to create nested rules.
Category	Use the **Object** category with user-defined attributes or, for example, if you need to create a selection filter based on GUIDs.
Property	Select an available property. All user-defined attributes can be selected as well.
Condition	Numeric, textual, and date properties all have different set of conditions.

Column	Description
Value	You can enter the value manually, or select the value from the model by clicking an object. For example, if you select **Part** as **Category**, and **Name** as **Property**, and click a part in the model, you get its **Name** property in the object group rule.
And/Or	Use the **And/Or** column when you create rules with several rows. An empty field is the same as if it had the **And** option selected.

Copy object groups to another model

1. Select the object group you want to copy.

   The object groups you have created are located in the model's `\attributes` folder, and they have the file name extension `.PObjGrp`.

2. Select where you want to copy the object group.

   - To make an object group available in another model, copy the file to the `\attributes` folder of the destination model.
   - To make an object group available in all models, copy the file to the project or firm folder, defined by the advanced option XS_PROJECT or XS_FIRM.

3. Restart Tekla Structures.

If you want to delete the object group file in the `\attributes` folder, select the `.PObjGrp` file and press **Delete**.

4 Check the model

You can use a variety of tools to check and view the model, and to ensure the model does not contain errors.

- To quickly search for objects in the entire model or within the selected model objects, see Search for model objects (page 666).
- To take snapshots and to create animations that demonstrate the design and build options of your model and use them in your presentations, see Visualize the model (page 669).
- To travel through the model by using the **Fly** command, see Fly through the model (page 682). If you need to focus in on required details in the model, see Create clip planes (page 683). To show parts in a selected view angle, see Show parts, components, or assemblies in a selected view angle Show parts, components, or assemblies in a selected view angle (page 685).
- Use the different **Inquire** commands to get particular information about model objects, as described in Inquire object properties (page 686).
- Use the **Measure** commands to measure, for example, distances, as described in Measure objects (page 692).
- To find colliding objects in a model, see Detect clashes (page 695).
- To check the inconsistencies in a model, you can compare selected parts or assemblies, view solid errors in a log file, or diagnose and repair the model to check and repair errors, or to find distant objects. For instructions, see Compare parts or assemblies (page 707), View solid errors (page 707), Diagnose and repair the model (page 708) and Find distant objects (page 710).

4.1 Search for model objects

You can quickly search for objects in the entire model or within the selected model objects by using the **Model search** toolbar.

When you run a search in the model, Tekla Structures searches for objects whose property values contain the search term, and then highlights and selects the objects that have matching property values. Tekla Structures searches for the following objects and checks the listed object properties for matches:

- **Parts and items**: name, profile or shape, material, part position number, assembly position number or cast unit position number, GUID
- **Pour objects**: pour type (if pour management is enabled (page 445))
- **Reinforcement**: name, grade, position number, cast unit position number, GUID
- **Surfaces**: name, GUID
- **Components**: name, running number (shown in the **Inquire object** dialog box), GUID
- **Assemblies**: name, assembly position number, GUID
- **Cast units**: name, cast unit position number, GUID
- **Pour units**: name (if pour management is enabled)
- **All other objects**: GUID

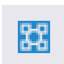
You can use the following wildcards (page 181) in the search term: *, ?, or [ ], or " " for exact match.

The search term may consist of several words. When a property value of an object contains all of those words, Tekla Structures finds that object.

If there is only one word in the search term, Tekla Structures finds all the objects whose property values contain that word. For example, with the search term `Plate` you can find objects with the name `Plate` or `End plate`, but objects with the name `Plate1` are not found.

If you enclose the search term in " ", Tekla Structures only finds the exact matches. For example, `"end plate"` does not find objects with the name `Stiffened end plate` or `Two sided end plate`.

The search is not case-sensitive, which means that you can use both lower-case and upper-case letters. For example, the search terms `beam` and `BEAM` give the same search results.

If the **Select assemblies**  selection switch is active, Tekla Structures searches for assemblies, cast units, and pour units with matching names or position numbers. Otherwise Tekla Structures searches for other objects with matching property values. Other selection switches do not affect the search results.

Search in the entire model

When you run a search in the entire model, Tekla Structures also searches for the hidden objects whose property values match the search criteria.

1. If pour management is enabled (page 445) and you are searching for pour objects or pour units, ensure that you are using a pour view (page 447).
2. On the **Model search** toolbar, enter the search term in the box.

   You can use wildcards *, ?, or [ ], or " " for exact match.
3. Click , or press **Enter**.

Tekla Structures highlights and selects the model objects whose property values match the search criteria, and shows the object properties in the property pane.

Search within the selected model objects

1. Select the objects within which you want to search.

   You can use area selection or a filter (page 150) to focus on certain objects.
2. On the **Model search** toolbar, enter the search term in the box.

   You can use wildcards *, ?, or [ ], or " " for exact match.
3. Click .

Tekla Structures highlights and selects the model objects whose property values match the search criteria, and shows the object properties in the property pane.

Review the search results

You may find these tips useful when you examine the search results and the selected model objects.

- To hide the model objects whose property values do not match the search criteria, do any of the following:
  - Press **Ctrl+5** to show the matching objects only.
  - Press **Shift+5** to show the matching component objects only.
  - Right-click and select **Show only selected (page 654)**.
- To narrow down your search further, you can run nested searches. Keep the first search result objects selected and then run another search within them by clicking .

- To review the matching objects as a list, use **Organizer**. **Object Browser** lists the selected objects and shows their properties in columns.
- To review or modify the properties of the selected objects, use the property pane.

Show or hide the Model search toolbar

By default, the **Model search** toolbar is visible and located at the bottom of the Tekla Structures main window.

If the toolbar is not visible, do one of the following:

- Go to the **File** menu, click **Settings**, and in the **Toolbars** list, select the **Model search toolbar** check box.
- Go to **Quick Launch**, start typing `toolbar`, and then select the **Model search toolbar** check box in the list that appears.

If you want to hide the **Model search** toolbar, clear the **Model search toolbar** check box in **File** --> **Settings** --> **Toolbars** or by using **Quick Launch**.

4.2 Visualize the model with Trimble Connect Visualizer

Trimble Connect Visualizer creates a basic rendering of an open Tekla Structures 3D model. You can use Trimble Connect Visualizer to take snapshots and create animations that demonstrate the design and build options of your model. You can then use the snapshots and animations in your presentations.

You can access Trimble Connect Visualizer directly from Tekla Structures. The Trimble Connect Visualizer commands are located on the **View** tab.

Visualize all model objects

1. On the **View** tab, click the arrow under **Visualize**.
2. On the menu, select  **Visualize all**.

   A rendering of the entire model opens in Trimble Connect Visualizer.

   Note that parts that are not visible in the Tekla Structures model view are also rendered.

Visualize the selected model objects

1. Select the objects that you want to visualize.

2. On the **View** tab, click the arrow under **Visualize**.

3. On the menu, select  **Visualize selected**.

   A rendering of the selected model objects opens in Trimble Connect Visualizer.

Working in Trimble Connect Visualizer

Before you take snapshots or create animations in Trimble Connect Visualizer, you can zoom, drag, and pan the model to get the desired views. You can also adjust the scene settings to control, for example, the brightness of the scene and the position of the sun.

Zoom, rotate, or pan the rendered model

- Do any of the following:

To	Do this
Zoom in	Do either of the following: • Scroll forward with the mouse wheel. • Press the **W** key on the keyboard.
Zoom out	Do either of the following: • Scroll backward with the mouse wheel. • Press the **S** key on the keyboard.
Rotate the model	1. Place the mouse pointer over the point around which you want to rotate the model. 2. Hold down the left mouse button. 3. Drag the model with the left mouse button. You can also press the arrow keys on the keyboard to rotate the model.
Pan the model	1. Hold down the middle mouse button. 2. Drag the model with the middle mouse button. You can also press the **A** and **D** keys on the keyboard to pan the model.

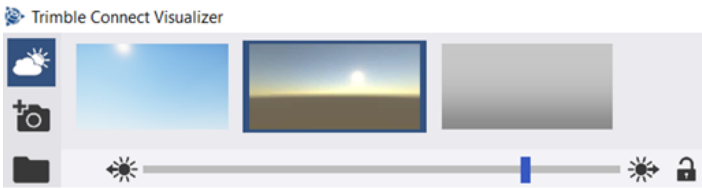
Adjust the scene

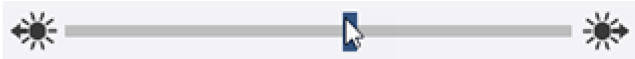
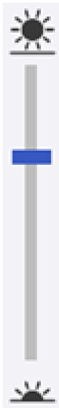
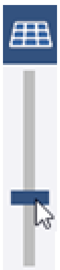
In Trimble Connect Visualizer, you can modify the following scene settings:

- The used skybox, or background, and its rotation

- The position of the sun
- The brightness of the scene
- The visibility and position of the ground plane

1. Click  on the Trimble Connect Visualizer side pane.
2. Do any of the following:

To	Do this
Change the skybox	The skybox that is currently in use is highlighted with blue. To change the skybox: • Click another skybox in the skybox selector at the top of Trimble Connect Visualizer.  You can also use your own images as skyboxes. Note that all skybox images need to be 360° images with a ratio of 2:1. Save the skyboxes in the C:\Users\<user>\Pictures\TrimbleConnectVisualizer\Skyboxes folder. The skyboxes will be available in the skybox selector the next time you open Trimble Connect Visualizer.
Rotate the skybox	• Move the rotation slider at the top of Trimble Connect Visualizer.  You can lock the position of the skybox and the direction of the sun together by clicking . After locking the direction of the sun and the position of the skybox, moving the rotation slider also moves the sun direction slider by the same amount.
Adjust the brightness of the scene	• Move the brightness slider at the upper-right corner. 
Adjust the position of the sun	Do any of the following:

To	Do this
	• To change the direction of the sun, move the sun direction slider at the upper-left corner.  You can lock the direction of the sun and the position of the skybox together by clicking . After locking the direction of the sun and the position of the skybox, moving the rotation slider also moves the sun direction slider by the same amount. • To change the height of the sun above the horizon, move the sun height slider on the left side of the rendered model. 
Show or hide the ground plane	• To switch between showing and hiding the ground plane, click  on the left side of the rendered model. When the ground plane is switched on, a slider appears below the  button. You can move the slider to adjust the height of the ground plane. 

Take and view snapshots

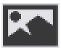
1. Adjust the position of the model and the scene as needed.
2. Click  at the upper-left corner of the Trimble Connect Visualizer window.

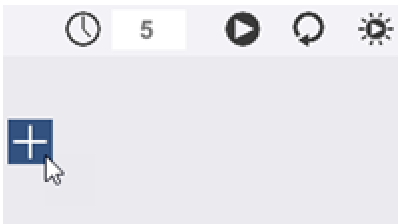
Trimble Connect Visualizer takes a snapshot of the current view of the rendered model. The snapshots are saved in the C:\Users\<username>\Pictures\TrimbleConnectVisualizer folder.

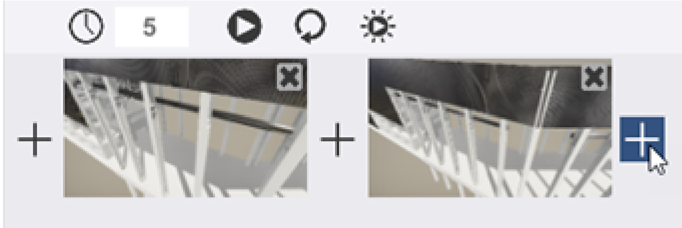
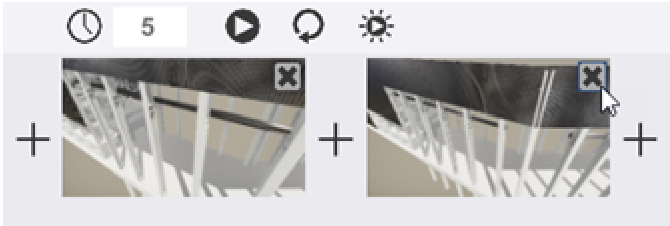
To open the C:\Users\<username>\Pictures\TrimbleConnectVisualizer folder and view the snapshots that you have taken, click  at the upper-left corner of the Trimble Connect Visualizer window.

Create animations

You can combine different views of the rendered model to create animations that show the model from different angles. Do the following:

1. Click  on the Trimble Connect Visualizer side pane.
2. Do any of the following:

To	Do this
Add a view	a. Move the model to get the view that you want to use. b. Click **+** on the toolbar below the view.  Repeat steps a to b to create all necessary views. Note that when you have added multiple views, you need to click **+** on the right side of the view after which you want to add the new view. Similarly, click **+** or on the left side of the view before which you want to add the new view.

To	Do this
	
Remove a view from the animation	• Click the **Close** button (**X**) at the upper-right corner of a view. 
Set the time between views	The time between views determines the speed of the animation. • Type the desired time in seconds in the box below the rendered model ().
Play or stop the animation	• To play the animation, click  on the toolbar below the rendered model. • To stop the animation and return to the first view, click .
Play the animation continuously	You can choose to play the animation in a loop without stopping. a. Click  on the toolbar below the rendered model. b. To play the animation, click .
Freeze the sun position in the animation	In each view you create, the sun is in a different position. Animations can look strange if the sun moves, so it might be helpful to freeze the sun position. • Click  on the toolbar below the rendered model.

3. To save the animation, use a screen recorder, such as the built-in free screen recorded in Windows 10.

   You can find more information, for example, here.

Return to the initial view of the model

1. Click on the Trimble Connect Visualizer side pane.

The model returns to the view that was opened when you started Trimble Connect Visualizer.

Enter or exit full screen mode

1. Click on the Trimble Connect Visualizer side pane.

Show or hide the Trimble Connect Visualizer side pane

1. Click at the bottom of the Trimble Connect Visualizer side pane.

Use Trimble Connect Visualizer in VR mode

The VR mode in Trimble Connect Visualizer allows you to easily switch to a virtual reality mode. In the VR mode, you can move either with the keyboard or with an Xbox One controller that is connected to your computer. Note that moving in the VR mode imitates the speed of walking or running.

The VR mode uses the OpenVR API. Because of this, using the VR mode in Trimble Connect Visualizer has some prerequisites:

- A VR device that is compatible with OpenVR must be attached to your computer and configured according to the instructions of the hardware vendor. Among others, HTC Vive, Oculus Rift, Windows MR, and VarjoVR devices are compatible with OpenVR.

  The VR mode in Trimble Connect Visualizer is heavier to render than the regular 3D mode. Check the technical specifications of the device to see if the device has enough capacity for working in the VR mode in Trimble Connect Visualizer. For more information, contact the vendor of your device.

- Steam and SteamVR need to be downloaded and installed onto your computer:

  1. Download Steam and install it onto your computer.
  2. Create a Steam account and sign in to Steam.
  3. Download the SteamVR package onto your computer.
  4. Start SteamVR and configure it.

  You do not need to be signed in to Steam to work in the VR mode in Trimble Connect Visualizer. It is enough to set up Steam once, and then sign in for updates occasionally.

1. Click in the Trimble Connect Visualizer side pane.

   If the VR mode icon ( ) is not visible, your computer does not meet the requirements of the VR mode.
2. Move in the VR mode as you wish.

To move in the VR mode with	Do this
Your keyboard	The keyboard controls are the same as in the regular 3D mode: • To move forward, press the **W** key. • To move backward, press the **S** key. • To move left, press the **A** key. • To move right, press the **D** key. • To move up and down, press the **Q** and **E** keys. • To rotate around the global vertical axis, press the **X** and **Z** keys. • To move at running speed, hold down **Shift** .
An Xbox One controller	When an Xbox One controller is connected to your computer via Bluetooth, use the following controls: • To move the viewer relative to the view direction, use the left stick. • To rotate around the global vertical axis, use the right stick. • To move up and down, use the left and right triggers. • To move at running speed, hold down the left stick.

3. To switch back to the 3D mode, click again.

Modify material mappings for Trimble Connect Visualizer

By default, Trimble Connect Visualizer attempts to map the materials used in Tekla Structures to the material types used in the Trimble Connect Visualizer material catalog. You can also set material type mappings manually to define how you want materials to look in Trimble Connect Visualizer. If necessary, you can override the materials of particular model objects, and use another material instead.

You can currently map the Tekla Structures materials to the following pre-defined material types in Trimble Connect Visualizer:

- Asphalt
- Concrete
- Dark gravel
- Default: renders objects with white matte material
- Glass
- Grass
- Ground
- Light Gravel
- Steel
- Timber
- Water

Pre-defined materials and their color and transparency information are read from the materials_ifc.xml file saved in the ..\TeklaStructures\<version>\bin\applications\Tekla\Tools\TrimbleConnectVisualizer\TrimbleConnectVisualizer_Data\StreamingAssets folder.

You can also map to your own user-defined materials. For more information, see Create and modify user-defined materials (page 679).

In Trimble Connect Visualizer, you can currently only render the material type of objects, not their material finish. This means that paints or other finishes are not visible in Trimble Connect Visualizer.

To set the material type mappings used in Trimble Connect Visualizer:

1. On the **View** tab in Tekla Structures, click the arrow on the right side of **Visualize**.
2. On the menu, select  **Visualizer material type mapping**.
3. In the lists on the right side of the related Tekla Structures materials, select the Trimble Connect Visualizer material types that you want to map to.

   Selecting the blank option renders the color of the Tekla Structures class as a material that is similar to plastic. The blank option can be useful if you need to show the class colors to communicate an aspect of the structural design.

4. To override the Tekla Structures material of particular model objects in Trimble Connect Visualizer:
    a. In the model, select the objects.
    b. In the **Override Visualizer materials of selected objects** list, select the material that you want to use for visualizing the selected objects.
    c. Click **Set**.

    The selected override material is stored to the model objects' `VISUALIZER_MATERIAL` user-defined attribute, which you can see by inquiring the object properties (page 686).
5. Click **OK** to update the material type mapping.
6. To save the material type mapping as an XML file, do either of the following:

<table>
<tr><th>To</th><th>Do this</th></tr>
<tr><td>Save the material mapping to the model folder</td><td>• When Tekla Structures asks you if you want to save the material mapping to the model, click Yes.

The material type mapping is saved in the model folder under the <code>\attributes</code> folder. The file name is <code>VisualizerMaterials.tcv11</code>.

The material type mapping is saved for all users of the model, including the users that edit the model using Tekla Model Sharing or the multi-user mode.

NOTE To avoid issues, do not modify the colors in the <code>VisualizerMaterials.tcv11</code> file manually.</td></tr>
<tr><td>Save the material mapping to local application data</td><td>• When Tekla Structures asks you if you want to save the material mapping to the model, click No.

The material type mapping is saved in the <code>\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\Trimble Connect Visualizer</code> folder. The file name is <code>VisualizerMaterials.tcv11</code>.

The material type mapping now applies to all projects that you are working on.

NOTE To avoid issues, do not modify the colors in the <code>VisualizerMaterials.tcv11</code> file manually.</td></tr>
</table>

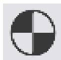
To	Do this

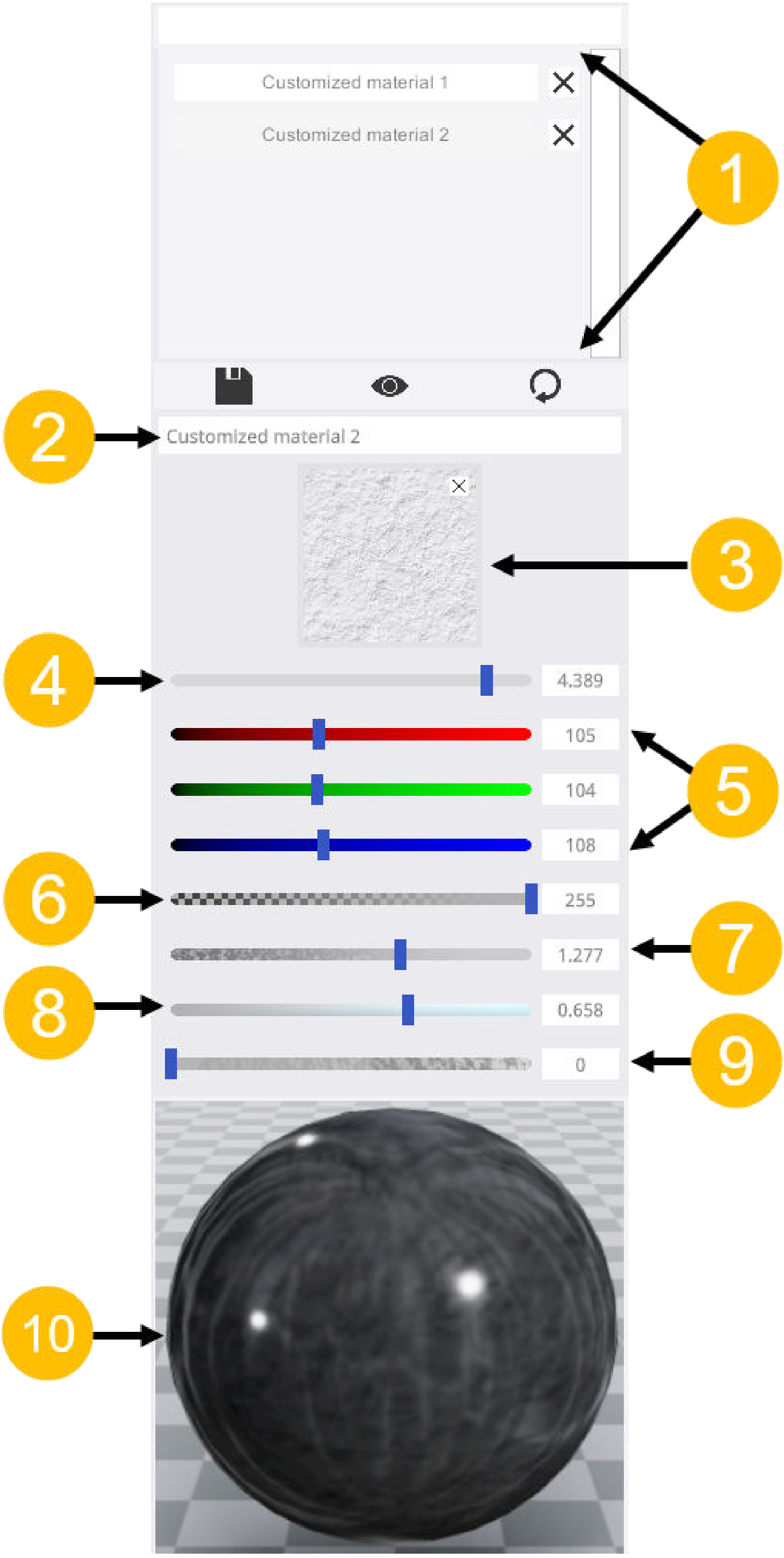
Create and modify user-defined materials

You can create your own customized materials in Trimble Connect Visualizer with the material editor. The customized materials can be used in Trimble Connect Visualizer material mapping in the same way as pre-defined materials.

NOTE User-defined materials are not shared to other users with Tekla Model Sharing. Any objects that use user-defined materials appear with their class colors to other users.

To create or modify user-defined materials:

1. Visualize the model. (page 669)
2. To open the material editor, click  on the Trimble Connect Visualizer side pane.



The material editor contains the following elements:

(1) List of user-defined materials

(2) Material name box

(3) Texture picker

(4) Scaling or tiling of the selected texture

(5) RGB color values

(6) Transparency

(7) Roughness

(8) Metallic look

(9) Bump strength

(10) Preview of the material

3. Do one of the following:
   - To create a new material, type a name in the material name box.
   - To modify an existing user-defined material, select the material in the list of user-defined materials.
4. In the material editor, do any of the following:

To	Do this
Add a texture	a. Click the texture picker. b. Click the texture that you want to use. Note that you can only use textures that are saved in the C:\Users\<user>\Pictures\TrimbleConnectVisualizer\Textures folder as .png or .jpg images. To reset the material texture, click **X** at the upper-right corner of the texture picker.
Adjust scaling or tiling of texture, RGB colors, transparency, roughness, metallic look, or bump strength	Either drag the related slider, or type the desired value in the related box.
Preview all objects with the current user-defined material settings	Click . To reset the objects to use their original materials, click .

5. To save the material, click .

User-defined materials are saved in the `C:\Users\<user>\AppData\Local\TrimbleConnectVisualizer\CustomMaterials` folder.

Each user-defined material is defined using the following files:

- <Material name>.xml
- <Material name>_DiffuseMap.png
- <Material name>_NormalMap.png
- <Material name>_RoughnessMap.png

Note that the name of the material file and the material name in the `.xml` file need to be the same.

To avoid issues, do not modify any of the material files manually.

4.3 Fly through the model

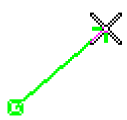
Using the **Fly** command, you can travel through a model, changing direction and varying the speed as you go. You can also adjust the field of view setting, which can be useful when flying in a tight space.

1. Set view projection to **Perspective**.
    a. Double-click the view to open the **View Properties** dialog box.
    b. In the **Projection** list, select **Perspective**.
    c. Click **Modify**.
2. If needed, adjust the field of view setting.

    The bigger the value, the more distance there is between the parts when you fly through the model.

    a. On the **File** menu, click **Settings** --> **Advanced options** , and go to the **Model views** category.
    b. Modify the advanced option XS_RENDERED_FIELD_OF_VIEW.
    c. Click **OK**.
3. On the **View** tab, click **Fly**.
4. Select a view.

    The mouse pointer changes into an arrow and a cross. The arrow indicates the current flying direction.



5. Drag the mouse to move around in the model.
   - To fly forward, move the mouse forward.
   - To change the flying direction, drag the mouse in the desired direction.

     The flying speed grows exponentially when you are approaching the model from a distance.
   - To move up or down, hold down **Ctrl** and drag the mouse forward or backward.
   - To change the camera angle, scroll with the mouse wheel.
   - To fly in the in the direction of the camera angle, hold down **Shift** and scroll forward or backward.
6. To stop flying, press **Esc**.

4.4 Create clip planes

Clip planes enable you to focus on the required detail in the model. You can either create clip planes by selecting an object face, or create clip planes that are based on the view depth.

Create a clip plane

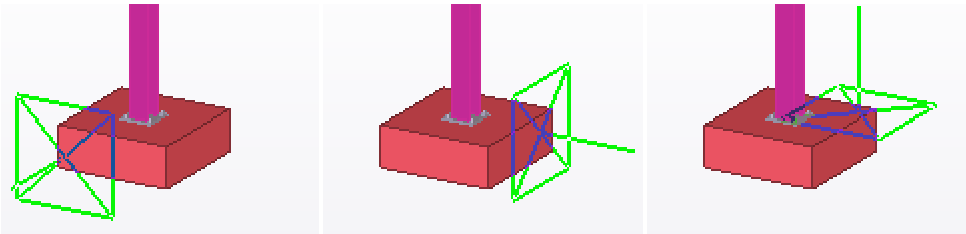
You can create up to six clip planes in any model view that shows the object faces.

1. When you create clip planes, ensure that you are using a model view that shows object faces.

   On the **View** tab, click **Rendering**, and use any of the following options:
   - **Parts shaded wireframe** (Ctrl+2)
   - **Parts grayscale** (Ctrl+3)
   - **Parts rendered** (Ctrl+4)
   - **Components shaded wireframe** (Shift+2)
   - **Components grayscale** (Shift+3)
   - **Components rendered** (Shift+4)
   - **References shaded wireframe** (Ctrl+Shift+2)
   - **References grayscale** (Ctrl+Shift+3)
   - **References rendered** (Ctrl+Shift+4)
2. On the **View** tab, click **Clip plane** .

3. Move the mouse pointer over the model objects.

   A green symbol indicates the object faces that you can select and align the clip plane with. The green line indicates the side that will be clipped out. For example:

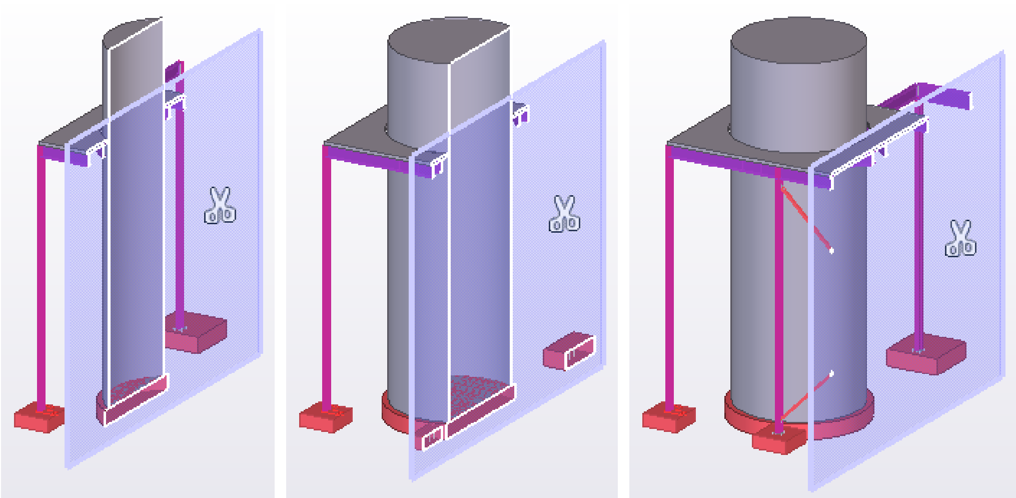
   Note that the snap depth setting on the **Snapping** toolbar affects the object faces you can select. Set snap depth to **3D** or **Auto** to select object faces in the entire 3D space.

4. Select an object face.

   The clip plane symbol appears in the model:

5. Repeat step 4 to create as many clip planes as needed.
6. To finish creating clip planes, press **Esc**.
7. To move a clip plane, select the clip plane scissor symbol and drag it to a new location.

8. If you want to move the clip plane scissor symbol to a new location on a clip plane, hold down **Shift** and drag the symbol.

   This does not move the clip plane, only the scissor symbol.

9. You can delete the clip planes when you do not need them anymore.

- To delete a single clip plane, select the clip plane symbol and press **Delete**.
- To delete all the clip planes in all the open views, click **View** --> **Clip plane** --> **Delete all clip planes** .

Create a view depth clip plane

If the model objects do not fit to the work area view depths, you can create view depth clip planes on the contextual toolbar.

Based on the view depth, you can, for example, easily isolate an entire floor from a building. View depth clip planes can be created both in native Tekla Structures models as well as in reference models and overlay models.

1. If needed, modify the view depth of the current view on the contextual toolbar.
2. On the contextual toolbar, click **Add or update view depth clip planes** .
3. If you want to remove the view depth clip planes, click **Remove view depth clip planes**  on the contextual toolbar, or click **View** --> **Clip plane** --> **Delete all clip planes**.

   Note that the work area box may have gaps to view depths, and when working with overlay models, you can clip models that are outside the work area.

Limitations:

- You cannot move the view depth clip planes by dragging the clip plane to a new location. This is because the view depth clip planes are strictly connected to the view depth.
- After you have adjusted the view depth, click the **Add or update view depth clip planes** button to manually update the view depth clip plane.

4.5 Show parts, components, or assemblies in a selected view angle

In some cases it is useful to examine parts, components, or assemblies in a selected view angle. For example, when you reinforce concrete elements, you can easily check the distances between the reinforcing bars.

Limitation:

The **View angle** option is not available for:

- detail system components
- detail custom components
- custom seams

1. Select an object.
2. Click **View angle** on the contextual toolbar.
3. Select a top, back, right, bottom, front, or left view.

   Tekla Structures displays the object in the selected view angle. Note that the full object is not always displayed.

   For parts, the view angle is based on the selected object's coordinate system, so that the top view is looking to the negative z-direction. For example, if you select the top view, Tekla Structures sets the current view angle from up to down in the coordinate system.

   For assemblies and components, the view angle is based on the assembly main part's coordinate system. Because of that, the object may not always appear as expected in the selected view angle.
4. To return to the original 3D view, click the button in the middle of the view angle options.

NOTE Tekla Structures shows the object in the current view, which is the view where your mouse has last been. If the contextual toolbar is located, for example, on top of two views, the object is shown in the view where your mouse has last been and not in the view where the object was selected.

See also

Adjust how model objects are displayed (page 642)

4.6 Inquire object properties

Use the **Inquire** commands to get information about a particular object, or a group of objects, within the model.

Do any of the following:

To inquire	Do this
Object properties	1. On the ribbon, click **Object**. 2. Select an object. Tekla Structures displays the object properties in a separate window.
Point coordinates	1. On the ribbon, click the down arrow next to , and then select **Point coordinates**. The **Inquire point coordinates** dialog box is displayed. 2. Click **Pick**, and then pick a point in the model to see the coordinates of the point in: • Local coordinates • Model coordinates (global) • Project base point coordinates • Current base point coordinates
Center of gravity	1. On the ribbon, click the down arrow next to , and then select **Center of gravity**. 2. Select one or more parts or reinforcement objects. Tekla Structures creates a point at the center of gravity of the selected objects, and displays information about the center of gravity in a separate window.
Object properties using custom reports	See Custom inquiry (page 689).
Welded parts	1. On the ribbon, click the down arrow next to , and then select **Welded parts**. 2. Select a part. Tekla Structures highlights the selected part and all the parts that are welded to it.
Primary welded parts	1. On the ribbon, click the down arrow next to , and then select **Primary welded part**.

To inquire	Do this
	2. Select a part. Tekla Structures highlights the primary part when you select a secondary part.
Assembly, cast unit, or rebar assembly objects	See Check and highlight objects in an assembly (page 428), Check and highlight objects in a cast unit (page 430), or Check and highlight objects in a rebar assembly (page 618).
Component objects	1. On the ribbon, click the down arrow next to , and then select **Component objects**. 2. Select a component. Tekla Structures highlights all objects belonging to the selected component.
Phases	On the ribbon, click the down arrow next to , and then select **Phases**. Tekla Structures displays information about objects in different phases in a separate window.
Model size	Tekla Structures displays the quantity of all objects in the current model in a separate window. On the ribbon, click the down arrow next to , and then select **Model size**.

See also

Object property report templates (page 688)

Object property report templates

When you view object properties using the **Inquire object** command, Tekla Structures uses the following report templates:

Object type	Template
Assemblies	TS_Report_Inquire_Assembly.rpt
Bolts	TS_Report_Inquire_Bolt.rpt
Cast units	TS_Report_Inquire_Cast_Unit.rpt
Parts	TS_Report_Inquire_Part.rpt
Pour breaks	TS_Report_Inquire_Pour_Break.rpt
Pour objects	TS_Report_Inquire_Pour_Object.rpt

Object type	Template
Pour units	TS_Report_Inquire_Pour_Unit.rpt
Rebar assemblies	TS_Report_Inquire_Rebar_Assembly.rpt
Reinforcement meshes	TS_Report_Inquire_Rebar_Mesh.rpt
Rebar sets	TS_Report_Inquire_Rebar_Set.rpt
Reinforcement strands	TS_Report_Inquire_Rebar_Strand.rpt
Reference models	TS_Report_Inquire_Reference.rpt
Reinforcement	TS_Report_Inquire_Reinforcement.rpt
Surfaces	TS_Report_Inquire_Surface.rpt
Welds	TS_Report_Inquire_Welding.rpt

When Tekla Structures is installed in the `..\Program Files` folder, these templates are by default available in the folder `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\system`.

You can copy these templates to your project or firm folder and then modify the copied templates according to your needs. For more information on how to use templates, see the Template Editor user documentation.

You can also create a custom template for connections and details by saving the template with the name `TS_Report_Inquire_Connection.rpt`.

See also

Inquire object properties (page 686)

Custom inquiry

You can use the **Custom inquiry** command to display information about the selected model object in the side pane. You can define what information you want to display.

Use the Custom Inquiry tool

1. Click the **Custom Inquiry** button in the side pane.

   Alternatively, you can click the down arrow next to on the ribbon, and then select **Custom inquiry**.

   The **Custom Inquiry** window opens in the side pane.
2. In the **Report type** list, select the report template you want to use for showing the object information.
3. Select a model object.

Tekla Structures displays the object properties in the side pane.

If you select several objects or object types, for example parts, bolts, and reinforcing bars, Tekla Structures displays the quantity of all the selected objects, regardless of the object types, or the report template used. For the object properties that differ, Tekla Structures shows **Varies**.

Define what information is shown by Custom Inquiry tool

You can define what information is displayed in the **Custom Inquiry** side pane window. You can add and modify report templates and the attributes in them.

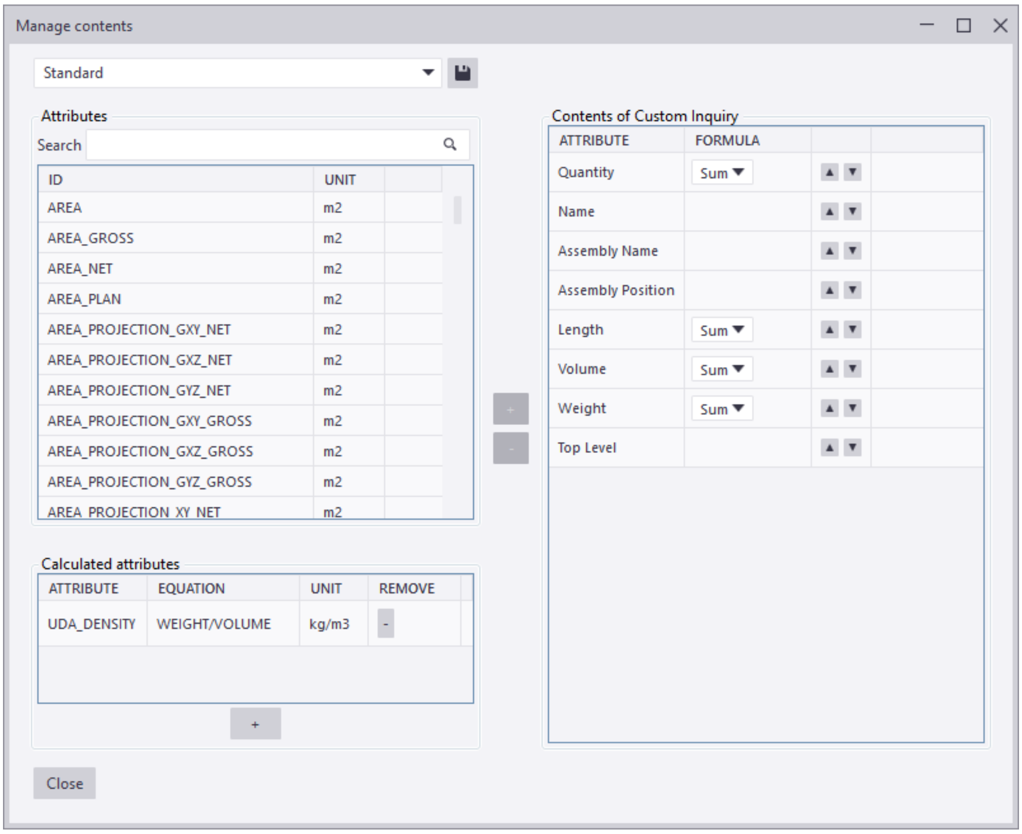
1. Click the **Custom Inquiry** button  in the side pane.

   Alternatively, you can click the down arrow next to  on the ribbon, and then select **Custom inquiry**.

   The **Custom Inquiry** window opens in the side pane.

2. Click the  button.

   The **Manage contents** dialog box appears.



   The **Attributes** list contains the attributes that are available by default. In the **Calculated attributes** area you can create your own attribute formulas. The **Contents of Custom Inquiry** list contains the attributes whose values will be displayed in the side pane.

3. Define which report templates and attributes are available.

   - To modify an existing report template, select it from the top left-hand list in the **Manage contents** dialog box.
   - To create a new report template, enter a name in the box next to the button, and then click .
   - To modify the default attributes, edit the `InquiryTool.config` file.

     For example, you can change the units of the attributes.
   - To create a new calculated attribute, click the + button below the **Calculated attributes** area. To modify a calculated attribute, double-click a cell in the **Calculated attributes** area. In the **Attribute** cell, type the name of the attribute. In the **Equation** cell, use attribute names and standard mathematical symbols (+, -, *, and /) to form equations.

4. Define which attributes are shown in the **Custom Inquiry** side pane window.

   - To add more attributes to the side pane, select an attribute in the **Attributes** or **Calculated attributes** list and then click the + button.
   - To remove attributes from the side pane, select an attribute in the **Contents of Custom Inquiry** list and then click the - button.
   - To change the order of the attributes, use the buttons.
   - To change the formula of an attribute, click the down arrow and select a different formula (**Sum**, **Average**, **Max**, or **Min**) from the list.

5. Click to save the changes.

Modify the default attributes in InquiryTool.config file

Use the `InquiryTool.config` file to control which attributes are shown as default attributes in the **Manage contents** dialog box in the **Custom Inquiry** tool.

NOTE	This section is for advanced users.

Tekla Structures searches for the `InquiryTool.config` file in the following folders in the following order:

1. `\attributes` folder in the model folder

2. `\CustomInquiry` subfolder in the folder defined by XS_PROJECT
3. `\CustomInquiry` subfolder in the folder defined by XS_FIRM
4. `\CustomInquiry` subfolder in the folder defined by XS_SYSTEM

If you specify several folders that have `\CustomInquiry` as a subfolder, Tekla Structures uses the first folder it finds.

To add new attributes to the `InquiryTool.config` file:

1. Open the `InquiryTool.config` file in any standard text editor.
2. Copy the entire contents of `[ATTR_CONTENT_??]` to the end of the file.
3. Change the position number of the new attribute.

   For example, change `[ATTR_CONTENT_??]` to `[ATTR_CONTENT_66]`.
4. Modify the `NAME`, `DISPLAY_NAME`, `DATATYPE`, `UNIT`, and `DECIMAL` values of the new attribute. Use the attribute names and definitions that are included in the `contentattributes_global.lst` or `contentattributes_userdefined.lst` file.
5. Change the `TOTAL_ATTR_CONTENT` value to reflect the total number of attributes in the file.

   For example, change `TOTAL_ATTR_CONTENT=65` to `TOTAL_ATTR_CONTENT=66`.
6. Save the file.

4.7 Measure objects

Use the **Measure** commands to measure angles, arcs, the distance between two points and between bolts in the model.

All measurements are temporary. The measurements appear in the model view window until you update or redraw (page 46) the window.

The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals** .

Measure distances

You can measure horizontal, vertical and user-defined distances in the model.

1. Press **Ctrl+P** to switch to the plane view.
2. On the **Edit** tab, click **Measure** and select one of the following commands:

- **Distance**

  This command measures the distance between any two points. Use this command to measure inclined or aligned distances. By default, the results contain the distance and the coordinates.

- **Horizontal distance**

  This command measures the distance between two points in the direction of the view plane x axis.

- **Vertical distance**

  This command measures the distance between two points in the direction of the view plane y axis.

3. Pick the start point.
4. Pick the end point.
5. Pick a point to indicate on which side of the dimension line you want the measurement to appear.

   The measurement is displayed until the next window update or redraw (page 46).

Measure angles

You can measure angles in the model.

1. On the **Edit** tab, click **Measure** --> **Angle** .
2. Pick the center point.
3. Pick the start point.
4. Pick the end point.

   The measurement is displayed until the next window update or redraw (page 46).

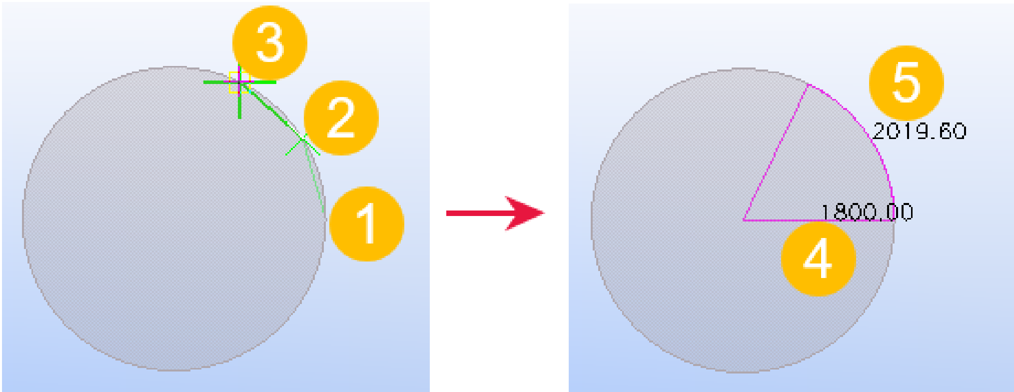
Measure arcs

You can measure the radius and length of an arc in the model.

1. On the **Edit** tab, click **Measure** --> **Arc** .
2. Pick the start point.
3. Pick the middle point.

   This can be any point along the arc between the start and end points.
4. Pick the end point.

The measurement is displayed until the next window update or redraw (page 46).



(1) Start point

(2) Middle point

(3) End point

(4) Arc radius

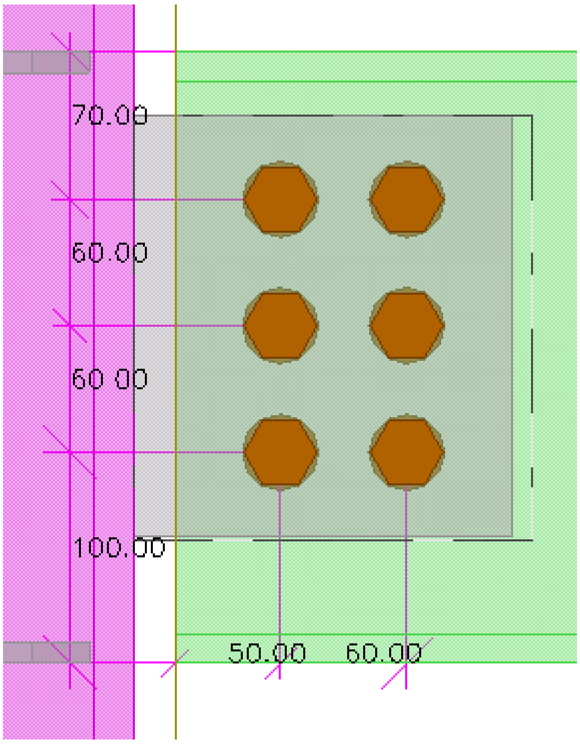
(5) Arc length

Measure bolt spacing

You can measure distances between bolts in a bolt group. Tekla Structures also gives you the edge distances between the bolts and a selected part.

1. On the **Edit** tab, click **Measure** --> **Bolt spacing**.
2. Select a bolt group.
3. Select a part.

The measurement is displayed until the next window update or redraw (page 46).



4.8 Detect clashes

Use the **Clash Check Manager** tool to find objects that collide or that are too close to each other.

Clash Check Manager can check clashes within native Tekla Structures models, within reference models, and between the objects in these models.

Use the clash check settings to define clearances between different model objects.

For information on point cloud clash check, see the instructions about modifying point cloud properties and visualization settings.

You can also use the sections and floors created in **Organizer** to run a controlled clash check.

If you want to use another Tekla Structures model as a reference model, you must export it in the IFC format to be able to use it in clash check. The following reference model file types are supported in clash check:

- IFC
- IFC4
- DWG

- DGN
- SKP
- .tekla (overlay models excluded)

See also

Find clashes in a model (page 696)

Modify, view, and print clash check results (page 703)

Find clashes in a model

You can use clash check to find clashes between reference model objects and pour objects, and between these objects and native Tekla Structures parts, and inside reference models and Tekla Structures models.

Check the model for clashes

1. On the **Manage** tab, click **Clash check**  .
2. In the **Clash Check Manager** dialog box, select the appropriate clash check settings to determine what is included in the clash check.
    - **Between reference models**

      Clashes between reference models are included in the clash check.
    - **Objects in reference models**

      Clashes within reference models are included in the clash check (bolts and welds are not included).
    - **Between reference models and components**

      Clashes between reference models and components are included in the clash check.

      When you have the whole component and this setting selected, the component parts are included in the clash check.

      Note that if you already have the **Select objects in components** selection switch active to select the parts in the component, the **Between reference models and components** setting does not have any effect, even if you have the check box selected in **Clash Check Manager**.
    - **Between parts**

      Clashes between native Tekla Structures parts are included in the clash check.

3. Set the **Minimum distance** between reference model objects and pour objects, and between these objects and native Tekla Structures parts.

   Objects that are closer to each other than the set minimum distance are reported in the clash check results. You can filter out minimum distance type clashes from the clash check results by setting the minimum distance value to `0.0`.

   You can define a minimum distance value within the range from `500 mm` to `-500.0 mm`. With a negative minimum distance, you can define how much overlap is allowed without reporting a clash.

   When checking the minimum distance, **Clash Check Manager** uses the Tekla Structures units set in **File** --> **Settings** --> **Options** --> **Units and decimals**.

   Note that Tekla Structures uses a hard-coded tolerance when checking clashes between native parts. If the native parts overlap less than 0.25 mm, clashes are not reported.

4. In the model, select the objects that you want to include in the clash check.

5. In the **Clash Check Manager** dialog box, click  to check the objects.

   The status bar in **Clash Check Manager** indicates that clash check is in progress.

   Tekla Structures also displays a dialog box with a clash check progress bar. The progress bar first shows the clash check progress for Tekla Structures objects, and then for reference model objects and pours. You can cancel the clash check in the dialog box.

   When the clash check is finished, the status bar in the **Clash Check Manager** dialog box shows a message about listing the clash check results and a progress bar that indicates how long it will take to list all the clashes. When the listing is complete, the status bar message changes to **Ready**.

6. If you want to include more objects in the clash check, select the desired model objects and re-run the clash check.

   New clashes are appended to the end of the list.

7. After removing or modifying objects, re-run the clash check to see if the clash still exists.

   a. Select the desired clashes in **Clash Check Manager**.

   b. Click  to re-run the clash check.

NOTE If you do not wish to check the entire model, you can use **Organizer** to select the sections and floors for which you want to run the clash check. Select a section or a floor in the category tree in **Organizer**, right-click and select **Select in the model**.

NOTE If you cannot find the clashes in the model, change the rendering of objects to **Show only selected part** (**Ctrl+5**) for better visibility.

Note that clashes are not detected if the profiles and positions are identical.

Symbols used in clash check

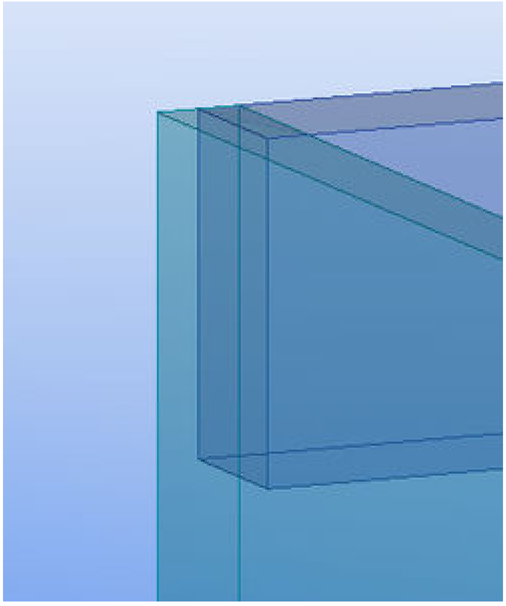
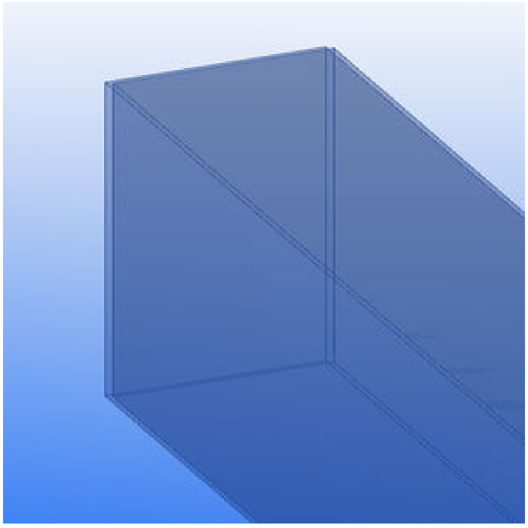
Clash Check Manager uses symbols to indicate the state of a clash.

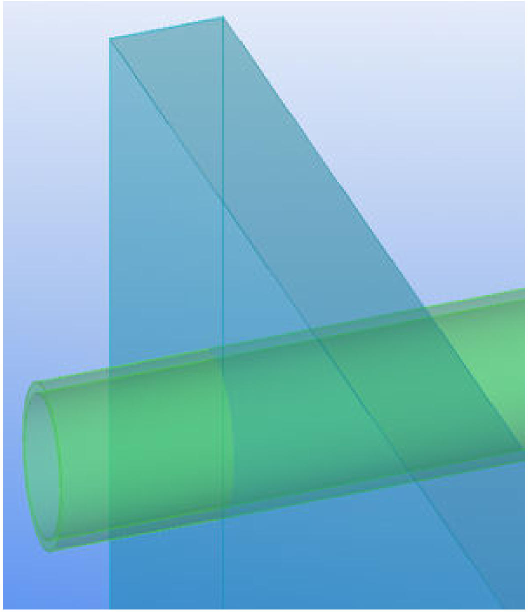
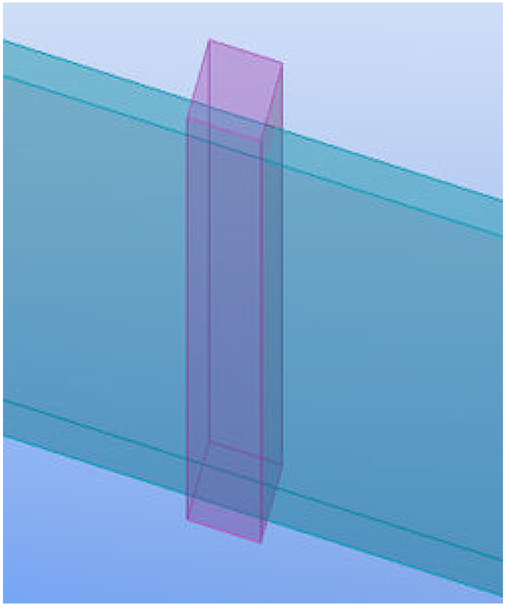
Flag symbol	State	Description
(none)	Active	The default state. The clash is not new, modified, resolved, or missing.
	New	All clashes are marked as new when they are found for the first time.
	Modified	If the object has been modified (for example, if the profile has been changed), the state changes to modified when you re-run the clash check. Only certain object properties affect this flag. To see which properties have an effect, right-click one of the column headings. Both visible and hidden properties affect the flag.
	Resolved	If the objects no longer clash, the state changes to resolved when you re-run the clash check.
	Missing	If one or both of the clashing objects have been removed from the model, the state changes to missing when you re-run the clash check.

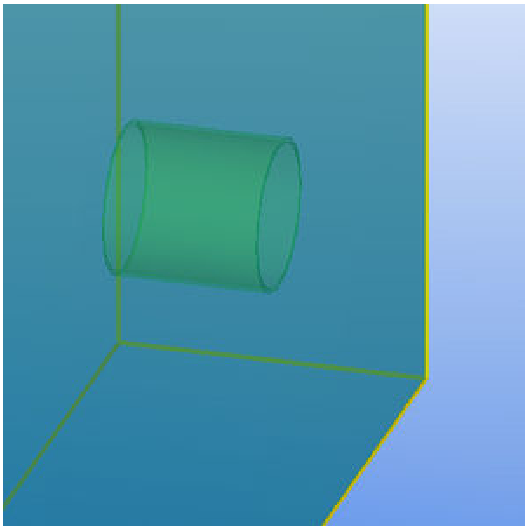
About clash types

Tekla Structures shows the type of each clash in the **Type** column in the **Clash Check Manager** dialog box.

The following clash types can occur:

Type	Description	Example
Clash	The object partly overlaps with another object.	
Clash	Two identical objects overlap completely.	

Type	Description	Example
Clash	Objects intersect with each other in several locations.	
Clash	The object cuts through another object.	

Type	Description	Example
Is inside	The object is inside another object.	
Min distance	The objects are closer to each other than the defined minimum distance. The minimum distance is used between reference model objects and pour objects, and between these objects and native Tekla Structures parts. Minimum distance is not used between native Tekla Structures parts only.	

Open and save clash check sessions

Clash check sessions are saved as `.xml` files in the `..\TeklaStructuresModels\<model>\Clashes` folder. Tekla Structures creates the folder automatically when you open **Clash Check Manager** for the first time.

To open or save sessions in **Clash Check Manager**, do any of the following:

To	Do this
Save the current session	Click in the **Clash Check Manager** dialog box.
Open a session	1. Click in the **Clash Check Manager** dialog box. 2. In the **Open** dialog box, select a session. 3. Click **Open**.
Start a new session	Click in the **Clash Check Manager** dialog box. **Clash Check Manager** clears the list of clashes without running a clash check. If you already have clashes listed in the **Clash Check Manager** dialog box, selecting or clearing the check box of the **Between reference models**, **Between reference models and components**, **Objects in reference models**, **Between parts**, or **Minimum distance** setting will start a new clash check session. Tekla Structures displays a dialog box asking whether you want to save your current clash check session.
Save the current session with another name or in another location	1. Click the arrow next to the button in the **Clash Check Manager** dialog box. 2. Click **Save as** . 3. In the **Save as** dialog box, browse to the folder where you want to save the session. 4. In the **File name** box, enter a new name. 5. Click **Save**.
Save only selected clashes	1. In the **Clash Check Manager** dialog box, select the clashes that you want to save.

To	Do this
	2. Click the arrow next to the button. 3. Click **Save selected** .

Modify, view, and print clash check results

You can modify, view, and print clash check results in the **Clash Check Manager** dialog box.

To open the **Clash Check Manager** dialog box, go to the **Manage** tab on the ribbon and click **Clash check** .

Modify clash check results

You can change the status and priority of clashes, group and ungroup clashes, and add comments for clashes in the **Clash Check Manager** dialog box.

To	Do this
Change the status of a clash	1. Select the clash whose status you want to change. 2. Right-click on the **Status** column. 3. Select a suitable status.
Change the priority of a clash	1. Select the clash whose priority you want to change. 2. Right-click on the **Priority** column. 3. Select a suitable priority.
Group and ungroup clashes	You can combine several clashes into a group so that the clashes are treated as a single unit. 1. Select the clashes that you want to group. 2. Right-click and select **Group** --> **Group** . 3. If you want to add clashes to a group that already exists, select

To	Do this
	the clashes and the group, and then repeat step 2. Note that you cannot create nested groups of clashes. 4. If you want to ungroup clashes: a. Select the clash group to ungroup. b. Right-click and select **Group** --> **Ungroup** .
Add, modify, and remove comments for a clash	You can add comments for clashes and clash groups. For example, you might want to use comments as reminders to yourself and other users. You can modify the comments and remove them. 1. Select the clash or clash group for which you want to add a comment. 2. Right-click and select **Clash Information**. 3. Go to the **Comments** tab. 4. Click  to open the **Add comment** dialog box. 5. Enter your comment in the **Comment** box. 6. Modify the author name and the date, if needed. 7. Click **OK**. To modify a comment, select the comment in the **Clash Information** dialog box and click . To delete a comment, select the comment in the **Clash Information** dialog box and click .

View clash check results

You can search for clashes, and view the history and other details of a clash in the **Clash Check Manager** dialog box.

To	Do this
Search for clashes	Use the **Search** box to find clashes. The more search terms you enter, the more refined your search will be. For example, if you enter `column 8112`, only the clashes that match both of these terms are displayed. 1. Open the clash check session from which you want to find clashes. 2. In the **Search** box, enter the words to search for. The search results are displayed as you type. 3. To narrow down your search, enter more characters. 4. To display all clashes again, click next to the **Search** box.
To zoom in the active view	To zoom in the active view so that the selected objects are shown in the center of the view, double-click a clash in **Clash Check Manager**.
To zoom in on and highlight a clashing object	To zoom in on and highlight a clashing object when there is only one clash, right-click the clash in **Clash Check Manager** and select **Clash Information**.
View the history of a clash	You can view the history of a particular clash. For example, you can view when and by whom the clash was detected. 1. Select a clash or a clash group. 2. Right-click and select **Clash Information**. 3. Go to the **History** tab to view the history information.
View the details of a clash	You can view the details of a clash, such as the profile, material and class of the objects that collide. This can be

To	Do this
	useful especially when you view clash groups that contain more than two objects. 1. Select the clash or clash group whose details you want to view. 2. Right-click and select **Clash Information**. You can view the details of one clash or clash group at a time. If you select more than one clash or clash group, the **Clash Information** option is disabled.
Show or hide a column in **Clash Check Manager**	1. Right-click one of the column headings to open a menu. 2. Click any of the list items to show or hide them. A check mark in front of the item indicates that it is visible. Click the heading of the desired column to alternate between ascending and descending sort order.

Print clash check results

You can print the clash check results from the **Clash Check Manager** dialog box. You can control the print settings in the same manner as in any standard Windows application.

To	Do this
Print the clash check results	1. Click to open the clash check session that you want to print. 2. Click **Print...**. 3. If needed, modify the print settings. 4. Click **Print...**.
Preview before printing	1. Click the down arrow next to and select **Print preview...**. 2. Click one of the page layout buttons to

To	Do this
	select how many pages to view at a time. 3. Click to print the current page.
Define the page setup	Click the arrow next to the button and select **Page setup**.

4.9 Compare parts or assemblies

You can compare two selected parts or assemblies to check, for example, if they get the same position number.

1. Select the objects you want to compare.
    - To compare parts, select two parts in the model.
    - To compare assemblies, select a part in each assembly.
2. On the **Edit** tab, click **Compare**, and then select either **Compare parts** or **Compare assemblies**.

    Tekla Structures displays the results on the status bar. For example, you may get a message that the part geometry differs or parts are positioned differently in the assembly.

See also

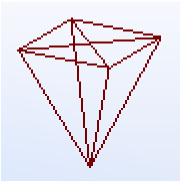
Check the model (page 666)

4.10 View solid errors

You can view solid errors in a log file. This may be needed, for example, if overlapping volumes and faces occur for parts and pour objects, and you need to analyze the error.

1. On the **File** menu, click **Logs** --> **Session history log** .
2. Look for rows that start with the phrase `Solid error`.
3. Click the corresponding `Solid failure position` row to show the solid object error.

A diamond-shaped position locator is displayed in the model to point you to the error.



TIP When you click a `Solid error` row in the log file, hold down the **z** key to center the view to the error location.

4. Redraw the view to hide the position locator.

See also

Troubleshoot pours (page 464)

4.11 Diagnose and repair the model

Use the **Diagnose & repair** commands to check and to repair errors and inconsistencies in the structure of model objects and the library database (`xslib`). By diagnosing and repairing the model you can ensure, for example, that empty assemblies are removed and that unused points and attributes are deleted. Repairing the model also corrects invalid object relations and hierarchies. We recommend you to diagnose and to repair your model regularly to maintain the consistency and integrity of your model databases.

1. On the **File** menu, click **Diagnose & repair**.
2. Select the appropriate diagnose command.

   The errors and inconsistencies found in the model are listed in a report. Some of them are automatically corrected, some of them are warnings that you need to correct manually.

   If a profile, material grade, bolt element or assembly, or reinforcement seems to be missing from the corresponding catalog, your Tekla Structures environment or a catalog file may not be the same as the model's original one.

   If there are solid errors (page 707) in the model objects, these errors are listed in the report. The report is updated every time you run the **Diagnose model** command. (Solid errors are also listed in the session history log file `TeklaStructures_<user>.log`.)

   The following table lists the most common errors and inconsistencies found when diagnosing the model.

Diagnosis result	Description	Action needed
Empty assembly	The assembly does not contain any objects.	a. On the **File** menu, click **Diagnose & repair**. b. Under **Model**, click **Repair model** to delete the assembly.
Missing assembly	A part is not included in any assembly.	a. On the **File** menu, click **Diagnose & repair**. b. Under **Model**, click **Repair model** to create an assembly and to move the part to it.
Illegal profile	An unknown profile has been found.	a. Ensure that you are using the correct Tekla Structures environment. b. Use the model's original `profdb.bin` and `profitab.inp` files and save them in the model folder. c. Reopen the model.
Illegal material	An unknown material grade has been found.	a. Ensure that you are using the correct Tekla Structures environment. b. Use the model's original `matdb.bin` file and save it in the model folder. c. Reopen the model.
Illegal bolt	An unknown bolt element or bolt assembly has been found.	a. Ensure that you are using the correct Tekla Structures environment. b. Use the model's original `screwdb.db` and `assdb.db` files and save them in the model folder. c. Reopen the model.
Illegal reinforcing bar size or grade Illegal reinforcement mesh	Reinforcement with invalid properties has been found.	a. Ensure that you are using the correct Tekla Structures environment. b. Use the model's original `rebar_database.inp` and `mesh_database.inp` files and save them in the model folder. c. Reopen the model.

Diagnosis result	Description	Action needed
Invalid rebar geometry	Reinforcement with undefined geometry has been found.	See Check the validity of reinforcement geometry (page 564).
Solid creation failed	A model object with invalid geometry has been found.	a. Ensure that you are using the correct Tekla Structures environment and profile catalog files. b. Try the following: • If there are cuts and/or fittings in the object, slightly adjust one of them or the object. • Check the pour geometry. (page 464) • Remodel the object.

If the model does not contain any errors or inconsistencies, a message is displayed on the status bar.

See also

Check the model (page 666)

4.12 Find distant objects

When the work area is huge, the model may contain some distant objects that are not easy to find. Use the **Find distant objects** command to find these objects.

1. On the **File** menu, click **Diagnose & repair**.
2. Under **Utilities**, click **Find distant objects**.

   Tekla Structures displays a list of object GUIDs. At the end of the list, Tekla Structures displays additional six objects that have the biggest and smallest x, y, or z coordinates.

```
Min x: Guid: e32a7a28-40db-4597-b160-031d15c1944a
Max x: Guid: 985a39e2-8097-4a9a-8706-9651d08f61c6
Min y: Guid: 8ccb2748-cfe8-4a97-be80-abf453008567
Max y: Guid: 08c8e02d-6a79-4b7e-be70-5370359a1ff5
Min z: Guid: 95eec6e2-d22b-4ae8-8c31-ee8009c028a6
Max z: Guid: f791c3d0-de62-4ced-8d79-03668296f862
```

3. Select an object in the list.

4. Right-click and select a command.

   You can, for example, inquire or delete the object.

See also

Check the model (page 666)

5 Number the model

This section explains how to change numbering settings and apply numbering in Tekla Structures.

Click the links below to find out more:

5.1 What is numbering and how to plan it

Before you can create drawings or accurate reports, you need to number all parts in the model. You do not need to number the model before you create general arrangement drawings.

Numbering is the key to the production output, for example, drawings, reports, and NC files. Numbers are also needed when you export models. Part numbers are vital in the fabrication, shipping, and erection stages of construction. Tekla Structures assigns a mark to each part and assembly/cast unit in a model. The mark includes part or assembly prefix and position number, and other elements, such as profile or material grade. It is useful to identify the parts with numbers to see which parts are similar and which different. Identical parts within a numbering series have the same number, which makes the planning of the production easier.

We recommend that you plan the numbering in an early phase of the project. If other users are using the same model, it is even more important to make a numbering plan that everybody in the project follows. You should have the numbering ready before you create the first drawings and reports.

When planning the numbering, it can be useful to number the model in phases, for example first floor of the building first, then the second, and so on.

Give start numbers in wide ranges so that you do not run out of numbers within a numbering series, and that any numbering series does not overlap with another. For example, start the first floor with the start number 1000 and the second with start number 2000.

If the numbering of a part or assembly is not up to date, a question mark (?) is displayed in the part label and in the **Inquire object** dialog box, for example:

```
 Assembly information
----------------------------
 Assembly Pos:           C/0(?)
 Main part profile:      HEA400
```

See also

Numbering series (page 713)

Identical parts (page 716)

Identical reinforcement (page 717)

Define what affects numbering (page 718)

User-defined attributes in numbering (page 719)

Family numbers (page 719)

Inquire object properties (page 686)

Numbering series

Use numbering series to divide steel parts, assemblies, and cast units into groups. For example, you can allocate a separate numbering series to each phase or part type. Using separate numbering series for different parts speeds up the numbering operation.

The name of a numbering series consists of a *prefix* and a *start number*. You do not always have to define a part prefix (for example, you may want to omit the part prefix for minor parts).

When you run numbering, Tekla Structures compares parts that belong to the same series with each other. All identical parts in the same numbering series are given the same part number.

NOTE Concrete parts are numbered according to the cast unit numbering settings. For example, if the cast unit prefix is **C** and the start number is **1**, concrete parts will get the part prefix **Concrete_C-1**.

This applies also to concrete components whose part position prefix is **Concrete** and start number is **1**.

Example

For example, if you define a numbering series with the prefix P and start number 1001, Tekla Structures numbers that series P1001, P1002, P1003, ...

See also

Plan your numbering series (page 714)

Assign a numbering series to a part (page 715)

Assign a numbering series to an assembly (page 715)

Overlapping numbering series (page 716)

Family numbers (page 719)

Plan your numbering series

Before you start modeling, it is a good idea to plan the numbering prefixes and start numbers you will use for the entire project. Careful planning prevents numbering conflicts.

To save time, include the numbering series in the default part properties for each type of part before you start modeling.

You may want to omit the part prefix for minor parts, such as plates. If you do this, ensure that you assign a **Start number** for that numbering series so that it will not overlap other parts.

Example

One way to plan the numbering series is to create a table:

Part type	Part Prefix	Part Start number	Assembly Prefix	Assembly Start number
Beam	PB	1	AB	1
Vertical brace	PVB	1	AVB	1
Horizontal brace	PHB	1	AHB	1
Rafter	PR	1	AR	1

Part type	Part Prefix	Part Start number	Assembly Prefix	Assembly Start number
Purlin	PP	1	AP	1
Column	PC	1	AC	1
Plate		1001	A	1

See also

Numbering series (page 713)

Overlapping numbering series (page 716)

Assign a numbering series to a part

1. Double-click a part to open the part properties in the property pane.
2. Go to the **Numbering series** section.

   If you are assigning a cast unit numbering series to a concrete part, go to the **Cast unit** section.
3. Define a part prefix and a start number.
4. Click **Modify**.

See also

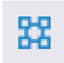
Assign a numbering series to an assembly (page 715)

Numbering series (page 713)

Assign a numbering series to an assembly

To assign a numbering series to an assembly:

To	Do this
To assign a numbering series to an assembly according to its main part	1. Check what is the main part of an assembly. a. On the ribbon, click the down arrow next to , and then select **Assembly objects**. b. Select the assembly. Tekla Structures highlights the main part in orange. c. Press **Esc**.

To	Do this
	2. Ensure that the  **Select objects in assemblies** selection switch is active. 3. Double-click the assembly main part to open the part properties in the property pane. 4. Under **Numbering series**, define an assembly prefix and a start number. 5. Click **Modify**.
To assign a numbering series to an assembly by using the assembly properties	1. Ensure that the  **Select assemblies** selection switch is active. 2. Double-click an assembly to open the assembly properties in the property pane. 3. In the **Assembly numbering** boxes, define an assembly prefix and a start number. 4. Click **Modify**.

See also

Assign a numbering series to a part (page 715)

Numbering series (page 713)

Overlapping numbering series

When you plan numbering, ensure that you reserve enough numbers for each series. If a series overlaps another, Tekla Structures numbers only one of the objects that would have overlapping numbers and leaves the other object unnumbered.

Tekla Structures warns you about series overlaps. View the numbering history log to check which numbers overlap, and then adjust the numbering prefixes and start numbers so that the series will not overlap anymore.

See also

Numbering series (page 713)

View the numbering history (page 729)

Identical parts

Tekla Structures gives parts the same number if the parts are identical **in the fabrication or casting**. If a part is deformed after fabrication or casting (for example if the part is cambered, shortened, or warped), the final geometry on site and in the model may be different.

Tekla Structures treats parts as identical and gives them the same number if the following basic part properties are the same:

- Part geometry
- Casting direction
- Numbering series
- Profile
- Material
- Finish
- Shortening

You can set the degree of tolerance for part geometry in the **Numbering Setup** dialog box. If the geometry of parts differs within this degree of tolerance, Tekla Structures treats the parts as identical for numbering purposes.

