Tekla Structures 2023

Create drawings

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1 Quick start to Tekla Structures drawings

Read this article if Tekla Structures drawings are new to you!

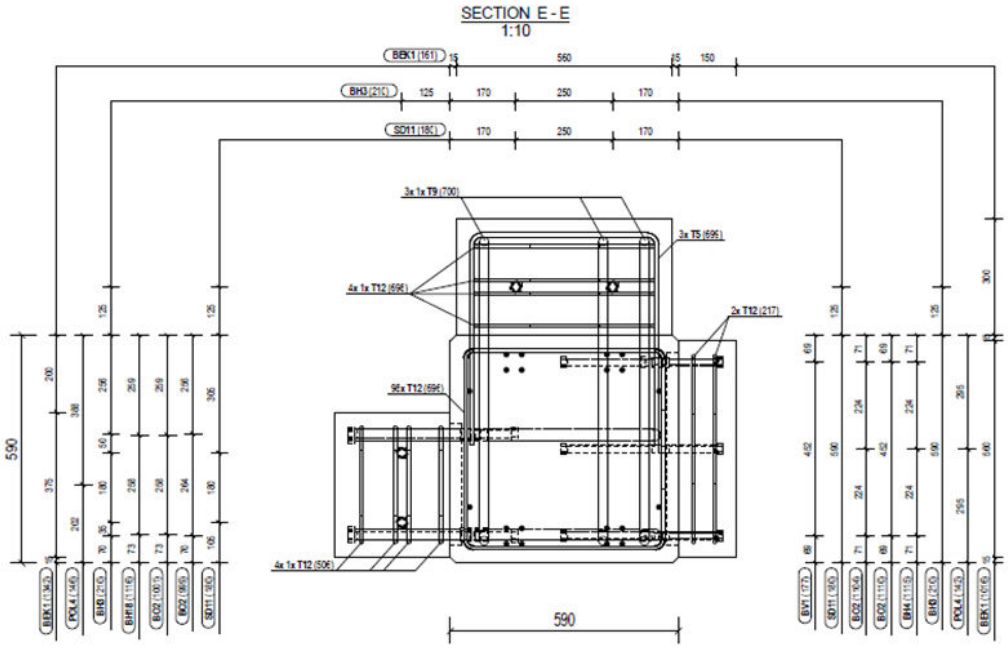
You will learn:

- What is specific to Tekla Structures drawings, and what is included in the drawings
- What to do before creating any drawings
- How to create drawings in your first project using the predefined settings in your environment
- How to modify the created drawings manually in the drawing mode

1.1 Basics of Tekla Structures drawings

- In Tekla Structures, a drawing is a window to the model presenting the 3D model structures in 2D, together with object dimensions and other

markings derived from model data. Below is an example of a drawing view containing building objects, marks and dimensions:



- From Tekla Structures, you can get general arrangement drawings (page 89), single-part drawings (page 94), assembly drawings (page 97), cast unit drawings (page 99), and multidrawings (page 102).

Up-to-date drawings

Tekla Structures drawings are always up to date (page 46) because:

- The building objects shown in the drawing are exactly the same as the model objects created in the model. You can change their representation (page 1004) in the drawing, but you cannot change the geometry or the location of the building object, or delete building objects; all changes to building objects are made in the model.
- Most objects in the drawing are associative (page 46) and automatically updated when the corresponding model objects change. For example, if you resize a model object, the associated dimension points move with the corresponding object in the drawing, the dimensions are recalculated, and the related data is updated in markings. Still, you do not lose any manual changes that you have made in the drawing. Note that if the dimension points do not move, they are not associated with any building objects.

Drawing objects, views and drawing layout

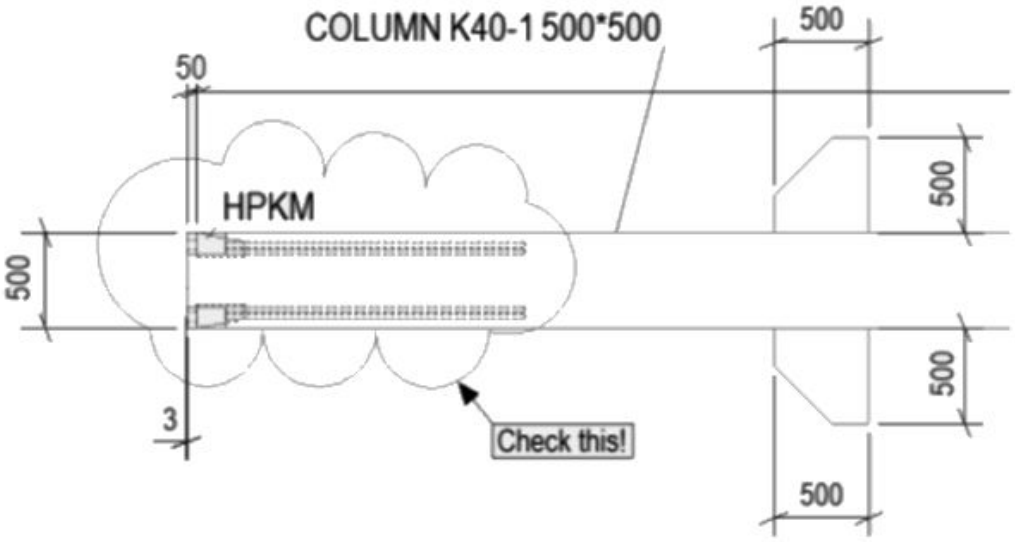
Drawing objects are organized inside *drawing views*, which are placed in the selected *drawing layout* according to the selected settings:

- Drawing objects (page 68) include *building objects* (parts, bolts, welds, chamfers, reinforcement, surface treatment, etc.), *annotation objects*

(marks, notes, dimensions, texts, linked objects, reference objects, etc.) and *sketch objects* (lines, rectangles, clouds, circles, etc.).

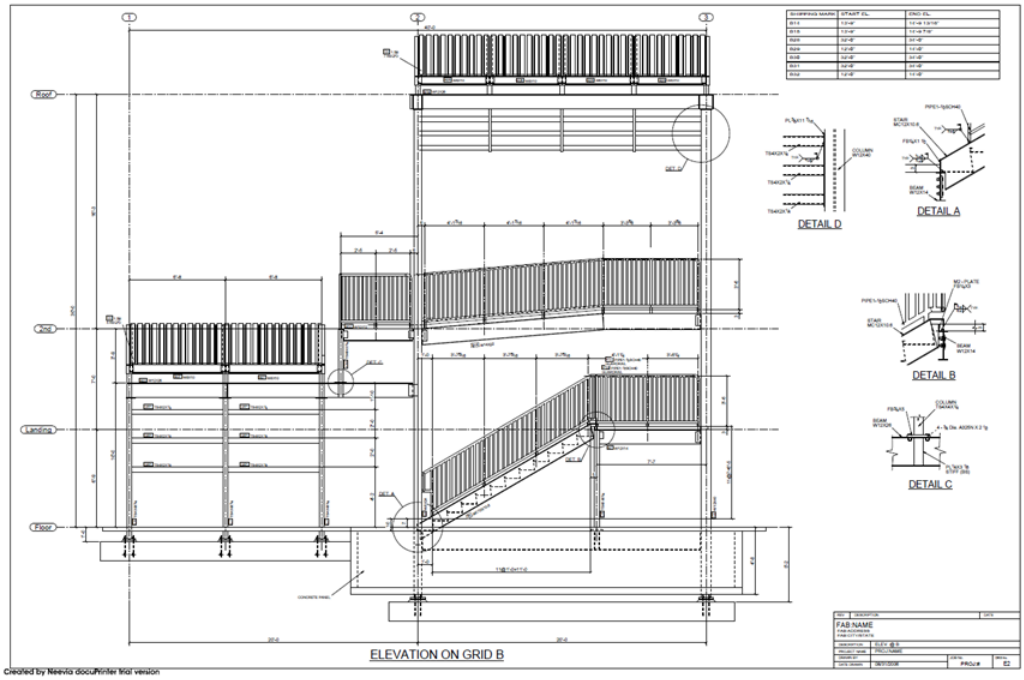
All these objects are adjustable.

Some examples building objects, dimensions, marks, texts, and clouds:

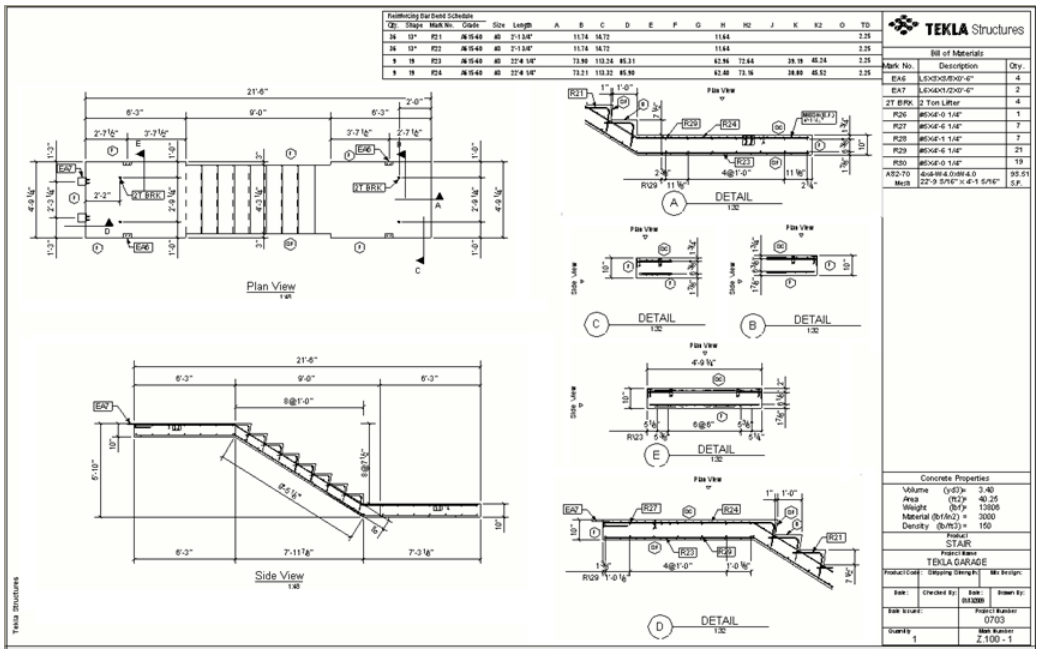


- Drawing views (page 179) act as containers for the building objects, or for areas in the model that you have selected to include in the drawing. The drawing view size adjusts automatically to fit in more content when necessary. Drawing views can show building objects from different directions (top, front, back, bottom), or as cross sections. View settings, for example the view depth and scale, are adjustable.

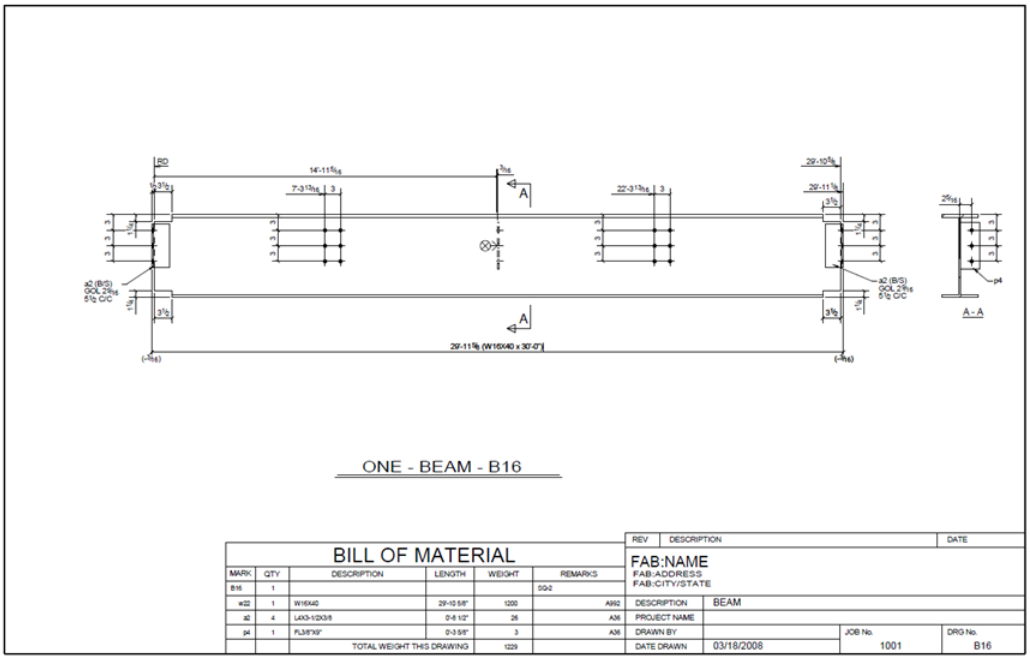
  Example of a GA drawing with an erection elevation view and detail views:



Example of a combined form and reinforcing bar drawing of stairs with landings with two main views and a couple of detail views:



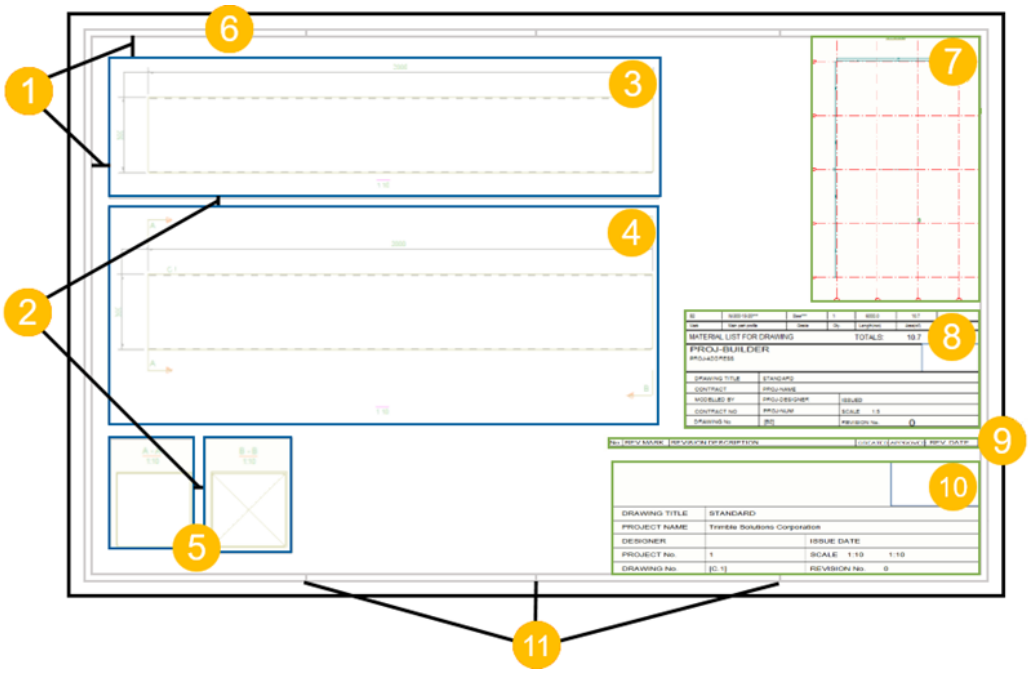
Example of a beam assembly drawing with one main view and a section view:



- A drawing layout (page 655) defines:
  - The drawing size, and spaces between the drawing frame and views, and between the drawing views (page 656)
  - The drawing frames and fold marks (page 656)
  - The table set (page 669) including a revision table, title block, material list, bill of materials, and general notes.

    The data in the tables is derived directly from the model data. You can create tables in Template Editor, and add tables in the drawing layout and adjust the layout settings in Layout editor (page 656).

  Below is an example of a typical fabrication drawing layout:



(1) Margins between the drawing frame and the outermost views

(2) Spaces between the views

(3) Top view

(4) Front view

(5) Section views A-A and B-B

(6) Drawing frame

(7) Key plan

(8) Material list

(9) Revision table

(10) Drawing title block

(11) Fold marks

Predefined drawing settings

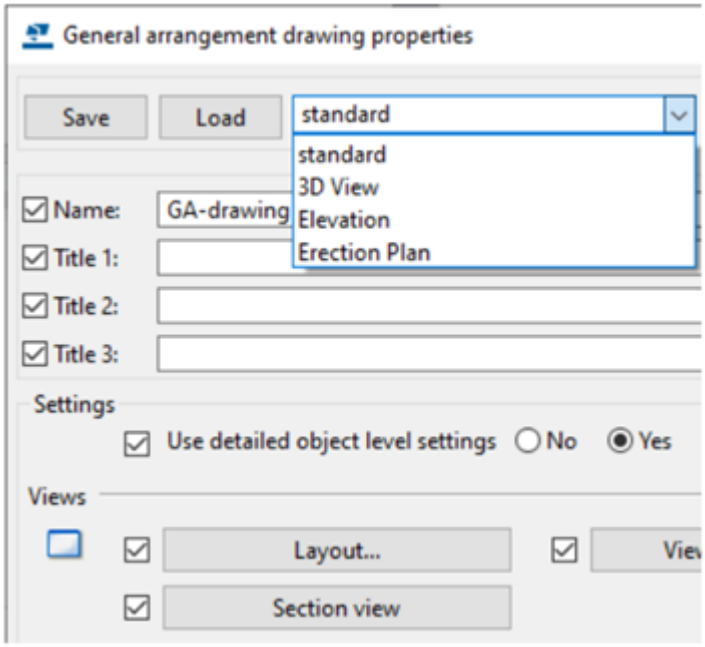
Your environment contains predefined drawing settings suited for various purposes, on multiple levels.

When you create a drawing for the first time, all you have to do is to **pick the most suitable predefined drawing settings file** from the list in the drawing properties dialog box.

Note that the drawing level settings also define which layout is used.

You can modify these settings and save your own settings files for your future drawings.

Below is an example of a list of drawing settings in a GA drawing:



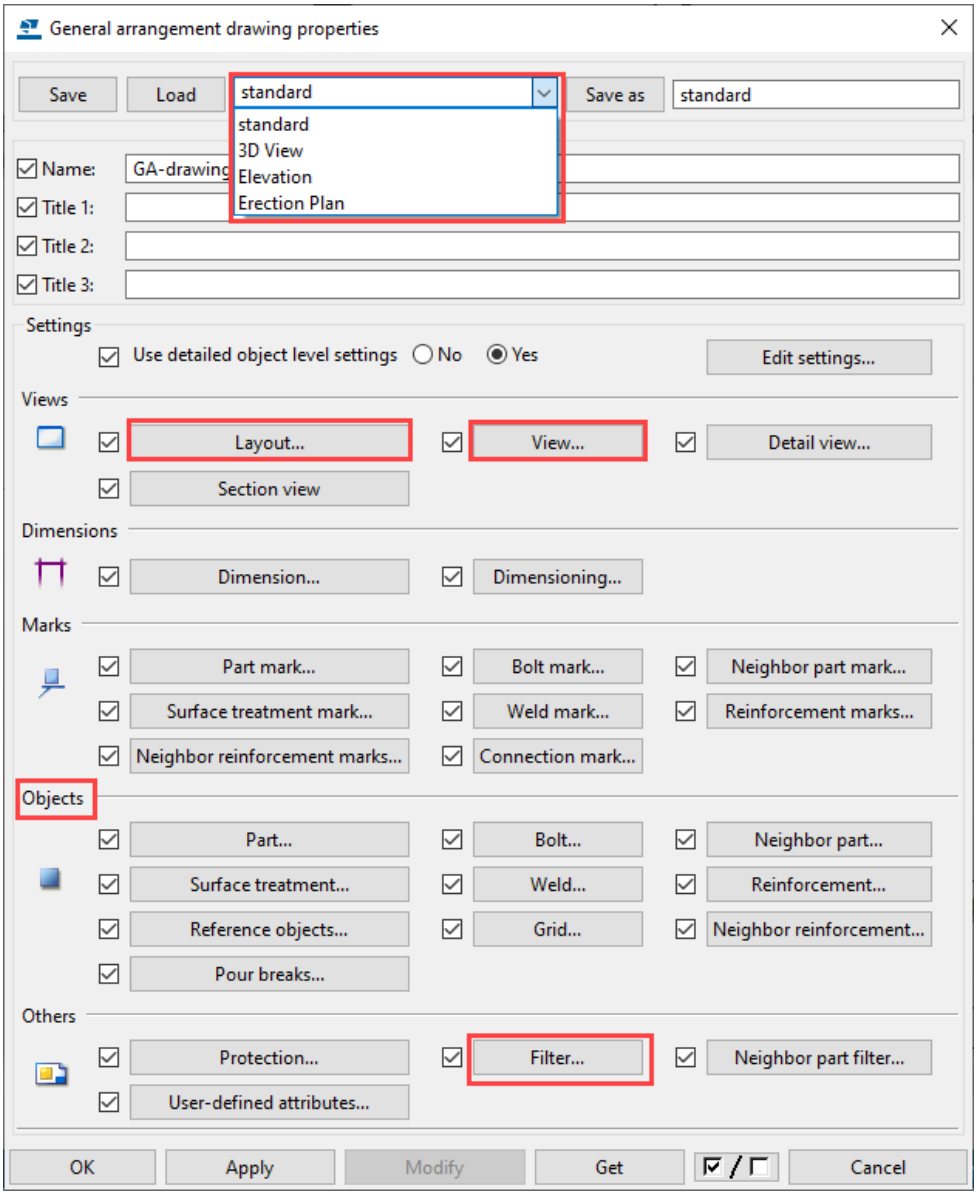
1.2 Before creating drawings in Tekla Structures

- You can create drawings at any stage of the project, but to minimize re-work, take the detailing as far as possible, and finish numbering before drawing creation (page 105). Numbering is a prerequisite for creating single-part, assembly, and cast unit drawings.
- Ensure that the object classifications and naming in your model matches filtering. Follow the company conventions.
- In the model, create all necessary views for GA drawings. GA drawing views will have the same orientation and content as the model view. For example, create an erection elevation view or a floor plan view. A good idea is to fit the work area in the model view using two points to select the area that you want to show in the drawing.
- Decide the view depth to use in GA drawings. Set the desired view depth in the initial model view for efficient and transparent workflow. The drawings will use the depth defined for the view in the model.
- Decide which predefined settings and drawing layout (page 656) you want to use. Select suitable default settings and a layout available in your environment.

1.3 Create a general arrangement drawing

In your first project, the idea is to create a general arrangement (GA) drawing using predefined settings from your environment, and then modify the drawing manually in the drawing mode as necessary.

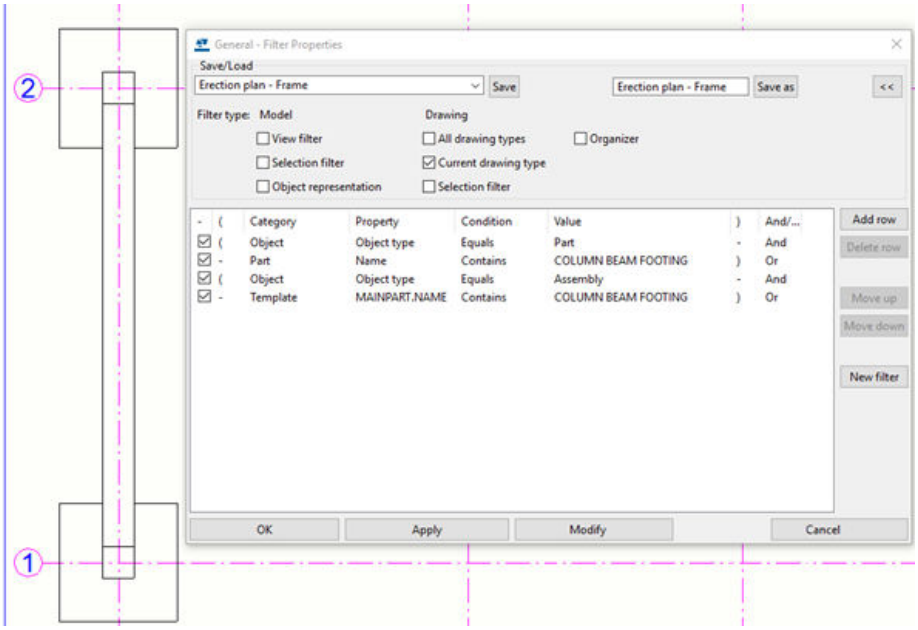
1. Go to the **Drawings & reports** tab on the ribbon, and select **Drawing properties** --> **GA drawing**.
   - Open the drawing settings list, and select the settings file with a name that suits your needs.



   - Check the general settings. If you modify the settings in the subdialog boxes, remember to click **OK**:
     - Define **Name** and **Title 1** - **Title 3**.

- Go to **Layout...** settings and change the layout to another one if necessary.
- Go to **View...** settings and select the **Scale** and the **Label**. The depth is taken automatically from the model view.
- Go to building object (part, reinforcement, surface treatment, etc.) settings and modify the representation.
- Go to **Filter...** settings and create filters. Decide which objects you want to show in the drawing, and filter other objects away.

  In the following example, you only want to show columns, beams, and footings:



- Click **Apply** in the **General Arrangement Drawing Properties** dialog box to apply the changes to the drawing that you will create. Also, save the drawing settings so that you can use the saved settings in your next projects.

2. On the **Drawings & reports** tab on the ribbon, click **Create drawings** --> **GA drawing**.
3. In the **Create general arrangement drawing** dialog box, select the view from which you want to create the drawing among the views that you created earlier in the model.

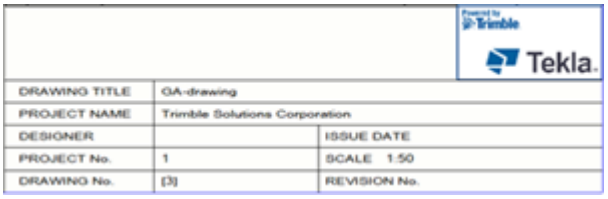
   GA drawings are based on model views, so select a suitable floor plan view or grid line view, for example.
4. To create the drawing, click **Create**.

1.4 Modify a GA drawing

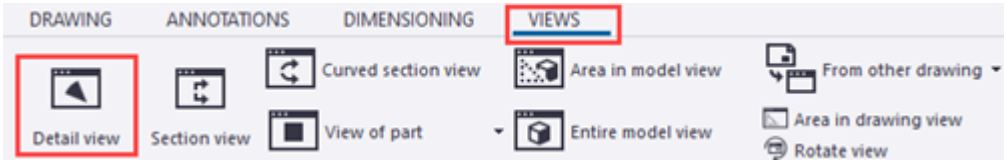
In the drawing mode, modify the drawing manually to get the desired result.

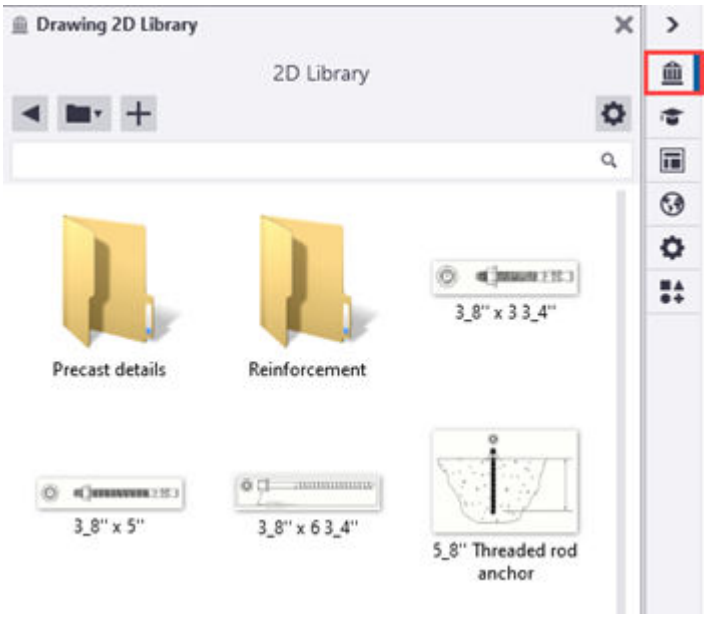
1. On the **Drawings & reports** tab on the ribbon, click **Document manager**, and select and open the GA drawing you created earlier.
2. Check the layout, tables and the title block; all content outside the views. To modify the layout, open Layout editor (page 656) by double-clicking a table in the layout.

   For example, double-click the title block:

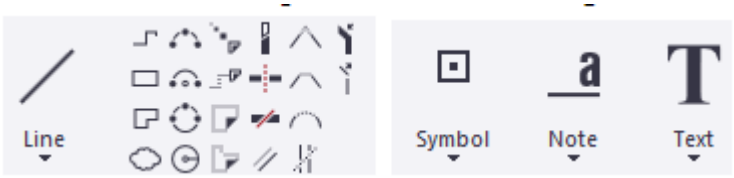


3. Check the view, and change the view settings (page 912) by **double-clicking the view frame**. The view frame is visible when your mouse pointer is within the view frame.
   - Is the scale appropriate?
   - Is the view label appropriate?
   - Does the view contain the building objects that you want? If not, change the object visibility settings and filtering.
   - Are you happy with the building object representations? If not, check the part, bolt, surface treatment, reinforcement, etc. representations.
   - When you are done, click **Modify**. Also save the view settings so that you can use them in your next projects.
4. Create the necessary detail views (page 180), and add 2D details from 2D Library (page 529). The view creation commands are located on the drawing **Views** tab, and 2D details in **Drawing 2D Library** in the side pane.





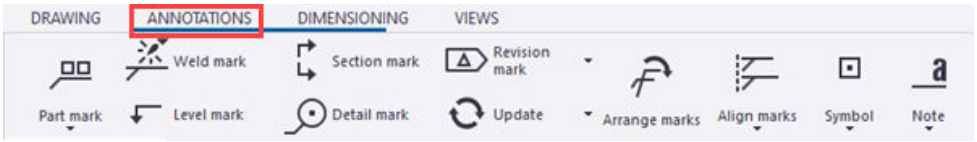
5. Check and modify the content of the views, one by one:
   - Note that after you start modifying the view content, you should not touch the view settings.
   - Add lines, graphics (page 388), symbols (page 371), and text (page 364). Tools for adding these are located on the drawing mode ribbon on the **Drawing** and **Annotations** tabs.

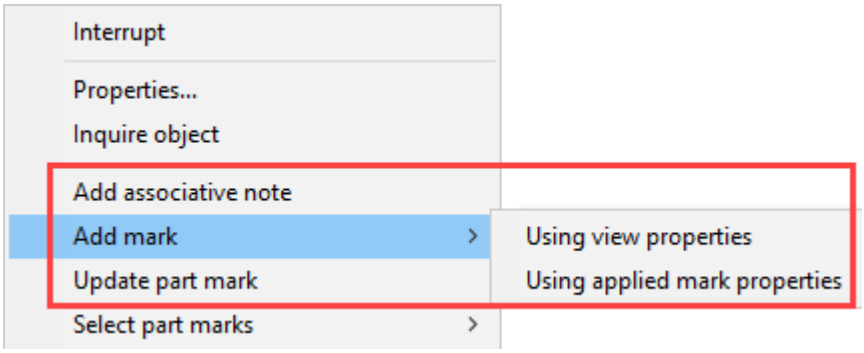


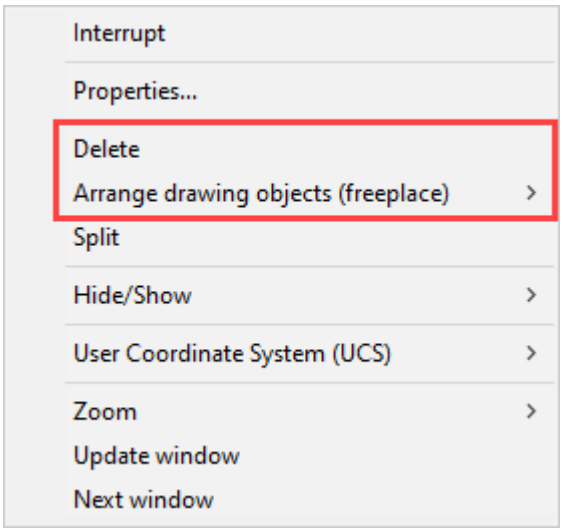
   - Remove undesired dimensions, add new dimensions (page 217), and double-click a dimension to change the dimension settings (page 260). Save the settings for later use. The dimensioning commands are located on the **Dimensioning** tab on the drawing ribbon.

   - Delete unnecessary marks (page 337), create new marks (page 301) and reinforcement marks (page 303), double-click a mark to change mark settings (page 335), and arrange (page 416) or align (page 417) the marks. Save the settings for later use. The related commands are

located on the **Annotations** tab on the drawing ribbon and in the context menu.

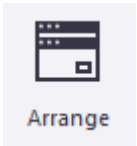






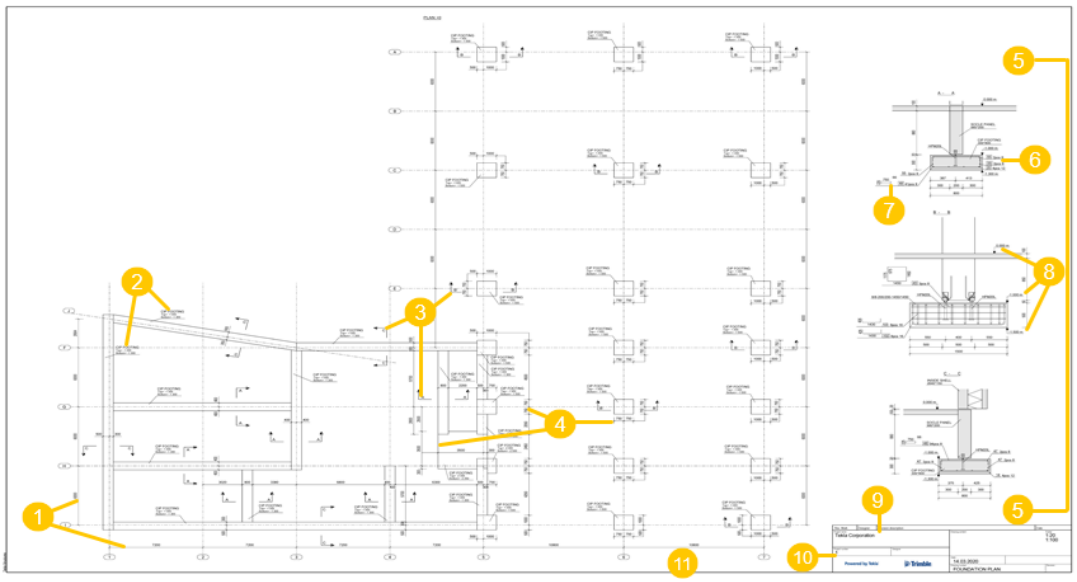
6. Arrange the views, or align all views vertically or horizontally with the main view. (page 200)

   The **Arrange** command is located on the **Views** tab on the drawing ribbon, and the alignment commands on the context menu.



When you are happy with the result, use this drawing as a template for creating drawings for other similar model content. You can also use drawings in the current project as templates in your future projects.

Below is an example of a foundation plan drawing.



(1) Grid dimensions

(2) Part marks

(3) Section marks A-A, B-B, and C-C

(4) Manual dimensions

(5) Section views A-A, B-B, and C-C

(6) Rebar group mark

(7) Rebar mark with pull-out picture

(8) Level marks

(9) Revision table

(10) Drawing title block table

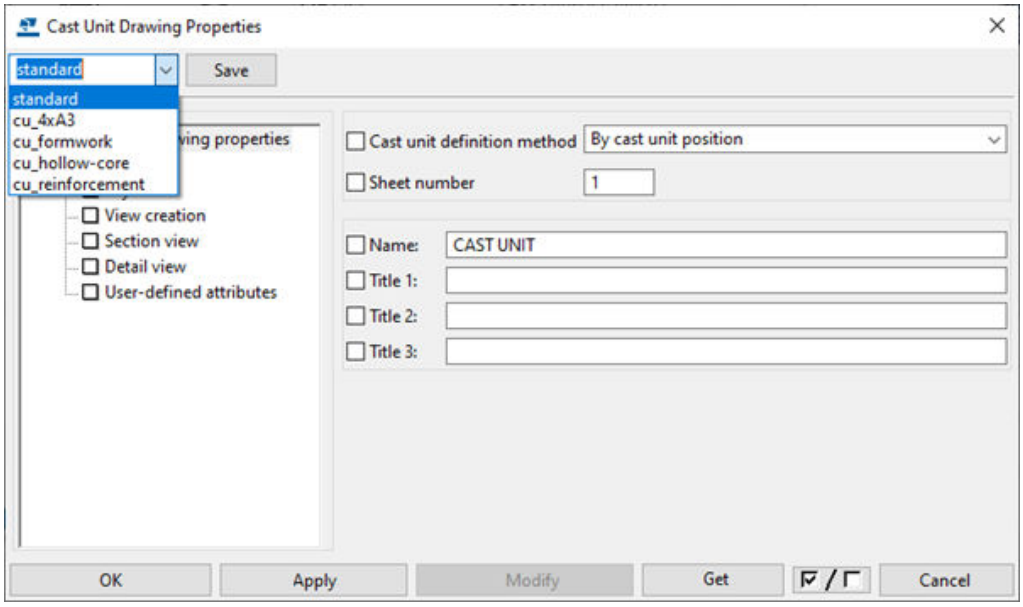
(11) Drawing frame and margin

1.5 Create a precast fabrication drawing

As this is your first project, we recommend that you create a precast fabrication drawing using predefined settings from your environment, and then modify the drawing manually in the drawing mode as necessary.

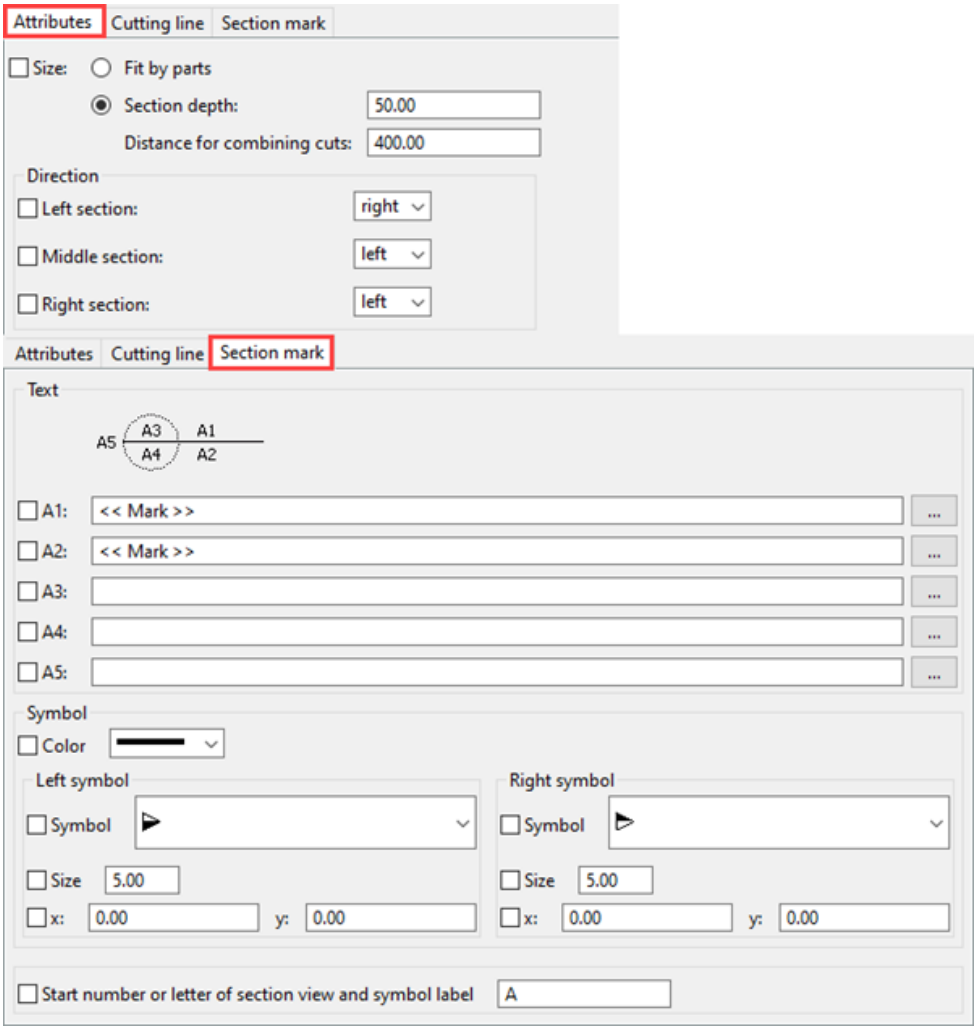
1. Go to the **Drawings & reports** tab on the ribbon, and select **Drawing properties** --> **Cast unit drawing**.

2. Open the drawing settings list, and select the settings file with a name that suits your needs.



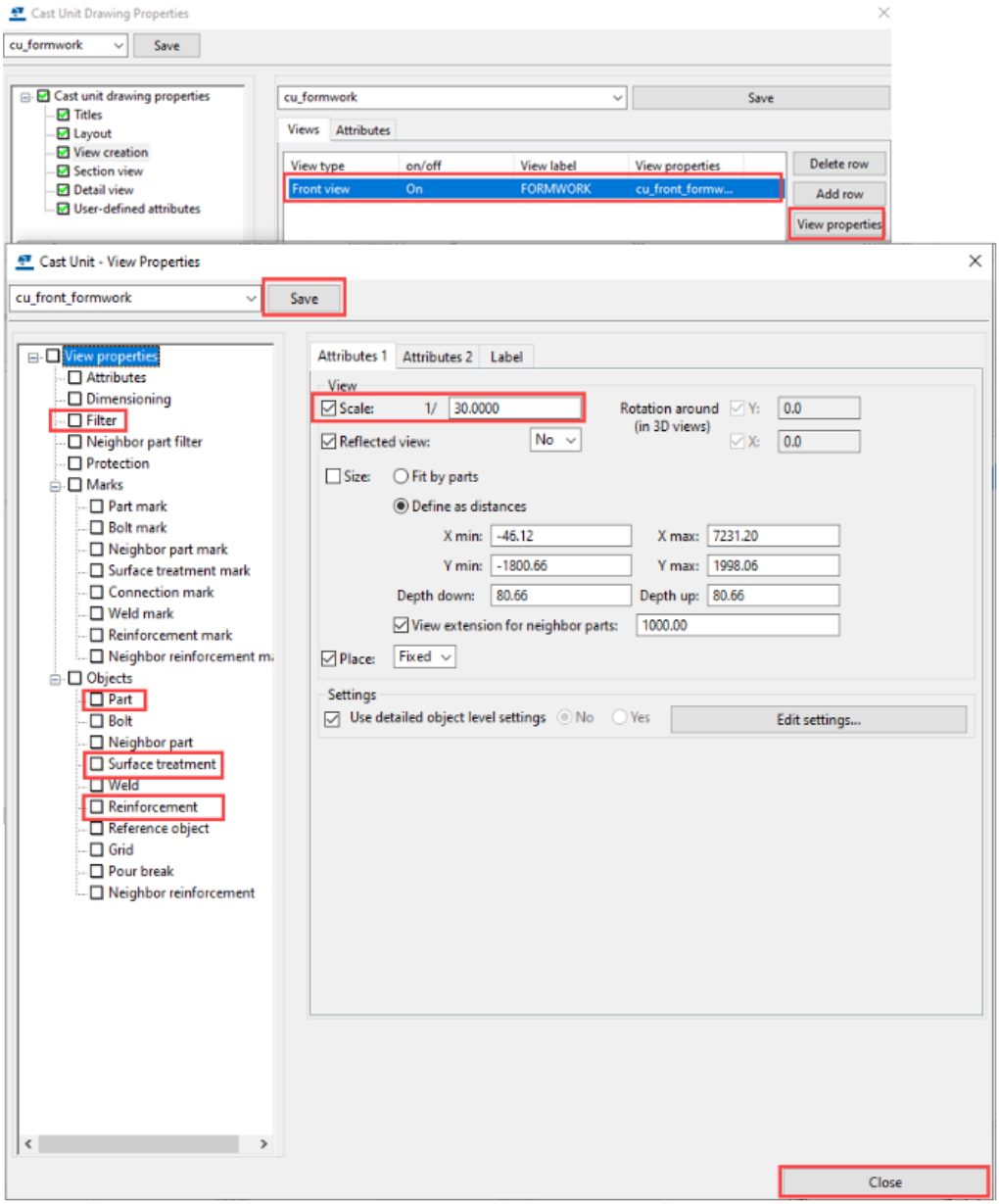
3. Check and change the general settings in **Cast Unit Drawing Properties**:
   - Define **Name** and **Title 1** - **Title 3**.
   - Go to **Layout** settings and change the layout to another one if necessary.

- Go to **Section view** and set the default section view depth and section mark content.



- Go to **View creation** and define at least one main view to create, for example, front view, and type a **Label** for the view.

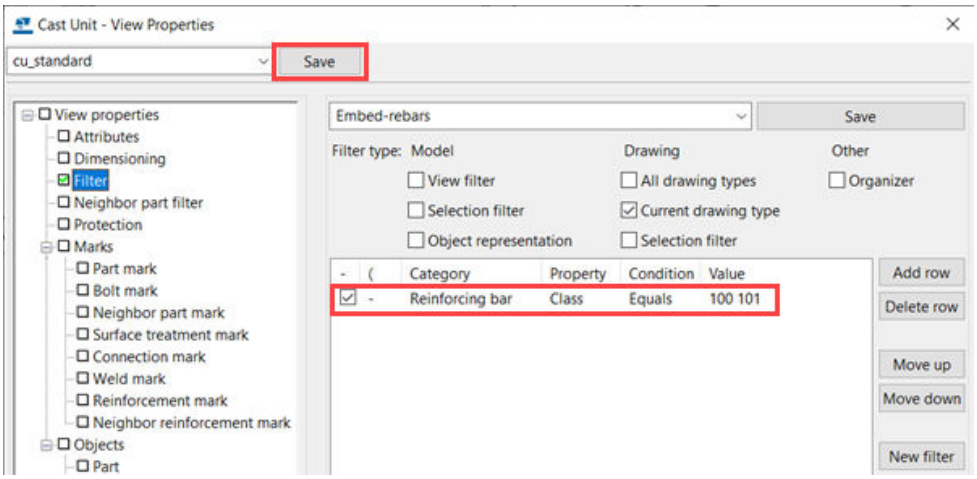
4. Select a view that you created from the view list and go to **View properties**. Modify the view settings (page 912). If you decide to create several main views, modify the settings for each view separately.



- Set the **Scale**.
- Modify building object settings and set the object representation.

  In your first project, you will probably have one concrete part and some embed parts. Set the part representation to either **Outline** or **Exact**. Set the reinforcement to **Visible**, and the reinforcement representation to **single line with filled ends**, for example. Also, remember to check surface treatment settings.
- Go to **Filter...** settings and create filters. Decide which objects you want to show in the drawing, and filter other objects away.

  In the following example, all other rebars have been filtered out, except embeds:



- When you are done, **Save** the view settings and click **Close**.

5. Click **Apply** in the **Cast Unit Drawing Properties** dialog box to apply the changes to the drawing that you will create. Also, save the drawing settings so that you can use the saved settings in your next projects.
6. Select the objects.

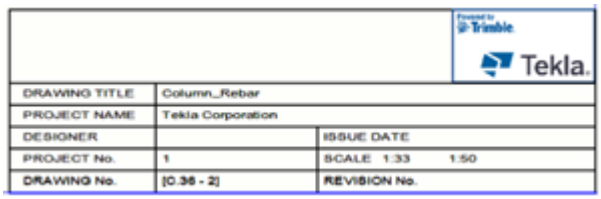
   Ensure that the correct selection switches are active, and select the entire model using area selection. You can also use selection filters to select objects.
7. On the **Drawings & reports** tab, click **Create fabrication drawing**, and ensure that the **Creation review** switch is enabled and click **Create fabrication drawing**.
8. In the **Creation review** dialog box, open the **Applied settings** list, select the settings that you modified earlier, and click **Create**.

1.6 Modify a precast fabrication drawing

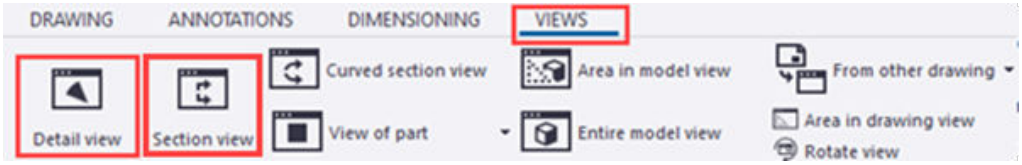
In the drawing mode, modify the precast fabrication drawing manually to get the desired result.

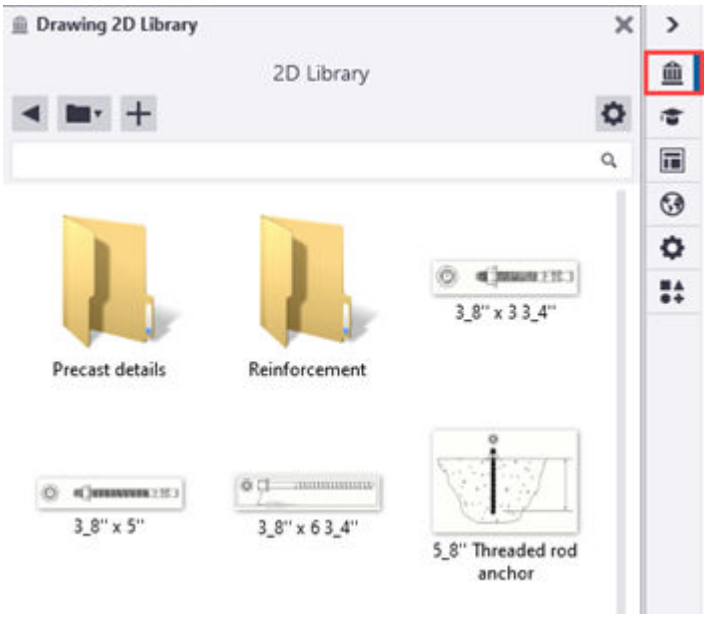
1. On the **Drawings & reports** tab on the ribbon, click **Document manager**, and select and open the precast fabrication drawing you created earlier.
2. Check the layout, tables and title blocks; all content outside the views. To modify the layout, go to Layout editor (page 656) by double-clicking a table in the layout.

For example, double-click the title block:

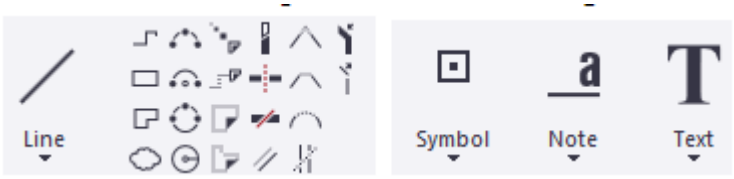


3. Check the settings in the main view that you created, and change the view settings (page 912) by **double-clicking the view frame**. The view frame is visible when your mouse pointer is within the view frame.
   - Is the scale appropriate?
   - Is the view label appropriate?
   - Does the view contain the building objects that you want? If not, change the object visibility settings and filtering.
   - Are you happy with the building object representations? If not, check the part, surface treatment, reinforcement, etc. representations.
   - When you are done, click **Modify**. Also save the view settings so that you can use them in your next projects.
4. Create other views (page 180) (section views, detail views), and check the view settings in the same way as for the main view you created. Also, add 2D details from 2D Library (page 529), and links to DXF files (page 380) and images (page 380). The view creation commands are located on the drawing **Views** tab, 2D details in **Drawing 2D Library** in the side pane, and the linking commands on the **Drawing** tab.



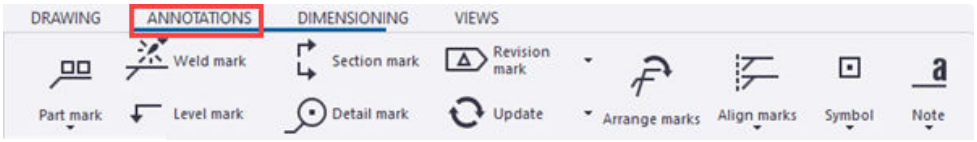


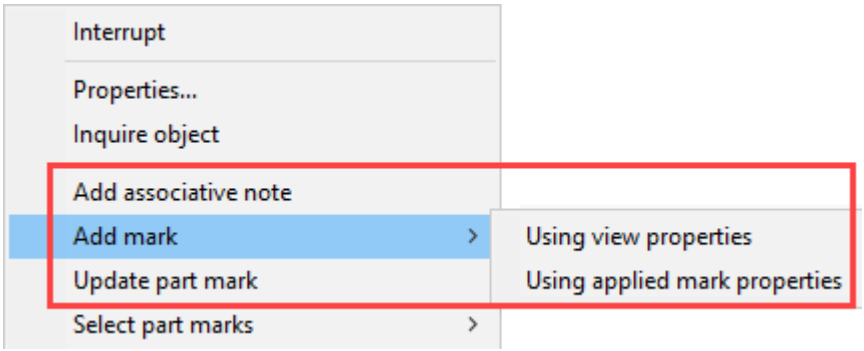
5. Check and modify the content of the views, one by one:
   - Note that after you start modifying the view content, you should not touch the view settings.
   - Add lines, graphics (page 388), symbols (page 371), and text (page 364). Tools for adding these are located on the **Drawing** and **Annotations** tabs on the drawing ribbon.

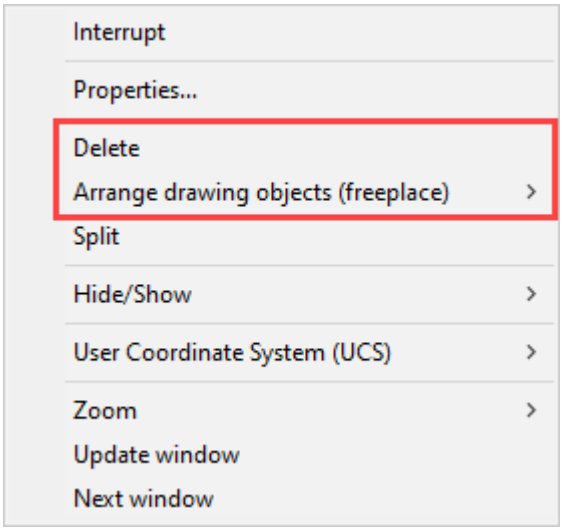


   - Remove undesired dimensions, add new dimensions to parts (page 207) and reinforcement (page 219), and change part dimension settings (page 260) and rebar dimension settings (page 932) by double-clicking a dimension. Save the settings for later use. The dimensioning commands are located on the **Dimensioning** tab on the drawing ribbon.

- Delete unnecessary marks (page 337), create new marks (page 301), reinforcement marks (page 303), and associative notes (page 329), double-click a mark to change mark settings (page 335), and arrange (page 416) or align (page 417) the marks. Save the settings for later use. The related commands are located on the **Annotations** tab on the drawing ribbon and in the context menu.

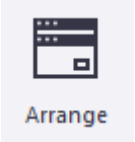






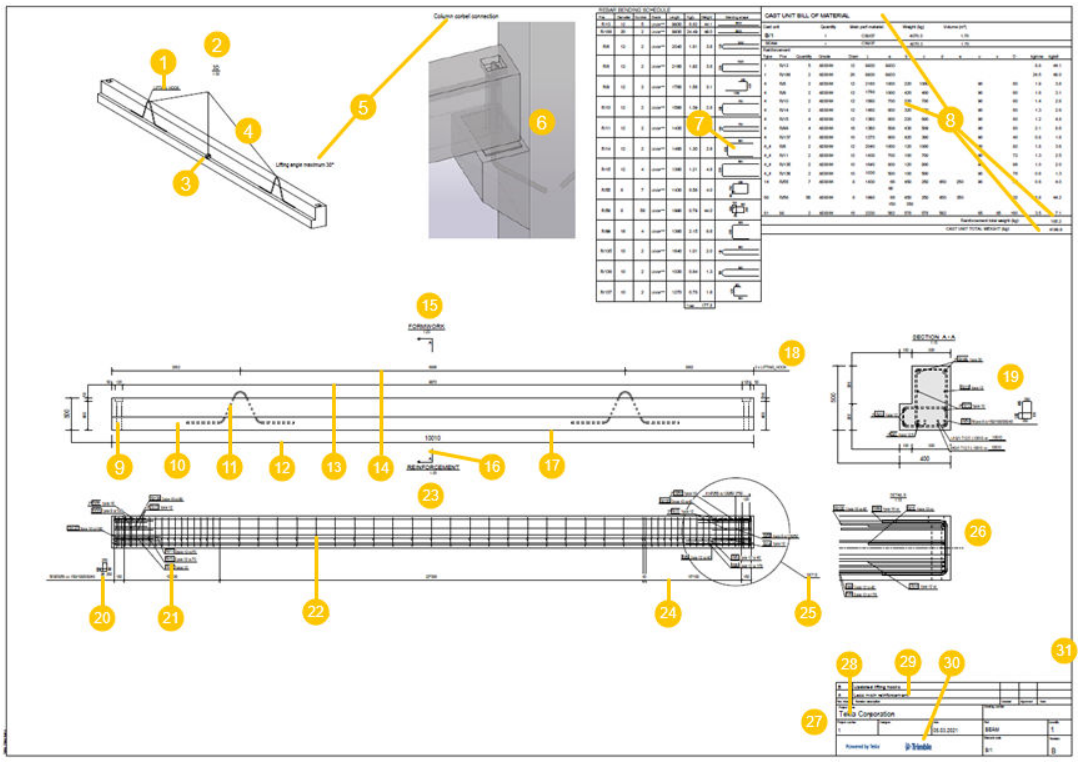
6. Arrange the views, or align all views vertically or horizontally with the main view. (page 200)

The **Arrange** command is located on the **Views** tab on the drawing ribbon, and the alignment commands on the context menu.



When you are happy with the result, use this drawing as a template for creating drawings for similar cast units (page 107).