Class does not affect numbering. Tekla Structures gives the same number to identical parts that belong to different classes.

If you have created NC files, pop marks and contour marking affect numbering.

See also

Casting direction (page 434)

Define what affects numbering (page 718)

User-defined attributes in numbering (page 719)

Identical reinforcement

Tekla Structures treats reinforcing bars as identical, and gives them the same number, if the following properties are the same:

- Bar geometry
- Numbering series
- Size
- Grade
- Bending radius

To specify how the geometry of reinforcing bars is taken into account in numbering and which type of bar geometry (modeled, rationalized, or fabrication) is used when bars are compared with each other, use the advanced option XS_REBAR_GEOMETRY_TYPE_IN_NUMBERING.

Class does not affect numbering. Tekla Structures gives the same number to identical reinforcing bars that belong to different classes.

The modeling direction of tapered reinforcing bar groups affects numbering. This means that identical bar groups with different modeling directions receive different numbers.

The bar length rounding, step tapering, and some other rebar set settings affect bar geometry and thus the numbering of rebar set bars. The rounding settings that are defined in the `rebar_config.inp` file for drawings and reports do not affect numbering.

See also

Create reinforcement (page 469)

Define what affects numbering

To define which properties affect numbering in your model, modify the settings in the **Numbering Setup** dialog box.

You can have Tekla Structures compare the following properties:

- Holes (if created with the **Bolt** command)
- Part name
- Beam orientation
- Column orientation
- Assembly name
- Assembly phase (XS_ENABLE_PHASE_OPTION_IN_NUMBERING set to `TRUE`)
- Reinforcement
- Embedded objects (affect only cast units)
- Surface treatment (affects only assemblies)
- Welds (affect only assemblies)

If these properties differ, Tekla Structures treats objects as different, and therefore numbers them differently.

For example, if two otherwise identical parts have different names and you select the **Part name** check box, Tekla Structures gives the parts different numbers.

By default, a part retains its number, as long as only one part has that particular number, regardless of the settings in the **Numbering Setup** dialog box.

See also

Adjust the numbering settings (page 721)

What is numbering and how to plan it (page 712)

Identical parts (page 716)

Identical reinforcement (page 717)

User-defined attributes in numbering (page 719)

General numbering settings (page 1005)

User-defined attributes in numbering

You can set in the `objects.inp` file whether a user-defined attribute affects numbering or not. Tekla Structures treats parts and reinforcing bars as different, and therefore numbers them differently, if the values of the user-defined attribute differ.

NOTE Only user-defined attributes of parts and reinforcement can affect numbering. User-defined attributes of other objects, such as phases, projects and drawings do not affect numbering.

If you want Tekla Structures to consider a user-defined attribute when numbering, set the `special_flag` option of the attribute to `yes` in the `Part attributes` section of `objects.inp`. For reinforcement, you need to set `special_flag` to `yes` **also** in the `Reinforcing bar attributes` section. Tekla Structures assigns different numbers to parts or reinforcement that are otherwise identical but have different values for this user-defined attribute.

If you want Tekla Structures to ignore a user-defined attribute when numbering, set the `special_flag` option to `no` in `objects.inp`.

See also

Examples of user-defined attributes (UDAs) for parts (page 323)

Select and change the profile or material of a part (page 320)

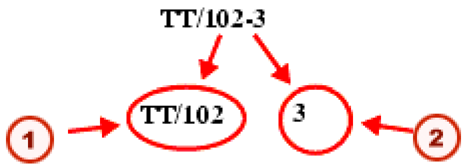
Define what affects numbering (page 718)

Family numbers

With family numbering you can group objects within the same numbering series into different “families”. This can be used, for example, to find similar precast cast units that can be cast in the same bed.

Family numbering is not available for cast-in-place cast units.

When you use family numbering, the cast unit position numbers consist of a *family number* and a *qualifier*. For example:



1. Family number
2. Qualifier

Assemblies and precast cast units that match the comparison criteria you define in the **Numbering Setup** dialog box get the same family number. However, if they have the same family number but different part geometry or materials, they get unique qualifier numbers.

See also

Numbering series (page 713)

Assign family numbers (page 720)

Change the family number of an object (page 721)

Example: Use family numbers (page 741)

Assign family numbers

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog box.
2. Go to the **Family numbering** tab.
3. Define which numbering series to assign family numbers to.
    a. Click **Add series** to open the **Add series** dialog box.

       Tekla Structures displays all the assembly and cast unit numbering series in the model.

    b. Select a numbering series from the list, and then click **Add**.

       The numbering series appears in the family numbering list.
4. Under **Compare**, select the properties that need to be identical for the members of the same family.

Define comparison criteria for each numbering series separately.

Select at least one check box, but not all of them. If you select all the check boxes, the family number will be the same as the normal assembly position, and the qualifier number will be 1 for all. If you do not select any check boxes, only one family number per series is assigned.

5. Click **Apply**.

   Tekla Structures stores the settings in the numbering database file (`<model_name>.db2`) in the current model folder the next time you save the model.

6. If you are assigning family numbers to parts that have already been numbered, clear the existing numbers.

7. Update numbering in the model.

   Tekla Structures assigns a family number to all objects in the numbering series.

See also

Family numbers (page 719)

Clear existing numbers (page 726)

Change the family number of an object

You can change the family number and/or family qualifier of an object.

1. Select the objects whose family numbers you want to change.
2. On the **Drawings & reports** tab, click **Change number** --> **Change family number** .
3. In the **Assign Family Number** dialog box, enter the desired values in the **Family number** and **Family qualifier** boxes.
4. Click **Assign**.

See also

Family numbers (page 719)

5.2 Adjust the numbering settings

You can adjust the numbering settings to better suit your needs. This should be done early in the project, before creating any drawings or reports. Do not change your numbering conventions in the middle of a project.

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog box.

2. Modify the settings (page 1005) if necessary.

   For example, you can define which part properties affect numbering in your model. Using the default settings is effective in most cases.

3. Click **Apply** or **OK**.

   NOTE Always check and repair numbering after you have changed the numbering settings.

See also

Define what affects numbering (page 718)

Numbering settings during a project (page 746)

Repair numbering errors (page 730)

5.3 Number parts

Use the **Number modified objects** command to number all parts that have been created or modified since the last numbering. If this is the first time you run numbering for this model, all parts in the model are new and will therefore be numbered.

To number new and modified parts:

- On the **Drawings & reports** tab, click **Perform numbering** --> **Number modified objects** .

  Tekla Structures numbers the parts according to the numbering settings (page 1005).

See also

Number a series of parts (page 722)

Number assemblies, cast units, and rebar assemblies (page 723)

Number reinforcement (page 724)

Number welds (page 725)

Save preliminary numbers (page 725)

Number parts by design group (page 738)

Number a series of parts

Use the **Number series of selected objects** command to only number parts that have a certain prefix and start number. This enables you to limit the numbering to certain series of objects only, which can be useful in large models.

Before you start, we recommend that you plan the numbering series carefully, and split the model into smaller numbering series, for example by area or phase.

1. Select the parts that have the desired prefix and start number.

   Only parts that have the same prefix and start number as the selected part will be numbered.

2. On the **Drawings & reports** tab, click **Perform numbering** --> **Number series of selected objects**.

   Tekla Structures numbers all parts in the specified numbering series.

See also

Example: Number selected part types (page 743)

Example: Number parts in selected phases (page 744)

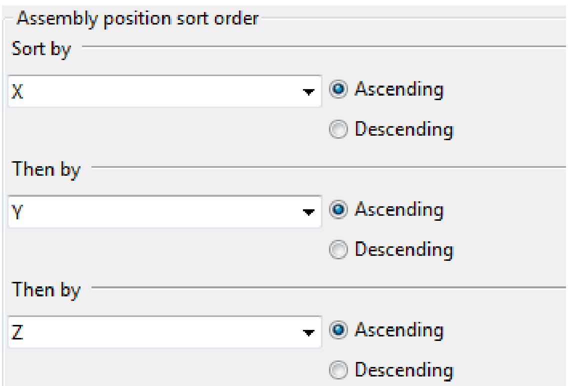
Number assemblies, cast units, and rebar assemblies

To number assemblies, cast units, and rebar assemblies, use the same numbering commands as for numbering parts. Before numbering, you can modify the sort order, which defines how assemblies, cast units, and rebar assemblies are given their position numbers. Sorting does not affect part position.

1. If needed, modify the sort order of assemblies and cast units.

   a. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog box.

   b. Modify the sort order by selecting options from the **Sort by** and **Then by** lists.

   The default sort order is XYZ. You have the following options:

- The x, y or z coordinates of the main part of the assembly or cast unit, or the main reinforcement object of the rebar assembly.

  The sorting is based on the center of gravity (COG) location of the assembly or cast unit. Tekla Structures finds the center of gravity for each assembly and cast unit and compares them in the order you defined.

- The user-defined attribute of an assembly or the main part

  If your sorting is based on user-defined attributes, Tekla Structures displays a list that includes all the available user-defined attributes.

   c. Click **Apply** or **OK** to save the changes.

2. If needed, modify the other numbering settings (page 1005).
3. On the **Drawings & reports** tab, click **Perform numbering** --> **Number modified objects** to number the model.

NOTE If you add new parts in the model, parts that have already been numbered are **not** renumbered to suit the sorting order. In this case you should check and repair the numbering of those parts.

See also

Repair numbering errors (page 730)

Number reinforcement

To number reinforcement, use the same numbering commands as for numbering parts.

To specify how the geometry of reinforcing bars is taken into account in numbering and which type of bar geometry is used when bars are compared with each other, use the advanced option XS_REBAR_GEOMETRY_TYPE_IN_NUMBERING.

Before numbering rebar assemblies, you may want to modify the sort order settings similarly as when numbering assemblies or cast units (page 723).

Note that reinforcement may affect the numbering of cast units. To force Tekla Structures to give otherwise identical cast units different numbers if they have different reinforcement, select the **Reinforcing bars** check box in the **Numbering Setup** dialog box.

Part numbering and cast unit numbering do not affect the numbering of reinforcement.

See also

Number parts (page 722)

Identical reinforcement (page 717)

Define what affects numbering (page 718)

User-defined attributes in numbering (page 719)

Assign running numbers to reinforcement (page 566)

Number welds

Use the **Number welds** command to assign numbers to welds. Weld numbers are displayed in drawings and reports.

1. On the **Drawings & reports** tab, click **Perform numbering** --> **Number welds** to open the **Weld Numbering** dialog box.
2. If needed, modify the weld numbering settings (page 1007).

   For example, you can define whether to assign numbers for **All welds** or **Selected welds**.
3. If you selected to assign numbers to certain welds only, select the welds.
4. Click **Assign numbers** to start numbering welds.

See also

Number parts (page 722)

Save preliminary numbers

Preliminary mark is a user-defined attribute that defines the part position number. You can save the current part position numbers as preliminary numbers for selected parts. The previous preliminary numbers are overridden.

1. Select the parts.
2. On the **Drawings & reports** tab, click **Numbering settings** --> **Save preliminary numbers** .

The preliminary numbers appear on the **Parameters** tab in the parts' user-defined attributes.

See also

Number parts (page 722)

5.4 Change existing numbers

Use the **Change number** commands to change the existing part, assembly, multi-position, or family numbers into something you have defined yourself. These commands do not change the numbering series of parts. To prevent drawing, modeling, and fabrication errors, Tekla Structures does not allow you to use identical numbers for two different assemblies or parts.

1. On the **Drawings & reports** tab, click **Change number** and select one of the following commands:

   - **Change part number**
   - **Change assembly number**
   - **Change part multinumber**
   - **Change assembly multinumber**
   - **Change family number**

   The corresponding dialog box appears. For example, if you select the **Change assembly number** command, the **Assign Assembly Number** dialog box appears.

2. Select a part in the model.
3. In the dialog box that appeared (for example **Assign Assembly Number**), click **Get** to view the current numbering properties of the part.
4. Enter the numbering properties you want to use for this part.

   Note that the position numbers you enter here are not absolute numbers. For example, if the start number of the series is `100`, the position numbers refer to numbers in that series. Therefore, position number `1` is actually `100`, position number `2` is `101`, position number `3` is `102`, and so on.

5. If you are changing the assembly number of selected parts, ensure that the **Assign to** option is set to **Selected objects only**.

   Otherwise, all parts with the same original number will be renumbered.

6. Click **Assign** to change the number.

   If the number you specified is already in use, Tekla Structures displays a warning and keeps the original number.

   Tekla Structures also displays a warning if the position number is higher than the highest current number. This is for information only and the number is still changed.

See also

Number parts (page 722)

5.5 Clear existing numbers

Use the **Clear** commands to permanently remove the current position numbers of parts. Next time you run numbering, Tekla Structures assigns new numbers to these parts, irrespective of what their previous numbers were.

1. Select the parts whose numbers you want to clear.
2. On the **Drawings & reports** tab, click **Change number** and select one of the following commands:
   - **Clear part and assembly numbers**
   - **Clear part numbers**
   - **Clear assembly numbers**
   - **Clear reinforcing bar numbers**

   Tekla Structures removes the position numbers of the selected parts.

See also

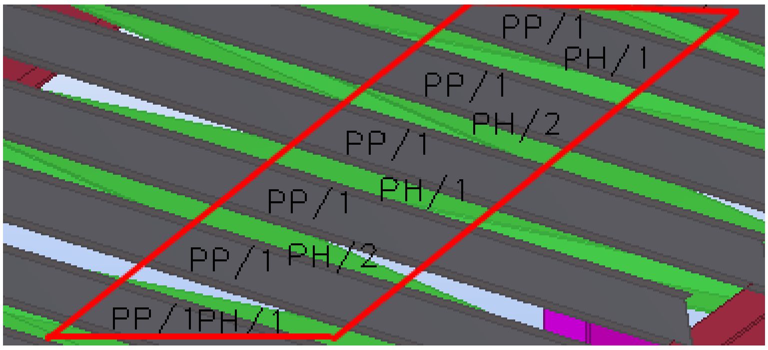
Number parts (page 722)

5.6 Check the numbering

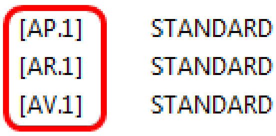
You can check the position numbers in many places:

- You can add the position numbers to the part labels.
  1. In the model, make sure the **Select views** selection switch is active.
  2. Double-click the background to open the **View Properties** dialog box.
  3. Click **Display...** to open the **Display** dialog box.
  4. Go to the **Advanced** tab and add **Part position** to the **Part label**.
  5. Click **Modify**.

  The part labels now contain the position numbers.



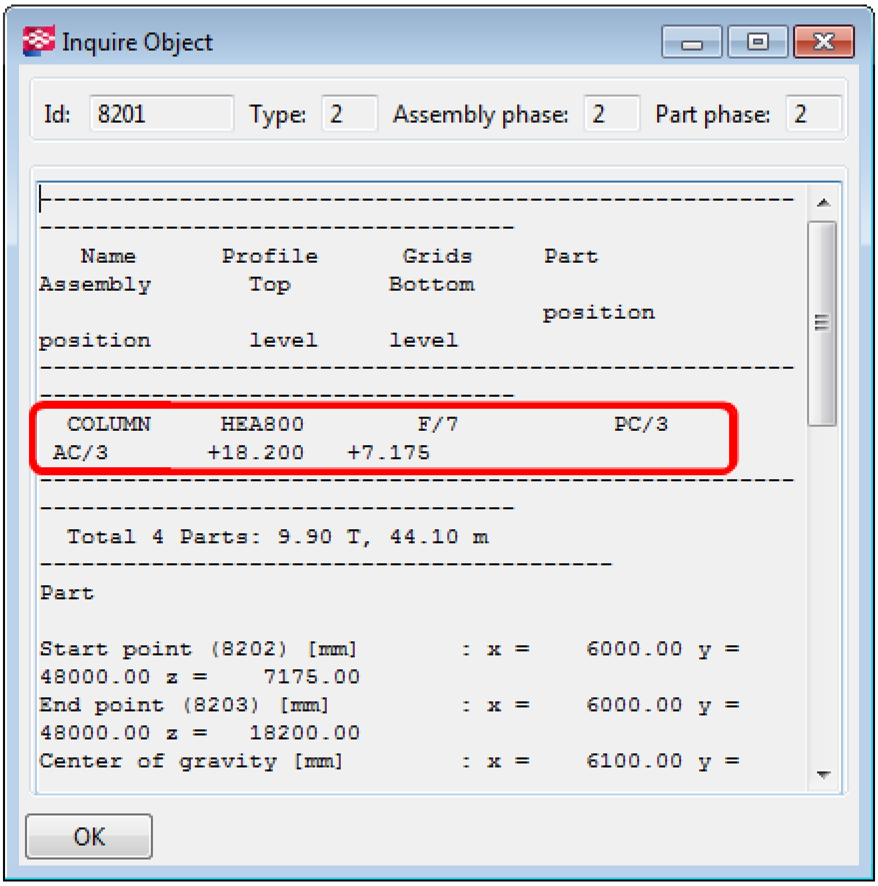
- You can check the part number in .



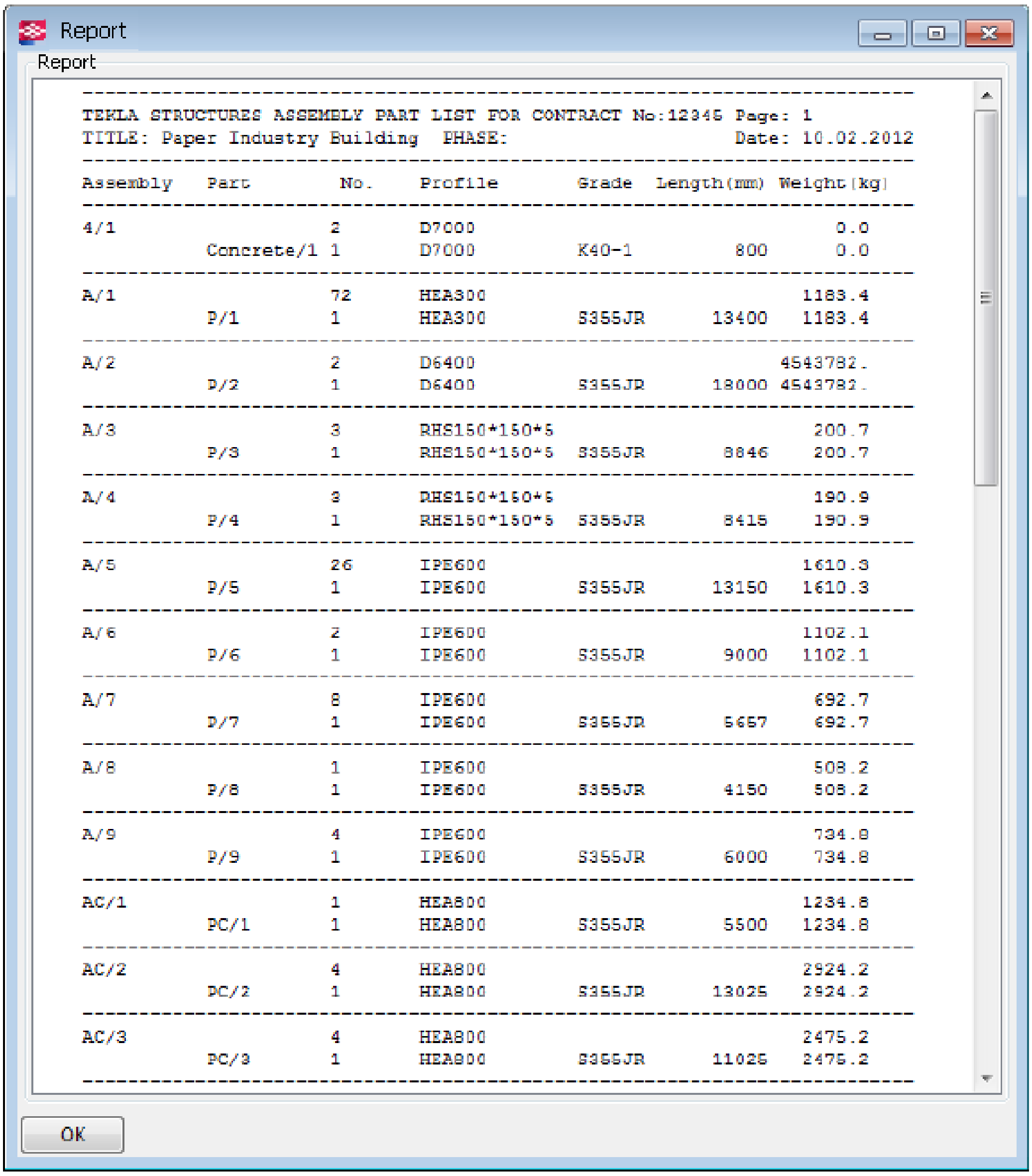
- The drawing label can show the position number and the amount of identical parts.

GENERAL NOTES:	ALL HOLES ARE		0.0	mm UNLESS NOTED		
	ALL WELDS ARE		0.0	mm F.W UNLESS NOTED		
MATERIAL LIST FOR ASSEMBLY MK'D			AC/5		3	No. Required
Mark	Profile	Material	No.	Length	Area	Weight
PC/5	HEA800	S355JR	1	18200	49.1	4086.1
				Total	49.1	4086.1

- You can use the **Inquire** commands.



• You can create reports that list your assembly and part positions.



TEKLA STRUCTURES ASSEMBLY PART LIST FOR CONTRACT No:12345 Page: 1
TITLE: Paper Industry Building PHASE: Date: 10.02.2012

Assembly	Part	No.	Profile	Grade	Length(mm)	Weight(kg)
4/1		2	D7000			0.0
	Concrete/1	1	D7000	K40-1	800	0.0
A/1		72	HEA300			1183.4
	P/1	1	HEA300	S355JR	13400	1183.4
A/2		2	D6400			4543782.
	P/2	1	D6400	S355JR	18000	4543782.
A/3		3	RHS150*150*5			200.7
	P/3	1	RHS150*150*5	S355JR	8846	200.7
A/4		3	RHS150*150*5			190.9
	P/4	1	RHS150*150*5	S355JR	8415	190.9
A/5		26	IPE600			1610.3
	P/5	1	IPE600	S355JR	13150	1610.3
A/6		2	IPE600			1102.1
	P/6	1	IPE600	S355JR	9000	1102.1
A/7		8	IPE600			692.7
	P/7	1	IPE600	S355JR	5657	692.7
A/8		1	IPE600			508.2
	P/8	1	IPE600	S355JR	4150	508.2
A/9		4	IPE600			734.8
	P/9	1	IPE600	S355JR	6000	734.8
AC/1		1	HEA800			1234.8
	PC/1	1	HEA800	S355JR	5500	1234.8
AC/2		4	HEA800			2924.2
	PC/2	1	HEA800	S355JR	13025	2924.2
AC/3		4	HEA800			2475.2
	PC/3	1	HEA800	S355JR	11025	2475.2

See also

Repair numbering errors (page 730)

5.7 View the numbering history

To view the numbering history:

• On the **File** menu, click **Logs** --> **Numbering history log** .

Tekla Structures displays the numbering log file.

5.8 Repair numbering errors

We recommend that you check and repair numbering in the model every now and then, especially before producing drawings or reports.

NOTE If you work with Tekla Model Sharing or in the multi-user mode, it is very important that you repair numbering regularly.

---

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog box.
2. Ensure that the option **Compare to old** is selected for **New** parts.
3. Ensure that one of the following options is selected for **Modified** parts:
   - **Compare to old**
   - **Keep number if possible**
4. Click **OK** to save the changes.
5. Unless you want to repair the entire model, select the objects whose numbering you want to repair.
6. On the **File** menu, click **Diagnose & repair** and select one of the following commands under **Numbering**:
   - **Diagnose and repair numbering: all**

     This command numbers all parts and assemblies, even the unmodified ones.
   - **Diagnose and repair numbering: series of selected objects**

     This command numbers all parts and assemblies that have the same prefix and start number as the selected part.

     Note that Tekla Structures assigns the position number of the oldest part or assembly to all identical parts, even if a newer part or assembly has a smaller position number.

TIP To manually assign a certain position number on a part or assembly, use the **Change number** command after repairing numbering in the model.

See also

Change existing numbers (page 725)

5.9 Renumber the model

Use the **Renumber all** option when the numbering needs to be started over. This option permanently removes the existing position numbers and resets them with new ones. Any existing drawings will also be removed.

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog box.
2. Select the **Renumber all** check box.
3. Click **Apply** or **OK**.
4. On the **Drawings & reports** tab, click **Perform numbering** --> **Number modified objects** .
5. When you are asked to confirm the renumbering of the model, click **Yes**.

   Tekla Structures renumbers the entire model.

See also

Change existing numbers (page 725)

Clear existing numbers (page 726)

5.10 Control numbers

Control numbers are additional numbers that can be used to identify parts in a model. Use control numbers if you need to give additional, unique numbers to assemblies or cast units, regardless of their position numbers.

Control numbers can be useful, for example, when delivering a large number of similar wall elements to the site. To successfully pack and unpack the load, you need to plan the order of the wall elements already when the order is shipped. Although all of the wall elements may have the same cast unit position number, you can assign a unique control number to each individual wall element.

See also

Assign control numbers to parts (page 731)

Control number order (page 732)

Display control numbers in the model (page 734)

Remove control numbers (page 735)

Lock or unlock control numbers (page 736)

Example: Use control numbers to indicate the erection order (page 736)

Assign control numbers to parts

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Assign control numbers** to open the **Create control numbers** dialog box.
2. Indicate which parts to assign control numbers to.
   - To number the entire model, do not select any parts.
   - To only number specific parts, select the parts you want to number.
3. If you want to assign control numbers only to parts in a specific numbering series:
   a. In the **Numbering** list, select **By numbering serie**.
   b. Enter the **Prefix** and **Start number** in the corresponding boxes.
4. Define the control numbers to be used.
   a. In the **Start number of control numbers** box, enter the first control number to be used.
   b. In the **Step value** box, define the control number interval.

      For example, to assign the control numbers 2, 5, 8, 11, and so on, enter 2 in the **Start number of control numbers** box and 3 in the **Step value** box.
5. Use the **Renumber** list to specify how to treat parts that already have control numbers.
   - Select **No** to keep the existing control numbers.
   - Select **Yes** to replace the existing control numbers with new ones.
6. Use the **First direction**, **Second direction**, and **Third direction** lists to define the order of control numbers.
7. In the **Write UDA to** list, select where to save the control numbers. The control number will appear on the **Parameters** tab in the user-defined attributes dialog box either for:
   - **Assembly**
   - **Main part**
8. Click **Apply** to save the changes.
9. Click **Create** to number the parts.

See also

Control number order (page 732)

Control number settings (page 1008)

Control number order

When you assign control numbers, you must define in what order to assign them. The order is based on the location of each part on the global coordinate system.

The options are:

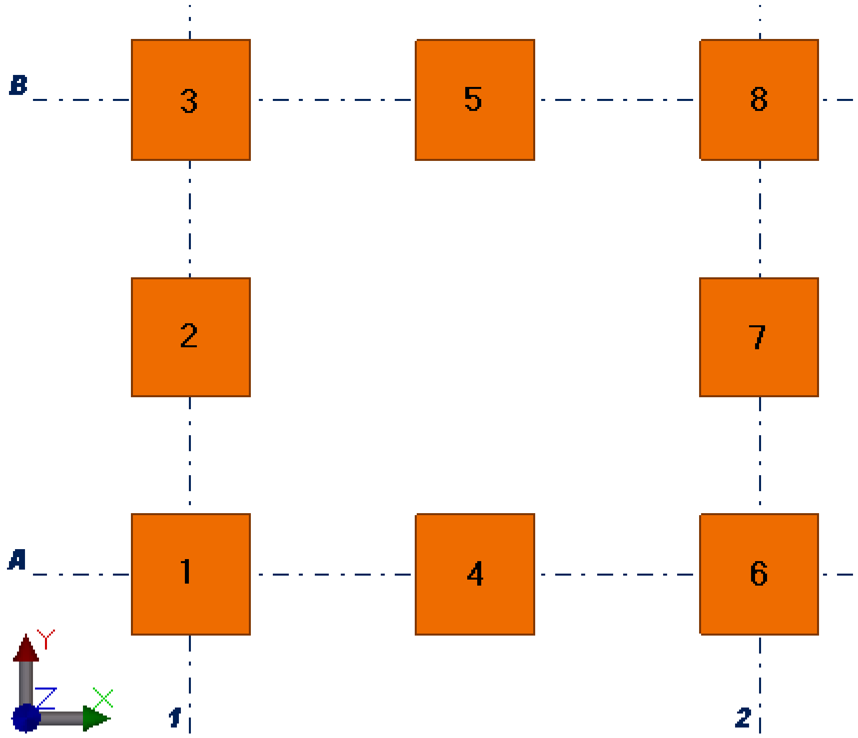
- **None**
- **X**
- **Y**
- **Z**
- **-X**
- **-Y**
- **-Z**

With positive directions (X, Y, and Z), the parts with the lowest coordinate value are numbered first. With negative directions (-X, -Y, and -Z), the parts with the highest coordinate value are numbered first.

For example, if the first direction is X, the second direction is Y, and the third direction is Z, numbering starts from the parts that have the lowest x coordinate value. If multiple parts have the same x coordinate, also their y coordinates are compared. If multiple parts have the same x and y coordinates, also their z coordinates are compared.

Example

In the following example, the first direction is X and the second direction is Y. The numbers 1–8 indicate the control numbers.



See also

Assign control numbers to parts (page 731)

Display control numbers in the model

If the control numbers are not displayed in the model, you can set them visible using the display settings.

1. Double-click the view to open the **View Properties** dialog box.
2. Click **Display...** and go to the **Advanced** tab.
3. Select the **Part label** check box.
4. In the **Properties** list, select **User-defined attributes**, and then click **Add >**.

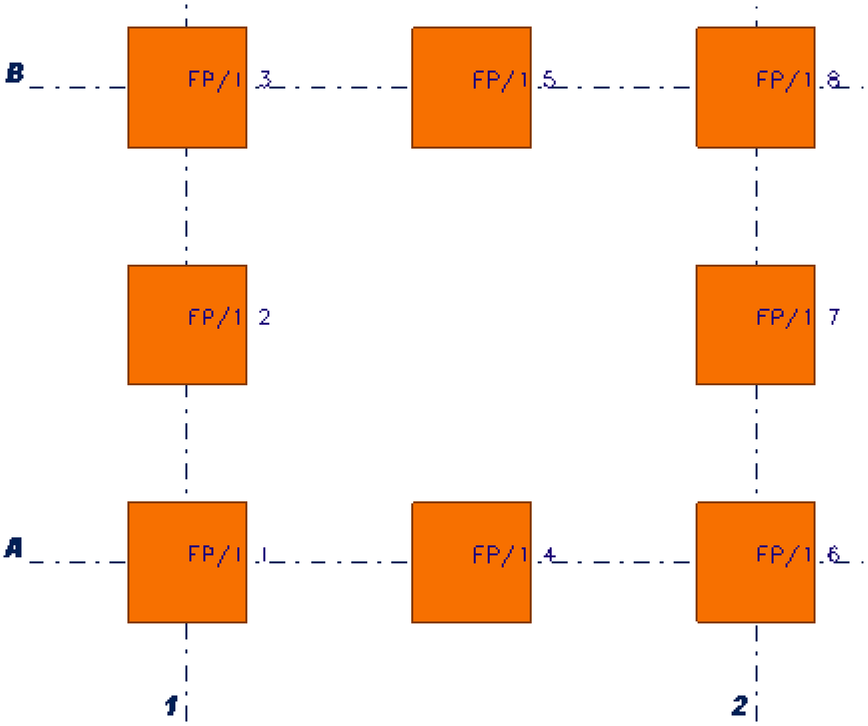
   The **Part label** dialog box is displayed.
5. Enter `ACN` and click **OK**.

   The property is moved to the **Part label** list.
6. Click **Modify**.

   The control numbers are displayed in the model, right after the part position numbers.

Example

In the following example, the numbers 1–8 indicate the control numbers.



See also

Control numbers (page 731)

Remove control numbers

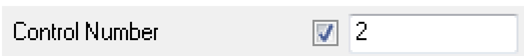
If needed, you can remove existing control numbers from all or some of the parts. Do not remove control numbers unless you are absolutely sure that they are no longer needed.

NOTE **Removing** of control numbers is different than **reassigning** of control numbers. If you only want to reassign new control numbers to parts that already have control numbers, use the **Renumber** option in the dialog box.

---

1. Double-click a part to open the part properties in the property pane.
2. Click the **User-defined attributes** button.

   The current control number of the part is displayed on the **Parameters** tab, in the **Control Number** box. For example:

3. Remove the existing control number from the box.
4. Click **Modify** to apply the change.

See also

Control numbers (page 731)

Lock or unlock control numbers

To prevent other users from changing the control numbers of some or all of the parts in the model, use the **Lock/unlock control numbers** command. Later on, if the control numbers need to be changed, you can use the same command for unlocking the numbers.

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Lock/ unlock control numbers** to open the **Lock/Unlock control numbers** dialog box.
2. Define which parts' control numbers to lock or unlock.
   - To lock or unlock the control numbers of all parts, do not select any parts in the model.
   - To lock or unlock the control numbers of specific parts only, select the parts in the model.
3. In the **Status** list, select **Lock** or **Unlock**.
4. Click **Apply** to save the changes.
5. Click **Create** to lock or unlock the numbers.

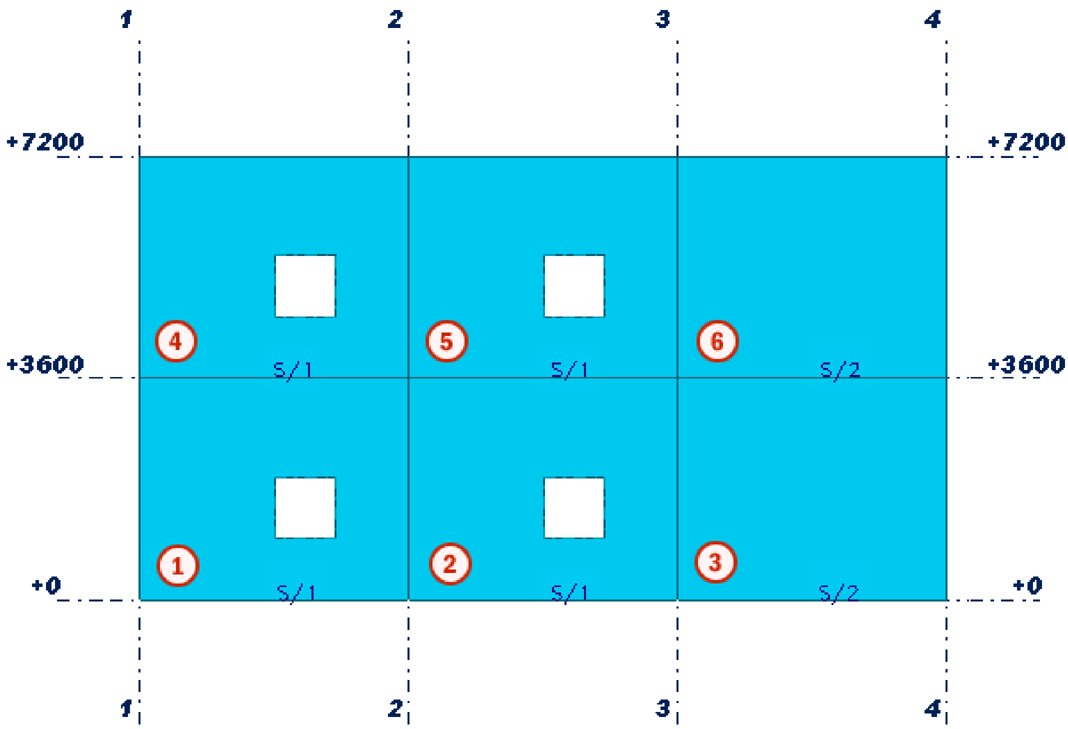
See also

Control numbers (page 731)

Example: Use control numbers to indicate the erection order

This example shows how to assign control numbers to six concrete walls. As four of these walls have the same cast unit position, you cannot make a clear distinction between the cast units based on their position number. That is why each wall will get a unique identifier that indicates their erection order at the site. The erection order also affects the order of shipping. For example, the wall number 1 needs to be shipped on top of the delivery, because it will be erected first on the site; the wall number 2 should be the second on the bundle, because it will be erected next; and so on.

The following image represents the desired end result.



1 Erected as 1st

2 Erected as 2nd

3 Erected as 3rd

4 Erected as 4th

5 Erected as 5th

6 Erected as 6th

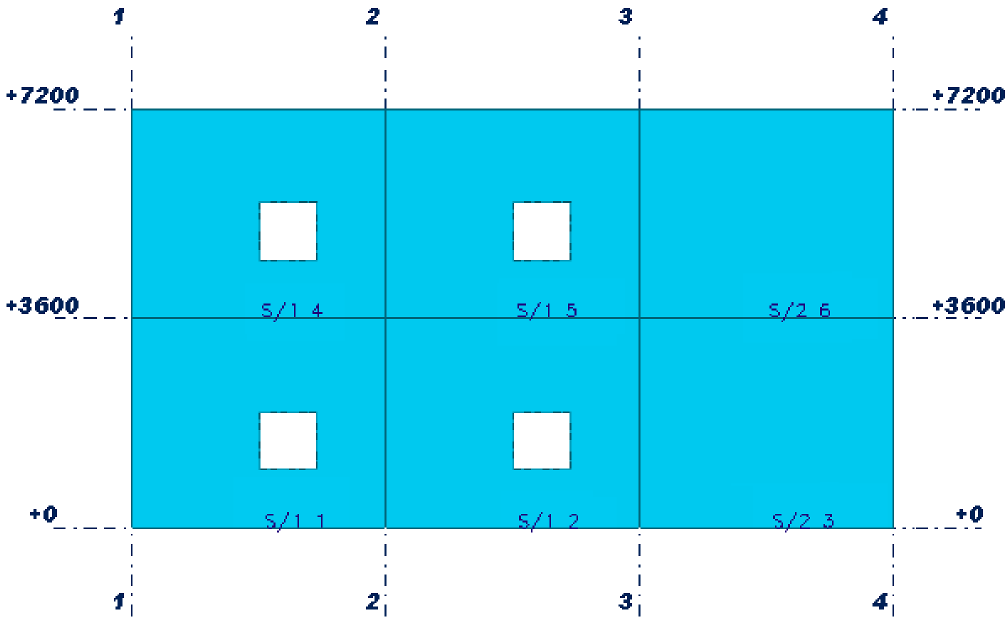
To assign control numbers to the concrete walls:

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Assign control numbers** to open the **Create control numbers** dialog box.
2. Select the six concrete walls.
3. Define that you want to assign control numbers only to parts in the numbering series S with the start number 1.
    a. In the **Numbering** list, select **By numbering serie**.
    b. In the **Prefix** box, enter `S`.
    c. In the **Start number** box, enter `1`.

4. Define that you want to use the numbers 1–6 as the control numbers for these concrete walls.
    a. In the **Start number of control numbers** box, enter `1`.
    b. In the **Step value** box, enter `1`.
5. Define that you want to first number the walls with identical z coordinates in the order they appear in the positive x axis.
    a. In the **First direction** list, select **Z**.
    b. In the **Second direction** list, select **X**.
6. Click **Apply** to save the changes.
7. Click **Create** to number the concrete walls.

    Each concrete wall gets a unique control number, as shown in the following image.

TIP If you cannot see the control numbers in the model, modify the display settings. For more information on the required settings, see Display control numbers in the model (page 734).



5.11 Number parts by design group

You can number parts by design groups so that you can differentiate the parts from each other in drawings and reports. The design group numbers can be used in engineering documents or as preliminary numbers.

Design group numbers consist of a prefix, a separator, and a number or a letter.

Use the **Design group numbering** application to assign prefixes and numbers or letters to parts on the basis of design groups. The **Design group numbering** application groups parts that match a selection filter into a design group, numbers them, and optionally compares the part lengths. The application also compares the user-defined attributes of parts that are set to affect numbering.

NOTE The **Design group numbering** application only numbers parts that have a profile extruded to create the length of the part, such as beams, columns, panels, and footings. Contour plates, slabs, or items are not numbered.

Before you start:

- Create the needed selection filters that define the design groups.
- If you want to use certain letters in design group numbering, list the allowed letters using the advanced option XS_VALID_CHARS_FOR_ASSEMBLY_POSITION_NUMBERS. By default, letters A–Z are allowed.
- In multi-user or Tekla Model Sharing models, make sure that only one of the users runs the **Design group numbering** application.

To number parts by their design group:

1. In the model, 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 **Design group numbering** to start the application.
4. In the **Design group numbering** dialog box:
    a. Click **Add group** to create design group numbering settings for parts matching a selection filter.
        - Select the filter in the **Group filter** column.

          The selection filters are read from specific folders in the standard folder search order.
        - Enter the design group prefix and start number or start letter you want to use for the parts in this group.
        - In the **Compare length** column, define whether the part lengths are compared or not.

    b. Repeat step 4a for all part groups that you want to number by design group.

c. If needed, change the order of the groups by using the **Move up** and **Move down** buttons.

   If a part belongs to several groups, the last group filter in the list overrides the previous ones.

d. If you want to compare part lengths, define the length tolerance.

   For example, if you enter 0, parts must be exactly the same length to receive the same design group number (or letter). If you enter 2, part lengths can differ 2 mm from each other.

   The default tolerance is 0.05 mm.

e. Enter a number separator that is used to separate the design group prefix and number or letter in drawing marks and in reports. For example, enter - .

   We recommend that you do not change the separator during the project.

f. To reuse old, unneeded numbers or letters, select the **Reuse old numbers** check box.

g. In **Number using letters**, select whether to use letters or not.

h. In **Renumber all**, select whether to renumber all parts or not.

i. To number the parts by design group, click **Perform numbering**.

   A design group number is saved as the user-defined attribute `DESIGN_GROUP_MARK` of each part.

   By default, the user-defined attribute `DESIGN_GROUP_MARK` is available in the `objects.inp` file in the Engineering configuration in the default and US environments.

j. To create a report to show the numbering results, select whether to create a report of all or the selected parts, and then click **Create report**.

   Tekla Structures shows the report in the **List** dialog box and also saves the report. The report is saved as `dgnReport.txt` in the folder defined by the advanced option XS_REPORT_OUTPUT_DIRECTORY. In the Default environment, the report is saved in the `\Reports` folder under the current model folder.

   When you select a row in the **List** dialog box, Tekla Structures highlights and selects the corresponding part in the model.

   If the numbering of a part is not up to date, meaning that the part has been modified after numbering, a question mark (?) is added after the design group number.

5. To show the design group numbers in drawing marks or in reports, use the user-defined attribute `DESIGN_GROUP_MARK`.

See also

Create new filters (page 153)

5.12 Numbering examples

This section gives some examples of numbering the model.

Click the links below to find out more:

Example: Number identical beams (page 741)

Example: Use family numbers (page 741)

Example: Number selected part types (page 743)

Example: Number parts in selected phases (page 744)

Example: Number identical beams

This example explains how different numbering settings result in different part numbers when you modify a part.

To number identical beams:

1. Create three identical beams with the numbering series prefix P and start number 1.
2. Number the model. All the beams have the part position number P1.
3. Modify one of the beams.
4. Number the model. You should now have two beams P1 and one P2.
5. Change beam P2 to be identical to the others.
6. Number the model.

   Depending on the numbering settings in the **Numbering Setup** dialog box, Tekla Structures assigns one of the following part position numbers to the modified part:

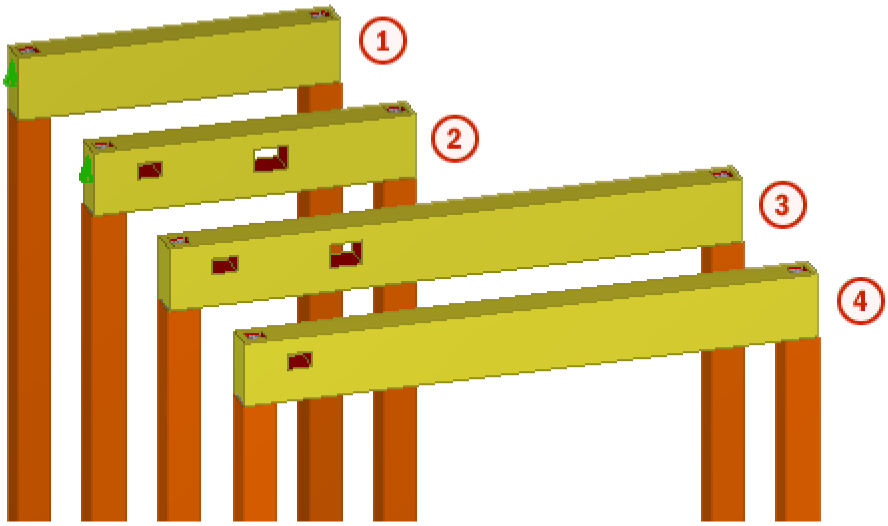
   - **Compare to old**: P1
   - **Keep number if possible**: P2
   - **Take new number**: P3

See also

Number parts (page 722)

Example: Use family numbers

In this example, the following four beams have the numbering series prefix B and the start number 1. The parts have the same main profile, and each pair has the same length, but the holes are different.



1 Assembly position: B/1

2 Assembly position: B/2

3 Assembly position: B/3

4 Assembly position: B/4

We use the following family numbering settings:

- **Use family numbering for series**: add series **B/1**
- **Compare**: select options **Main part profile** and **Overall length**

With the given family numbering criteria, Tekla Structures divides the beams into two families. All beams have the same profile, but each pair has a different length. Within both families the beams get different qualifiers because they have different holes.

- The first beam gets the assembly position number B/1-1
- The second beam gets the assembly position number B/1-2
- The third beam gets the assembly position number B/2-1
- The fourth beam gets the assembly position number B/2-2

See also

Family numbers (page 719)

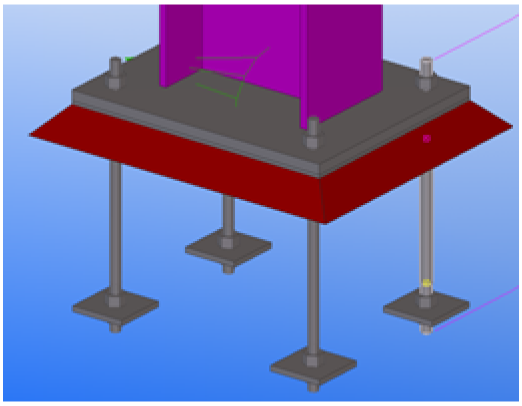
Example: Number selected part types

This example shows how different numbering settings can be used for different part types. We will use one set of numbering settings for steel anchor rods, and another one for steel columns. Note that the **Number series of selected objects** command numbers all parts that have the same assembly prefix.

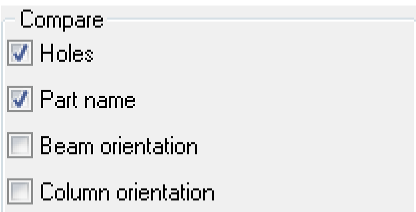
To number anchor rods and columns:

1. Create steel columns.
2. Create anchor rods with the numbering series prefix AR and start number 1.

   Ensure that this numbering series is different to any other parts or assemblies in the model.

3. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog box.
4. Ensure that the **Column orientation** check box is not selected, and then click **Apply**.

5. Select one of the anchor rods in the model.
6. On the **Drawings & reports** tab, click **Perform numbering** --> **Number series of selected objects**.

   All parts with the AR prefix and start number at 1 are numbered.

7. Wait until the anchor rod numbering is complete.
8. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog box.
9. Select the **Column orientation** check box, and then click **Apply**.
10. Select one of the steel columns in the model.
11. On the **Drawings & reports** tab, click **Perform numbering** --> **Number series of selected objects**.

    All columns belonging to the same numbering series as the selected column are numbered.

See also

Number a series of parts (page 722)

Example: Number parts in selected phases

This example shows how to number a model that consists of multiple phases, each phase having a different detailing and submittal schedule. This enables you to release drawings for a particular phase at any time.

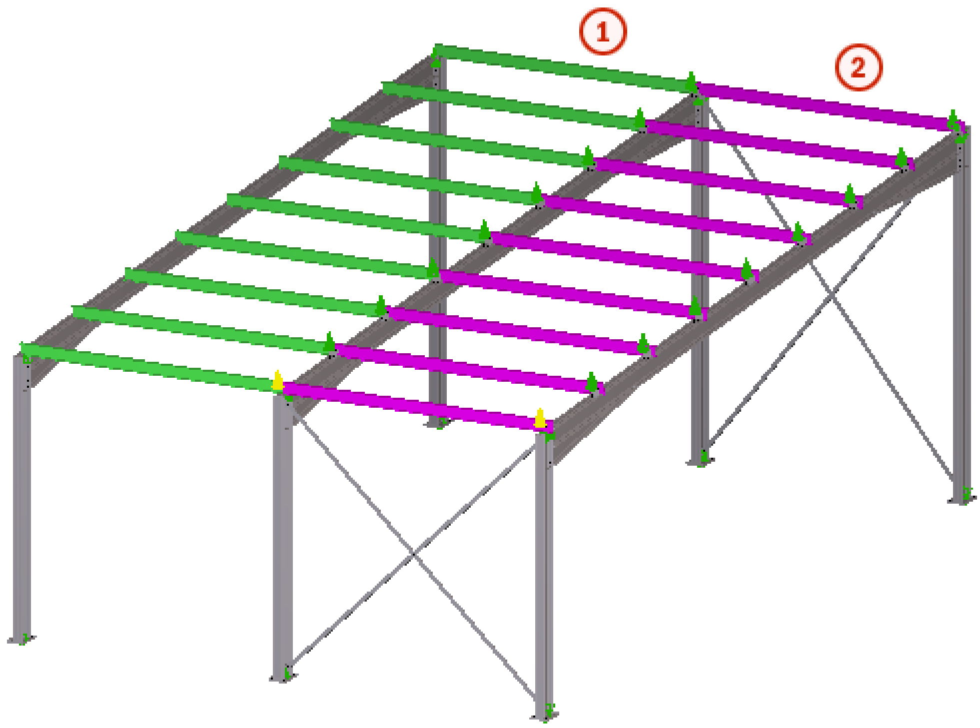
Before you start, divide the model into phases.

To number parts in selected phases:

1. Apply a specific numbering series prefix and start number for parts in each phase.

   For example:

   - The beams in phase 1 get the numbering series prefix B and start number 1000.
   - The beams in phase 2 get the numbering series prefix B and start number 2000.



(1) Phase 1: green

(2) Phase 2: magenta

2. Ensure that the numbering series are not overlapping.

   For example, to avoid numbering overlaps with the beams in phase 2, phase 1 should not contain more than 1000 position numbers.

3. Select the parts you want to number.

   TIP Use selection filters to easily select parts belonging to a certain phase or parts with a specific start number series. You can also use selection filters to ignore specific phases that are already completed or phases that are not ready for numbering.

4. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog box.

5. Modify the numbering settings, and then click **Apply**.

6. Select one of the parts you want to number.

7. On the **Drawings & reports** tab, click **Perform numbering** --> **Number series of selected objects**.

   All parts belonging to the same numbering series as the selected part are numbered.

See also

Number a series of parts (page 722)

Numbering settings during a project (page 746)

5.13 Tips for numbering

- It is a good idea to have some kind of a routine in numbering. For example, number the model when you start your working day or when you finish for the day.
- To save time, include the numbering series in the default part properties for each type of part before you start modeling.
- Numbering is not just another way of classifying parts. To classify, use **Organizer**, user-defined attributes, or colors.
- If you have overlapping position numbers, Tekla Structures will warn you about it.

  You can have a closer look at the overlapping position numbers in the numbering history log. To display the log, click **File menu** --> **Logs** --> **Numbering history log**.

See also

Numbering settings during a project (page 746)

Numbering examples (page 741)

Create a standard-part model (page 747)

Numbering settings during a project

You can use different numbering settings at different times in a project.

For example:

- Before releasing a phase of the project for fabrication, you might use the **Re-use old numbers** option for numbering the entire model.
- If a phase has already been released for fabrication in a project, you might use the **Take new number** option for new and modified parts.
- If you are numbering other phases of the project at earlier stages of detailing, you might use the **Compare to old** option and try to combine as many position numbers as possible.

See also

Example: Number parts in selected phases (page 744)

General numbering settings (page 1005)

Create a standard-part model

A standard-part model contains only standard parts with specific part prefixes. You can use these prefixes when numbering parts in another model. The prefixes that you define will be used as actual part position numbers in the other model.

The standard-part model is only used for comparing parts when you number parts in a project model. It cannot be used for creating parts in the project model.

NOTE This functionality only applies to steel parts. Assemblies are not affected.

---

1. Create a new model and give it a descriptive name.

   For example, `StandardParts`.

2. Create the objects you want to use as standard parts.
3. Explode all components.

   You can explode the components if you plan to delete the unnecessary parts such as duplicate angles and main parts.

4. Delete all unnecessary items.
5. Give the objects part prefixes that are not used elsewhere (for example, STD1, STD2, and so on).

   Ensure that the standard-part model does not contain duplicate part prefixes. You do not need to define the assembly prefix or the part or assembly start numbers.

6. Save the standard-part model.

   To use the standard-part model with Tekla Model Sharing, save the standard-part model in a separate folder under the current model folder.

   To use the standard-part model with a multi-user model, save the standard-part model so that all users have access to it.

7. Open a project model that you wish to number.
8. On the **File** menu, click **Settings** --> **Advanced options** --> **Numbering** .
9. Check that the advanced option XS_STD_PART_MODEL points to the correct standard-part model.

   For example:

   `XS_STD_PART_MODEL=C:\TeklaStructuresModels\StandardParts\`

10. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to display the **Numbering Setup** dialog box.

11. If you have selected the **Part name** check box, ensure that the project model has the same part names as the standard-part model.
12. Select the **Check for standard parts** check box.
13. Click **Apply** to save the changes.
14. On the **Drawings & reports** tab, click **Perform numbering** --> **Number modified objects** to number the project model.

    As the parts are being numbered, Tekla Structures compares all parts in the project model to the standard-part model. Any part prefixes found in the standard-part model are applied to all identical parts found in the project model. The numbering series of the oldest numbered standard part in the project model is applied to all identical parts found in the project model.

See also

Number parts (page 722)

6 Applications

All available applications, macros, and drawing plugins are located in the section **Applications** of the **Applications & components** catalog. You can also record macros of your own and show them on the list.

Macros

Macros (page 751) are saved as `.cs` files in the `\drawings` or `\modeling` folder under the folders that have been defined with the advanced option . By default, this advanced option is set to `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\macros`.

In addition to this global folder, you can define a local folder and store local macros there, for example, environment macros or firm macros. Specify the local macro folder for the advanced option `XS_MACRO_DIRECTORY` in addition to the global folder. Define the global folder first and then the local folder. When you create a macro you need to set it as global or local, and the macro will be placed in the global or local folder according to your selection. The macros in the global folder are read first.

Example of the definition for `XS_MACRO_DIRECTORY` containing a global folder and a local folder:

```
%XSDATADIR%environments\common\macros;%XSDATADIR%environments\uk\General
\user-macros
```

Macros in the modeling mode

Macro	Description
AutoConnectSelectedParts (page 778)	Use to automatically create connections without opening the **AutoConnection** dialog box.
AutomaticSplicingTool (page 564)	Use to split long reinforcing bars and bar groups that exceed the stock length, and create splices in split locations.
ContinuousBeamReinforcement	Use to reinforce a continuous beam. The macro creates main top and

Macro	Description
	bottom bars, stirrups, fittings, and additional top and bottom bars using system components.
CreateSurfaceView (page 33)	Use to create an automatically aligned surface view.
CreateSurfaceView_wEdge (page 33)	Use to create a surface view and align the work plane along the edge you select.
DesignGroupNumbering (page 738)	Use to number parts by design groups so that you can differentiate the parts from each other in drawings and reports.
DirectoryBrowser	Use to to find and modify the location of the various Tekla Structures files and folders, and customize user settings.
RebarClassificator (page 567)	Use to classify the reinforcing bars and reinforcement meshes by their order of depth in concrete slabs and panels.
RebarSeqNumbering (page 566)	Use to assign cast unit specific running numbers (1, 2, 3...) to the reinforcement in the model.
RebarSplitAndCoupler	Use to split a reinforcing bar group and add couplers in relation to the direction of the picked points.
UpdateRebarAttributes	Use to manage the user-defined attributes (UDAs) of the couplers and the end anchor parts created by **Rebar Coupler and Anchor Tools**.

Macros in the drawing mode

Macro	Description
Add surfacing symbols in drawings	Use to add surface treatment symbols in cast unit drawings.
Moment connection symbols	Use to create moment connection symbols to show the beams that are connected to columns with rigid connections.
Create dimensions marks for all rebars	Use to add dimension marks to all rebars at one go.
Exaggerate selected dimensions	Use to exaggerate narrow dimensions to make them easier to read.

Macro	Description
Rebar layering marker	Use to mark reinforcing bar layers with different marking styles and line types in a drawing.
Rebar mesh view creator	Use to create drawing views each containing one reinforcement mesh.
Remove change clouds	Use to remove dimension change symbols, mark change symbols, and associative note change symbols at one go in an open drawing.

Extensions (.tsep)

You can download Tekla Structures extensions that have the `.tsep` file extension from Tekla Warehouse and import (page 755) these extensions to the **Applications & components** catalog. When you restart Tekla Structures, the imported extensions are installed and added to the **Ungrouped items** group in the catalog. You can move them to a suitable group.

Publish groups in the Applications & components catalog

You can collect content to a group that you create in the **Applications & components** catalog. You can then publish the group (page 756) as a catalog definition file to make it available for other Tekla Structures users.

See also

Working with applications (page 751)

6.1 Working with applications

You can run, add, edit, rename, save as and delete applications, macros and plugins in the **Applications** section of **Applications and Components** catalog. You can also record and edit macros.

To	Do this
Record a macro	1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog. 2. Click the **Access advanced features** button and then click **Record macro > Global** or **Local** depending on whether you

To	Do this
	want to save the macro in the global or local macros folder. The **Local** command is only available if you have defined a location for the local macros using the advanced option `XS_MACRO_DIRECTORY`. 3. Enter a name for the macro in the **Macro name** box. 4. Click **OK** and perform the actions you want to record. 5. Click **Stop recoding** to stop recording. The recorded macro is saved under global or local macros in the `macros\drawings` or `macros\modeling` folder depending on the mode (drawing or modeling) you were using while you were recording the macro.
Create a macro file and add contents later	1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog. 2. Click the **Access advanced features** button  and then click **New macro > Global** or **Local** depending on whether you want to save the macro in the global or local macros folder. The **Local** command is only available if you have defined a location for the local macros using the advanced option `XS_MACRO_DIRECTORY`. 3. Enter a name for the macro in the **Macro name** box.

To	Do this
	4. Click **OK**. This creates an empty macro file that is displayed in the **Applications** list. 5. Right-click the empty macro file and select **Edit**. 6. Add the macro content, for example, by copying commands from other macro files and save the file.
View or edit a macro	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. Right-click the macro you want to edit and click **Edit**. The macro can be opened in any text editor. 4. If needed, edit the macro and save the macro file.
Run an application	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 the application you want to run.
Save an application with another name	1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.

To	Do this
	2. Click the arrow next to **Applications** to open the applications list. 3. Right-click the application that you want to save with another name and click **Save as**. 4. Enter a new name for the application and click **OK**. The application will be added in the list.
Rename an application	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. Right-click the application you want to rename and click **Rename**. 4. Enter a new name for the application and click **OK**. The name of the application changes.
Delete an application	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. Right-click the application you want to delete and click **Delete**. The application is removed from the list.

See also

Applications (page 749)

6.2 Import a .tsep extension to the Applications & components catalog

You can import Tekla Structures .tsep extensions (Tekla Structures extension package) to the **Applications & components** catalog.

Before you begin, download the .tsep extension from Tekla Warehouse. For more information, see Download content from Tekla Warehouse.

NOTE Some Tekla Structures extensions have an .msi installation file. You must install these extensions separately. Download the .msi installation file from Tekla Warehouse, then double-click the file to run the installation.

System administrators can copy multiple .tsep extension files to the \Tekla Structures\<version>\Extensions\To be installed folder on a Tekla Structures user's computer. The extensions are installed the next time the user restarts Tekla Structures.

1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
2. Click **Manage extensions** --> **Extension manager**.
3. In the **Extension manager**, click **Import**.
4. Browse to the location where you saved the .tsep extension, then double-click the .tsep extension.

   Tekla Structures opens a dialog box that lists the installed Tekla Structures versions that are compatible with the extension. If none of the installed Tekla Structures versions are compatible, the compatible versions are listed.
5. Select the Tekla Structures versions to which you want to import the extension.
6. Click **Import**.

   The extension is shown in the **Extension manager** in all the Tekla Structures versions that you selected.
7. Restart Tekla Structures to install the imported extension.

The extension is shown in the **Ungrouped items** group in the **Applications & components** catalog. You can move the extension to a more suitable group, or create a new group.

TIP In **Extension manager**, you can list the extensions by type: extension or environment. You can also search for content based on name, author, description, and type of extension.

See also

How to use the Applications & components catalog (page 766)

6.3 Remove .tsep extensions from the Applications & components catalog

You can uninstall installed extensions in **Extension manager**

Installing and uninstalling extensions creates a log file to `\Tekla Structures \<version>\Extensions\TSEP Logs`.

1. Select one or several extensions.

   Use **Ctrl** or **Shift** to select several extensions.

2. Click **Remove**.
3. Restart Tekla Structures to remove the extensions.

6.4 Copy .tsep extensions to a new Tekla Structures version

When starting to use a new Tekla Structures version, you can use the Migration Wizard tool to copy the installed `.tsep` extensions to the new version.

1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
2. Click **Manage extensions** --> **Migrate extensions**.
3. Restart Tekla Structures to install the copied extensions.

6.5 Publish a group in the Applications & components catalog

You can collect content, such as macros, extensions, and system and custom components to a group that you create in the **Applications & components** catalog. You can then publish the group as a catalog definition file to make it available for other Tekla Structures users. For the published content to work correctly in another Tekla Structures installation, the content must also exist in that installation.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Create a new group:
    a. Right-click in the catalog and select **New group**.
    b. Enter a name for the group.
    c. Select the group and click the small arrow on the right to add a description to the group.
    d. Add content to the group.

       Some content in the **Applications & components** catalog may be hidden. To publish the hidden content, select the **Show hidden items** check box at the bottom of the catalog.

       Note that the model-specific items you add to the group are visible in the modeling mode and the drawing-specific items in the drawing mode.
    e. Add the needed information to the items in the group: description, tags, and additional thumbnail images.

       Use a thumbnail image from the \Tekla Structures\<version>\Bitmaps folder to ensure that the image is available for other Tekla Structures users.
3. Right-click the group and select **Publish group** to create a catalog definition file.

   The file contains the following information:
    - Name and description of the published group
    - Names and descriptions of the subgroups
    - References to the items that you have added to the group

      The file does not contain the actual items. When other users use the group, they must ensure that the referenced items exist in their Tekla Structures installation and model.
    - Descriptions, tags and thumbnail references of the items in the group

      The file does not contain the actual thumbnail image files.
4. Add a unique prefix to the filename in the **Publish group** dialog box.

   The filename format must be <prefix>_ComponentCatalog.ac.xml.
5. Click **Save**.

   The file is by default saved to the model folder.

6. Make the group available for other Tekla Structures users by moving the <prefix>_ComponentCatalog.ac.xml catalog definition file to an appropriate folder:
   - Project, firm or system folder defined in XS_PROJECT, XS_FIRM, or XS_SYSTEM.
   - \attributes folder under the current model folder
   - Extensions folder in \Tekla Structures\<version>\environments\common\extensions, or to any of the folders defined in XS_EXTENSION_DIRECTORY.

     The **Applications & components** catalog also searches the subfolders of these folders. We recommend that you use the extension folders if you have created your own extensions and have included them in the group.
7. Check that the catalog definition file works correctly:
   a. Delete the published group from your **Applications & components** catalog.
   b. Click > **Catalog management** > **Reload catalog** to load and view the published group.

When you have checked the group, other users can start using it:

- If the group content is already included in other users' Tekla Structures installation, they can use the group immediately after reloading the catalog by clicking > **Catalog management** > **Reload catalog**.
- If the group content, for example extensions, is not included in other users' Tekla Structures installation, they have to download the missing extensions from Tekla Warehouse first, and then re-open the model where they are going to use the group.

7 Components

Components are tools that you can use to connect parts in the model. Components automate tasks and group objects so that Tekla Structures treats them as a single unit. You can save the properties of a component and use them in other projects.

Components adapt to changes in the model, which means that Tekla Structures automatically modifies a component if you modify the parts it connects. When you copy or move objects, Tekla Structures automatically copies or moves all the associated components with the objects.

All components are stored in the Applications & components catalog. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.

System components

Tekla Structures contains a wide range of predefined system components by default. There are three types of system components:

- **Connection** components connect two or more parts, and create all the required objects such as cuts, fittings, parts, bolts, and welds.

  For example, end plates, clip angles, and bolted gussets are connections.

  In the **Applications & components** catalog, the connection symbol is .

- **Detail** components add a detail or a reinforcement to the main part. A detail is only connected to one part.

  For example, stiffeners, base plates, and lifting hooks are steel details, and beam reinforcement and pad footing reinforcement are concrete details.

  In the **Applications & components** catalog, the detail symbol is .

- **Detailing** components automatically create and assemble the parts to build a structure, but do not connect the structure to existing parts.

  For example, stairs, frames, and towers are detailing components.

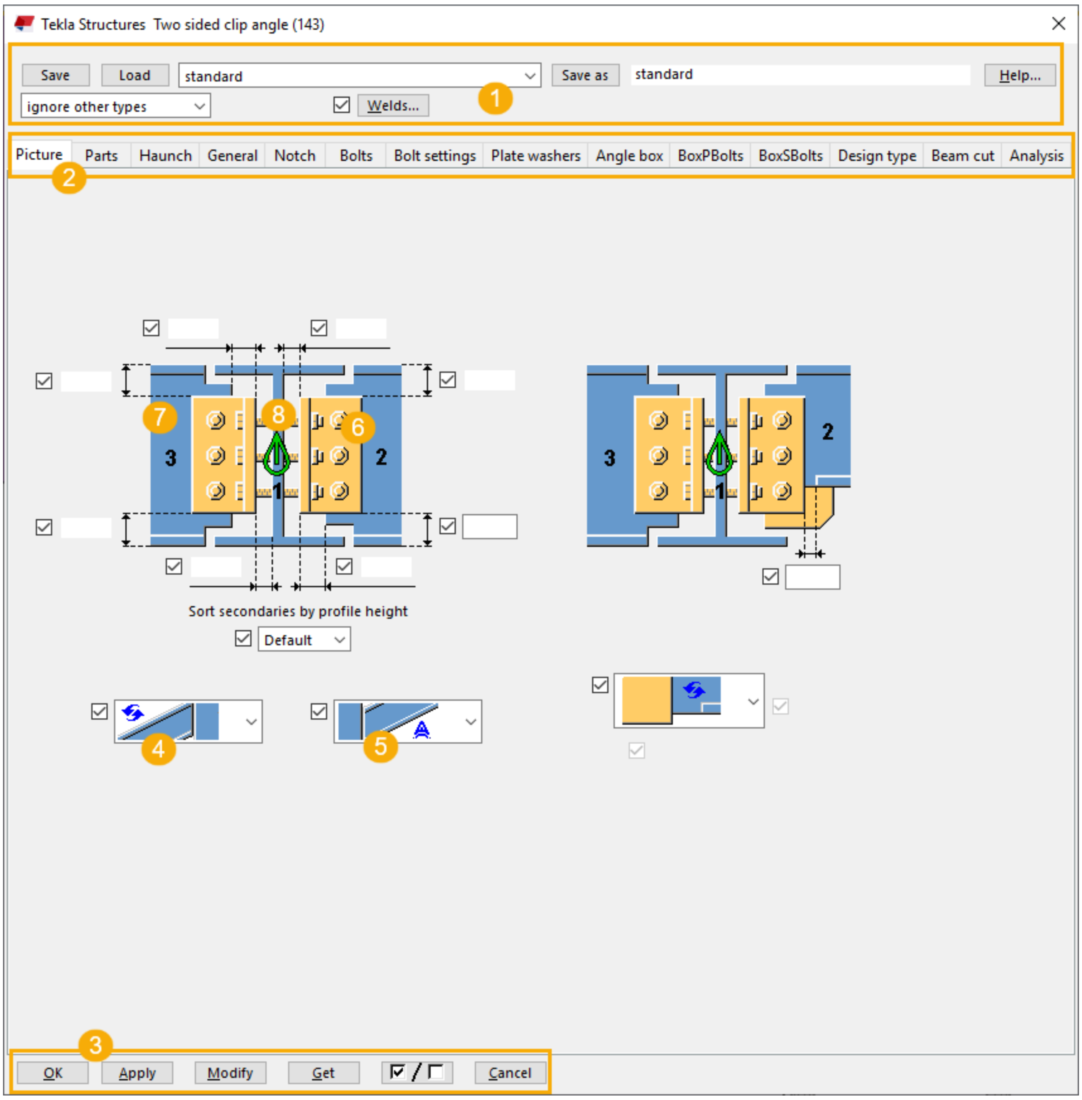
In the **Applications & components** catalog, the detailing symbol is .

You can also create your own components, custom components (page 823), if you do not find a system component that suits your needs. You can use custom components in the same way as system components.

7.1 Component properties

Each component has a dialog box where you can define the properties of the component. You can open the dialog box by double-clicking the component in the **Applications & components** catalog.

The image below shows a typical example of a steel connection, the **Two sided clip angle (143)** connection. Concrete and reinforcement component dialog boxes may have different options.



	Description
1	In the upper part of the dialog box, you can save and load predefined settings. Some components have buttons for accessing bolts, welds and DSTV properties. When modifying connections and details, you can select whether Tekla Structures ignores other types of connections and details, or modifies all selected connections and details irrespective of their type. With the modify option, the type of the selected components is changed to match the type of the component you are currently modifying. For more information, see and Model folder files and file name extensions.
2	On the tabs, you can define the properties of the parts and bolts that the component creates. You can enter values manually, use system default values, AutoDefaults values, automatic values, or for some steel connections, the values in the `joints.def` file. Manually entered values, AutoDefaults, automatic values and the properties defined in the `joints.def` file all override the system defaults. System default values are used if you do not manually enter a value or select any other type of property value. You cannot change the system default values. For more information on `joints.def`, see Define connection properties in the joints.def file (page 794).
3	For information on the dialog box buttons, see .
4	If you select an AutoDefaults option, Tekla Structures uses the property defined in the AutoDefaults rules. The image in the AutoDefaults option is an example and does not necessarily match the outcome in the model. For more information on AutoDefaults, see AutoDefaults (page 780).
5	If you select an automatic option, Tekla Structures automatically determines which option to use for a property. For example, when you use the automatic option for the stiffener in **End plate (144)**, the connection automatically adds the stiffener to a beam-to-column connection, but does not add it to a beam-to-beam connection. For more information on AutoConnection, see AutoConnection (page 775).
6	The parts that are yellow in the component dialog box are created by the component.
7	The parts that are blue in the component dialog box should already exist in the model before you create the component.

	Description
8	Up direction indicates how the connection is rotated around the secondary part, relative to the current work plane. The symbol on the **Picture** tab of the component dialog box indicates the correct up direction. If there are no secondary parts, Tekla Structures rotates the connection around the main part. The options are: `+x, -x, +y, -y, +z, -z`. You can change the default up direction on the **General** tab of the component dialog box. Try changing the positive directions first.

7.2 Add a component to a model

When you add a component to a model, you either attach the component to existing parts in the model, or pick positions to indicate the location or length of the component.

Connections and details have a main part that you select first. Connections also have one or more secondary parts that you select after you have selected the main part. Detailing components do not always have a main part and secondary parts. Instead, they automatically create and assemble the parts to build a structure when you pick a position in the model.

If you use a component you are unfamiliar with, use the default properties of the component. Then check what needs to be modified, and modify only few properties at a time to see how the modifications affect the component. This is quicker than trying to set all the component properties before seeing what the component actually creates.