Below is an example of a cast unit formwork drawing.



(1) Associative note

(2) 3D view created automatically

(3) Symbol (center of gravity)

(4) Line

(5) Text

(6) 2D detail from **Drawing 2D Library** or image

(7) Rebar bending schedule with graphical fields for pull-out pictures

(8) Material list table with table header row, table content rows, total reinforcement weight row and total cast unit weight sum row

(9) Recess with hidden lines

(10) Concrete part

(11) Embed part/rebar

(12) **Overall dimensions** created automatically

(13) **Recess dimensions** created automatically

(14) **Filter dimensions** created automatically

(15) Formwork front view

(16) Section mark A-A

(17) Part contour

(18) Dimension tag

(19) Section view A-A created automatically

(20) Rebar mark with pull-out picture

(21) Rebar group mark

(22) Rebar

(23) Reinforcement front view

(24) Rebar group dimension

(25) Detail mark

(26) Detail view

(27) Drawing title block table

(28) Attribute (Tekla Corporation) and fixed text (Product name) in table

(29) Revision table

(30) Image in table

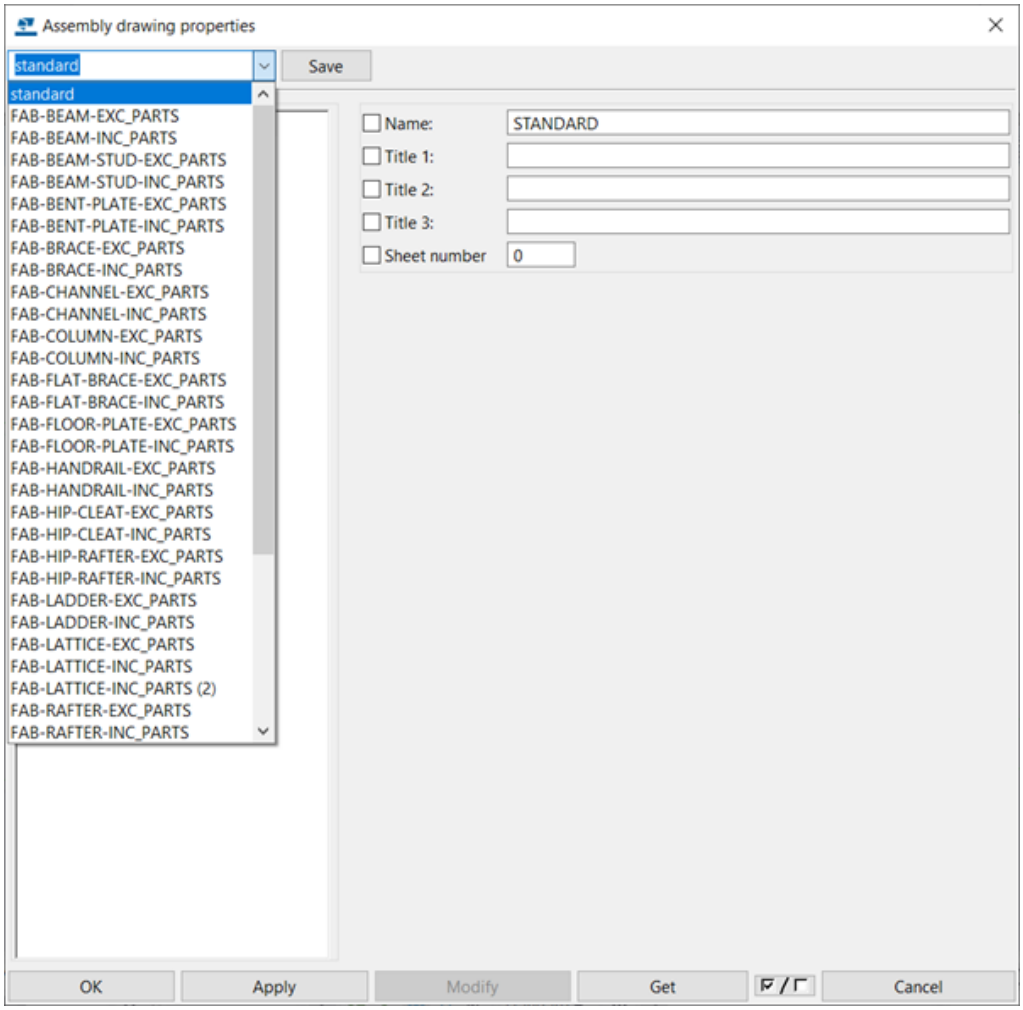
(31) Drawing frame and margin

1.7 Create a steel fabrication drawing

As this is your first project, we recommend that you create a steel fabrication drawing using predefined settings from your environment, and then modify the drawing manually in the drawing mode as necessary. The instructions below apply to assembly drawings.

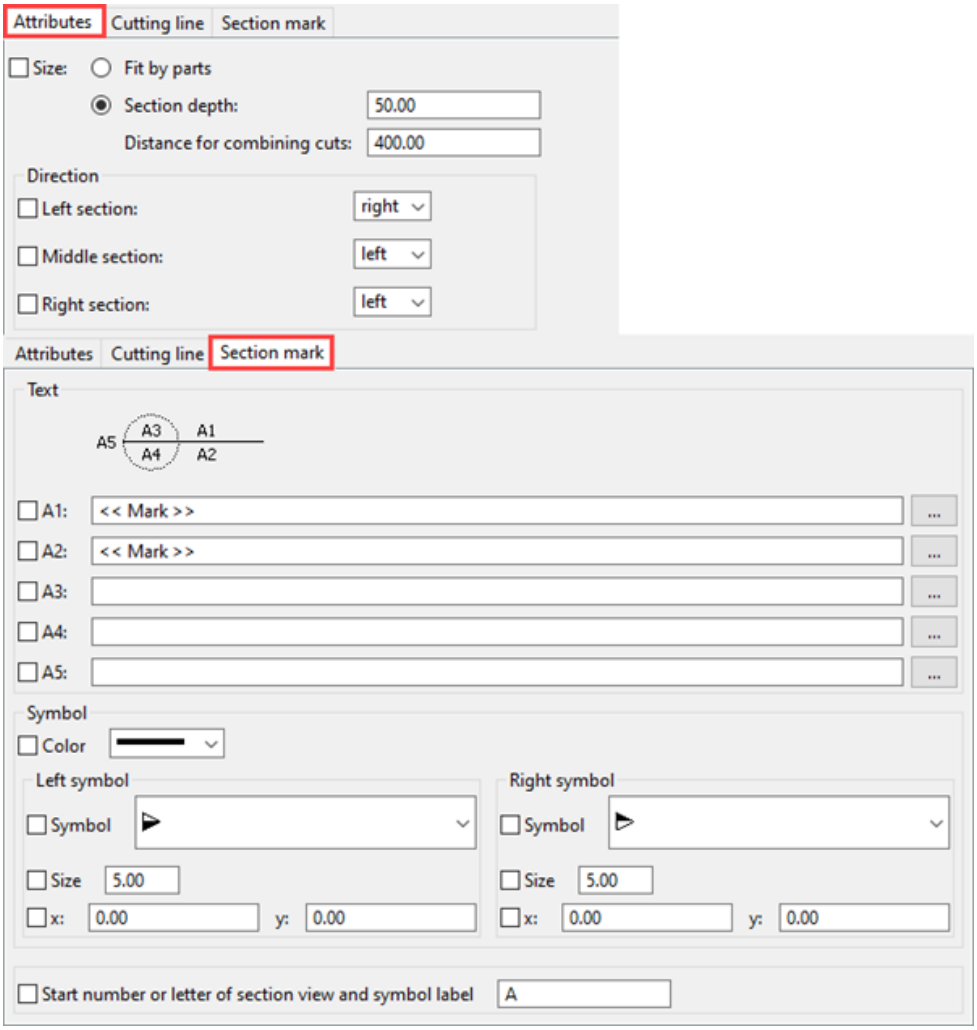
1. Go to the **Drawings & reports** tab on the ribbon, and select **Drawing properties** --> **Assembly drawing**.

2. Open the drawing settings list, and select the settings file with a name that suits your needs.



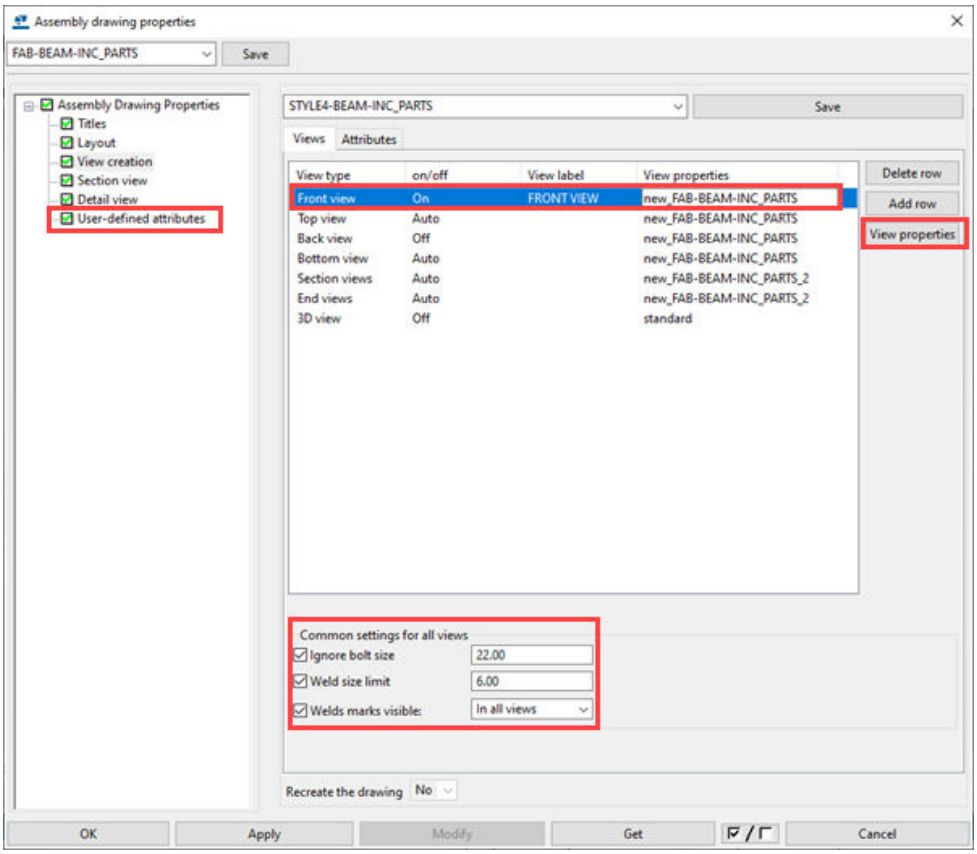
3. Check and change the general settings in **Assembly Drawing Properties**:
   - Define **Name** and **Title 1** - **Title 3**.
   - Go to **Layout** settings and change the layout to another one if necessary.

- Go to **Section view** and set the default section view depth and section mark content.

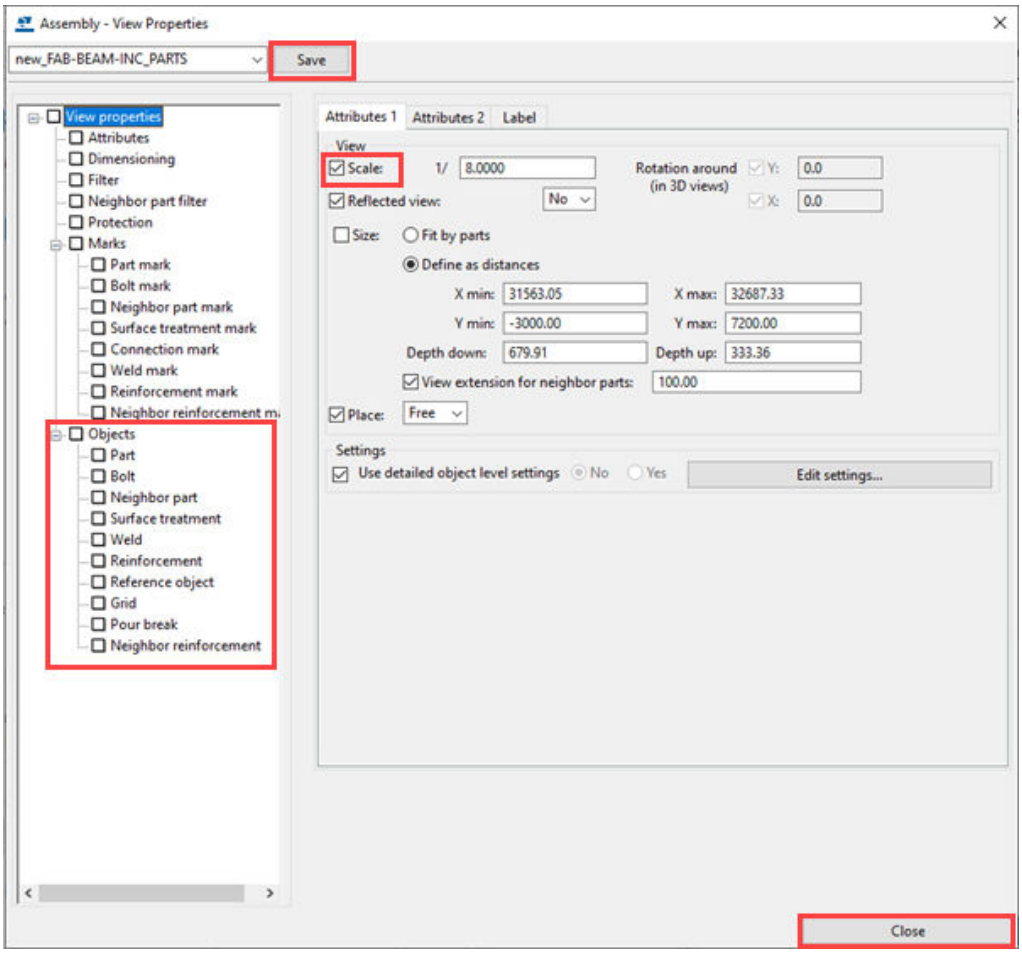


- Go to **View creation** and define at least one main view to create (page 689), for example, front view, and type a **Label** for the view. You can also create section views and end views here.

  Also, define the bolt and weld visibility settings (page 906) common to all views. Check and fill in the user-defined attributes (page 906) that are common to all your fabrication drawings.



4. Select a view that you created from the view list, for example the front view, and go to **View properties**. Modify the view settings (page 912). If you decided to create several main views, modify the settings for each view separately.



- Set the **Scale**.
- Go to building object (part, neighbor part, bolt, weld, reference object, grid, etc.) settings and set the object visibility and representation.

  For parts, **Outline** usually works for parts shown in main views. **Exact** is ideal for parts in detail, section, and end views, because it shows the true contours of hot-rolled profiles, for example. For welds, you can define whether to show the modeled welds or not (page 1036). For bolts, you can define whether to show bolt holes and bolt axis, or actual bolts (page 1010). If you prefer it simple, use the hole representation.
- When you are done, **Save** the view settings and click **Close**.

5. Click **Apply** in the **Assembly Drawing Properties** dialog box to apply the changes to the drawing that you will create. Also, save the drawing settings so that you can use the saved settings in your next projects.
6. Select the objects.

   Use selection filters to select objects. When you create assembly drawings, ensure that the assembly selection switch is active.

7. On the **Drawings & reports** tab, click **Create fabrication drawing**, and ensure that the **Creation review** switch is enabled and click **Create fabrication drawing**.
8. In the **Creation review** dialog box, open the **Applied settings** list, select the settings that you modified earlier, and click **Create**.

1.8 Modify a steel fabrication drawing

In the drawing mode, modify the steel fabrication drawing manually to get the desired result.

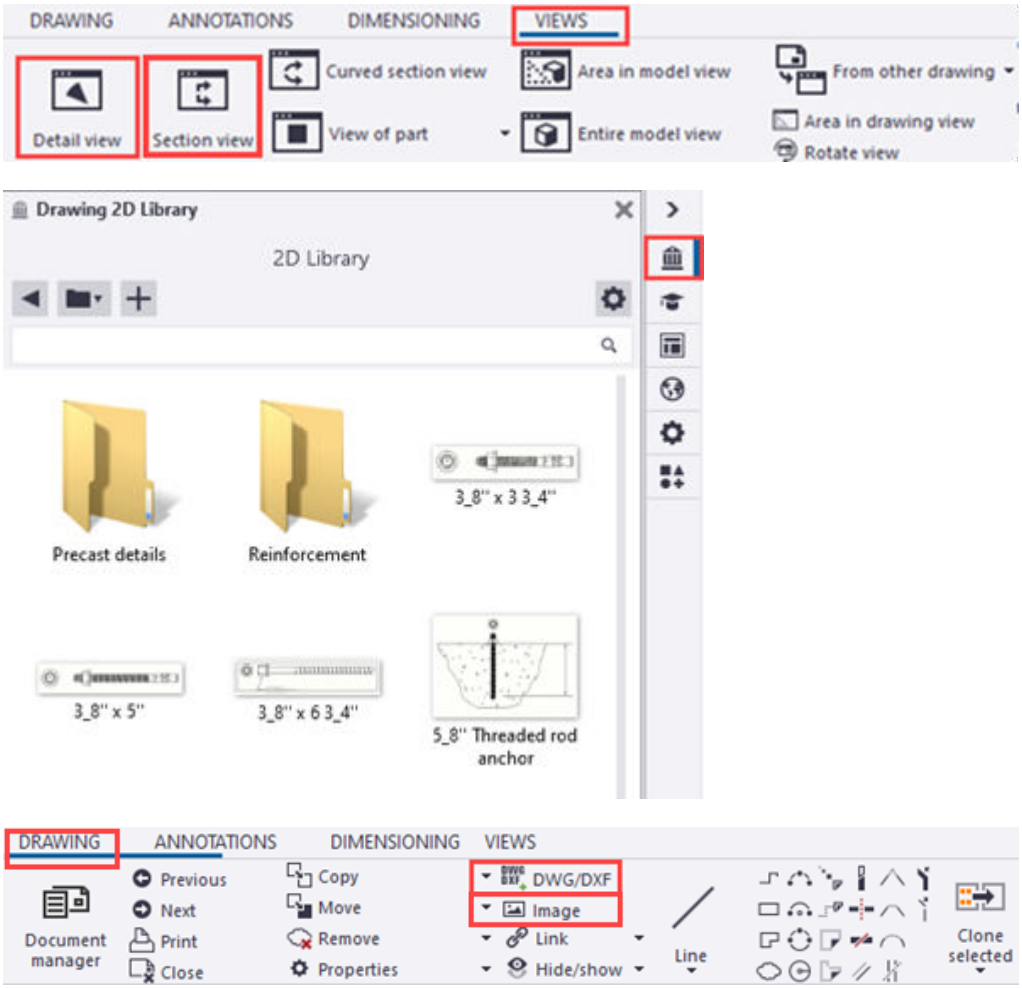
1. On the **Drawings & reports** tab on the ribbon, click **Document manager**, and select and open the steel fabrication drawing you created earlier.
2. Check the layout, tables and title blocks; all content outside the views. To modify the layout, go to Layout editor (page 656) by double-clicking a table in the layout.

   For example, double-click the shop material list:

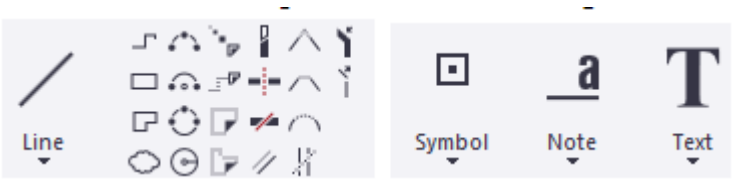
SHOP MATERIAL LIST FOR 1 ASSEMBLY						
Mark	Size	Grade	No.	Length (mm)	Area (m^2)	Weight (kg)
1001	PL10*140	S235JR	2	140	0.087	3.0
1002	L150*100*10	S235JR	4	200	0.419	15.1
1004	PL20*350	S235JR	1	450	0.347	24.7
c/1	HEA400	S235JR	1	7180	14.089	860.8
				Total	14.943	903.6

3. Check the settings in the main views (top, front, bottom, back) that you created, and change the view settings (page 912) by **double-clicking the view frame**. The view frame is visible when your mouse pointer is within the view frame.
   - Is the scale appropriate?
   - Is the view label appropriate?
   - Does the view contain the building objects that you want? If not, change the object visibility settings.
   - Are you happy with the building object representations? If not, check the part, surface treatment, bolt, weld, etc. representations.
   - When you are done, click **Modify**. Also, save the view settings so that you can use them in your next projects.
4. Create other views (page 180) (section views, detail views), and check the view settings in the same way as for the main views you created. Also, add 2D details from 2D Library (page 529), or links to DXF files (page 380). The view creation commands are located on the drawing **Views** tab, the

Drawing 2D Library in the side pane, and the commands for adding links on the **Drawing** tab.



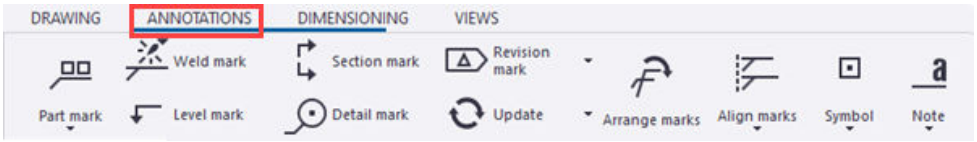
5. Check and modify the content of the views, one by one:
   - Note that after you start modifying the view content, you should not touch the view settings.
   - Add lines, graphics (page 388), symbols (page 371), and text (page 364). Tools for adding these are located on the drawing mode ribbon on the **Drawing** and **Annotations** tabs.

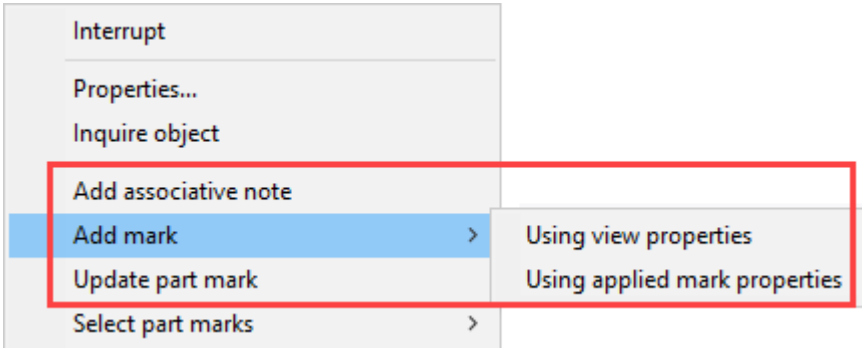


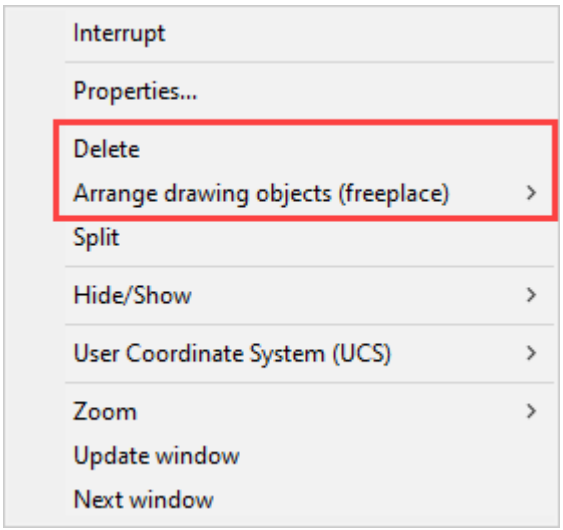
   - Remove undesired dimensions, add new dimensions to parts and bolts (page 207), and double-click a dimension to change dimension settings (page 260). Save the settings for later use. The dimensioning

commands are located on the **Dimensioning** tab on the drawing ribbon.

- Delete unnecessary marks (page 337), create new marks (page 301), double-click a mark to change mark settings (page 335), and arrange (page 416) or align (page 417) the marks. Also, add manual section marks (page 343). Save the settings for later use. The related commands are located on the **Annotations** tab on the drawing ribbon and in the context menu.

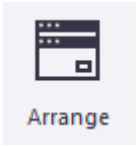






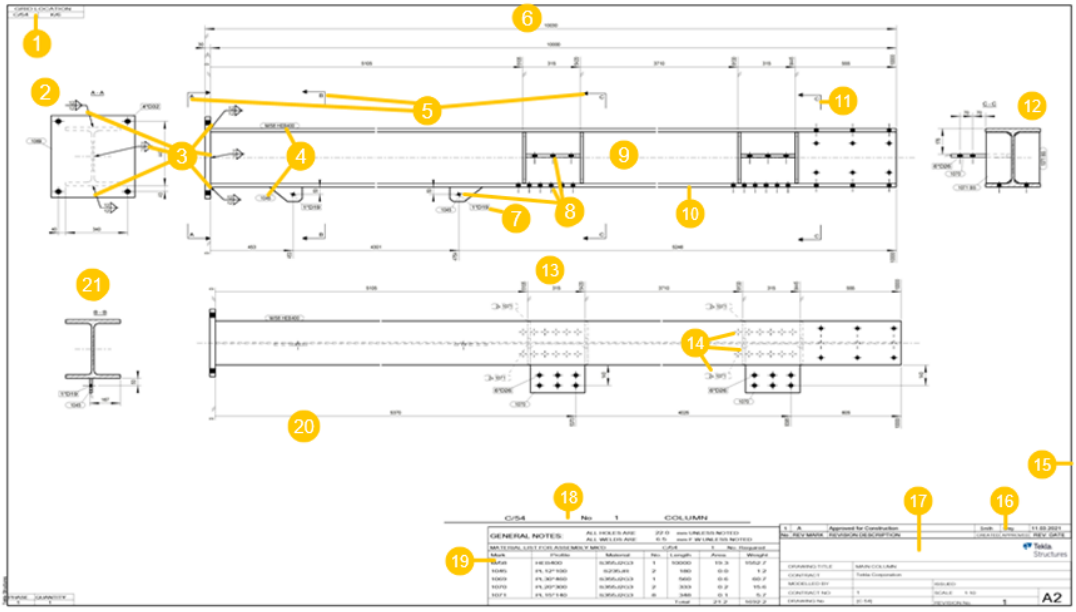
6. Arrange the views, or align all views vertically or horizontally with the main view. (page 200)

The **Arrange** command is located on the **Views** tab on the drawing ribbon, and the alignment commands on the context menu.



When you are happy with the result, use this drawing as a template when you are creating drawings for similar assemblies (page 107).

Below is an example of a column assembly drawing.



(1) Grid location table tells on which grid line the assembly is located in the model

(2) Section A-A created automatically, manual marks and dimensions

(3) Model weld marks

(4) Part marks

(5) Section marks A-A, B-B, and C-C

(6) Front view

(7) Bolt mark

(8) Bolts

(9) Column C/54

(10) Column contour

(11) Manual section mark C-C (without a section view). The identifier is C-C to indicate that this section is identical to the C-C section that has a section view

(12) Section C-C created automatically, manual marks and dimensions

(13) Top view

(14) Bolts, plates and marks hidden by other parts

(15) Drawing frame and margin

(16) Revision table

(17) Drawing title block

(18) Call off table indicating the total number of main parts in the drawing

(19) Material list table

(20) Dimensions: Most dimensions are automatic **Integrated dimensions**. The section view dimensions are manual.

(21): Section B-B created automatically, manual marks and dimensions

2 Drawings in Tekla Structures

Tekla Structures drawings contain a large variety of features and tools that help you to create and manage your drawings efficiently.

Main drawing features

- The model is the single source of information for drawings. The drawing is just another view to the model, usually a 2D view.
- Creating drawings is fast, efficient and controlled.
- Automatic workshop single-part and assembly drawings, and cast unit drawings of selected parts with predefined settings for layout, views, dimensions, marks, and building objects. View properties are defined separately for each view before a drawing is created.
- Automatic general arrangement drawings and anchor bolt plans of selected views.
- Drawing objects are associated with model objects, and updated when the model changes.
- Changing some of the properties requires the recreation of the drawings.
- If there are several identical parts, cast units or assemblies in the model, Tekla Structures creates only one drawing for each of them.
- You can modify drawing properties on three levels; drawing, view and object level, depending on the drawing type and the desired results.
- The drawings consist of three main types of elements: drawing layout, drawing views, and drawing objects. You can select what to include in the drawing before you create it, and also add, modify and delete objects in an existing drawing.
- You can check previews of the drawings, and print your drawings to pdf, printer or plot files.
- You can control drawing revisions, and issue, lock, and freeze drawings.
- You can use interactive editing tools for adding dimensions, various sketch objects, marks, notes, texts, symbols, images and links in the drawings.

Drawings are integrated with models

Tekla Structures integrates the drawings with the model. A drawing is a window to the model presenting 3D structures in 2D. The building objects shown in the drawing are model objects you create in the model. You can change their representation in the drawing but you cannot change the geometry or the location of the building object, or delete building objects; all changes to building objects are made in the model. That is why the drawings are always up to date, and dimensions and marks in drawings are always correct, for example. You can filter out parts and bolts in drawings using the filtering tools, or make them invisible by hiding them.

You can create drawings at any stage of the project. Creating single-part, assembly and cast unit drawings requires that the model is numbered, so you need to plan and perform the numbering before creating drawings. For more information about numbering, see Number the model.

If the model changes, Tekla Structures notifies in **Document manager** that you need to update the related drawings. You cannot open a drawing that is not up to date. For more information about updating the model, see Update drawings when the model changes (page 575).

Drawings are associative

Tekla Structures drawings are *associative*. The objects in the drawing are linked to model objects, which means that most objects in the drawing are automatically updated when the corresponding model objects change. For example, if a model object is resized, the dimension points move with the corresponding object in the drawing, and the dimensions are recalculated. Still you do not lose any manual changes that you have made in the drawing. This applies to all drawing types.

Tekla Structures updates the following drawing objects to reflect the changes in the model:

- Parts
- Marks
- Dimensions
- Welds
- Views
- Section marks
- Detail marks
- Associative notes
- Lines and other shapes
- Tables

Tekla Structures retains the following manual changes made to drawings:

- Base points of objects; for example, if you drag an object to a new location

- Object properties; for example, color, font, and line type

Advanced options related to associativity:

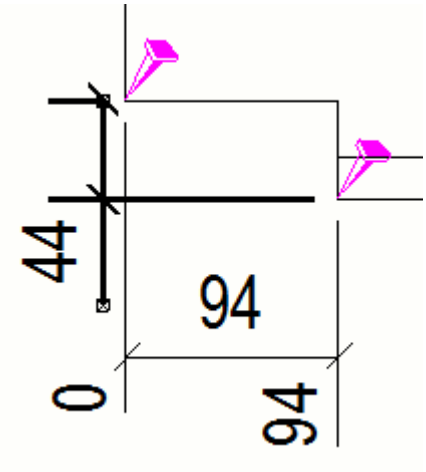
XS_ASSOCIATIVE_CHANGE_HIGHLIGHT_SIZE

XS_ASSOCIATIVE_CHANGE_HIGHLIGHT_SYMBOL

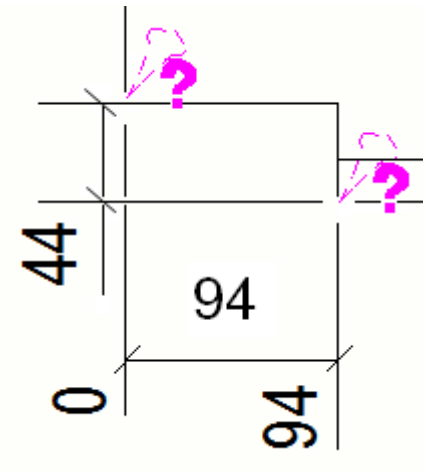
XS_HIGHLIGHT_ASSOCIATIVE_DIMENSION_CHANGES

Associativity symbol

In drawings, the associativity symbol indicates which drawing objects are associative and automatically updated. Associativity symbols are shown only when you select a drawing object, for example, a dimension.



Objects that do not have valid association get a ghost associativity symbol and a question mark. These symbols are shown constantly, even though the drawing object is not selected. This makes it easier to find objects that need attention.



The associativity symbols are not shown in printed drawings.

TIP To hide associativity symbols in drawing views, on the **File** menu, click **Settings** and select the **Associativity symbol** check box, or press **Shift +A** on the keyboard.

Find out more about drawings

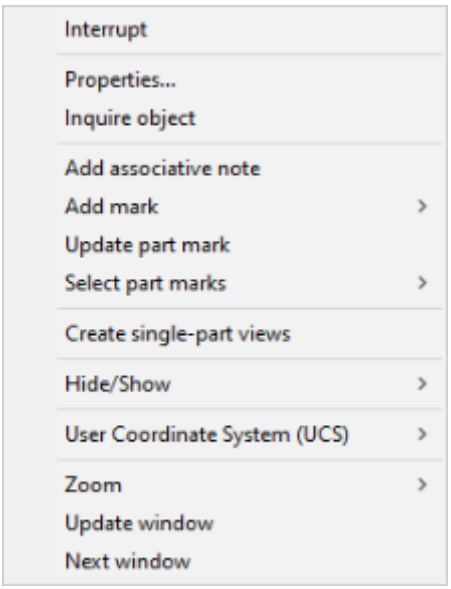
To	Click the links below to find out more
Learn what is specific to Tekla Structures drawings, what to do before creating any drawings, how to create drawings in your first project using the predefined settings in your environment, and how to modify the created drawings manually in the drawing mode	Quick start to drawings (page 15)
Check the controls available in the drawing mode, see how you can use snapping, zooming and panning in drawings, and	Introduction to drawing mode (page 50)
How to use snapping in drawings	Snapping in drawings (page 53)
How to zoom and pan in drawings	Zooming and panning in drawings (page 58)
Get familiar with colors and color modes available in drawings	Colors in drawings (page 61)
Learn what drawings consist of	Drawing objects, views and layouts (page 68)
Find out what happens when a drawing needs to be updated	How to keep your drawings up to date? (page 70)
Learn where you can change the drawing properties and in which order	Different levels of setting up and modifying drawing properties (page 71)
Find out the cases when a drawing needs to be updated	Recreation of drawings (page 88)
What kind of drawings can you produce in Tekla Structures	Drawing types (page 89)
Check the different drawing types available and learn how to create them using different methods	Create drawings in Tekla Structures (page 104)
Create drawings by cloning, and find out the situations when cloning is the best choice	Clone drawings (page 153)
Find the drawings that you need using searching and filtering in **Document**	Document manager (page 552)

To	Click the links below to find out more
manager, select drawings, and open drawings.	
Edit your drawings manually: add objects, modify properties, etc.	Edit drawings (page 166)
Update drawings when the model changes, lock drawings, freeze drawings, issue drawings, revise drawings, and delete drawings	Manage drawings (page 551)
Print drawings as PDF files, save them as plot files (.plt) for printing with printer/plotter, or print them on a selected printer	Print drawings (page 602)
Set up and modify automatic drawing settings	Define automatic drawing settings (page 650)
Check the available drawing properties and setting	Drawing settings (page 902)

2.1 Introduction to drawing mode

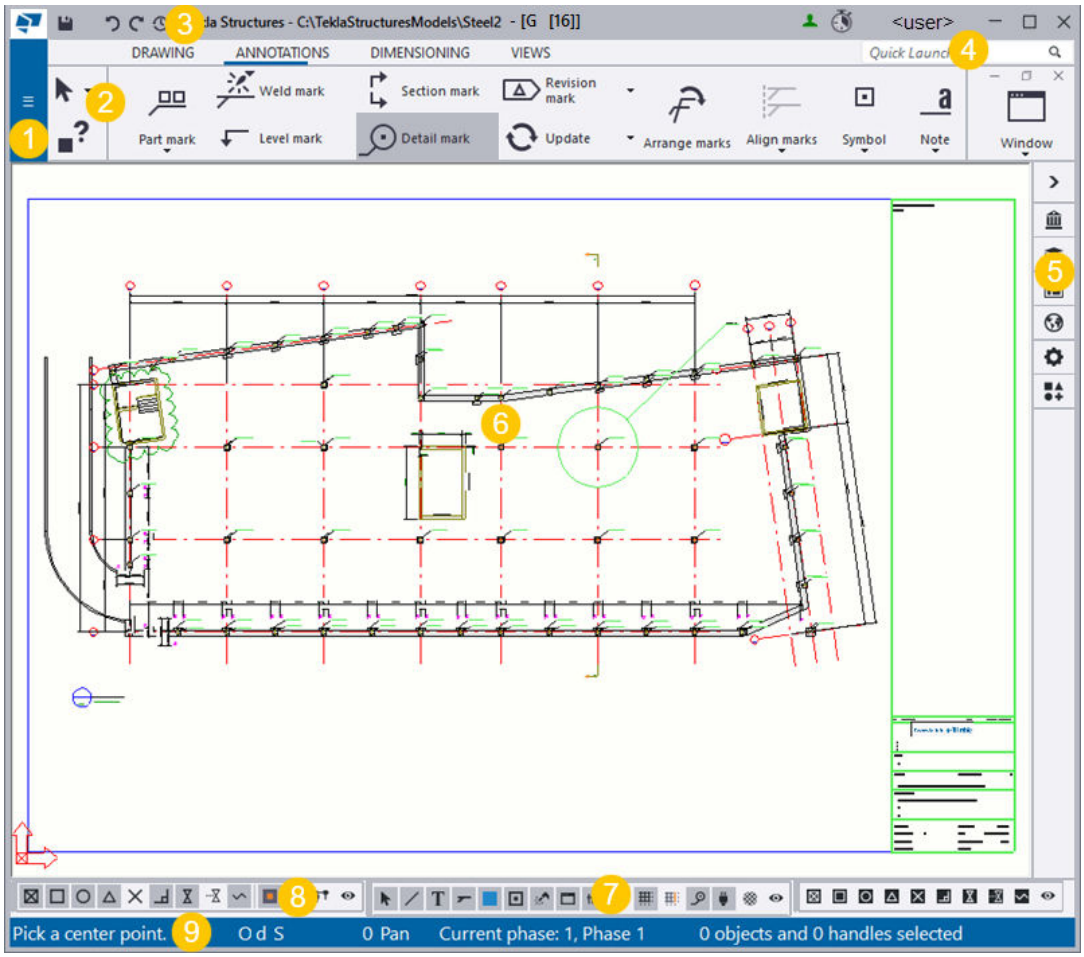
When you open a drawing, Tekla Structures activates the drawing mode. In the drawing mode:

- The ribbon, **File** menu, side pane, selection switches, and snap switches for the modeling mode are replaced by those of the drawing mode. Model views remain on the screen in the background. When you close the drawing (page 170), you automatically return to the modeling mode.
- The contextual toolbar and context menu contain commands and controls that are specific to drawings and drawing objects, and they vary depending on the drawing content that you have selected.



- **Quick Launch** contains drawing-specific commands, and you can use it to find commands, dialog boxes, and toolbars. Start typing the name of the command, for example, `dimension`, and start the command by clicking the name of the dimensioning command in the displayed list.

The following image identifies the locations of the controls in the a drawings window:



(1) The **File** menu contains commands and basic settings that can be used in the drawing mode.

(2) The ribbon and the tabs mostly contain commands available for drawings only, some commands are common with the modeling mode. When you click a command on the ribbon, the related help opens in the **Instructor** side pane.

(3) By default, the Quick Access Toolbar contains the **Save**, **Undo**, **Redo**, and **Undo history** shortcuts icons. You can customize the **Quick Access Toolbar** according to your needs.

(4) Use the Quick Launch for searching drawing-related commands.

(5) Use the side pane on the right side of the screen to check instructions for the currently active ribbon command, view and modify object properties, open **2D Library**, open **Drawing content manager**, or to find direct access to Tekla Online services.

(6) This is your Tekla Structures drawing.

(7) Use the selection switches to scale down the amount of selectable objects. Click the **Selection filter** button to create new selection filters.

(8) Use the Snap switches (page 53) to control which points you can snap to and pick.

(9) The status bar shows you instructions and error messages.

2.2 Snapping in drawings

In drawings, you can snap in positions in the same way as in the model. You can also snap to orthogonal angles while placing drawing objects or sketching. Zoom level affects free snapping in the way that the closer you zoom, the more exact you can snap. You can also place a drawing sketch object at a specified distance in the indicated direction.

Note that you cannot snap to blank lines.

Drawing snap switches and snap settings

Drawing snap switches:

Use the **Drawing snap settings** dialog box (**File** --> **Settings** --> **Snap settings**) to view and modify the snap settings in the drawing. These settings are user specific.

Snap to orthogonal points in a drawing

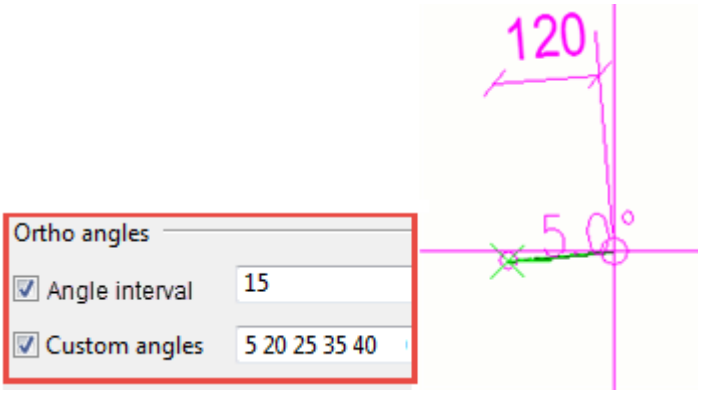
Use the **Ortho** tool to snap to the closest orthogonal point in a drawing. You can specify the orthogonal angle in **Drawing snap settings** (**File** --> **Settings** --> **Snap settings**). Using ortho snapping is useful if you need to place associative notes in a consistent manner, or sketch a polygon using a certain angle, for example. You can use predefined ortho angles and specify custom angles.

1. Open a drawing.
2. To activate orthogonal snapping, on the **File** menu, click **Settings** and select the **Ortho** check box.

   By default, the keyboard shortcut is **O**.
3. On the **File** menu, click **Settings** --> **Snap settings** .

   Note that you have the same snap settings available in the model, but these settings have no effect on drawings.
4. Set the snapping angles using one or both of the following ways:

- **Angle interval**: Select the check box next to **Angle interval** and then select a predefined angle: 10, 15, 30, 45, 90.
- **Custom angles**: Select the check box next to **Custom angles** and define the custom angles you want to snap to, for example, 12.5 or 17.5.



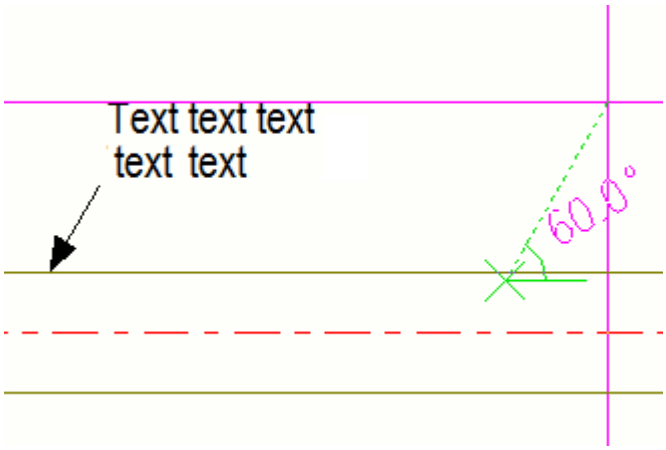
5. You can save different snap settings by entering a unique name in the **Save as** box and clicking **Save as**.

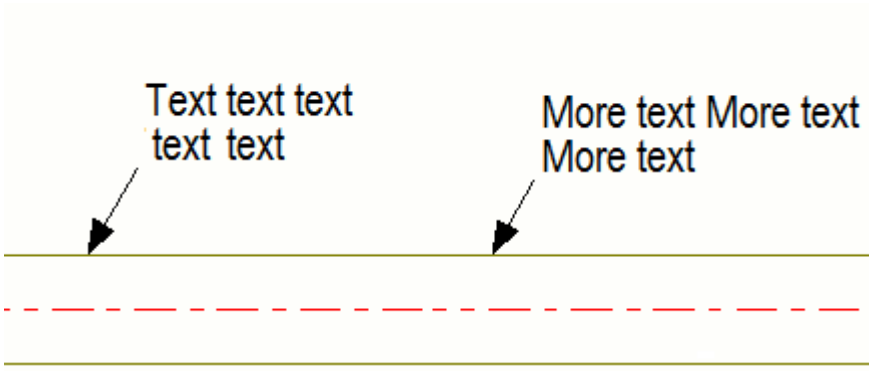
   The settings are saved in the `\attributes` folder under the model folder.

6. Click to save and apply the new settings.

In the example below, you first add a text with a leader line using a 60 degree angle to the part:

Then you will add a new text using the same angle:





Free snapping

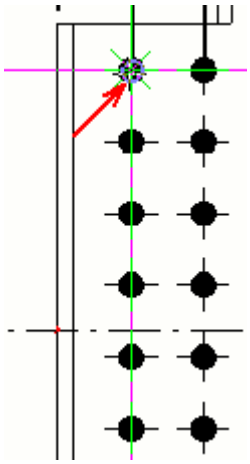
Free snapping in drawings is based on the zooming level in drawings: The closer you zoom, the more exact you can sketch. For example, you can more easily create rectangles that are of exact length when you zoom closer. The snapping step changes from 1 to 1000 (1/16" - 5') depending on the zoom level. You can follow the dimensions while you sketch.

Place a sketch object at a specified distance

You can snap to a specified distance in the indicated direction and place a sketch object in that position. You can specify the distance coordinate in the **Enter a Numeric Location** dialog box. In the following, you will ad a line.

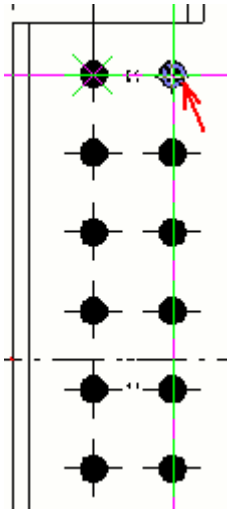
1. On the **Drawing** tab, click **Line** to activate the line tool.

2. Hold down **Ctrl** and pick an origin.



3. Point to the direction where you want to place the start point of the line.

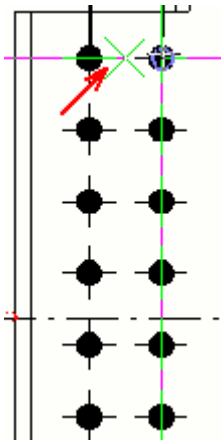
   Here, the bolt group needs to be moved 30 mm to the right and the line will indicate the new position for the group.



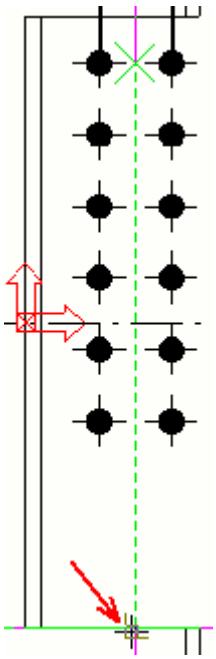
4. Start entering the distance, for example `30`.

   The **Enter a Numeric Location** dialog box is displayed.

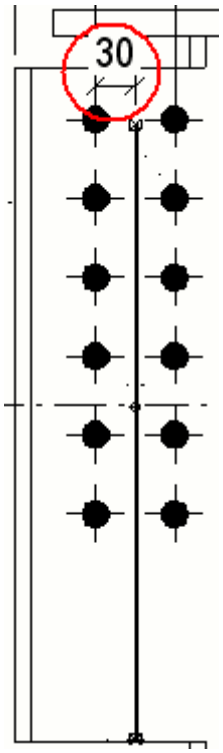
5. When you have entered the distance, click **OK**. Tekla Structures indicates the start point of the line.



6. Pick an end point for the line.



7. To check that the distance is correct, create a dimension.



2.3 Zooming and panning in drawings

The zooming commands in the context menu in drawings allow you to focus on a particular area, or pull out for a wider view. You can also use the mouse, keyboard shortcuts or **Quick Launch**. You can also pan (move) the drawings.

TIP Try the advanced option XS_USE_SMART_PAN: To activate optimized zooming and panning in drawings, set this advanced option to `TRUE`. When the smart pan is enabled while you pan or zoom a heavy drawing with lots of graphics, the whole screen is not drawn, empty space is left on the edges, and zooming and panning is faster. The default setting is `FALSE`, because using the optimization sometimes results in an unwanted checkerboard effect.

NOTE The point at which the level of detail is reduced in drawings was set earlier starting from the Tekla Structures version 2020. This means that zooming out in drawings drops details earlier than before. As a result, overall zooming and panning is smoother, while some drawing content is simplified and some is left out when zoomed out to certain extent. The effect is easy to notice in large drawings with a high amount of content.

The change in the level of detail affects:

- Hidden lines, which are not drawn
- Very short lines, which are not drawn
- Hatches, which are drawn as a cloud of points
- Text, which is replaced by a line

Zoom in and out

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

To	Do this
Zoom in	Do one of the following: • Scroll forward with the mouse wheel. • Press **Page Up** on the keyboard. • Right-click the drawing and select **Zoom** --> **Zoom in**. Then click the location in the view that you want to zoom in. • Go to **Quick Launch**, enter **Zoom in** and select the command from the list.
Zoom out	Do one of the following: • Scroll backwards with the mouse wheel. • Press **Page Down** on the keyboard. • Right-click the drawing and select **Zoom** --> **Zoom out**. Then click the location in the view that you want to zoom out. • Go to **Quick Launch**, enter**Zoom out** and select the command from the list.
Zoom to selected objects	1. Select the objects in the drawing. To select multiple objects, hold down **Ctrl**. 2. Right-click and select **Zoom** --> **Zoom selected**.

To	Do this
	Shift+Space is the default shortcut for the **Zoom selected** command. You can also go to **Quick Launch**, type **Zoom selected** and select the command from the list. **Zoom selected** works for any drawing objects that you select inside and outside the drawing views, and also for selected views. Note that **Zoom selected** also works for templates.
Restore the original zoom level	Do one of the following: • Right-click the drawing and select **Zoom** --> **Zoom original**. • Press **Home** on the keyboard. • Go to **Quick Launch**, enter **Zoom original** and select the command from the list.
Return to the previous zoom level	Do one of the following: • Right-click the drawing and select **Zoom** --> **Zoom previous**. • Press **End** on the keyboard. • Go to **Quick Launch**, enter **Zoom previous** and select the command from the list.
Create a zoom window	1. Right-click the drawing and select **Zoom** --> **Create zoom window**. 2. Click a starting corner for the zoom window, and then drag the pointer to size the window. You can also go to **Quick Launch**, enter **Create zoom window** and select the command from the list.

Pan in drawings

To	Do this
Activate panning	Do one of the following: • Press **P** on the keyboard.

To	Do this
	• Go to **Quick Launch**, enter **Pan** and select the command from the list. In an open drawing, the **Pan** command moves the whole drawing. The pointer will change to a hand. Click and drag the mouse to move the drawing anywhere within the window. • To deactivate panning, press **Esc**.
Activate middle mouse button pan	Do one of the following: • Select **File** --> **Settings** --> **Middle button pan**. • Press **Shift+M** on the keyboard. • Go to **Quick Launch**, enter **Toggle middle button pan** and select the command from the list. When middle mouse button panning is active, the text **Pan** is shown at the bottom of the Tekla Structures window. Hold down the middle mouse button and drag the drawing anywhere within the window. • To deactivate middle mouse button panning, clear the **Middle button pan** check box or press **Shift+M** again.

2.4 Colors in drawings

You have three basic color modes for line colors in drawings: **Black and white**, **Grayscale**, and **Color**.

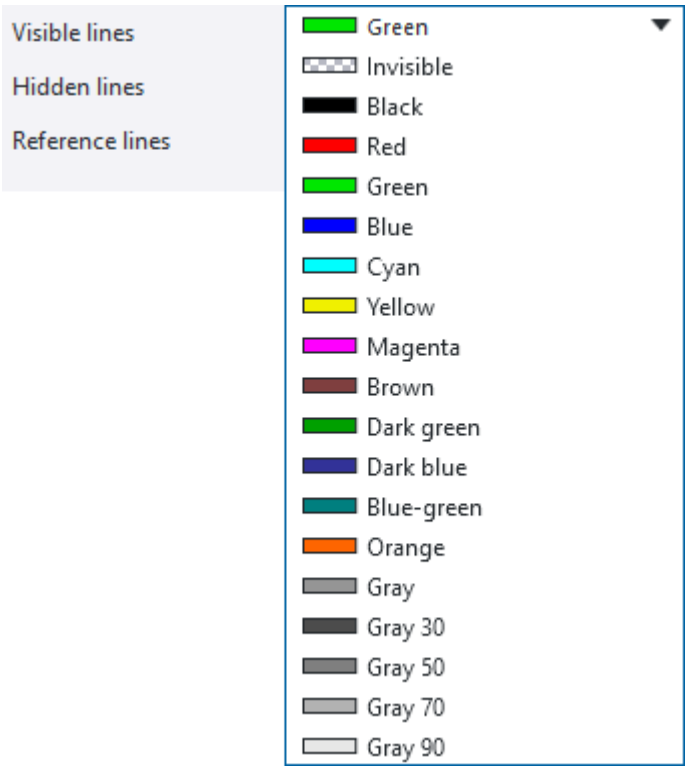
Colors available in drawings

The table below shows the basic colors available in Tekla Structures drawings, and how the different colors are shown in black and white drawings and in grayscale drawings. The corresponding pen numbers referring to the line width used in the printed drawings are listed in the **Pen** column.

Some of the different gray shades are true colors in the way that they will keep their color regardless of the drawing color mode, all the way to printing.

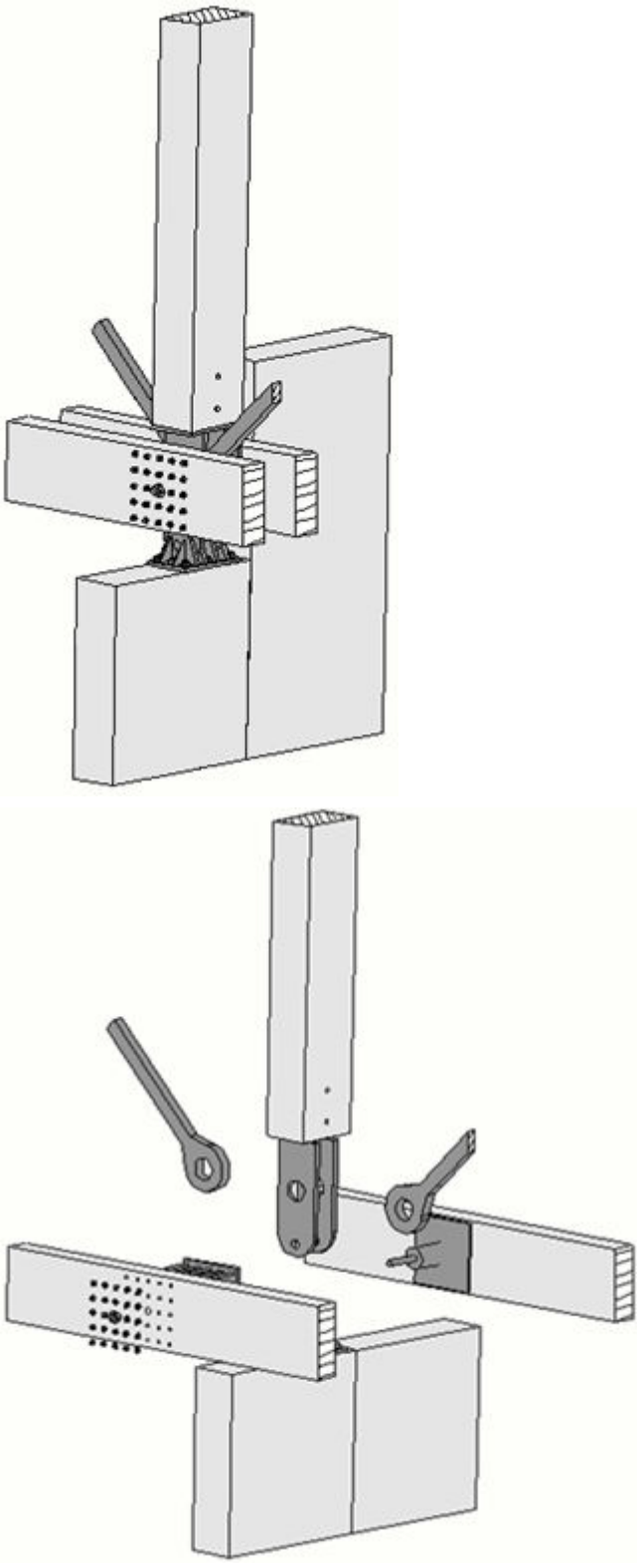
Name	Pen	Tekla Structures color	Black and white	Gray scale	Lightness
Invisible	9				Invisible
Black	7				0%
Red	1				0%
Green	3				0%
Blue	5				0%
Cyan	4				0%
Yellow	2				0%
Magenta	6				0%
Brown	15				30%
Dark green	110				50%
Dark blue	141				70%
Blue-green	111				90%
Orange	31				100% white
Gray	8				60%
Gray 30	251				30%
Gray 50	252				50%
Gray 70	253				70%
Gray 90	254				90%
Special	-				-

You can select the color for a line in the drawing object properties. In the property pane, the color names are also indicated.


Visible lines
Hidden lines
Reference lines
Green
Invisible
Black
Red
Green
Blue
Cyan
Yellow
Magenta
Brown
Dark green
Dark blue
Blue-green
Orange
Gray
Gray 30
Gray 50
Gray 70
Gray 90

Colors in drawing hatches (fills)

You can select the color for a hatch (fill) in part and shape properties on the **Fill** tab after you have selected a fill type from the **Type** list.