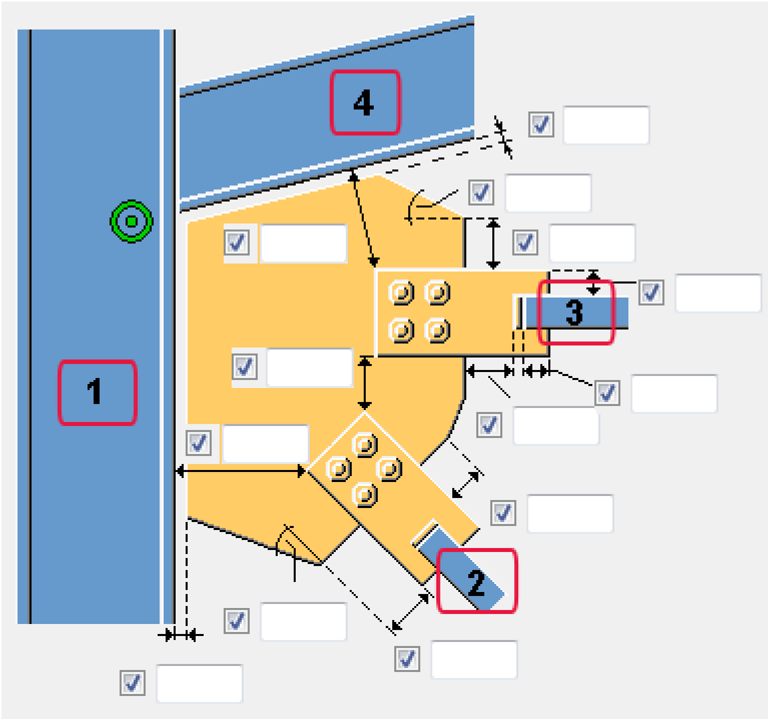
Tekla Structures opens a command prompt when you add a component. Do not close the prompt window, because it displays information on adding the component. This information can be useful in problem situations.

1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.

   You can also press **Ctrl + F**.

2. Select a component and do one of the following:

To	Do this
Add a connection	1. Select the main part. 2. Select the secondary part or parts. • If there is one secondary part, the connection is automatically created when you select the secondary part.

To	Do this
	• If there are several secondary parts, click the middle mouse button to finish selecting the parts and to create the connection. In the example image below, the numbers from 1 to 4 show the selection order of parts. The blue parts should already exist in the model before you create the component. 
Add a detail	1. Select the main part. 2. Pick a position in the main part to determine the location of the detail.
Add a detailing component	Pick one to three positions to determine the location of the objects that the detailing component creates.

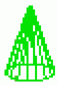
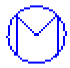
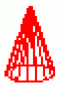
When you have added components to the model, you can use the property pane to list the components:

- If you select one component in the model, the property pane shows the name and number of that component. You can open the component properties dialog box by clicking the **Component properties** button in the property pane.
- If you select several different components in the model, the property pane shows lists that have the text **Varies**. Open the lists to show the names and numbers of the selected components.

- If you select components and other model objects, click the **Object type list** button  in the property pane to open a list of the selected object types, and select **Component** to list the components.

Component status

When you have added the component, Tekla Structures shows the component status using the symbols shown in the table below. Double-click the symbol to open the component properties.

Color	Status
	The green symbol shows that the component was created successfully. The detailing component symbol in the model is  .
	The yellow symbol shows that the component was created, but has problems. This often occurs when bolts or holes have an edge distance that is less than the default value.
	The red symbol shows that the component was not created. Common reasons are that the properties are incorrect or that the up direction is not appropriate.

7.3 Modify a component in a model

You can modify the properties of a component after you have added the component in the model, for example, if you need to change the number of bolts or plate dimensions.

1. Double-click the component symbol in the model to open the component dialog box.
2. Modify the properties.
3. If needed, define which connections and details are modified:
   - Ignore other types: Tekla Structures only modifies the connections and details that are of the same type as the connection or detail you are modifying.
   - Modify connection type: If you have selected several connections and details, click **Modify** to modify all the selected connections and details irrespective of their type. If the connection type of a selected connection is not the same as in the connection dialog box, Tekla Structures changes the connection type.
4. Click **Modify**.

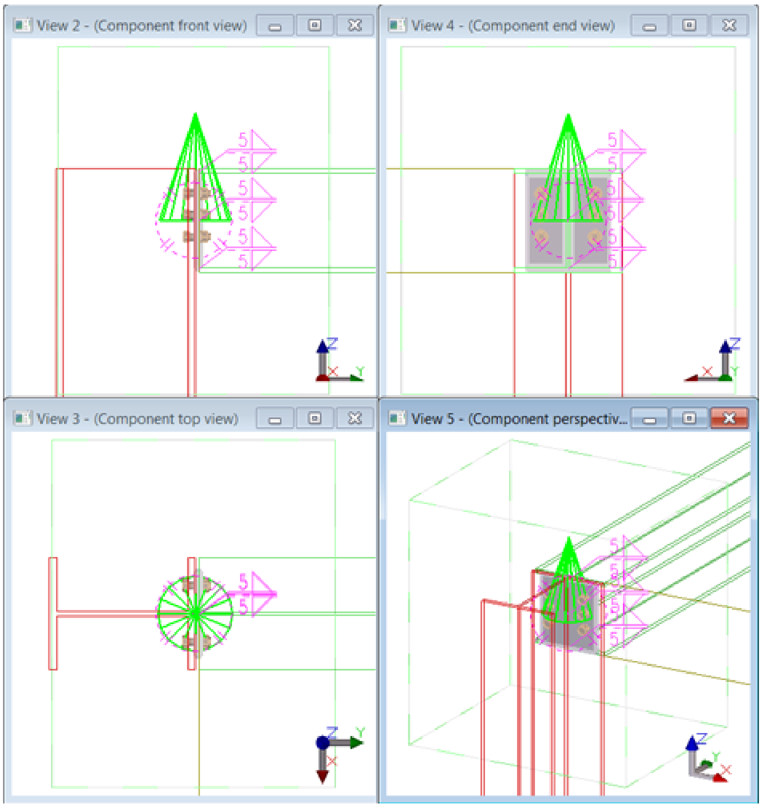
7.4 View a component in a model

You can create several views of a component to view it from different viewpoints.

1. Click the component symbol in the model to select the component.
2. Right-click and select **Create view** --> **Default views of component** .

   Tekla Structures creates four views: front, end, top, and perspective.

   The example image shows the default views of the **End plate (144)** connection.



NOTE You can check the dimensions, such as bolt locations and edge distances, using the **Measure** tool in the **Component front view**.

7.5 Component tips

Default properties

If you use a component you are unfamiliar with, use the default properties of the component. Then check what needs to be modified, and modify only few properties at a time to see how the modifications affect the component. This is quicker than trying to set all the component properties before seeing what the component actually creates.

Valid profiles

Some components work with certain profiles only. If a component is not created successfully, try entering a valid profile.

Select components switch

You can switch on the **Select components** switch to be able to select any object that belongs to the component.

Component is not added to the model

If the component is not added to the model, check the status bar. For example, you may need to click the middle mouse button to stop selecting parts before Tekla Structures creates the component.

Using thickness to create needed parts

If a component does not by default create the parts that you need, look for options to create them. If there are no options, try entering a thickness value for the parts.

If a component creates parts that you do not need, look for options to remove them. If there are no options, enter a zero (0) as the thickness of the parts.

Many secondary parts are found

If you are using a connection that only allows one secondary part, you may see the message `Many parts found` on the status bar. This means that Tekla Structures cannot determine which parts to connect. You may have several parts in the same location, or the view may be set too deep.

7.6 How to use the Applications & components catalog

Components are stored in the **Applications & components** catalog and organized into two different types of groups: default groups are automatically available and predefined groups depend on your environment.

Click the **Applications & components** button in the side pane to open the **Applications & components** catalog. You can also press **Ctrl + F**.

To use a component (page 759), select it in the catalog and follow the instructions on the status bar to add the component to the model. Double-click the component in the catalog to open the component properties dialog box.

Groups in the catalog

Default groups and predefined groups are shown against different background color in the catalog.

Default groups are automatically available:

- **Recent** contains the 12 components and applications that you have most recently used in the model.
- **Ungrouped items** contains the components and applications that are not in any predefined group.

  Ungrouped items can be, for example, imported components that have not been moved to any other group yet.
- **Applications** contains applications (page 749), macros and drawing plugins.

  If you create your own macros, you can add them to this group.
- **Connections** contains connections and seams.
- **Detailing** contains detailing components.
- **Details** contains details.
- **Parts** contains custom parts.
- **Legacy catalog** shows the folder structure of the **Component Catalog** used in previous Tekla Structures versions if the catalog definition files are found in the standard folder search paths.

Depending on your environment, the catalog may also contain **predefined groups** for specific usage, such as **Steel** --> **Beam to beam connections**. You can create your own groups according to your needs, for example, for your own favorite connections. This way you can find these connections quickly and easily. You can also hide the groups that you are not using so that only the groups you use are visible in the catalog.

Model-specific components are shown in the modeling mode and drawing-specific components are shown in the drawing mode.

Search for a component in the catalog

To search for a component in the catalog, enter the search term in the search box. The search is case-insensitive.

Note that the search does not find catalog content that has been hidden. Select the **Show hidden items** check box to show the hidden content.

The search uses the following rules:

- Non-numeric search terms find partial matches, for example, `bolt` shows both `bolt` and `bolted` in the search result.

  If you have more than one word in the search term, for example `bolt plate`, the words are automatically combined so that the search result shows the components that contain both bolt and plate in their name, description, or tags.
- Numeric (integer) search terms find the exact match, for example, `121` shows component number **121** in the search result.

  You can use the `*`, `?` and `[ ]` wildcards to search for partial numeric matches. For example, `10*` finds components number **10**, **110**, **104**, **1040**, and so on.
- You can restrict the search to specific tags, groups and types of components by using the keywords `tag`, `group`, or `type`. For example, `10 tag:advanced` finds the number **10** components that have `advanced` in the tag, and `type:custom` finds all custom components.

Change the view in the catalog

- Click  to show the thumbnail view.
- Click  to show the list view.
- Click  to show the compact view.

  The compact view shows the thumbnail images of the group that you select from the list above the search box. You can use the compact view to have more space on the screen.
- Click  to show the normal view.

Show selected components in the catalog

Click **Show selected** to show a **Selected components** group that contains the components that are selected in the model or in the drawing.

Click **Show selected** again to hide the **Selected components** group.

The **Show selected** button is not available when you are using the search in the catalog.

TIP You can use the property pane to list the components that you have selected in the model. If you select one component, the property pane shows the name and number of the component. If you select several different components, the property pane shows lists that have the text **Varies**. Open the lists to show the names and numbers of the selected components.

If you select components and other model objects, click the **Object type list** button in the property pane to open a list of the selected object types, and select **Component** to list the components.

View and modify component information in the catalog

Each component has an information box that shows the type of the component and the groups the component belongs to. You can add a description for the component and tags that can be used in the search.

1. Select a component in the catalog and click the small arrow on the right to open the component information box.
2. Type a description in the **Description** box.
3. Click + to add a tag and enter a tag in the box.
4. If needed, click + again to add more tags. You can also remove tags.
5. Click outside the information box to close it.

The descriptions and tags that you add are by default saved in the `ComponentCatalog.xml` file in the model folder.

Add a thumbnail image for a component in the catalog

Components have a default thumbnail image that shows a typical situation where the component can be used. You can add several thumbnails for a component and select which thumbnail is shown in the thumbnail view in the **Applications & components** catalog.

1. Select a component in the catalog.
2. Right-click and select**Thumbnails**.
3. Click **Add thumbnail**.
4. Select an image and click **Open**. You can use any standard image format, for example, `.png`, `.jpeg`, `.gif`, `.tiff`, and `.bmp`.
5. Select the check boxes of the thumbnails that you want to show in the component information box. You can also remove thumbnails, except for the default thumbnail.

6. Click **Close**.

The thumbnail information that you add is by default saved in the `ComponentCatalog.xml` file in the model folder.

Publish a component in the catalog

You may need to use the same component with different settings in different situations. To easily use the component, you can define the settings for each situation and publish the component in the catalog.

For example, you may need **End plate (144)** in three different situations. Add **End plate (144)** once to each situation in the model. Define the settings needed and then publish each **End plate (144)** in the catalog. You will then have **End plate (144)** saved in the catalog as three separate components, each with different settings. You can use these components from the catalog in the same way as other components.

1. Add a component once to all needed situations in the model.
2. Define the desired settings for each situation.
3. Select one of the components you added in the model, right-click and select **Publish in Catalog...**.
4. Enter a descriptive name for the component and click **OK**.
5. Repeat steps 3-4 for each component you added.

The components are first placed in the **Ungrouped items** group in the catalog. They have the name you entered and the thumbnail image of the original component.

You can move the components to a more appropriate group in the catalog and change the thumbnail image. For example, you can create a basic component view in each situation, and use an image of the view as the thumbnail.

Create and modify groups in the catalog

You can create groups and subgroups, and move groups to different locations in the predefined groups section in the catalog. You can add and remove components from the groups, rename the groups, and add descriptions for the groups.

To	Do this
Create a group	Right-click in the catalog and select **New group...**. Drag the group to the desired location.
Create a subgroup	Right-click a group in the catalog and select **New group...**.

To	Do this
Name a group	Right-click a group, select **Rename...** and type the name.
Add components to the group	• Select components in the catalog and drag them to another group. • Select components in the catalog, right-click and select **Add to group**. Then select the group to which the components are added. • Right-click a group, select **Add all to group** and select the group to which all the components in the group are added. Note that the components are copied, not moved, to the other groups.
Remove a group Remove a component from a group	Right-click a group or a component in a group, and select **Remove from group**.

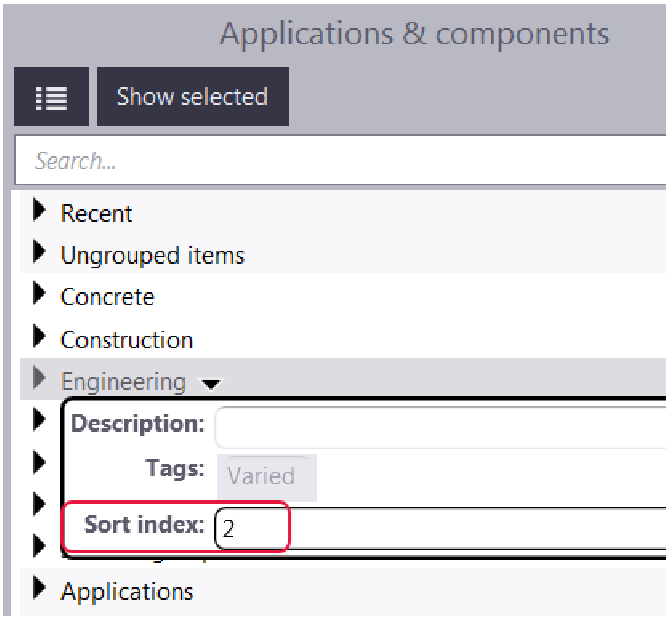
The groups that you create are by default saved in the `ComponentCatalog.xml` file in the model folder.

NOTE You cannot add or remove groups in the default groups, and you cannot modify the content of the default groups. However, you can hide the default groups and the individual items in the groups.

Change the order of groups in the catalog

You can change the order of the predefined groups in the **Applications & components** catalog. Predefined groups depend on your environment, for example, **Steel** and **Concrete** can be such groups. Note that you cannot change the order of the default groups, for example, **Applications**, **Connections**, and **Detailing**.

You can control the order with a sort index. The **Sort index** option is available in the group information of each predefined group in the **Applications & components** catalog. Sort indexes are saved in the catalog definition files.



You can change the sort index by entering either a negative or a positive integer number, or `0`, in the **Sort index** option box. A negative sort index moves a group towards the top and a positive sort index moves a group towards the bottom in the predefined groups section. Enter `0` or clear the value to revert to the default order. By default, the groups are in alphabetical order.

The sort index changes you make are model specific and they are saved in the `ComponentCatalog.xml` file in the `\model` folder. Administrators can define the order of groups for an environment or a project using the catalog definition files in the environment, firm and project folders. Do not edit these files if you are not an administrator.

Note that even if administrators have defined the order, you can still make model-specific changes to the order of the groups by entering a different sort index value for a group. If you need to revert to the default order, enter `0` as the sort index.

To change the order:

1. Select a predefined group.
2. Click the small arrow on the right to open the group information box.
3. Enter a number in the **Sort index** box.

   The group is immediately moved.
4. Save the model to keep the order.

Hide groups and components in the catalog

1. Select a group or a component in the catalog.
2. Right-click and select **Hide / Unhide** to hide the group or the component.
3. To view the hidden group or component again, select the **Show hidden items** check box at the bottom of the catalog. The hidden group or component is shown as dimmed.
4. To show the hidden group or component normally, right-click it and select **Hide / Unhide**.

Show the catalog message log

If there are errors or warnings, for example, in the catalog definition files, the **Message Log** button is shown in the lower right corner in the catalog. The button is not shown if there are no errors or warnings.

To view the error log, click the **Message Log** button.

Errors and warnings are also written to the `ComponentCatalog_<user>.log` file in the `\logs` folder under the model folder.

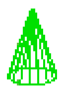
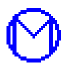
Catalog definitions

The commands in **Access advanced features**  > **Catalog management** are used for modifying catalog definitions. Generally, there is no need to modify catalog definitions. Do not modify the definition files if you are not an administrator. For more information on administrator tasks, see Customize the Applications & components catalog.

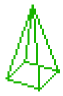
7.7 Convert components to conceptual or detailing components

Depending on the Tekla Structures configuration you are using, you can create either detailing or conceptual components.

- Detailing components include all the information needed for fabrication, such as assemblies, cast units, and reinforcing bars.

  Detailing components have a round symbol in the model:  or .
- Conceptual components look similar to detailing components but do not include the option to change part numbering or assembly numbering

settings. Conceptual components are meant to be used as reference information for further fabrication detailing.

Conceptual components have a rectangular symbol in the model:  or .

You can create conceptual components in the **Tekla Structures Graphite** configuration, and in the **Engineering**, **Rebar Detailing**, and **Construction Modeling** configurations.

You can edit conceptual components and convert them to detailing components in the **Tekla Structures Diamond** configuration, and in the **Full**, **Primary**, **Steel Detailing**, and **Precast Concrete Detailing** configurations.

Modifying part properties, such as the size of the component main part, does not automatically convert a detailing component to a conceptual component, or vice versa. For example, if you use the **Engineering** configuration and modify the model, detailing components are not converted to conceptual components. However, detailing components will adapt to the changes done in the model, but you cannot modify any properties using the component dialog box. Detailing components will also remain as detailing components unless you convert them one by one.

You can convert components in the **Applications & components** catalog. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.

Do one of the following:

To	Do this	Configuration
Convert a conceptual component to a detailing component	1. Click > **Convert to detailing component**. 2. Select the component symbol.	**Tekla Structures Diamond** **Full**, **Primary**, **Steel Detailing**, **Precast Concrete Detailing**
Convert a detailing component to a conceptual component	1. Click >**Convert to conceptual component**. 2. Select the component symbol.	**Tekla Structures Graphite** **Engineering**, **Construction Modeling**, **Rebar Detailing**

7.8 Automate connection creation

This section describes the tools that you can use to automate connection creation in the model.

Click the links below to find out more:

AutoConnection (page 775)

AutoDefaults (page 780)

AutoConnection and AutoDefaults rules (page 785)

AutoConnection

Use the AutoConnection tool to automatically select and apply connections with predefined properties to selected parts in a model. With AutoConnection, Tekla Structures automatically creates similar connections for similar framing conditions.

You can use AutoConnection to quickly add connections individually, in phases, or project-wide. This is useful when you are working on a large project using many connections, modifying a model, or importing modified profiles.

NOTE Before using AutoConnection in a working model, we recommend that you create a test model, and create all the connection conditions in it that you need for a particular project. You can then use the test model to check the rules and properties of various connection types. The model also acts as a quick reference for connection information.

See also

Define AutoConnection settings and rules (page 775)

Create a connection using AutoConnection (page 778)

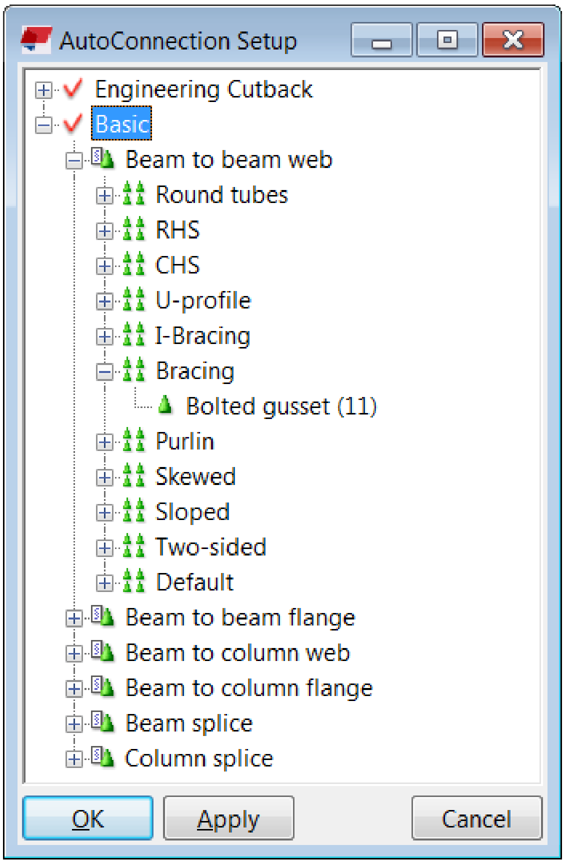
AutoConnection and AutoDefaults rules (page 785)

Define AutoConnection settings and rules

With AutoConnection you can define groups of rules which Tekla Structures automatically applies when creating connections in a model. By using a rule group to select connections and connection properties you do not have to select each connection and define its properties separately. For example, you can create separate rules for different standards, projects, manufacturers, and even individual models.

AutoConnection settings

To open the **AutoConnection Setup** dialog box, on the **File** menu, click **Catalogs --> AutoConnection settings**.



Icon	Setup level	Description
	Rule group	You can use rule groups to organize connections and connection properties according to different standards, projects, manufacturers, and models. You can create, modify and delete rule groups.
	Framing condition	Framing conditions are predefined connection types that you cannot change. Tekla Structures creates the framing conditions automatically: • Beam to beam web • Beam to beam flange • Beam to column web • Beam to column flange • Beam splice

Icon	Setup level	Description
		• Column splice
	Rule set	You can use rule sets to define which connection to use in a certain situation. You can create additional rule sets.
	Connection	The connection to apply if the rule set criteria are met. To apply a particular connection, the conditions in the model have to match all the rules in the branch that contains the connection.

Rules.zxt file

When you use AutoConnection, Tekla Structures saves the AutoConnection information in a zipped `rules.zxt` file in the `\attributes` folder under the current model folder.

You can copy the `rules.zxt` file to the project or firm folder to make it available in other models. Each time you modify the AutoConnection setup you need to recopy this file to the firm and project folders. To use the modified setup in other models, restart Tekla Structures.

Limitation

You can have a maximum of two secondary parts in the connection (for example, you cannot use complex gussets with several secondary parts). AutoConnection uses the profile height and ID number as the criteria for determining the first secondary and second secondary parts.

Create a rule group for AutoConnection

You can define rule groups for AutoConnection to organize connections and connection properties according to different standards, projects, manufacturers, and models.

1. On the **File** menu, click **Catalogs** --> **AutoConnection settings**.
2. Right-click an existing rule group and select **New Rule Group**.
3. Click the **New** group and enter a name.

   Give the rule group a name that reflects the group of connections that you want to create (page 778). For example, use the fabricator's name, the project name, or any name that clearly identifies the connection rules that you want to use for a specific model.

When you create a new rule group, Tekla Structures automatically adds the existing framing conditions in the group.

Create a rule set for AutoConnection

You can create AutoConnection rule sets under framing conditions to specify which connection properties to use when specific conditions in the model are met.

You only need to create AutoConnection rule sets if you plan to use different connections (page 778) to connect similar framing conditions. For example, in the model, some beam-to-beam connections require clip angles, others need shear tabs. You need to define rule sets to determine where each connection type should be used.

1. On the **File** menu, click **Catalogs** --> **AutoConnection settings**.
2. Click the plus icon in front of the rule group to open the tree structure.
3. Right-click the relevant framing condition and select **Create Additional Rule Sets**.
4. Right-click the new rule set and select **Edit Rule Set...**.
5. Enter a name for the rule set.
6. Select a rule from the **Available rules** list.
7. Click the right arrow button to move the selected rule into the list of **Rules in rule set**.
8. Enter the values used in the rule: either an exact value, or minimum and maximum values.
9. Click **OK**.

NOTE The order of the rules in the tree structure is important. Tekla Structures uses the first rule that matches the conditions in the model so you should place the most limiting rule highest in the tree, and the most generic rule lowest.

You can change the priority of a rule set by right-clicking the rule set and selecting **Move up** or **Move down**.

Change a connection in an AutoConnection rule set

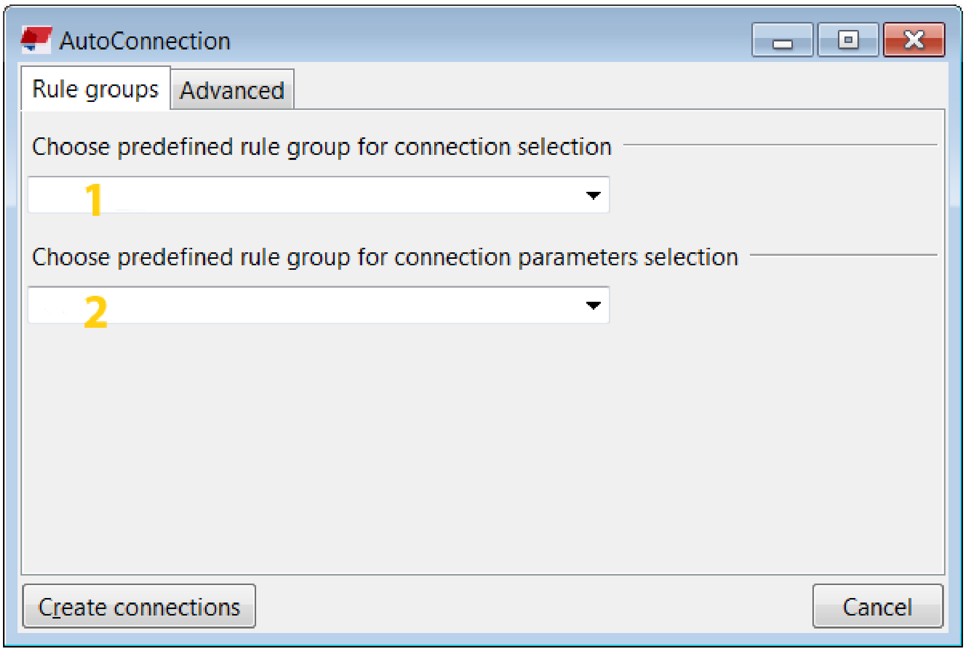
You can change the connection in a rule set by selecting a connection in the **Applications & components** catalog.

1. On the **File** menu, click **Catalogs** --> **AutoConnection settings**.
2. Click the plus icon in front of the relevant framing condition and rule set to find the connection that you want to change.
3. Right-click the connection and select **Select Connection Type...**.
4. Double-click a connection in the **Select component** dialog box.
5. Click **OK** in the **AutoConnection Setup** dialog box.

Create a connection using AutoConnection

Use AutoConnection to have Tekla Structures automatically create connections using the properties of predefined rules. When you use AutoConnection, Tekla Structures ignores the properties in the connection dialog boxes. Tekla Structures does not modify the existing connections.

1. In the model, select the parts to connect.
2. On the **Edit** tab, click **Components** --> **Create AutoConnections**.
3. Select the rule groups from the lists on the **Rule groups** tab.



1	Rule group for AutoConnection
2	Rule group for AutoDefaults

4. If needed, go to the **Advanced** tab to change the rules used in the framing conditions:
   a. Select the connection in the **Connection selection** option:
      - **AutoConnection** applies the connection defined in the rule group that you have selected in the first list on the **Rule groups** tab.
      - **None** does not create a connection.
      - Click **Select...** to select a connection from the **Applications & components** catalog. Tekla Structures creates the connection using the default properties.

   b. Select the connection properties in the **Parameters selection** option:
      - **Autodefaults** applies the properties of the rule group that you have selected in the first list on the **Rule groups** tab.

- **No autodefaults** applies the default connection properties.

5. Click **Create connections**.

TIP You can also use the **Auto connect selected parts** macro to automatically create connections using the current properties without opening the **AutoConnection** dialog box.

Macros are located in the **Applications** group in the **Applications & components** catalog.

See also

Define AutoConnection settings and rules (page 775)

AutoDefaults

Use AutoDefaults to set up properties for existing connections. AutoDefaults allows you to modify the default connection properties and save them for use in specific circumstances. When you use AutoDefaults, Tekla Structures automatically creates the connections with the predefined AutoDefaults properties. You can also use AutoDefaults for a single connection.

For example, you can use AutoDefaults to automatically adjust the thickness of each base plate you create, according to the main part profile. If the main part profile changes, Tekla Structures automatically adjusts the thickness of the base plate.

NOTE Before using AutoDefaults in a working model, we recommend that you create a test model, and create all the connection conditions in it that you need for a particular project. You can then use this test model to check the rules and properties of various connection types. It also acts as a quick reference for connection information.

See also

Define AutoDefaults settings and rules (page 780)

Modify a connection using AutoDefaults (page 785)

AutoConnection and AutoDefaults rules (page 785)

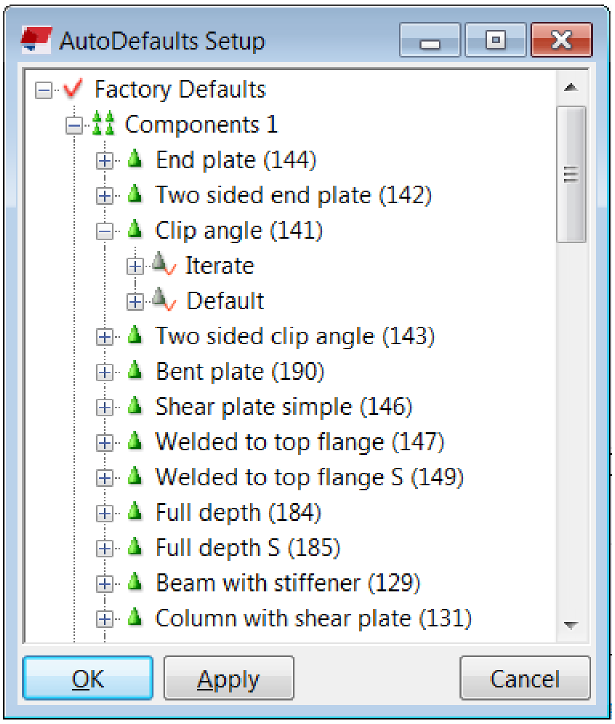
Define AutoDefaults settings and rules

Use AutoDefaults to set up properties for existing connections. AutoDefaults chooses connection properties based on the framing condition. With AutoDefaults you can create rules that define the situations where the predefined properties are used.

To apply the configured AutoDefaults settings and rules to connections, see Modify a connection using AutoDefaults (page 785).

AutoDefaults settings

To open the **AutoDefaults setup** dialog box, click **File** --> **Catalogs** --> **AutoDefaults settings**.



Icon	Setup level	Description
	Rule group	You can use rule groups to organize settings according to different standards, projects, manufacturers, and models. You can create, modify and delete rule groups.
 	Components	The component tree structure shows the connections that are available on component toolbars in Tekla Structures.
	Rule set	Rule sets control which properties to use in certain situations. You can create additional rule sets. Tekla Structures processes AutoDefaults rule sets in the order in which they are in the tree, so you can control the selection of properties.
	Properties file	The properties files are under the rule sets. By default, each connection has a standard properties file that defines the standard

Icon	Setup level	Description
		properties, for example, `standard.j144` or `standard.j1042`. You can create additional properties files for the properties that you want to use again and give the files distinctive names.

Defaults.zxt file

When you use AutoDefaults, Tekla Structures saves the AutoDefaults rules in a zipped `defaults.zxt` text file in the `\attributes` folder under the current model folder.

You can copy the `defaults.zxt` file to the project or firm folder to make it available in other models. Each time you modify the AutoDefaults setup, you need to recopy this file to the firm or project folder. To use the modified setup in other models, restart Tekla Structures.

NOTE We do not recommend that you edit the `defaults.zxt` file using a text editor, but if you do, ensure that you are using the right syntax. The easiest way to unzip the `.zxt` file is to change the file extension `.zxt` to `txt.gz` and unzip the file. Change the extension back to `.zxt` when you have finished. You do not need to zip the file after editing it, Tekla Structures can also read the unzipped file.

Limitation

AutoDefaults only affects connection parts (clip angles, shear tabs, end plates, ...), bolts, and welds. AutoDefaults cannot change beam profiles or the connection number.

Create a rule group for AutoDefaults

You can define rule groups for AutoDefaults to group the rules according to different standards, projects, or manufacturers, for example.

1. On the **File** menu, click **Catalogs** --> **AutoDefaults settings**.
2. Right-click an existing rule group and select **New Rule Group**.
3. Click the **New** group to rename it.

   Give the rule group a name that reflects the contents of the group. For example, use the fabricator's name, the project name, or any name that clearly identifies the rules that you want to use for a specific model.

When you create a new rule group, Tekla Structures automatically adds the existing components to the group.

Create a rule set for AutoDefaults

You can create rule sets to define which connection properties are used when specific conditions in the model are met.

1. On the **File** menu, click **Catalogs** --> **AutoDefaults settings**.
2. Click the plus icon in front of the rule group to open the tree structure.
3. Click the plus icon in front of the relevant group of components and connection .
4. Right-click an existing rule set and select **New Rule Set**.
5. Right-click the new rule set and select **Edit Rule Set...**.
6. Enter a name for the rule set.
7. Select a rule from the **Available rules** list.
8. Click the right arrow button to move the selected rule into the list of **Rules in rule set**.
9. Enter the values used in the rule: either an exact value, or minimum and maximum values.
10. Select from the **Parameter files selection in rule set** list how the properties are selected in the rule set.

Option	Description
Use combination of first parameters	Tekla Structures uses the properties files it finds in the first matching sub-rule set and does not check other rule sets.
Iterate until connection symbol is green	Tekla Structures checks sub-rule sets until it finds matching properties.
Iterate until connection symbol is yellow	Tekla Structures checks sub-rule sets until it finds matching properties.
Use combination of all parameters	Tekla Structures checks all rule sets and uses the properties files in all matching rule sets. The order of the properties files is important. When Tekla Structures combines the properties files, the most recent files (the lowest in the tree) override previous ones. If you do not enter any values for the properties, Tekla Structures does not override the previous property values.

11. Click **OK**.

NOTE The order of the rules (page 788) in the tree structure is important. Tekla Structures uses the first rule that matches the conditions within the model so you should place the most limiting rule highest in the tree, and the most generic rule lowest.

You can change the priority of a rule set by right-clicking the rule set and selecting **Move up** or **Move down**.

Modify connection properties for AutoDefaults

Each connection has a default standard property file that defines the properties for the connection. You can modify the properties that the standard file uses. Save the connection properties that you want to use and set the standard file to use these properties (page 785) in the AutoDefaults settings.

1. On the **File** menu, click **Catalogs** --> **AutoDefaults settings**.
2. Click the plus icon in front of the rule group to open the tree structure.
3. Click the plus icon in front of the relevant group of components and connection .
4. Right-click the `standard.j` connection file that you want to modify, for example, `standard.j144` and select **Edit Connection Parameters...**.
5. In the connection dialog box, set the properties that you want to save.

   Such properties could be, for example, bolt properties, profiles, and materials.
6. Enter a descriptive name for the properties in the box next to the **Save as** button.
7. Copy this name in the **Connection code** option on the **General** tab.

   Using the same name allows you to check which properties Tekla Structures used in specific situations. Tekla Structures does not automatically show the AutoDefaults values in the connection dialog box.
8. Click **Save as**.

   Tekla Structures saves the properties file in the `\attributes` folder under the current model folder. The filename consists of the name you entered in **Save as** and the file extension `.jXXX`, where `XXX` is the connection number, for example, `sec_0-190.j144`.
9. Click **Cancel** to close the connection dialog box and return to the **AutoDefaults setup** dialog box.

   If you click **OK** to close the connection dialog box, you need to load the default properties the next time you use the connection. Using the default properties ensures that AutoDefaults can modify the properties.

10. Right-click the standard.j file again and select **Select Connection Parameters...**.

    The **Attribute File List** dialog box that opens contains the properties that have been set and saved in the connection dialog box.

11. Select a file in the **Attribute File List** dialog box.
12. Click **OK**.

Modify a connection using AutoDefaults

When you use a connection that you are unfamiliar with, first use the default properties. Then use AutoDefaults to modify the properties.

1. Double-click a connection symbol in the model to open the connection dialog box.
2. On the **General** tab, select a rule group from the **AutoDefaults rule group** list.
3. On all tabs, select the AutoDefaults options marked with the arrow symbol  for the properties in which you want to use AutoDefaults.
4. Click **Apply**.

If you manually modify the properties after using AutoDefaults, Tekla Structures uses the manually modified properties.

For example, you have manually set the base plate thickness of a connection to 20 mm. AutoDefaults is active and sets the plate thickness according to the main part profile. If you modify the main part profile, Tekla Structures does not update the base plate thickness. It remains at 20 mm.

NOTE You can view which AutoDefault rules and properties are used:

- To view AutoDefaults rules, select the connection symbol in the model, right-click and select **Inquire** .

  Tekla Structures shows the rule group, rule sets and properties files used.

- To view the AutoDefaults properties, double-click the connection symbol in the model, select <AutoDefaults> in the list box next to the **Load** button and click **Load**.

See also

Define AutoDefaults settings and rules (page 780)

AutoConnection and AutoDefaults rules

You can create your own AutoConnection and AutoDefaults rules for project and company defaults. By defining rules you can accurately select connections and connection properties when using AutoConnection and AutoDefaults.

General rules

- **Profile name** is the name in the profile catalog.
- **Profile type**

Profile type	Number
I	1
L	2
Z	3
U	4
Plate	5
Round bar	6
Pipe	7
Square pipe	8
C	9
T	10
ZZ	15
CC	16
CW	17
Polygon plate	51

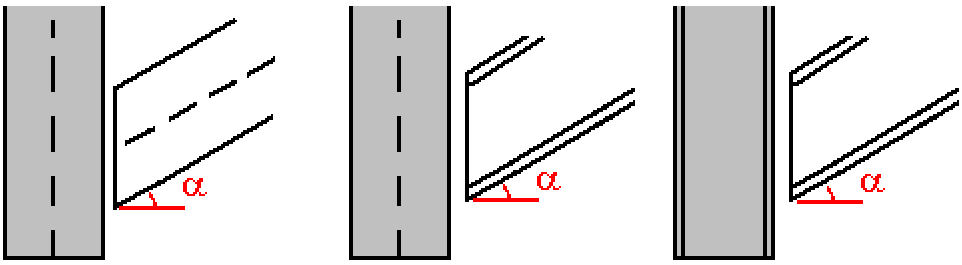
- Number of secondary parts
- Number of main parts
- Material name

Orientation rules

Depending on the relative angle of a beam, the connections can be classified as sloped, skewed, or cant. The angle value can be between - 90 and 90 degrees.

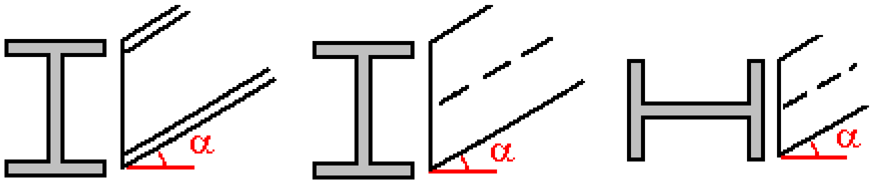
- **Sloped** angle (relative to main part cross section)

  The longitudinal axis of the secondary part follows the slope of the longitudinal axis of the main part.



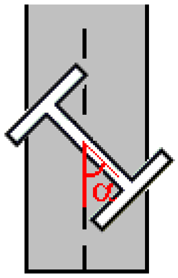
- **Skewed** angle (relative to main part longitudinal axis)

The longitudinal axis of the secondary part is skewed according to the main part cross section. The angle is the smaller of the angles between the longitudinal axis of the secondary part and the main part Z or Y axis.



- **Cant** angle

  For rotated secondary parts



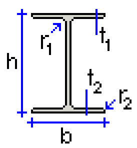
Dimension rules

- **Profile depth**
- **Web depth**

  For profiles with an upper and lower flange, the web depth is: `h-t1-t2-2*r1`

  Or, if t2 is zero: `h-2*t-2*r1`

  For profiles with one flange, the web depth is `h-t-r1-r2`.



- **Web thickness**
- **Flange thickness**

Forces and strengths

- Shear force
- Axial force
- Bending moment

See also

Combining and iterating properties for AutoDefaults (page 788)

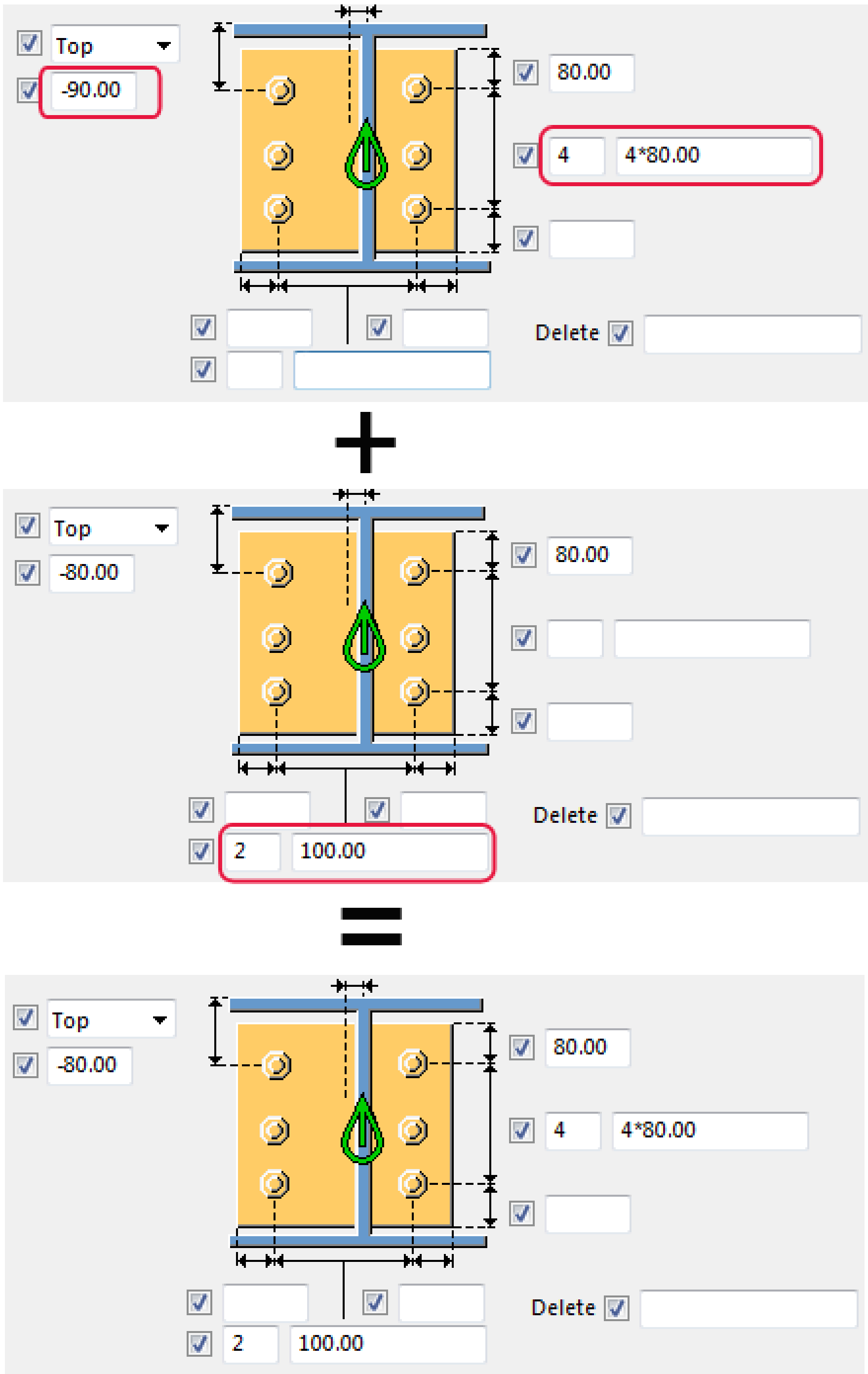
AutoDefaults example: Using iteration with connection check (page 790)

Using reaction forces and UDLs in AutoDefaults and AutoConnection (page 792)

Combining and iterating properties for AutoDefaults

Combining properties

You can save properties files that cover different groups of properties and use these files to define many rules. For example, you can have one file for bolt properties and another for profile properties. AutoDefaults combines the separate files into one file. This means that you can define fewer files because you can use one file for several rules. If the files contain different values for the same property, Tekla Structures uses the last property it finds, see the example image below.



Iterating properties

Tekla Structures tests the properties until the connection symbol is yellow or green. Iteration changes the properties automatically if the connection is not created successfully, even if the rules match. If connection check is active, the iteration results in properties that have passed the check.

Limitations

- Tekla Structures cannot iterate property files directly. Use a single iteration rule set with sub-rule sets.
- You cannot have many parallel iteration rule sets. Use a single iteration rule set and place it just before the default rule set.
- Place the combination rule sets above the iteration rule set in the AutoDefaults tree structure.
- Combination rule sets can only be one level deep.
- Tekla Structures disregards empty rule sets, so include at least one rule in each rule set.

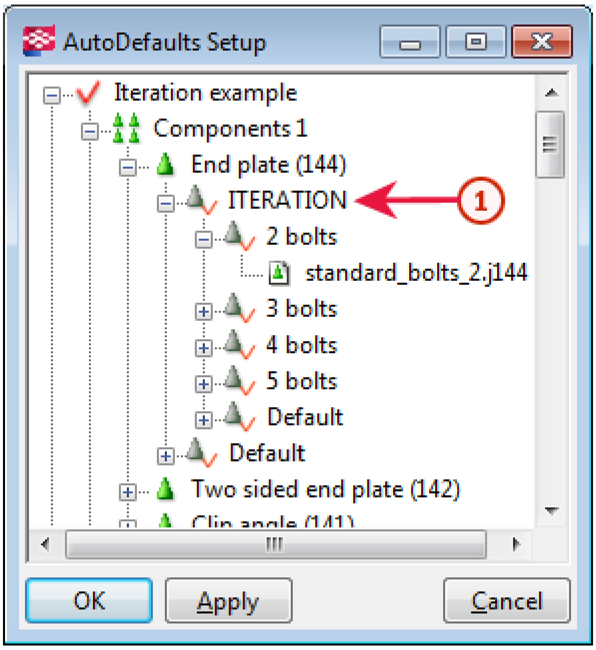
See also

Define AutoDefaults settings and rules (page 780)

AutoDefaults example: Using iteration with connection check

You can use the connection check result when using AutoDefaults with iteration. If an iteration rule matches, but the connection does not pass the connection check and the connection symbol remains red, AutoDefaults continues testing other rules and properties until the connection symbol is green.

In this example, you will create iteration rules to set the number of bolts according to the result of the connection check. After this, you will use the rules group and connection check together for a connection. The example image below shows the rules in the **AutoDefaults setup** dialog box.



To create iteration rules for use with connection check:

1. On the **File** menu, click **Catalogs** --> **AutoDefaults settings**.
2. Right-click the tree and select **New Rule Group**.
3. Click the new rule group and rename it to `Iteration example`.
4. Browse the `Iteration example` tree to find **End plate (144)**, right-click it, and select **Create Additional Rule Sets**.
5. Right-click the **New** rule set and select **Edit Rule Set**.
6. Change the rule set name to `ITERATION`.
7. Set the **Parameter file(s) selection in rule set** option to **Iterate until connection symbol is green**.
8. Click **OK**.
9. Right-click the `ITERATION` rule set and select **Create Additional Rule Sets**.
10. Right-click the **New** rule set and select **Edit Rule Set**.
11. Change the rule set name to `2 bolts`.
12. Select the rule **Secondary 1 depth** and set the minimum and maximum depth values for two bolts.
13. Set the **Parameter file(s) selection in rule set** option to **Use combination of first parameters**.
14. Click **OK**.

15. Right-click the connection properties file `standard.j144` under `2 bolts` and select **Select Connection Parameters**.
16. Select a properties file for two bolts in the **Attribute File List** and click **OK**.

    TIP If there is no suitable properties file, you can create a new file. Right-click the `standard.j144` file and select **Edit Connection Parameters**. Save the needed properties and click **Cancel** to close the dialog box. The saved properties are now available in the **Attribute File List**.

    ---

17. Click **Apply** to have the changes available in the connection dialog box.
18. Repeat steps 9 to 16 for other rule sets.
19. Open the **End plate (144)** dialog box.
20. Select `<Defaults>` from the list next to the **Load** button and click **Load**.
21. On the **General** tab, set the **AutoDefaults rule group** option to the `Iteration example` you created.
22. On the **Design type** tab, set the **Check connection** option to **Yes**.
23. Enter the load from secondary members in the **Shear**, **Tension**, and **Moment** options.
24. Click **OK**.

See also

Define AutoDefaults settings and rules (page 780)

Combining and iterating properties for AutoDefaults (page 788)

Using reaction forces and UDLs in AutoDefaults and AutoConnection

You can set reaction forces for AutoConnection and AutoDefaults in the user-defined attributes of a part, and for AutoDefaults also on the **Design** tab in the connection dialog box.

Reaction forces

When you use reaction forces in a rule and AutoDefaults is activated, Tekla Structures first searches for reaction forces in the corresponding connection's properties. If the properties do not contain reaction forces, Tekla Structures searches the user-defined attributes of the secondary part of the connection. If Tekla Structures does not find forces there, you cannot use reaction force rules.

Shear force calculation

If you have not given any reaction force values, shear force is calculated using the UDL (uniformly distributed load) shear force routine. The UDL calculation is mainly intended for use with imperial units. It uses the yield stress value, profile dimensions, and UDL percentage to calculate the maximum shear force allowed.

- Yield stress is defined in the material catalog.
- Profile dimensions come from the profile catalog.
- UDL percentage is taken either from the connection dialog box or from an advanced option.

Tekla Structures compares the result with the shear force rule in AutoDefaults.

To use UDLs for AutoConnection and AutoDefaults:

To	Do this
Use UDL for AutoConnection	1. On the **Design** tab in the connection dialog box, set the UDL option to **Yes**. 2. Enter the UDL percentage in the **UDL%** box. If you do not enter any value, Tekla Structures uses a default percentage set with the `XS_AUTODEFAULT_UDL_PERCENT` advanced option.
Use UDL for AutoDefaults	1. On the **Design** tab in the connection dialog box, set the **Use UDL** option to **Yes**. 2. Enter the UDL percentage in the **UDL %** box. If you do not enter any value, Tekla Structures uses a default percentage set using the `XS_AUTODEFAULT_UDL_PERCENT` advanced option.

See also

Design and Design type tabs (page 818)

7.9 Advanced component settings

This section describes how to set default properties for different connection types, how to use Excel spreadsheets in connection design, and advanced component properties, such as analysis and design properties.

Click the links below to find out more:

Define connection properties in the joints.def file (page 794)

Excel spreadsheets in connection design (page 806)

General tab (page 816)

Design and Design type tabs (page 818)

Analysis tab (page 820)

Define connection properties in the joints.def file

The `joints.def` file contains general connection settings and connection-specific settings for different connection types. You can use the `joints.def` file to set the default properties for different connection types. `Joints.def` is a text file that you can open and edit in any standard text editor.

Tekla Structures uses the values defined in the `joints.def` file for the properties that do not have values in the connection dialog boxes. If you manually enter values in the connection dialog boxes, the manually entered values are used instead of the values in the `joints.def` file. AutoDefaults also override the values defined in the `joints.def` file.

Tekla Structures stores the `joints.def` file in the system folder. Tekla Structures searches for the `joints.def` file in the standard search order: model, project, firm, and system folder.

How to use the joints.def file

The `joints.def` file contains general connection settings and connection-specific settings for different connection types in separate sections. You can modify the `joints.def` file using any standard text editor.

When you modify the file:

- Enter absolute values or names.
- Do not use feet and inch symbols.
- Ensure that the profiles exist in the profile catalog.
- Ensure that the bolts exist in the bolt catalog.
- You can set the measurement units at the beginning of the file.
- You can define in the `JOINTDEFAULT` line whether Tekla Structures uses the default values in the `joints.def` file or the system default values, for example, as follows:

  ```
  // is default file available (1) or not (0)
  JOINTDEFAULT   1
  ```

  - Value `1` means that the default values defined in the `joints.def` file are used.
  - Value `0` means that the system default values are used.
  - The `//` characters at the beginning of a line mean that the line is a comment line. Tekla Structures does not use the information on these lines.

- You can force Tekla Structures to use the system default for a particular property by entering the value `-2147483648` for the property.

Connection-specific properties

The properties for clip angles, shear tabs, end plates, gusset connections and diagonal connections are in separate sections. Each section begins with a header row that contains the column labels, for example as follows:

`joints.def`

```
// name          part     lproflength    diameter   number_of_bolts
BOLTHEIGHT       GUSSET   100            20.0       2
```

Do not add columns to the file. If Tekla Structures cannot find a property in the connection-specific section, it searches for the default property in the general defaults section.

Connections that use the joints.def file

The following connections use the `joints.def` file:

- **Welded gusset (10)**
- **Bolted gusset (11)**
- **Bracing cross (19)**
- **Tube gusset (20)**
- **Tube crossing (22)**
- **Two sided angle cleat (25)**
- **Corner tube gusset (56)**
- **Corner bolted gusset (57)**
- **Wraparound gusset (58)**
- **Hollow brace wraparound gusset (59)**
- **Wraparound gusset cross (60)**
- **Wrapped cross (61)**
- **Gusseted cross (62)**
- **Corner wrapped gusset (63)**
- **Beam with stiffener (129)**
- **Column with shear plate (131)**
- **Bolted moment connection (134)**
- **Clip angle (141)**
- **Two sided end plate (142)**
- **Two sided clip angle (143)**
- **End plate (144)**
- **Shear plate simple (146)**

- **Welded to top flange (147)**
- **Welded to top flange S (149)**
- **Moment connection (181)**
- **Column with stiffeners W (182)**
- **Full depth (184)**
- **Full depth S (185)**
- **Column with stiffeners (186)**
- **Column with stiffeners S (187)**
- **Column with stiffeners (188)**
- **Shear plate tube column (189)**
- **Bent plate (190)**

Example: How Tekla Structures uses the joints.def file

This example explains how Tekla Structures calculates the bolt diameter and other properties of the **Bolted gusset (11)** connection using the `joints.def` file.

The height of the diagonal profile is 10". Tekla Structures calculates the bolt size and the number of bolts according to the profile height. It searches the `BOLTHEIGHT` rows for a profile height of 10".

As the profile height is greater than `8.0` but under `12.0`, Tekla Structures uses the row with profile height `8.0`. This sets the bolt diameter to `0.75`.

```
// DIAGONAL JOINTS
// diagonal default boltdiameters depending on prof height, higher prior than
//
// name             part        profileheight     diameter  number_of_bolts
BOLTHEIGHT          DIAGONAL    3.0               0.75      1
BOLTHEIGHT          DIAGONAL    8.0               0.75      2
BOLTHEIGHT          DIAGONAL    12.0              0.75      3
BOLTHEIGHT          DIAGONAL    16.0              0.75      4
BOLTHEIGHT          DIAGONAL    18.0              0.75      5
```

Tekla Structures uses the bolt diameter to assign the bolt and part properties. It searches the `DIAGBOLTPART` rows for bolt diameter `0.75`.

```
//              bolt      angle       conn.plate |    horizontal bolts          |  vertical bolts
// name         diameter  profile     thickness  number   pitch    edge_dist  number       pitch        edge_
DIAGBOLTPART    0.5       L4X3X1/2    0.375      2        1.5      1.0        -2147483648  -2147483648  1.0
DIAGBOLTPART    0.75      L4X4X1/2    0.375      2        2.5      1.5        -2147483648  -2147483648  1.5
DIAGBOLTPART    1.0       L5X5X1/2    0.375      2        3.0      2.0        -2147483648  -2147483648  2.0
```

The following property values are used:

Bolt diameter	0.75
Number of bolts horizontally	2
Edge distance horizontally	1.5

Edge distance vertically	1.5
Distance between bolts horizontally	2.5
Distance between bolts vertically	System default is used.

Tekla Structures does not use the connection plate thickness or angle profile properties in this connection.

General defaults in the joints.def file

Tekla Structures uses the general defaults in the `joints.def` file if it cannot find a connection property in the connection-specific section.

For example, for clip angles, Tekla Structures determines the bolt diameter and the number of bolts according to the secondary beam height. If the secondary beam is higher than the highest value in the clip angle section in the `joints.def` file, Tekla Structures uses the default bolt diameter in the general defaults.

The properties in the general defaults section in the `joints.def` file are:

Property	Description
`boltdia`	Bolt diameter
`pitch`	Distance from the center of one bolt to the center of the next bolt
`clipweld`	Weld size
`angle-cc-inc`	Tekla Structures adds bolt to bolt distance and web thickness, then rounds up the result using this value. Complies with the US AISC standard.
`lprofgapinc`	Tekla Structures rounds up the angle profile gap using this value. Complies with the US AISC standard.
`lsize`	Size of the angle profile
`copedepth`	Notch size
`copelength`	Notch size
`boltedge`	Edge distance
`webplatelen`	Haunch plate height (h)
`webplatewid`	Haunch plate width (b)
`beamedge`	Setback distance between the end of the beam and the main part
`knifeclr`	No longer used
`clipedge`	Edge distance for bolts (clip angles only)

Property	Description
gap	No longer used
shearplatethk	Shear tab thickness
endplatethk	End plate thickness
shearweld	Size of weld
cliplsize	Size of angle profile (clip angles only)
flangecutclear	Flange cut clearance
slotsize	Size of slotted hole
clipslots	Part with slotted holes: • 1 = beam • 2 = angle profiles • 3 = both This property is the **Slots in** option on the **Bolts** tab.
clip_attac	Clip angle attached to the main part and secondary parts: • 1 = both parts bolted • 2 = main part bolted/secondary part welded • 3 = main part not welded • 4 = main part welded/secondary part bolted • 5 = both parts welded • 6 = main part not bolted • 7 = secondary part not welded • 8 = secondary part not bolted • 9 = both parts bolted/welded This property is the bolt attachment option on the **Bolts** tab where the location of bolts is defined.
copedepth_inc	Tekla Structures rounds up notch depth using this value.
copelength_inc	Tekla Structures rounds up notch length using this value.

Bolt diameter and number of bolts in the joints.def file

In the `joints.def` file, the `BOLTHEIGHT` rows in each connection-specific section show the default bolt diameter and default number of bolt rows for the connection type.

Tekla Structures determines the bolt diameter and the number of bolts according to the connection type based on the following properties:

For	According to
Clip angles	Secondary beam height
Shear tabs	Secondary beam height
End plates	Secondary beam height
Gusset connections	Angle profile length
Diagonal connections	Profile height

Clip angle, shear tab, and end plate connections

Tekla Structures calculates the default bolt diameter and the number of bolt rows vertically according to the height of the secondary beam. You can enter the following properties:

Property	Description
`name`	`BOLTHEIGHT`
`part`	`ANGLECLIP`
`sec.beam.height`	Minimum height of the secondary beam for a certain number of bolts
`diameter`	Bolt diameter. The diameter must exist in the bolt catalog.
`number_of_bolts`	Number of bolts vertically

Gusset connections

Tekla Structures calculates the default bolt diameter and the number of bolt rows horizontally according to the length of the angle profile. You can enter the following properties:

Property	Description
`lproflength` or `angleproflength`	Length of the angle profile
`name`	`BOLTHEIGHT`
`part`	`GUSSET`
`diameter`	Bolt diameter. The diameter must exist in the bolt catalog.
`number_of_bolts`	Number of bolts horizontally

Diagonal connections

Tekla Structures calculates the default bolt diameter and number of bolt rows horizontally according to the profile height. You can enter the following properties:

Property	Description
name	BOLTHEIGHT
part	DIAGONAL
conn.pl.height or profileheight	Profile height
diameter	Bolt diameter. The diameter must exist in the bolt catalog.
number_of_bolts	Number of bolts horizontally

Bolt and part properties in the joints.def file

Once Tekla Structures has used the joints.def file to calculate the bolt diameter, it uses the result to assign other properties to bolts and parts, according to the connection type.

In clip angle connections, for example, the default properties for bolts and parts are in the rows that begin with ANGLECLBOLTPART in the CLIP ANGLE section of the joints.def file.

The table below lists the properties that you can assign for bolts and parts in each connection type.

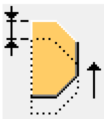
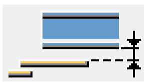
Property	Description	Clip angle	Shear tab	End plate	Gusset	Diagonal
name	Identifies the connection type. For example, GUSSETBOLTPART for gusset connections.	*	*	*	*	*
bolt diameter	The bolt diameter must exist in the bolt catalog.	*	*	*	*	*
shear plate thickness	Thickness of the shear tab		*			
end plate thickness	Thickness of the end plate			*		

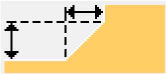
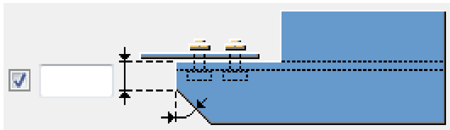
Property	Description	Clip angle	Shear tab	End plate	Gusset	Diagonal
gusset thickness	Thickness of the gusset plate				*	
conn. plate thickness	Thickness of the connection plate					*
angle profile or L profile	The name of the used angle profile must exist in the profile catalog. Enter the exact profile, for example: L100*100*10.	*			*	*
number	Number of bolts in each row vertically and horizontally.	*	*	*	*	*
pitch	Distance between the bolts from the center of each bolt for vertical and horizontal bolts	*	*	*	*	*
edge distance	Distance from the center of a bolt to the edge of the part for vertical and horizontal bolts	*	*		*	*
vert. bolt firshole	Position of the first vertical row of bolts	*	*		*	

Gusset connection properties in the joints.def file

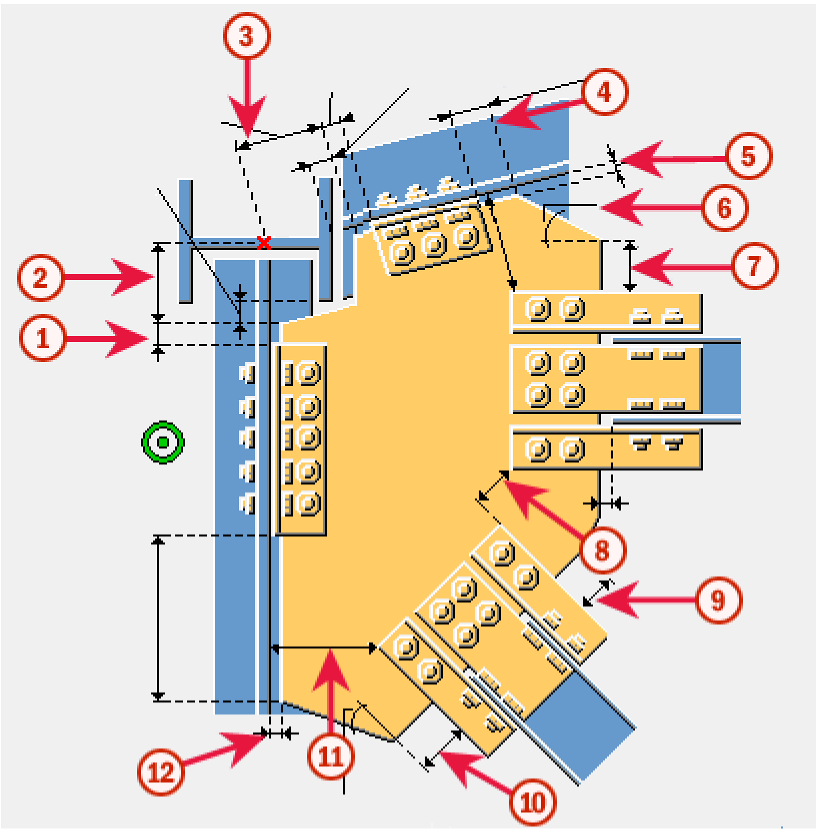
Enter the additional default properties for gusset connections in the GUSSETDEFDIM row . All gusset connections do not use all the properties.

Property	Description	Affects plate shape
name	GUSSETDEFDIM	
boltdia_def	Bolt diameter for all bolt groups Tekla Structures uses this value if the **Bolt size** box is empty in the connection dialog box.	
tol_prim	Tolerance between the gusset and main part web	

Property	Description	Affects plate shape
tol_sec	Tolerance between the gusset and secondary part web	
dist_diag_prim	Clearance between the first secondary part selected and the main part	
dist_diag_sec	Perpendicular distance from the last secondary part selected to the nearest secondary part	
angle_first_corner	Corner angle dimension	Yes
angle_sec_corner		
dist_between_diag	Clearance between braces	
first_bolt_from_line	Bolt edge distance for the bolt groups on the **Gusset** tab	
corner_dx	Corner dimension	
corner_dy	Corner dimension	
movey	 option on the **Gusset** tab	
movez	 option on the **Gusset** tab	
dist1	Edge length of the gusset plate perpendicular to the lowest brace	Yes
dist2	Edge length of the gusset plate perpendicular to the braces	Yes
dist3	Edge length of the gusset plate perpendicular to the uppermost brace	Yes
tol_lprof	Edge tolerance from gusset plate to connection plate	
tol_stiffener	Stiffener tolerance	
chamfer_dx	Stiffener chamfer dimension on the **Gusset** tab	
chamfer_dy	Stiffener chamfer dimension on the **Gusset** tab	

Property	Description	Affects plate shape
`chamfer_corner_dx`		
`chamfer_corner_dy`		
`side_length`	Side length	
`diafit_length`	Fit length in the **Bracing cross (19)** connection. Tekla Structures uses this value if the option on the **Parameters** tab is empty. 	

The example image below shows the properties of the **Wraparound gusset (58)** connection on the **Picture** tab.

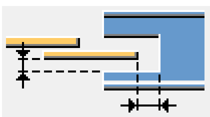
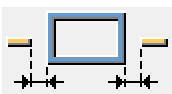


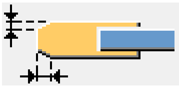
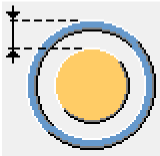
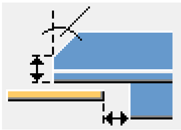
1. `tol_lprof`
2. `corner_dy`
3. `corner_dx`

4. dist_diag_sec
5. tol_sec
6. angle_sec_corner
7. dist3
8. dist_between_diag
9. dist2
10. dist1
11. dist_diag_prim
12. tol_prim

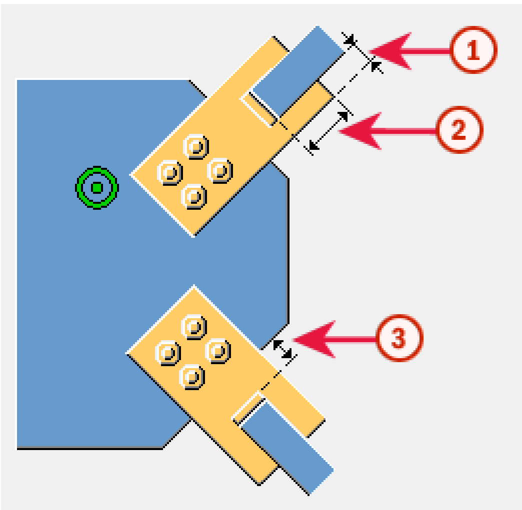
Diagonal connection properties in the joints.def file

Enter the additional default properties for bolts and parts in the DIAGDEFDIM row. All diagonal connections do not use all the properties.

Property	Description
name	DIAGDEFDIM
boltdia_def	Bolt diameter for all bolt groups Tekla Structures uses this value if the **Bolt size** box is empty in the connection dialog box.
dist_gus_diag	Gap between the gusset plate and the brace If the tube profiles are closed with end plates, dist_gus_diag is the gap between the gusset plate and the end plate. See the **Tube crossing (22)** image below.
dist_in	Cut depth in the brace. Enter a negative value to prevent the connection plate from being inside the tube brace. See the **Tube crossing (22)** image below.
dist_dv	Brace edge distance to the edge of the connection plate. This dimension changes the width of the connection plate. See the **Tube crossing (22)** image below.
sec_cut_tol slot_length_tol	On the **Brace conn** tab: 
tube_cut_tol	On the **Brace conn** tab: 

Property	Description
conn_cut_dx	On the **Brace conn** tab: 
conn_cut_dy	
round_plate_tol	On the **Brace conn** tab: 
flanges_cut_angle	On the **Brace conn** tab: 
dist_flanges_cut	
dist_skew_cut	
end_plate_thk	End plate thickness

The example image below shows the properties of the **Tube crossing (22)** connection on the **Picture** tab:



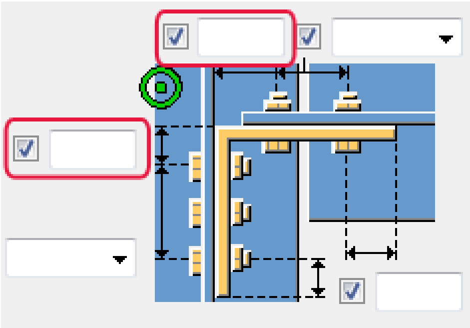
1. dist_dv
2. dist_in
3. dist_gus_diag

Profile dependent bolt dimensions in the joints.def file

For some connections, such as **Clip angle (141)** and **Two sided clip angle (143)**, Tekla Structures calculates the bolt size according to the profile size.

For these connections, Tekla Structures takes the bolt size from the PROFILEBOLTDIM rows of the PROFILE TYPE-DEPENDENT BOLT

DIMENSIONS section in the joints.def file if you leave the corresponding options empty on the **Bolts** tab.



Property	Description
width	Profile width
one bolt firsthole	For single bolts, distance from the edge of the profile angle to the first hole
two bolts firsthole	For two bolts, distance from the edge of the profile angle to the first hole
pitch	Distance between bolts from the center of each bolt, for vertical and horizontal bolts

For example, to find the bolt dimensions to be used with an L6X6X1/2 profile in a clip angle connection:

1. Tekla Structures first searches the PROFILEBOLTDIM rows for L6X6X1/2 in the PROFILE TYPE-DEPENDENT BOLT DIMENSIONS section.
2. If there is no match, Tekla Structures then searches the ANGLECLBOLTPART rows in the CLIP ANGLE section.

Excel spreadsheets in connection design

You can use Excel spreadsheets in connection design for all steel connections that have the **Design** or **Design type** tab in the connection dialog box.

You can link connections to Excel spreadsheets by selecting **Excel** in the **External design** option on the **Design** or **Design type** tab. The connection information is transferred to the connection type-specific spreadsheet where the needed calculations are made. The calculated properties are saved to an output file and the modified component property values are transferred back to the connection. The connection is modified according to the changes.

You can create an Excel spreadsheet for a connection type using the component_template.xls file available in the ..\Tekla Structures\<version>\Environments\common\exceldesign folder, or use a predefined file.