The true gray shades are marked with red in the image below.

The gray scale colors (130 - 133) are also available for automatic hatches.

Color modes in drawings

You have three basic color modes for line colors in drawings: **Black and white**, **Grayscale**, and **Color**.

Color is the default color mode. To change the default color mode in drawings when Tekla Structures is started, use the XS_USE_COLOR_DRAWINGS advanced option.

In addition to the three basic color modes, you can have a black background and colored lines in drawings (advanced option XS_BLACK_DRAWING_BACKGROUND).

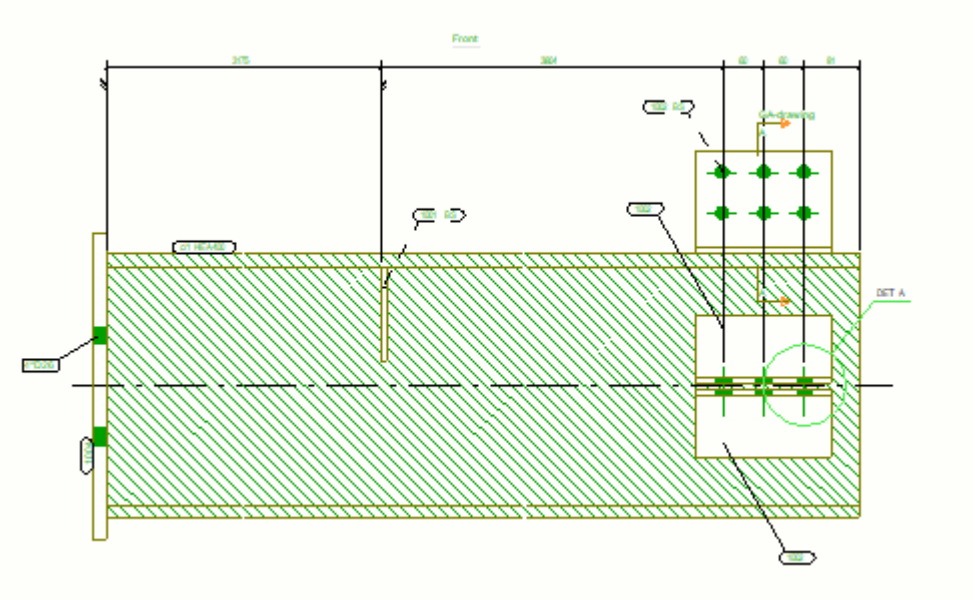
In addition to the actual colors, you can specify a special color that is not converted to black in printouts. For more information, see "Specify a special color for printing" later in this article.

Note that the drawing color mode does not affect the colors in the printouts, you can select the printing color on the **Options** tab in the **Print Drawings** dialog box. You can change the line width of the colors (page 603) on the **Line properties** tab in the **Print Drawings** dialog box. This setting affects the line width in printouts, and also the line width on the screen in black and white color mode. To also show the defined line width on the screen in color and grayscale modes, go to **File** --> **Settings**, and select the **Printer line widths** check box. In the printing preview, the actual line widths are shown in all modes.

Color

The **Color** mode does not show different line widths by default. To show the line widths defined in the **Print Drawings** dialog box on the screen, go to **File** --> **Settings**, and select **Printer line widths**.

Below is an example of a drawing in the **Color** mode.

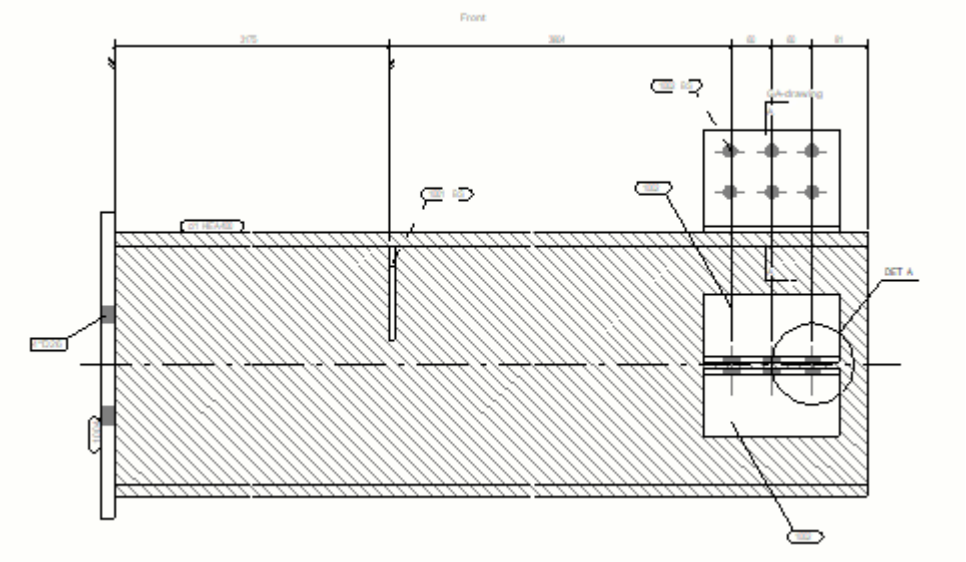


Grayscale

In the **Grayscale** mode, the colors from 1 to 7 (black, red, green, blue, cyan, yellow, magenta) are shown in black, and the colors from 8 to 14 (brown, dark green, dark blue, blue-green, orange, gray) are shown in different shades of gray.

The **Grayscale** mode does not show different line widths by default. To show the line widths defined in the **Print Drawings** dialog box on the screen, go to **File** --> **Settings**, and select **Printer line widths**.

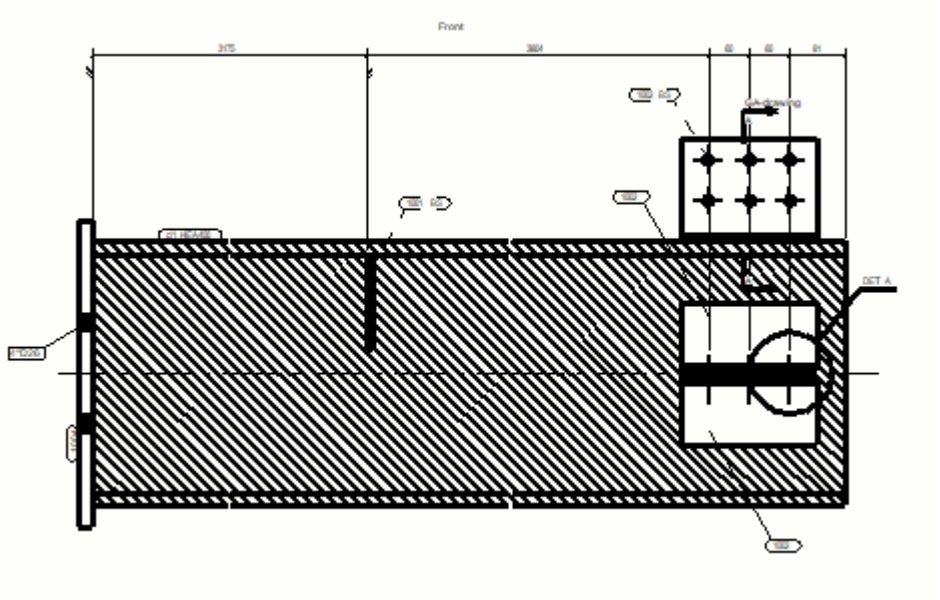
Below is an example of a drawing in the **Grayscale** mode.



Black and white

The **Black and white** mode shows different line widths by default.

Below is an example of a drawing in the **Black and white** mode.



Change drawing color mode

You can change the color mode of the drawings.

1. Open a drawing.
2. On the **File** menu, click **Settings** and, under **Color mode**, select **Black and white**, **Grayscale** or **Color**.

   You can toggle between the color modes in a drawing by pressing **B** on the keyboard.

Specify a special color for printing

You can define a special color that is not converted to black when printed. This color will be printed as color or grayscale, depending on the selected printer settings. The special color is defined using RGB (Red Green Blue) values in a scale of 0 to 255. The special color is applied on parts or sketch objects as a hatch.

1. On the **File menu**, click **Settings** --> **Advanced options** and go to the **Hatching** category.
2. Define the color using the following advanced options:
   - XS_HATCH_SPECIAL_COLOR_R
   - XS_HATCH_SPECIAL_COLOR_G

- XS_HATCH_SPECIAL_COLOR_B

  The default value for all of the above advanced options is 230. The smaller the value is, the darker the shade.

3. Click **OK** or **Apply**.
4. Open a drawing.
5. Double-click a drawing object to open object properties. For example, click a building object or a rectangle.
6. Select a fill type.
7. Select the **Special** fill color.
8. Click **Modify**.

The object that you selected now uses the special color that you specified.

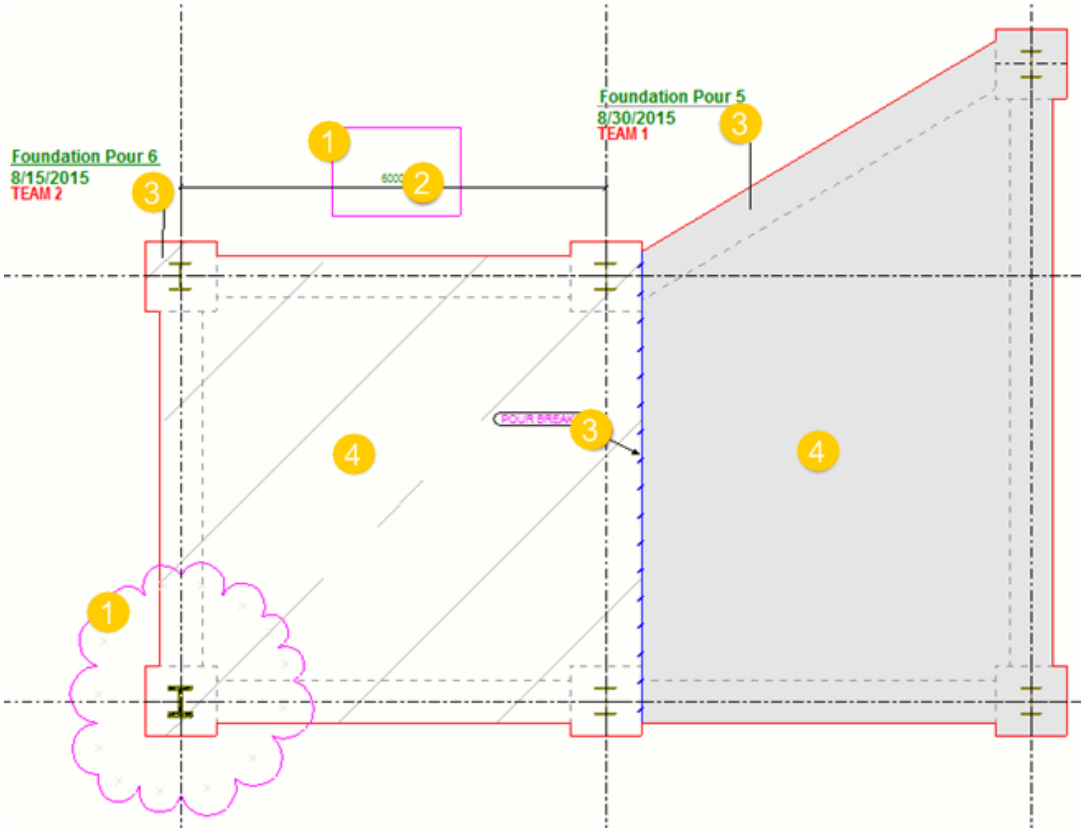
2.5 Drawing objects, views and layouts

Drawing objects are organized inside *drawing views*, which are placed in the selected *drawing layout* according to the selected settings.

Drawing objects

Drawings may contain several types of objects. Some of them come from the model and represent something that will exist in the real building or will be closely related to it. Others are objects that represent information that is only relevant in the drawing, or add extra information to the information in the model. Drawings may include the following object types:

- Building objects (page 426): parts, bolts, welds, chamfers, reinforcing bars, surface treatment, reference objects, etc. Building objects are 2D representations of the 3D objects in the model.
- Associative annotation objects (page 289): dimensions, marks, associative notes.
- Independent annotation objects (page 289): objects that are not linked to the model: text objects, `.rtf` files, symbols, links, hyperlinks, and DWG/DXF files. These objects become associative if they have associativity points, i.e. they are associated to building objects.
- Sketch objects (page 387): graphical objects you create using the various sketching tools. These objects can be used for highlighting parts of the drawing (clouds, lines, rectangles, etc.), for example. These object are associative if they have associativity points, i.e. they are associated to building objects.



(1) Sketch objects: clouds and rectangles

(2) Dimensions

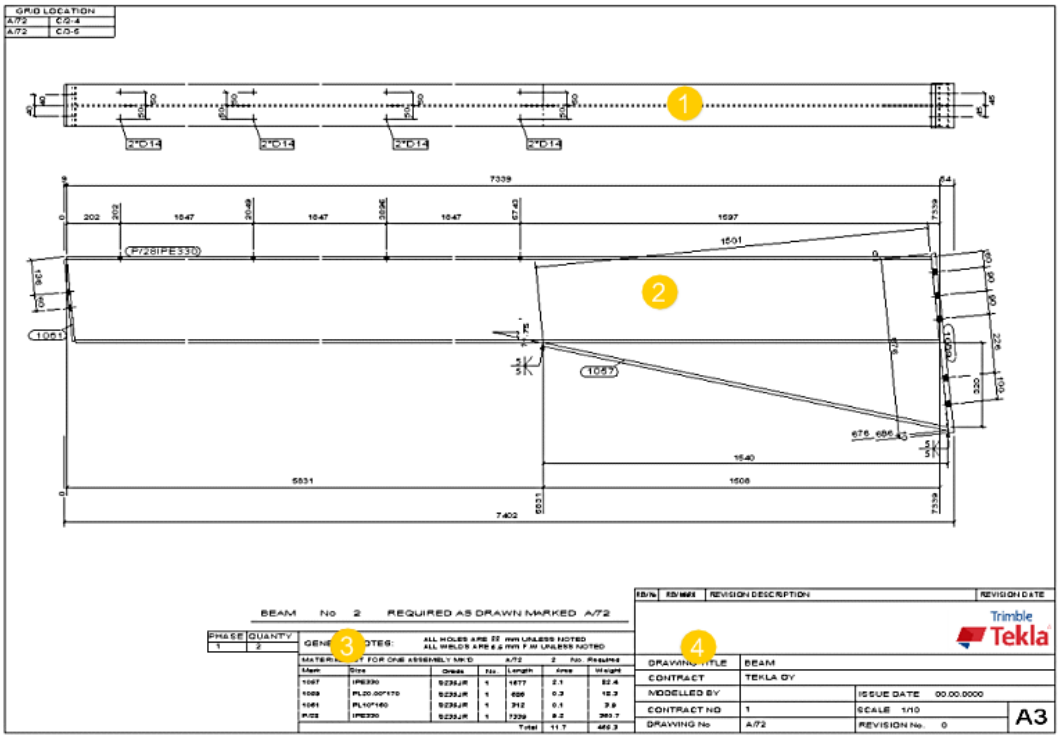
(3) Marks, associative notes

(4) Building objects

Drawing views and layouts

- Drawing views (page 179) are views to the whole model, to a part of the model, or to individual parts in the model. Views can show building objects from different directions (top, front, back, bottom) and cross sections. Drawing views act as containers for the building objects or areas in the model that you have selected to be included in the drawing.
- A drawing layout (page 655) defines the drawing size, drawing view margins, spaces between drawing views, a table set including tables, such as revision tables, title blocks, material lists, key plans, DWG references, or bills of materials, and also drawing frames and foldmarks. The data in the tables is derived from the model data. Tekla Structures has a range of predefined layouts, and you can also create your own layouts. Different tables can be included in your drawing according to the selected drawing layout and drawing size.

Below is an example of the drawing layout and views in a workshop drawing.



(1) Top view to a part. Here you have selected to include the top view (page 689) in the **View creation** pane.

(2) Front view to a part. Here you have selected to include the front view (page 689) in the **View creation** pane.

(3) Material list. This is defined in your drawing layout (page 656).

(4) Drawing title block. This is defined in your drawing layout (page 656).

2.6 How to keep your drawings up to date?

When a model changes, the related drawings need to be updated. Tekla Structures takes care of updating and notifies you if updating is needed.

WARNING When you start using a newer version of Tekla Structures, updating drawings created with the older version may cause problems. We recommend that you complete any drawings you have started using the older version, or recreate the drawings using your new version of Tekla Structures.

You need to update drawings when:

- The geometry of a model object changes.
- Some other model object properties, for example, material and class change.

- Model objects are added or deleted.
- The number of identical model objects changes.

Tekla Structures automatically updates drawings each time you number the model. If you have not numbered the model, you are prompted to do so when you create a drawing. Furthermore, if you have changed the model and go to the **Document manager** to open drawings, Tekla Structures has marked the outdated drawings, and you need to update them before you can open them.

General arrangement drawings are always updated when you open them if the model has changed. You do not need to number the model to update general arrangement drawings.

Click the following links to find out more about drawing update, when recreation is needed, and how the drawings are numbered:

Update drawings when the model changes (page 575)

Recreation of drawings (page 88)

2.7 Different levels of setting up and modifying drawing properties

In Tekla Structures, you can set up and change drawings and drawing properties on different levels, depending on how permanent and extensive changes are needed, and which drawing type you are working with. You can change drawings on the drawing, view, and object level.

Single-part, assembly, and cast unit drawings

- On the highest level, you can define drawing properties on the *drawing level*:
  - The drawing-specific properties that you define in the **Drawing properties** apply to the whole drawing: user-defined attributes for the drawing, the detail view start number/letter, drawing-specific section view settings, some drawing-specific view attributes, drawing titles, and drawing layout settings.
  - On the drawing level, you can also select the views that you want to create, and set view, dimensioning, protection, building object, and mark settings **separately for each view** by going further to **View properties** for the selected view. For example, you can define that all marks have blue frame in the top view, or that model weld marks are shown in the front view. It is very important to save the view-level properties files in the **View properties** to be able to connect the desired view properties to the views that you are creating.

    - You can modify the drawing level properties before you create the drawing and also change them in the existing drawing.
- You can also change drawing properties in single-part, assembly and cast unit drawings on the *view level*:
    - Double-click a view frame in an open drawing, which opens the **View properties**. You can also modify several views by holding down Ctrl and then double-clicking the view frame of one of the selected views.
    - Change a property, for example, a mark property. Select the properties that you want to change from the options tree on the left. The change takes place only in the views that you have selected in the open drawing. For example, if you change the mark line color, the mark line changes in all marks in the selected views.
- Finally, you can change drawing properties on the *object level*:
    - Click an object in an open drawing, which opens the object-specific properties. For some objects, you need to double-click the object. You also need to double-click the object if the property pane is not open.
    - Properties change for the selected object only. You can change select several objects and change the properties in all of them.
    - The properties changed on the object level are not affected by any property changes on higher levels.

General arrangement drawings

- On the highest level, you can change general arrangement drawing properties on *drawing level* **Drawing properties** and its subdialog boxes:
    - This way you can change the properties of all building objects, protected areas, marks, dimensions, and views in the drawing at the same time. For example, you can define that all marks have blue frame.
    - You can modify the drawing properties before you create the drawing and also change them in the created drawing.
    - The object properties change in all views and objects in that drawing, except for new views that you create after the drawing is created.
- You can also change general arrangement drawing properties on *view level*:
    - Double-click a view frame in an open drawing, which opens **View properties**.
    - The changes take place only in the views that you have selected in an open drawing.
    - The object properties change in all objects of the particular type in the selected views. For example, if you change the mark line color, the mark line changes in all marks in the selected views.
    - If you change a property on the view level, do not change the same setting on the drawing level, because drawing-level changes override view-level changes.

- Finally, you can change drawing properties on *object level*:
  - Double-click an object in an open drawing, which opens object-specific properties.
  - Properties change for the selected object only.
  - The properties changed on the object level are no longer affected by any property changes on higher levels.

Detailed object level properties

By saving the object property settings and combining them with drawing or view filters and drawing object types into *detailed object level settings*, you can also apply the object level properties on the drawing and view level. Note that detailed object levels properties on drawing level are available only in general arrangement drawings. Object level settings are a powerful tool: you can use the same property file for creating drawings and quickly change a particular property before you create drawings, for example, reinforcement color or mark frame shape. The object level settings override the property settings in the view and drawing properties. Changes in the object level settings applied on the drawing level are inherited to the view level if there are no object level settings defined on the view level. If you apply object level settings on the view level, they override the drawing level settings.

Recommended workflow

The recommended way of working is to work from top to bottom, from drawing level to object level:

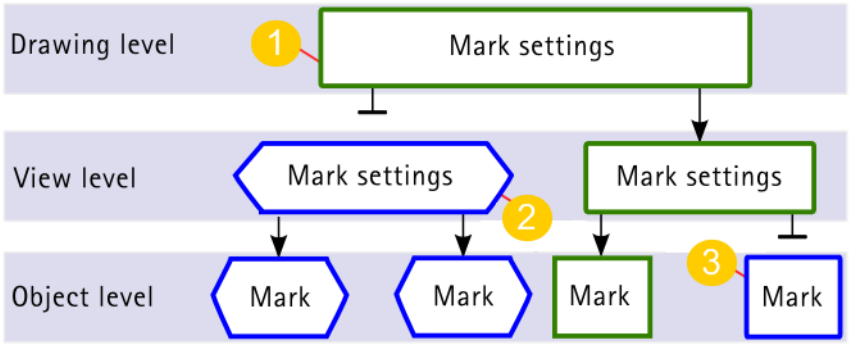
1. Set your drawing and automatic drawing view properties as close as possible to what you want by first attempting this on the drawing level.
2. Then modify anything else that needs to be changed on the view level.
3. Finally, if any further cleanup or adjusting is necessary, you can make changes on the individual object level.

The modifications made on the drawing level remain when the drawing is recreated due to a model change.

Once you change properties on the view level you should not go to the drawing level properties and modify drawing properties there. This is because once you have changed a setting on the view level, changing the same setting on the drawing level may negate the changes you have just made. For example, if you change the mark settings on the view level in a GA drawing, and use differents settings for different views, and then change the mark settings on the drawing level, the drawing-level changes override the view-level changes in all views.

Example workflow in a general arrangement drawing

The following image illustrates the idea of the three property levels in a GA drawing. The frame color and the shape of the mark are used as an example.



1. When you change the mark frame color and shape for the whole drawing on the drawing level, the changes propagate to the view level and to the object level.
2. When you change the mark frame color and shape in the selected views, the changes take place in the selected views only. The properties do not change in the whole drawing.

   Note that if you change the frame color and shape on the drawing level after changing them on the view level for some of the views, the drawing-level changes override the view-level changes in all views. Note that the view attributes and label settings behave differently: they are not overridden, for example, the view scale stays as you have set it for individual views.
3. When you change the mark frame color and shape in the selected marks, the properties do not change anywhere else, only in the selected marks. If you try to change the mark frame color and shape on the view or drawing level, the properties do not change in the marks that you changed separately on the object level.

Click the following links to find out more:

Set automatic drawing properties before creating drawings (page 74)

Modify drawing properties of an existing drawing (page 77)

Modify drawing view-level properties (page 76)

Modify drawing object properties (page 78)

Detailed object level settings (page 79)

How Tekla Structures applies drawing properties in drawing creation (page 87)

Recreation of drawings (page 88)

Set automatic drawing properties before creating drawings

Tekla Structures creates drawings using the automatic drawing properties defined separately for each type of drawing.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. If possible, load drawing properties closest to the ones you need by selecting a properties file from the list at the top.

   In general arrangement drawing properties, you need to click **Load**.
3. Do one the following, depending on the type of the drawing:

   Single-part, assembly and cast unit drawings:

   a. Go through the various properties in the drawing properties options tree and adjust the values as requires.

      The check boxes next to the options are automatically selected when you change something for that particular option.

   b. To save properties for future use, give a unique name for the properties in the **Save** box and click **Save**. Note that you should not include spaces or special characters in the name.

   c. To adjust view-level settings (views, dimensions, filters, protected areas, marks and objects), click **View creation**, select the desired view and view properties file, and then click **View properties**.

   d. Adjust the properties for the selected view as required.

   e. To save properties, for example, part properties for future use, give a unique name for the properties in the **Save** box and click **Save**. Note that you should not include spaces or special characters in the name.

   f. Click **Save** in **View Properties** to save the properties in the view properties file, when you are done.

   g. Click **OK** in **View Properties** to return to the drawing properties.

   General arrangement drawings:

   a. Switch off the all the check boxes by clicking the on/off switch at the bottom and only select the check boxes for the options that you wish to change.

   b. Go through the various properties in the drawing properties dialog box, and make the necessary changes.

   c. Click **OK** in each subdialog box where you changed the properties to save the changes and to return to drawing properties.

   d. To save properties, for example, part properties, for future use, give a unique name for the properties in the **Save as** box and click **Save as**. Note that you should not include spaces or special characters in the name.
4. Click **OK** to save the properties.

If you want to save the changes in another drawing properties file, enter the new name.

Note that you should not use spaces in the properties file name, this may cause problems. For example, when you use the drawing properties file in a rule set in the **Master Drawing Catalog**, the drawings are not created if the properties file name contains spaces.

In general arrangement drawing properties, click **Save as**.

Now you can create a drawing using the drawing properties file that you just set up.

Modify drawing view-level properties

You can modify the view-level properties after you have created the drawing. Most of the properties that a single-part, assembly and cast unit drawing have are specified for each of the drawing views separately already before creating a drawing in the automatic drawing properties. General arrangement drawings do not have automatic view properties, and the view properties can only be modified in an open drawing.

The changes you make in view properties only apply to the view that you have selected to modify.

For a list of view properties and their values, see View properties in drawings (page 912).

1. Open the drawing.
2. To open the drawing view properties in the property pane, click the drawing view frame. If the property pane is not open, you need to double-click the frame. This also opens the view properties dialog box.
3. Go through the properties in the drawing view property pane, modify the properties and click **Modify** when you are done.
4. If you need to modify settings available in the view properties dialog box only, click the **View** button in the property pane.
5. Make the necessary changes. The check boxes in the options tree are off by default. When you change a property, for example, select another option for a setting, Tekla Structures adds a check mark to the check box next to the option in the option tree.
6. Click **Modify** when you are ready with the changes.

The view changes according to the changes that you made in the drawing view properties.

NOTE Some of the changes in you make require a drawing recreation. For more information, see Recreation of drawings (page 88).

See also

Modify drawing properties of an existing drawing (page 77)

Modify drawing properties of an existing drawing

If you are not satisfied with the drawing properties after creating the drawing and checking it, you can modify the automatic drawing properties in the created drawing.

1. Open the drawing.
2. Double-click the drawing background.
3. Do one the following, depending on the type of the drawing:

   Single-part, assembly and cast unit drawings:

   a. Go through the various options in the drawing property options tree and adjust the values as required.

      The check boxes next to the options are automatically selected when you change something for that particular option.

   b. To adjust view-level settings (views, dimensions, filters, protected areas, marks and objects), click **View creation**, select the desired view and view properties file and click **View properties**.

   c. Adjust the view properties as required.

   d. Click **Save** in **View Properties** to save the properties in the view properties file, when you are done.

   e. Click **OK** in **View Properties** to return to the drawing properties.

   General arrangement drawings:

   a. Switch all the check boxes off by clicking the on/off switch at the bottom of the dialog box and only select the check boxes for the options that you wish to change.

   b. Go through the various options in the drawing properties dialog box, and make the necessary changes.

   c. Click **OK** in each subdialog box where you changed the properties to save the changes and to return to drawing properties.

4. Click **Modify**.

   The drawing is changed according to the changes that you made in the drawing properties.

NOTE Some of the changes in you make require a drawing recreation. For more information, see Recreation of drawings (page 88).

TIP You can also modify the properties of several drawings by selecting them from **Document manager**, right-clicking and selecting **Properties**

See also

Modify drawing view-level properties (page 76)

Set automatic drawing properties before creating drawings (page 74)

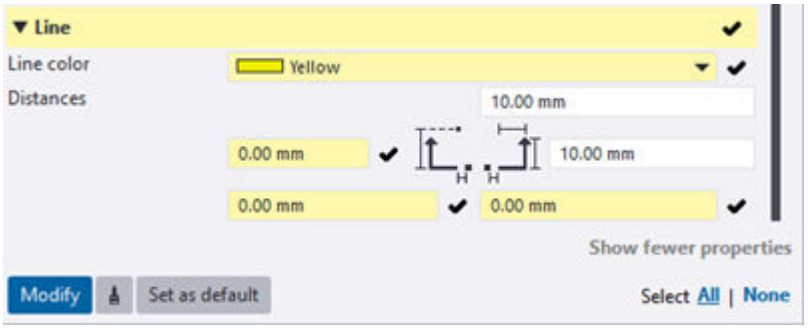
Modify drawing object properties

You can change the properties of the objects in the drawings such as parts, marks, bolts, and welds, and save the object properties files for later use.

1. Open a drawing.
2. Click the object that you want to modify.

   For example, click a part, bolt, or a reinforcing bar.