Files used in Excel spreadsheet connection design

The following files are used in the connection design with Excel spreadsheets:

File	Description
Visual Basic script file	The Excel.vb file links Tekla Structures with the external software and defines the Excel spreadsheet file names and the locations. The file is located in the ..\Tekla Structures\<version>\Environments\common\exceldesign folder.
Component type-specific Excel spreadsheet	The component type-specific spreadsheet contains predefined calculations. When you run the connection design, the connection properties and information of the main and secondary parts are transferred to the **Input** and **Component** sheets of the Excel spreadsheet. Excel searches for the relevant spreadsheet file in the following order: 1. From the \exceldesign folder in the current model folder: file named as component_ + number or name + .xls, for example, ..\test_model\exceldesign\component_144.xls. 2. From the location defined with the XS_EXTERNAL_EXCEL_DESIGN_PATH advanced option as follows: XS_EXTERNAL_EXCEL_DESIGN_PATH (= %XS_DIR%\environments\common\exceldesign\) + "component_" + number + ".xls"
Connection specific result file	The result file contains the modified connection properties. • The result file is created automatically from the **Calculation** sheet. • The file is stored in the \exceldesign folder in the model folder and named with the GUID, Globally Unique Identifier. • The file is updated each time you modify the connection. • The calculation results can be stored as an Excel spreadsheet, or in HTML or PDF format, depending on how the calculation spreadsheet is configured.

File	Description
Template spreadsheet	The `..\Tekla Structures\<version>\Environments\common\exceldesign` folder contains a `component_template.xls` spreadsheet you should use to create your own spreadsheet applications to use with Tekla Structures components.

Example of an Excel spreadsheet in connection design

The images in this example show the Excel spreadsheet that is used for the **End plate (144)** connection.

The sample spreadsheet has the following sheets:

The **Calculation** sheet contains a report of the calculations.

Endplate Unity Check

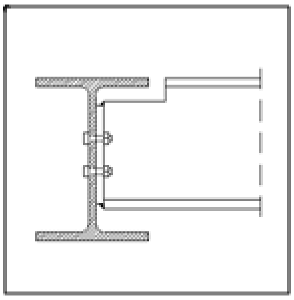
Contract Tekla Sample

Ref: User

Date: 17.03.05 4:52:44 PM

Calculated according to
BS

Connection Referen 130



Framing Condition Beam - Beam

Section		
Primary Section	IPE300	S235JR
Secondary Section	IPE300	S235JR
Endplate Size	10 mm	S275JR
Plate (length*width)	200 * 180 mm	

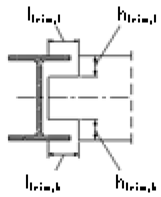
Bolt	
Bolt	20
Bolt Grade	7990
Shear area	Thread
Screw Thread	Rolled
Endplatelength in calculation	

Parameters of Connection

$e_{1,t}$ = 40 mm	a_{weld} = 6 mm	e_{bolt} = 66 mm
$e_{1,b}$ = 40 mm	s_1 = 60 mm	$n_{r,prim}$ = 3
e_2 = 40 mm	s_2 = 67,09999	$n_{c,prim}$ = 2

Notch

$l_{trim,t}$ = 82 mm
$h_{trim,t}$ = 26 mm
$l_{trim,b}$ = 82 mm
$h_{trim,b}$ = 26 mm



Shear of the endplate
F = 261 kN [BS 5950-1:2000 6.2.3/6.2.4]

Bearing of the endplate
F = 606 kN [BS 5950-1:2000 6.3.3.3]

Shear of beam near the weld
F = 200 kN [BS 5950-1:2000 6.8.7.3]

Shear (& Tension) on the bolts
F = 230 kN F = 23 kN [BS 5950-1:2000 6.3.2/6.3.4.3]

The **Inputs** sheet contains the properties of the connection from the connection dialog box.

	A	B	C	D	E	F
1				**Attribute**	**Value**	**Type**
2		**Plate**				
3			Material	mat		string
4			Thickness	tpl1	10	double
5			Depth	hpl1	-2147483648	double
6			Width	bpl1	180	double
7						
8		**Bolt**				
9			Diameter	diameter		double
10			Grade	screwdin		string
11				lbd	-2147483648	string
12				lwd	-2147483648	string
13				lba	-2147483648	double
14				nb	-2147483648	int
15				nw	-2147483648	int
16				rb1	-2147483648	double
17				rb2	-2147483648	double
18				rw1	-2147483648	double
19				rw2	-2147483648	double
20						
21		**Weld**				
22				w3_size	-2147483648	double
23						
24		**Notch**				
25				t_cut_length	-2147483648	double
26				t_cope_length	-2147483648	double
27				b_cut_length	-2147483648	double
28				b_cope_depth	-2147483648	double
29						
30		**Loading**				
31				designcode	0	int
32				END		

The **Outputs** sheet contains the design results. These values are transferred to the connection and the connection in the model is modified accordingly.

The **Component** sheet contains calculations, information on the connection geometry, and on the main part and the secondary parts. The component attributes in the spreadsheet are the same as in the corresponding `.inp` file. See more about `.inp` files in .

	A	B	C	D
1	**Connection**	**Attribute**	**Value**	
2	Connection id in model	id	130	
3	Connection class	group	99	
4		flags	50	
5	Number of the connection	jointnumber	144	
6	Local x-coordinate of Connection up directio	up.x	0	
7	Local y-coordinate of Connection up directio	up.y	0	
8	Local z-coordinate of Connection up directio	up.z	1000	
9	Model Directory	ModelDirectory	C:\TeklaStructuresModels\	
10		END		
11			**Primary**	**Secondaries**
12		attribute	value	value 1
13	Primary and secondary ids	id	108	70
14	**PartCoordinateSystem**	x.x	-9,11626E-13	6000
15	y-coordinate of part origin (first end) point	x.y	8000	-9,13758E-13
16	z-coordinate of part origin (first end)point	x.z	-150	-150
17	x-coordinate of second end point of part	y.x	12000	6000
18	y-coordinate of second end point of part	y.y	8000	8000
19	z-coordinate of second end point of part	y.z	-150	-150
20	x-coordinate of parts up direction point	z.x	-9,11626E-13	6000
21	y-coordinate of parts up direction point	z.y	8000	-9,13758E-13
22	z-coordinate of parts up direction point	z.z	850	850
23	**PartExtrema**			
24	Minimum x value of primary or secondary pa	min.x	-9,11626E-13	5925
25	Minimum y value of primary or secondary pa	min.y	7925	-9,13758E-13
26	Minimum z value of primary or secondary pa	min.z	-300	-300
27	Maximum x value of extrema	max.x	12000	6075
28	Maximum y value of extrema	max.y	8075	8000
29	Maximum z value of extrema	max.z	0	0
30	**FramingCondition**			
31	Member type (Column, Beam)	Type	1	1
32	Profile name	Name	COLUMN	BEAM
33	Profile type	ProfileType	1	1
34	Skew angle between primary/sec	SkewAngle		0
35	Slope angle between primary/sec	SlopeAngle		0
36	Cantilever angle between primary/sec	AngleCant		90
37		Offset		0
38	Shear force at connection end of the beam	ShearForce		-2147483648
39	Axial force at connection end of the beam	AxialForce		-2147483648
40	Moment at connection end of the beam	BendingMoment		-2147483648
41	Use uniformly distributed load	UseUDL		0
42	How many percents from maximum uniformly	UDLPercent		0

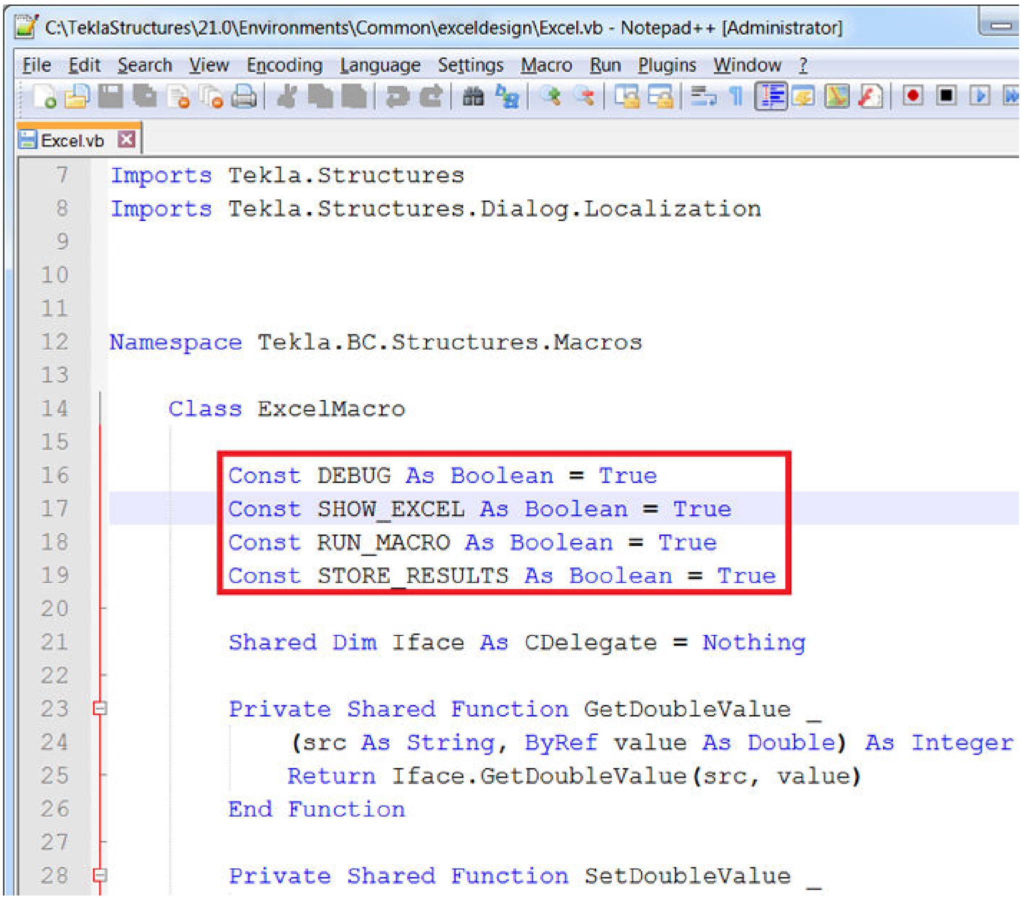
The following sheets are also included:

- **Data** shows catalog information.
- **Norm** shows the code selection and references to codes.
- **Language** shows translations.

Example of visualizing the Excel connection design process

You can define in the `Excel.vb` file how the Excel connection design process is visualized. The `Excel.vb` file links Tekla Structures with the external software and defines the Excel spreadsheet file names and the locations.

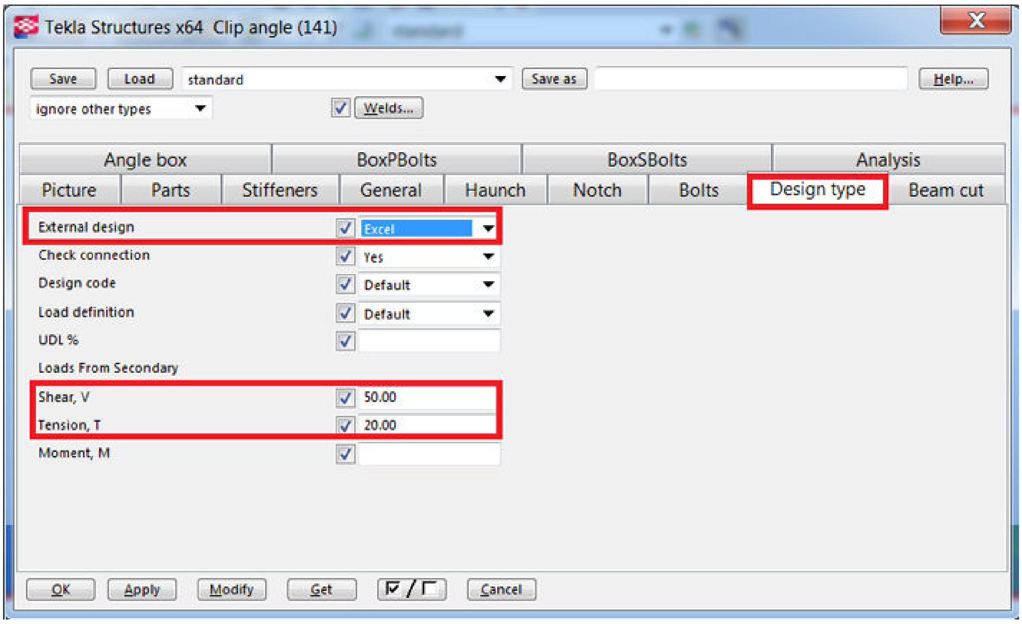
1. Open the `Excel.vb` file located in `..\Tekla Structures\<version>\Environments\common\exceldesign`.
2. Configure the `Excel.vb` file as follows:



   - Visualizing control - `Const DEBUG As Boolean= True`
   - Visualizing using Excel - `Const SHOW_EXCEL As Boolean = True`
   - Storing the output - `Const STORE_RESULTS As Boolean = True`
3. Save the file.
4. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
5. Search for **Clip angle (141)** and double-click it to open the properties dialog box.
6. On the **Design type** tab:

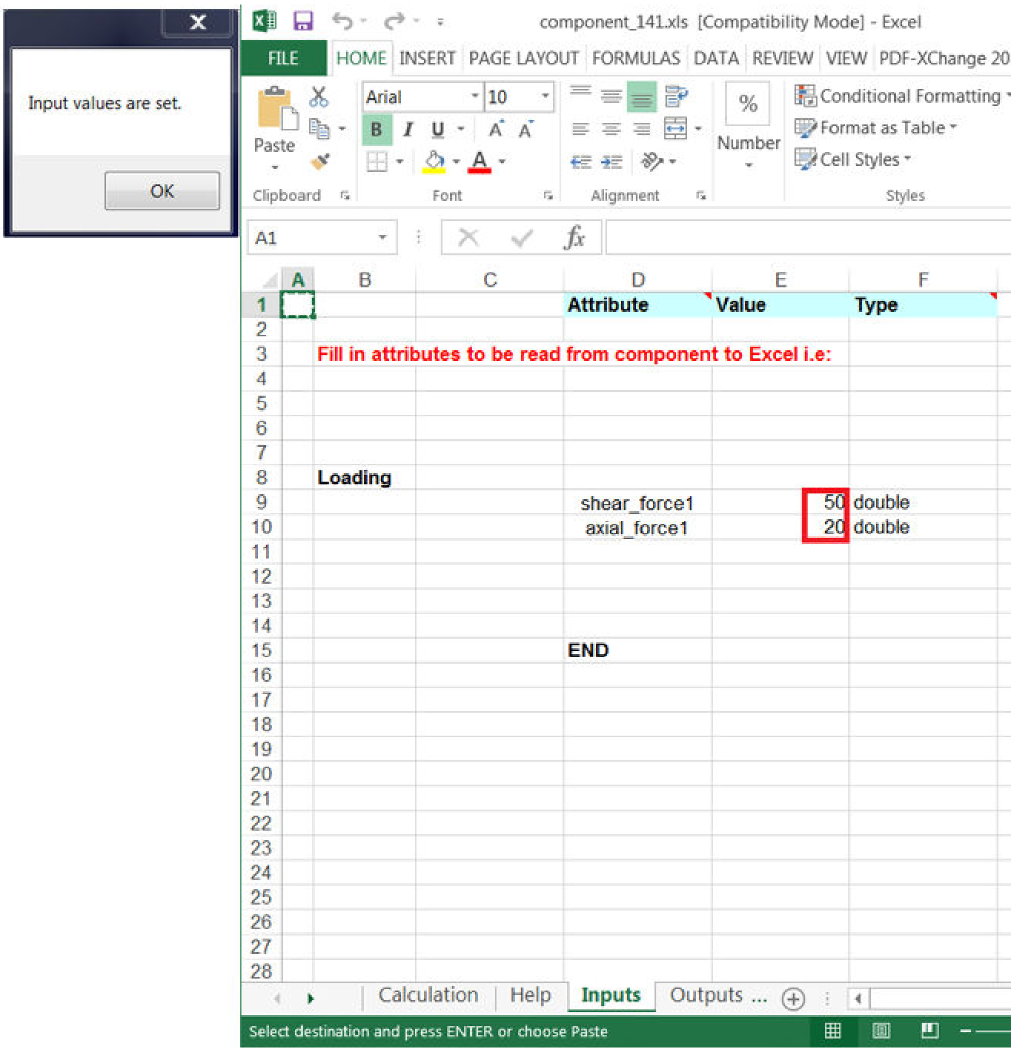
a. Select **Excel** in the **External design** option.

b. Enter the load values.



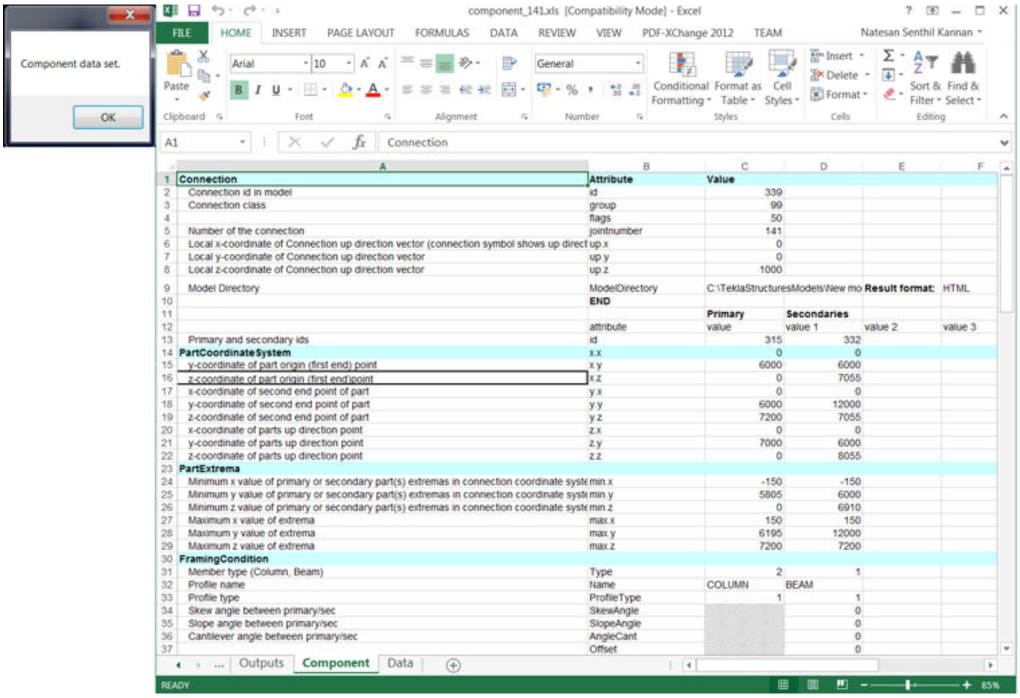
7. Click **Modify**.

The Excel design file opens and shows the **Inputs** sheet.



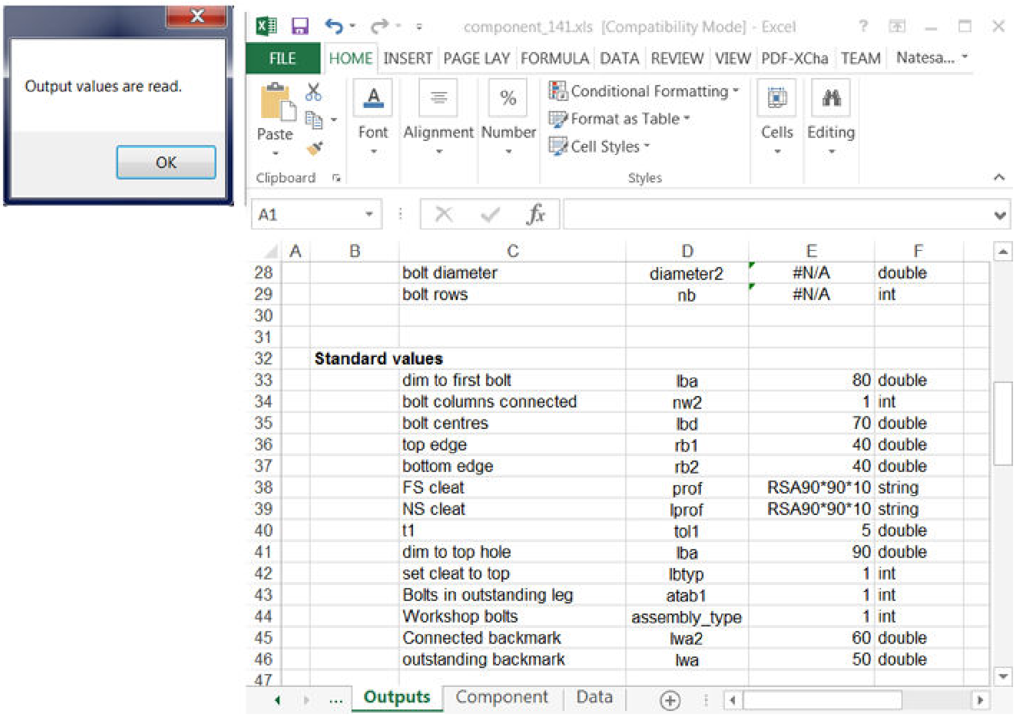
8. Click **OK** to continue.

The Excel design process now calculates the data that is shown on the **Component** sheet, and then opens the **Component** sheet.



9. Click **OK** to continue.

The Excel design process now calculates the result output values and then opens the **Outputs** sheet. The calculated results values are transferred to the connection.



10. Save the file in the model folder.
11. When you click **OK**, the design process is completed and the Excel design file is closed.

Showing connection status in Excel connection design

When you use Excel spreadsheets in connection design, you can have Tekla Structures use different colors in component symbols to show the status of a component in the model.

You can do this by including an error attribute on the **Outputs** sheet of the component's Excel spreadsheet. The type of the attribute is `int`.

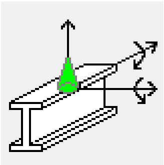
The possible values are:

Value	**Color**	**Status**
1	Green	Bolt edge distances are sufficient. The connection passes the connection design check using the UK and US design codes embedded in the system.
2	Yellow	Bolt edge distances are insufficient according to the value defined in the **Components** settings in **File** --> **Settings** --> **Options**.
3	Red	Tekla Structures cannot calculate the component properties. The possible reasons are: • The connection direction is not correct. • The work plane is not correct. • The selected connection is not appropriate for the situation. • The connection design check was carried out using the embedded UK and US design codes and the connection cannot support the loading you have defined.

NOTE The component symbol color can only be controlled for system components, not for custom components.

General tab

The **General** tab is available in steel connections and steel details.

Option	Description
Up direction 	Rotates the connection around the secondary part or the detail around the main part. You can define the rotation angle around the x- and y-axis of the secondary part. The upper box is for the y-axis and the lower box for the x-axis.
Position in relation to primary part	Available only for details. The check boxes next to the images indicate the position of the definition point of the detail, relative to the main part. **Horizontal offset** and **Vertical offset** define the horizontal and vertical alignment of the detail, relative to the main part.
Detail type	Available only for details. This option determines on which side of the part the detail will be created, but it depends on where you pick the input point of the detail. • **End detail** Pick a point on a part. The detail will be created from the picked point towards the start or end point of the part depending on which one of them is further from the picked point. If you pick the mid point of the part, the detail will be created towards the end (magenta handle) point of the part. If you pick the start point of the part, the detail will be created towards the end point (and vice versa). • **Intermediate detail** Pick a point on a part. The detail will be created from the picked point towards the start (yellow handle) point of the part. Do not pick the start (yellow handle) point of the part. • **Intermediate detail (reverse)** Pick a point on a part. The detail will be created from the picked point towards the end (magenta handle) point of the part. Do not pick the end (magenta handle) point of the part.
Locked	Prevents modifications. You can use the `privileges.inp` file to control the access to the **Locked** attribute.
Class	A number given to all parts the connection creates. You can use class to define the color (page 657) of the parts in the model.

Option	Description
Connection code	Identifies the connection. Enter a suitable code. You can display the connection code next to the component in the model and in connection marks in drawings. 1. Open the **View Properties** dialog box by double-clicking in the model and click **Display...**. 2. Check that the **Component symbols** check box is selected on the **Settings** tab. 3. On the **Advanced** tab, select the **Connection text** check box. 4. Click **Modify**. If you have not entered a connection code, the connection name will be displayed next to the component.
AutoDefaults rule group	Automatically sets connection properties according to the selected rule group. Rule group **None** switches AutoDefaults off.
AutoConnection rule group	Automatically switches the connection to another according to the selected rule group.

See also

AutoDefaults (page 780)

AutoConnection (page 775)

Design and Design type tabs

Some component dialog boxes include a **Design** tab, others include a **Design type** tab. You can use the options on these tabs to check if the component will bear the uniform distributed load (UDL). Some **Design** tabs include only the design check. Tekla Structures saves the design summary as a `.txt` file in the model folder.

You can use AutoDefaults rule groups and Excel files in the design check:

- AutoDefaults rule groups automatically modify component properties to take the calculated load. To define which AutoDefaults rule group to use, go to the **General** tab and select the rule in the **AutoDefaults rule group** list box.

  For more information, see Using reaction forces and UDLs in AutoDefaults and AutoConnection (page 792).

- The information in an Excel file checks the connection design and automatically updates component properties to bear the UDL. This is useful when you want to check connection design according to other design codes. See Excel spreadsheets in connection design (page 806).

Design tab

This design check is intended to be used with imperial units.

To check the design:

1. Go to the **Design** tab and select **Yes** in the **Use UDL** list.
2. To use information in an Excel spreadsheet in the UDL calculation, select **Excel** in the **External design** list.
3. Enter the information you want to use in the calculation.
4. Select the connection in the model and click **Modify**.

   Tekla Structures checks the component. A green component symbol indicates that the connection will bear the UDL, red indicates that it will not.
5. To view the results of the check, right-click the component symbol and select **Inquire** from the pop-up menu.

   The **Inquire object** dialog box shows the summary of the design check and related information.

See also Excel spreadsheets in connection design (page 806).

Design type tab

This design check is intended to be used with imperial units.

To check the design:

1. Go to the **Design type** tab and select **Yes** in the **Check connection** list.

   Tekla Structures checks the connection each time it is used or changed in the model.
2. Enter the information you want to use in the calculation.
3. Select the connection in the model and click **Modify**.

   Tekla Structures checks the component. A green component symbol indicates that the connection will bear the UDL, red indicates that it will not.
4. To view the results of the check, right-click the component symbol and select **Inquire** from the pop-up menu.

   The **Inquire object** dialog box shows the summary of the design check: the part checked, the name of the check, the applied and allowed force and how much capacity has been used, the results and possible solutions.

Design tab for check design only

The design is based on the British standard BS5950.

The design has the following limitations:

- Design only works in the UK environment.
- Design is available only if the main part and the secondary parts are perpendicular.
- Design is available only with two bolts horizontally.
- Design is available only when vertical bolts are defined from the top.
- Design is valid for I profiles only.

To check the design:

1. Go to the **Design** tab and select **On** in the **Design** list.
2. Enter the **Tie Force** in kilo Newtons (kN).

   Tie force is required if the design check is turned on and the framing type of the connection is beam-to-column. If there is no tie force, enter `0`.
3. Enter the **Shear Force** in kN.

   If the design check is turned on, enter a positive value. If there is no shear force, enter `0`.
4. Select the connection in the model and click **Modify**.

   The connection symbol shows the design check status:

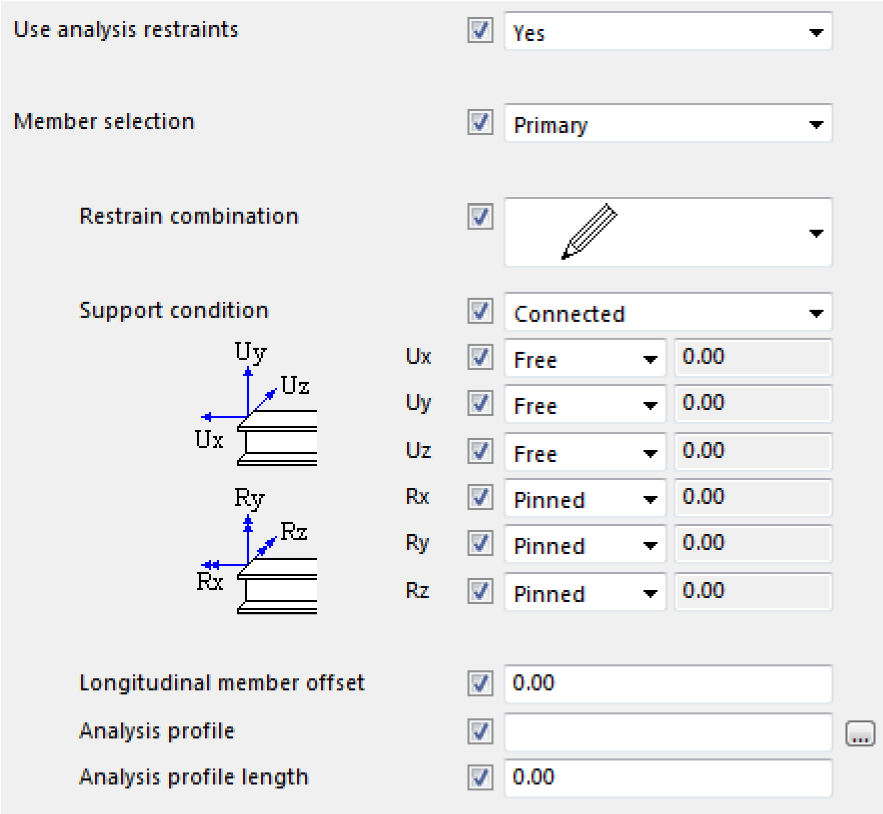
   - Green means that the design check was successful.
   - Yellow means that a warning occurred in the design check.
   - Red means that a fatal error occurred in the design check.
5. To view the results of the check, right-click the connection symbol and select **Inquire** from the pop-up menu.

   The **Inquire object** dialog box shows the summary of the design check and related information.

NOTE If the message **Numbering not up to date** is shown in the **Inquire object** dialog box, the marks will not be correct. You need to renumber the model to ensure that the marks are up-to-date. After that use the **Inquire** command again to get the correct marks to the design check summary.

Analysis tab

Use the **Analysis** tab in the steel connection or detail dialog box to define how Tekla Structures handles connections and details in the analysis.



Option	Description
Use analysis restraints	Set to **Yes** to use the analysis properties of the connection or detail in the analysis instead of the analysis properties of the parts in the connection. You also need to set **Member end release method by connection** to **Yes** in the **Analysis Model Properties** dialog box when you create the analysis model. For more information, see Analysis model properties.
Member selection	Use to associate the analysis properties with each connection part (**Primary**, **1. secondary**, **2. secondary**, and so on).
Restraint combination	For more information, see Defining support conditions.
Support condition	
Longitudinal member offset	For more information, see Analysis part properties.
Analysis profile	Tekla Structures uses this profile in the analysis instead of the one in the physical model to take

Option	Description
	the stiffness of the connection or detail into account.
Analysis profile length	In the analysis, Tekla Structures overrides the profile of the part in the physical model for this length.

8 Custom components

You can define customized connections, parts, seams, and details for your project. These are called *custom components*. You can use custom components in the same way as any Tekla Structures system component. By editing the custom components you can create intelligent, parametric custom components that automatically adjust to changes in the model.

When to use

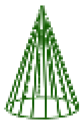
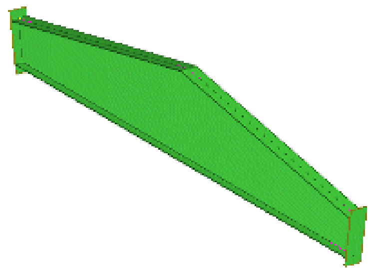
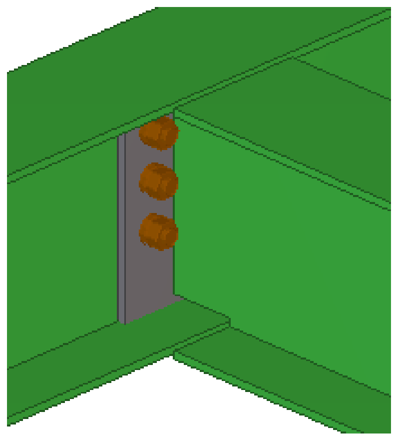
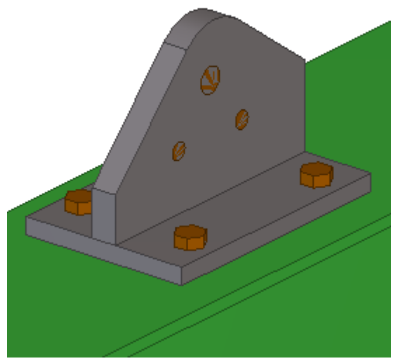
Define a custom component if you cannot find a predefined system component (page 759) that meets all your needs. Custom components are useful especially if you need to create a large number of complex model objects and copy them across several projects.

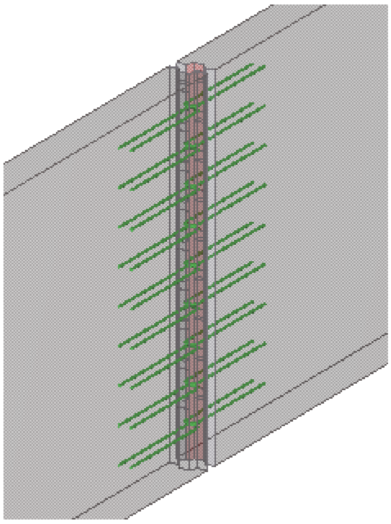
Benefits

Once you define and store a custom component in the **Applications & components** catalog, you can easily access it from the catalog and use it in another location in the same model. If you must modify the custom component, you only need to make the changes once. When you save the changes, they will be automatically applied to all copies of that custom component in the model. You can also import and export custom components as `.uel` files between models and share the custom components with your colleagues, or store the custom components in a model template (page 192) so that they are available with each new model that is based on the used template.

Custom component types

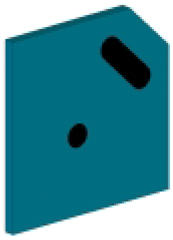
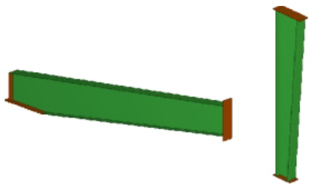
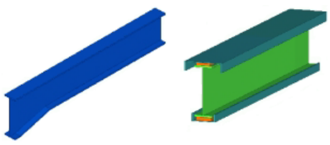
You can create four types of custom components:

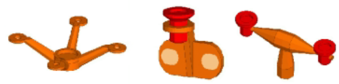
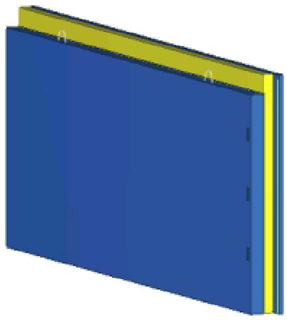
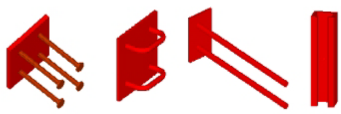
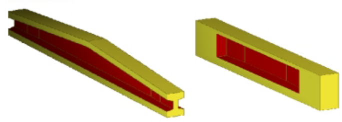
Type	Description	Example
Custom part	Creates a group of objects that may contain connections and details. **Note:** Unlike other custom components, custom parts are **not** marked with a  component symbol in the model. Custom parts have the same position properties as beams have.	
Custom connection	Creates connection objects and connects the secondary parts to the main part. The main part may be continuous at the connection point.	
Custom detail	Creates detail objects and connects them to a single part at the location you picked.	

Type	Description	Example
Custom seam	Creates seam objects and connects the parts along a line that you create by picking with two points. The parts are usually parallel.	

8.1 Examples of custom parts

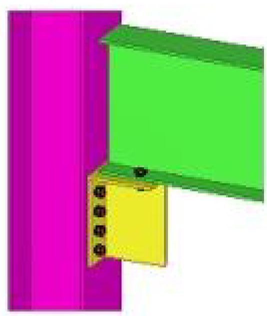
Custom parts may consists of a single part or a group of parts, and they often have a complex composition. The following images show some examples of custom parts:

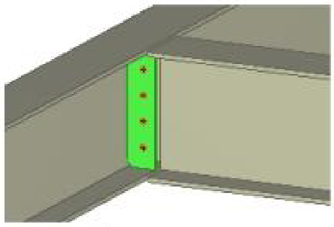
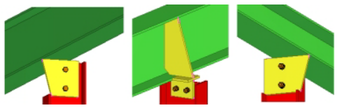
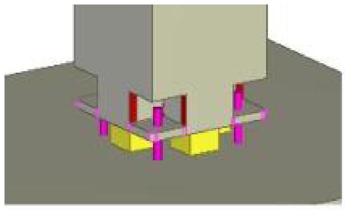
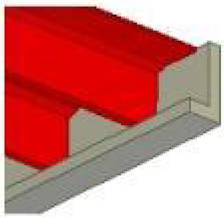
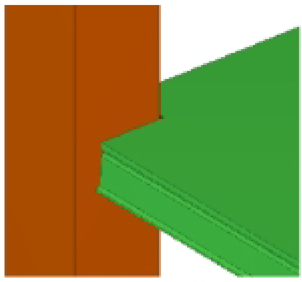
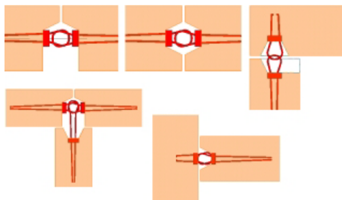
Steel	Company standard bracing plates	
	Castellated beam and cell beam	
	Built-up beams/columns	
	Built-up beams	

	Standard glazing fixings	
Precast concrete	Sandwich panel	
	Lifters	
	Standard embeds/ inserts	
	Standard beams	

8.2 Examples of custom connections

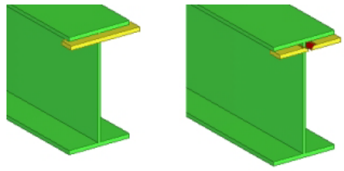
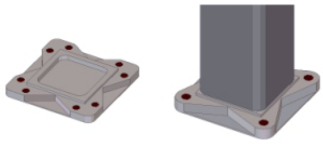
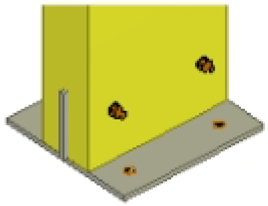
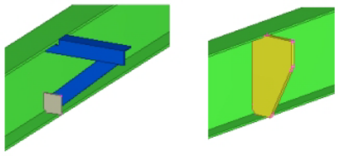
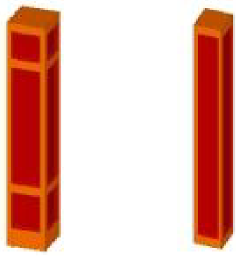
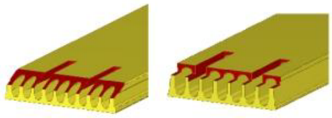
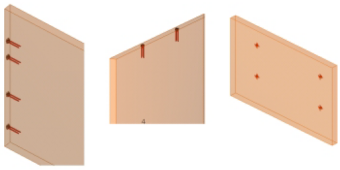
Custom connections can be used to connect a main part to up to 30 secondary parts. The connection is made between the main part and the ends of the secondary parts. The following images show some examples of custom connections:

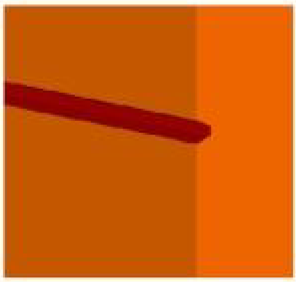
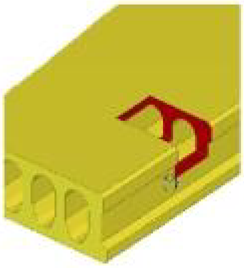
Steel	Built-up plate seat	

	Shear plate	
	Typical japanese post connections	
Precast concrete	Base detail	
	Double tee to L profile	
	Column cut-out	
	Wall panel connections	

8.3 Examples of custom details

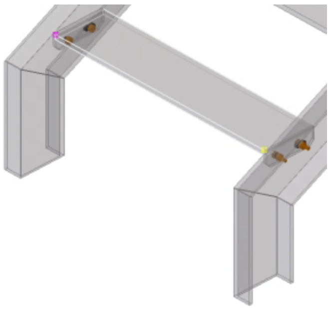
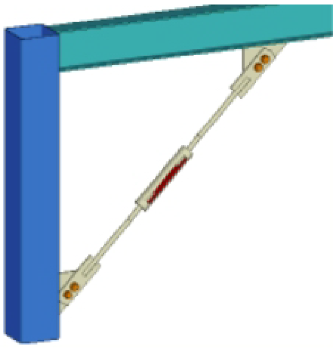
Custom details can be used to add more information to a single part, such as extra plates or cut-outs. The following images show some examples of custom details:

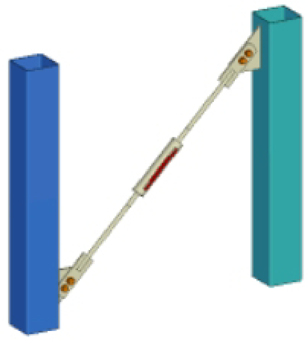
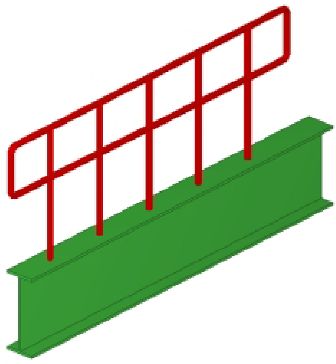
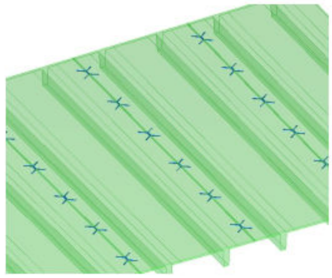
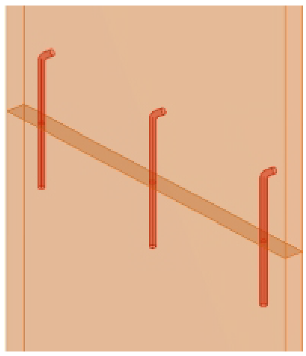
Steel	Backing plates	
	Cast base	
	Timber base	
	Out rigger (stiffeners) and out rigger plate (stiffeners)	
Precast concrete	Door and window	
	Column patterns	
	Hollow core end details	
	Lifting details	

	Mock joint/reveal	
	Side pocket	

8.4 Examples of custom seams

Custom seams can be used to connect a main part to up to 30 secondary parts. They can also be used on one main part only. The seam is made along the length of the part. The following images show some examples of custom seams:

Steel	Steel stair step	
	Turnbuckles	

		
	Handrail	
Precast concrete	Double tee connection	
	Panel to panel grout tube connection	

8.5 Define custom components

You can define customized components that have all the details you need.

Start by defining a simple custom component which you can modify later. Defining a simple custom component typically takes only few minutes. You can invest more time in defining your custom components if you are planning to use them in future projects.

By editing the custom components (page 840) even further you can define self-adjusting, parametric custom components (page 869) that automatically adjust to changes in the model. This is more time-consuming but can pay off later when you have a group of parametric custom components which you can use across several models or projects.

Explode an existing component

When you start defining a custom component, we recommend that you first apply a similar system component in the model and then explode it. Exploding means that you ungroup the objects of an existing component. Once the objects are detached, you can modify, remove or add objects to suit your needs, and then create new custom components using these objects. Exploding a component and using the detached objects as a basis for a new custom component can be useful when you want to create custom components more quickly.

Alternatively, you can create individually the component objects that are needed in a custom component, such as parts, cuts, fittings, and bolts.

1. In the model, select the component you want to explode.
2. Right-click and select **Explode Component**.

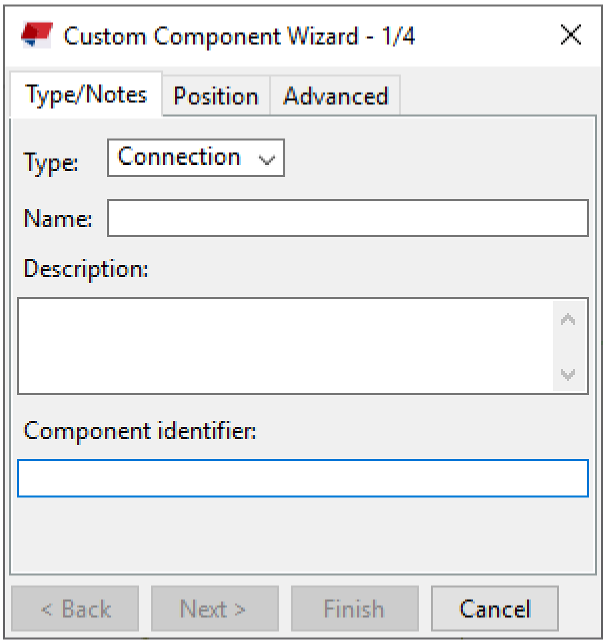
   Tekla Structures separates the component objects. You can modify the objects and use them when you define new custom components.

Define a custom component

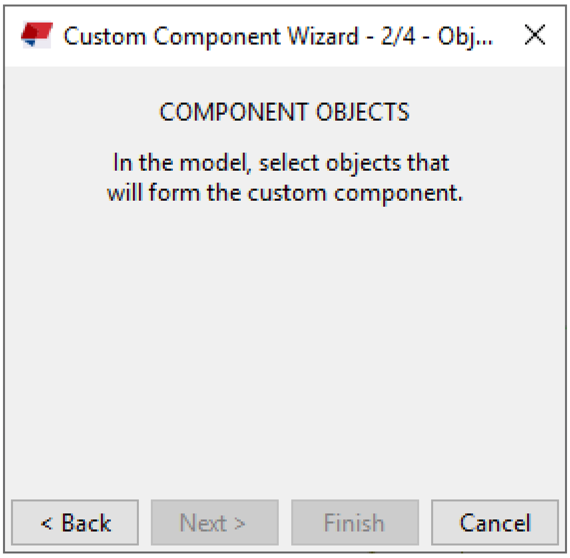
The following example shows how to define a simple custom connection.

1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
2. Click the **Access advanced features** button and select **Define custom component...**.

The **Custom Component Wizard** dialog box opens.



3. In the **Type** list, select the component type (page 823): connection, detail, seam, or part.
4. In the **Name** box, enter a unique name for the component.
5. Modify the other properties (page 953) on the **Type/Notes** tab, **Position** tab, and **Advanced** tab, and then click **Next >**.
6. In the model, select the objects you want to include in the custom component.



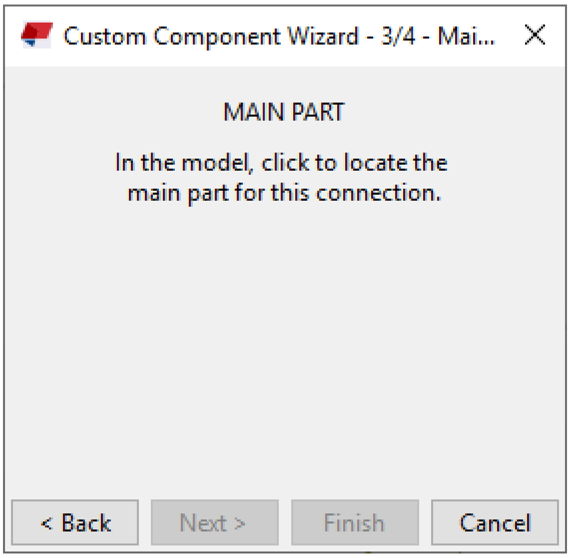
You can use area selection to select multiple objects simultaneously (page 113). The main and secondary parts and the grids are ignored when you select objects for the custom component.

NOTE If you cannot select desired objects in the model, check the selection switches and the selection filter settings (page 153).

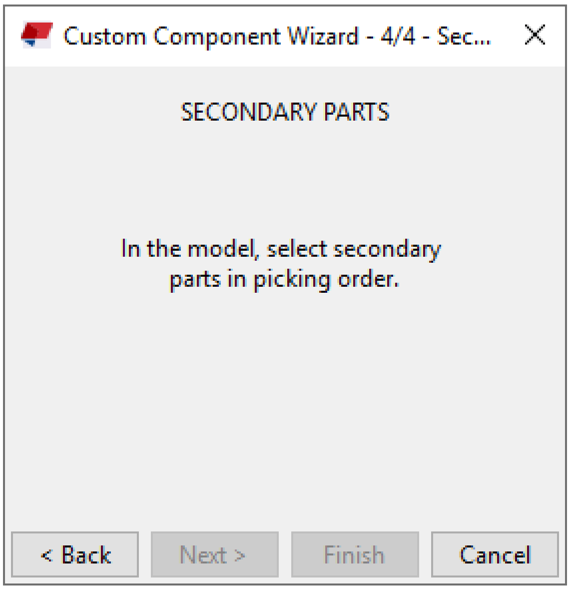
If you want to include rebar set modifiers (page 533) in the custom component:

- Ensure that **Direct modification** is switched off when you select the modifiers.
- Hold down **Shift** and select the modifiers one by one. Area selection does not select modifiers.

7. Click **Next >**.
8. Select the main part for the component.



9. Click **Next >**.
10. Select the secondary parts for the component.



To select multiple secondary parts, hold down **Shift** when you select them. The maximum number of secondary parts in a custom component is 30.

NOTE Pay attention to the order in which you select secondary parts. Tekla Structures will use the same picking order when you use the custom component in a model.

---

11. Define any other properties required for this custom component, such as detail or seam position.

    The properties depend on the component type that you selected in step 4.

12. If you want to adjust any of the settings at this stage, click **< Back** to return to the previous page of the **Custom Component Wizard**.

13. When you are happy with the settings, click **Finish** to create the custom component.

    The custom component is added in the model and in the **Applications & components** catalog.

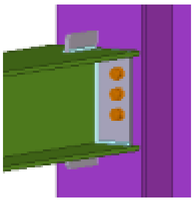
14. If needed, add a thumbnail image for the custom component.

    The thumbnail image is displayed in the **Applications & components** catalog. In the thumbnail image you can show a typical situation where the component can be used.

    a. Take a screenshot of the custom component.

    b. Add a thumbnail image (page 769) of the custom component in the **Applications & components** catalog.

Tekla Structures shows the thumbnail image in the **Applications & components** catalog:



End plate with haunches

15. If you want to modify custom component settings afterwards:
    a. On the custom component editor toolbar (page 840), click the **Modify custom component settings** button.
    b. Change the settings.
    c. Click **OK**.

The defined custom component is not intelligent and Tekla Structures does not adjust dimensions to suit any changes in the model. To have the component adapt to changes in the model, edit (page 840) the component in the custom component editor, where you can build dependencies between component objects and model objects.

Define a nested custom component

You can define more complex custom components by joining two or more components together as a nested component. This allows you to create smaller, simpler components and wrap them up into a single component. The original components become sub-components in the nested component. Nested components are typically used with precast and cast-in-place components, such as with embeds.

Do not nest components more than necessary. If you nest more than two levels, you might run across some limitations. To view the different component levels in a nested custom component (page 121), hold down the **Shift** key and scroll with the mouse wheel.

1. In the model, create the components and other model objects that you want to include in the nested component.

2. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.

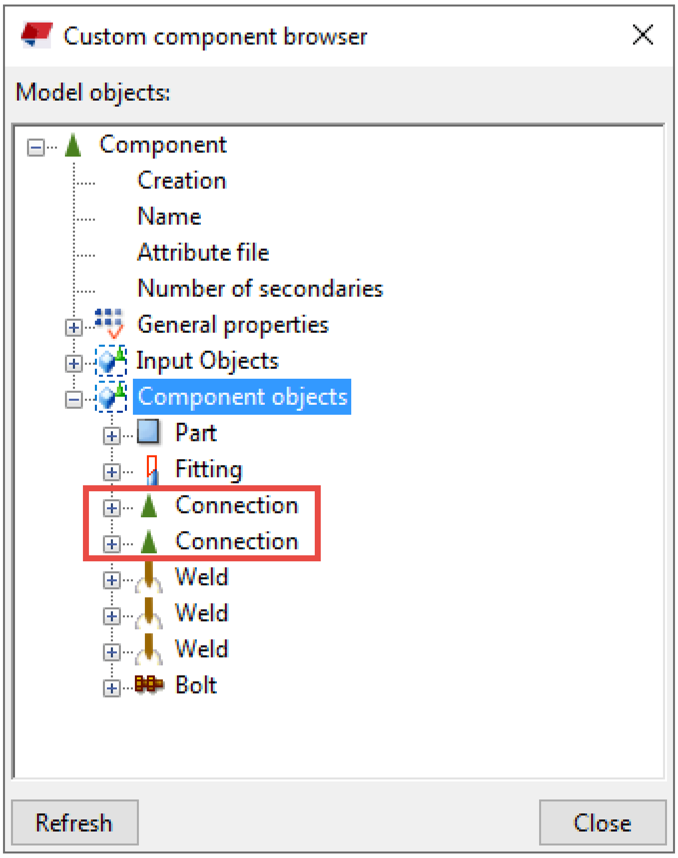
3. Click the **Access advanced features** button and select **Define custom component...**.

   The **Custom Component Wizard** dialog box opens.

4. In the **Type** list, select the type of the nested custom component.
5. In the **Name** box, enter a unique name for the nested component.
6. Modify the other properties on the **Type/Notes** tab, **Position** tab, and **Advanced** tab, and then click **Next >**.
7. Select the components and any other objects you want to include in the nested component, and then click **Next >**.
8. Follow the instructions in the **Custom Component Wizard** to continue.

   You will be asked to select the main and secondary parts for the nested component. Depending on the component type that you selected in step 4, you may also have to define other properties, such as detail or seam position.
9. When you are happy with the settings, click **Finish** to create the nested component.

   The component is added in the model and in the **Applications & components** catalog. The sub-components are shown in the **Custom component browser**, together with the other component objects:



10. If you want to change the settings afterwards:

a. In the custom component editor (page 840), click the **Modify custom component settings** button .

b. Change the settings.

c. Click **OK**.

WARNING If you use a component of the type plug-in as a sub-component of a nested component, and change the sub-component's properties in the custom component editor, note that those changes may be lost when you save the nested component and use it in a model.

To prevent losing any properties, link a variable to each plug-in property that you want to keep. You can also use component attribute files to do this. For more information, see Examples of parametric variables and variable formulas in custom components (page 884).

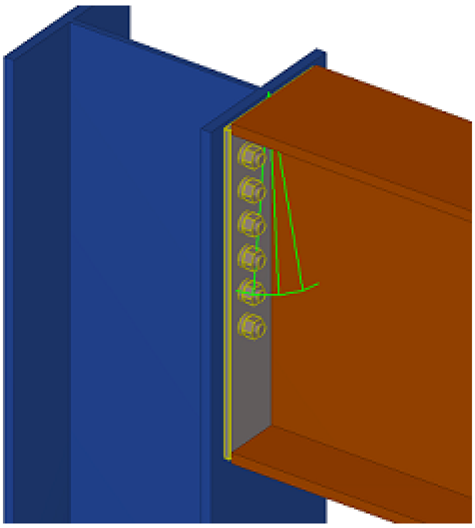
Example: Define a custom end plate component

This example shows how to define a simple custom component based on an existing end plate component.

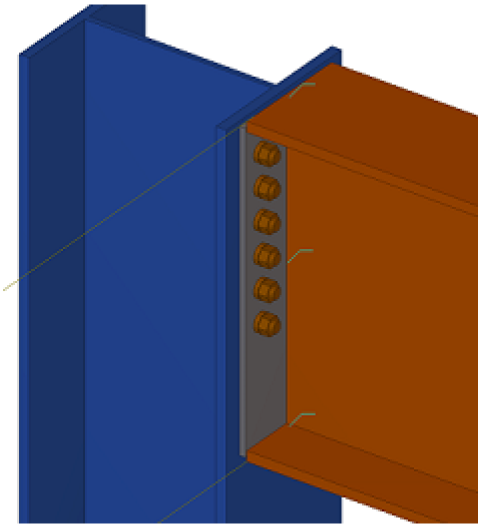
1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.

2. Click the **Access advanced features** button and select **Explode component**.

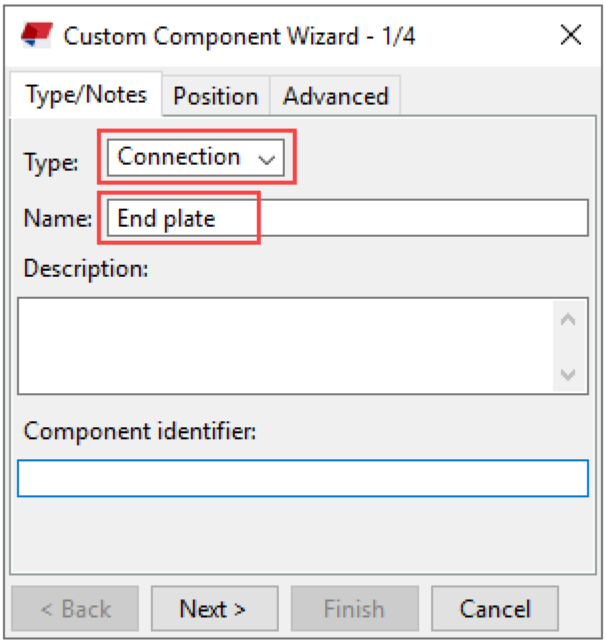
3. Select the end plate component in the model.



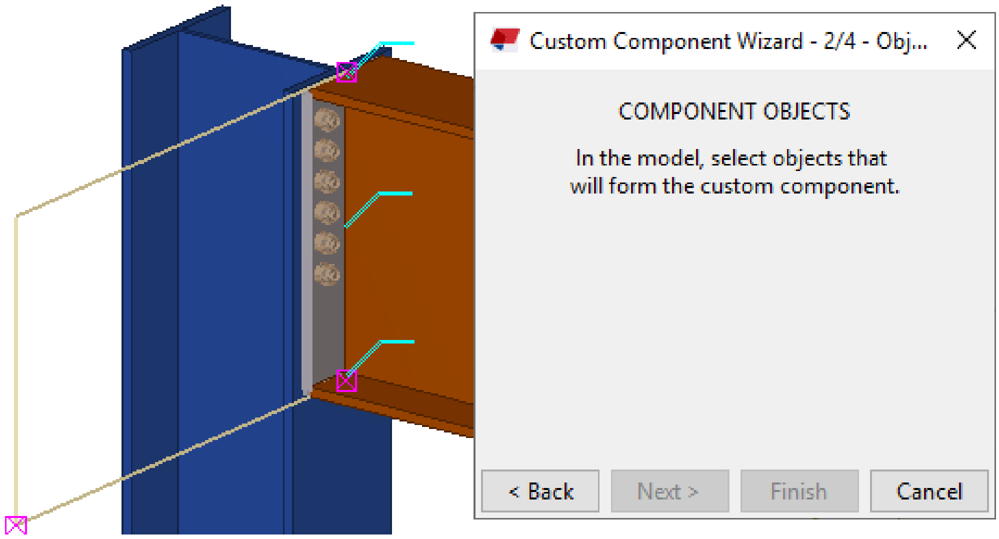
Tekla Structures separates the objects in the component.



4. Click the **Access advanced features** button and select **Define custom component...**.
5. In the **Type** list, select **Connection**.
6. In the **Name** box, enter a unique name for the custom component.



7. Click **Next >**.
8. Select the objects you want to use in the custom component, and then click **Next >**.



You can use area selection (left to right) to select the objects. Tekla Structures ignores the main part and secondary parts and the grids when you are selecting objects to include in the custom component.

9. Select the column as the main part, and then click **Next >**.

   The main part supports the secondary part.

10. Select the beam as the secondary part.

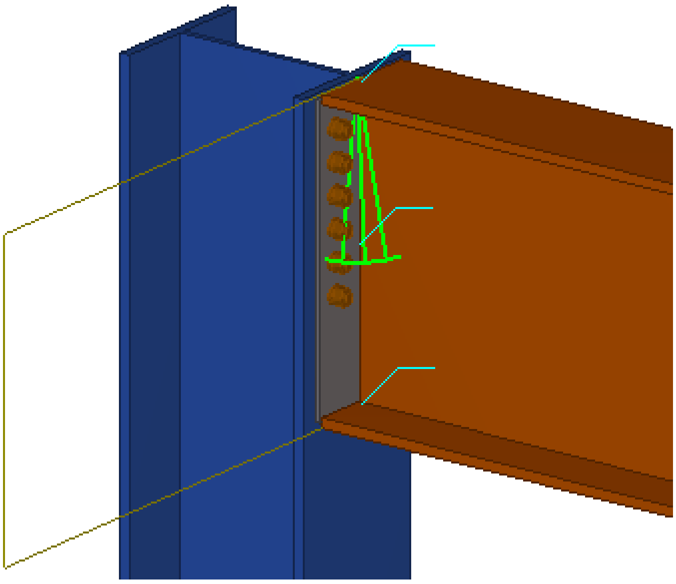
    The secondary part is supported by the main part.

NOTE When you select multiple secondary parts, pay attention to the order of selection. The custom component will use the same selection order when you add the component in a model. The maximum number of secondary parts in a custom component is 30.

---

11. Click **Finish**.

Tekla Structures displays a component symbol for the new component.

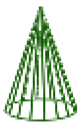


You have now defined a simple custom component, which you can use in locations similar to where it was originally created. This component is not intelligent and Tekla Structures does not adjust dimensions to suit any changes in the model. To make the custom component intelligent, you need to edit (page 840) it in the custom component editor.

8.6 Edit and save custom components

Use the custom component editor to fine-tune existing custom components, and to make the components parametric. When you edit a custom component, Tekla Structures updates all instances of that component throughout the model with the changes you have made.

Edit a custom component

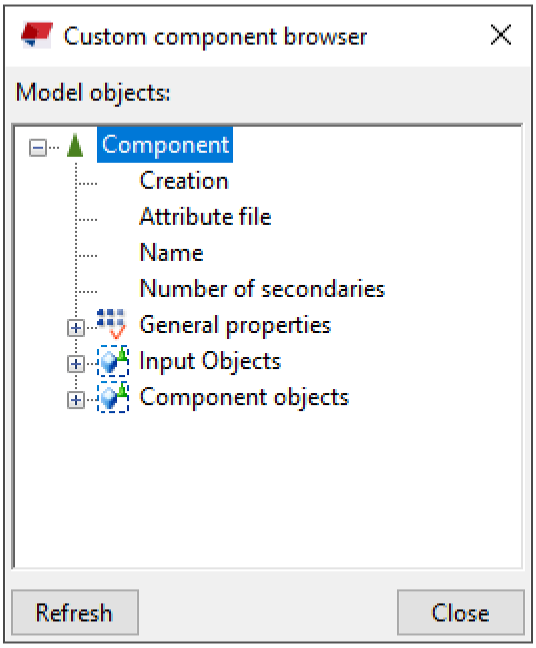
1. In the model, select the custom component by clicking the green component symbol .

NOTE Custom parts do not have a component symbol in the model. To select custom parts, ensure that the **Select components**  selection switch is active.

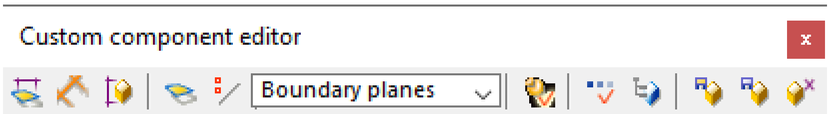
2. Right-click and select **Edit Custom Component**.

   The custom component editor opens. It consists of the following parts:

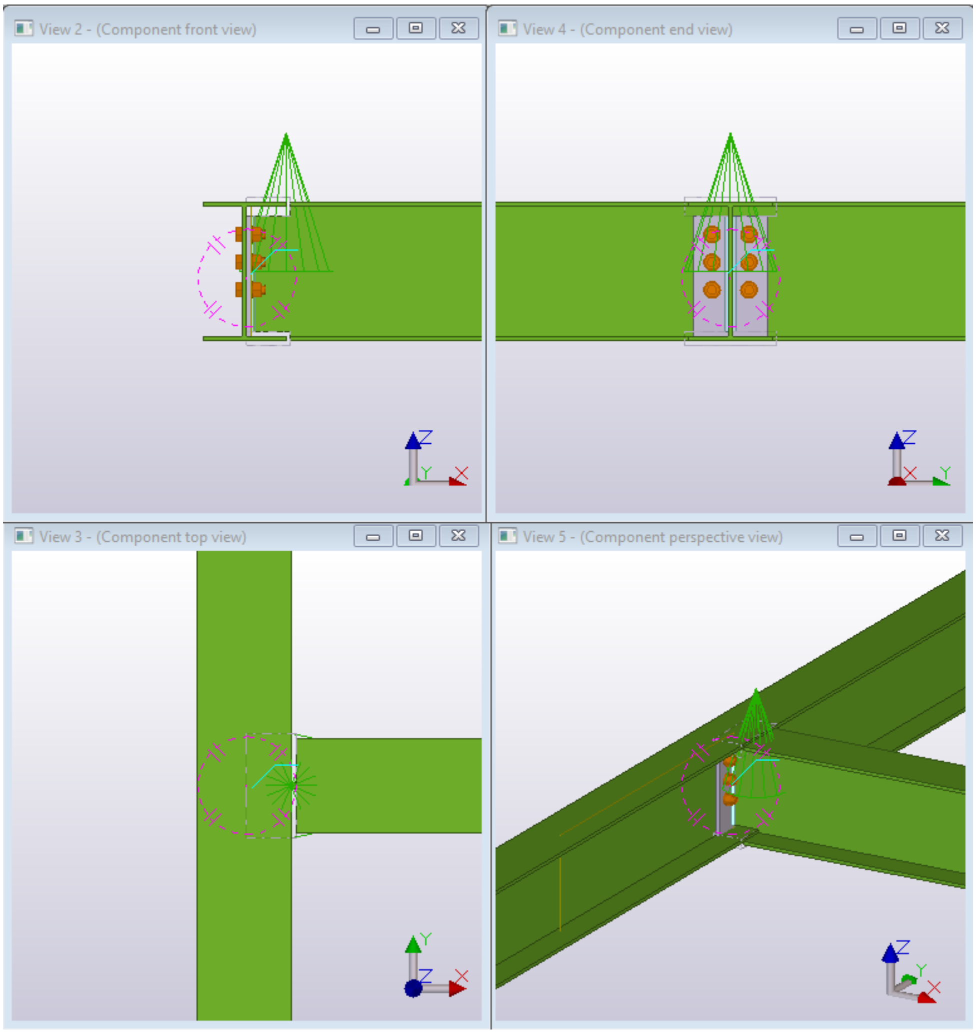
   • The **Custom component browser**



   • The **Custom component editor** toolbar



- Four different **views** of the custom component



3. Modify the custom component in one of the four custom component views. You can, for example:

   - Add or remove component objects

     For example, add extra bolts or stiffeners to the component. Only component objects, not the main or secondary parts, can be modified in the custom component editor.

   - Bind component objects to a plane (page 850)
   - Add a distance between component objects (page 862)
   - Set object properties by using parametric variables (page 864)

4. Save the custom component.

   Click **Yes** when prompted to replace all occurrences of the custom component in the model. All instances of the custom component are now updated with the changes you made.

Custom component browser

Custom component browser shows the contents of a custom component in a hierarchical, tree-like structure. **Custom component browser** lists the model objects, to which custom component is attached, and the objects the custom component creates. You can create links between custom component variables and component object properties.

The **Custom component browser** works with the views. When you select a part in the view, Tekla Structures highlights it in the **Custom component browser**, and vice versa.

You can copy names, values, and references (page 868) from main and secondary parts in the component in the **Custom component browser**, and then use them in the **Variables** dialog box to define the properties of custom components.

Custom component editor toolbar

Use the tools on the **Custom component editor** toolbar, for example, to create distances, select planes, and to save the component.

Icon	Description
	Creates a distance. Select the plane first and then the handle or chamfer where to bind.
	Creates a reference distance.
	Creates distances automatically. Tekla Structures binds the selected part to their handles' contact planes (page 960). Tekla Structures selects planes in the following order: 1. construction planes 2. planes of the main and the secondary parts
	Creates a construction plane (page 859).
	Creates a construction line (page 859).
Plane types	Shows plane types you can use when defining distance variables.
	Edits the type or the position of a custom component (page 953), or add notes after you have created the component.

Icon	Description
	Shows all created variables (page 849).
	Opens **Custom component browser**.
	Saves the custom component with another name.
	Saves and update the existing components in the model.
	Closes the editor.

Save a custom component

After modifying a custom component, save the changes.

Tekla Structures saves the custom component in the current model folder, in the `xslib.db1` file, which is a library file containing custom components and sketches.

To	Do this
Save changes to all copies of the custom component	1. In the custom component editor, click the **Save component** button. 2. In the **Save confirmation** dialog box, click **Yes**. Tekla Structures saves the changes and applies them to all copies of the custom component in the model.
Save the component with a new name	1. In the custom component editor, click the **Save with new name** button. 2. Enter a new name for the component.
Save and close the component	1. In the custom component editor, click the **Close** button. 2. In the **Close custom component editor** message, click **Yes**. If you click **No**, the custom component editor closes without saving the changes.

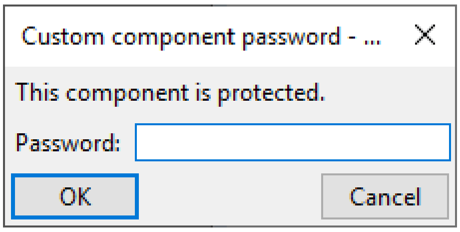
Protect a custom component with a password

You can set a password to prevent others from modifying a custom component. Password-protected custom components can still be added to models as usual.

1. In the model, select a custom component.
2. Right-click the custom component symbol and select **Edit Custom Component**.
3. On the **Custom component editor** toolbar, click the **Display variables**  button.

   The **Variables** dialog box opens.
4. Click **Add** to create a new parametric variable.
5. In the **Name** box, enter `Password`.
6. In the **Formula** box, enter the desired password.
7. Save the custom component.

The next time someone tries to edit this custom component, they will be asked for the password.



NOTE If you have a password protected custom component and you explode the custom component in the **Custom component editor**, you can either use the **Explode Component** or **Explode component with parameters** command.

The **Explode Component** command explodes the custom component without creating the component parameters and mappings.

The **Explode component with parameters** command will ask you to enter the password. Only after entering the correct password will the component be exploded, and the component parameters and mappings created.

8.7 Add custom components to a model

Use the **Applications & components** catalog to add your custom component to a model.

Add a custom connection, detail, or seam to a model

1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
2. To search for a component, browse the catalog or enter a search term in the search box.

   Custom components have the following symbols in the catalog:

Type	Symbol
Custom part	
Custom connection or seam	
Custom detail	

3. Select the custom component you want to add.
4. Follow the instructions on the status bar to add the custom component in the model.
5. To modify the properties, double-click the custom component in the model.

Example: Add a custom connection to a model

This example shows how to add a previously created custom end plate connection (page 837) to a model. Because you have not modified the custom component to adapt to different situations in the model, you need to add it to a similar location where it was created. Otherwise the custom component may not work as required.

1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
2. In the catalog, select the custom end plate connection you want to add.

   Tekla Structures displays instructions on the status bar.
3. Select the column as the main part.
4. Select the beam as the secondary part.

   Tekla Structures adds the end plate connection to the model.

Add or move a custom part in the model

Use the direct modification handles and dimensions when you add or move custom parts. If you are unable to select custom parts in the model, ensure that the **Select components** selection switch is active.

NOTE This method cannot be used when adding custom parts to faces that have cuts or edge chamfers. You need to hide the cutting parts and edge chamfer objects from the view before you add custom parts on cut or chamfered faces using direct modification.

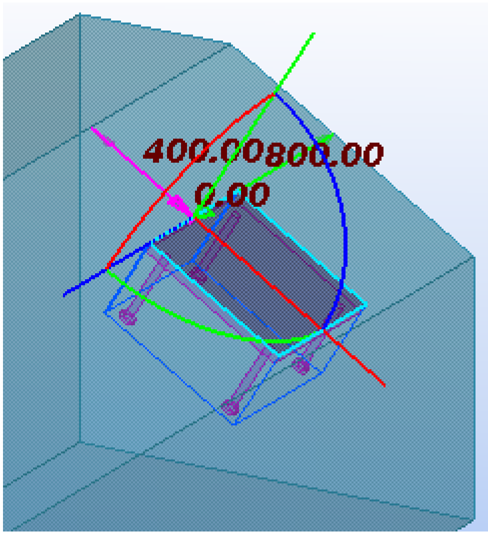
We do not recommend using this method with custom parts that are parametric, and in which the input points define the dimensions of the custom part. The preview is simplified, based on the default custom part dimensions, and snapping has a different focus than usually.

1. Ensure that **Direct modification** is switched on.
2. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
3. In the catalog, select the custom part you want to add.
4. Move the mouse pointer over part faces and edges in the model, and see how the custom part turns over and adjusts to the part faces.

   If you are adding the custom part to another object, Tekla Structures shows location dimensions to the nearest object edges.

   If you are adding a custom part that has only one input point, press **Tab** to rotate it in 90-degree steps around the work plane Y axis.
5. Depending on the number of custom part input points, pick one or two points to place the custom part in the model.

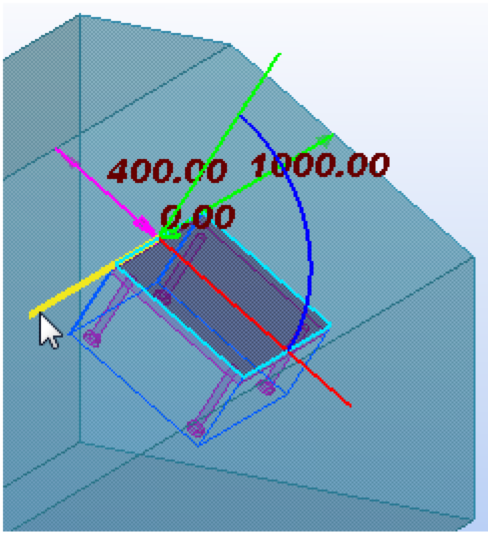
   Tekla Structures shows the coordinate axes, rotation handles, and location dimensions that you can use to fine-tune the location and rotation of the custom part. The handles are red, green, and blue, according to the local coordinate system of the custom part.



6. Click the middle mouse button to confirm the location and rotation.

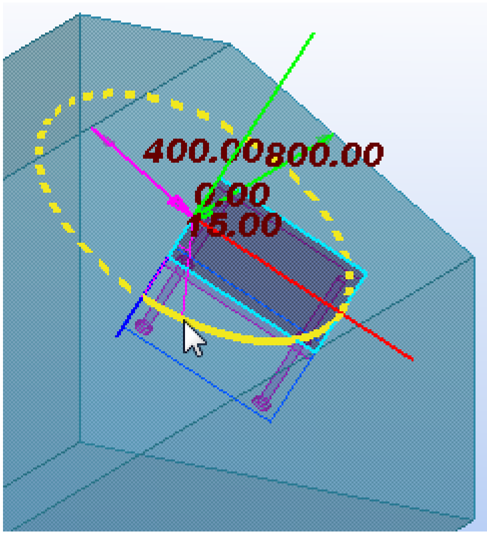
   Tekla Structures adds the custom part to the model.

7. To move the custom part along any of its coordinate axes, drag the relevant axis handle to a new location.



8. To rotate the custom part around any of its coordinate axes, drag the relevant rotation handle to a new location.

Press **Tab** to rotate the custom part in 90-degree steps in the direction of the selected rotation handle.



9. To move or rotate the custom part by specifying a distance or angle:

   a. Select an axis handle, a rotation handle, or a dimension arrowhead.

   b. Type the value by which you want the dimension to change.

      When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog box.

   c. Click **OK** to confirm the new dimension.

10. To stop modifying, press **Esc**.

8.8 Add variables to a custom component

Variables are the properties of a custom component. You can create variables in the custom component editor and use them to adapt custom components to changes in the model. Some of the variables appear in the custom component's dialog box, while others are hidden and are only used in calculations.

Variable types

There are two types of variables:

- **Distance variable**: The distance between two planes, or between a point and a plane. A distance variable binds parts together, or works as a reference distance.
- **Parametric variable**: Controls all the other properties in a custom component, such as name, material grade, and bolt size. Parametric variables are also used in calculations.

Distance variables

Use distance variables to bind custom component objects to a plane (page 850), so that the component objects stay at a fixed distance even if the surrounding objects change. You can create distance variables manually or automatically.

You can bind the following objects to a plane:

- construction planes
- reference points of parts (only custom component objects)
- reference points of bolt groups
- chamfers
- part and polygon cut handles
- line cuts
- reference points of reinforcing bars
- reference points of reinforcement meshes and strands
- fittings

You can decide which distance variables are shown in the custom component's dialog box. Show the variables if you want to edit their values in the dialog box. Hide the variables if you only use them to bind objects to a plane.

Parametric variables

Use parametric variables to set properties for any object the custom component creates (page 864). After creating the variable, you will be able to change the value directly in the custom component's dialog box.

You can also create formulas (page 884) to calculate values. For example, you can calculate the position of a stiffener relative to the beam length.

You can decide which parametric variables are shown in the custom component's dialog box. Show the variables if you want to edit their values in the dialog box. Hide the variables if you only use them in calculations.

NOTE There are some limitations concerning the variable names.

- To be able to correctly reference a variable in your formula, the variable name must be 19 characters or shorter. Variables with longer names will not work correctly when referenced.
- Variable names cannot contain mathematical operators (`+`, `-`, `*`, `/`).
- You cannot use a mathematical constant, such as `PI` or `e`, as a variable name.

Bind component objects to a plane

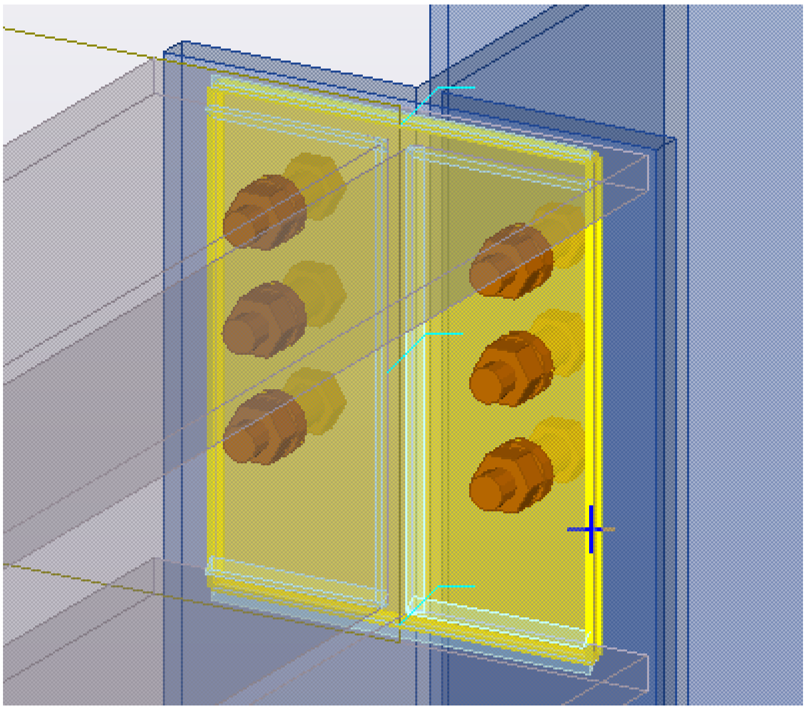
Use *distance variables* to bind component objects to a plane. Binding keeps the custom component at a fixed distance from the plane even if the surrounding objects change. Distance variables automatically get the prefix **D** (distance), which is shown in the **Variables** dialog box.

Bind objects automatically

You can bind objects automatically to the main and secondary parts of a connection or detail. The selected objects, or their handles, are bound to existing planes if the objects (or handles) are located exactly on the plane.

NOTE You cannot bind custom parts (page 825) automatically, because they do not have a main part.

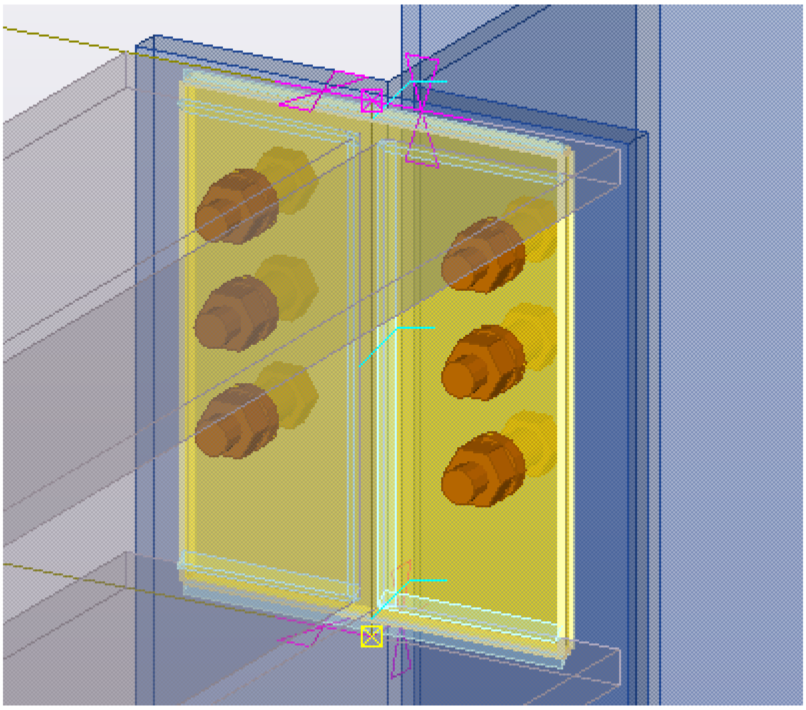
1. In the custom component editor, click the **Automatically create distances for picked components' handle points**  button.
2. Select an object that has handles (page 307).



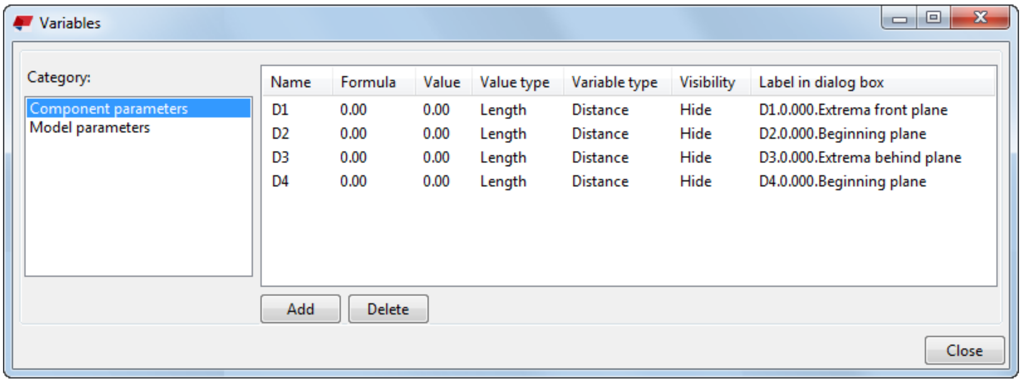
3. Click the middle mouse button to bind the object.

   Tekla Structures binds the object from a maximum of three directions to the existing planes.

   Tekla Structures displays a distance symbol for each binding. Select the object to see the bindings.



The corresponding distance variables are shown in the Variables (page 964) dialog box:



Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
D1	0.00	0.00	Length	Distance	Hide	D1.0.000.Extrema front plane
D2	0.00	0.00	Length	Distance	Hide	D2.0.000.Beginning plane
D3	0.00	0.00	Length	Distance	Hide	D3.0.000.Extrema behind plane
D4	0.00	0.00	Length	Distance	Hide	D4.0.000.Beginning plane

Bind objects manually

Create the bindings manually if you want to bind a custom component from specific handles only. You can bind an object to a maximum of three planes.

1. Ensure that **Direct modification**  is switched off.

   The selection of handles is easier when **Direct modification** is off.

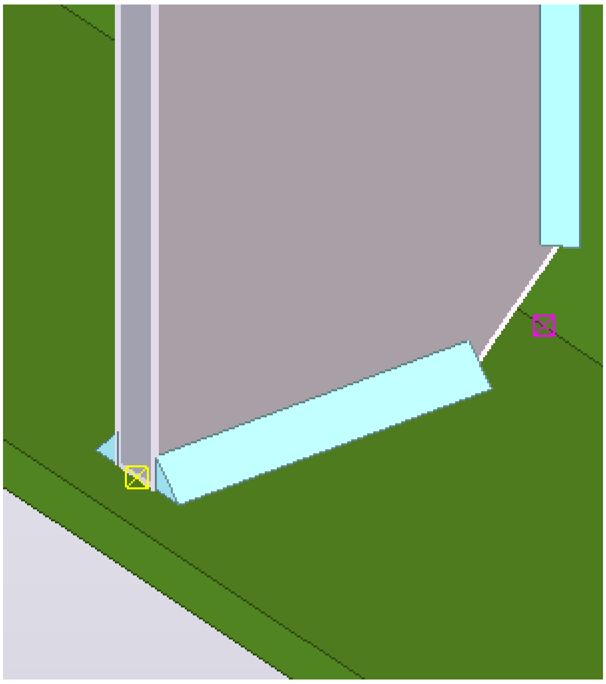
2. Ensure that you are using a model view that shows object faces.

   On the **View** tab, click **Rendering**, and use one of the following options:

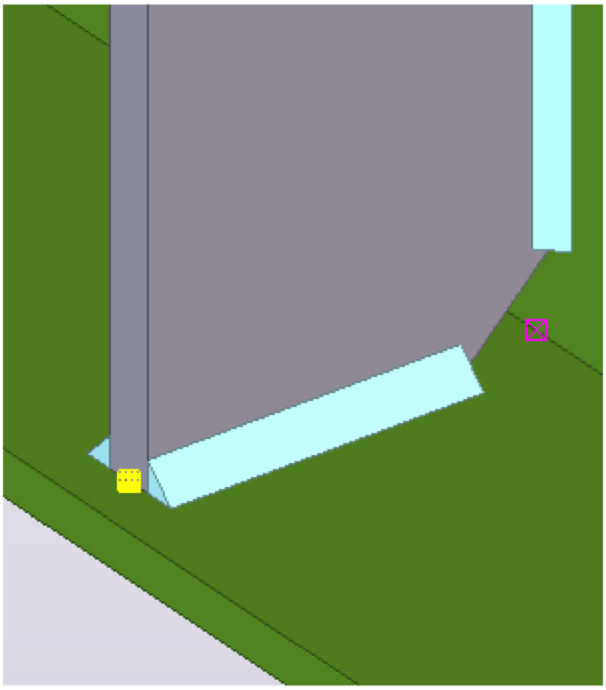
   - **Parts grayscale** (Ctrl+3)

- **Parts rendered** (Ctrl+4)

3. In a custom component view, select the custom component to see its handles (page 307).



4. Select the handle that you want to bind to a plane.

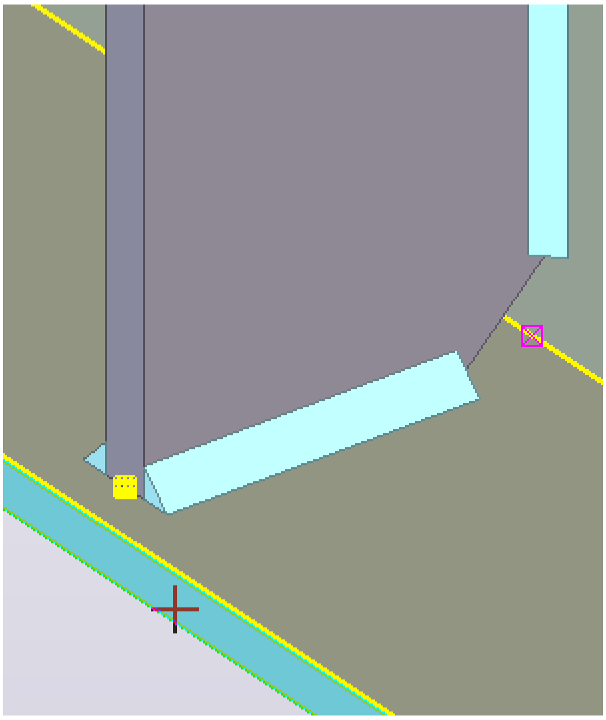


5. In the custom component editor, click the **Create distance** button.

   You can also right-click and select **Bind to plane**.

6. Move the mouse pointer in a custom component view to highlight the plane that you want to bind with the handles.

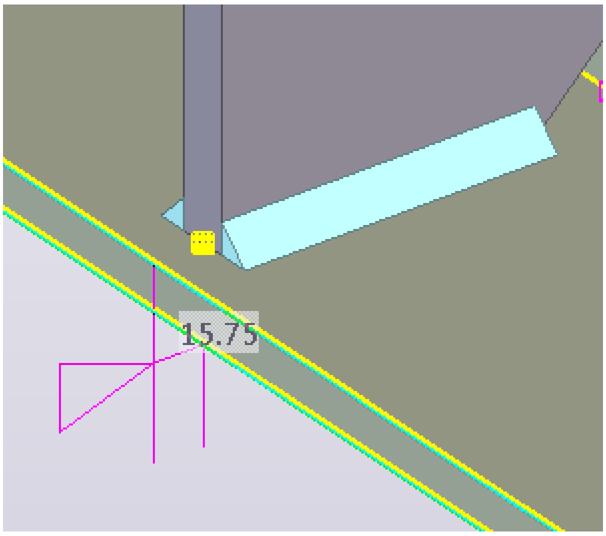
   For example:

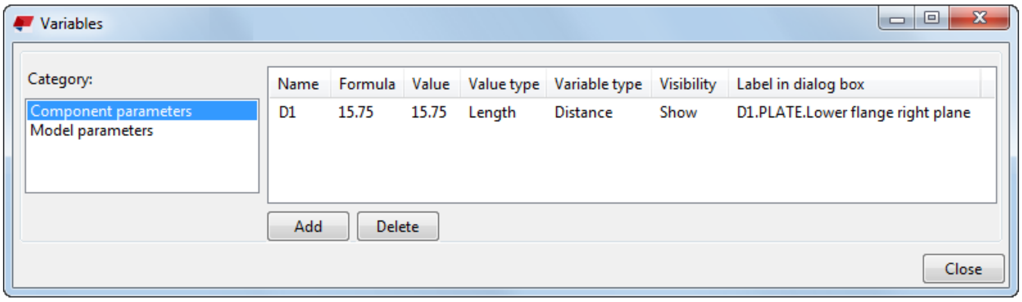
NOTE	If you cannot highlight the correct plane, change the plane type (page 960) on the **Custom component editor** toolbar. Boundary and component planes work for most profile types, so try to use them whenever you can.

7. Click the plane to create the binding.

Tekla Structures displays a distance symbol for the binding.



The corresponding distance variable is shown in the **Variables** dialog box:



NOTE If you have created a nested custom component and have used a component of the type plug-in as a sub-component of a nested component, or another custom component as a sub-component of a nested component,

the bindings may be lost or do not work as desired when you save the nested component and use it in a model.

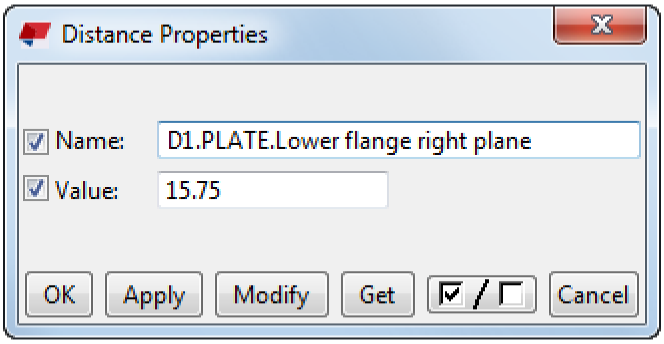
Test a binding

Test all bindings to see that they work correctly.

To be able to select distances in the model, ensure that the **Select distances**  selection switch active.

1. Double-click the distance symbol in a custom component view.

The **Distance Properties** dialog box opens.



2. In the **Value** box, enter a new value.
3. Click **Modify**.

   You should see the binding change in the model.

TIP Alternatively, you can test the binding in the Variables (page 964) dialog box:

a. Enter a new value in the **Formula** box.

b. Press **Enter**.

   You should see the binding change in the model.

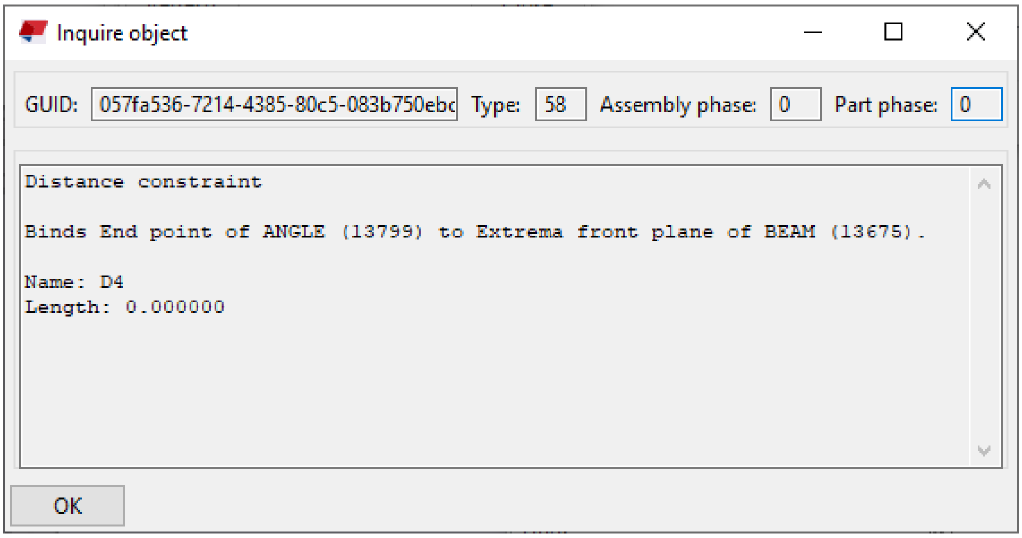
Check a binding

You can check what is bound to what by using the **Inquire objects** command.

To be able to select distances in the model, ensure that the **Select distances**  selection switch active.

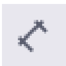
1. On the ribbon, click  **Object**.
2. Select a distance symbol in a custom component view.

The **Inquire object** dialog box shows information about the binding.



Delete a binding

Bindings cannot be modified, but you can delete the existing bindings and then create new ones to rebind the objects.

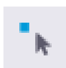
To be able to select distances in the model, ensure that the **Select distances**  selection switch active.

1. Select the binding in a custom component view.
2. Press **Delete**.

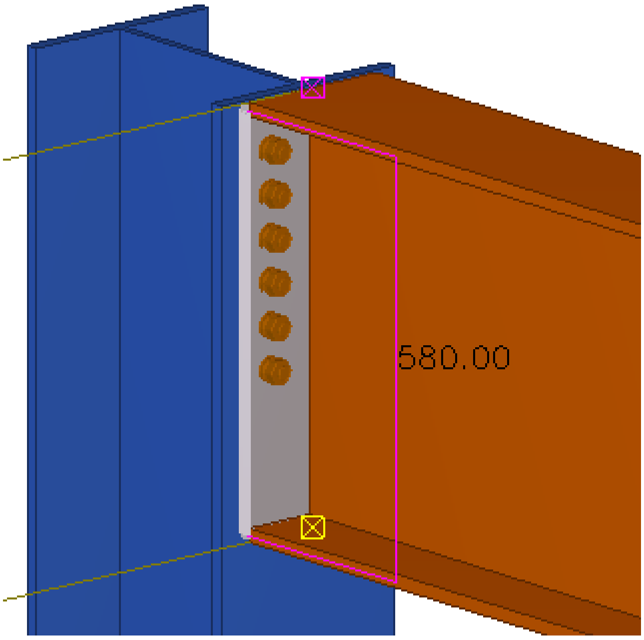
   You can also select the binding in the Variables (page 964) dialog box and then click the **Delete** button.

Example: Bind an end plate to a plane

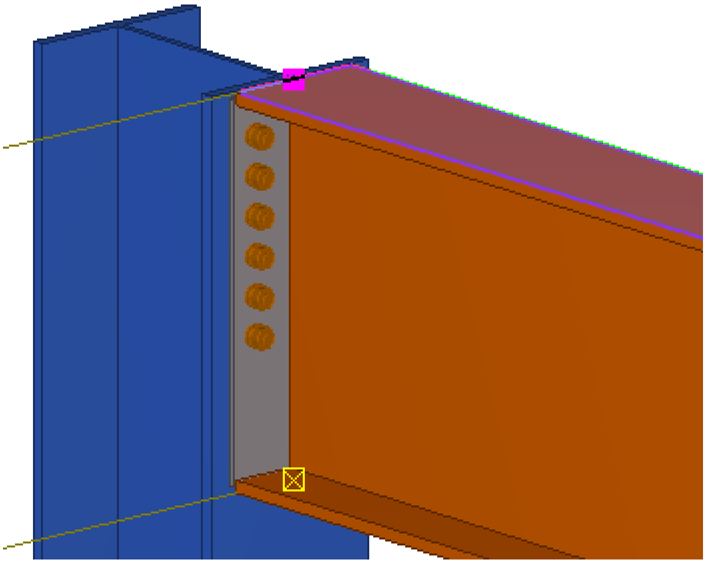
This example shows how to bind the end plate top to the upper side of the beam.

1. Ensure that **Direct modification**  is switched off.

   The selection of end plate handles is easier when **Direct modification** is off.
2. In a custom component view, select the end plate to see the end plate handles.



3. Select the top handle of the end plate.
4. Right-click the top handle and select **Bind to plane**.
5. Move the pointer over the upper side of the beam flange to highlight it.

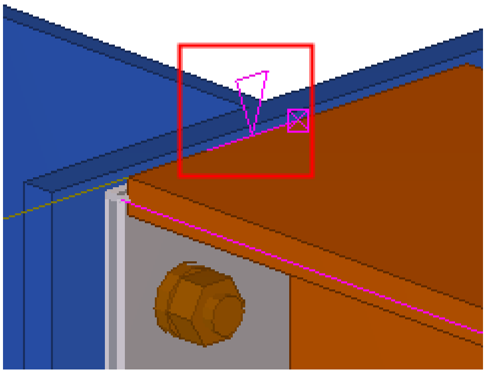


Here we are using the boundary plane type. If the part profile changes, the boundary plane is always found.

NOTE If you cannot highlight the desired plane, change the plane type (page 960) on the **Custom component editor** toolbar.

6. Click the upper side of the beam flange.

A distance symbol appears in the custom component views.



7. Give a descriptive name for the binding you created:

   a. In the custom component editor, click the **Display variables** button .

      The **Variables** dialog box opens.

   b. In the **Label in dialog box** box, enter `Plate Top to Flange Top` as the name of the new binding.

Bind component objects using magnetic construction planes or lines

Instead of binding each component object handle to a plane separately, you can use magnetic construction planes and lines. The objects that are directly on a magnetic construction plane (or line) will move with the plane (or line), which means you only need to create one distance variable instead of 8, for example.

Bind handles using a magnetic construction plane

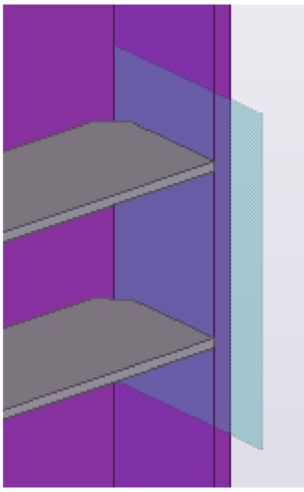
1. In the custom component editor, click the **Create construction plane** button .

2. Pick four points to define the shape of the construction plane.

   For example, create a plane that goes through all the handles and chamfers of the custom component.

3. Click the middle mouse button.

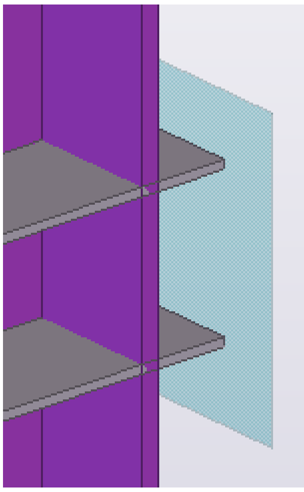
Tekla Structures creates a construction plane. For example:



4. Double-click the plane.

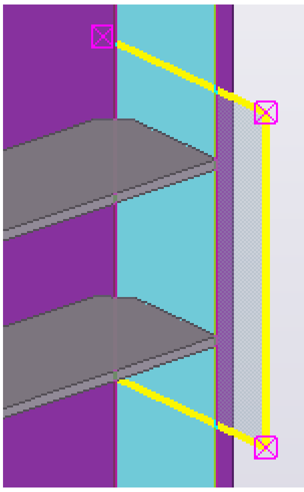
   The plane properties are shown in the property pane.
5. Enter a name for the plane.
6. In the **Magnetic** list, select **Yes**.
7. Click **Modify**.

   Now when you move the construction plane, all handles that are on the plane are moved as well:



8. Bind the construction plane to a part face:
   a. Select the construction plane, right-click and select **Bind to plane**.
   b. Select a suitable part face.

For example, the inner flange of the column:



Tekla Structures displays a distance symbol for the binding. Now if you move the part face, the handles on the magnetic construction plane will follow.

NOTE Only the objects whose reference points are directly on the magnetic construction plane are affected. By default, the magnetic distance is `0.2` mm. To change this setting, use the advanced option XS_MAGNETIC_PLANE_OFFSET.

Bind handles using a magnetic construction line

1. In the custom component editor, click the **Add construction line** button .
2. Pick the starting point of the construction line.
3. Pick the end point of the construction line.

   Tekla Structures creates a construction line.
4. Double-click the line.

   The line properties are shown in the property pane.
5. Enter a name for the line.
6. In the **Magnetic** list, select **Yes**.
7. Click **Modify**.

   Now when you move the construction line, all handles that are on the line are moved as well.
8. Bind the construction line to a part face:

   a. Select the construction line, right-click and select **Bind to plane**.

b. Select a suitable part face.

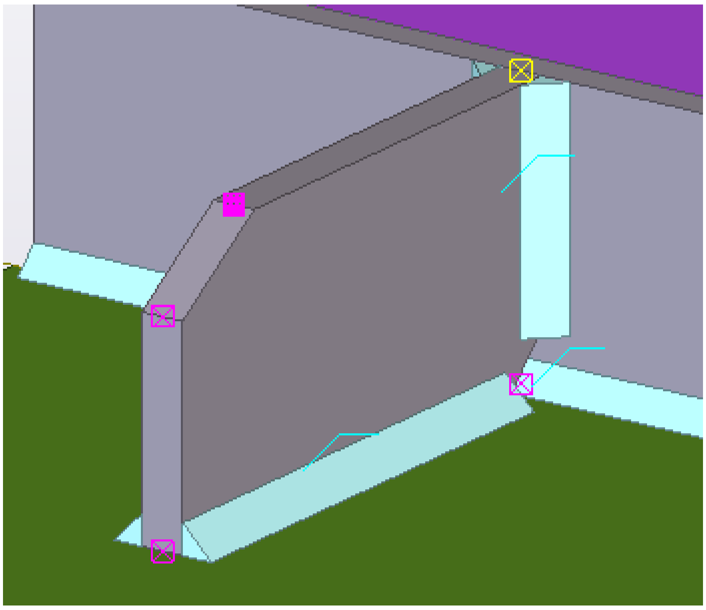
   Tekla Structures displays a distance symbol for the binding. Now if you move the part face, the handles on the magnetic construction line will follow.

Add a distance between component objects

Use *reference distance variables* to add a distance between two points or a point and a plane. The reference distance changes as you move the objects it refers to. You can use reference distances in calculations, for example, to determine the spacing of rungs on a ladder. Reference distance variables automatically get the prefix **D** (distance), which is shown in the **Variables** dialog box.

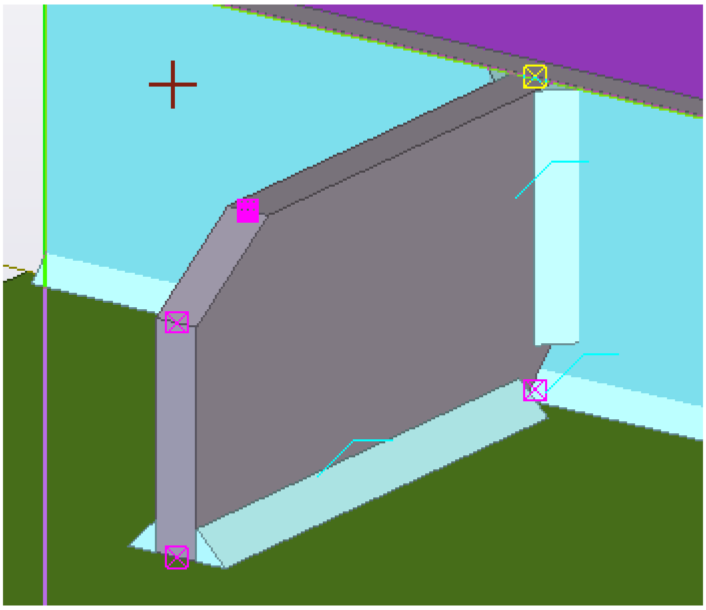
1. In a custom component view, select a handle (page 307).

   This is the starting point for your measurement.

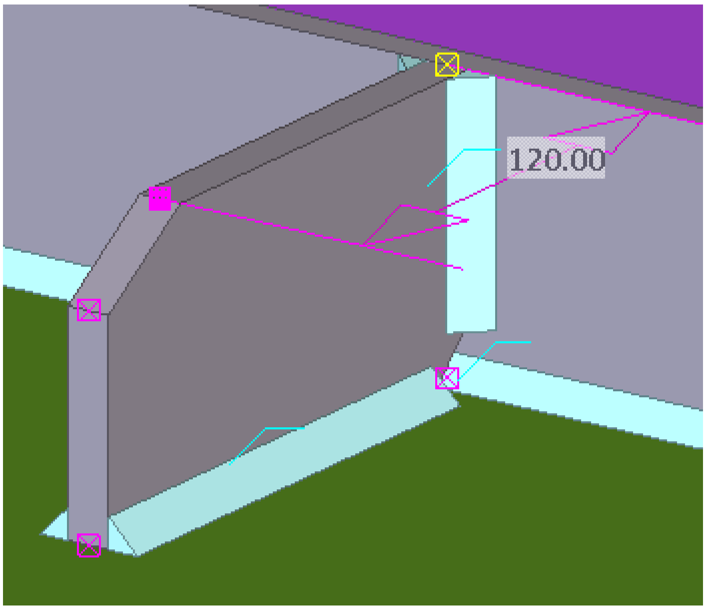
2. In the custom component editor, click the **Create reference distance** button .
3. Move the mouse pointer in the view to highlight a plane.

This will be the end point for your measurement. If you cannot highlight the correct plane, change the plane type (page 960) on the **Custom component editor** toolbar.

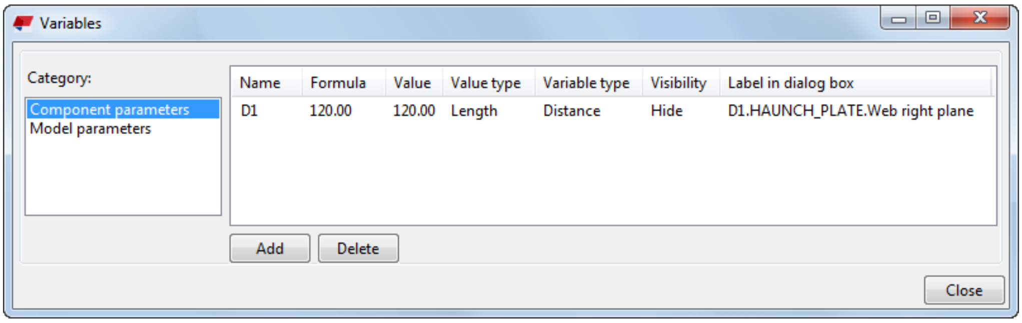


4. Click the plane to select it.

   Tekla Structures displays the distance.



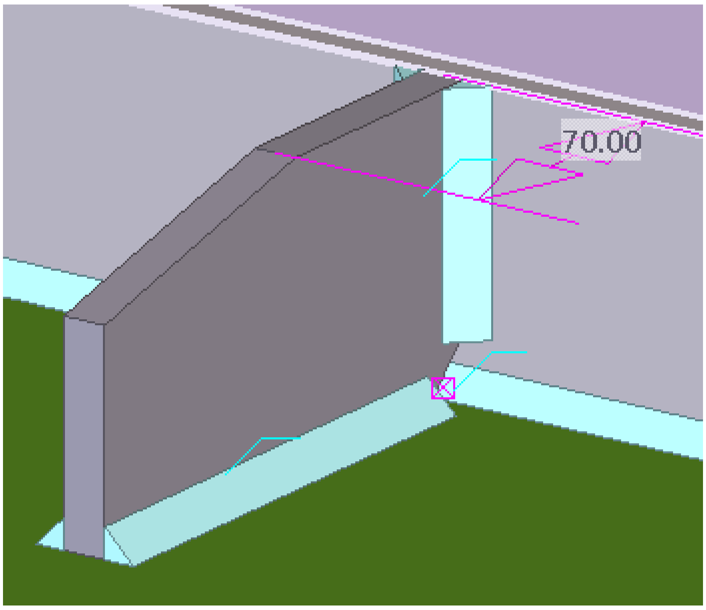
The corresponding reference distance variable is shown in the **Variables** dialog box:



Note that the **Create reference distance** command remains active. You can click more planes if you want to measure other distances.

5. To stop measuring, press **Esc**.
6. To check that the reference distance works correctly, move the handle.

   The distance changes accordingly. For example:



See also

Add variables to a custom component (page 849)

Set object properties by using parametric variables

Use *parametric variables* to set basic properties (such as name, material, profile, position number, and so on) for any object the custom component creates. Parametric variables automatically get the prefix **P** (parameter), which is shown in the **Variables** dialog box.

The following example shows how to create a variable that sets all welds in a custom component to a given size. After creating the variable, you are able to change the weld size directly in the custom component's dialog box.

1. In the custom component editor, click the **Display variables** button .

   The **Variables** dialog box opens.

2. Click **Add** to create a new parametric variable.

3. In the **Name** box, enter a name for the variable.

   You can also use the default name, such as `P1`. In this example, enter `Weldsize` as the name of the variable.

4. In the **Value type** list, select a suitable value type (page 964).

   The type determines what kind of values can be used with this variable. In this example, select **Length**, which is suitable for lengths and distances.

5. In the **Formula** box, enter a value or variable formula.

   Leave this box empty.

6. In the **Label in dialog box** box, enter a descriptive name for the parametric variable.

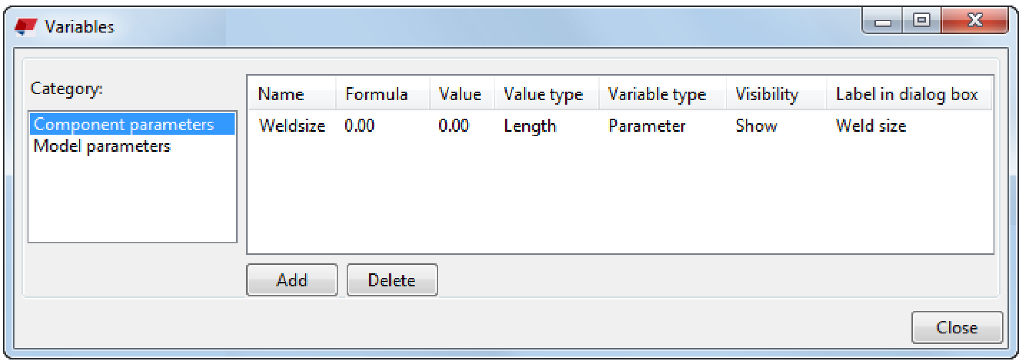
   This label will be shown in the custom component's dialog box. In this example, enter `Weld size` as the label.

7. In the **Visibility** list, define whether the variable will be visible in the custom component's dialog box.

   Hide the variable if you only use it in calculations. Show the variable if you want to be able to edit the value in the custom component's dialog box. In this example, select **Show**.

8. Click **Close**.

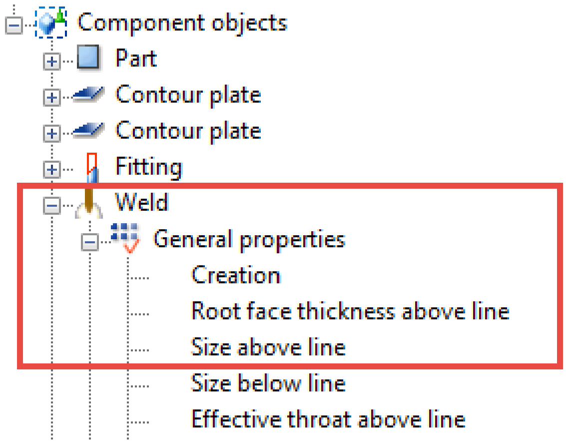
   You have now created a parametric variable with the following settings:



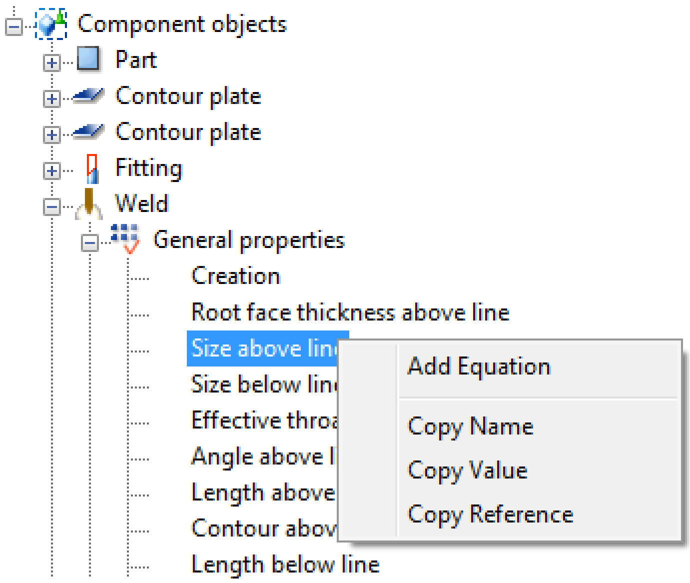
9. In the **Custom component browser**, link the variable to the desired object property.

a. Select the property.

Select the **Size above line** property of the uppermost weld.

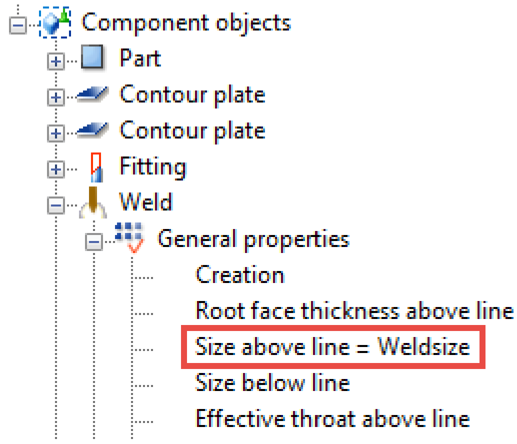


b. Right-click the property and select **Add Equation**.



c. After the equal sign, enter the name of the parametric variable.

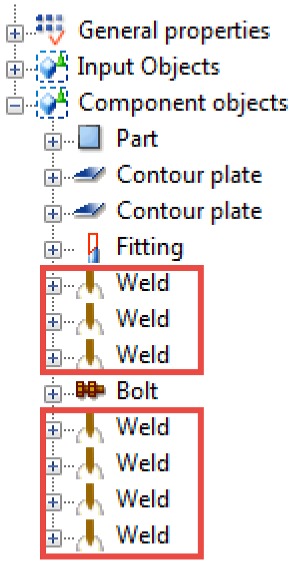
Enter `Weldsize` here.



You can now modify the **Size above line** property by using the **Weld size** box in the custom component's dialog box.

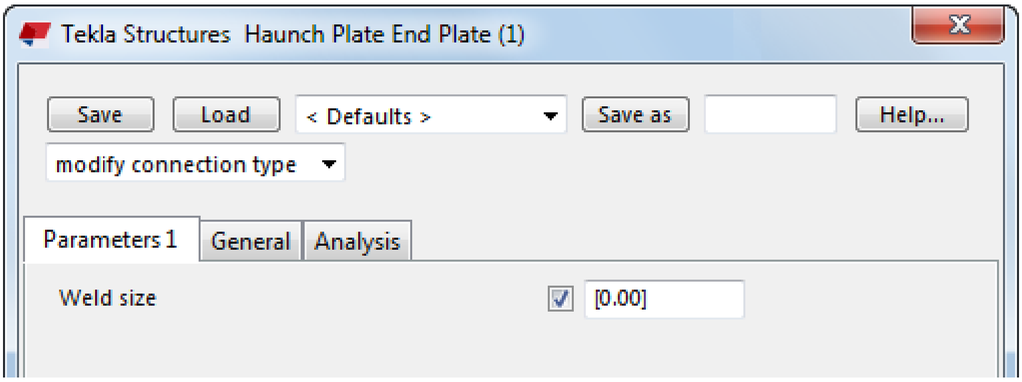
10. Repeat step 9 for any other property of the same type, if needed.

    Repeat the procedure for the other welds as well, so that they will all be linked to the **Weld size** box in the custom component's dialog box.



11. Save the custom component. (page 844)

The variable is now displayed in the custom component's dialog box, unless you set the visibility of the variable to **Hide** in step 7.



If you change the weld size value now, the size of all welds within the custom component will change accordingly.

See also

Copy properties and property references from another object (page 868)

Copy properties and property references from another object

You can copy properties, such as names and values, from other objects and use them to determine the properties of a custom component. You can also copy property *references*. The link is dynamic, so when the property changes, the reference reflects the change. For example, you can use a beam length reference in variable formulas. Even if the length changes, the correct value is always used in calculations.

1. In the **Custom component browser**, browse for the object property you want to copy.

   To find the required component object more easily, select it in a custom component view. Tekla Structures highlights the selected object in the **Custom component browser**.

2. Right-click the property and select one of the following:

   - **Copy Name**

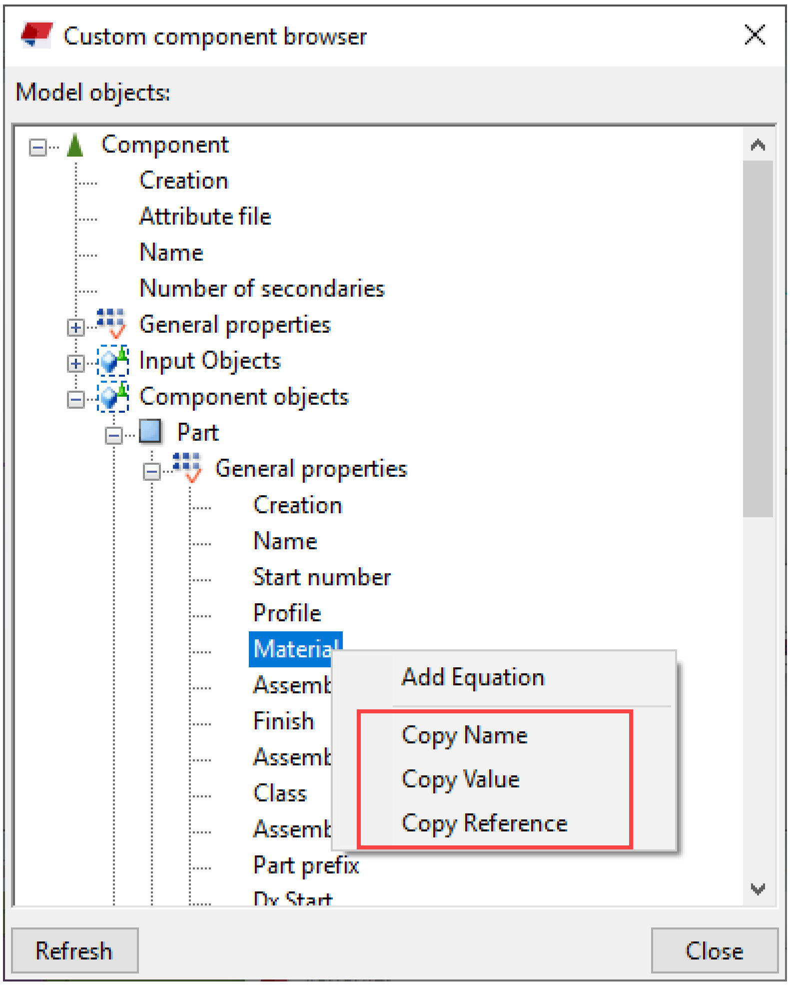
     Copies the name of the object. For example, `Material`.

   - **Copy Value**

     Copies the value the object currently has. For example, `S235JR`.

   - **Copy Reference**

     Copies the link to the property. For example, `fP(Material,"ID57720EEE-0000-000E-3134-363730393237")`.



3. Right-click where you want to insert the object property, and then select **Paste**.

   For example, you can paste a reference to the **Formula** box in the Variables (page 964) dialog box to use it in a calculation.

See also

Examples of parametric variables and variable formulas in custom components (page 884)

Create a variable formula

Use variable formulas to add more intelligence to your custom components. Variable formulas always begin with the equal sign (=). At its simplest, a formula can be a simple dependency between two variables, stating that P2 equals half of P1 (`P2=P1/2`), for example. To create more complex calculations, you can use functions and operators inside the formula. For example, you can add mathematical expressions, **if** statements, references to object properties, and so on. When you create formulas, note that multiplication is faster than division, for example, `P1*0.5` is faster than `P1/2`.

The following example shows how to create a formula that sets the weld size to half the thickness of the secondary part flange. When the component is used in a model, Tekla Structures will use the thickness of the secondary part flange to calculate the size of the weld.

1. In the custom component editor, click the **Display variables** button .

   The **Variables** dialog box opens.

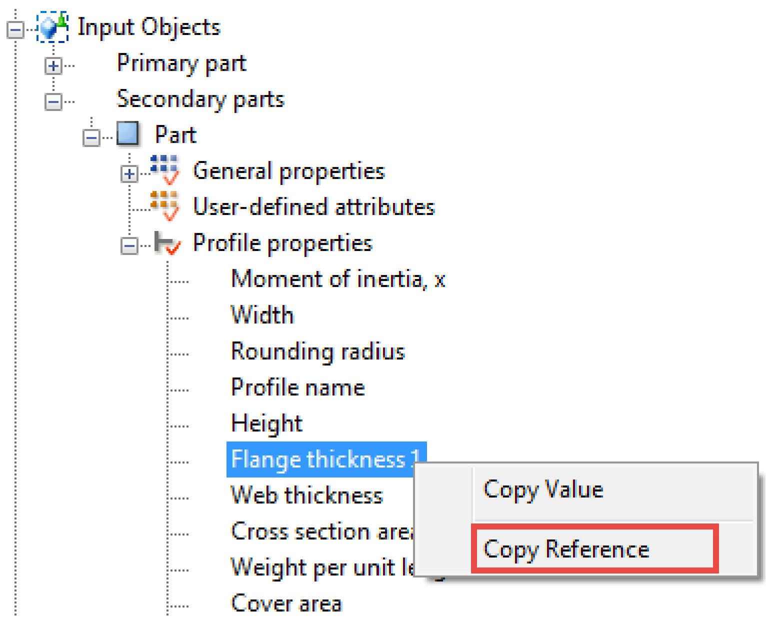
2. Click **Add** to create a new parametric variable.

3. In the **Name** box, enter a name for the variable.

   In this example, enter `w` as the name of the variable.

4. In the **Custom component browser**, go to **Input objects** --> **Secondary parts** --> **Part** --> **Profile properties** .

5. Right-click **Flange thickness 1** and select **Copy Reference**.



6. In the **Formula** box, type =, right-click, and select **Paste**.

   Tekla Structures pastes the reference to flange thickness from the clipboard.

7. After the flange thickness formula, enter `*0.5`.

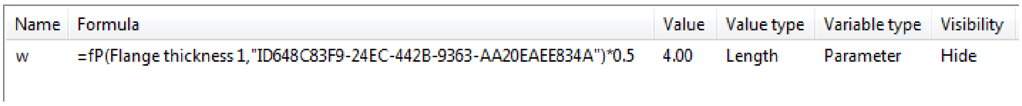
   The formula should now read:

   `=fP(Flange thickness 1,"GUID")*0.5`

8. Set the other values as follows:

   a. In the **Value type** list, select **Length**.

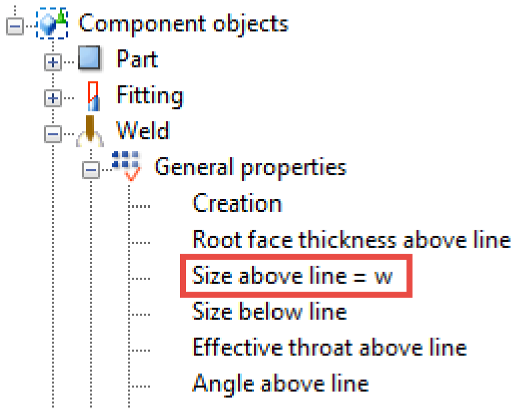
b. In the **Visibility** list, select **Hide**.



Name	Formula	Value	Value type	Variable type	Visibility
w	=fP(Flange thickness 1,"ID648C83F9-24EC-442B-9363-AA20EAEE834A")*0.5	4.00	Length	Parameter	Hide

9. In the **Custom component browser**, go to **Component objects** --> **Weld** --> **General properties** .

10. Right-click **Size above line**, select **Add Equation** and type `= w`.



See also

Functions in variable formulas (page 871)

Functions in variable formulas

You can use functions to calculate values for parametric variables. Variable formulas always begin with the equal sign (=).

For more information, see Set object properties by using parametric variables (page 864).

Arithmetic operators

Use arithmetic operators to combine expressions that return numeric values. You can use the following arithmetic operators:

Operator	Description	Notes
+	addition	Use also to create strings of parameters.
-	subtraction	
*	multiplication	Multiplication is faster than division. `=D1*0.5` is faster than `=D1/2`
/	division	

Logical and comparison operators

Use logical and comparison operators inside **if** statements. You can use **if-then-else** statements to test a condition and to set the value according to the result.

For example:

```
=if (D1>200) then 20 else 10 endif
```

You can use the following operators inside if statements:

<table>
<tr><th>Operator</th><th>Description</th><th>Example</th></tr>
<tr><td>==</td><td>both sides are equal</td><td></td></tr>
<tr><td>!=</td><td>sides are not equal</td><td></td></tr>
<tr><td><</td><td>left side is smaller</td><td></td></tr>
<tr><td><=</td><td>left side is smaller or equal</td><td></td></tr>
<tr><td>></td><td>right side is smaller</td><td></td></tr>
<tr><td>>=</td><td>right side is smaller or equal</td><td></td></tr>
<tr><td>&&</td><td>logical AND

both conditions must be true</td><td><code>=if (D1==200 && D2<40) then 6 else 0 endif</code>

If D1 is 200 and D2 smaller than 40, the result is 6, otherwise 0.</td></tr>
<tr><td>||</td><td>logical OR

only one condition must be true</td><td><code>=if (D1==200 || D2<40) then 6 else 0 endif</code>

If D1 is 200 or D2 is smaller than 40, the result is 6, otherwise 0.</td></tr>
</table>

Reference functions

Use reference functions to refer to the property of another object, such as the plate thickness of a secondary part. Tekla Structures refers to the object on the system level, so if the object property changes, so does the reference function value.

You can use the following reference functions:

Function	Description	Example
`fTpl("template attribute", "object GUID")`	Returns the template attribute value of an object that has a given object GUID.	`=fTpl("WEIGHT","ID50B8559A-0000-010B-3133-353432373038")` returns the weight of an object whose GUID is

Function	Description	Example
		ID50B8559A-0000-010B-3133-353432373038.
`fP("user-defined attribute", "object GUID")`	Returns the user-defined attribute value of an object that has a given object GUID.	`=fP("comment", "ID50B8559A-0000-010B-3133-353432373038")` returns the user-defined attribute **comment** of an object whose GUID is ID50B8559A-0000-010B-3133-353432373038.
`fValueOf("parameter")`	Returns the value of the parameter.	If the equation is `=P2+"*"+P3`, the result is `P2*P3` With `=fValueOf("P2")+"*"+fValueOf("P3")`, where P2=780 and P3=480, the result is `780*480`
`fRebarCatalogValue(BarGrade, BarSize, Usage, FieldName)`	Returns the reinforcing bar catalog value of an object. `Usage` can be either 2 ("Tie") or 1 ("Main"). `FieldName` must be one of the following: • 0 NominalDiameter • 1 ActualDiameter • 2 Weight • 3 MinRadius • 4 Hook1Radius • 5 Hook1Angle • 6 Hook1Length • 7 HookRadius • 8 Hook2Angle	`fRebarCatalogValue("A500HW", "10", 1, 2)` returns the size, usage, and weight of an object whose reinforcing bar grade is A500HW.

Function	Description	Example
	• 9 Hook2Length • 10 Hook3Radius • 11 Hook3Angl • 12 Hook3Length • 13 Area	

ASCII file as a reference function

You can refer to ASCII files to get data. Tekla Structures searches for the files in the following order:

1. model
2. `..\TeklaStructuresModels\<model>\CustomComponentDialogFiles\`
3. project (set with advanced option `XS_PROJECT`)
4. firm (set with advanced option `XS_FIRM`)
5. system (set with advanced option `XS_SYSTEM`)

The format for reading files is the following:

`fVF("filename", "key_value_of_row", column_number)`

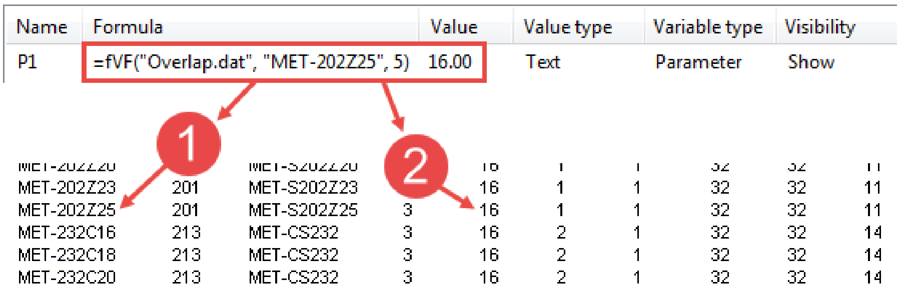
- Key value of row is a unique text value.
- Column number is an index starting from 1.

You can specify a character for data separation: `fVF(data file, lookup value, column#[, separator character])`.

- You can use a preferred column separator of choice. This enables the support for spaces in names, profiles, shapes, and so on, as well as the use of distance lists as input.
- You can use blank or empty strings as input.
- Only a single character can be used as a separator. For example, you cannot use a more complex separator such as `"/+/"`, because only the first character would be considered as a column separator.

Example

The `=fVF("Overlap.dat", "MET-202Z25", 5)` function is in the **Formula** box in the **Variables** dialog box. The function gets the value `16.0` for the profile `MET-202Z25`, from the `Overlap.dat` file.

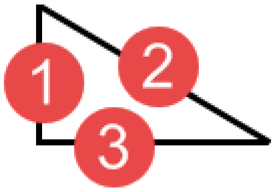


1. Key value of the row (MET-202Z25)
2. Column number (5)

Mathematical functions

Use mathematical functions to create more complex mathematical expressions. You can use the following functions:

Function	Description	Example
fabs(parameter)	Returns the absolute value of the parameter	`=fabs(D1)` returns 15 if D1 = -15
exp(power)	Returns `e` raised to the power `e` is Euler's number.	`=exp(D1)` returns 7.39 if D1 = 2
ln(parameter)	Returns the natural logarithm of the parameter (base number `e`)	`=ln(P2)` returns 2.71 if P2 = 15
log(parameter)	Returns the logarithm of the parameter (base number 10)	`=log(D1)` returns 2 if D1=100
sqrt(parameter)	Returns the square root of the parameter	`=sqrt(D1)` returns 4 if D1 = 16
mod(dividend, divider)	Returns the modulo of the division	`=mod(D1, 5)` returns 1 if D1 = 16
pow(base number, power)	Returns the base number raised to the specified power	`=pow(D1, D2)` returns 9 if D1 = 3 and D2 = 2

Function	Description	Example
hypot(side1,side2)	Returns the hypotenuse  1. side1 2. hypotenuse 3. side2	`=hypot(D1, D2)` returns 5 if D1 = 3 and D2 = 4
n!(parameter)	Returns the factorial of the parameter	`=n!(P2)` returns 24 if P2 = 4 (1*2*3*4)
round(parameter, accuracy)	Returns the parameter rounded off to the given accuracy	`=round(P1, 0.1)` returns 10.600 if P1 = 10.567
PI	Returns the value of pi to 31 decimal places	`=PI` returns 3.1415926535897932 384626433832795

Statistical functions

Use statistical functions to calculate sums and averages, and to round values. You can use the following statistical functions:

Function	Description	Example (P1 = 1.4 P2 = 2.3)
ceil()	Returns the smallest whole number greater than or equal to the parameter	`=ceil(P1)` returns 2
floor()	Returns the largest whole number less than or equal to the parameter	`=floor(P1)` returns 1
min()	Returns the smallest parameter	`=min(P1, P2)` returns 1.4
max()	Returns the largest parameter	`=max(P1, P2)` returns 2.3
sum()	Sum of the parameters	`=sum(P1, P2)` returns 3.7
sqsum()	Sum of the squared parameters: (parameter1)2 + (parameter2)2	`=sqsum(P1, P2)` returns 7.25
ave()	Average of the parameters	`=ave(P1, P2)` returns 1.85

Function	Description	Example (P1 = 1.4 P2 = 2.3)
sqave()	Average of the squared parameters	`=sqave(P1, P2)` returns 3.625

Example: Ceil and floor statistical functions

In this example, you have the following parametric variables:

- Beam length: P1 = 3500
- Post spacing: P2 = 450

P1 / P2 = 7.7778

You can use the `ceil` and `floor` statistical functions to round the value and then use the rounded value as the number of beam posts:

- `=ceil(P1/P2)` returns 8
- `=floor(P1/P2)` returns 7

Data type conversion functions

Use data type conversion functions to convert values into another data type. You can use the following data type conversion functions:

Function	Description	Example
int()	Converts data to integer	Useful especially for calculating profile dimensions: `=int(100.0132222000)` returns 100, if decimals are set to 0 in the **Options** dialog box
double()	Converts data to a double	
string()	Converts data to string	
imp()	Converts imperial units Use this function in calculations instead of imperial units. You cannot use imperial units directly in calculations.	For the following examples, length unit is set to mm and decimals are set to 2 in the **Options** dialog box. `=imp(1,1,1,2)` meaning 1 foot 1 1/2 inch returns 342.90 mm `=imp(1,1,2)` meaning 1 1/2 inches returns 38.10 mm `=imp(1,2)` meaning 1/2 inches returns 12.70 mm `=imp(1)` meaning 1 inch returns 25.40 mm `=3'/3"` is not possible, but `=imp(36)/imp(3)` is ok

Function	Description	Example
vwu(value, unit)	Converts the length values and angle values. The available units are: • "ft" ("feet", "foot") • "in" ("inch", "inches") • "m" • "cm" • "mm" • "rad" • "deg"	`=vwu(4.0,"in")` returns 101.60 mm, if length unit is set to mm and decimals are set to 2 in the **Options** dialog box `=vwu(2.0,"rad")` returns 114.59 degrees, if angle is set to degrees and decimals are set to 2 in the **Options** dialog box

NOTE The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals** .

String operations

Use string operations to manipulate character strings. Strings must be inside quotation marks in variable formulas.

You can use the following string operations:

Operation	Description	Example (P1 = "PL100*10")
match(parameter1, parameter2)	Returns 1 if parameters are equal and 0 if different. You can also use wildcards `*`, `?`, and `[ ]` with the match function.	`=match(P1, "PL100*10")` returns 1 Accept all profiles starting with PFC: `=match(P4, "PFC*")` Accept profiles starting with PFC, and height starts with 2, 3, 4 or 5: `=match(P4, "PFC[2345]*")` Accept profiles starting with PFC, heights are 200, 300, 400 or 500 and width starts with 7: `=match(P4, "PFC[2345]00?7*")`
length(parameter)	Returns the number of characters in the parameter.	`=length(P1)` returns 8
find(parameter, string)	Returns the order number (starting at zero) of the specified string and -1 if the specified string is not	`=find(P1, "*")` returns 5

Operation	Description	Example (P1 = "PL100*10")
	found from the parameter.	
getat(parameter, n)	Returns the n:th (starting at zero) character from the parameter.	`=getat(P1, 1)` returns "L"
setat(parameter, n, character)	Sets the n:th (starting at zero) character to the specified character in the parameter.	`=setat(P1, 0, "B")` returns "BL100*10"
mid(string, n, x)	Returns x characters from the string starting from n:th (starting at zero) character. If you leave out the last argument (x), returns the last part of the string.	`=mid(P1,2,3)` returns "100"
reverse(string)	Reverses the given string.	`=reverse(P1)` returns "01*001LP"

Example 1

To define profile size PL100*10 with two variables P2 = 100 and P3 = 10, enter the formula as follows:

```
="PL"+P2+"*"+P3
```

Example 2

Tekla Structures handles bolt spacings as strings. To define bolt spacing, set **Value type** to **Distance list** and enter the formula as follows:

```
=P1+" "+P2
```

This results in 100 200, if P1 = 100 (**length**) and P2 = 200 (**length**).

Trigonometric functions

Use trigonometric functions to calculate angles. You can use the following trigonometric functions:

Function	Description	Example
sin()	Returns the sine value	`=sin(d45)` returns 0.71
cos()	Returns the cosine value	`=cos(d45)` returns 0.71
tan()	Returns the tangent value	`=tan(d45)` returns 1.00
asin()	Inverse function of sin(), return value in radians	`=asin(1)` returns 1.571 rad

Function	Description	Example
acos()	Inverse function of cos(), return value in radians	`=acos(1)` returns 0 rad
atan()	Inverse function of tan(), return value in radians	`=atan(1)` returns 0.785 rad
sinh()	Returns the hyperbolical sine value	`=sinh(d45)` returns 0.87
cosh()	Returns the hyperbolical cosine value	`=cosh(d45)` returns 1.32
tanh()	Returns the hyperbolical tangent value	`=tanh(d45)` returns 0.66
atan2()	Returns the angle whose tangent is the quotient of the two numbers. Return value in radians	`=atan2(1,3)` returns 0.32

NOTE When you use trigonometric functions in variable formulas, you need to include a prefix to define the unit. If you do not include a prefix, Tekla Structures uses radians as the default unit.

- d is degree. For example, `sin(d180)`
- r is radians (default). For example, `sin(r3.14)` or `sin(3.14)`

Market size function

Use the market size function in a custom component to select a suitable plate dimension (usually plate thickness) from the available market sizes. For example, a plate's thickness should match the web of a beam.

Function	Description	Example
fMarketSize(material, thickness, extrastep)	Returns the next available market size for the material from the `marketsize.dat` file, based on the thickness you specify. The file must be in the `..\environments\your_environment\profil` folder or the system folder. For an extra step, enter a number to define the increment to the next size (default is 0).	`=fMarketSize("S235JR", 10, 0)`

Example

In this example, you have the following data in `marketsize.dat`:

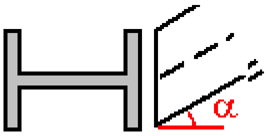
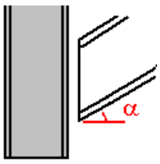
```
S235JR,6,9,12,16,19,22
SS400,1.6,2.3,3.2,4.5,6,9,12,16,19,22,25,28,32,38
DEFAULT,6,9,12,16,19,22,25,28,32,38
```

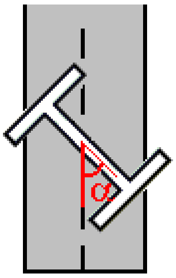
The first item in a row is a material grade followed by available plate thicknesses in millimeters. The DEFAULT line lists the thicknesses available in all other material grades.

With the above data, the function `=fMarketSize("S235JR",10,0)` would return 12, and `=fMarketSize("S235JR",10,1)` would return 16 (one size up).

Framing condition functions

Use the framing condition functions to return the skew, slope, and cant angle of the secondary beam relative to the main part (column or beam). You can use the following framing condition functions:

Function	Description	Example
fAD("skew", GUID)	Returns the skewed angle of the secondary part whose GUID is given. 	`=fAD("skew","ID50B8559A-0000-010B-3133-353432373038")` returns 45 ID50B8559A-0000-010B-3133-353432373038 is the GUID of the secondary part, which is at a 45 degree angle to the main part.
fAD("slope", GUID)	Returns the sloped angle of the secondary part whose GUID is given. 	`=fAD("slope","ID50B8559A-0000-010B-3133-353432373038")`

Function	Description	Example
fAD("cant", GUID)	Returns the cant angle of rotated secondary part whose GUID is given. 	`=fAD("cant","ID50B8559A-0000-010B-3133-353432373038")`

NOTE

- These functions do not return positive and negative slope and skew values. It is not possible to determine up or down slope and left or right skew with these functions.
- The maximum skew angle to return is 45 degrees.
- Tekla Structures calculates the angles in 2D so that slope and skew are isolated from each other. For example, the skew angle is not taken into consideration when calculating the slope angle, which means that the slope angle value stays the same regardless of the secondary part's rotation around the primary part.

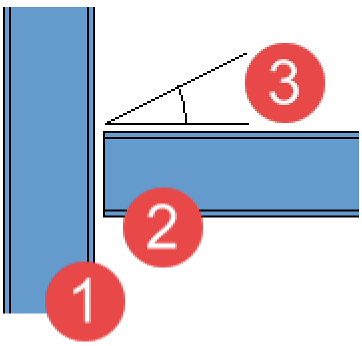
  To find out the true 3D slope with the skew included, you can use the following mathematical formula:

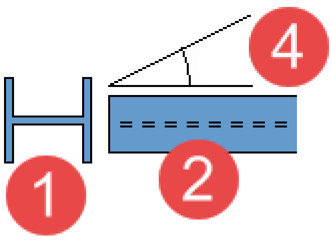
  ```
  TRUE_SLOPE = atan( tan(SLOPE) * cos(SKEW))
  ```

Example 1

The slope and skew are relative to a beam framing into a column.

Side view **Top view**



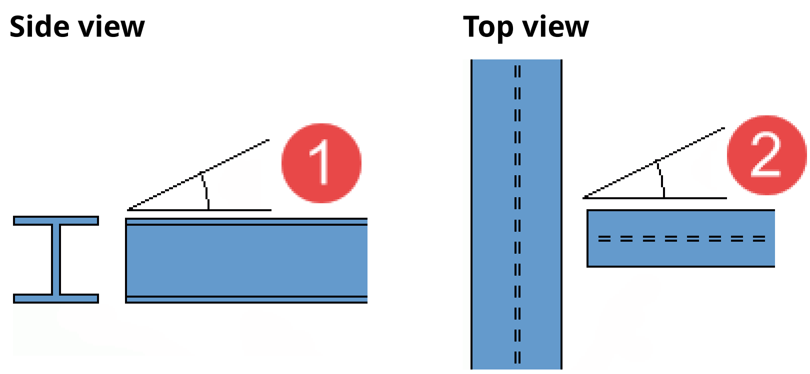


1. Column
2. Beam

3. **Slope**
4. **Skew**

Example 2

With two beams, the **slope** is actually the horizontal skew of the beam framing into the other beam, and the vertical slope of the beam relative to the main is actually the **skew** angle.



1. **Skew**
2. **Slope**

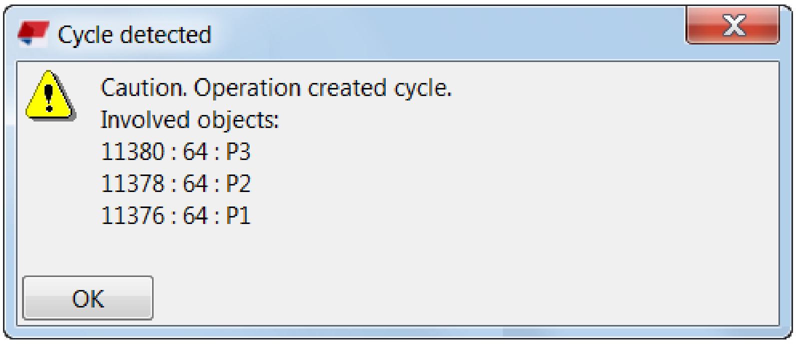
How to avoid cyclic dependencies in formulas

Be careful not to create cyclic dependencies between variables, or else the custom component will not work correctly. A cyclic dependency chain contains formulas that make a variable eventually dependent on itself.