   If the drawing property pane is not open, you double-click the object.
3. Change the desired properties.



4. To apply the changes in the object, click **Modify**.

You can also give a new name for the property file and save the file. You should not use spaces or special characters in the property file names. You can load the properties in this file in other drawing objects, in this case, parts. You can also use the object property files with filters to apply detailed object level settings on object groups.

See also

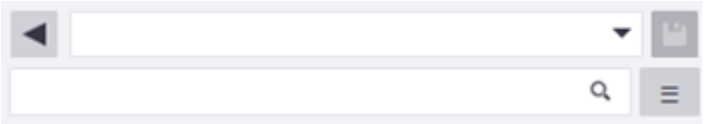
Load saved drawing object properties (page 78)

Load saved drawing object properties

You can change the object properties in an existing drawing, for example the part color, by loading saved object properties.

1. Open a drawing.
2. Click the drawing object you want to change. For example, click a part.

   If the drawing property pane is not open, double-click the object.
3. Select the desired object properties file from the list.



4. Click **Modify**.

Tekla Structures changes the drawing object according to the settings in the loaded object properties file.

See also

Modify drawing object properties (page 78)

Detailed object level settings

In addition to defining automatic drawing mark and object properties, you can also set up a special representation for marks and building objects in drawings and use these detailed object level settings for specific purposes. For example, you may want to define that all columns in a specific general arrangement drawing are shown in a specific color, and in all other general arrangement drawings in the default part color.

To create the detailed object level settings, you first need to define the part properties and filters. For example, in part properties, set the part color to blue and save the properties.

To create detailed objects settings, you need to have:

- A filter that selects the objects that you want to be affected
- Object properties that you want to apply on the objects

When you want to apply detailed object level setting on view level, create object properties on drawing view level in drawing view properties. When you want to apply detailed object level settings on drawing level, like in general arrangement drawings, create object properties on drawing level in drawing properties.

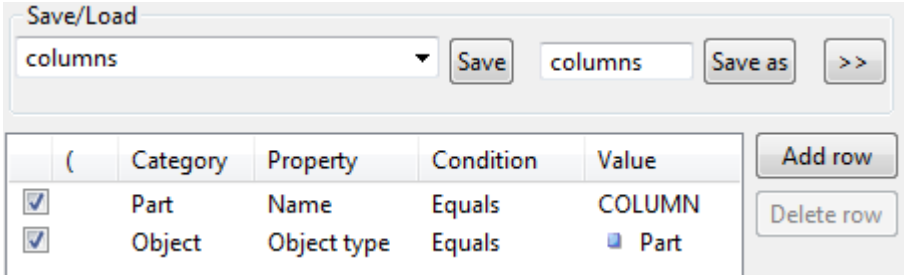
Create detailed object level settings in a general arrangement drawing

To create detailed object level settings on drawing level in general arrangement drawings:

1. On the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing** .

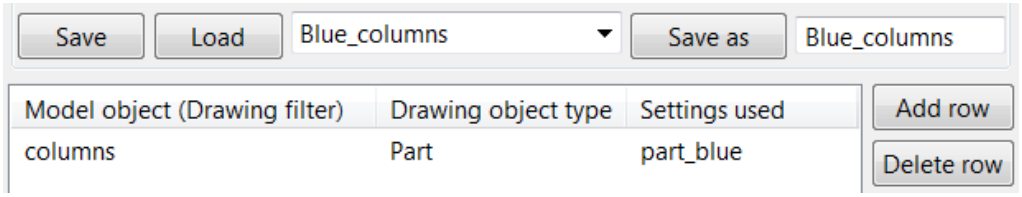
2. Click **Part** and create part properties that define the object appearance that you want.
3. Save the part properties by entering a unique name in the box next to the **Save as** button and click the button.
4. Click **Cancel** to close the dialog box.
5. In the drawing properties dialog box, click **Filter** and create a drawing filter that selects the objects that you want to have special representation.

   For example, create a filter that selects columns.



6. Save the filter by entering a name for it and click **Save as**.
7. Click **Cancel** to close the dialog box.
8. In the drawing properties dialog box, click **Edit settings...**.
9. In the **Object level settings for general arrangement drawing** dialog box, click **Add row** and select the filters to be used, the drawing object types and the drawing object properties to be used.

   By doing this, you are combining the filter, object type, and object property settings into detailed object level settings.



   You can add several rows in detailed object level settings.
10. Give the object level settings a name and save using **Save as**.

    Note that you should not include spaces or special characters in the name.
11. Click **Cancel** to close the dialog box.

Now you have new detailed object level settings to be applied on a drawing.

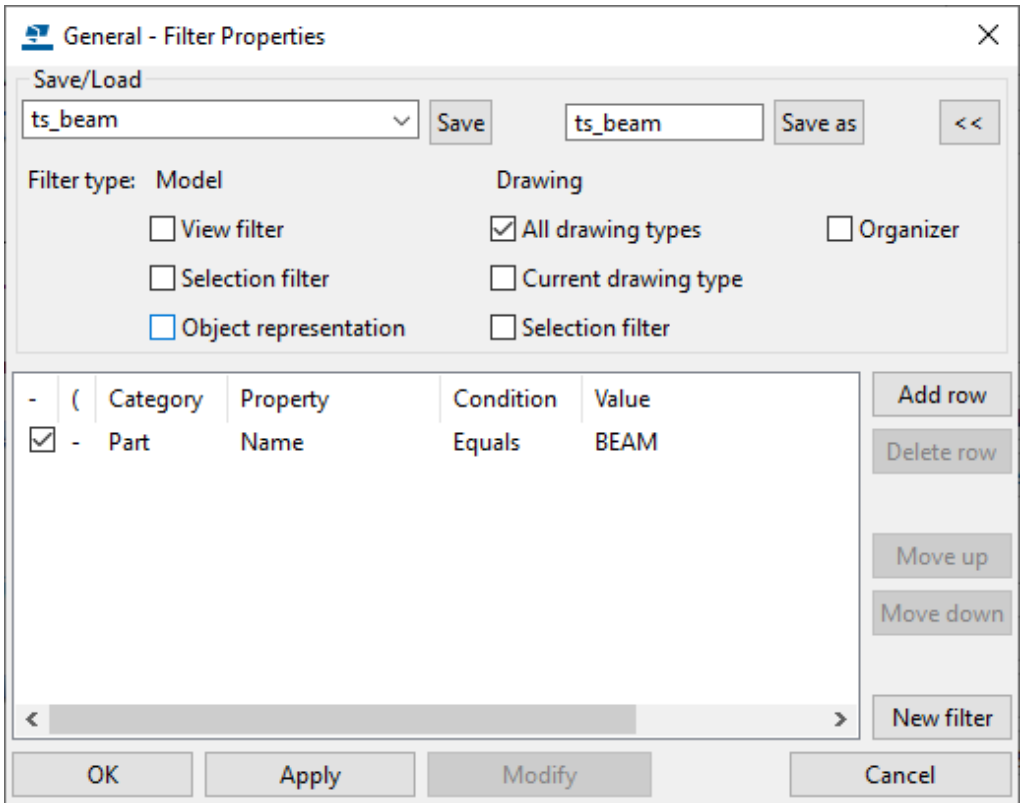
Example: Apply detailed object level settings on drawing level in a GA drawing

Before doing this, drawing object properties for beams, columns and braces have been created and saved with a unique name in the drawing part properties dialog box so that these parts have different colors. For details about creating drawing part properties, see the instructions above.

First you will make a general arrangement drawing filter, then save detailed object level settings in a settings file and finally, apply the new detailed object level settings on drawing level.

In this example, the filter uses part names defined in the model, similarly as in modeling filters.

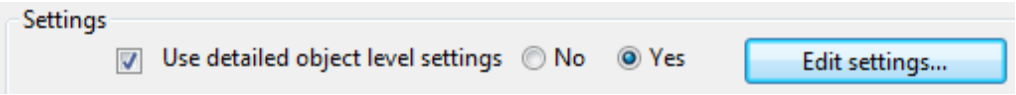
1. On the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing**
2. Click **Filter** in the drawing properties dialog box.
3. Create filters by **Part** - **Name** for column, beam, and brace and save the settings using **Save as** with different names, such as `ts_column`, `ts_beam` and `ts_brace`.



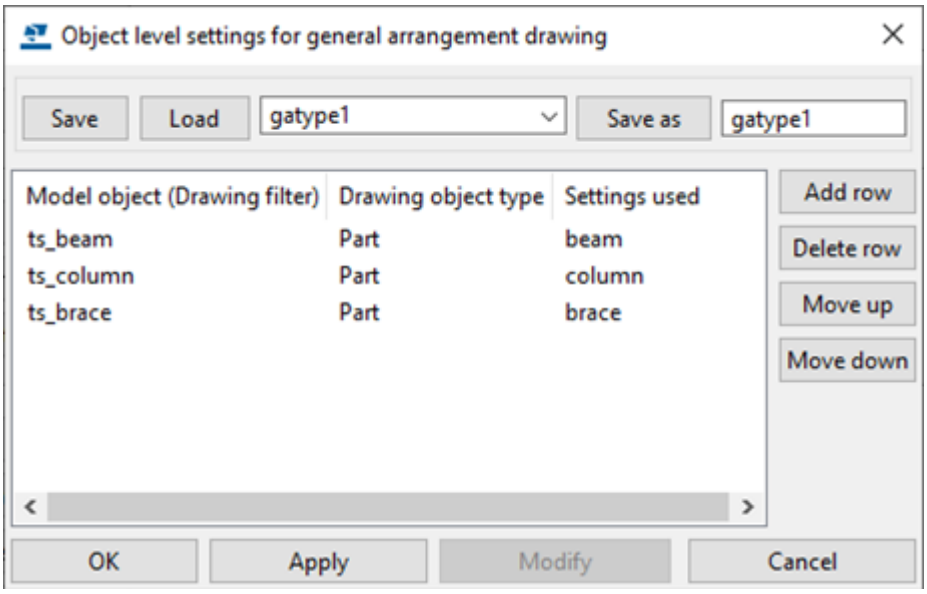
4. Click **Cancel** to close the dialog box.

   Now you have created the required filters needed for selecting the desired parts.

5. In the **General arrangement drawing properties** dialog box, click **Edit settings...**.

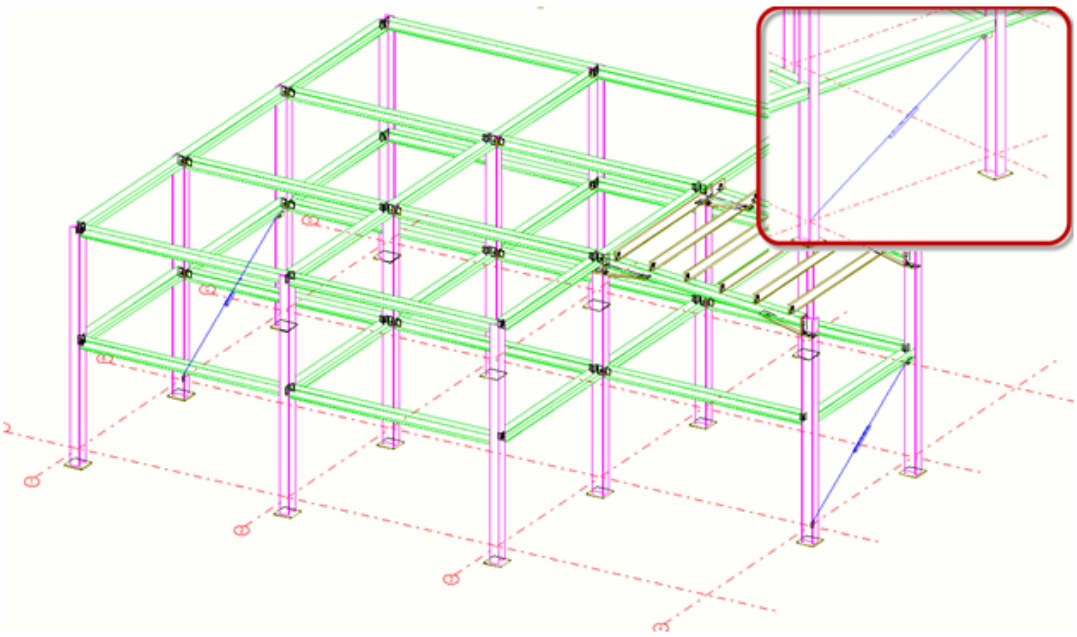


6. In the **Object level settings for drawing** dialog box, select the filters to be used, the drawing object types and the drawing object properties to be used:
   a. Click **Add row** and select `ts_column` as **Model object (Drawing view filter)**, **Part** as **Drawing object type**, and `column` as **Settings used**.
   b. Click **Add row** and select `ts_beam` as **Model object (Drawing view filter)**, **Part** as **Drawing object type**, and `beam` as **Settings used**.
   c. Click **Add row** and select `ts_brace` as **Model object (Drawing view filter)**, **Part** as **Drawing object type**, and `brace` as **Settings used**.
7. Save the detailed object level settings as `gatype1` using **Save as**.



   The same detailed object level settings can be used in other general arrangement properties, you do not need to create new ones.
8. Click **Cancel** to exit the dialog box.
9. Open a created general arrangement drawing and double-click the drawing background to open the drawing properties.
10. Click **Edit settings...**.
11. Load object level settings you created, in this example `gatype1`, and click **Load**.
12. Click **OK**.
13. Click **OK**.

14. Ensure that **Use detailed object level settings** is set to **Yes**.
15. Click **Modify** to apply the changes in the drawing.
16. Confirm the detailed object level changes by clicking **Yes** in the on **Object level settings changed.** message box.



NOTE If a row in the **Object level settings for view** dialog box is shown in red, either the object properties file defined for **Settings used**, or the filter file defined for **Model object (Drawing view filter)** is missing from the `\attributes` folder in the current model folder. See an example below:

Model object (Drawing filter)	Drawing object type	Settings used
test_column_filter	Part	test_properties

Create detailed object level settings in cast unit drawings

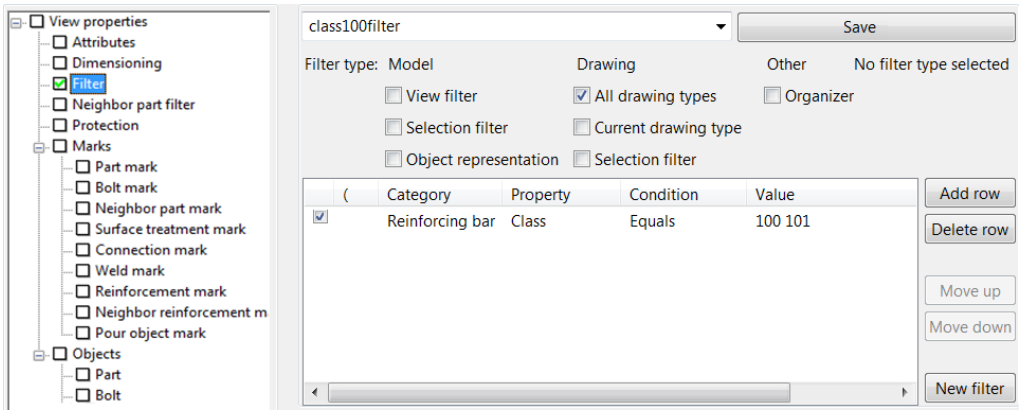
Before creating the detailed object level settings, create the part properties and needed filters in drawing view properties. For example, in reinforcement properties, set color to red and save the properties. You create detailed object level settings in the same way in single-part drawings and assembly drawings.

To create detailed object level settings on view level:

1. On the **Drawings & reports** tab, click **Drawing properties** --> **Cast unit drawing** .
2. Click **View creation**, and click the**View properties** button.
3. Click **Part** in the options tree and create part properties that define the object appearance that you want.

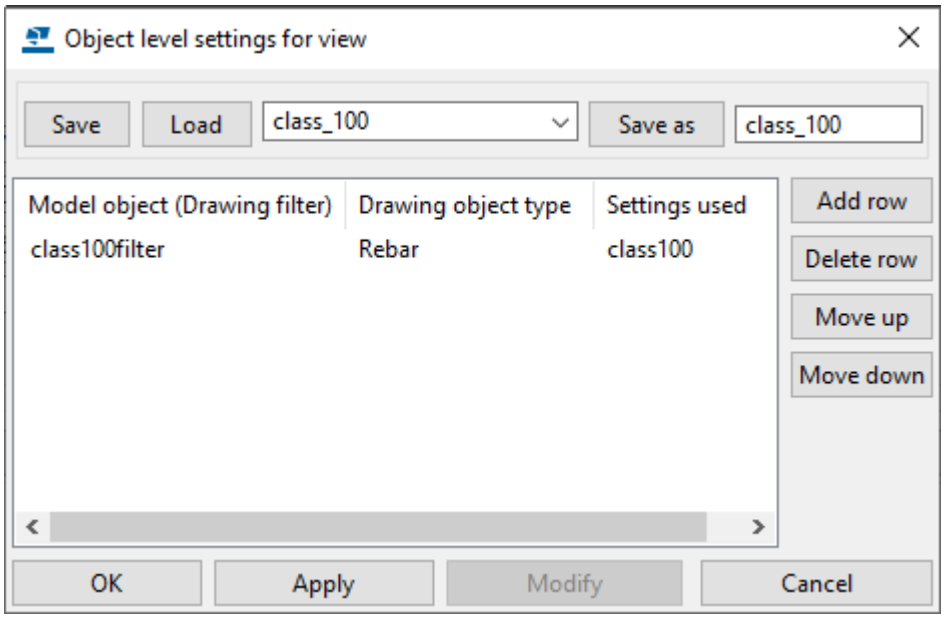
4. Save the part properties by entering a unique name in the box next to the **Save** button and click the button.
5. Click **Filter** in the options tree and create a view filter that selects the objects that you want to have special representation.

   For example, create a filter that selects reinforcing bars in class 100 and 101.



6. Save the filter by entering a name in the box at the top and clicking **Save**.
7. Click **View properties** in the options tree and click **Edit settings...**.
8. In the **Object level settings for view** dialog box, click **Add row** and select the filters to be used, the drawing object types and the drawing object properties to be used.

   By doing this, you are combining the filter, object type, and object property settings into detailed object level settings.



   You can add several rows.

9. Give the detailed object level settings a name (here `class_100`) and save using **Save as**.

   Note that you should not include spaces or special characters in the name.

10. Click **Cancel** to close the dialog box.

Now you have new detailed object level settings to be applied on a drawing.

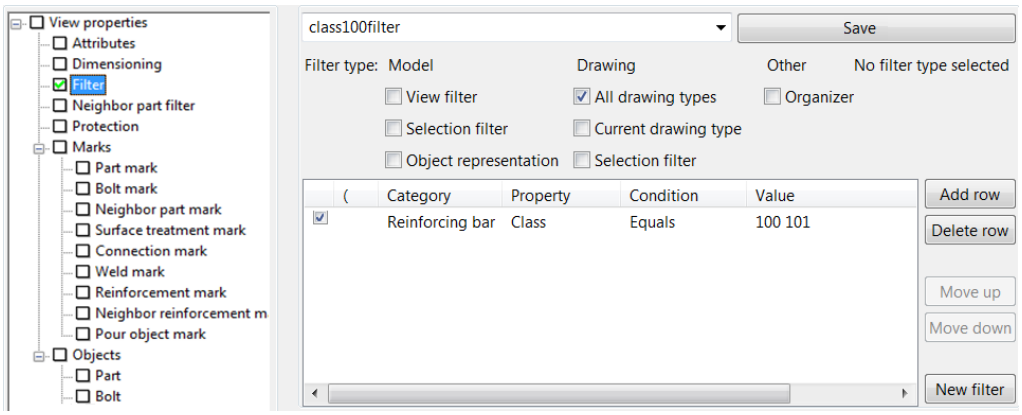
Example: Apply detailed object level settings on view level in a cast unit drawing

In this example, detailed object level settings will be set up for lifting anchors in a cast unit drawing.

Before doing this, drawing object properties on drawing view level for lifting anchors have been created so that they use different color. For details about creating drawing object properties, see the instructions above.

First you will make a cast unit drawing view filter, then save detailed object level settings in a settings file and finally, apply the new detailed object level settings in a view.

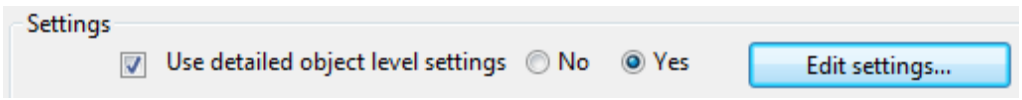
1. On the **Drawings & reports** tab, click **Drawing properties** --> **Cast unit drawing** .
2. Click **View creation**, and click the**View properties** button.
3. Click **Filter** in the options tree.



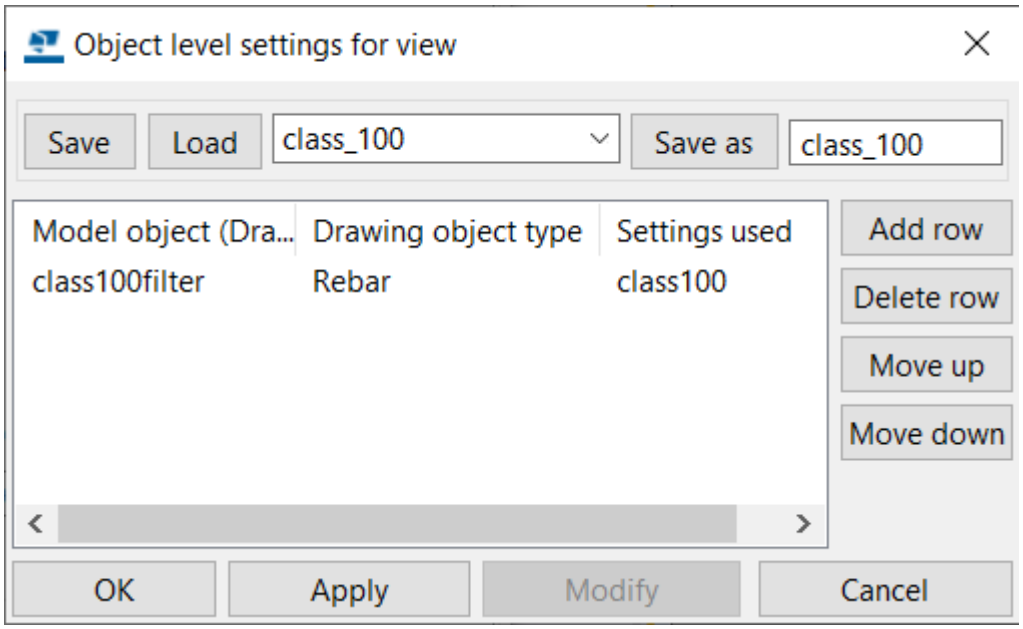
4. Create a filter by **Reinforcing bar** - **Class** and enter value 100 101.
5. Save the filter as `class100filter` and click **Save**.

   Now you have the filter needed for selecting the reinforcing bar.

6. Click **View properties** in the options tree and click **Edit settings...**.

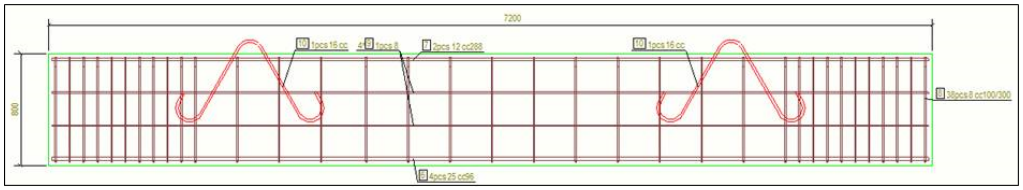


7. In the **Object level settings for view** dialog box, select the filter to be used, the drawing object type and the drawing object properties to be used:
   a. Click **Add row** and define the following values:
      Model object (Drawing view filter): `class100filter`
      Drawing object type: **Reinforcing bar**
      Settings used: `class100`
8. Save the detailed object level settings as `class_100` using **Save as**.



   The same detailed object level settings can be used in other cast unit drawing properties, you do not need to create new ones.
9. Click **Cancel** to close the dialog box.
10. Create a cast unit drawing containing lifting anchors and open the drawing.
11. Double-click the view frame to open the **View properties** dialog box.
12. Click **Edit settings...**.
13. Load object level settings you created, in this example `class_100`, and click **Load**.
14. Click **OK**.
15. Ensure that **Use detailed object level settings** is set to **Yes**.
16. Click **Modify** to apply the changes in the view.
17. Confirm the detailed object level changes by clicking **Yes** in the on **Object level settings changed.** message box.

The lifting anchors now have a different color, they are red.



NOTE If a row in the **Object level settings** dialog box is shown in red, either the object properties file defined in the **Settings used** box, or the filter file defined in the **Model object (Drawing filter) box** is missing from the `\attributes` folder in the current model folder. See an example below:

Model object (Drawing filter)	Drawing object type	Settings used
test_column_filter	Part	test_properties

How Tekla Structures applies drawing properties in drawing creation

Tekla Structures generates drawings according to the properties defined for each drawing type. You define the desired properties before you create the drawings. The way Tekla Structures connects the properties to the drawing you create depends on the method you select for creating the drawing.

- If you create single-part, assembly, or cast unit drawings drawings by using the **Create fabrication drawing** command on the ribbon, Tekla Structures creates the drawings by using the best matching drawing settings or applied drawing settings, according to your selection.
- If you create general arrangement drawings using the **Create GA Drawing** command on the ribbon, Tekla Structures uses the current GA drawing properties to create the drawings.
- If you create the drawings in the **Master Drawing Catalog** by using saved settings, Tekla Structures uses the properties defined in the saved settings file you select. Saved settings file in the **Master Drawing Catalog** is the same thing as the drawing properties file you save in various drawing properties dialog boxes.
- If you create the drawings in the **Master Drawing Catalog** by using a rule set, Tekla Structures creates the drawing according to the properties defined in the saved settings file or cloning template file used in the rule set.
- If you create the drawings in the **Master Drawing Catalog** by using cloning templates, Tekla Structures creates the drawing using the properties that you have defined for the drawing used as a cloning template together with the manual modifications that you have made in the drawing.

See also

Define automatic drawing settings (page 650)

Create fabrication drawings (page 107)

Create drawings in Master drawing catalog (page 121)

Create drawings in Tekla Structures (page 104)

Create general arrangement drawings (page 105)

Recreation of drawings

Changing some of the drawing settings requires the drawing to be recreated. The changes will be applied, and the drawing will be recreated only if and when you click **Modify**.

Settings affecting the recreation of drawings

In the **View creation** panel:

- **Coordinate System**
- **Around X**
- **Around Y**
- **Around Z**
- **Undeformed**
- **Unfolded**
- New views added into the **Views** list.

If at least one of the above mentioned settings is changed, the **Recreate the drawing** option is automatically set to **Yes** and the drawing will be recreated when you click **Modify**. If you manually set **Recreate the drawing** to **Yes**, the drawing will also be recreated. When **View creation** settings have changed and drawings will be recreated, you will get a warning message.

In the **Section view** panel:

- **Section depth**
- **Distance for combining cuts**
- **Left section**
- **Middle section**
- **Right section**

If at least one of the above mentioned settings is changed, Tekla Structures automatically recreates the drawing without giving a warning message when you click **Modify**.

If you only want to modify the view properties for one of the created views:

- If the changed properties are saved in the same file for all created views, the only way to modify the properties for one view is to set **Recreate the drawing** to **Yes**.
- If the changed properties are saved in a separate file not used for other created views, and that file is selected for the changed view in the **Views** list, then drawing will only be updated on **Modify**, not recreated.

Prevent automatic drawing updates and recreation

There are some ways to prevent drawings from being updated and/or recreated.

- Freeze the drawings where you do not want to update all associative objects on top of the drawing views. The building objects (parts, bolts, welds, etc) are always updated in frozen drawings. For more information about freezing, see Freeze drawings (page 577)
- Lock the drawings that you do not want to update. For details about locking, see Lock drawings (page 576)
- If you update a drawing that **has not been modified**, the drawing is recreated. Drawings are automatically recreated unless the drawing has been edited and then saved, or the drawing has been issued using the **Issue** functionality in **Document manager**. Use the advanced option XS_RECREATE_UNMODIFIED_DRAWINGS to control the recreation of unmodified drawings.
- To prevent Tekla Structures from automatically updating drawings if the model changes, set the advanced option XS_INTELLIGENT_DRAWING_ALLOWED to `FALSE`.

2.8 Drawing types

You can create many types of drawings in Tekla Structures according to your needs.

Click the links below to find out more:

- Examples of general arrangement drawings (page 89)
- Examples of single-part drawings (page 94)
- Examples of assembly drawings (page 97)
- Examples of cast unit drawings (page 99)
- Examples of multidrawings (page 102)

Examples of general arrangement drawings

A general arrangement drawing (GA drawing) is a contract document, which records information needed to understand the general arrangement structural elements on a project. A GA drawing is created in BIM workflows from one or more model views, with associated schedules and on a project title sheet. This is the most common form of drawing used by consulting structural engineers to communicate a structure sufficiently to other legal teams, public authorities, along with the design, costing and construction teams.