In the following example, variable P1 becomes dependent on itself, through variables P2 and P3:

Name	Formula
P1	=P2
P2	=P3/4
P3	=P1*2

Cyclic dependencies may also occur when binding handles to other objects or when using magnetic construction planes. When you create new formulas, bindings, or magnetic construction planes, Tekla Structures checks if they create cyclic dependency chains in a custom component. If that happens, a warning message "Caution. Operation created cycle." is displayed.



Tekla Structures also writes the message "Cycle detected in parametric solver" into the session history log file and lists the objects involved in the cyclic dependency, to help you find and remove the cyclic dependency. If you do not remove it, the custom component will not work correctly.

8.9 Examples of parametric variables and variable formulas in custom components

Here you will find some examples that demonstrate how to use parametric variables and variable formulas to create intelligent custom components that adapt to changes in the model.

NOTE There are some limitations concerning the variable names.

- In some of the examples below, we reference variables by name. To be able to correctly reference a variable in your formula, variable name must be 19 characters or shorter. Variables with longer names will not work correctly when referenced.
- Variable names cannot contain mathematical operators (`+`, `-`, `*`, `/`).
- You cannot use a mathematical constant, such as `PI` or `e`, as a variable name.

The examples are independent from each other.

- Example of a variable formula: Set the end plate material (page 886)

  This example shows how to link a parametric variable to the end plate material of a component object.

- Example of a variable formula: Create new component objects (page 890)

  This example shows how to create a parametric variable that adds bolts to the custom component.

- Example of a variable formula: Replace sub-components (page 891)

  This example shows how to create a parametric variable that replaces sub-components with other sub-components.

  This example shows how to create a parametric variable that modifies a sub-component on the basis of a component attribute file.

  This example shows how to use construction planes for determining the position of the stiffeners. You will position the stiffeners so that they divide the beam into three equally long sections.

  This example shows how to create two parametric variables that determine the bolt size and bolt standard.

  This example shows how to create a variable formula that calculates the bolt group distance from the beam flange.

  This example shows how to create a variable formula that calculates the number of bolt rows based on the beam height. You will use `if` statements in the calculations.

  This example shows how to link parametric variables to the user-defined attributes of panels. You can then use the user-defined attributes in view filters to show or hide the panels.

  This example shows how to create a variable formula that calculates the number of handrail posts based on the length template attribute of the beam. The handrail posts are created at both ends of the beam and one of them is copied with the **Array of objects (29)** component.

  This example shows how to link a parametric variable to an Excel spreadsheet. For example, you can use Excel spreadsheets to check connections.

- Examples of a variable formula: Rebar set modifiers in custom components (page 907)

  These examples show you how to use rebar set modifiers for defining the properties and hooks of rebar set bars in custom components.

Example of a variable formula: Set the end plate material

This example shows how to link a parametric variable to the end plate material of a component object.

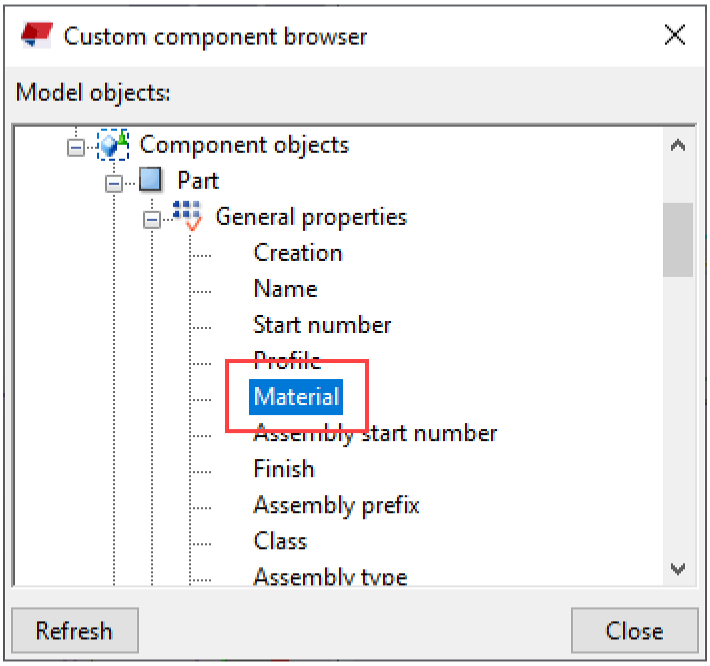
1. On the **Custom component editor** toolbar, click the **Display variables** button.

   The **Variables** dialog box opens.
2. Click the **Add** button.

   A new parametric variable appears.
3. In the **Value type** list, change the variable's value type to **Material**.
4. In the **Label in dialog box** box, enter `End Plate Material`.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	0.00	0.00	Material	Parameter	Show	End Plate Material

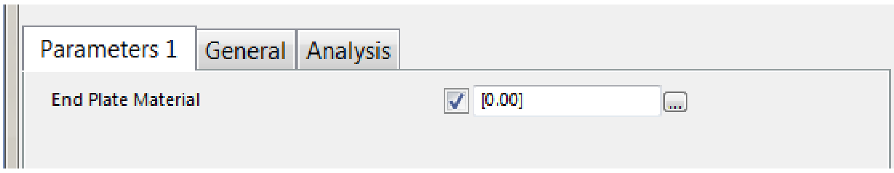
5. In the **Custom component browser**, browse for the end plate material.

6. Right-click **Material** and select **Add Equation**.
7. Enter `P1` after the equal sign, and then press **Enter**.

8. Save the custom component.
9. Close the custom component editor.

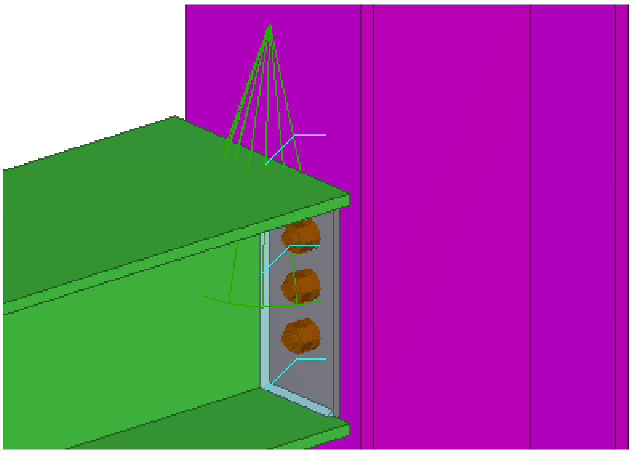
You can now change the end plate material in the custom component's dialog box.



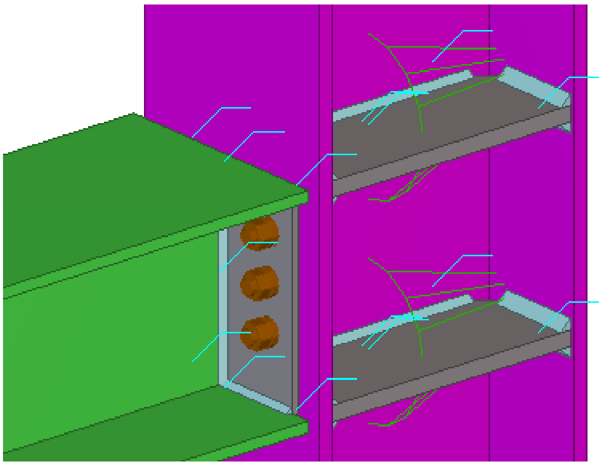
Example of a variable formula: Create a nested connection with stiffeners

This example shows how to create a nested custom connection that consists of and end plate, a bolt group, welds, and two **Stiffeners (1003)** components. The stiffeners are optional, which means that you can choose whether or not to create them when using the component in a model.

1. Add an **End plate (144)** component.



2. Explode the end plate component.
3. Add two **Stiffeners (1003)** components.



4. Create a nested custom component that contains the stiffeners and the end plate objects.

    a. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.

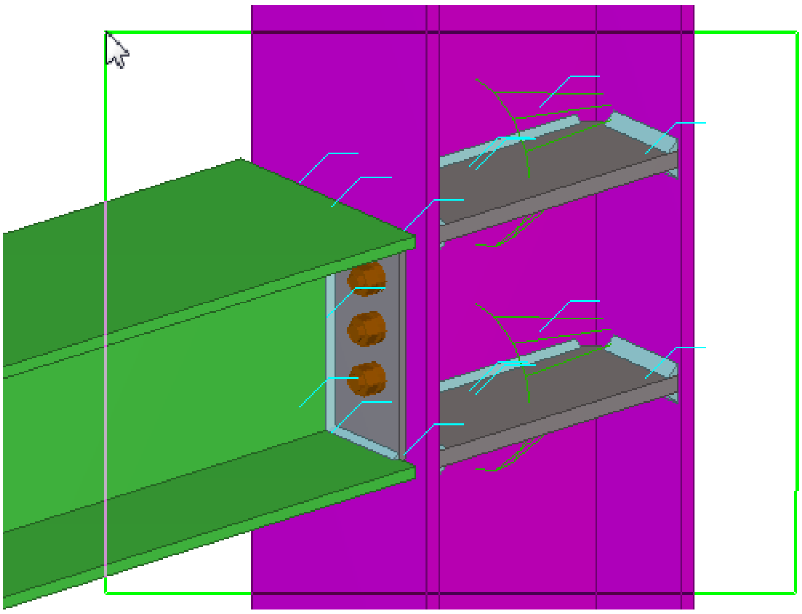
    b. Click the **Access advanced features** button and select **Define custom component...**.

    c. In the **Type** list, select **Connection**.

    d. In the **Name** box, enter `End plate with stiffeners`.

    e. Click **Next >**.

    f. Make an area selection (from right to left) to include the following objects in the nested component: the column, the beam, the stiffener components, and all the end plate objects.

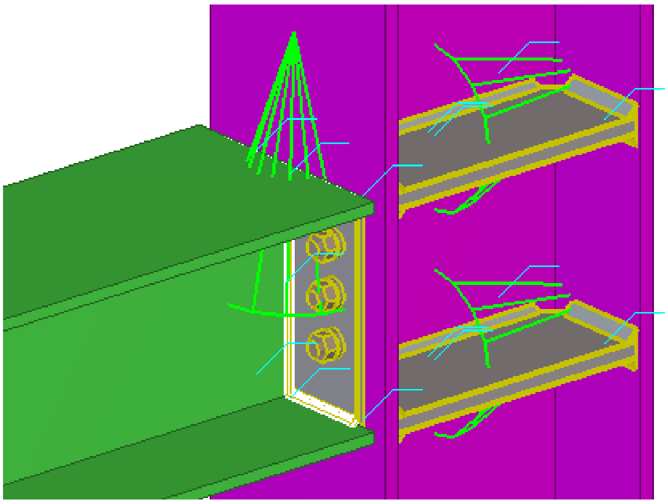


g. Click **Next >**.

h. Choose the column as the main part of the nested component, and then click **Next >**.

i. Choose the beam as the secondary part of the nested component, and then click **Finish**.

   Tekla Structures creates the nested component.



5. Select the nested component you just created.

6. Right-click and select **Edit Custom Component**.

7. In the custom component editor, click the **Display variables**  button.

   The **Variables** dialog box opens.

8. Create the following parametric variables:

   a. Click **Add** to create a new parametric variable **P1**.

   b. In the **Value type** list, select **Yes/No**.

   c. In the **Label in dialog box** box, enter `Create Stiffener 1`.

   d. Click **Add** to create a new parametric variable **P2**.

   e. In the **Value type** list, select **Yes/No**.

   f. In the **Label in dialog box** box, enter `Create Stiffener 2`.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	0	0	Yes/No	Parameter	Show	Create Stiffener 1
P2	0	0	Yes/No	Parameter	Show	Create Stiffener 2

9. Link the variables to the **Creation** property of the two stiffeners:

   a. In the **Custom component browser**, browse for the uppermost **Connection**.

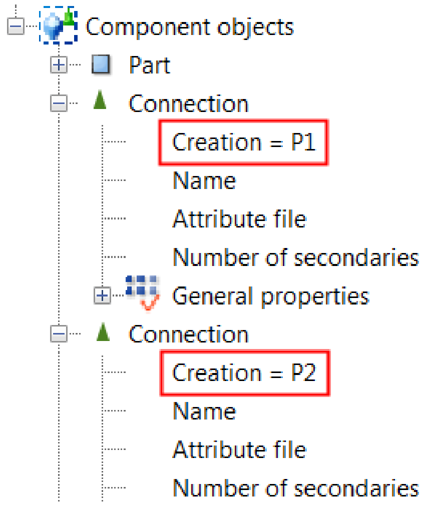
b. Right-click **Creation** and select **Add Equation**.

c. Enter `P1` after the equal sign, and then press **Enter**.

d. Browse for the second **Connection**.

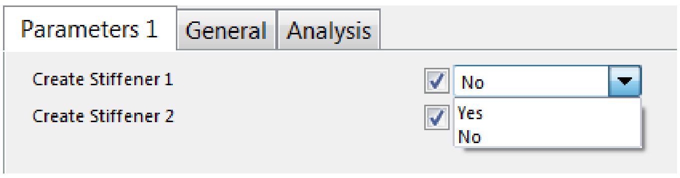
e. Right-click **Creation** and select **Add Equation**.

f. Enter `P2` after the equal sign, and then press **Enter**.



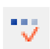
10. Save and close (page 844) the nested component.

You now have the following options in the nested component's dialog box:



Example of a variable formula: Create new component objects

This example shows how to create a parametric variable that adds bolts to the custom component.

1. On the **Custom component editor** toolbar, click the **Display variables**  button.

   The **Variables** dialog box opens.

2. Click **Add** to create a new parametric variable.

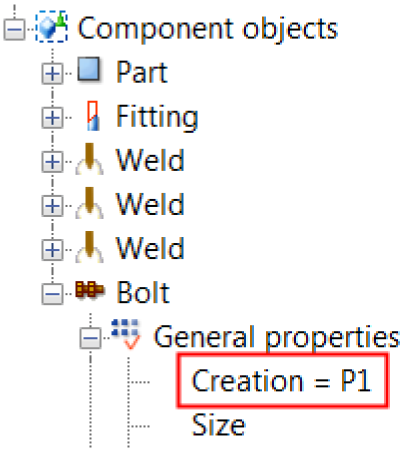
3. Modify the variable as follows:

   a. In the **Value type** list, select **Yes/No**.

   b. In the **Label in dialog box** box, enter `Create bolts`.

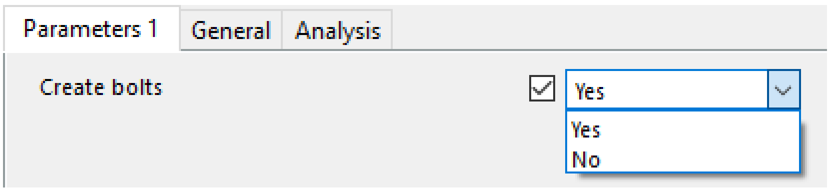
Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	0	0	Yes/No	Parameter	Show	Create bolts

4. Select the bolt group in a custom component view to highlight it in the **Custom component browser**.
5. In the **Custom component browser**, browse for **Bolt**.
6. Right-click **Creation** and select **Add Equation**.
7. Enter `P1` after the equal sign, and then press **Enter**.



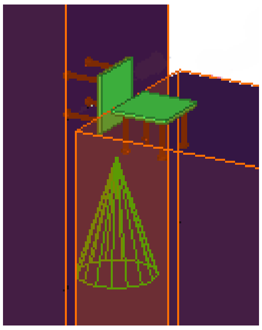
8. Save the custom component.
9. Close the custom component editor.

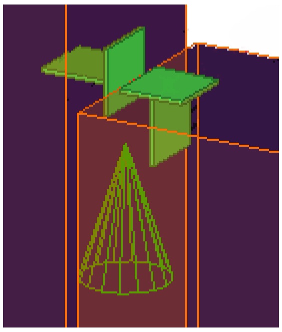
You now have the following option in the custom component's dialog box:



Example of a variable formula: Replace sub-components

This example shows how to create a parametric variable that replaces sub-components with other sub-components.





1. On the **Custom component editor** toolbar, click the **Display variables**  button.

   The **Variables** dialog box opens.

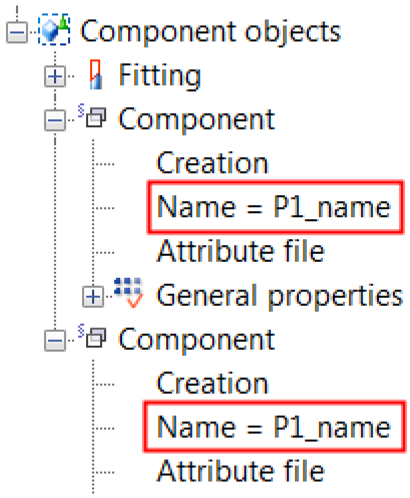
2. Click **Add** to create a new parametric variable.
3. Modify the variable as follows:
   a. In the **Value type** list, select **Component name**.

      Tekla Structures automatically adds the suffix `_name` in the variable name. Do not delete the suffix.

   b. In the **Formula** box, enter the name of the sub-component.
   c. In the **Label in dialog box** box, enter `Cast-in plate`.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1_name	castin1	castin1	Component name	Parameter	Show	Cast-in plate

4. Link the variable to the **Name** property of both sub-components:
   a. In the **Custom component browser**, browse for the **Name** attribute of the first sub-component.
   b. Right-click **Name** and select **Add Equation**.
   c. Enter `P1_name` after the equals sign.
   d. Repeat steps 4b–4c for the other sub-component.



5. Save the custom component.
6. Close the custom component editor.

You can now change the sub-components by using the **Cast-in-plate** option in the custom component's dialog box.

Example of a variable formula: Modify a sub-component by using a component attribute file

This example shows how to create a parametric variable that modifies a sub-component on the basis of a component attribute file.

1. On the **Custom component editor** toolbar, click the **Display variables** button.

   The **Variables** dialog box opens.
2. Click **Add** to create a new parametric variable.
3. In the **Value type** list, select **Component attribute file**.

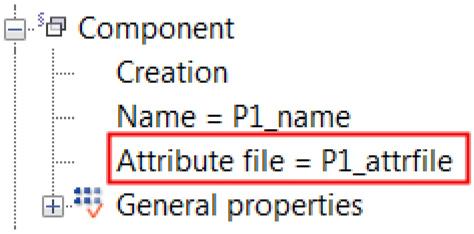
   Tekla Structures automatically adds the suffix `_attrfile` in the variable name. Do not delete the suffix.
4. In the **Formula** box, enter the name of the component attribute file.
5. In the **Name** box, ensure that the variable has the same prefix as the variable that is linked to the component name.

   In this example, the prefix is P1.

   NOTE The component name and the component attribute file variables must always have the same prefix, otherwise they do not work.
6. In the **Label in dialog box** box, enter `Properties file`.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1_name	castin1	castin1	Component name	Parameter	Show	Cast-in plate
P1_attrfile	prop1	prop1	Component attribute file	Parameter	Show	Properties file

7. In the **Custom component browser**, browse for the component attribute file property of the sub-component.
8. Right-click **Attribute file** and select **Add Equation**.
9. Enter `P1_attrfile` after the equal sign, and then press **Enter**.

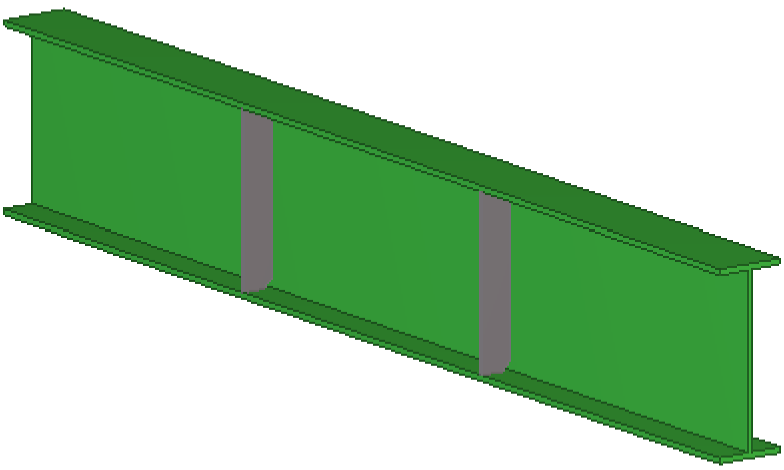


10. Save the custom component.
11. Close the custom component editor.

You can now modify the sub-component by using the **Properties file** option in the custom component's dialog box.

Example of a variable formula: Define the stiffener position using construction planes

This example shows how to use construction planes for determining the position of the stiffeners. You will position the stiffeners so that they divide the beam into three equally long sections.



1. Ensure that **Direct modification** is switched off.

   The selection of handles is easier when **Direct Modification** is off.

2. On the **Custom component editor** toolbar, click the **Display variables** button.

   The **Variables** dialog box opens.

3. Click **Add** to create a new parametric variable.

4. Get the GUID of the beam.

   a. On the ribbon, click **Inquire objects** .

   b. Select the beam.

   c. In the **Inquire Object** dialog box, check the GUID of the beam.

5. Modify the variable as follows:

   a. In the **Formula** box, enter `=fTpl("LENGTH","ID4C8B5E24-0000-017D-3132-383432313432")`.

      `ID4C8B5E24-0000-017D-3132-383432313432` is the GUID of the beam.

      The value of the variable is now the same as the beam length. If you change the beam length, also the value changes.

   b. In the **Label in dialog box** box, enter `Beam Length`.

6. Click **Add** to create another parametric variable.

7. Modify the new variable as follows:

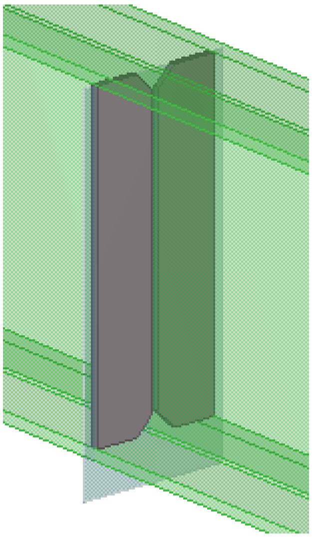
   a. In the **Formula** box, enter `=P1/3`.

   b. In the **Label in dialog box** box, enter `3rd Points`.

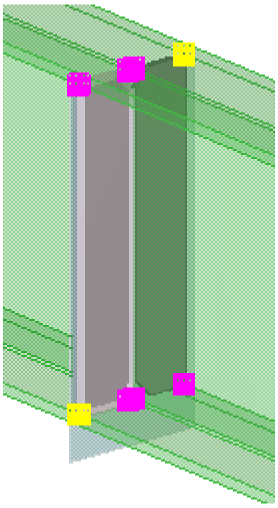
8. Create a construction plane:

   a. In the custom component editor, click the **Add construction plane** button.

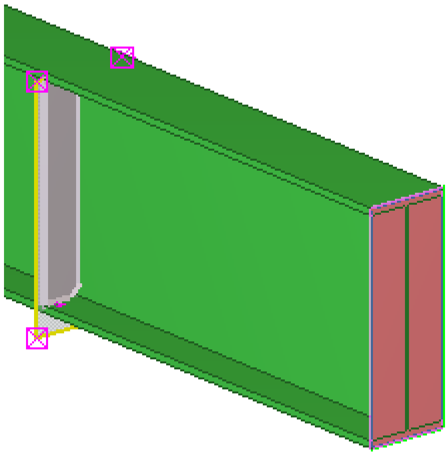
   b. Pick the required points and then click the middle mouse button to create a construction plane in the center of a stiffener at one end.



9. Bind the stiffener to the construction plane:
   a. Select the stiffener.
   b. Hold down **Alt** and use area selection (from left to right) to select all stiffener handles.



   c. Right-click and select **Bind to plane**.
   d. Bind the stiffener handles to the construction plane.
10. Bind the construction plane to the beam end:
   a. Select the construction plane.
   b. Right-click and select **Bind to plane**.
   c. Bind the construction plane to the beam end.

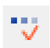


11. Repeat steps 9–11 for the stiffener at the other end.
12. In the **Formula** box, enter `=P2` for the two distance variables that bind the construction planes to the beam ends.
13. Save the custom component.
14. Close the custom component editor.

If you now change the beam length, the position of the stiffeners changes so that the stiffeners divide the beam into three equally long sections.

Example of a variable formula: Determine the bolt size and bolt standard

This example shows how to create two parametric variables that determine the bolt size and bolt standard.

1. On the **Custom component editor** toolbar, click the **Display variables**  button.

   The **Variables** dialog box opens.
2. Click **Add** twice to create two new parametric variables.
3. Modify the first variable as follows:
   - In the **Value type** list, select **Bolt size**.

     Tekla Structures automatically adds the suffix `_diameter` to the name of the variables. Do not delete the suffix.
   - In the **Label in dialog box** box, enter `Bolt Size`.
4. Modify the second variable as follows:

a. In the **Value type** list, select **Bolt standard**.

Tekla Structures automatically adds the suffix `_screwdin` to the name of the variable. Do not delete the suffix.

b. In the **Name** box, change the prefix of the second variable so that the prefixes for the two variables are the same.

In this example, the prefix is P1.

Name	Formula	Value	Value type	Variable type	Visibility
P1_diameter	0.00	0.00	Bolt size	Parameter	Show
P1_screwdin	0.00	0.00	Bolt standard	Parameter	Show

NOTE The bolt size and bolt standard variables must always have the same prefix, otherwise they do not work.

c. In the **Label in dialog box** box, enter `Bolt Standard`.

5. Link the parametric variables to the bolt group properties:

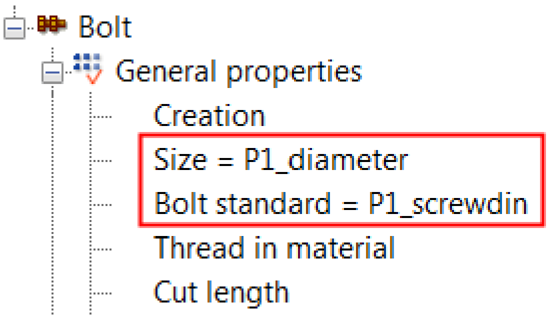
a. In the **Custom component browser**, browse for the size property of the component object.

b. Right-click **Size** and select **Add Equation**.

c. Enter `P1_diameter` after the equal sign, and then press **Enter**.

d. Right-click **Bolt standard** and select **Add Equation**.

e. Enter `P1_screwdin` after the equal sign, and then press **Enter**.



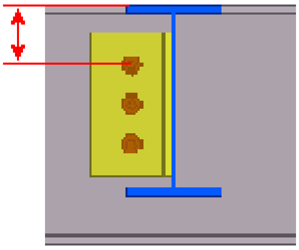
6. Save the custom component.

7. Close the custom component editor.

You can now determine the bolt size and bolt standard for the custom component in the custom component's dialog box.

Example of a variable formula: Calculate the bolt group distance

This example shows how to create a variable formula that calculates the bolt group distance from the beam flange.



1. Modify the bolt group properties as follows:

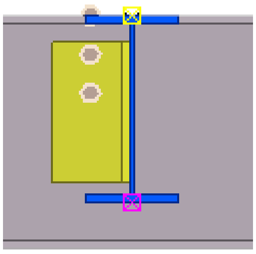
   a. In the custom component editor, double-click the bolt group.

      The **Bolt** properties open.

   b. Clear all values that are in the **Offset from** section.

   c. Click **Modify**.

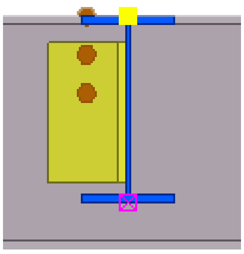
      The bolt group moves to the same level with the start point handle of the bolt group.

2. Bind the bolt group to the beam flange:

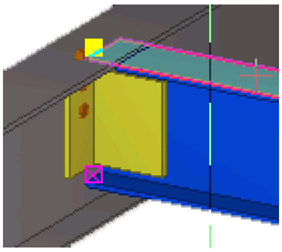
   a. In the custom component editor, select the bolt group.

   b. Select the yellow top handle.

   c. Right-click the handle and select **Bind to plane**.

   d. Select the top flange of the beam.



A new distance variable appears in the **Variables** dialog box.

3. On the **Custom component editor** toolbar, click the **Display variables**  button.

   The **Variables** dialog box opens.

4. Click **Add** to create a new parametric variable.
5. Modify the variable as follows:
   a. In the **Formula** box, enter a distance value.
   b. In the **Label in dialog box** box, enter `Vertical distance to bolt`.
6. In the **Formula** box, enter `=-P1` for the distance variable.

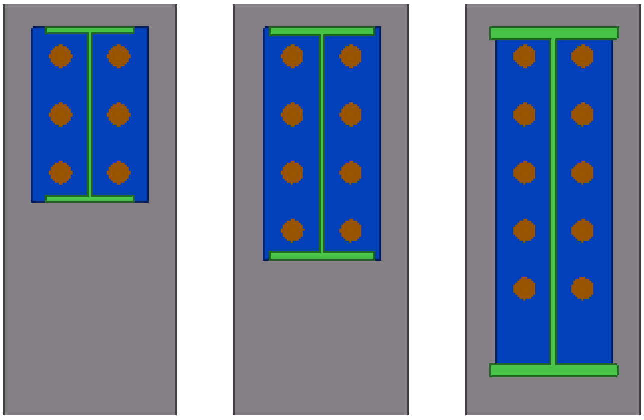
Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
D1	=-P1	-75.00	Length	Distance	Hide	D1.BOLT.BEAM
P1	75.00	75.00	Length	Parameter	Show	Vertical distance to bolt

7. Save the custom component.
8. Close the custom component editor.

You can now determine the bolt group distance from the beam flange by changing the **Vertical distance to bolt** value in the custom component's dialog box.

Example of a variable formula: Calculate the number of bolt rows

This example shows how to create a variable formula that calculates the number of bolt rows based on the beam height. You will use `if` statements in the calculations.



1. On the **Custom component editor** toolbar, click the **Display variables**  button.

   The **Variables** dialog box opens.

2. Click **Add** to create a new parametric variable.
3. In the **Value type** list, select **Number**.
4. In the **Custom component browser**, browse for height property of the beam.
5. Right-click **Height** and select **Copy Reference**.
6. In the **Formula** box, enter the following `if` statement for the parametric variable:

```
=if (fP(Height,"ID50B8559A-0000-00FD-3133-353432363133")< 301) then 2
else (if (fP(Height,"ID50B8559A-0000-00FD-3133-353432363133")>501) then 4
else 3 endif) endif
```

   In the formula, `fP(Height,"ID50B8559A-0000-00FD-3133-353432363133")` is the beam height reference copied from the **Custom component browser**. The variable gets its value in the following way:

   - If the beam height is under 301 mm, the value is 2.
   - If the beam height is over 501 mm, the value is 4.
   - If the beam height is between 300 and 500 mm, the value is 3.

7. Click **Add** to create another parametric variable.
8. In the **Value type** list, select **Distance list** for the new variable.
9. In the **Formula** box, enter `=P1+"*"+100` for the new variable.

In the formula, `100` is the bolt spacing and the `P1` value is the number of bolt rows.

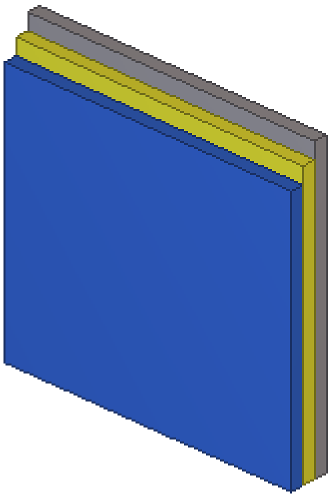
Name	Formula	Value	Value type
P1	=if (fP(Height,"ID50B8559A-0000 ...	2	Number
P2	=P1+"*"+100	2*100.00	Distance list

10. In the **Custom component browser**, browse for **Bolt group distance x**.
11. Right-click **Bolt group distance x** and select **Add Equation**.
12. Enter `P2` after the equal sign, and then press **Enter**.
13. Save the custom component.
14. Close the custom component editor.

When you now change the beam height, the number of bolt rows changes as well.

Example of a variable formula: Link variables to user-defined attributes

This example shows how to link parametric variables to the user-defined attributes of panels. You can then use the user-defined attributes in view filters to show or hide the panels in the model.



1. On the **Custom component editor** toolbar, click the **Display variables**  button.

   The **Variables** dialog box opens.
2. Click **Add** to create a new parametric variable.

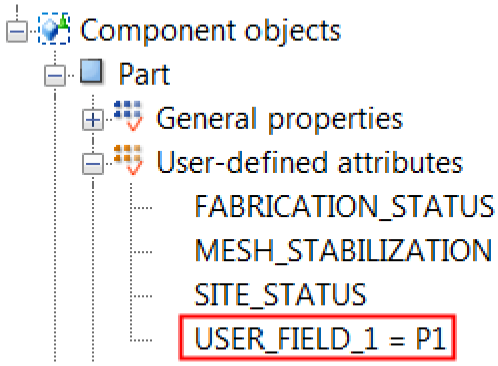
3. Modify the variable as follows:
   a. In the **Value type** list, select **Text**.
   b. In the **Formula** box, enter `Type1`.
   c. In the **Label in dialog box** box, enter `Panel1`.
4. In the **Custom component browser**, browse for the user-defined attributes of the first panel.

   You will link the **P1** variable to the **USER_FIELD_1** attribute. However, the attribute is not visible in the **Custom component browser**.
5. Make the user-defined attribute visible in the **Custom component browser**:
   a. Double-click the first panel.

      The panel properties open in the property pane.
   b. Click **More**.

      The dialog box for user-defined attributes opens.
   c. Go to the **Parameters** tab.
   d. Enter text in the **User field 1** box.
   e. Click **Modify**.
6. In the **Custom component browser**, click **Refresh**.

   USER_FIELD_1 appears under **User-defined attributes** in the **Custom component browser**.
7. Link **P1** to **USER_FIELD_1**.
   a. Right-click **USER_FIELD_1** and select **Add Equation**.
   b. Enter `P1` after the equal sign, and then press **Enter**.



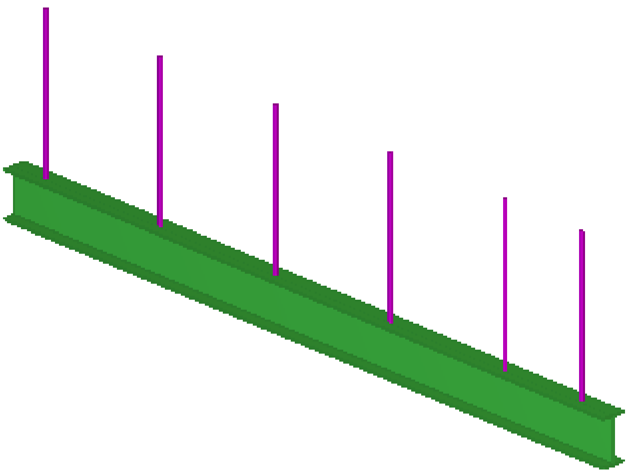
8. Create two new parametric variables and link them to the user-defined attributes of the other two panels.
9. Save the custom component.

10. Close the custom component editor.

You can now create a view filter (page 153) in the model to hide or show panels using the **User field 1** attribute and the **Formula** values you entered for the parametric variables in the filter.

Example of a variable formula: Calculate the number of handrail posts using a template attribute

This example shows how to create a variable formula that calculates the number of handrail posts based on the length template attribute of the beam. The handrail posts were created at both ends of the beam and one of them was copied with the **Array of objects (29)** component.



1. On the **Custom component editor** toolbar, click the **Display variables**  button.

   The **Variables** dialog box opens.
2. Create three new parametric variables by clicking **Add**.
3. Modify the variable **P1** as follows:
   - In the **Formula** box, enter `250`.
   - In the **Label in dialog box** box, enter `End Distance`.
4. Modify the variable **P2** as follows:
   - In the **Formula** box, enter `900`.
   - In the **Label in dialog box** box, enter `Spacing`.
5. Modify the variable **P3** as follows:
   - In the **Value type** box, select **Number**.
   - In the **Label in dialog box** box, enter `Number of Posts`.

6. Inquire the GUID of the beam:

   a. On the ribbon, click **Inquire objects** .

   b. Select the beam.

   c. Check the GUID of the beam in the **Inquire Object** dialog box.

7. In the **Formula** box of the **P3** variable, enter `=(fTpl("LENGTH","ID50B8559A-0000-010B-3133-353432373038")-(P1*2))/P2`.

   `fTpl("LENGTH","ID50B8559A-0000-010B-3133-353432373038")` is the length template attribute of the beam and `ID50B8559A-0000-010B-3133-353432373038` is the GUID of the beam.

   The number of the posts is calculated as follows: First the end distances are subtracted from the beam length, and then the result is divided by the post spacing.

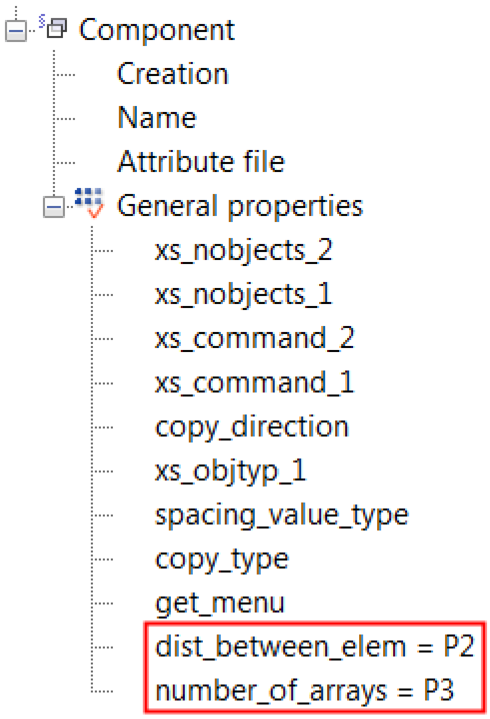
8. In the **Custom component editor**, link the variables **P2** and **P3** to the properties of **Array of objects (29)**.

   a. Right-click **dist_between_elem** and select **Add Equation**.

   b. Enter `P2` after the equal sign, and then press **Enter**.

   c. Right-click **number_of_arrays** and select **Add Equation**.

   d. Enter `P3` after the equal sign, and then press **Enter**.

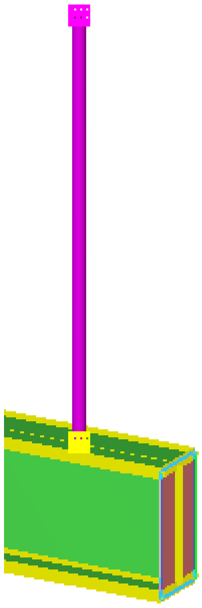


9. Bind the first post to the beam end.

a. Select the post in the custom component view.

b. Hold down **Alt** and use area selection (from left to right) to select the post handles.

c. Right-click and select **Bind to plane**.



10. Bind the last post to the other beam end by following the instructions in step 9.
11. Modify the distance variables as follows:

a. In the **Formula** box, enter `=P1`.

b. In the **Visibility** list, select **Hide**.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	250.00	250.00	Length	Parameter	Show	End Distance
P2	900.00	900.00	Length	Parameter	Show	Spacing
P3	=(fTpl(...	4	Number	Parameter	Show	Number of Posts
D1	=P1	250.00	Length	Distance	Hide	D1.COLUMN.BEAM
D2	=P1	250.00	Length	Distance	Hide	D2.COLUMN.BEAM
D3	=P1	250.00	Length	Distance	Hide	D3.COLUMN.BEAM
D4	=P1	250.00	Length	Distance	Hide	D4.COLUMN.BEAM

12. Save the custom component.
13. Close the custom component editor.

You can now change the spacing and the end distance of the handrail posts in the custom component dialog box. Tekla Structures calculates the number of posts based on the spacing, end distance, and length of the beam.

Example of a variable formula: Link an Excel spreadsheet to a custom component

This example shows how to link a parametric variable to an Excel spreadsheet. For example, you can use Excel spreadsheets to check connections.

1. Create an Excel spreadsheet.

   The name of the spreadsheet file must be `component_"component_name".xls`. For example, `component_stiffener.xls` for a custom component whose name is stiffener.

2. Save the Excel spreadsheet in the model folder: `..\<model>\exceldesign\`.

   Alternatively, you can save the spreadsheet in the folder defined with the `XS_EXTERNAL_EXCEL_DESIGN_PATH` advanced option.

3. On the **Custom component editor** toolbar, click the **Display variables**  button.

   The **Variables** dialog box opens.

4. Click **Add** to create a new parametric variable.

5. Modify the variable as follows:

   a. In the **Value type** list, select **Yes/No**.

   b. In the **Name** box, enter `use_externaldesign`.

   c. In the **Label in dialog box** box, enter `Use external design`.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
use_externaldesign	0	0	Yes/No	Parameter	Show	Use external design

6. Save the custom component.

7. Close the custom component editor.

The custom component dialog box now contains the **Use external design** option.

Examples of a variable formula: Rebar set modifiers in custom components

You can use rebar set modifiers in custom components. You can define the rebar set bar and modifier properties using parametric variables.

For each modifier property that you want to parametrize, the corresponding **Apply** property is also needed. By using the **Apply** property you can override an existing property value with an empty value. Clearing an existing value is not possible without the **Apply** property.

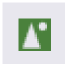
Example: Define the class and size of rebar set bars using a property modifier

This example shows how to use a rebar set property modifier for defining the class and size of certain rebar set bars in a custom component. You will define the class and size using parametric variables and modifier-specific **Apply** properties.

The parametric variable for class will be defined so that if the class is set to 0, the class value is not applied, but the original class of the rebar set is used.

1. In the model, select a previously created custom component (page 830) that contains a rebar set and a property modifier.

   NOTE Custom parts do not have a component symbol in the model.

   To select custom components, ensure that the **Select components**  selection switch is active.

   ---

2. Right-click and select **Edit Custom Component**.
3. On the **Custom component editor** toolbar, click the **Display variables** button .

   The **Variables** dialog box opens.
4. In the **Variables** dialog box, create and define parametric variables as follows:

   a. Click **Add** three times to create three new parametric variables.

      The variable names will be **P1**, **P2**, and **P3**.

   b. Modify the variable **P1** for class number input as follows:

      - In the **Value type** list, select **Number**.
      - In the **Label in dialog box** box, enter `Class`.

   c. Modify the variable **P2** for **Apply** property control as follows:

      - In the **Formula** box, enter `=if (P1==0) then 0 else 1 endif`.

        This means that if **P1** (**Class**) is set to 0, the class property is not applied when you use the custom component. If **P1** is set to any other value, then the class property is applied.
      - In the **Value type** list, select **Yes/No**.

- In the **Visibility** list, select **Hide**.

  This means that the variable **P2** will not be visible in the custom component dialog box.

- In the **Label in dialog box** box, enter `Apply class`.

d. Modify the variable **P3** for bar size input as follows:

- In the **Name** box, change the name to `P3_size`.
- In the **Value type** list, select **Rebar size**.
- In the **Label in dialog box** box, enter `Bar size`.
- In the **Formula** box, replace zero with a valid bar size value.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	0	0	Number	Parameter	Show	Class
P2	=if (P1==0) then 0 else 1 endif	0	Yes/No	Parameter	Hide	Apply class
P3_size	12	12	Rebar size	Parameter	Show	Bar size

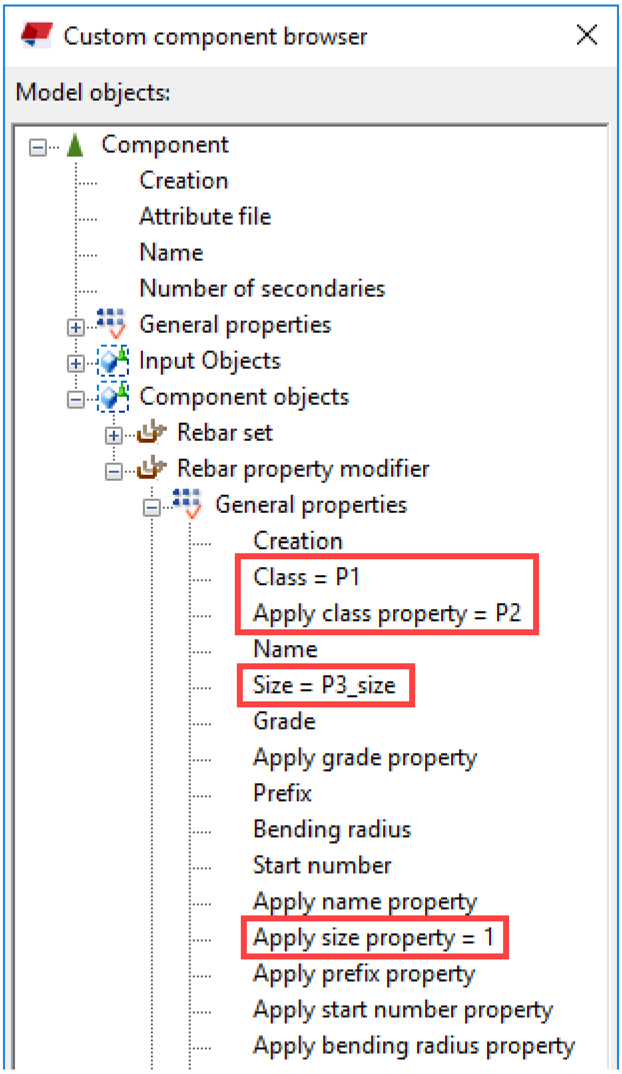
5. In the **Custom component browser**, link the parametric variables to the property modifier properties:

   a. Browse for **Component objects** --> **Rebar property modifier** --> **General properties** .

   b. Right-click **Class**, select **Add Equation**, enter `P1` after the equal sign (=), and then press **Enter**.

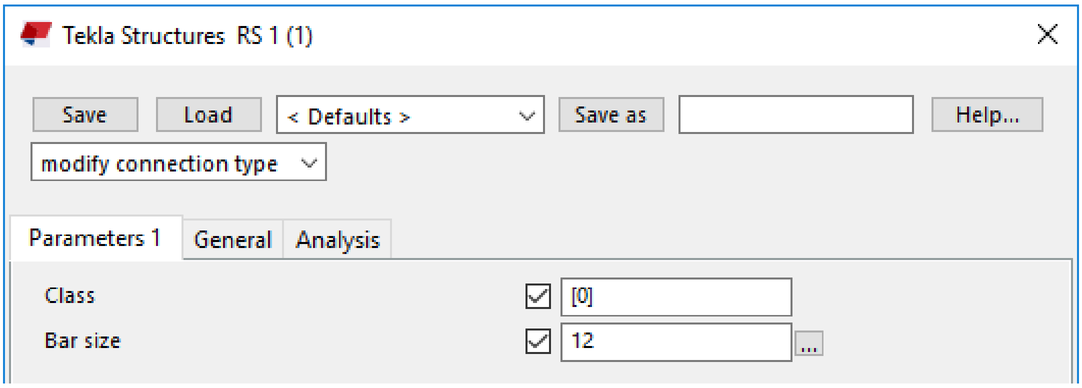
   Similarly, link the other variables and properties as follows:

   - **Apply class property** = `P2`
   - **Size** = `P3_size`
   - **Apply size property** = `1`



6. Save and close (page 844) the modified custom component.

Now you have the following properties available in the custom component's dialog box and you can modify the class and size of those rebar set bars that are affected by the property modifier:



You can use the component in locations similar to where the component was originally created. This component is not adaptive and Tekla Structures does not adjust the component dimensions to suit any changes in the model. To make the custom component adaptive, you need to modify (page 840) it in the custom component editor.

Example: Create and modify rebar hooks using an end detail modifier

This example shows how to use a rebar set end detail modifier for creating hooks at certain rebar set bar ends in a custom component. You will define the hook properties using parametric variables and modifier-specific **Apply** properties.

1. In the model, select a previously created custom component (page 830) that contains a rebar set and an end detail modifier.

    NOTE Custom parts do not have a component symbol in the model.

    To select custom components, ensure that the **Select components**  selection switch is active.

    ---

2. Right-click and select **Edit Custom Component**.
3. On the **Custom component editor** toolbar, click the **Display variables** button .

    The **Variables** dialog box opens.

4. In the **Variables** dialog box, create and define parametric variables as follows:

    a. Click **Add** four times to create four new parametric variables.

       The variable names will be **P1**, **P2**, **P3**, and **P4**.

    b. Modify the variable **P1** for hook type input as follows:

- In the **Value type** list, select **Rebar hook type**.
- In the **Label in dialog box** box, enter `Hook type`.
- In the **Formula** box, enter `4` for a custom hook.

  The different hook types are identified with numbers: 1 = 90 degree hook, 2 = 135 degree hook, 3 = 180 degree hook, 4 = custom hook.

c. Modify the variable **P2** for hook angle input as follows:

- In the **Value type** list, select **Number**.

  Note that although **Angle** is available as a value type, the **Number** option must be used for the hook angle.

- In the **Label in dialog box** box, enter `Custom hook angle`.

d. Modify the variable **P3** for hook length input as follows:

- In the **Value type** list, select **Number**.
- In the **Label in dialog box** box, enter `Custom hook length`.

e. Modify the variable **P4** for hook radius input as follows:

- In the **Value type** list, select **Number**.
- In the **Label in dialog box** box, enter `Custom hook radius`.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	4	4	Rebar hook type	Parameter	Show	Hook type
P2	0	0	Number	Parameter	Show	Custom hook angle
P3	0	0	Number	Parameter	Show	Custom hook length
P4	0	0	Number	Parameter	Show	Custom hook radius

5. In the **Custom component browser**, link the parametric variables to the end detail modifier properties:

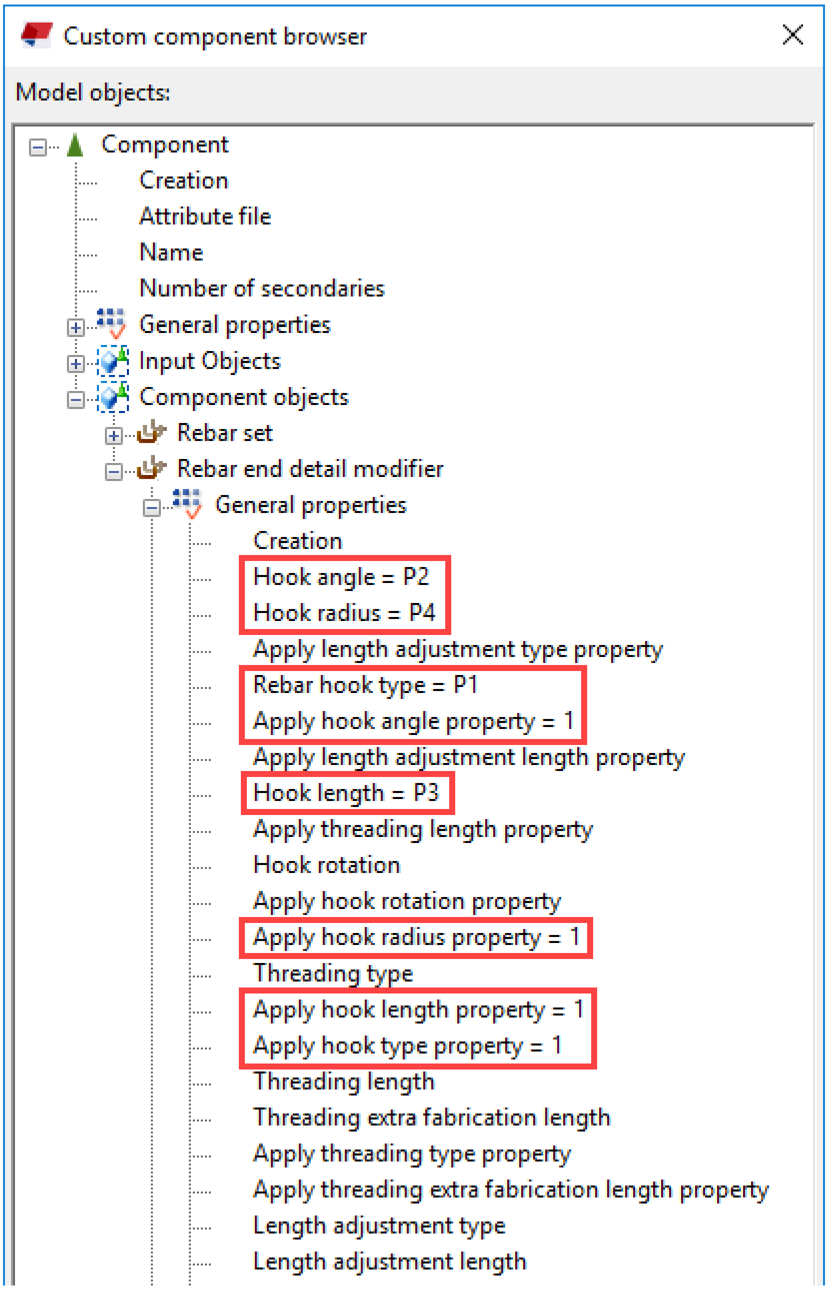
a. Browse for **Component objects** --> **Rebar end detail modifier** --> **General properties**.

b. Right-click **Hook angle**, select **Add Equation**, enter `P2` after the equal sign (=), and then press **Enter**.

   Similarly, link the other variables and properties as follows:

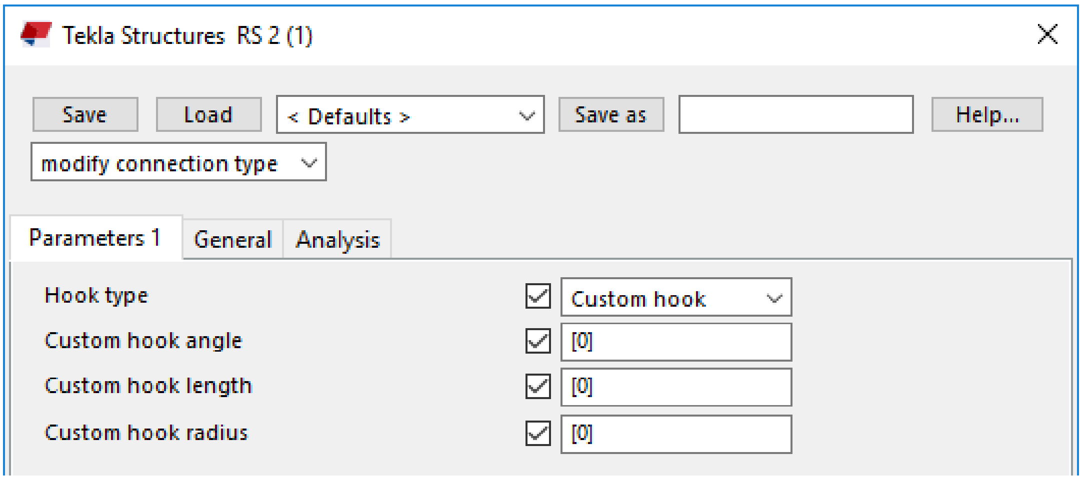
- **Hook radius** = `P4`
- **Rebar hook type** = `P1`
- **Apply hook angle property** = `1`
- **Hook length** = `P3`
- **Apply hook radius property** = `1`
- **Apply hook length property** = `1`

- **Apply hook type property** = 1



6. Save and close (page 844) the modified custom component.

Now you have the following properties available in the custom component's dialog box and you can modify the hooks of those rebar set bars that are affected by the end detail modifier:



You can use the component in locations similar to where the component was originally created. This component is not adaptive and Tekla Structures does not adjust the component dimensions to suit any changes in the model. To make the custom component adaptive, you need to modify (page 840) it in the custom component editor.

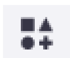
8.10 Import and export custom components

You can import and export custom components as `.uel` files between models.

NOTE You can share your custom components in Tekla Warehouse, and also download custom components made by other users.

Export a custom component

You can export custom components in a `.uel` file.

1. Click the **Applications & components**  button in the side pane to open the **Applications & components** catalog.
2. In the catalog, select the custom components you want to export.
3. Right-click the selection, and then select **Publish**.
4. Browse for the folder where you want to save the file.
5. Enter a name for the export file.

   The file name extension is `.uel`. Do not change the file name after exporting the custom component. If the file name differs from the name in the **Applications & components** catalog, it may be difficult to find the right component later on.

The data files that are used by the custom component are included in the exported `.uel` file. The data files need to be located in the model folder or in the `CustomComponentDialogFiles` subfolder to be exported. Only files specified directly in `fVF` functions are exported. For example: `=fVF("myData.dat", ...)` directly specifies the file, but `=fVF(P1, ...)` does not.

6. Click **Save** to export the custom components.

> **TIP** If you want to export custom components as separate files, select the custom components in the **Applications & components** catalog, right-click, and then select **Publish separately**.

Import a custom component

You can import previously made custom components to another model.

1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
2. Click the **Access advanced features** button , and then select **Import**.
3. Browse for the folder that contains the export file.

   The location depends on where you saved the file when you exported the custom component.
4. Select the export file.
5. Click **Open** to import the custom components.

> **TIP** You can import custom components to a new model automatically by using the `XS_UEL_IMPORT_FOLDER` advanced option. Export all custom components to certain folders and enter these folders as the value for the `XS_UEL_IMPORT_FOLDER` advanced option to easily import the custom components to new models.

8.11 Hints and tips for using and sharing custom components

Here you will find some useful tips on how to create and use custom components more efficiently.

Tips for creating custom components

- **Enter short, logical names for custom components.**

  Use the description field to describe the component and to explain what it does.

- **Create simple components for specific situations.**

  Simple components are easier and faster to model, and also much easier to use. Avoid creating a single, complex component which you will use for every possible purpose.

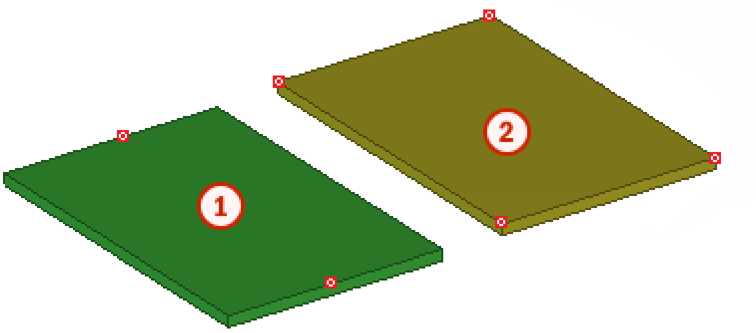
- **Consider creating a separate component model.**

  Use that model when you create and test custom components.

- **Use the simplest part you can with as few handles as possible.**

  For example, if all you need is a rectangular shape, use a rectangular plate, not a contour plate. Rectangular plates only have two handles, so you only need to create a few bindings to manipulate them. Contour plates require more bindings because they have four handles.

  Excessive bindings can cause a slower, less responsive model.

  1. Rectangular plate
  2. Contour plate

- **Model parts only as accurately as you need.**

  If the only part information required is a part mark in a general arrangement drawing, plus a quantity on a materials list, create a simple bar or plate. If you need to include the part in a detailed view later on, simply re-model the part more accurately at that point.

- **Model embeds as custom parts and include them in components.**

  The majority of embeds are standard embeds, which are designed and manufactured as mass production, and stored to be readily available at the factory. The other type of embeds is a bespoke embed, which is manufactured for a specific project, and needs its own workshop drawing, and which are quantified more accurately.

To have suitable embeds in you model, you can either

- create your own embeds as custom parts
- create your own embeds as items (page 300)
- use Tekla Warehouse to find embeds.

If you create your own embed custom components, be careful with the sub-assembly hierarchy. It is better to have a clear sub-assembly main part and then the other parts added to the main part.

- **You can use custom connections in AutoConnections**
- **You can use custom components in AutoDefaults**

Custom components are listed in the **Other components** group. Newly created custom components are not listed in the **Other components** group until you have reopened the model. Custom connections, details and seams work in AutoDefaults.

Tips for sharing custom components

- **Use Tekla Warehouse to share and store custom components.**
- **Provide essential information.**

  If you distribute your component to other users, remember to list the profiles it works with.
- **Use fixed profiles (page 320) whenever possible.**
- **If your custom component contains user-defined profile cross sections, remember to include them when you copy the custom component to a new location.**

Tips for updating custom components to a new version

When you start using a new version of Tekla Structures, always check that custom components created in older versions work correctly in the new version.

When you edit custom components created with an older version of Tekla Structures, and the new version contains improvements that require an update, Tekla Structures asks whether you want to update the component. If you do not update the component, it works in the same manner as in the version where it was originally created, but you do not gain the benefits of the new improvements.

If you choose to update the component, you need to check and sometimes recreate dimensions depending on the improvements. When you delete a dimension and create a new one (even with the same name), the equations that contain the dimension also need to be modified, because the dependency created by the equation is lost when a dimension is deleted. You can recreate dimensions and modify equations in the custom component editor.

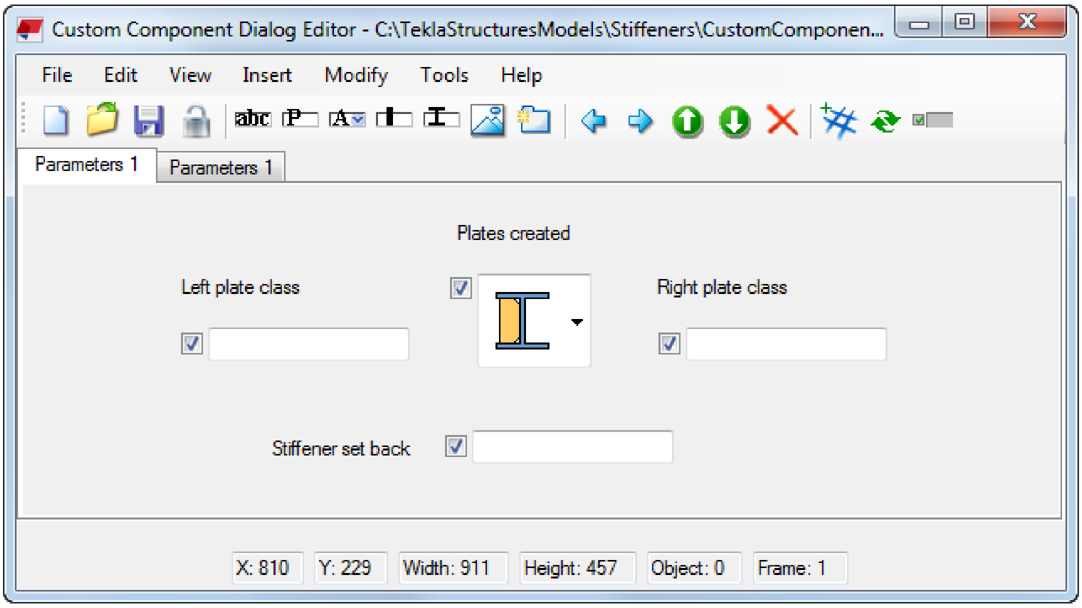
8.12 Customize the dialog boxes of custom components

Tekla Structures automatically creates a dialog box for each custom component you define. Each custom component has an input (`.inp`) file that defines the contents of the custom component's dialog box. You can customize the dialog box by using the **Custom Component Dialog Editor** tool.

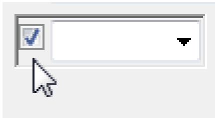
Alternatively, if you are an advanced user, you can modify the dialog box input (`.inp`) files manually (page 923) in a text editor.

Modify the dialog box of a custom component

To open the **Custom Component Dialog Editor**, select a custom component in the model, right-click, and select **Edit Custom Component Dialog Box**.



To	Do this
View and edit object properties	1. Select a dialog box element. For example, a text box. 2. Click **Modify** --> **Properties** . Now you can view and edit the current properties of the dialog box element. For example, you can check that you have the

To	Do this
	correct text box under each label in the dialog box. Alternatively, you can double-click the dialog box element. If the dialog box element will not open for viewing and editing, try double-clicking the space right underneath the check box: 
Add a dialog box element	Click **Insert** and select a suitable element from the list. The options are: • **Tab Page**: add a new tab • **Label**: add a label for a text box or list • **Parameter**: add a text box • **Attribute**: add a list • **Part**: add some basic part properties • **Profile**: add some basic profile properties • **Picture**: add an illustrative image of the custom component
Add an image	1. Click **Insert** --> **Picture** to show the contents of the **Image Folder** set in **Tools** --> **Options** . 2. Select an image. The image must be in the bitmap (`.bmp`) format. 3. Click **Open**. 4. Drag the image to the desired location.
Add a tab	1. Click **Insert** --> **Tab Page** . 2. Double-click the new tab. 3. Enter a new name, and then press **Enter**.

To	Do this
	NOTE Each tab may contain up to 25 fields. If you have more than 25 fields visible, Tekla Structures automatically creates another tab.
Show or hide the pixel grid	Click . Tekla Structures displays a pixel grid that makes it easier to align elements in the dialog box.
Move a dialog box element	Drag the dialog box element to a new location. You can also use the keyboard shortcuts **Ctrl+X** (cut), **Ctrl+C** (copy), and **Ctrl+V** (paste). For example, to move a dialog box element to another tab: select the dialog box element, press **Ctrl+X**, go to another tab, and press **Ctrl+V**.
Select multiple dialog box elements	Hold down the **Ctrl** key and click the dialog box elements, or use area selection.
Rename a tab or text box label	1. Double-click the tab or text box label. 2. Type a new name. 3. Press **Enter**.
Remove a dialog box element	1. Select the dialog box element you want to remove. 2. Press **Delete**.
Remove a tab	1. Select the tab. 2. Right-click and select **Delete**.
Add images to a list	1. Select the list element. 2. Click **Modify** --> **Properties** . 3. Click **Edit Values**. 4. Click **Browse Add**. 5. Select the image you want to use and click **Open**.

To	Do this
	6. Repeat steps 4–5 for any other images you want to use. 7. Click **OK** to save the changes.
Save the changes	Click **File** --> **Save** .

Custom component input files

Each custom component has an input (`.inp`) file that defines the contents of the custom component's dialog box.

When you create a new custom component, Tekla Structures automatically creates an input file for the component. The input file is located in the `\CustomComponentDialogFiles` folder under the model folder. The input file has the same name as the custom component, and the file name extension is `.inp`.

When you modify a custom component (page 840), you will lose any changes you have made to the input file. However, when you modify the custom component, Tekla Structures automatically creates a backup copy of the input file. The backup copy has the file name extension `.inp_bak`, and it is located in the `\CustomComponentDialogFiles` folder under the model folder. Tekla Structures displays a notification when the backup file is created.

Lock or unlock the custom component input file

You can lock the custom component's input (`.inp`) file to prevent accidental modifications. If the file is unlocked, and someone else updates the custom component in the custom component editor, all your modifications to the dialog box will be lost.

1. In the model, select the custom component whose input file you want to lock or unlock.
2. Right-click and select **Edit Custom Component Dialog Box**.
3. In the **Custom Component Dialog Editor**, click the **Lock/Unlock** button .

If someone modifies the custom component in the custom component editor when the `.inp` file is locked, the `.inp` file will not be updated. You can still modify the dialog box in the **Custom Component Dialog Editor** even if the `.inp` file is locked.

Custom component dialog editor settings

In the **Custom Component Dialog Editor**, click **Tools** --> **Options** to view and modify basic settings in the custom component dialog editor. Click **Tools** --> **Change Language** to change the language of the custom component dialog editor.

Option	Description
Image folder	The location of the image folder. To restore the default folder setting, click **Default**.
Project folder	The location of the project folder. When you create a completely new input file by clicking **File** --> **New** and then save it, the file is saved in the project folder. Note that existing input files are saved under the model folder.
Parameter width	The default width for text boxes.
Attribute width	The default width for lists.
Grid spacing X **Grid spacing Y**	The spacing of the pixel grid in the X and Y directions. The default value is `5`.
Snap to grid	Select to show or hide the pixel grid.

Option	Description
Language	Select a language from the list. Close and reopen the dialog editor for the change to take effect. You have the following options: • **Auto**: the dialog editor follows the language of the Tekla Structures user interface • **English** • **Dutch** • **French** • **German** • **Italian** • **Spanish** • **Japanese** • **Chinese Simplified** • **Chinese Traditional**

Option	Description
	• **Czech** • **Portuguese Brazilian** • **Hungarian** • **Polish** • **Russian** • **Korean**

Customize the dialog boxes of custom components by using a text editor

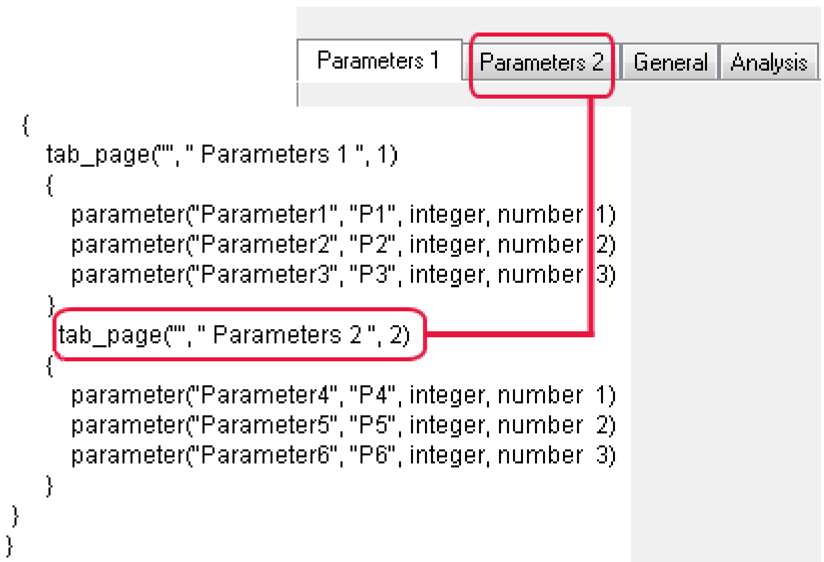
Each custom component has an input (`.inp`) file that defines the contents of the custom component's dialog box. If you are an advanced user, you can customize the dialog box input files manually in a text editor. Be careful when modifying an input file, as errors may cause the dialog box to disappear.

Note that the **General** tab is reserved for predefined general properties. You cannot rename the **General** tab or add more parameters to it.

Alternatively, you can use the **Custom Component Dialog Editor** tool to customize the dialog box (page 918).

Add new tabs

1. Open the `.inp` file in a text editor.
2. Add a new tab definition, as shown below:

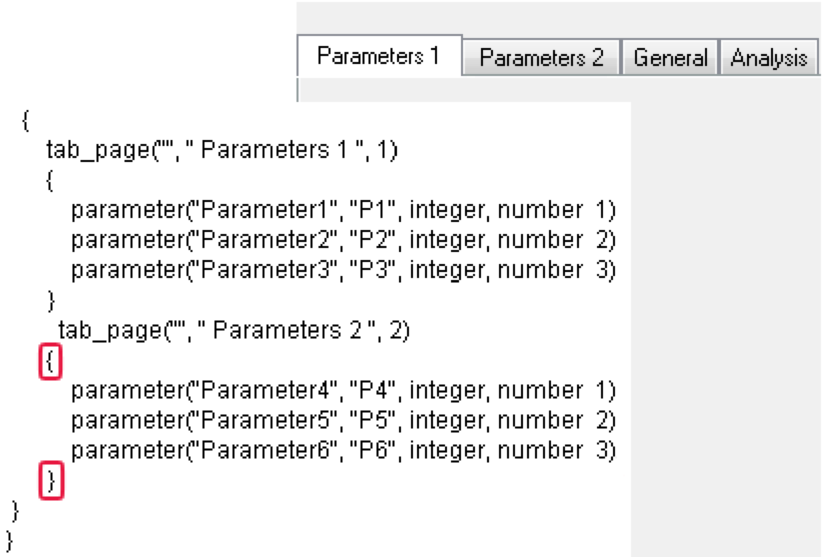


3. Save the `.inp` file.

NOTE The fourth tab is reserved for the **General** properties, so you cannot add your own parameters to it.

Add text boxes

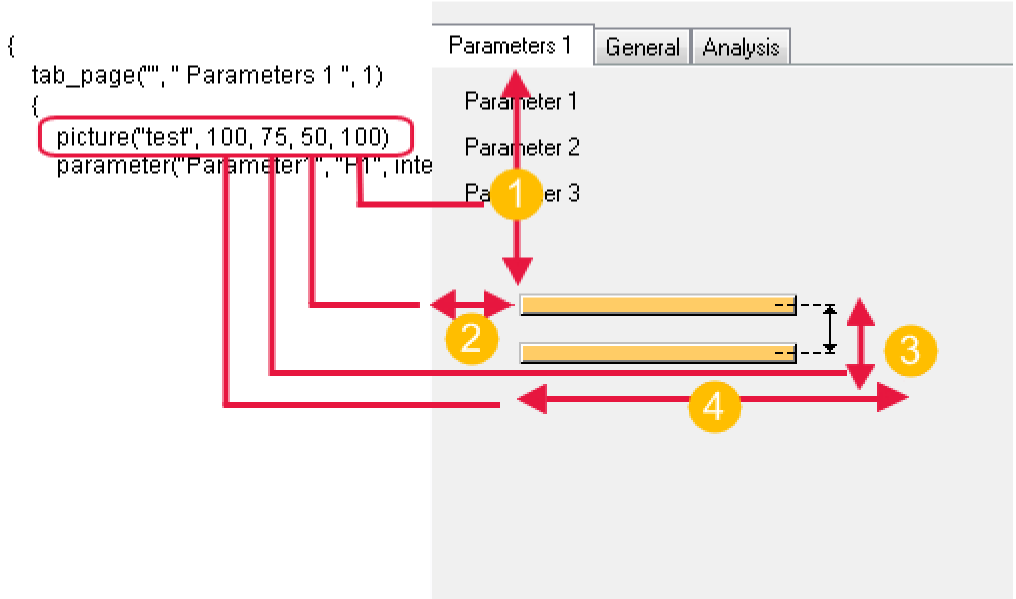
1. Open the .inp file in a text editor.
2. Add parameter elements and enclose them in curly brackets, as shown below:



3. Save the .inp file.

Add images

1. Create an image and save it in bitmap (.bmp) format in the ..\ProgramData\Trimble\Tekla Structures\<version>\Bitmaps folder.
2. Open the .inp file in a text editor.
3. Add an image definition, as shown below:



(1) y = 100

(2) x = 50

(3) height = 75

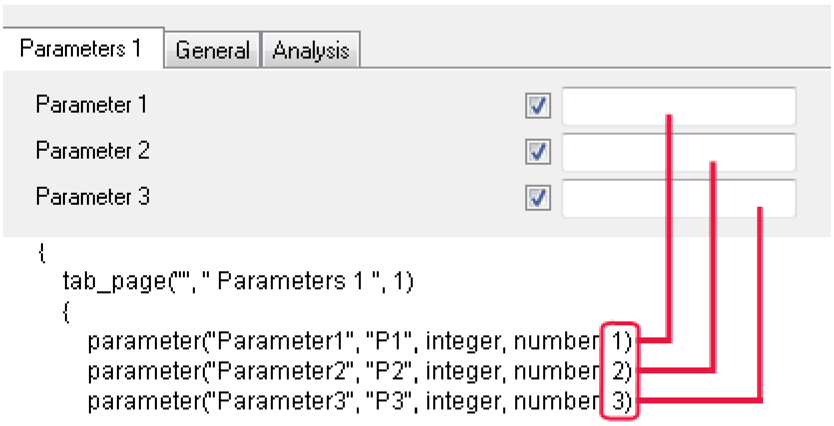
(4) width = 100

4. Save the `.inp` file.

Change the order of boxes

1. Open the `.inp` file in a text editor.
2. Change the last number in the parameter definition.

   The boxes are listed from top to bottom, as shown below:



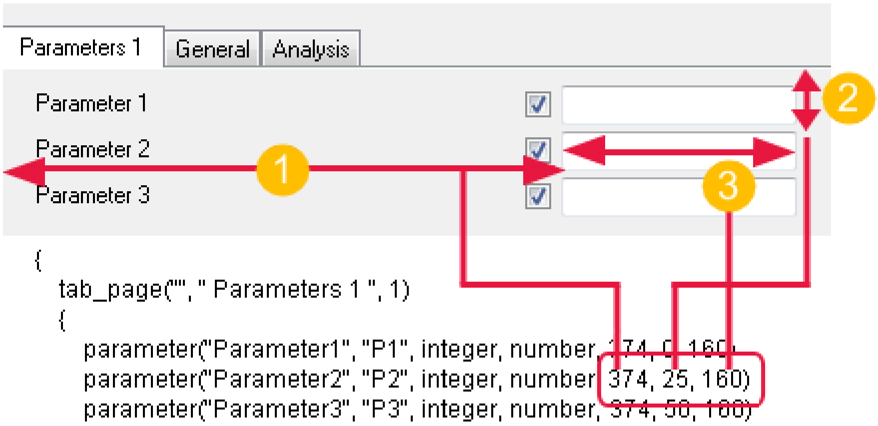
3. Save the `.inp` file.

Change the location of boxes

You can define an exact location for each text box.

1. Open the `.inp` file in a text editor.
2. Define the exact location of the box using three values: the x coordinate, y coordinate, and width of the box.

   For example:

   (1) x = 374

   (2) y = 25

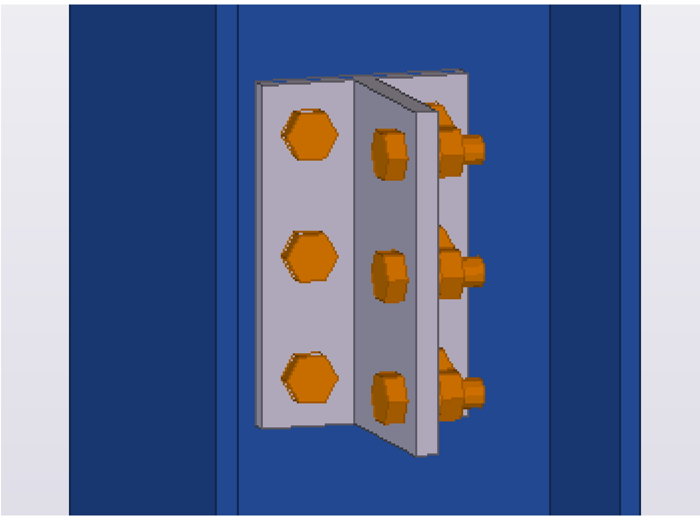
   (3) width = 160
3. Save the `.inp` file.

Example: Add a group of check boxes in the custom component dialog box

This example shows how to add a check box for each bolt group in a custom component by modifying the `.inp` file. When the component is used in a model, you can select which bolts to create by selecting the desired check boxes.

1. Define a custom component (page 830) that contains bolts.

For example, create a custom tee connection that creates one bolt group and three single bolts:



2. Create parametric variables that control the creation of bolts.

   With check box groups, the **Value type** of these variables must be **Yes/No**. For example, create three variables **P1**, **P2**, and **P3**, one for each single bolt in the custom tee connection.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	0	0	Yes/No	Parameter	Show	Parameter1
P2	0	0	Yes/No	Parameter	Show	Parameter2
P3	0	0	Yes/No	Parameter	Show	Parameter3

3. Link the variables (page 864) to the **Creation** property of the bolts.

   For example, link the variable **P1** to the **Creation** property of the first bolt, the variable **P2** to the **Creation** property of the second bolt, and so on.

- Component objects
  - Part
  - Bolt
    - General properties
      - Creation = P1
      - Size
      - Parts with slotted holes
      - Cut length
      - Bolt standard
      - Bolt group distance x
      - Thread in material
      - Extra length

4. Save the custom component.

5. In the model, click **File** --> **Open the model folder** to open the current model folder.
6. Go to the `\CustomComponentDialogFiles` folder.
7. Open the `.inp` file in a text editor.
8. Add an image definition (page 924).

   For example:

```
page("TeklaStructures","")
{
   detail(1, "tee")
   {
       tab_page("", " Parameters 1 ", 1)
       {
   picture("CustomTee",100,100,200,100) /*Bolts*/
       }
   }
}
```

   If you use a custom image, save it in bitmap (`.bmp`) format in the `..\TeklaStructures\<version>\Bitmaps` folder.

9. Add a `toggle_group` element to define the toggle group origin, that is, the position of the group of check boxes in the custom component dialog box.

   Use the x and y coordinate values to define the position. For example:

```
page("TeklaStructures","")
{
   detail(1, "tee")
   {
       tab_page("", " Parameters 1 ", 1)
       {
   picture("CustomTee",100,100,200,100) /*Bolts*/
   toggle_group(200,320,
       )
       }
   }
}
```

   (1) x = 200

   (2) y = 320

10. Within the `toggle_group` element, add a line for each check box you want to add.

Use the same parametric variables that you created in step 2.

```
page("TeklaStructures","")
{
   detail(1, "tee")
   {
       tab_page("", " Parameters 1 ", 1)
       {
   picture("CustomTee",100,100,200,100) /*Bolts*/
   toggle_group(200,320,

     "P1", 160, -165, "0",
     "P2", 160, -135, "0",
     "P3", 160, -105, "0")
       }
   }
}
```

The two numeric values after the variable name are offsets from the toggle group origin. For example, the first definition `"P1", 160, -165, "0"` means that the check box for the variable **P1** is located 160 steps right and 165 steps up from the toggle group origin.

Direction	Negative values	Positive values
X	left	right
Y	up	down

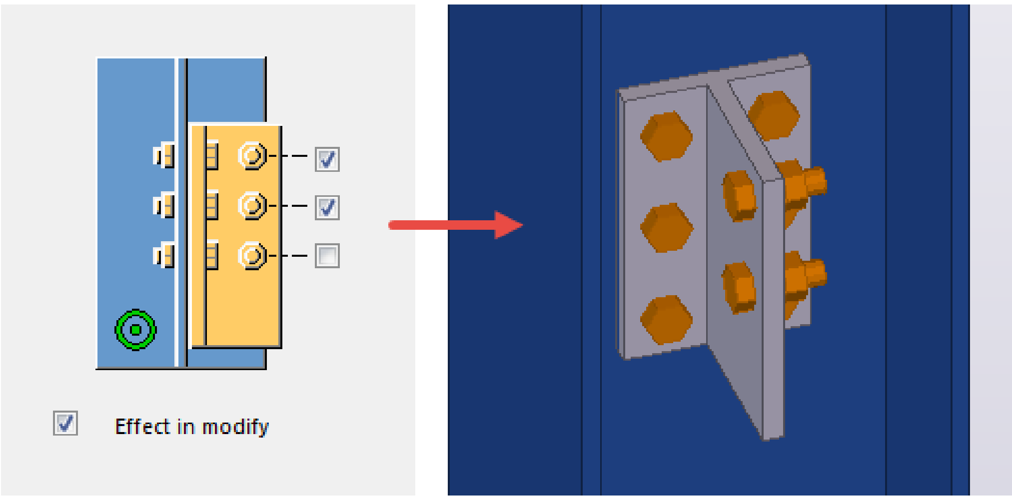
```
page("TeklaStructures","")
{
   detail(1, "tee")
   {
       tab_page("", " Parameters 1 ", 1)
       {
   picture("CustomTee",100,100,200,100) /*Bolts*/
   toggle_group(200,320,

     "P1", 160, -165, "0",
     "P2", 160, -135, "0",
     "P3", 160, -105, "0")
       }
   }
}
```

(1) offset in X direction

(2) offset in Y direction

11. Save the `.inp` file.
12. Close and reopen the model for the change to take effect.

Now when you select and clear check boxes in the dialog box, the number of bolts changes accordingly in the model. For example:

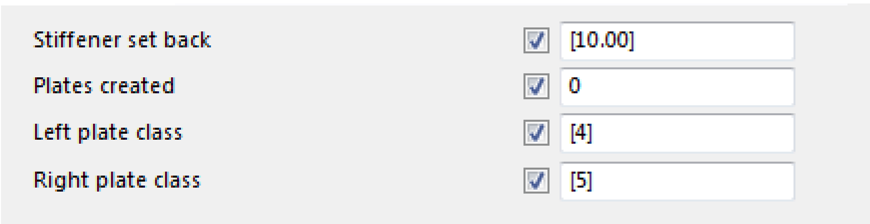


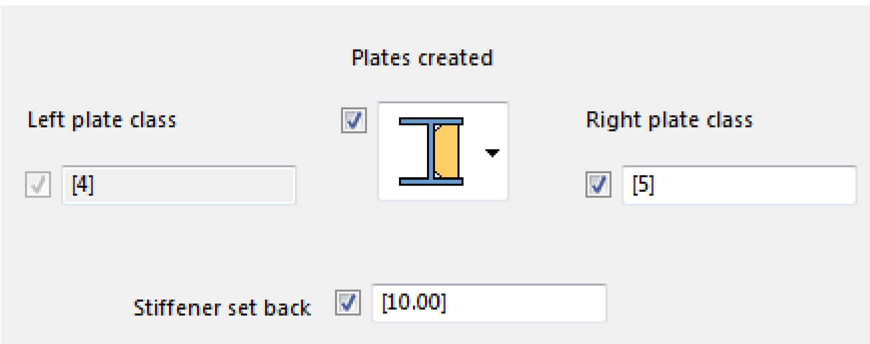
NOTE Tekla Structures automatically adds the **Effect in modify** label and check box for each toggle group you create.

Example: Customize the dialog box of a custom stiffener detail

This example shows how to customize the dialog box of a custom stiffener detail to make it easier to adjust the settings later on.

In the beginning, the custom component dialog box looks like the following:





Do this

1. Create a custom stiffener detail (page 931)with all the needed variables that control the creation of stiffener plates.
2. Add a list with images. (page 942)
3. Arrange text boxes and labels. (page 947)
4. Dim unavailable options. (page 949)

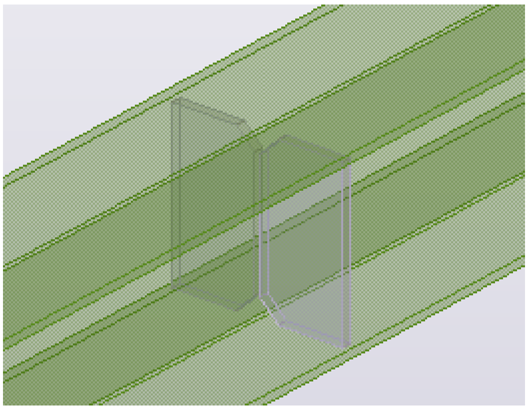
Example: Create a custom stiffener detail with variables

This example shows how to create a custom stiffener detail with variables that control the shape and position of the stiffeners.

Create a basic custom stiffener detail

This example shows how to create a basic stiffener detail.

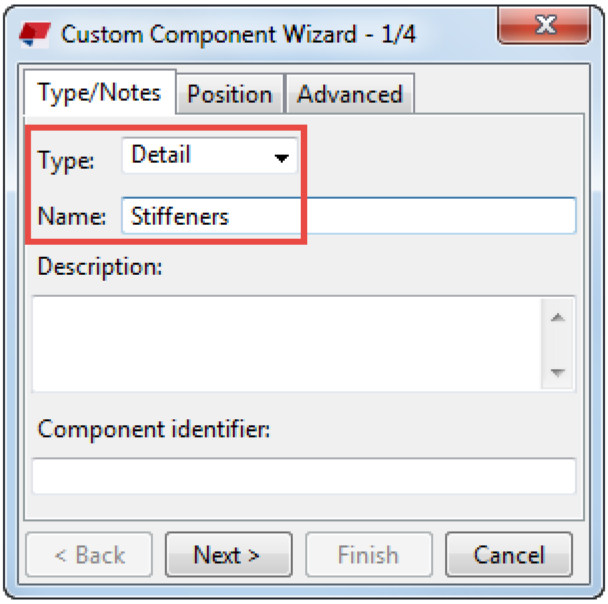
1. Create a beam with two stiffeners.



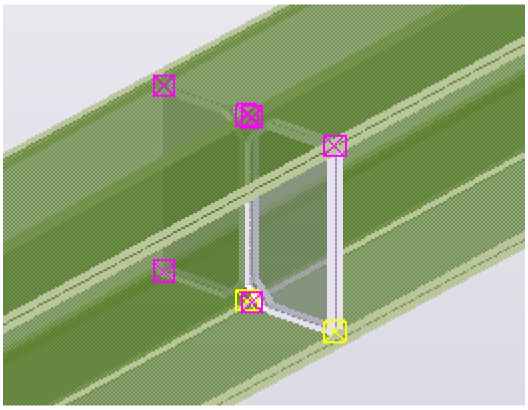
TIP To create the stiffeners, you can use the **Stiffeners (1003)** component and then explode the component.

2. Click the **Applications & components**  button in the side pane to open the **Applications & components** catalog.
3. Click the **Access advanced features**  button and select **Define custom component...**.

   The **Custom Component Wizard** dialog box opens.
4. In the **Type** list, select **Detail**.
5. In the **Name** box, type `Stiffeners`.

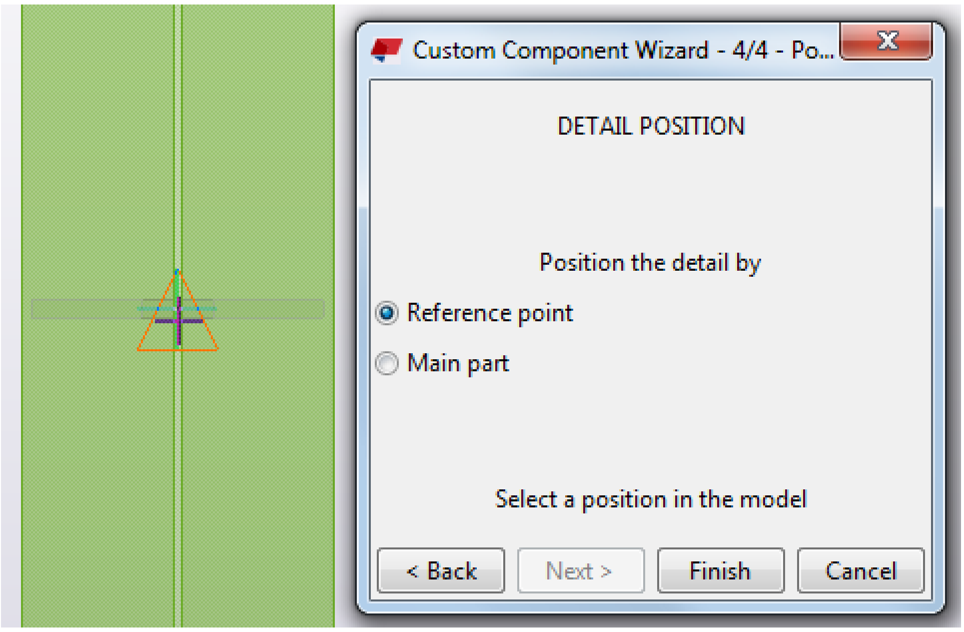


6. Click **Next >**.
7. Select the stiffeners and the beam as the objects that form the custom component.



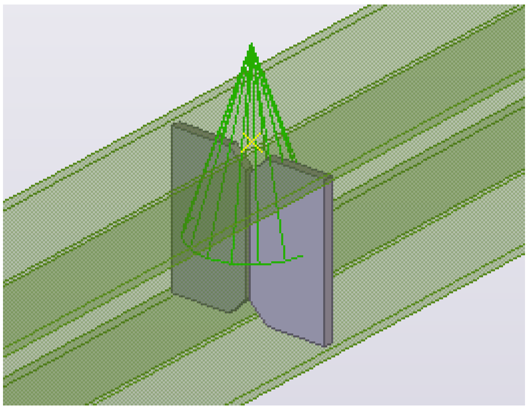
8. Click **Next >**.
9. Select the beam as the main part.
10. Click **Next >**.
11. Select the middle point of the beam as the reference point.

TIP Switch to the plane view (page 45) to select the middle point more easily.



12. Click **Finish** to finish creating the stiffener detail.

    Tekla Structures displays a component symbol for the new custom component and adds the stiffener detail to the component catalog.



Create bindings to control the stiffener shape

This example shows how to bind custom component handles to a plane to control the shape of the stiffeners.

1. Open the stiffener detail in the custom component editor.

    a. Right-click the custom component in the model.

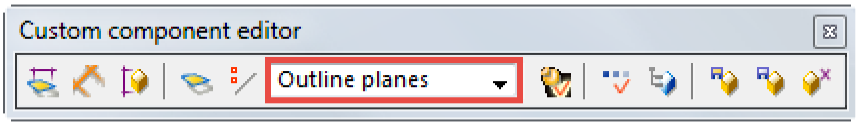
    b. Select **Edit Custom Component**.

    The custom component editor opens showing the custom component editor toolbar, the component browser, and four views of the custom component.

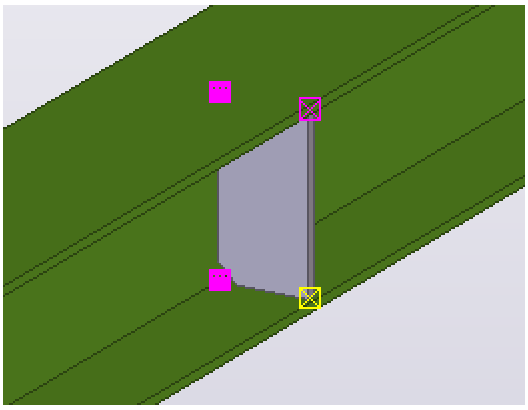
2. On the **View** tab, click **Rendering** --> **Parts rendered**.

   Part surfaces and available planes can be selected only when they are rendered.

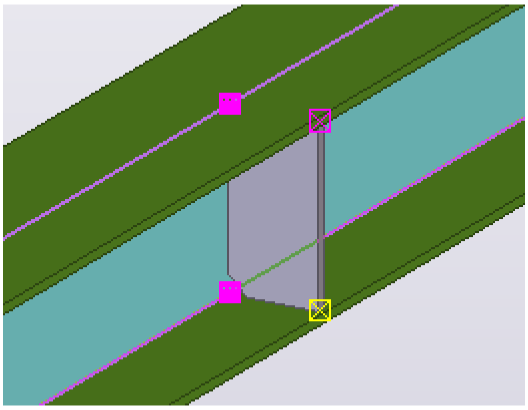
3. On the **Custom component editor** toolbar, select **Outline planes** from the list.

4. In a custom component view, select the stiffener on the right.
5. Bind the two inside handles of the stiffener to the beam web.
   a. Select the two handles next to the beam web.

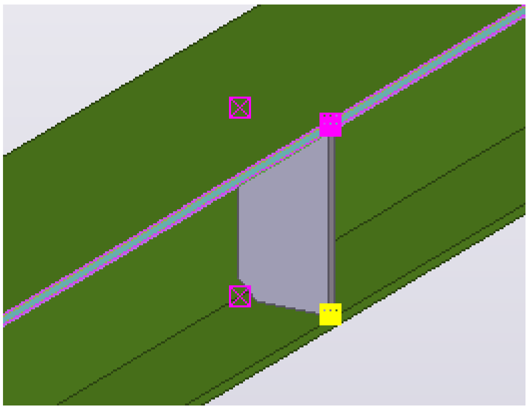
   

   b. Right-click and select **Bind to plane**.
   c. Move the pointer over the face of the web to highlight it.

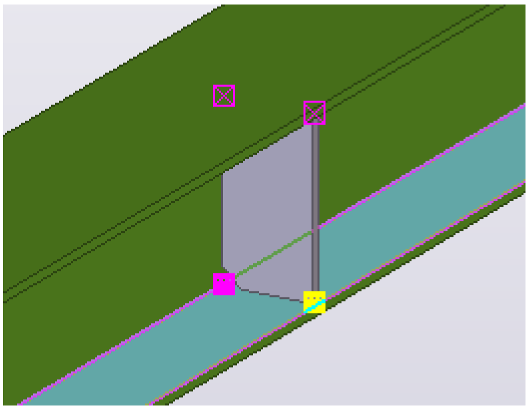
   d. Click the web to bind the handles.
6. Bind the two outside handles of the stiffener to the face of the top flange.

Use the same method as in step 5.



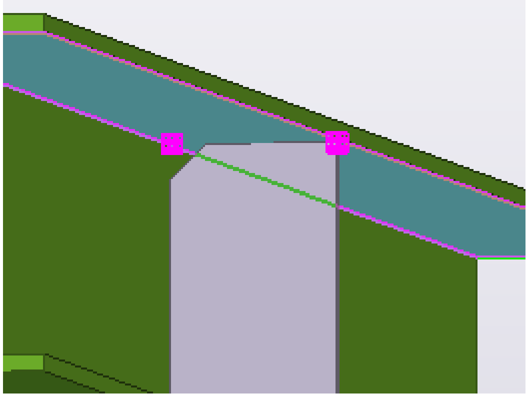
7. Bind the two bottom handles of the stiffener to the inside face of the bottom flange.

   Use the same method as in step 5.



8. Bind the two top handles of the stiffener to the inside face of the top flange.

   Use the same method as in step 5.



9. Repeat steps 4 –11 for the stiffener on the left.

10. On the **Custom component editor** toolbar, click the **Display variables** button .

    The **Variables** dialog box opens.

11. Click **Add** to create a new parametric variable **P1**.
12. Modify the variable **P1** as follows:
    a. In the **Formula** box, enter `10`.
    b. In the **Label in dialog box** box, enter `Stiffener set back`.
13. In the **Formula** box, enter `=P1` for all variables that got values during the binding of the handles.

    For example:

Name	Formula	Value	Value type
D1	0.00	0.00	Length
D2	0.00	0.00	Length
D3	10.00	10.00	Length
D4	10.00	10.00	Length

    The variable **P1** now controls the distances of these variables.

14. In the **Visibility** list, set the variable **P1** to **Show** and the other variables to **Hide**.

You have now created distance variables that control the stiffener shape.

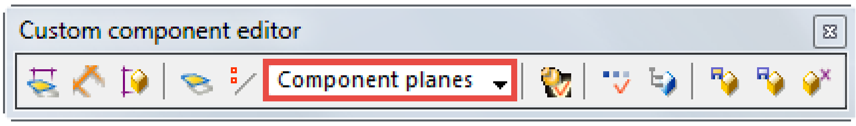
Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
D1	0.00	0.00	Length	Distance	Hide	D1.PLATE.Web right plane
D2	0.00	0.00	Length	Distance	Hide	D2.PLATE.Web right plane
D3	=P1	10.00	Length	Distance	Hide	D3.PLATE.Upper flange right plane
D4	=P1	10.00	Length	Distance	Hide	D4.PLATE.Upper flange right plane
D5	0.00	0.00	Length	Distance	Hide	D5.PLATE.Lower flange top right plane
D6	0.00	0.00	Length	Distance	Hide	D6.PLATE.Lower flange top right plane
D7	0.00	0.00	Length	Distance	Hide	D7.PLATE.Upper flange bottom right plane
D8	0.00	0.00	Length	Distance	Hide	D8.PLATE.Upper flange bottom right plane
D9	0.00	0.00	Length	Distance	Hide	D9.PLATE.Web left plane
D10	0.00	0.00	Length	Distance	Hide	D10.PLATE.Web left plane
D11	=P1	10.00	Length	Distance	Hide	D11.PLATE.Upper flange left plane
D12	=P1	10.00	Length	Distance	Hide	D12.PLATE.Upper flange left plane
D13	0.00	0.00	Length	Distance	Hide	D13.PLATE.Lower flange top left plane
D14	0.00	0.00	Length	Distance	Hide	D14.PLATE.Lower flange top left plane
D15	0.00	0.00	Length	Distance	Hide	D15.PLATE.Upper flange bottom left plane
D16	0.00	0.00	Length	Distance	Hide	D16.PLATE.Upper flange bottom left plane
P1	10.00	10.00	Length	Parameter	Show	Stiffener set back

Create bindings to control the stiffener position

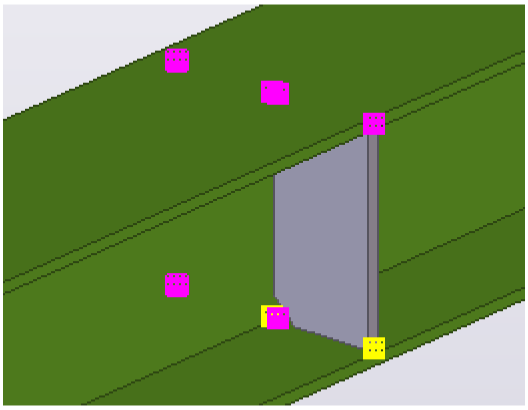
This example shows how to bind custom component handles to a plane to control the position of the stiffeners.

1. Open the stiffener detail in the custom component editor.
    a. Right-click the custom component in the model.
    b. Select **Edit Custom Component**.

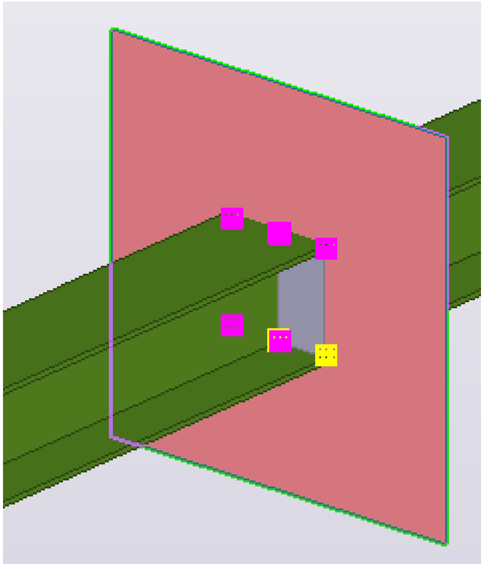
        The custom component editor opens showing the **Custom component editor** toolbar, the **Custom component browser**, and four views of the custom component.
2. On the **Custom component editor** toolbar, select **Component planes** from the list.



3. Select all the handles of both stiffeners.



4. Right-click and select **Bind to plane**.
5. Bind the handles to the vertical component plane.



You have now created distance variables that control the position of the stiffeners.

Create variables to control the stiffener thickness

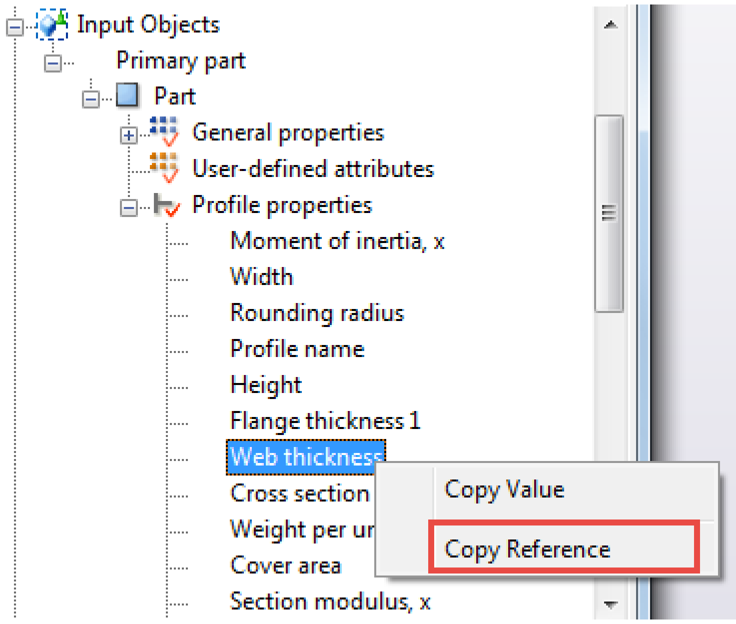
This example shows how to control the stiffener thickness so that it is one and a half times the web thickness rounded up to the nearest available plate thickness. The available thickness values are 10, 12, and 16 mm.

1. Open the stiffener detail in the custom component editor.
    a. Right-click the custom component in the model.
    b. Select **Edit Custom Component**.

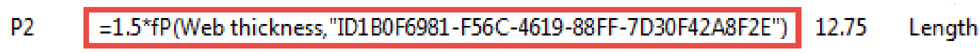
       The custom component editor opens showing the **Custom component editor** toolbar, the **Custom component browser**, and four views of the custom component.
2. On the **Custom component editor** toolbar, click the **Display variables** button .

   The **Variables** dialog box opens.
3. Click **Add** to create a new parametric variable **P2**.
4. Modify the variable **P2** as follows:
    a. In the **Formula** box, enter `=1.5*`.
    b. In the **Visibility** list, select **Hide**.
    c. In the **Label in dialog box** box, enter `Plate calculation`.
5. Select the beam in a custom component view to highlight the beam (main part) in the **Custom component browser**.
6. In the **Custom component browser**, select **Web thickness** of the main part.

7. Right-click and select **Copy Reference**.



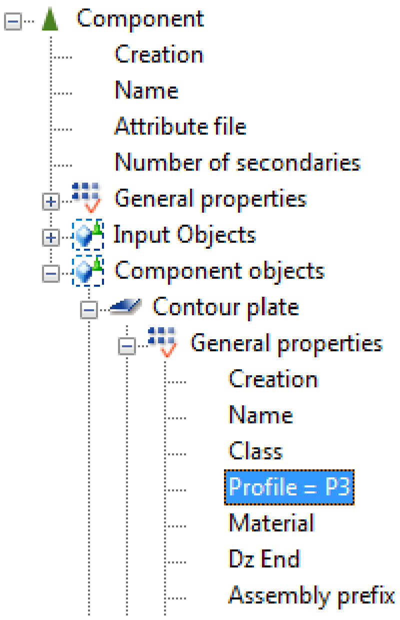
8. Paste the reference value to **Formula** after `=1.5*`.



NOTE A reference function refers to the property of an object, such as the web thickness of a part. If the object property changes, so does the reference function value.

9. Click **Add** to create a new parametric variable **P3**.
10. Modify the variable **P3** as follows:
    a. In the **Value type** list, select **Number**.
    b. In the **Formula** box, enter `=if (P2 < 12 && P2 > 10) then 12 else if (P2 > 12) then 16 else 10 endif endif`.

    This means that if **P2** is less than 12 and greater than 10, the thickness is 12. If **P2** is greater than 12, the thickness is 16. If none of these conditions are met, the thickness is 10.
11. In the **Custom component browser**, link the variable **P3** to the **Profile** property of the first contour plate.



12. Repeat step 11 for the second contour plate.

You have now created and linked all required variables that control the stiffener thickness according to the web thickness.

Create variables to control the creation of stiffener plates

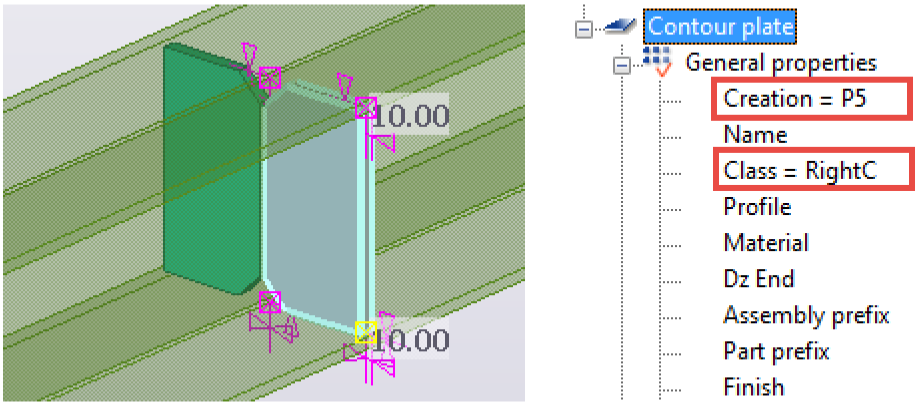
This example shows how to create five variables to control which stiffener plates are created and what is the class of the plates.

1. Open the stiffener detail in the custom component editor.
   a. Right-click the custom component in the model.
   b. Select **Edit Custom Component**.

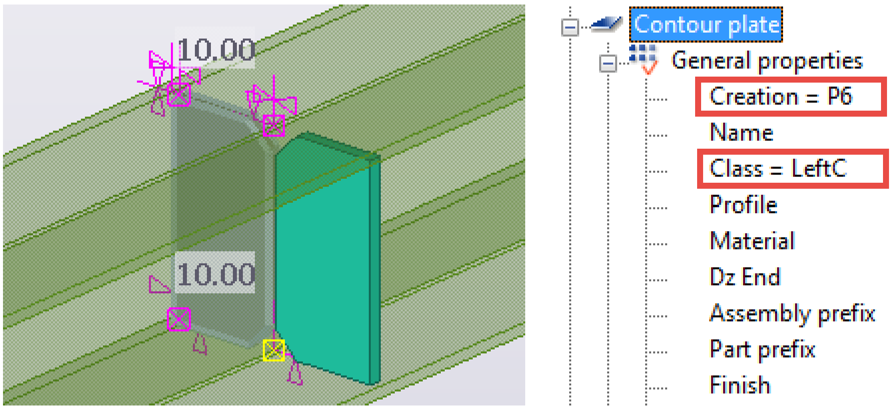
      The custom component editor opens showing the **Custom component editor** toolbar, the **Custom component browser**, and four views of the custom component.
2. On the **Custom component editor** toolbar, click the **Display variables** button .

   The **Variables** dialog box opens.
3. Click **Add** to create a new parametric variable **P4**.
4. Modify the variable **P4** as follows:
   a. In the **Formula** box, enter `2`.
   b. In the **Value type** list, select **Number**.
   c. In the **Visibility** list, select **Show**.
   d. In the **Label in dialog box** box, enter `Plates created`.
5. Click **Add** to create a new parametric variable **P5**.

6. Modify the variable **P5** as follows:
    a. In the **Formula** box, enter `=if P4==0 then 0 else 1 endif`.
    b. In the **Value type** list, select **Yes/No**.
    c. In the **Visibility** list, select **Hide**.
    d. In the **Label in dialog box** box, enter `Do not create right`.
7. Click **Add** to create a new parametric variable **P6**.
8. Modify the variable P6 as follows:
    a. In the **Formula** box, enter `=if P4==1 then 0 else 1 endif`.
    b. In the **Value type** list, select **Yes/No**.
    c. In the **Visibility** list, select **Hide**.
    d. In the **Label in dialog box** box, enter `Do not create left`.
9. Click **Add** to create a new parametric variable P7.
10. Modify the variable **P7** as follows:
    a. Rename **P7** as `LeftC`.
    b. In the **Formula** box, enter `4`.
    c. In the **Value type** list, select **Number**.
    d. In the **Visibility** list, select **Show**.
    e. In the **Label in dialog box** box, enter `Left plate class`.
11. Click **Add** to create a new parametric variable P8.
12. Modify the variable **P8** as follows:
    a. Rename **P8** as `RightC`.
    b. In the **Formula** box, enter `5`.
    c. In the **Value type** list, select **Number**.
    d. In the **Visibility** list, select **Show**.
    e. In the **Label in dialog box** box, enter `Right plate class`.
13. In the **Custom component browser**, link the variables **P5** and `RightC` to the right stiffener plate.



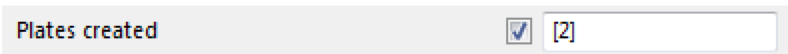
14. Link the variables **P6** and `LeftC` to the left stiffener plate.



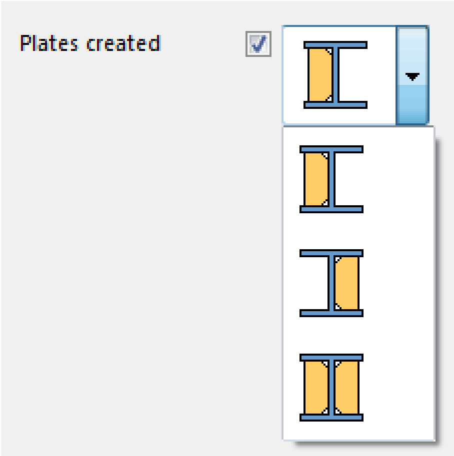
Example: Add a list with images in a custom component dialog box

This example shows how to add an illustrative list in the custom stiffener detail dialog box. You can do this either in the custom component dialog editor or by editing the input (`.inp`) file manually.

In the beginning of this example the dialog box has the text box shown below, and the you need to know the values (0 is left, 1 is right, and 2 is both plates) that control the creation of stiffener plates.



Replace the text box with a list that is easier to use:



Add a list by using the dialog editor

1. Create a custom stiffener detail (page 931) with all the needed variables that control which stiffener plates are created.

   In this example, the variable is called **Plates created**.

2. Open the stiffener dialog box for editing.

   a. In the model, select the custom stiffener detail.

   b. Right-click and select **Edit Custom Component Dialog Box**.

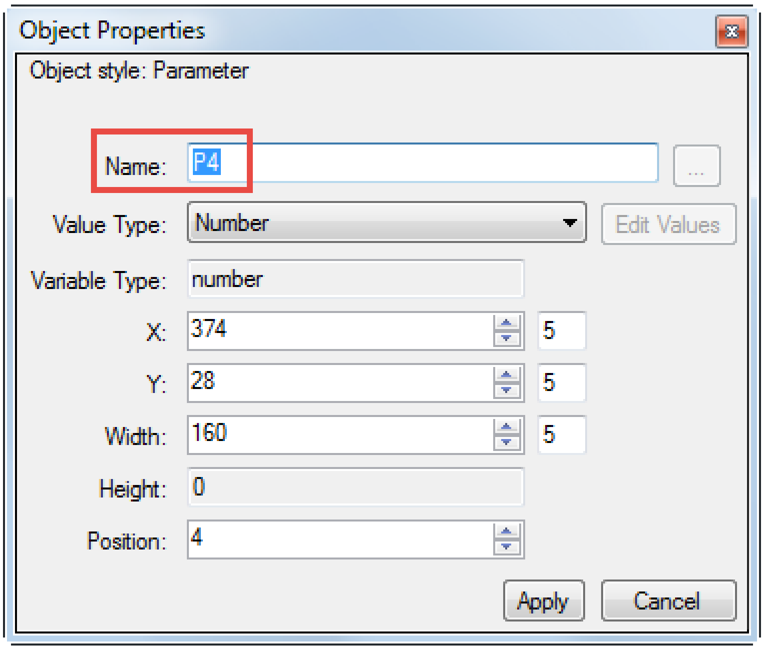
3. Check the name of the parametric variable that controls the plate creation.

   a. In the dialog editor, double-click the **Plates created** box.

      The **Object Properties** dialog box opens.

   b. Check the name of the parametric variable.

In this example, the name is **P4**.



c. Click **Cancel** to close the dialog box.

4. Select the **Plates created** text box and click **Delete**.
5. Click **Insert** --> **Attribute** to add a new attribute list.
6. Drag the attribute list to a suitable location, next to the **Plates created** label.
7. Select the attribute list and then click **Modify** --> **Properties** to edit its properties.
8. Enter **P4** as the **Name** of the attribute.

   Now the attribute list is linked to the parametric variable that controls the plate creation.
9. Click **Edit Values** to add the list items.
10. In the **Edit Attribute Values** dialog box, add an image for the left plate.

    a. Click **Browse Add**.

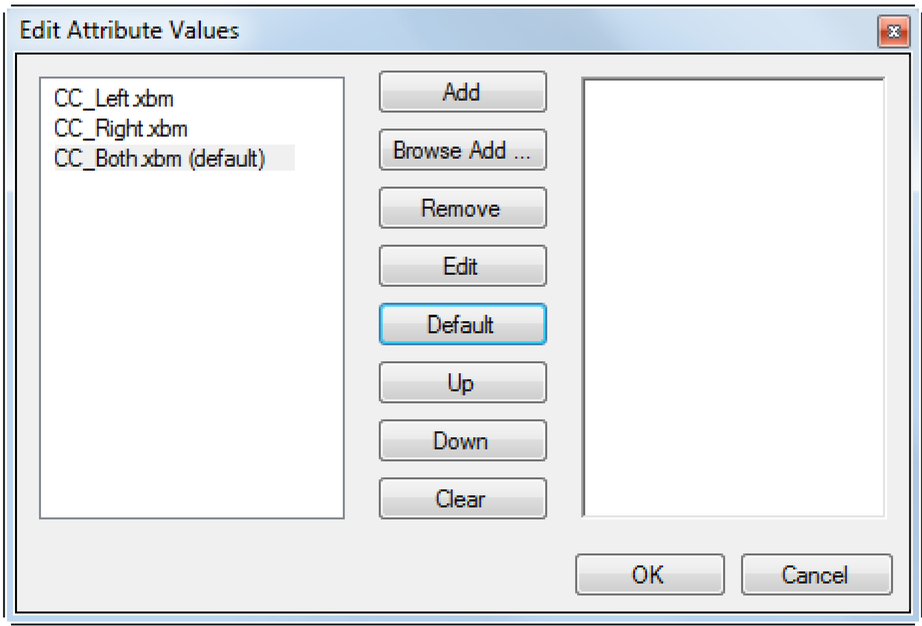
    b. Browse for a suitable image.

    If you create new images, make sure they are in the bitmap (`.bmp`) format. Save the images in the `..\ProgramData\Trimble\Tekla Structures\<version>\Bitmaps` folder.

    The maximum image size is 245x245 pixels.

    c. Click **Open**.
11. Repeat step 9 to add an image for the right plate, and then for both plates.

12. In the **Edit Attribute Values** dialog box, select the image of both plates and then click **Default** to make the attribute the default value.



13. Click **OK**.
14. Click **Apply** in the **Object Properties** dialog box, and then click **Cancel** to close the dialog box.
15. In the dialog editor, click **File** --> **Save** to save the changes.
16. Close and reopen the model for the change to take effect.

Add a list by editing the .inp file

1. Create a custom stiffener detail (page 931) with all the needed variables that control which stiffener plates are created.

   In this example, the variable is called **Plates created**.
2. In the model, click **File** --> **Open the model folder** to open the current model folder.
3. Go to the `\CustomComponentDialogFiles` folder.
4. Open the `.inp` file in a text editor.

```
page("TeklaStructures","")
{
    detail(1, "Stiffeners")
    {
        tab_page("", " Parameters 1 ", 1)
        {
            parameter("Stiffener set back", "P1", distance, number, 1)
            parameter("Plates created", "P4", integer, number, 2)
            parameter("Left plate class", "LeftC", integer, number, 3)
            parameter("Right plate class", "RightC", integer, number, 4)
        }
    }
}
```

5. Remove the following line:

```
parameter("Plates created", "P4", integer, number, 2)
```

6. Add a new **Plates created** attribute with the following settings:

```
page("TeklaStructures","")
{
    detail(1, "Stiffeners")
    {
        tab_page("", " Parameters 1 ", 1)
        {
            parameter("Stiffener set back", "P1", distance, number, 1)
            parameter("Left plate class", "LeftC", integer, number, 3)
            parameter("Right plate class", "RightC", integer, number, 4)
            attribute("", "Plates created", label, "%s", none, none, "0", "0", 334, 118)
        }
    }
}
```

7. Add a new P4 attribute with the following settings:

```
page("TeklaStructures","")
{
    detail(1, "Stiffeners")
    {
        tab_page("", " Parameters 1 ", 1)
        {
            parameter("Stiffener set back", "P1", distance, number, 1)
            parameter("Left plate class", "LeftC", integer, number, 3)
            parameter("Right plate class", "RightC", integer, number, 4)
            attribute("", "Plates created", label, "%s", none, none, "0", "0", 334, 118)
            attribute("P4", "", option, "%s", none, none, "0.0", "0.0", 360, 151, 90)
          {
            value ("Left", 0)
            value ("Right", 0)
            value ("Both", 1)
          }
        }
    }
}
```

The list now contains three options, and **Both** is the default value. The list options are linked to the variable P4 that controls the creation of the stiffener plates.

8. Edit the line numbers so that there are no empty rows between the variables in the dialog box.

```
page("TeklaStructures","")
{
   detail(1, "Stiffeners")
   {
       tab_page("", " Parameters 1 ", 1)
       {
           parameter("Stiffener set back", "P1", distance, number, 1)
           parameter("Left plate class", "LeftC", integer, number, 2)
           parameter("Right plate class", "RightC", integer, number, 3)
           attribute("", "Plates created", label, "%s", none, none, "0", "0", 334, 118)
           attribute("P4", "", option, "%s", none, none, "0.0", "0.0", 360, 151, 90)
        {
           value ("Left", 0)
           value ("Right", 0)
           value ("Both", 1)
        }
       }
   }
}
```

9. Browse for the images you want to use in the dialog box.

   If you create new images, make sure they are in the bitmap (`.bmp`) format. Save the images in the `..\ProgramData\Trimble\Tekla Structures\<version>\Bitmaps` folder.

   The maximum image size is 245x245 pixels.

10. Replace the option texts with the actual file names of the images, but with the file name extension `.xbm`.

```
page("TeklaStructures","")
{
   detail(1, "Stiffeners")
   {
       tab_page("", " Parameters 1 ", 1)
       {
           parameter("Stiffener set back", "P1", distance, number, 1)
           parameter("Left plate class", "LeftC", integer, number, 2)
           parameter("Right plate class", "RightC", integer, number, 3)
           attribute("", "Plates created", label, "%s", none, none, "0", "0", 334, 118)
           attribute("P4", "", option, "%s", none, none, "0.0", "0.0", 360, 151, 90)
        {
           value ("CC_Left.xbm", 0)
           value ("CC_Right.xbm", 0)
           value ("CC_Both.xbm", 1)
        }
       }
   }
}
```

11. Save the `.inp` file.

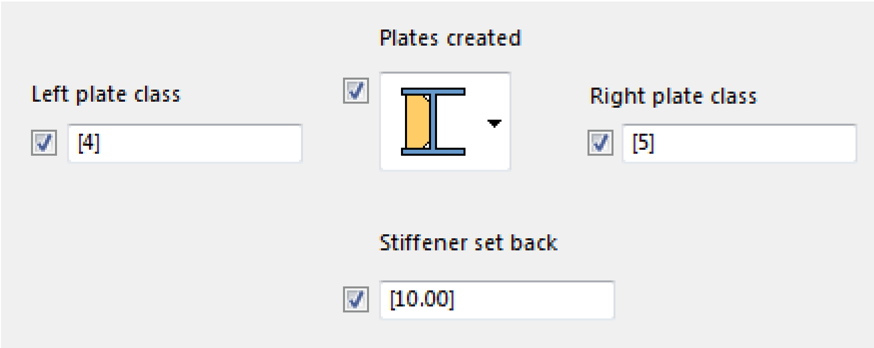
12. Close and reopen the model for the change to take effect.

Example: Arrange text boxes and labels in a custom component dialog box

This example shows how to arrange the text boxes and labels around a list in the custom component dialog box. You can do this either in the custom component dialog editor or by editing the input (`.inp`) file manually.

In the beginning of this example the dialog box looks like the following:

Arrange the dialog box elements more nicely, in the following manner:



Arrange the elements by using the dialog editor

1. Create a custom stiffener detail (page 931) with all the needed variables that control the creation of stiffener plates.
2. Open the stiffener dialog box for editing.
    a. In the model, select the custom stiffener detail.
    b. Right-click and select **Edit Custom Component Dialog Box**.
3. Drag the **Plates created** label above the list with images.
4. Drag the **Left plate class** label and the corresponding text box to the left side of the list.
5. Drag the **Right plate class** label and the corresponding text box to the right side of the list.
6. Drag the **Stiffener set back** label and the corresponding text box underneath the list.
7. In the dialog editor, click **File** --> **Save** to save the changes.
8. Close and reopen the model for the change to take effect.

Arrange the elements by editing the .inp file

1. Create a custom stiffener detail (page 931) with all the needed parametric variables that control the creation of stiffener plates.

2. In the model, click **File** --> **Open the model folder** to open the current model folder.
3. Go to the `\CustomComponentDialogFiles` folder.
4. Open the `.inp` file in a text editor.
5. Edit the file as follows:

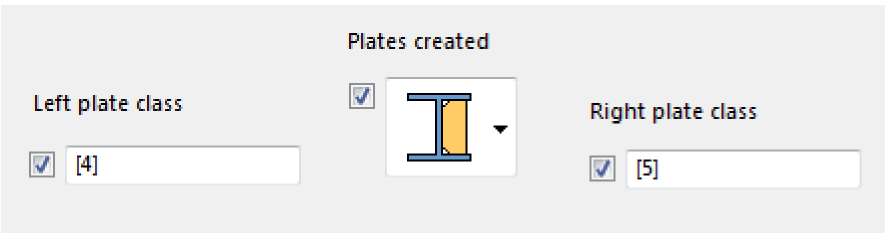
```
page("TeklaStructures", "")
{
    detail(1, "Stiffeners")
    {
        tab_page("", " Parameters 1 ", 1)
        {
            attribute("", "Plates created", label, "%s", none, none, "0", "0", 334, 118)
            attribute("P4", "", option, "%s", none, none, "0.0", "0.0", 360, 151, 90)
            {
                value("CC_Left.xbm", 0)
                value("CC_Right.xbm", 0)
                value("CC_Both.xbm", 1)
            }
            attribute("", "Left plate class", label, "%s", none, none, "0", "0", 125, 157)
            attribute("", "Right plate class", label, "%s", none, none, "0", "0", 497, 160)
            parameter("", "LeftC", integer, number, 146, 192, 160)
            parameter("", "RightC", integer, number, 522, 194, 160)
            parameter("", "P1", distance, number, 357, 289, 160)
            attribute("", "Stiffener set back", label, "%s", none, none, "0", "0", 330, 255)
        }
    }
}
```

6. Save the `.inp` file.
7. Close and reopen the model for the change to take effect.

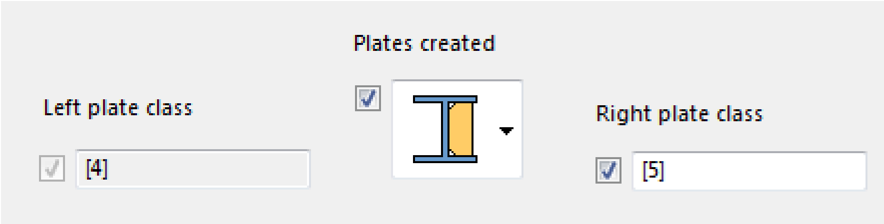
Example: Dim unavailable options in a custom component dialog box

This example shows how to dim the unavailable options in the custom stiffener detail dialog box based on conditions. You can do this either in the custom component dialog editor or by editing the input (`.inp`) file manually.

In the beginning of this example all the options are available:



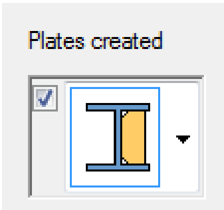
Define that the **Left plate class** text box is unavailable if only the right plate is created, and vice versa.



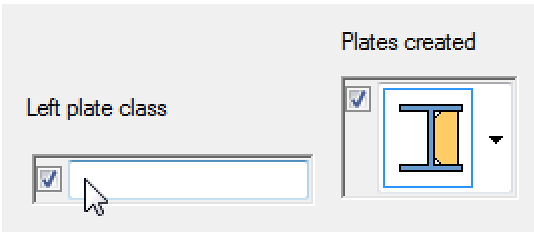
Dim unavailable options by using the dialog editor

1. [Create a custom stiffener detail (page 931)](#) with all the needed parametric variables that control the creation of stiffener plates.
2. Open the stiffener dialog box for editing.
   a. In the model, select the custom stiffener detail.
   b. Right-click and select **Edit Custom Component Dialog Box**.
3. Define that the **Left plate class** text box must be dimmed if only the right stiffener plate is created.
   a. In the **Plates created** list, select the image for the right plate class.

      Note that a blue selection border must be displayed for the image:

   b. Hold down the **Ctrl** key and click the **Left plate class** text box.

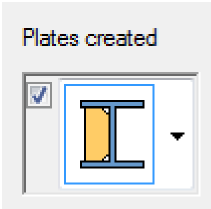
   c. Click the **Toggle visibility**  button.

      The **Left plate class** text box is now dimmed:

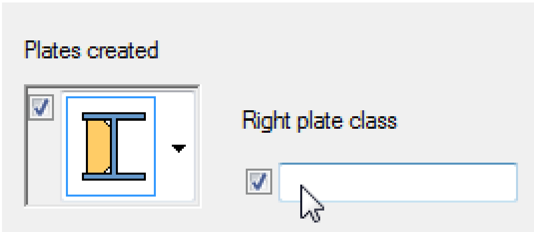
      

4. Unselect the **Left plate class** text box by clicking the **Right plate class** text box.
5. Define that the **Right plate class** text box must be dimmed if only the left stiffener plate is created.
    a. In the **Plates created** list, select the image for the left plate class.

    Note that a blue selection border must be displayed for the image:

    b. Hold down the **Ctrl** key and select the **Right plate class** text box.

    c. Click the **Toggle visibility**  button.

    The **Right plate class** text box is now dimmed:

6. In the dialog editor, click **File** --> **Save** to save the changes.
7. Close and reopen the model for the change to take effect.

Dim unavailable options by editing the .inp file

1. Create a custom stiffener detail (page 931) with all the needed parametric variables that control the creation of stiffener plates.
2. In the model, click **File** --> **Open the model folder** to open the current model folder.
3. Go to the `\CustomComponentDialogFiles` folder.
4. Open the `.inp` file in a text editor.
5. Add the following line to the end of the attribute P4 line:

"toggle_field:LeftC=0;RightC=1"

```
page("TeklaStructures", "")
{
    detail(1, "Stiffeners")
    {
        tab_page("", " Parameters 1 ", 1)
        {
            attribute("", "Plates created", label, "%s", none, none, "0", "0", 334, 118)
            attribute("P4", "", option, "%s", none, none, "0.0", "0.0", 360, 151, 90, "toggle_field:LeftC=0;RightC=1")
            {
                value("CC_Left.xbm", 0)
                value("CC_Right.xbm", 0)
                value("CC_Both.xbm", 1)
            }
            attribute("", "Left plate class", label, "%s", none, none, "0", "0", 125, 157)
            attribute("", "Right plate class", label, "%s", none, none, "0", "0", 497, 160)
            parameter("", "LeftC", integer, number, 146, 192, 160)
            parameter("", "RightC", integer, number, 522, 194, 160)
            parameter("", "P1", distance, number, 357, 289, 160)
            attribute("", "Stiffener set back", label, "%s", none, none, "0", "0", 330, 255)
        }
    }
}
```

The logic is the following:

Selecting the image **CC_left** returns 0, **CC_right** returns 1, and **CC_both** returns 2.

`toggle_field:RightC=1`

When 0 (left) is selected, **RightC** is dimmed.

`toggle_field:LeftC=0`

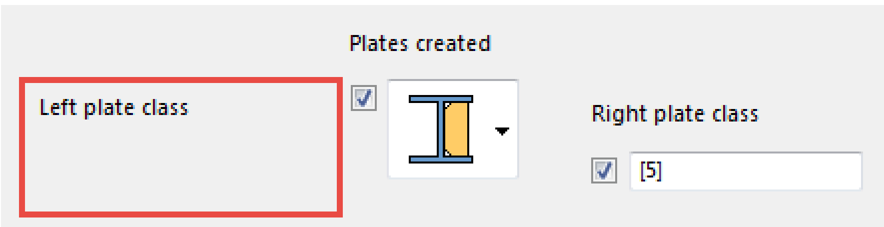
When 1 (right) is selected, **LeftC** is dimmed.

6. Save the `.inp` file.
7. Close and reopen the model for the change to take effect.

TIP If you want to hide unavailable options instead of dimming them in the stiffener dialog box, add an exclamation mark in the conditions:

`"toggle_field:!LeftC=0;!RightC=1"`

The option is now completely hidden when unavailable:



8.13 Custom component settings

Here you will find more information about the various custom component properties and plane types.

- Custom component properties in the Custom Component Wizard (page 953)

  You must define these properties when you create new custom components. You can change some of these properties when you modify an existing custom component.

- Default properties of a custom component dialog box (page 956)

  Each custom component has a dialog box that you can modify. By default, the dialog box has a **Position** tab for custom parts and a **General** tab for custom connections, details, and seams.

- Plane types (page 960)

  When you create distance variables for a custom component, you must select a plane type. The plane type defines what planes you can select.

- Variable properties (page 964)

  Use the **Variables** dialog box to define properties for distance and parametric variables.

Custom component properties in the Custom Component Wizard

You must define these properties when you define new custom components with the **Custom Component Wizard**. You can change some of these properties when you modify an existing custom component.

For more information, see Define custom components (page 830) and Edit and save custom components (page 840).

Type/Notes tab properties

On the **Type/Notes** tab, you have the following options:

Option	Description
Type	Select the type of the custom component. Type affects how you insert the custom component in the model. Type also defines if the custom component connects to existing parts.
Name	Enter a unique name for the custom component.
Description	Enter a short description for the custom component. Tekla Structures shows the description in the **Applications & components** catalog.
Component identifier	Enter an additional name or reference for the component, for example a design code reference.

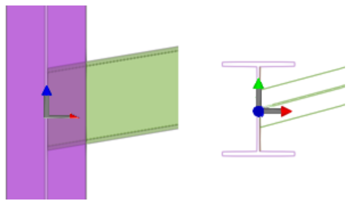
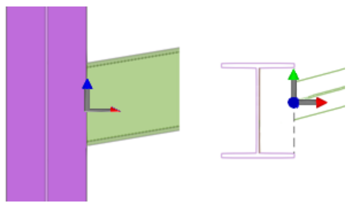
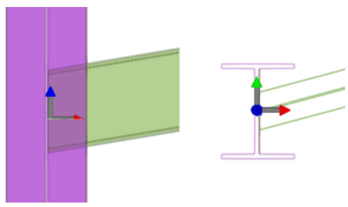
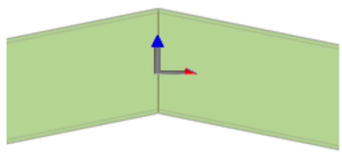
Option	Description
	This can be shown in general arrangement and assembly drawings, and in lists. To show this in drawings, include **Code** in the **Connection Mark Properties** dialog box.

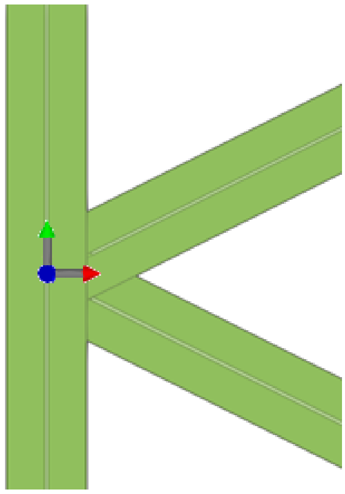
Position tab properties

On the **Position** tab, you have the following options:

Option	Description	Note
Up direction	Sets the default up direction.	Not available for parts.
Position type	The position (or origin) of the component, relative to the main part.	Not available for details and parts.

You can define the position for custom connections and seams. You have the following options:

Option	Description	Example
Middle	Where the center lines of the main and secondary parts intersect.	
Box plane	Where the main part bounding box and the center line of the secondary part intersect.	
Collision plane	Where the main part and the center line of the secondary part intersect.	
Endend plane	Where the center line of the secondary part hits the end of the main part.	

Option	Description	Example
Gusset plane	Where the center lines of the main part and the first secondary part intersect. The x direction is perpendicular to the center line of the main part.	

Advanced tab properties

On the **Advanced** tab, you have the following options:

Option	Description	Note
Detail type	Determines on which side of the main part the component is located. The options are: • **Intermediate detail** Tekla Structures creates all components on the same side of the main part • **End detail** Tekla Structures creates all components on the side of the main part closest to the details Only affects asymmetric components.	Only available for details and seams
Definition point position in relation to primary part	Determines the position you pick to create the detail, relative to the main part.	Only available for details

Option	Description	Note
Definition point position in relation to secondary part	Determines where the component is created, relative to the secondary part.	Only available for connections and seams
Allow multiple instances of connection between same parts	Select this option to create many components to the same main part, in different locations.	Only available for connections and seams
Exact positions	Select this option to position the seam based on the positions that you pick in the model. Clear the check box to let Tekla Structures use automatic seam recognition to position the seam. This is useful especially with warped seams.	Only available for seams
Use the center of the bounding box in positioning	Select to position the custom part based on the center of its bounding box (the box that surrounds the actual part profile).	Only available for parts

Default properties of a custom component dialog box

Each custom component has a dialog box that you can modify. By default, the dialog box has a **General** tab for custom connections, details, and seams, and a **Position** tab for custom parts.

For more information, see Customize the dialog boxes of custom components (page 918).

To view the current properties, double-click the custom component in the model.

Default properties of custom connections, details, and seams

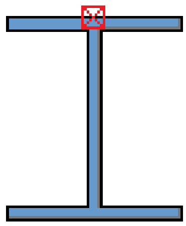
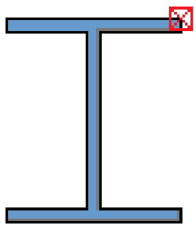
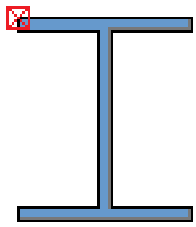
By default, the dialog box of a custom connection, detail, or seam has the following options:

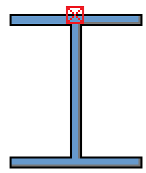
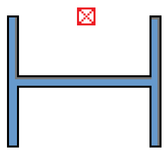
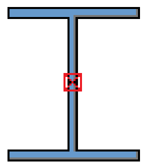
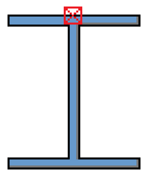
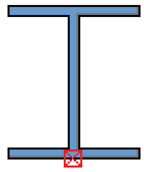
Option	Description	Note
Up direction	Indicates how the component is rotated around the secondary part, relative to the current work plane. If there are no secondary parts, Tekla Structures rotates the connection around the main part.	
Position in relation to primary part	The creation point of the component, relative to the main part.	Only available for details.
Position in relation to secondary part	Tekla Structures automatically places the component according to the selected option.	By default, only available for seams. If you want to use this property in connections, select the **Allow multiple instances of connection between same parts** check box on the **Advanced** tab when you create the component.
Place to picked positions	Select this option to place the seam at the points you pick.	Only available for seams.
Detail type	Determines on which side of the main part the component is located. The options are: • **Intermediate detail** Tekla Structures creates all components on the same side of the main part. • **End detail** Tekla Structures creates all components on the side that is closest to the details. Only affects asymmetric components.	Only available for details.
Locked	Select **Yes** to prevent other users from modifying the properties.	

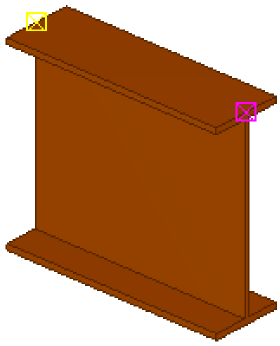
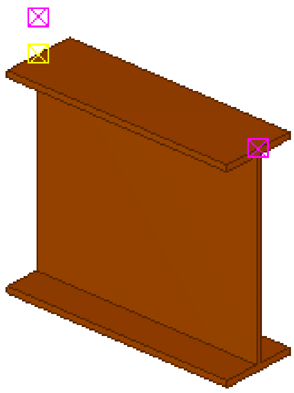
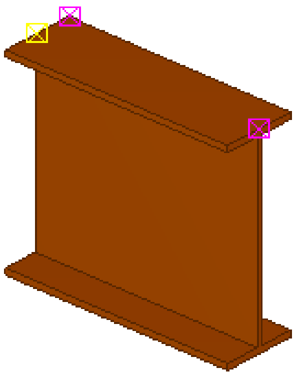
Option	Description	Note
Class	The class of the parts that the custom component creates.	
Connection code	Identifies the component. You can display this connection code in connection marks in drawings.	
AutoDefaults rule group	The rule group used for setting the connection properties.	
AutoConnection rule group	The rule group Tekla Structures uses to select the connection.	

Default properties of custom parts

By default, the dialog box of a custom part has the following options:

Option	Description	Example
On plane	Changes part location on the work plane.	**Middle** 
		Right 
		Left 

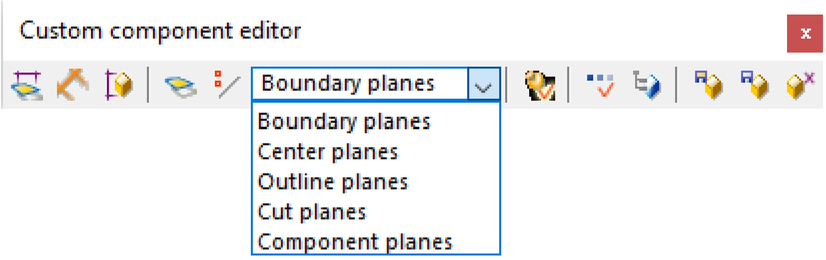
Option	Description	Example
Rotation	Rotates the part in steps of 90 degrees.	**Top** and **Below** 
		Front and **Back** 
At depth	Changes part location perpendicular to the work plane.	**Middle** 
		Front 
		Behind 

Option	Description	Example
Show third handle	Sets the third handle of a nested custom part visible in the desired direction. You can bind the third handle in the desired direction and thus force the part to follow the rotation of another part.	**None**  **Above**  **On the left** 

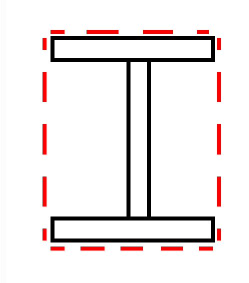
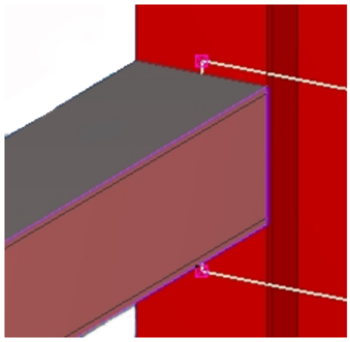
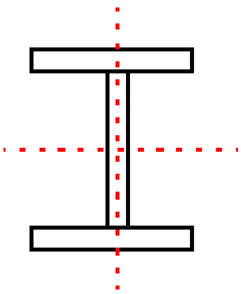
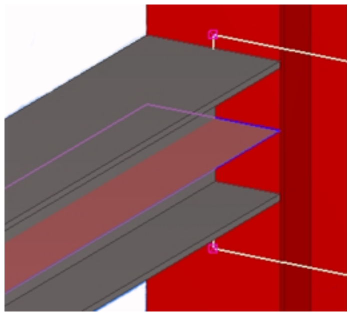
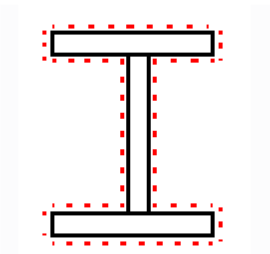
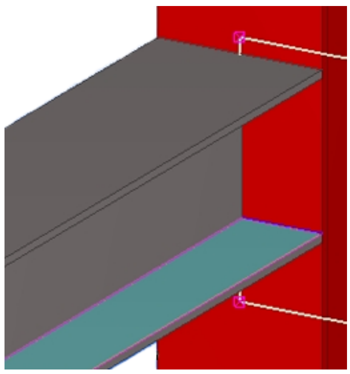
Plane types

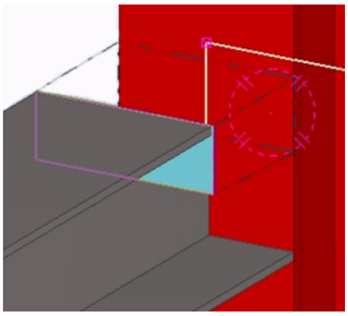
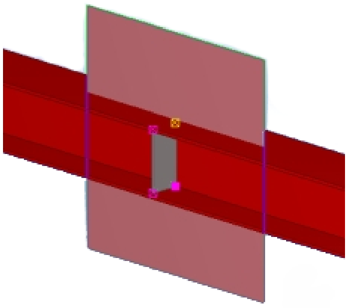
When you add distance variables to a custom component, or to a model, you must select a plane type. The plane type defines what planes you can select.

You have the following options on the **Custom component editor** toolbar:



For more information, see Add variables to a custom component (page 849).