General arrangement drawings show the model from the most suitable direction. For example, in plan drawings, you are looking from the top of a building or floor down towards the ground. In elevation drawings you are looking from one of the sides of the building, like along a grid line. General arrangement drawings often contain enlarged views of complex areas or details, and other additional information that helps in the approval process and during the installation phase.

Create general arrangement drawings when you need

- Several views in one drawing, including the entire model or a part of it
- Plan drawings (foundation, floor, deck layout, and anchor bolt plans)
- Erection elevation drawings
- Information from model views, including 3D views

Click the links below to find out more about creating general arrangement drawings:

Quick start to Tekla Structures drawings (page 15)

Create general arrangement drawings (page 105)

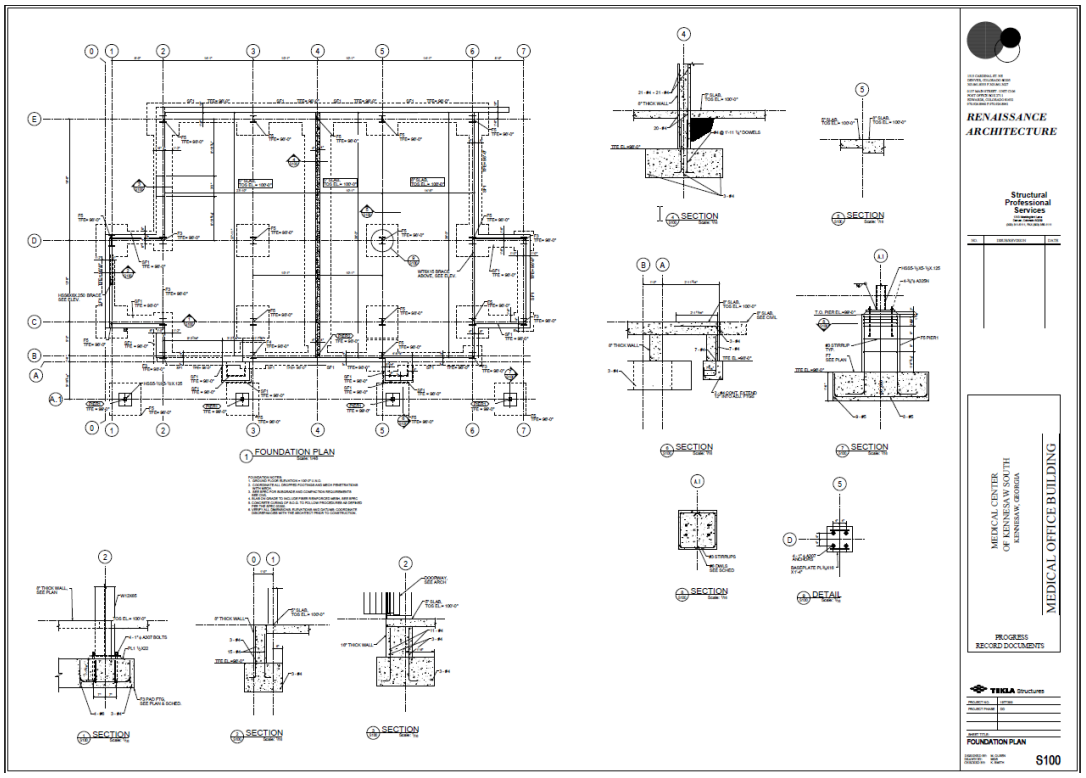
Create general arrangement drawings using saved settings in Master drawing catalog (page 126)

Create anchor bolt plans using saved settings (page 127)

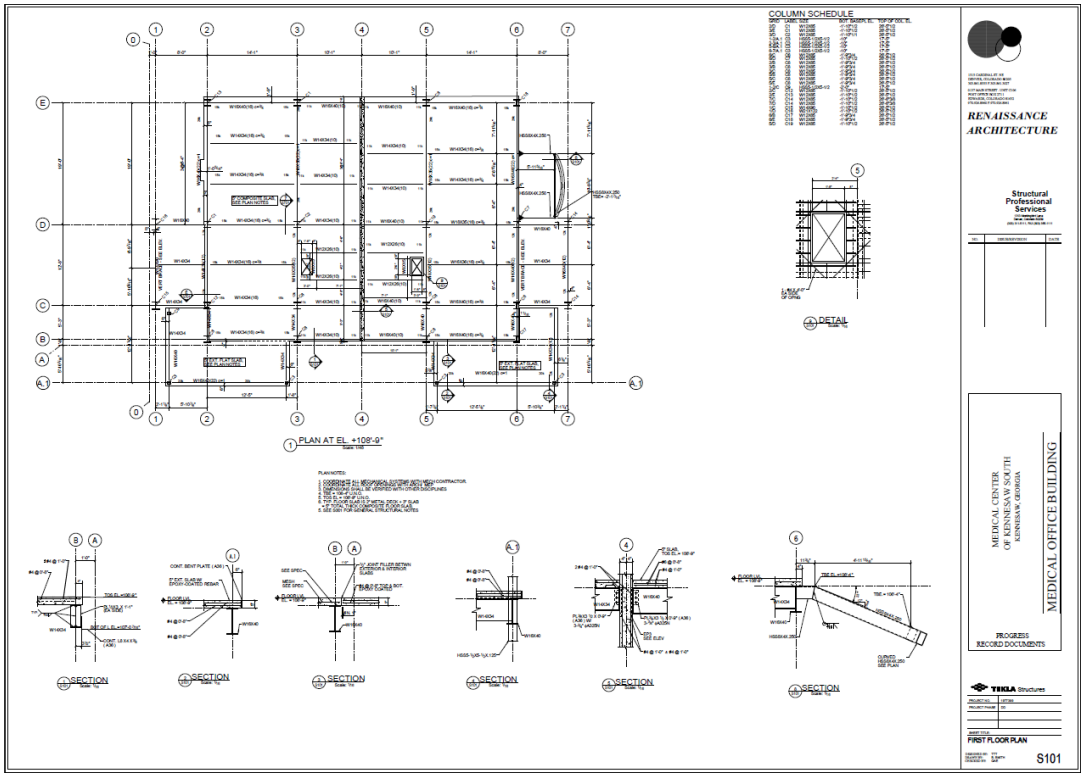
For details on about GA drawing properties, see General arrangement drawing properties (page 903).

Examples of GA drawings

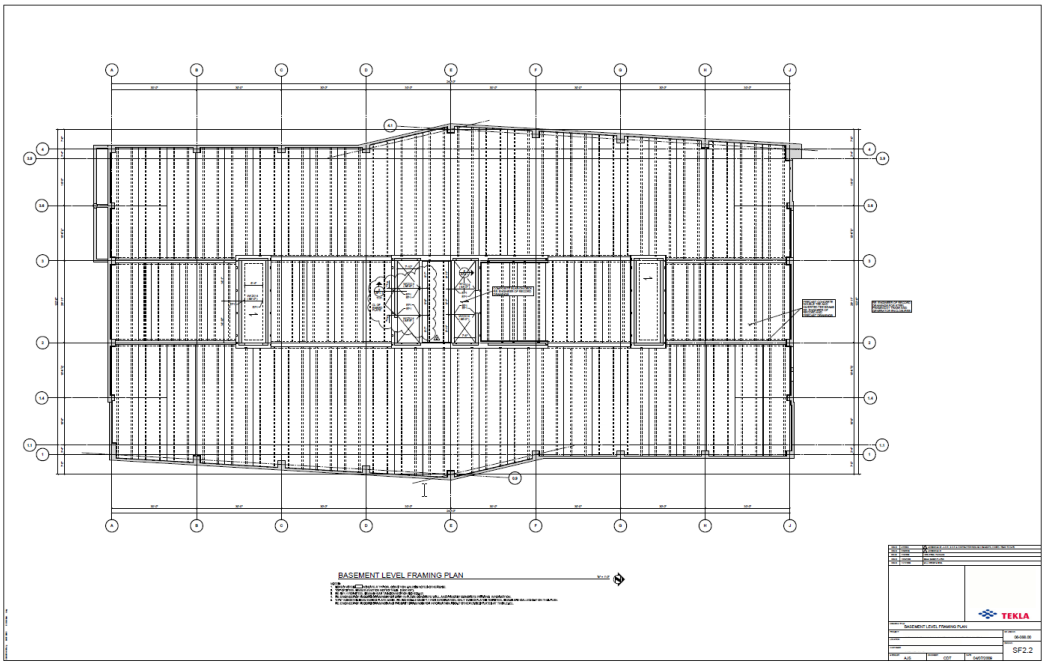
Foundation plan:



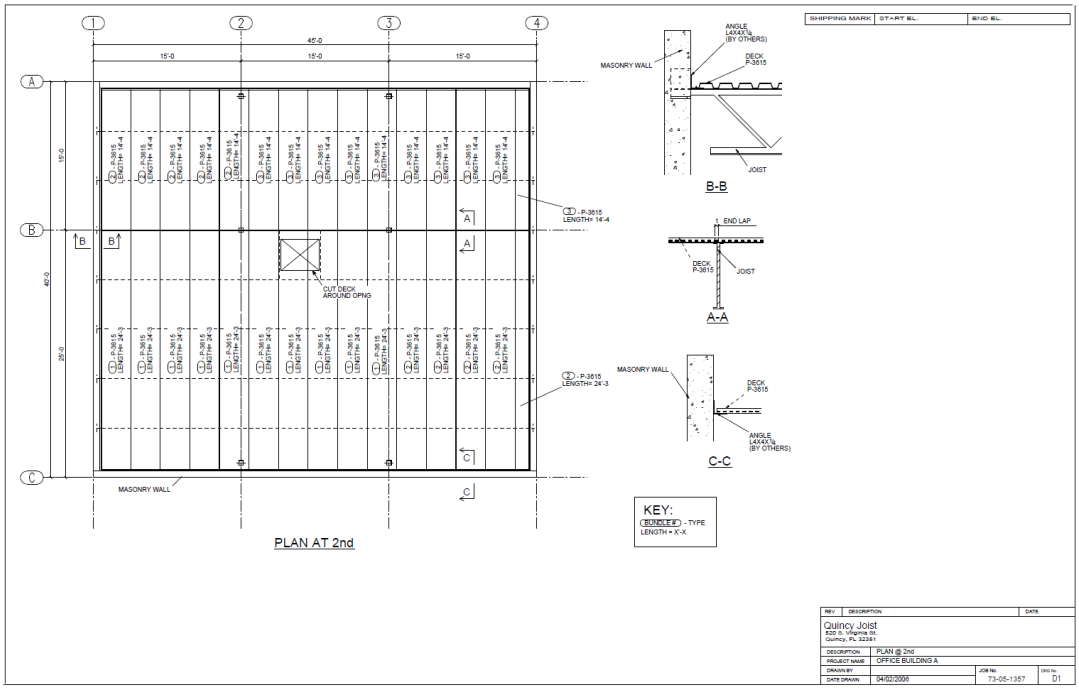
Slab plan:



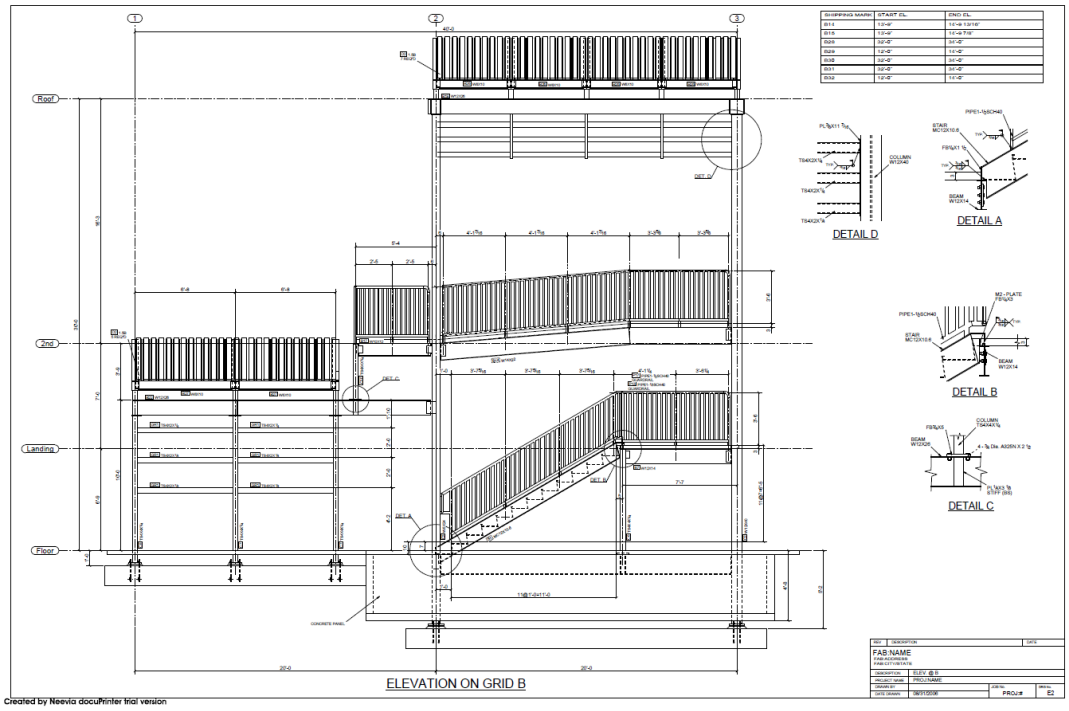
Framing plan:



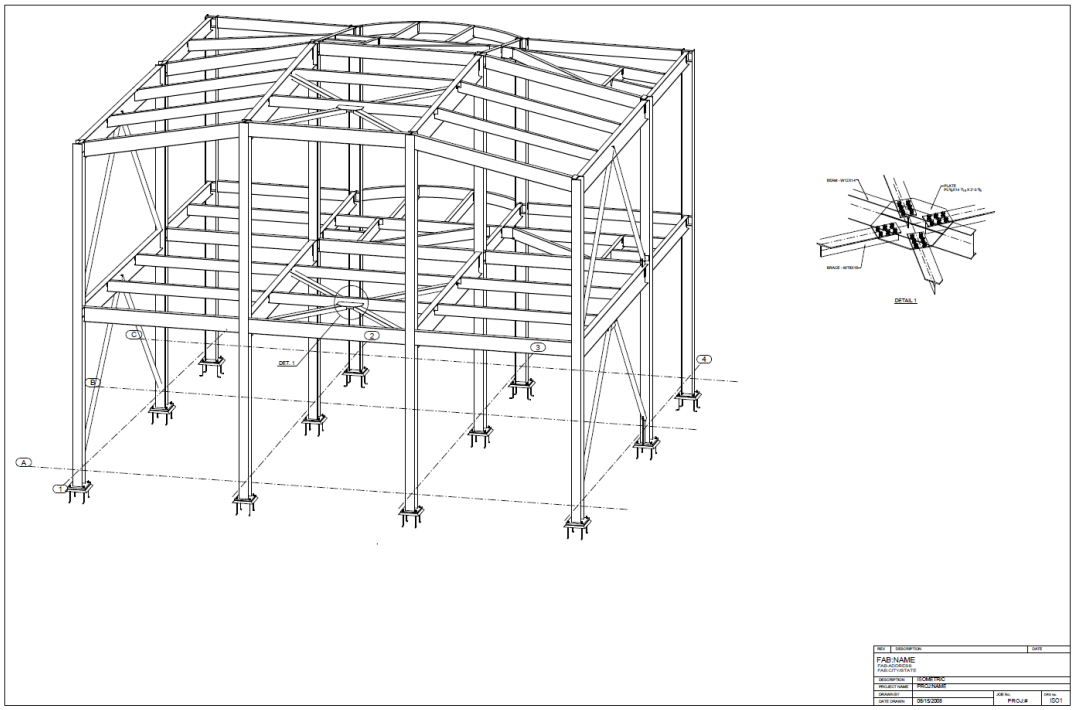
Deck plan:



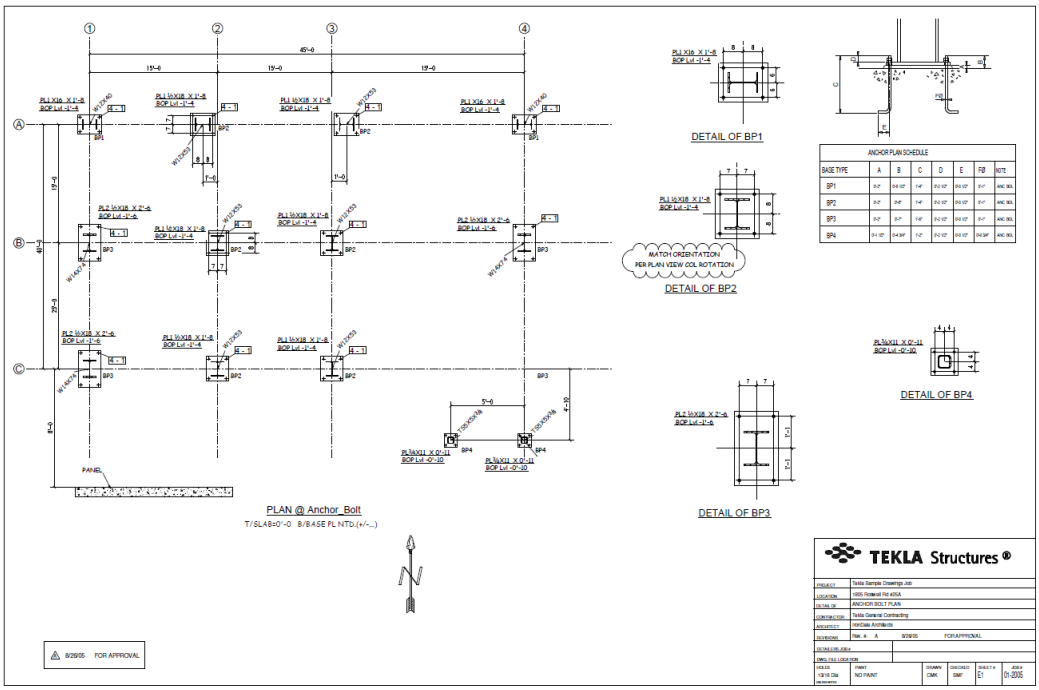
Erection elevation drawing:



Isometric drawing:



Anchor bolt plan:



Examples of single-part drawings

Single-part drawings are workshop drawings showing fabrication information for one part (usually without welds). Single-part drawings usually use small sheet sizes, for example, A4 or imperial standard size 8.5" x 11.5".

Click the links below to find out more about creating single-part drawings:

Quick start to Tekla Structures drawings (page 15)

Create fabrication drawings (page 107)

Create single-part, assembly, or cast unit drawings through Quick launch (page 113)

Create single-part, assembly and cast unit drawings using saved settings in Master drawing catalog (page 130)

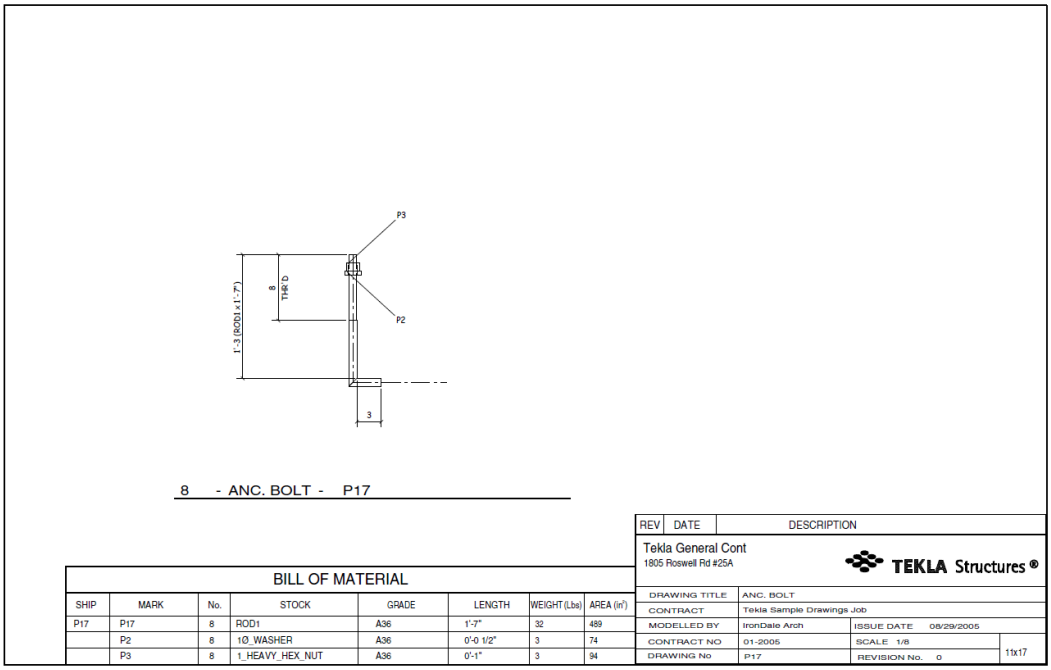
Create drawings using rule sets or wizards in Master drawing catalog (page 135)

Create AutoDrawings (page 115)

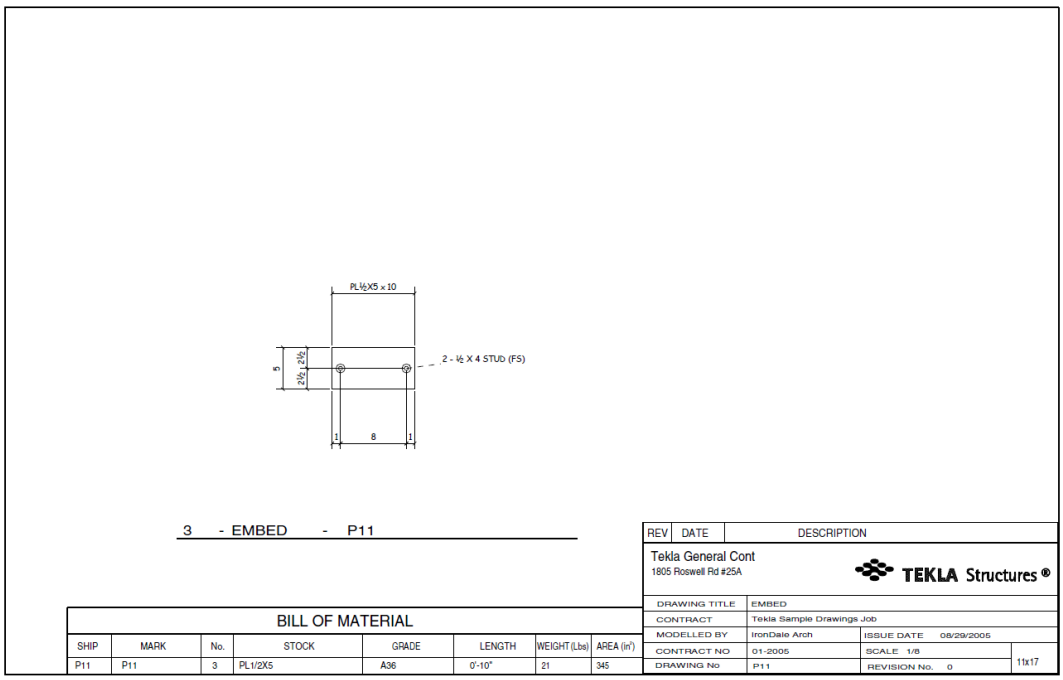
For details about single-part drawing properties, see Single-part drawing properties (page 906).

Examples of single-part drawings

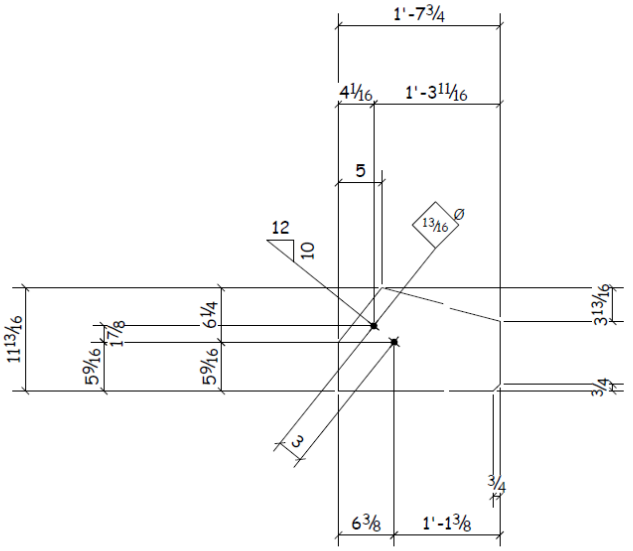
Single-part drawing presenting an anchor bolt:



Single-part drawing presenting an embed:



Single-part drawings presenting plates (three examples):

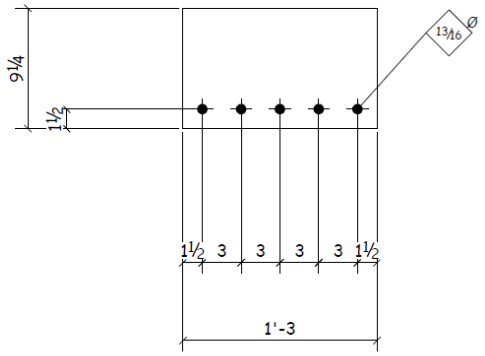


No	In assembly
1	100C1
1	C7
1	C9
1	C10

p2	PL1 X11 13/16	A36	4	1'-7 3/4"	435.36	53.8
MARK	SIZE	GRADE	No.	LENGTH	AREA	WEIGHT
MATERIAL LIST FOR WORKSHOP				TOTAL	1741.42	215.1

TEKLA Structures ®

DRAWING NUMBER:	W [p.2]		
CONTRACT NO:	01-2005	DESCRIPTION:	PLATE
ISSUE NO:		ISSUE DATE:	00/00/0000
REVISION No:		REVISON DATE:	

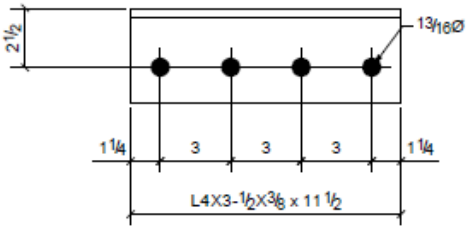


No	In assembly
2	100C1
4	C5
4	C6
2	C7

p6	PL3/8X9 1/4	A36	12	1'-3"	296.55	14.8
MARK	SIZE	GRADE	No.	LENGTH	AREA	WEIGHT
MATERIAL LIST FOR WORKSHOP				TOTAL	3558.57	177.2

TEKLA Structures ®

DRAWING NUMBER:	W [p.6]		
CONTRACT NO:	01-2005	DESCRIPTION:	PLATE
ISSUE NO:		ISSUE DATE:	00/00/0000
REVISION No:		REVISON DATE:	



2	101B6
2	101B11
4	101B12
Qty.	In Assembly

MARK	SIZE	GRADE	QTY.	LENGTH	AREA	WEIGHT
a1	L4X3-1/2X3/8	A36	8	0'-11 1/2"	178	9
MATERIAL LIST FOR WORKSHOP				TOTALS	1423 in²	70 lb/ft

FAB:NAME FAB:ADDRESS FAB:CITY/STATE	DRAWING No.	a1		
	JOB No.	1001	DESCRIPTION	ANGLE
	DRAWN BY		REVISION No.	
	DATE DRAWN	03/18/2008	REVISION DATE	

Examples of assembly drawings

Assembly drawings are typically workshop drawings showing fabrication information for one assembly. In most cases, an assembly consists of a main part and secondary parts. The secondary parts are either welded or bolted to

a main part. Assembly drawings usually use larger sheet sizes than single-part drawings, for example, A3 or imperial standard size 11" x 17".