Plane type	Description	Example
Boundary planes	You can select the edges of a bounding box that surrounds the profile. 	
Center planes	You can select the center planes of a profile. 	
Outline planes	You can select the outer and inner surfaces of a profile. 	

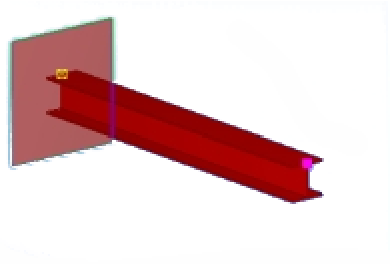
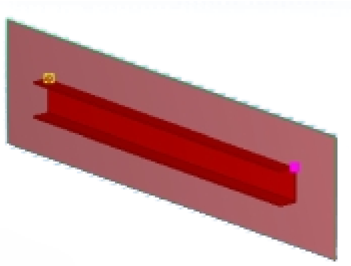
Plane type	Description	Example
Cut planes	If the part contains line, part, or polygon cuts, this option enables you to select cut surfaces. Fittings cannot be selected.	
Component planes	What you can select depends on the component type and the **Position type** of the custom component.	

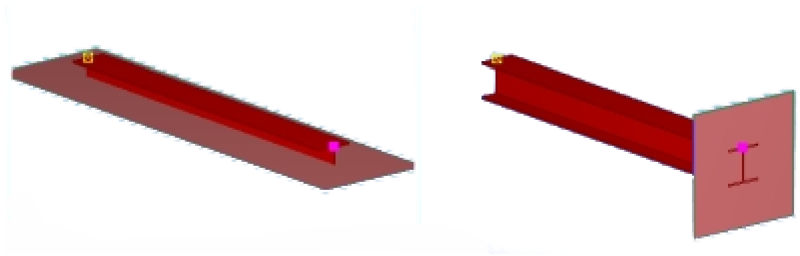
TIP When you create distance variables in a model and bind model object reference points to different planes, select the plane type from the third list on the **Snapping** toolbar. Most of the plane type options there are the same as above, but **Grid planes** is available instead of **Component planes**.

Examples of component planes

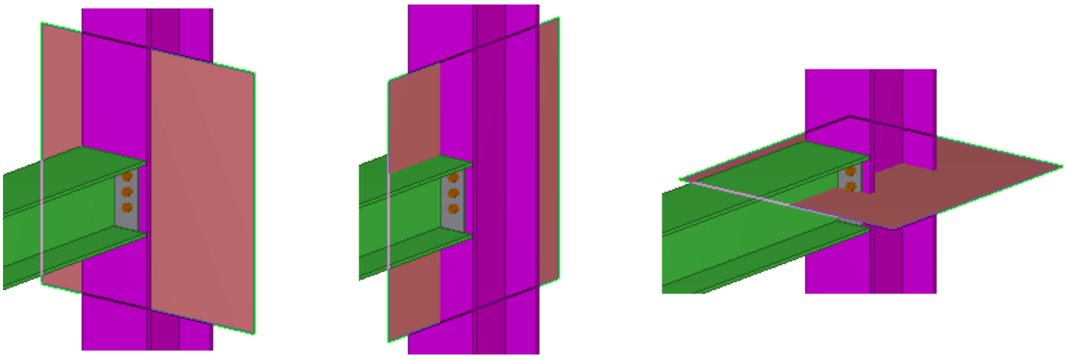
See below for examples of possible component planes. What you can select depends on the component type and the **Position type** of the custom component.

Part component planes

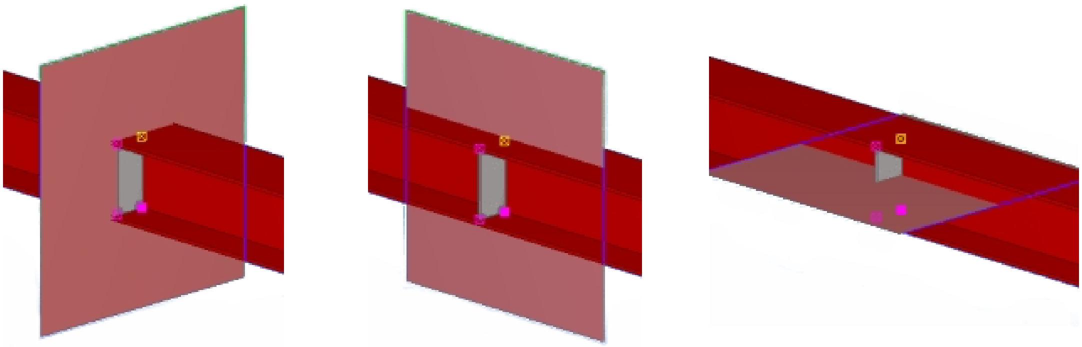





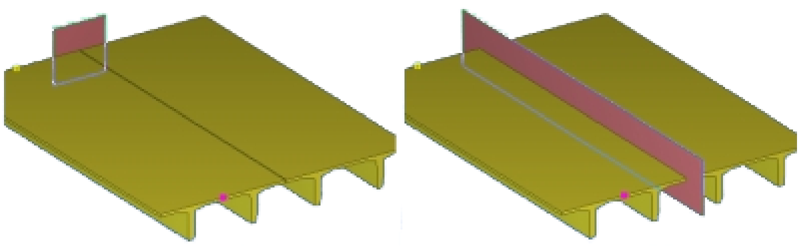
Connection component planes

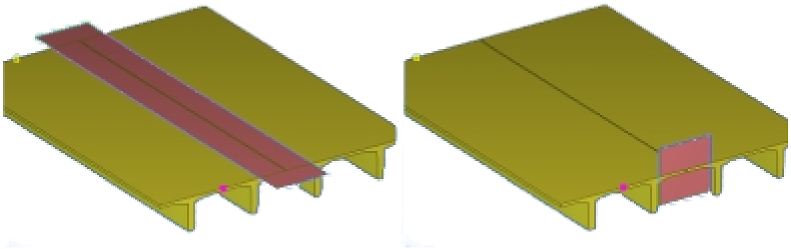


Detail component planes



Seam component planes





Variable properties

Use the **Variables** dialog box to view, modify, and create parametric variables, and to view fixed and reference distance variables.

Tekla Structures uses variables with custom components (page 849), sketched cross sections, and parametric modeling. The examples below are given for custom components, but the same principles apply also to sketched cross sections and parametric modeling.

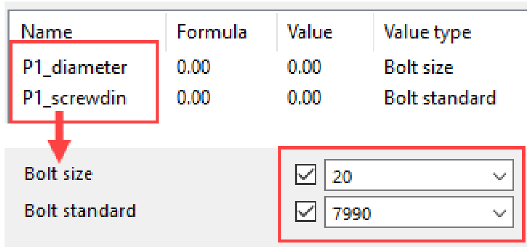
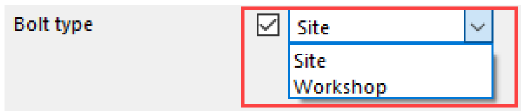
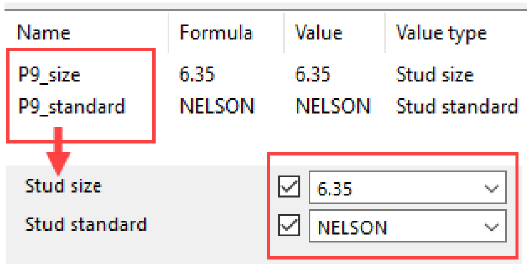
Option	Description
Category	**Component parameters** lists all variables in the component. **Model parameters** lists variables in the current model (such as bindings between a part's end point and a grid plane).
Name	A unique name of a variable. Use this name to refer to the variable in the custom component editor. To be able to correctly reference a variable, its name must be 19 characters or shorter. Variables with longer names will not work correctly when referenced.
Formula	Use this box to enter a value or a formula (page 869). Formulas begin with =.
Value	Shows the current value of **Formula**.
Value type	Select a value type from the list. The type determines what kind of value you can enter for the variable.
Variable type	This property can be either **Distance** or **Parametric**.

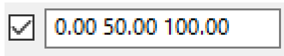
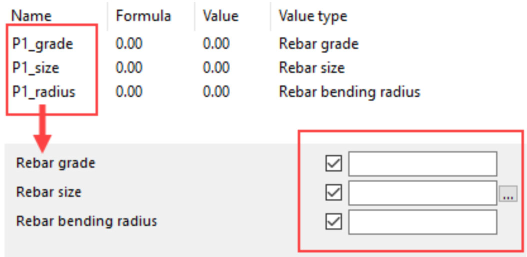
Option	Description
Visibility	Use this setting to control the visibility of a variable. Set to **Show** to display the variable in the custom component dialog box.
Label in dialog box	The name of the variable that Tekla Structures displays in the custom component dialog box. The maximum length is 30 characters.

Value types

You have the following options for the value type:

Option	Description
Number	A whole (integer) number. Use for quantity and multiplier.
Length	A decimal (floating point) number. Use for lengths and distances. Length numbers have unit (mm, inch, etc.) and they are rounded to two decimal places.
Text	A text (ASCII) string.
Factor	A decimal value without a unit. You can set the number of decimals for the value type in **File menu** --> **Settings** --> **Options** --> **Units and decimals** .
Angle	A decimal number type for storing angles, stored to one decimal place, in radians.
Material	A data type associated with the material catalog. Use to select a material grade from the **Select Material** dialog box.
Profile	A data type associated with the profile catalog. Use to select a profile from the **Select Profile** dialog box.
Bolt size **Bolt standard**	Data types linked to the bolt catalog. **Bolt size** works with **Bolt standard**. They have a fixed naming format: `Px_diameter` and `Px_screwdin`. Do not change the fixed name. To show values for these in the component's dialog box, `x` must be

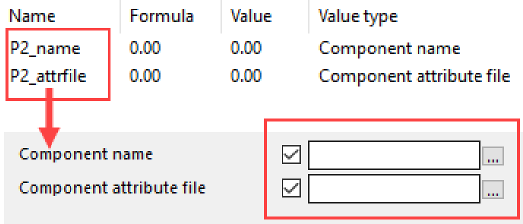
Option	Description
	the same for both, for example, `P1_diameter` and `P1_screwdin`. 
Bolt type	For determining the bolt type (site/ workshop) in the custom component dialog box. Linked to the **Bolt type** property of bolts in the **Custom component browser**. 
Stud size **Stud standard** **Stud length**	Data types linked to the bolt catalog. **Stud size**, **Stud standard** and **Stud length** work together. They have a fixed naming format: `Px_size`, `Px_standard` and `Px_length`. Do not change the fixed names. To show values for these in the component's dialog box, `x` must be the same for all of them. For example, `P9_size`, `P9_standard`, and `P9_length`. 
Hole type	A data type for determining whether holes are special holes and for selecting the special hole type (slotted/oversized/no hole).
Plain hole type	A data type for determining whether holes are through or blind holes.

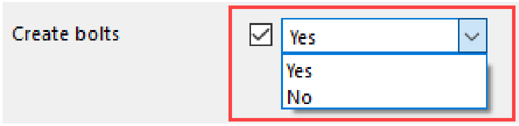
Option	Description
Distance list	Use with options that have several length values, such as bolt spacings. Use space as a separator between the distances. 
Distance list total	Use for calculating the total value of several listed length values, such as bolt spacings. Use space as a separator between the distances.
Weld type	A data type for selecting the weld type. 
Chamfer type	A data type for determining the shape of a chamfer. For more information, see Chamfer part corners and edges (page 397).
Welding site	A data type for determining the welding place: workshop or building site.
Rebar grade **Rebar size** **Rebar bending radius**	Data types linked to reinforcement catalog. **Rebar grade**, **Rebar size**, and **Rebar bending radius** work together. They have a fixed naming format: `Px_grade`, `Px_size`, and `Px_radius`. Do not change the fixed name. To show values for these in the component's dialog box, `x` must be the same for all, for example, `P1_grade`, `P1_size`, and `P1_radius`. 

Option	Description
Rebar hook type	Used for rebar set end detail modifiers, to specify the hook type.
Length adjustment type	Used for rebar set end detail modifiers, to specify how the bar length is extended or shortened, according to a specified end offset or leg length.
Bars affected	Used for rebar set modifiers, to specify how many bars are to be modified (1/1, 1/2, and so on).
Rebar stagger type	Used for rebar set splitters, to specify the stagger type (left/right/middle).
Rebar lap side	Used for rebar set splitters, to specify the side of the overlap (left/right/middle).
Rebar lap placement	Used for rebar set splitters, to determine whether the lapping bars are parallel to each other or on top of each other.
Rebar lap type	Used for rebar set splitters, to determine whether the reinforcing bars are kept straight at lap splices by offsetting entire bars, or placed slanted by offsetting bar ends.
Reinforcement mesh	For determining meshes in custom components. Linked to the **Catalog name** property of reinforcement meshes in the **Custom component browser**.
Cross bar location	Used for rebar meshes, to determine whether the crossing bars are located above or below the longitudinal bars.
Component name **Component attribute file**	Use **Component name** for replacing a sub-component inside a custom component with another sub-component. Linked to the **Name** property of objects in the **Custom component browser**. Use **Component attribute file** for setting the properties of a sub-component inside a custom component. **Component name** and **Component attribute file** work together. They

<table>
<tr><th>Option</th><th>Description</th></tr>
<tr><td></td><td>have a fixed naming format: <code>Px_name</code> and <code>Px_attrfile</code>. Do not change the fixed name.

To show values for these in the component's dialog box, <code>x</code> must be the same for both, for example, <code>P2_name</code> and <code>P2_attrfile</code>.

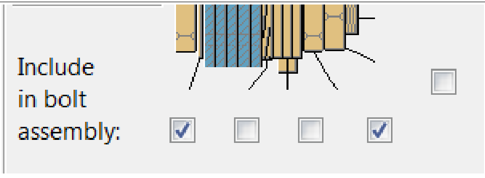
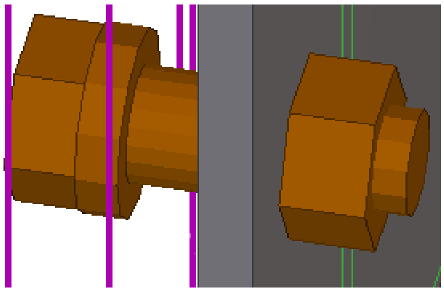

</td></tr>
<tr><td>Yes/No</td><td>For determining whether or not Tekla Structures creates an object in a custom component. Linked to the Creation property of objects in the Custom component browser.


</td></tr>
<tr><td>Shape</td><td>A data type associated with the shape catalog. Use for selecting a shape from the Shape Catalog dialog box.</td></tr>
<tr><td>Bitmask</td><td>For defining bolt assembly (nuts and washers) and parts with slotted holes. Linked to the Bolt structure and Parts with slotted holes properties of bolts in the Custom component browser.

The value is a five-digit series of ones and zeros. This relates to the check boxes in the bolt properties. 1 means that a check box is selected, 0 means that a check box is clear.

In the example below, the value of 10010 means that a bolt with a washer and a nut is created in the bolt assembly.


</td></tr>
</table>

Option	Description
	 

9 Predefined parametric profiles available in Tekla Structures

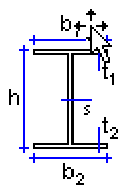
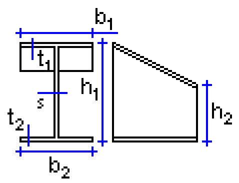
The below predefined parametric profiles are available in Tekla Structures.

The profiles are listed in the same order as they appear in the profile catalog in the default environment.

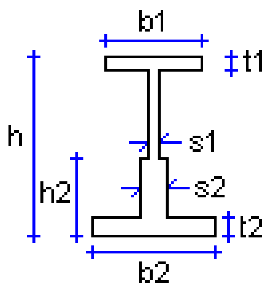
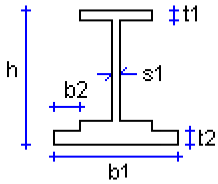
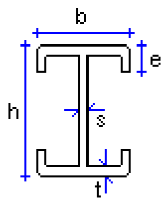
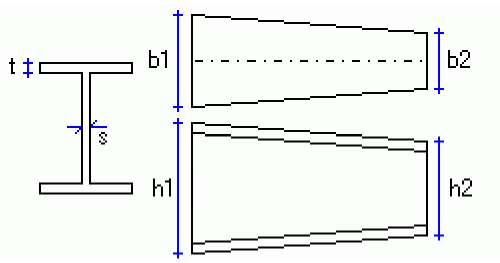
To change how the profiles are grouped in the profile catalog, modify the profile catalog rules.

You can download additional predefined profiles from Tekla Warehouse.

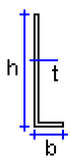
9.1 I profiles

	HIh-s-t*b (symmetric) HIh-s-t1*b1-t2*b2
	HIh1-h2-s-t*b HIh1-h2-s-t1*b1-t2*b2

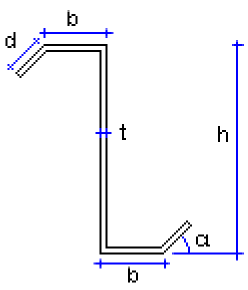
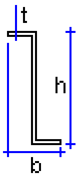
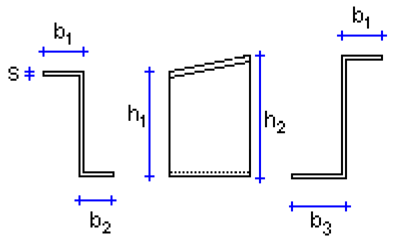
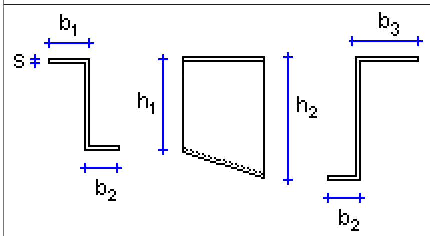
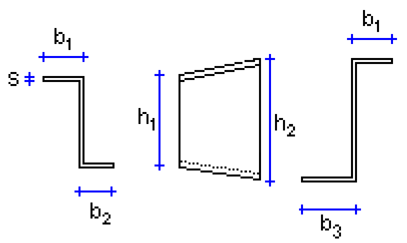
9.2 I beams (steel)

	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

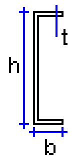
9.3 L profiles

	Lh*b*t

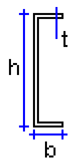
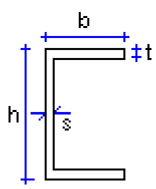
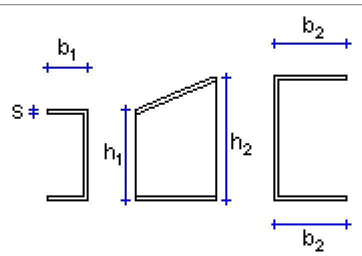
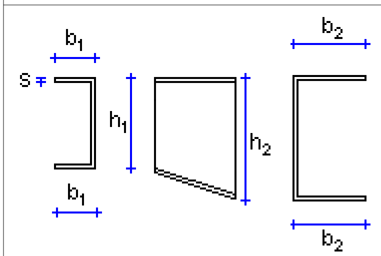
9.4 Z profiles

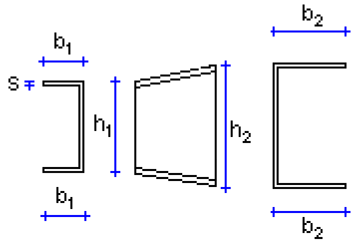
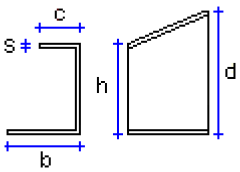
	BENTZ h*b*d*t[-a]
	Z h*b*t
	Z_VAR_A h1*b1*b2-s-h2*b3
	Z_VAR_B h1*b1*b2-s-h2*b3
	Z_VAR_C h1*b1*b2-s-h2*b3

9.5 U profiles

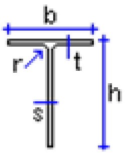
<table>
<tr><td>
</td><td>Uh*b*t</td></tr>
</table>

9.6 C profiles

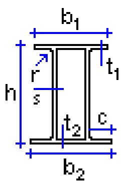
<table>
<tr><td>
</td><td>Ch*b*t</td></tr>
<tr><td>
</td><td>C_BUILTh*b*s*t</td></tr>
<tr><td>
</td><td>C_VAR_Ah1*b1-s-h2*b2</td></tr>
<tr><td>
</td><td>C_VAR_Bh1*b1-s-h2*b2</td></tr>
</table>

	C_VAR_Ch1*b1-s-h2*b2
	C_VAR_Dh-b-d-c-s

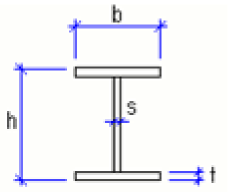
9.7 T profiles

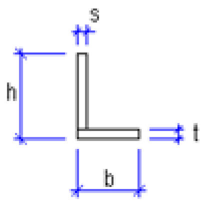
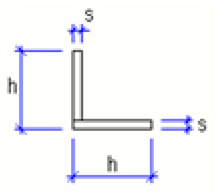
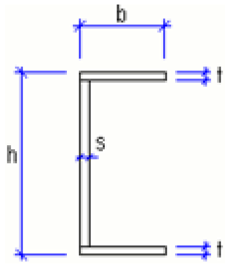
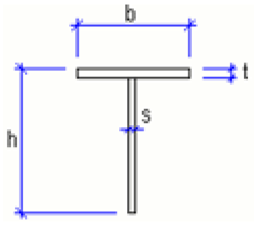
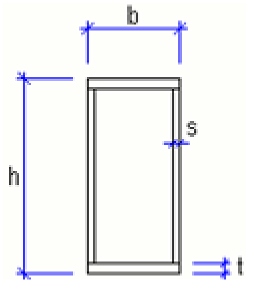
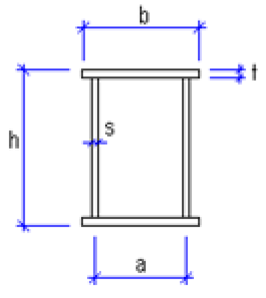
	Th-s-t-b

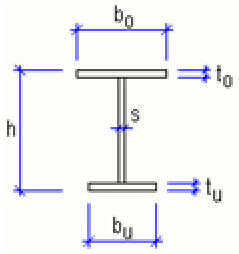
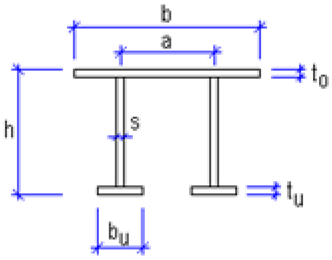
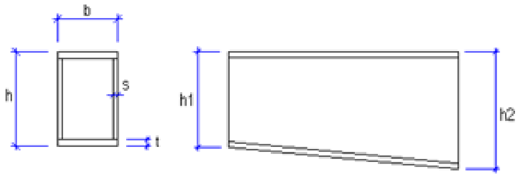
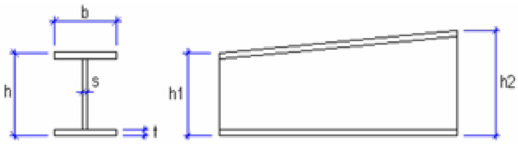
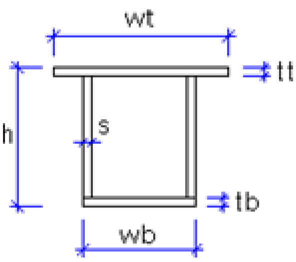
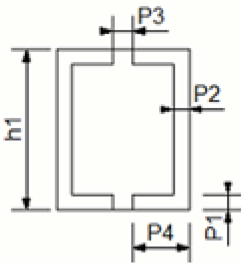
9.8 Welded box profiles

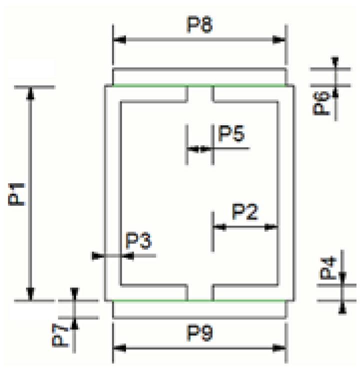
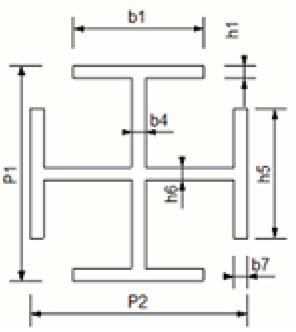
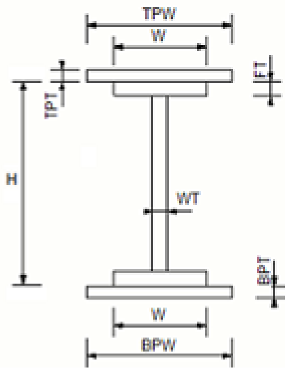
	HK h-s-t*b-c HKh-s-t1*b1-t2*b2-c

9.9 Welded beam profiles

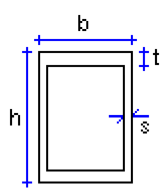
	B_WLD_A h*b*s*t

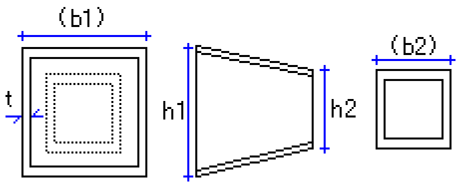
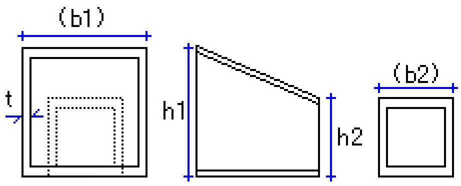
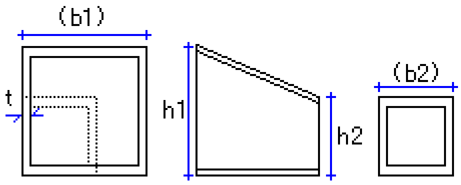
	B_WLD_B h*b*s*t
	B_WLD_C h*s
	B_WLD_D h*b*s*t
	B_WLD_E h*b*s*t
	B_WLD_F h*b*s*[t]
	B_WLD_G h*b*s*t*a

	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

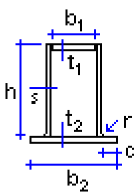
	B_WLD_N p1*p2*p3*p4*p5*p6*p7*p8*p9
	B_WLD_O b1*h1*b4*h5*b7*h6*P1*P2
	B_WLD_P W*H*FT*WT*TPT*TPW*BPT*BPW

9.10 Box profiles

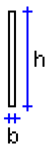
	B_BUILTh*b*s*t

	B_VAR_Ah1-h2*t
	B_VAR_Bh1-h2*t
	B_VAR_Ch1-h2*t

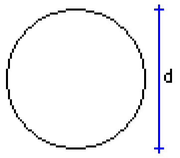
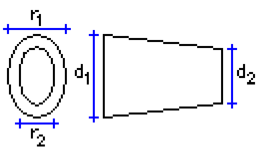
9.11 WQ profiles

	HQh-s-t1*t2*b2 HQh*s-t1*b1-t2*b2-c

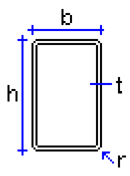
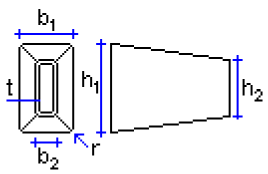
9.12 Rectangular sections

	PLh*b h=height b=thickness (smaller=b)

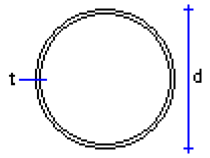
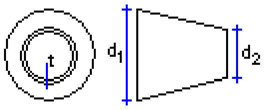
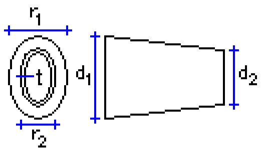
9.13 Circular sections

	Dd
	ELDd1*r1*d2*r2

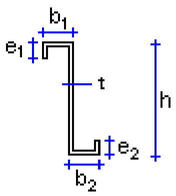
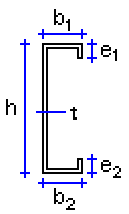
9.14 Rectangular hollow sections

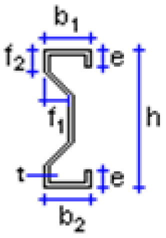
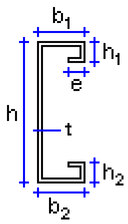
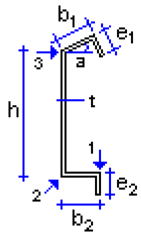
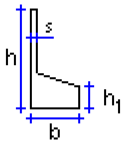
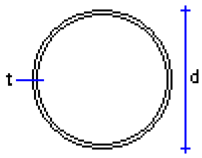
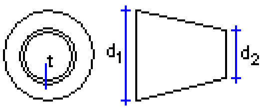
	Ph*t (symmetric) Ph*b*t
	Ph1*b1-h2*b2*t

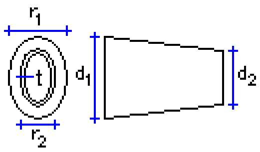
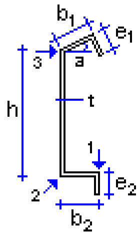
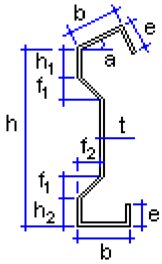
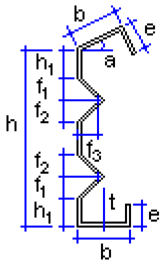
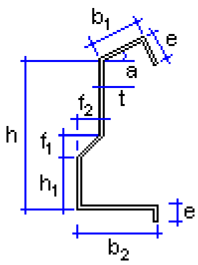
9.15 Circular hollow sections

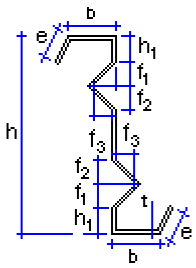
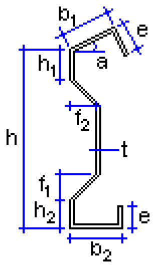
	PDd
	PDd1*d2*t
	EPDd1*r1*d2*r2*t

9.16 Cold rolled profiles

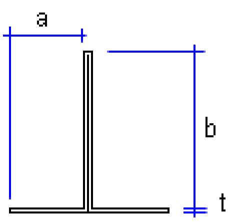
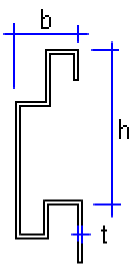
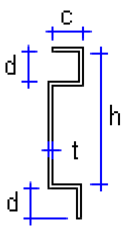
	ZZh-t-e-b (symmetric) ZZh-t-e1-b1-e2-b2
	CCh-t-e-b (symmetric) CCh-t-e1-b1-e2-b2

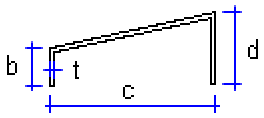
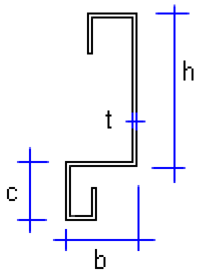
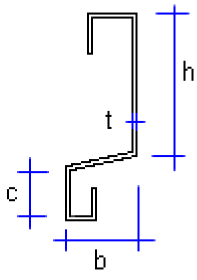
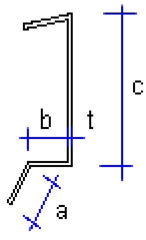
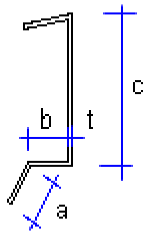
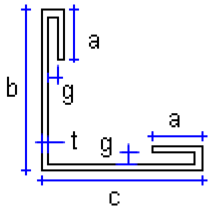
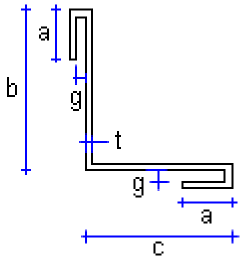
	CW h-t-e-b-f-h1 (symmetric) CW h-t-e1*b1-f1-f2-e2*b2
	CUh-t-h1-b-e (symmetric) CUh-t-h1-b1-h2-b2-e
	EBh-t-e-b-a EBh-t-e1-b1-e2-b2-a Reference points: 1=right 2=left 3=top
	BFh-s-b-h1
	SPDd*t
	SPDd2*d2*t

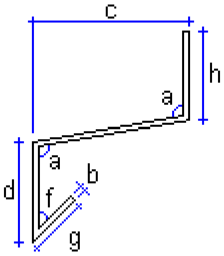
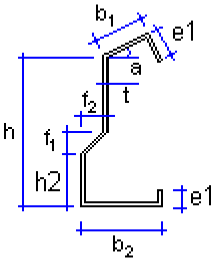
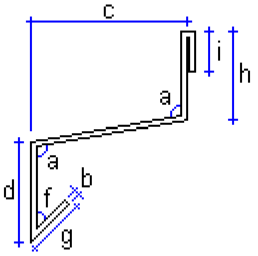
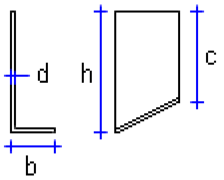
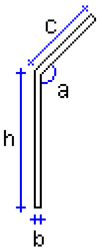
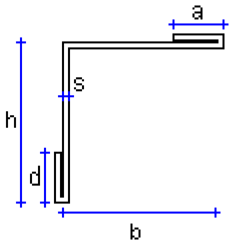
	ESPD d1-d2*t
	ECh-t-e-b-a ECh-t-e1-b1-e2-b2-a
	EDh-t-b-e-h1-h2-f1-f2-a
	EEh-t-e-b-f1-f3-h1-f2-a
	EFh-t-e-b1-b2-f1-f2/h1-a

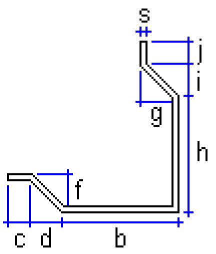
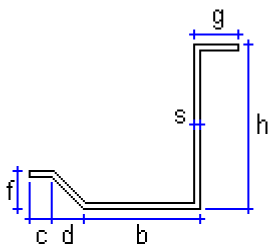
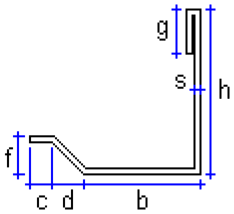
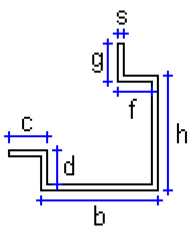
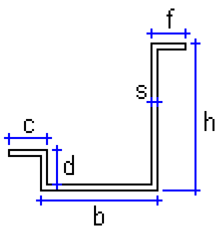
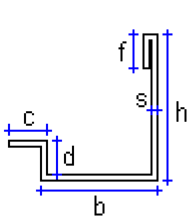
	EZh-t-e-b-f1-f3-h1-f2-a
	EWh-t-e-b1-b2-f1-f2-h2-h1-a

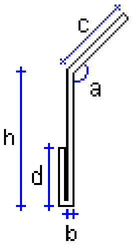
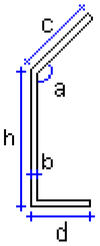
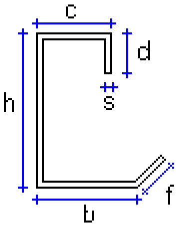
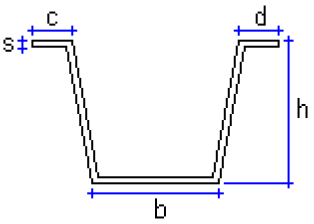
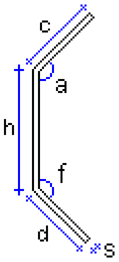
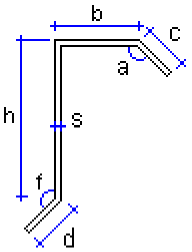
9.17 Folded plates

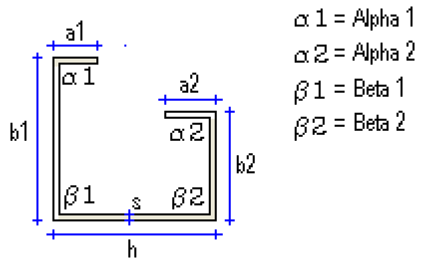
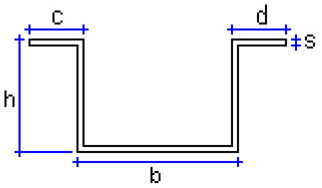
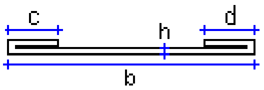
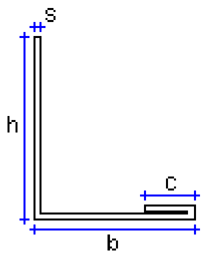
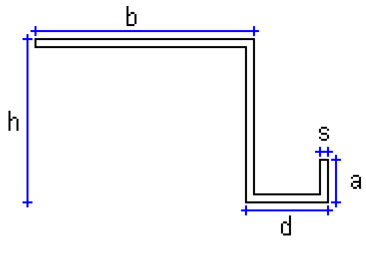
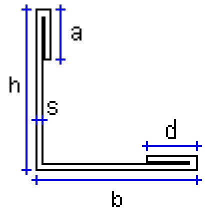
	FFLAa-b-t
	FPANBh-b-t FPANB_-b-t FPANBAh-b-t FPANBA_h-b-t
	FPANBBh-c-d-t

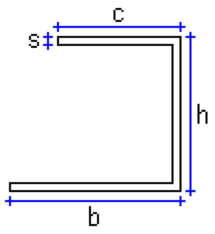
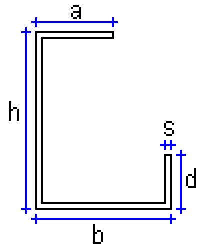
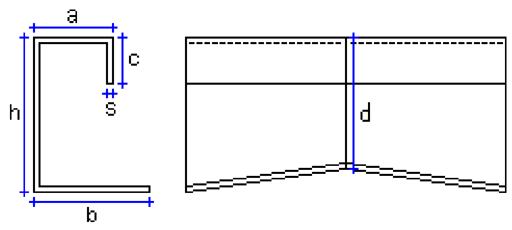
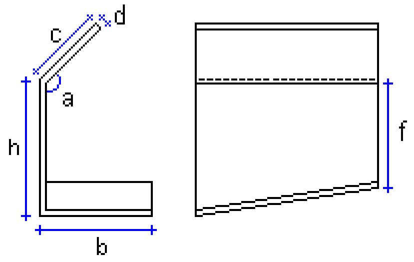
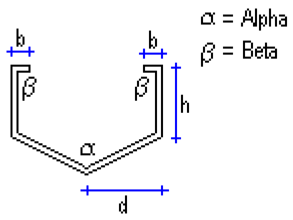
	FPANCVb-c-d-t
	FPANGh-b-c-t
	FPANGAh-b-c-t
	FPANJa-b-c-t
	FPANJa-b-c-t
	FPAN a-b-c-t-g
	FPANVVa-b-c-t-g

	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

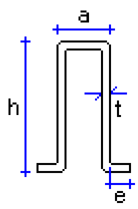
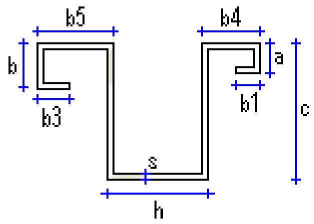
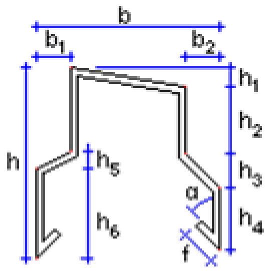
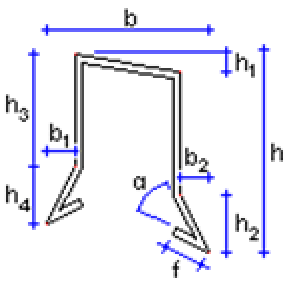
	FP_Db-h-c-d-f-g-i-j-s
	FP_Eb-h-c-d-f-g-s
	FP_Fb-h-c-d-f-g-s
	FP_Gb-h-c-d-f-g-s
	FP_Hb-h-c-d-f-s
	FP_Ib-h-c-d-f-s

	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

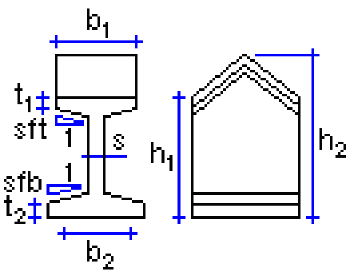
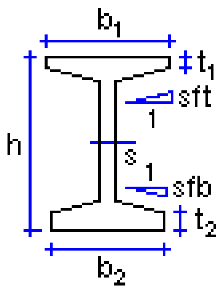
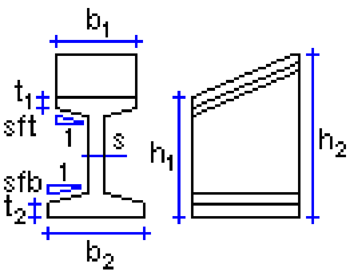
	FP_Pa1*a2*h-b1*b2-Alpha1-Alpha2-Beta1-Beta2-s
	FP_Qb-h-c-d-s
	FP_Rb-h-c-d
	FP_Sb-h-c-s
	FP_Tb-h-a-d-s
	FP_Ub-h-a-d-s

	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

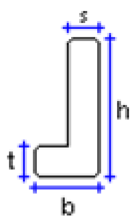
9.18 Hat profiles

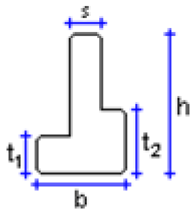
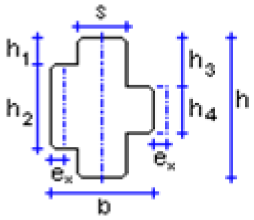
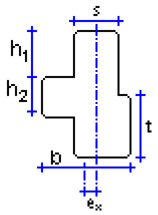
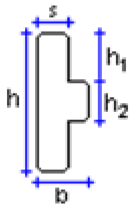
	HAT h*a*c*t
	HATCa-b-c-b1-h-b3-b4-b5-s
	HATAb1*h1*h2*h3*h4*h5*h6*b2*t*f *a*h*b
	HATBb*b1*b2*h*h1*h2*h3*h4*t*f*a

9.19 I beams (concrete)

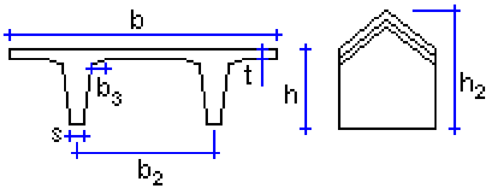
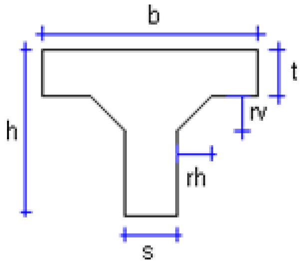
	HIIh1*b1*t1-h2-s-b2*t2[-sft[-sfb]]
	IIh*b1*t1-s-b2*t2[-sft[-sfb]]
	SIIh1*b1*t1-h2-s-b2*t2[-sft[-sfb]]

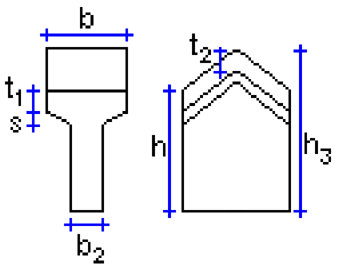
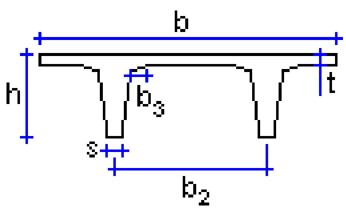
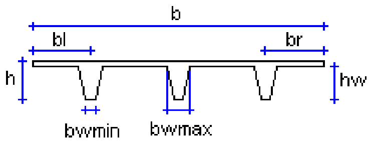
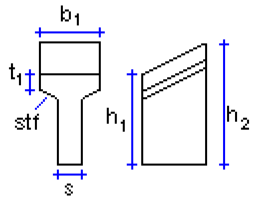
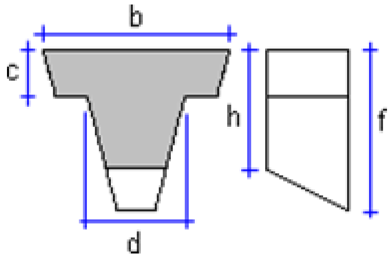
9.20 Ledger beams (concrete)

	RCLs*h-b*t

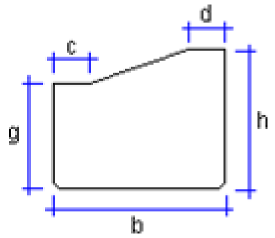
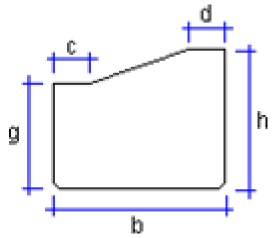
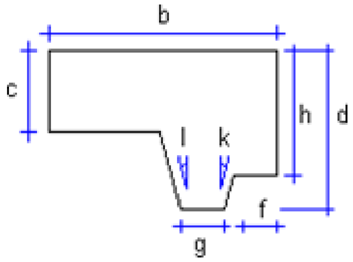
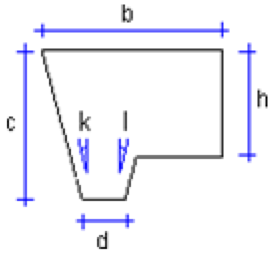
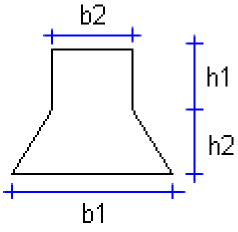
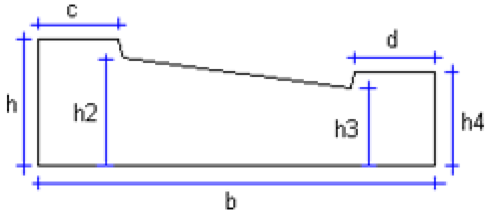
	RCDLs*h-b*t RCDLs*h-b*t1*t2
	RCDXs*h-b*h2*h1 RCDXs*h-b*h4*h3*h2*h1 RCDXs*h-b*h4*h3*h2*h1-ex
	RCXXs*h-b*t*h1-h2-ex
	RCXs*h-b*h2*h1

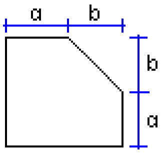
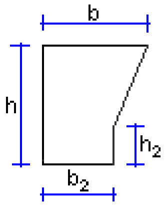
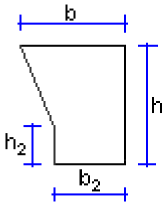
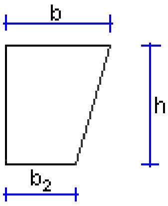
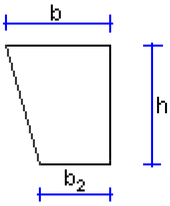
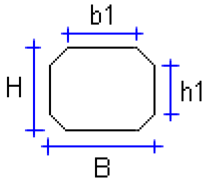
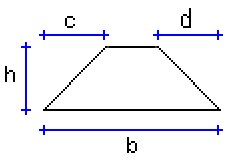
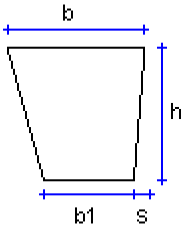
9.21 T profiles (concrete)

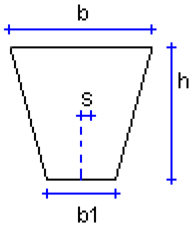
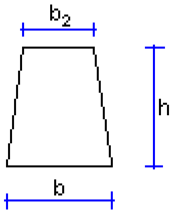
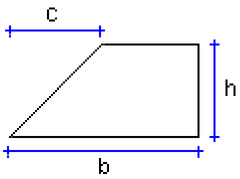
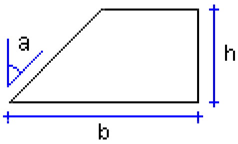
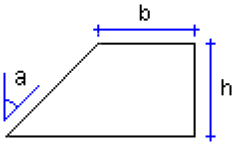
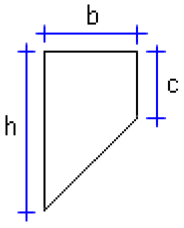
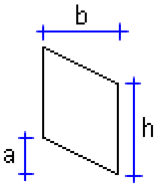
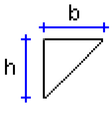
	HTTh*b-s-t-b2-h2
	TCh-b-t-s

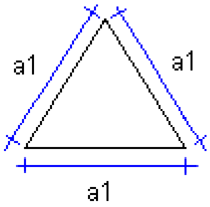
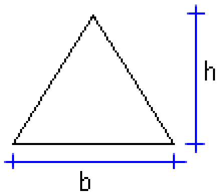
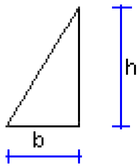
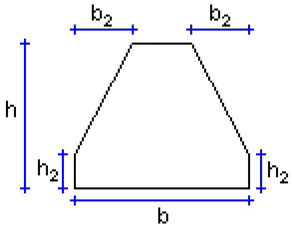
	TRIh*b-b2*t1-h3-t2
	TTh*b-s-t-b2
	TTTh*b-bl-br-hw-bwmin-bwmax
	T_VAR_Ah1*h2*s*b1*t1-sft
	T_VAR_Bh-b-c-d

9.22 Irregular beams (concrete)

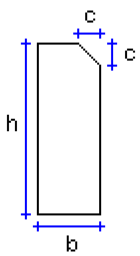
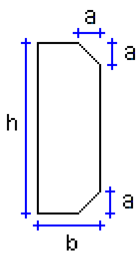
	IRR_Ab-h-g-c-d
	IRR_Bh-b-c-d-f-g
	IRR_Ch-b-c-d
	IRR_Db1*b2-h1*h2
	IRR_Eh-b-c-d-h2-h3-h4

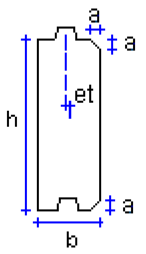
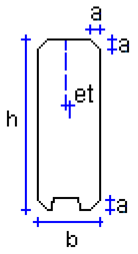
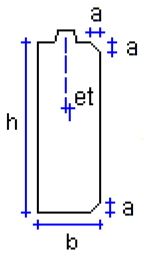
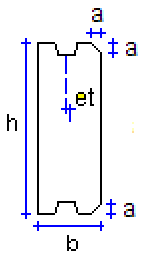
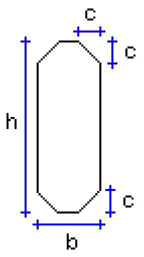
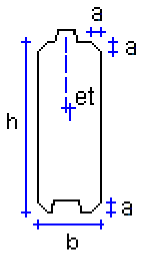
	IRR_Fa*b
	IRR_Gh*b*h2*b2
	IRR_Hh*b*h2*b2
	IRR_Ih*b*b2
	IRR_Jh*b*b2
	OCTB*b1-H*h1
	REC_Ah-b
	REC_Bh-b-b1

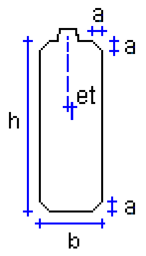
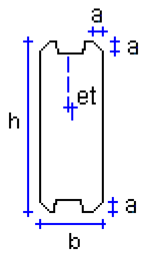
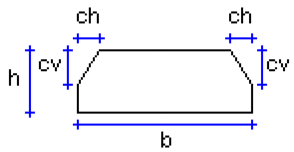
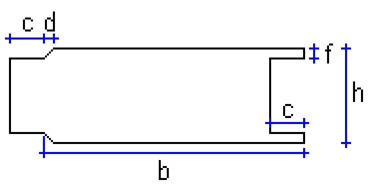
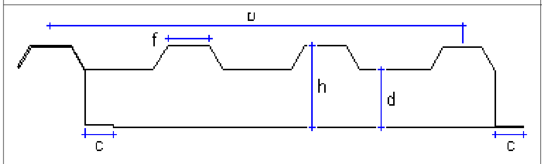
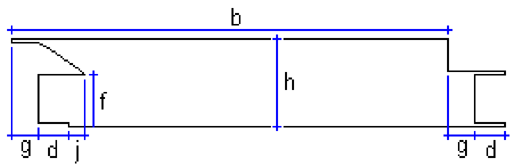
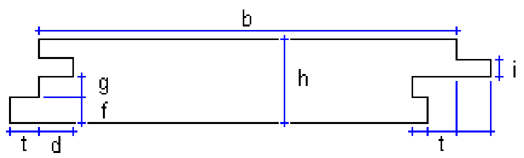
	REC_Ch-b-b1
	REC_Dh-b-b2
	REC_Eh-b
	REC_Fh-b
	REC_Gh-b
	REC_Hh-b
	REC_I a-b*h
	TRI_Ah-b

	TRI_Ba1
	TRI_Cb-h
	TRI_Dh*b
	TRI_Eb*h*h2*b2

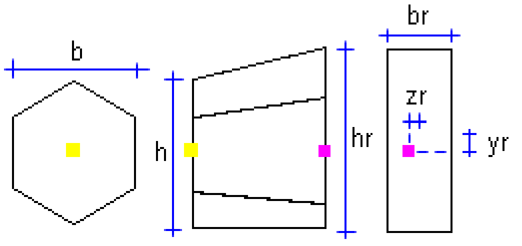
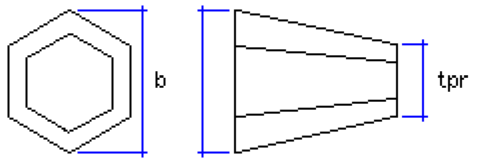
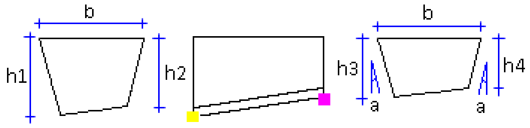
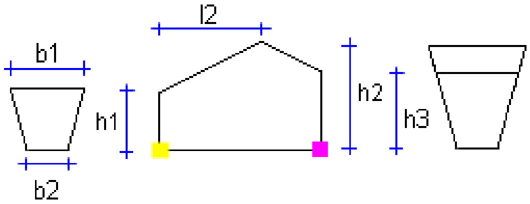
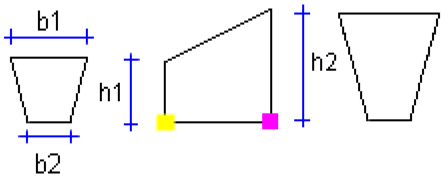
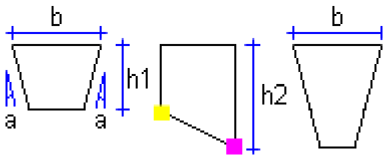
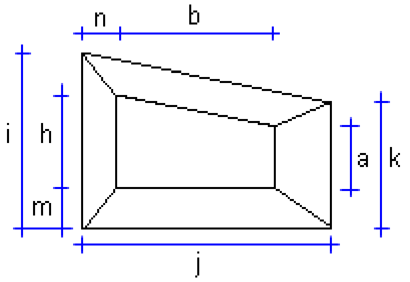
9.23 Panels

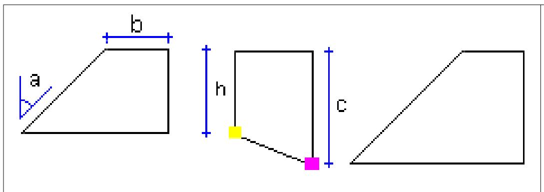
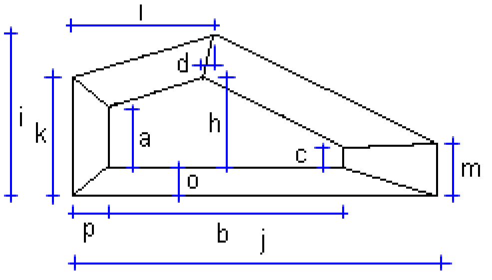
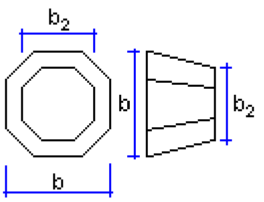
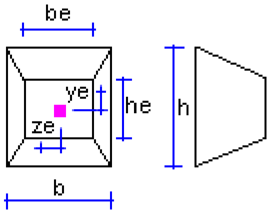
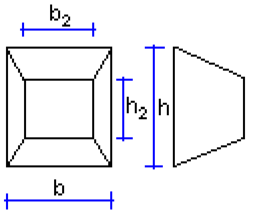
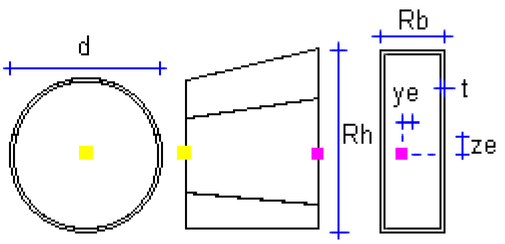
	PNL_Ah*b
	PNL_Bh*b

	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

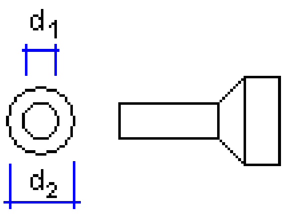
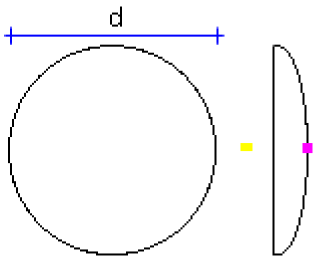
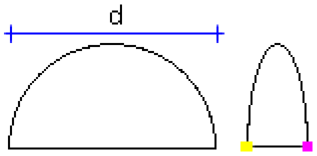
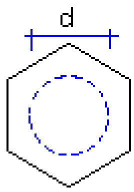
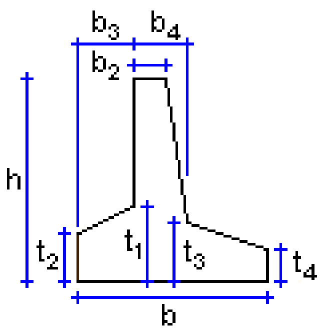
	PNL_Ih*b-a-ht*bt
	PNL_Jh*b-a-ht*bt
	PNL_Kh*b
	PNL_Lh-b-c-f
	PNL_Mh-b-c-f-d
	PNL_Nh-b-d-f-g-j
	PNL_Oh-b-d-f-g-i-t

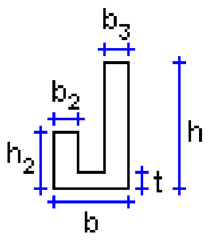
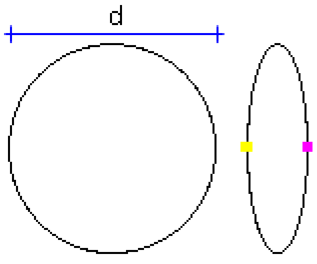
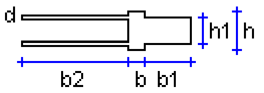
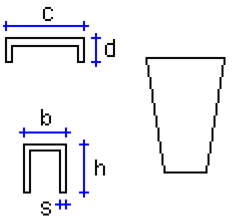
9.24 Variable cross sections

	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

9.25 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

10 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:

- Modify the position of a part (page 310)
- Numbering settings (page 1005)
- Reinforcement settings (page 1009)

10.1 Numbering settings

This section provides more information about specific numbering settings.

Click the links below to find out more:

- General numbering settings (page 1005)
- Weld numbering settings (page 1007)
- Control number settings (page 1008)

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.

Setting	Description
Check for standard parts	If a separate standard-part model (page 747) 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 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 is set to `TRUE`. The assembly phase affects numbering.
Reinforcing bars	Reinforcing bars affect numbering.

Setting	Description
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 Number assemblies, cast units, and rebar assemblies (page 723).
Family numbering	See Assign family numbers (page 720).

See also

Adjust the numbering settings (page 721)

Numbering settings during a project (page 746)

Numbering examples (page 741)

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

Number welds (page 725)

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.
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.

See also

Control numbers (page 731)

10.2 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 1009)

Reinforcement mesh properties (page 1011)

Rebar set properties (page 1016)

Reinforcement strand properties (page 1032)

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 (page 496)
- `.rbg` for groups (page 498)
- `.rci` for circular groups (page 507)
- `.rcu` for curved groups (page 505)

General, Hooks, Cover thickness, IFC export, More

The following properties are available for single reinforcing bars and reinforcing bar groups:

<table>
<tr><th>Option</th><th colspan="2">Description</th></tr>
<tr><td>Name</td><td colspan="2">User-definable name of the bar.

Tekla Structures uses bar names in reports and drawing lists, and to identify bars of the same type.</td></tr>
<tr><td>Grade</td><td>Steel grade of the bar.</td><td rowspan="2">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</td></tr>
<tr><td>Size</td><td>Diameter of the bar.

Depending on the environment, the nominal diameter of the bar, or a mark that defines the diameter.</td></tr>
</table>

<table>
<tr><th>Option</th><th colspan="2">Description</th></tr>
<tr><td>Bending radius</td><td>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.</td><td>for the chosen grade. You can also select whether the bar is a main bar, or a stirrup or tie.</td></tr>
<tr><td>Class</td><td colspan="2">Use to group reinforcement.

For example, you can display bars of different classes in different colors.</td></tr>
<tr><td>Numbering</td><td colspan="2">Mark series of the reinforcing bar.</td></tr>
<tr><td>Hook type</td><td>Shape of the hook.</td><td rowspan="4">The rebar catalog (<code>rebar_database.inp</code>) contains the predefined minimum bending radius and minimum hook length for all standard hooks.

See Add hooks to reinforcing bars (page 555).</td></tr>
<tr><td>Angle</td><td>Angle of a custom hook.</td></tr>
<tr><td>Radius</td><td>Internal bending radius of a standard hook or custom hook.</td></tr>
<tr><td>Length</td><td>Length of the straight part of a standard or custom hook.</td></tr>
<tr><td>Cover thickness on plane</td><td>Distances from the part surfaces to the bar on the same plane as the bar.</td><td rowspan="4">See Define the reinforcement cover thickness (page 558).</td></tr>
<tr><td>Cover thickness from plane</td><td>Distance from the part surface to the bar, or to the bar end, perpendicular to the bar plane.</td></tr>
<tr><td>Start</td><td>Concrete cover thickness or leg length at the first end of the bar.</td></tr>
<tr><td>End</td><td>Concrete cover thickness or leg length at the second end of the bar.</td></tr>
</table>

Option	Description
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 (page 509) groups
- curved reinforcing bar groups
- circular reinforcing bar groups

Option	Description	
Rebar group type	What is the type of the group.	See Create a tapered or spiral reinforcing bar group (page 509).
Number of cross sections		
Creation method	How the bars are spaced.	See Distribute bars in a reinforcing bar group (page 547).
Number of reinforcing bars		
Target spacing value		
Exact spacing value		
Exact spacing values		
Exclude	Which bars are omitted from the group.	See Delete bars from a reinforcing bar group (page 550).

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.
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 (page 516), select the **Custom mesh** option and define the properties (page 1013).
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 Add hooks to reinforcing bars (page 555). 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.

Option	Description
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	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)**.
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.

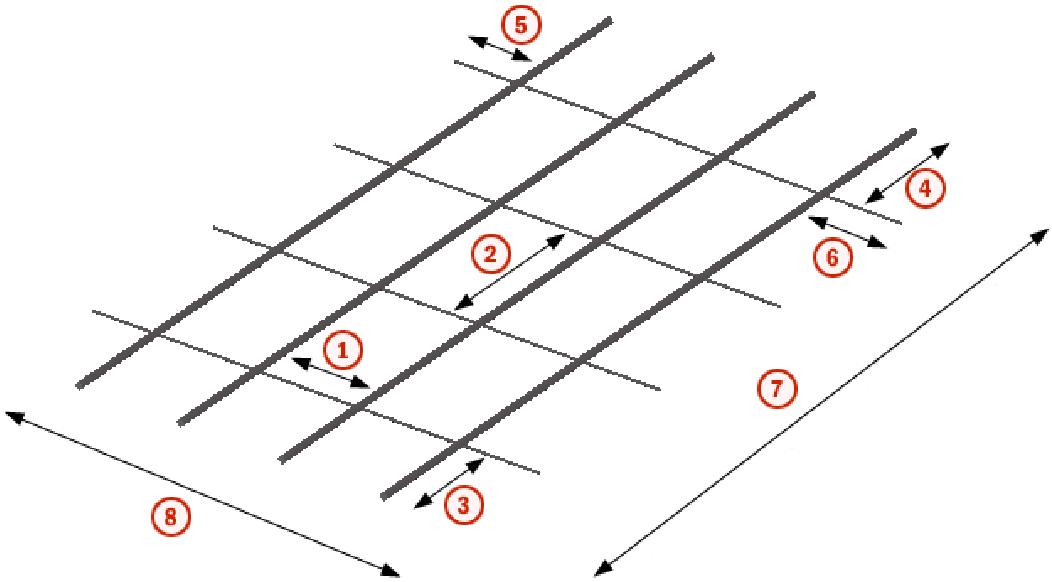
See also

Create a reinforcement mesh (page 511)

Custom reinforcement mesh properties

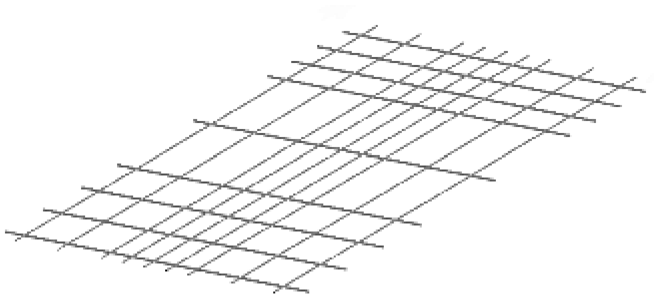
Use the **Rebar mesh** properties to view and modify the properties of customized reinforcement meshes. The file name extension of a reinforcement mesh properties file is `.rbm`.

You can define the following properties for the customized reinforcement meshes (page 516):



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.

Option	Description
	• **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. 
Left overhang	Extensions of crossing bars over the outermost longitudinal bars.
Right overhang	Extensions of longitudinal bars over the outermost crossing bars.
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.

Option	Description
Grade	Steel grade of the bars in the mesh.

See also

Create a reinforcement mesh (page 511)

Reinforcement mesh properties (page 1011)

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

<table>
<tr><th>Option</th><th colspan="2">Description</th></tr>
<tr><td>Name</td><td colspan="2">User-definable name of the bars.

Tekla Structures uses bar names in reports and drawing lists, and to identify bars of the same type.</td></tr>
<tr><td>Grade</td><td>Steel grade of the bars.</td><td rowspan="3">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.</td></tr>
<tr><td>Size</td><td>Diameter of the bars.

Depending on the environment, the nominal diameter of the bars, or a mark that defines the diameter.</td></tr>
<tr><td>Bending radius</td><td>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 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].</td></tr>
</table>

Option	Description
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 (page 522) 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.
End offset	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.
Creation method	How the bars are spaced. For more information, see Spacing options.
Number of reinforcing bars	
Target spacing value	
Exact spacing value	

Option	Description
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**.

Option	Description
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

Create a rebar set (page 470)

Modify a rebar set (page 521)

Secondary guideline properties (page 1019)

Leg face properties (page 1020)

Leg surface properties (page 1021)

Property modifier properties (page 1022)

End detail modifier properties (page 1026)

Splitter properties (page 1029)

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.
End offset	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.
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

Modify a rebar set locally using modifiers (page 533)

Distribute bars in a rebar set (page 541)

Rebar set properties (page 1016)

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

Modify a rebar set using leg faces and leg surfaces (page 525)

Rebar set properties (page 1016)

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

Modify a rebar set using leg faces and leg surfaces (page 525)

Rebar set properties (page 1016)

Leg face properties (page 1020)

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

<table>
<tr><th>Option</th><th colspan="2">Description</th></tr>
<tr><td>Name</td><td colspan="2">User-definable name of the bars.
Tekla Structures uses bar names in reports and drawing lists, and to identify bars of the same type.</td></tr>
<tr><td>Grade</td><td>Steel grade of the bars.</td><td rowspan="3">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.</td></tr>
<tr><td>Size</td><td>Diameter of the bars.
Depending on the environment, the nominal diameter of the bars, or a mark that defines the diameter.</td></tr>
<tr><td>Bending radius</td><td>Internal radius of the bends in the bars.
Bending radius complies with the design code you are using. Main bars, stirrups,</td></tr>
</table>

Option	Description
	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. 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.

Option	Description
Affect whole bar plane	Select whether the property modifier affects all bars of a rebar set that are in the same plane even 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

Modify a rebar set locally using modifiers (page 533)

Rebar set properties (page 1016)

End detail modifier properties

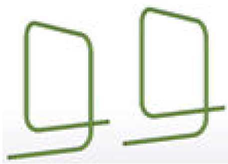
Use the property pane or the contextual toolbar to view and modify the properties of the rebar set end detail modifiers. The file name extension of the property file is `.rst_edm`.

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.

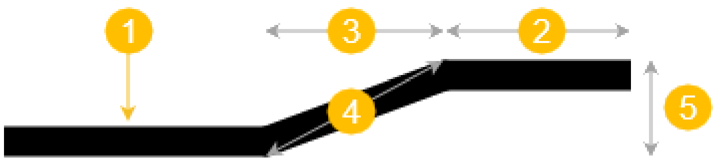
Hook

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 [Add hooks to reinforcing bars (page 555)](#).
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: 

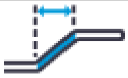
Cranking

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`.

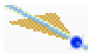
See also

Modify a rebar set locally using modifiers (page 533)

Rebar set properties (page 1016)

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

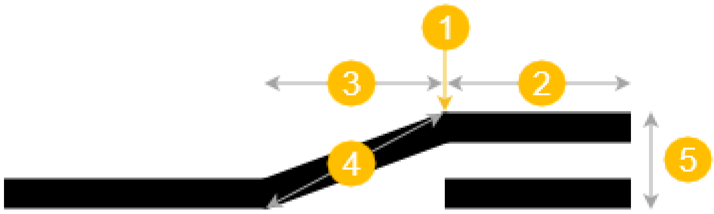
These properties are available when **Split type** is **Lapping**.

Option	Description
Lapping type	Select **Standard lapping** or **Custom lapping**.

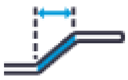
Option	Description
Lap length	With custom lapping, enter the length of the lap splice. With standard lapping, the lap length is read from the rebar catalog (`rebar_database.inp`).
Lap side	Select the side of the lap splice from the splitter: • **Lap left** • **Lap right** • **Lap middle**
Lap placement	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.

Option	Description
	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

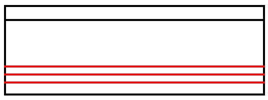
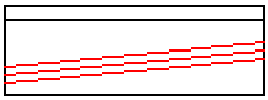
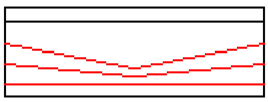
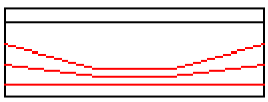
Modify a rebar set locally using modifiers (page 533)

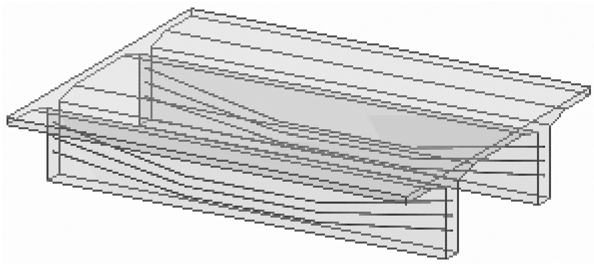
Rebar set properties (page 1016)

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.

Option	Description
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).
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:

Option	Description
	
Debonding	
Debonded strands	Enter the strand number. The strand number is the selection order number of the strand.
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	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)**.
Subtype (IFC4)	
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.

See also

Create a reinforcement strand pattern (page 517)

Debond reinforcement strands (page 518)

11 Disclaimer

To see the third party open source software licenses, go to Tekla Structures, click **File menu** --> **Help** --> **About Tekla Structures** --> **3rd party licenses** and then click the option.

The elements of the software described in this Manual are protected by several patents and possibly pending patent applications in the United States and/or other countries. For more information, go to page http://www.tekla.com/tekla-patents.

Index

B

C

D

E

I

J

L

M

N

Q

R

S

T

U

V

W

X

Z