Click the links below to find out more about creating assembly drawings:

Quick start to Tekla Structures drawings (page 15)

Create fabrication drawings (page 107)

Create single-part, assembly, or cast unit drawings through Quick launch (page 113)

Create single-part, assembly and cast unit drawings using saved settings in Master drawing catalog (page 130)

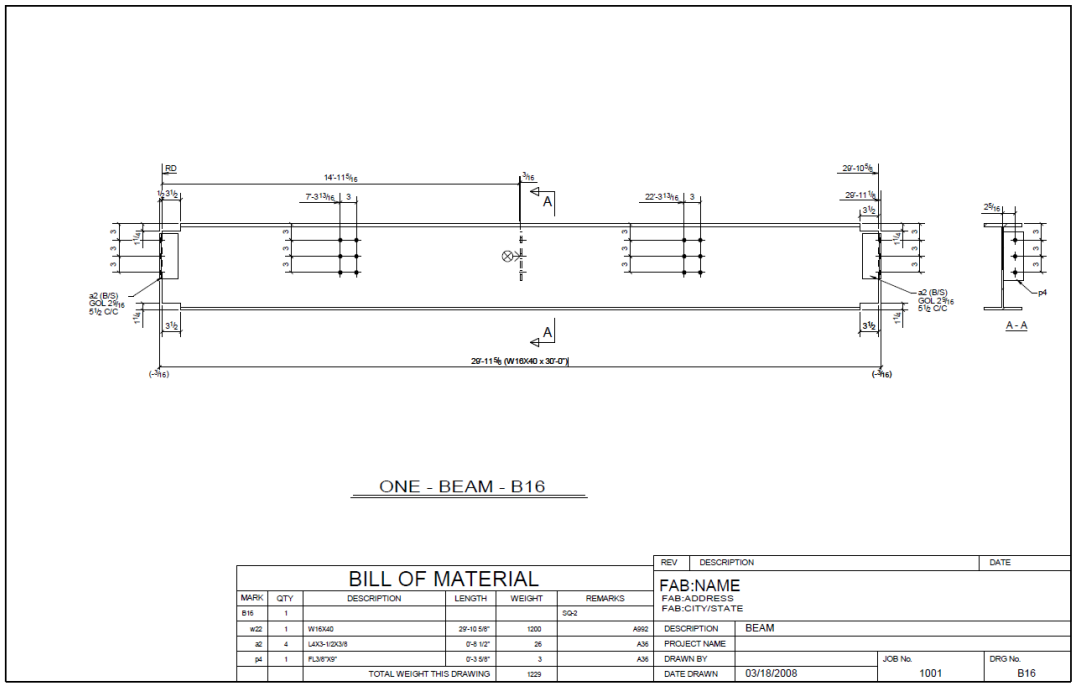
Create drawings using rule sets or wizards in Master drawing catalog (page 135)

Create AutoDrawings (page 115)

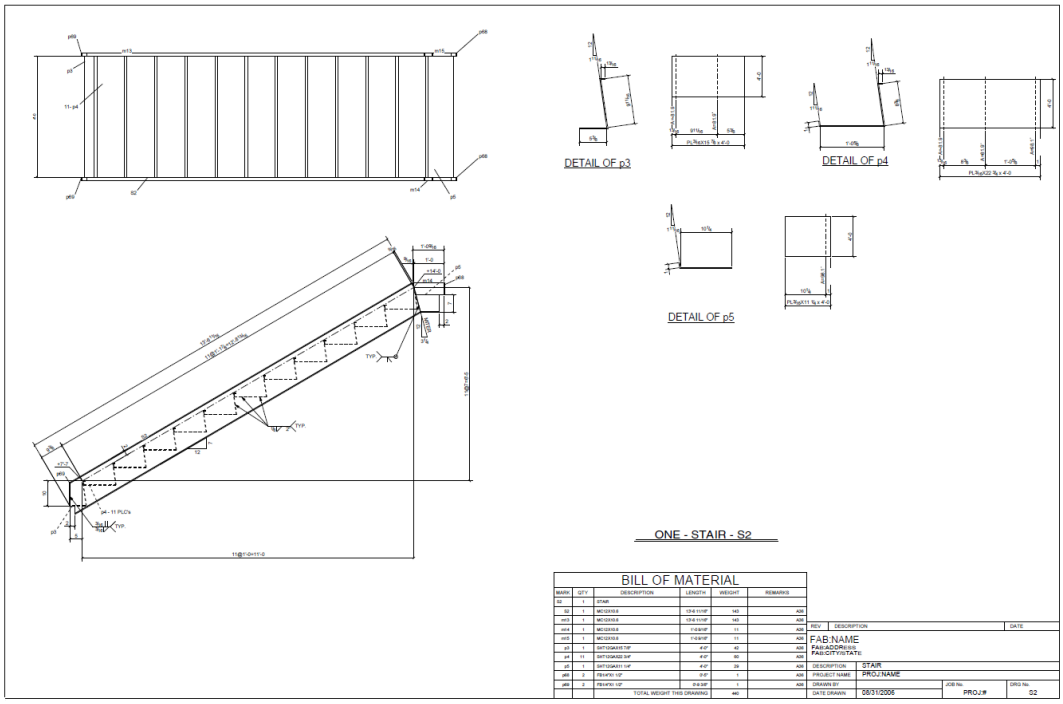
For details about assembly drawing properties, see Assembly drawing properties (page 906)

Examples of assembly drawings

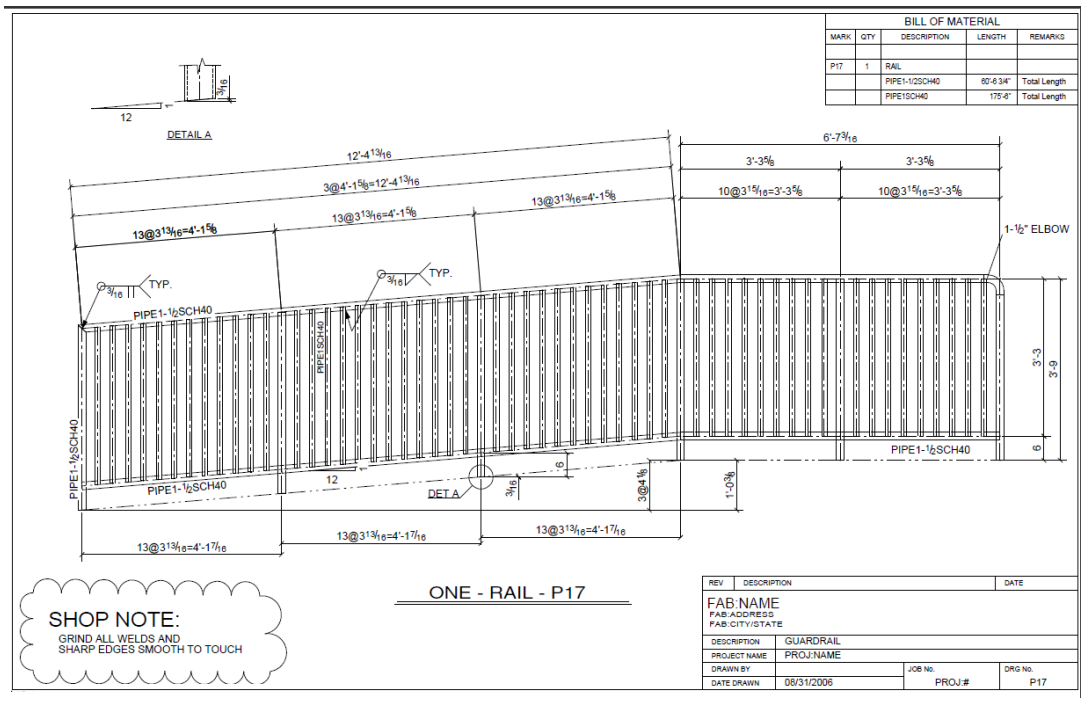
Beam assembly drawing:



Stairs assembly drawing:



Rail assembly drawing:



Examples of cast unit drawings

Cast unit drawings are dimensional, form work, or reinforcement drawings used in concrete design and construction. They show cast-in embeds, edge chamfers, and hard and soft insulation. Cast unit drawings that show cast-in-place concrete structures usually use large sheet sizes, for example, A1 or imperial standard size 24" x 36". Drawings that show precast structures usually use smaller sheet sizes, for example, A3 or imperial standard size 11" x 17".

You can only create cast unit drawings of concrete parts and cast-in embeds (steel parts or assemblies added to a cast unit). Cast unit drawings also show bolts and welds in steel parts. The volume and weight information of the cast unit remains accurate even if there are cuts inside the cast unit.

Click the links to find out more about creating cast unit drawings:

Quick start to Tekla Structures drawings (page 15)

Create fabrication drawings (page 107)

Create single-part, assembly, or cast unit drawings through Quick launch (page 113)

Create single-part, assembly and cast unit drawings using saved settings in Master drawing catalog (page 130)

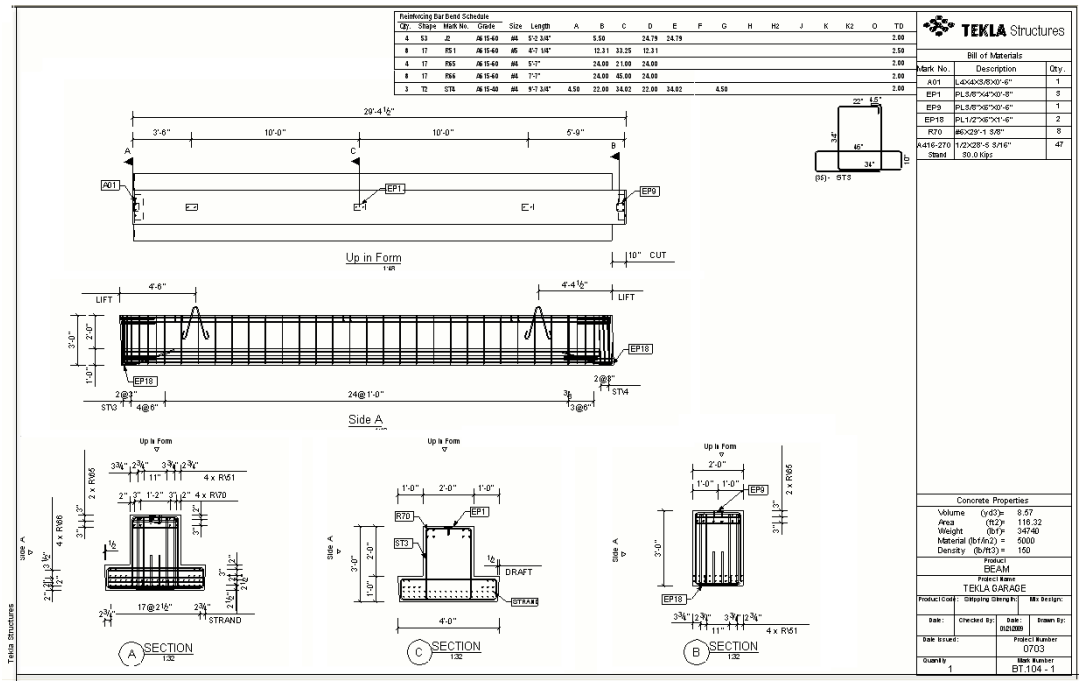
Create drawings using rule sets or wizards in Master drawing catalog (page 135)

Create AutoDrawings (page 115)

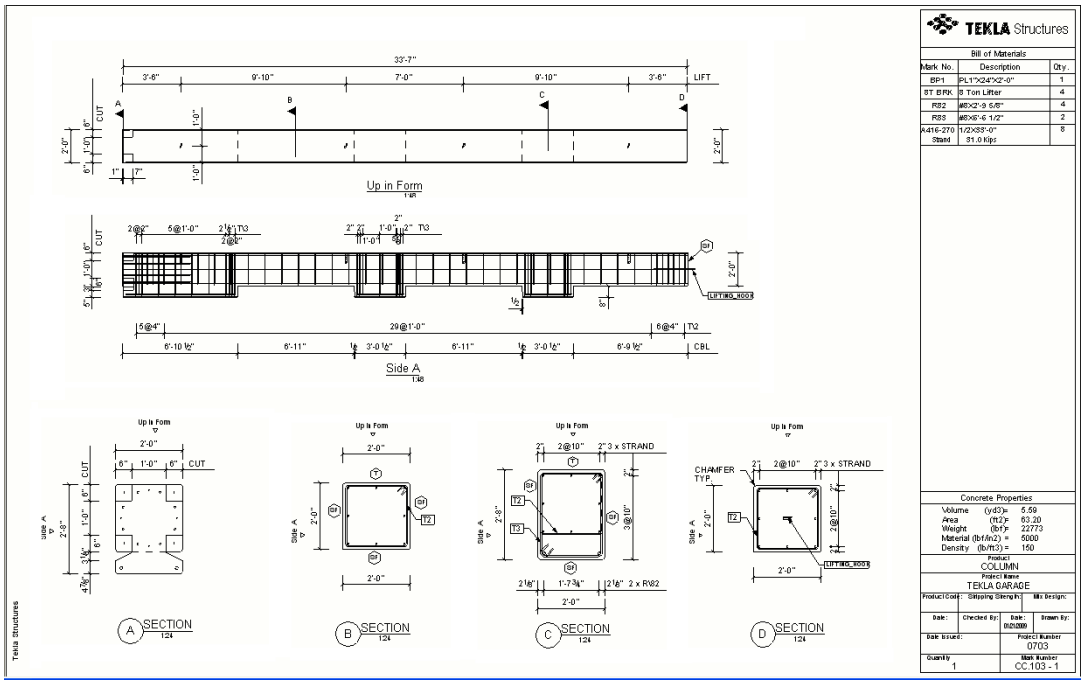
For details about cast unit drawing properties, see Cast unit drawing properties (page 906)

Examples of cast unit drawings

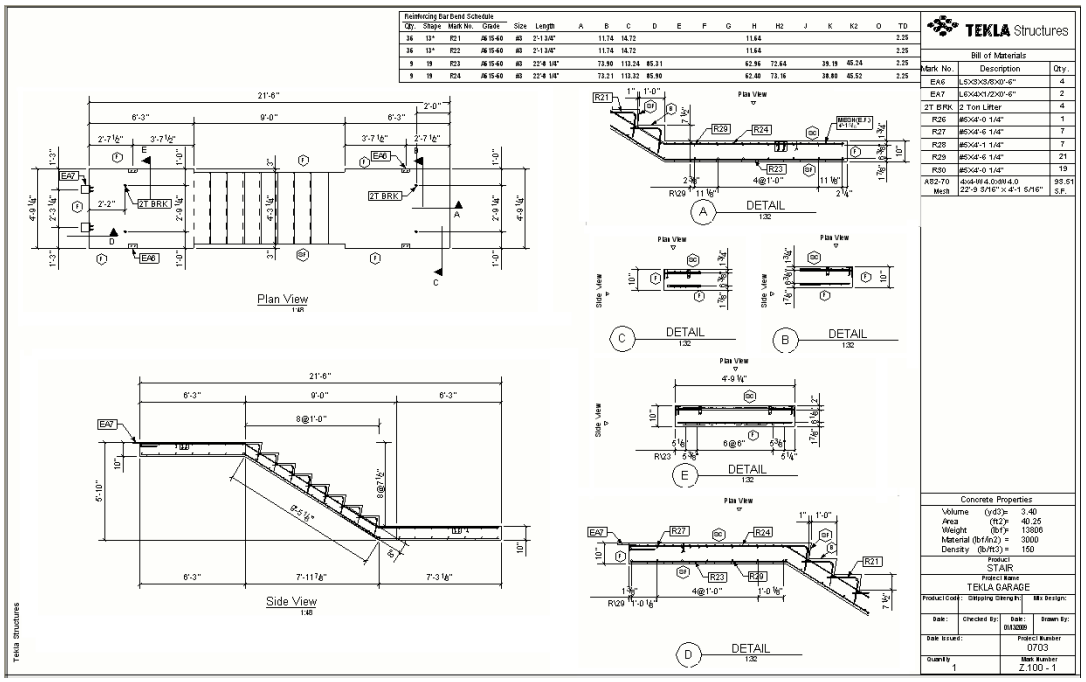
A combined form and reinforcing bar drawing of an inverted tee beam:



A combined form and reinforcing bar drawing of an exterior column with no corbels or plats for beams:



A combined form and reinforcing bar drawing of stairs with landings:



Examples of multidrawings

Multidrawings are workshop drawings that gather several single-part or assembly drawings on one sheet. Multidrawings usually demand large sheet sizes such as A1 or imperial standard size 24" x 36".

Create multidrawings when you want to:

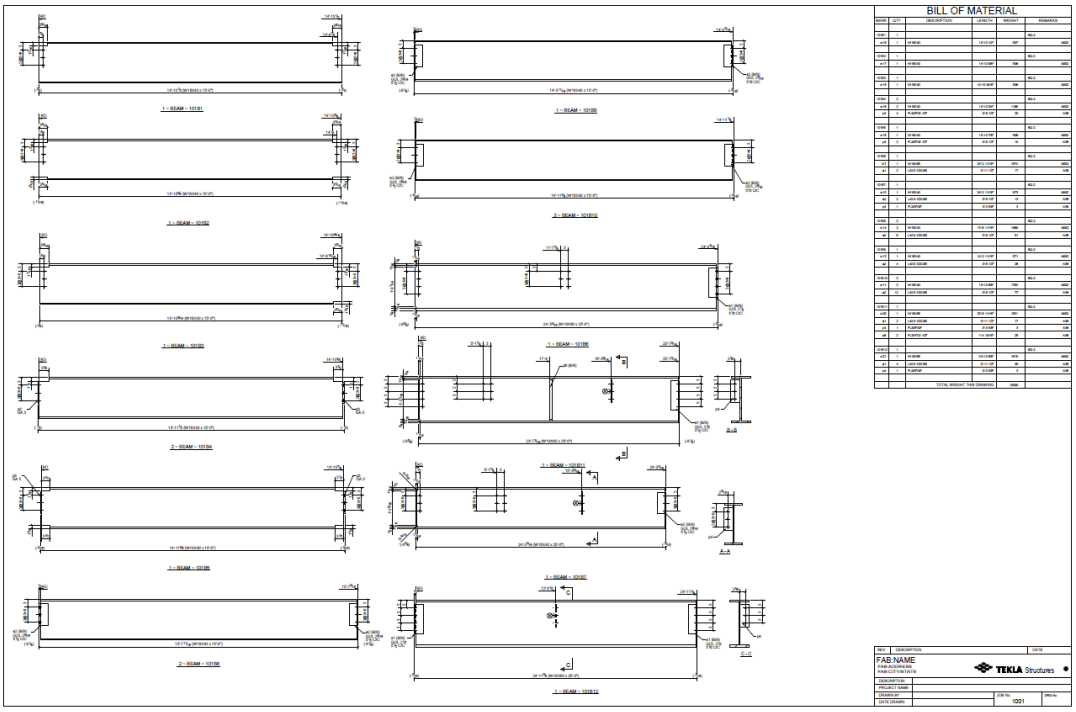
- Have more than one assembly on a sheet
- Collect multiple single-part drawings on a large sheet

You can create multidrawings in the following ways:

- Create empty multidrawings and link or copy drawing views to the created empty multidrawings
- Create multidrawings of selected parts
- Create multidrawings of selected drawings

For details about the different ways of creating multidrawings, see Create multidrawings (page 118).

Below is an example of a multidrawing with several assemblies included.


BILL OF MATERIAL
FAB:NAME
TEKLA Structures
1001

3 Create drawings in Tekla Structures

Drawing creation is always based on drawing properties, no matter which way you use to create them. Planning and implementing the most suitable drawing settings carefully is very important. You can create drawings one by one, in groups, or you can create all drawings automatically.

Choose the method that suits you the best:

- The **Create fabrication drawing** command is an easy way to create single-part, assembly, or cast unit drawings, if you have drawing templates available for use, or if you have saved settings that you want to apply.
- You can find drawing creation commands on the ribbon, context menu, contextual toolbar, and in **Quick Launch**.
- You can also create drawings using the various master drawing types in the **Master drawing catalog**. Using rule sets is a highly automated process of creating several drawings of different type in one go.
- You can clone drawings on the basis of cloning templates in the current model or in the cloning template models. You can clone drawings in the **Master drawing catalog** and in **Document manager**.
- You can increase the level of automation even more by applying detailed object level settings to all of the above mentioned methods. When the level of automation increases, the need for manual modifications decreases. We recommend that you put extra effort in making the automated settings as efficient as possible.

Click the links below to find out more:

- Drawing types (page 89)
- What to do before creating drawings (page 105)
- Quick start to Tekla Structures drawings (page 15)
- Create general arrangement drawings (page 105)
- Create fabrication drawings (page 107)

- Create single-part, assembly, or cast unit drawings through Quick launch (page 113)
- Create multidrawings (page 118)
- Create drawings in Master drawing catalog (page 121)
- Clone drawings (page 153)
- Detailed object level settings (page 79)

3.1 What to do before creating drawings

The list below contains some things that you may need to do before you create or clone drawings in Tekla Structures:

- For assembly, single-part, cast unit and multidrawings, the model needs to be numbered. If the model changes, the numbering needs to be updated. Numbering ensures that Tekla Structures connects the right objects to the right drawings. If you try to create drawings without numbering the model, Tekla Structures prompts you to do so. Once the model is numbered, part positions can also be shown in general arrangement drawings. Prior to numbering, if the part position is included in a mark on a general arrangment drawing, it will be shown as a question mark (?).
- Check that the detailing is correct.
- You may want to create test drawings of different types of parts to see how well the predefined drawing and view properties, layouts, rule sets or cloning templates suit your needs.
- You may want to modify drawing and view properties, layouts, or rule sets and save the modified settings for future usage.
- If you are going to use a drawing as a cloning template, check that it contains all the necessary elements for this purpose.

See also

Create fabrication drawings (page 107)

Create general arrangement drawings (page 105)

Create single-part, assembly and cast unit drawings using saved settings in Master drawing catalog (page 130)

Create multidrawings (page 118)

Define automatic drawing settings (page 650)

3.2 Create general arrangement drawings

When you create general arrangement drawings, remember to load the closest possible predefined properties in the drawing properties first, then modify the properties if needed, and then create the drawing.

Before you create general arrangement drawings, create the model views that you need and ensure that the views are as you want them to be in drawings. The drawing views will have the same orientation and content as the model view you select. A good idea is to fit the work area in the model view using two points to select the area that you want to show in the general arrangement drawing.

1. Create the model views you need.

   If you want to switch between 3D view and plane view in the created view, press **Ctrl+P**.

2. On the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing** .

3. Select the appropriate predefined drawing properties (saved settings) from the list at the top and click **Load**.

   Always load predefined properties when you are creating a drawing. When you need to modify drawing properties, save the new changes to the properties file when necessary.

4. If needed, modify drawing properties and, if necessary, apply object-level settings.

5. Click **Apply** or **OK**.

6. On the **Drawings & reports** tab, click **Create GA Drawing**.

   Alternatively, you can select the model views from the model (selected views get a yellow frame), right-click and click **Create general arrangement drawing...**

7. If you have not selected the views yet, select them from the displayed list.

   You can use the **Shift** and **Ctrl** buttons for selecting several views.

8. In the **Options** list, select if you want to create one drawing for each selected view or add all selected views in one drawing.

   By selecting **Empty drawing**, you can create an empty drawing and add drawing views in it afterwards. For more information about copying, moving or linking drawing views to another drawing, see Copy, move and link drawing views (page 188).

9. If you want to open the created drawings, select **Open drawing**.

10. Click **Create**.

Tekla Structures creates the drawings and adds them in **Document manager**. You can now open the drawings and also change the drawing properties.

See also

Define automatic drawing settings (page 650)

Examples of general arrangement drawings (page 89)

Create general arrangement drawings using saved settings in Master drawing catalog (page 126)

General arrangement drawing properties (page 903)

Show pours in drawings (page 485)

3.3 Create fabrication drawings

Create assembly, single-part or cast unit drawings depending on the type of the selected model objects, based on the best matching drawings or applied settings.

You can create drawings by using saved settings predefined for different model objects, or by using existing drawings in your model as templates. You can efficiently manage the creation process, and select settings for every individual model object or, alternatively, select the best matching drawing template automatically detected in the current model.

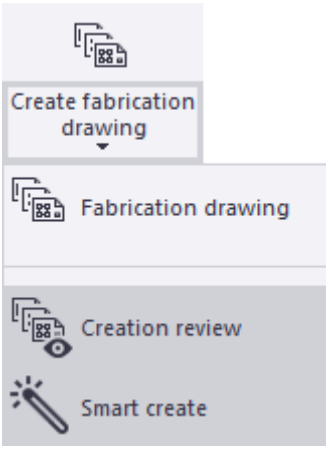
Before you start

Before you start creating fabrication drawings, consider the following:

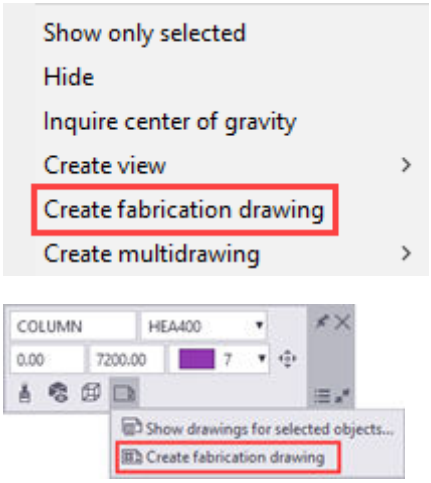
- When you create assembly drawings, ensure that the **Select assemblies** selection switch is active in the model before you select any objects. Alternatively, you can use appropriate selection filters and the **Object visibility** options in the **Creation review** dialog box to define the objects for which you want to create drawings.

  If you have selected parts in the model, a message is displayed asking you to confirm if you want to create single-part drawings. If you want to create single-part drawings, click **OK**, otherwise click **Cancel**.

  This is how the ribbon drop-down looks by default with the **Create fabrication drawing** command and with both of the switches enabled:



- Enable or disable the **Creation review** and **Smart create** switches depending on whether you want to create drawings immediately, or open the **Creation review** dialog box for more controlled creation process.
- You can select the **Create fabrication drawing** command from the ribbon, context menu or contextual toolbar.



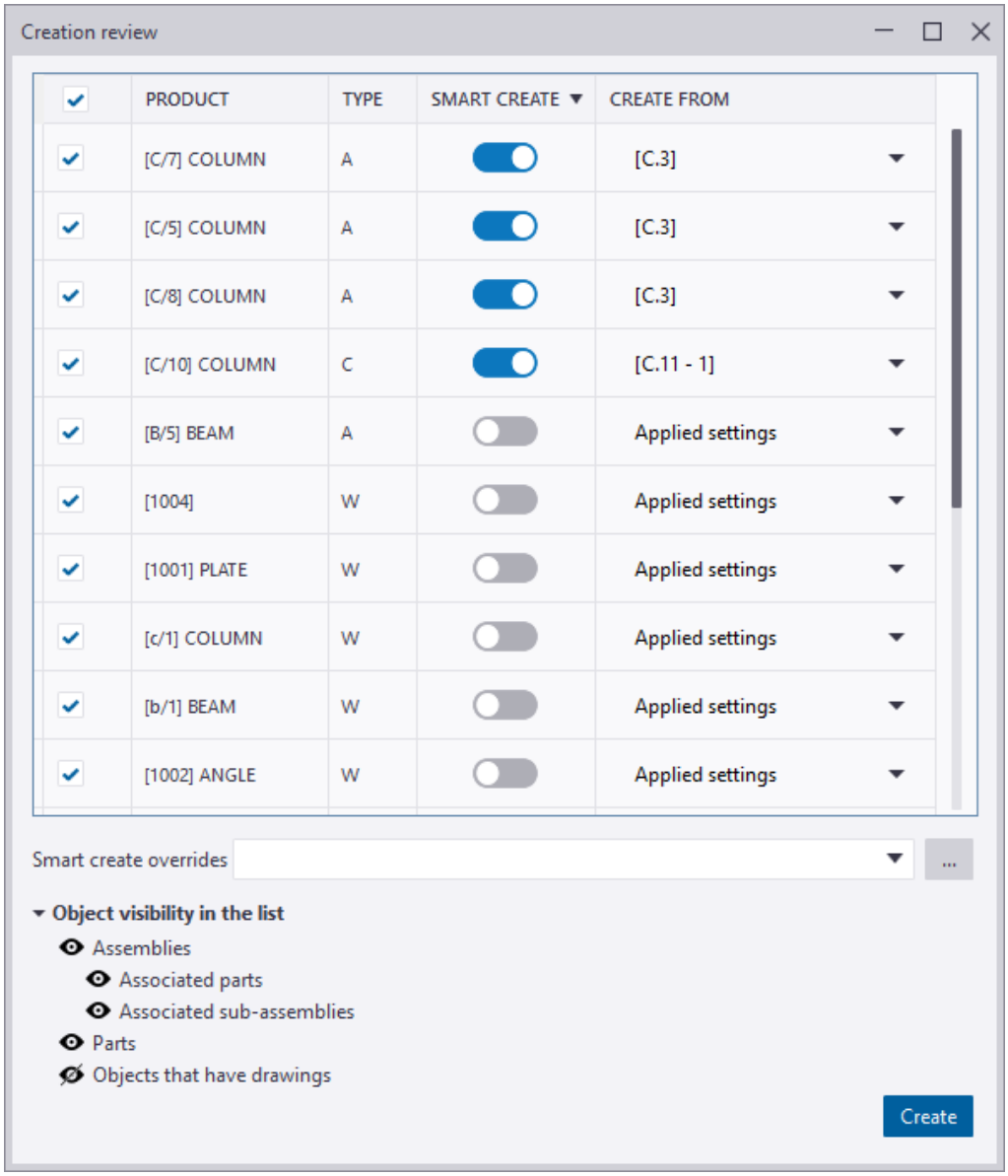
Control drawing creation with Creation review

To have more control over the drawing creation process, enable both the **Creation review** and the **Smart create** switch, and select the **Create fabrication drawing** command. Using **Creation review** is especially useful if you do not have any drawings in your model to use as templates, or you know exactly which saved settings you want to use.

1. Select objects in the model.
2. Enable the **Creation review** and **Smart create** switches. If you do not have drawing templates that you want to use, or if you want to use specific drawing settings, enable the **Creation review** switch only.

3. Select **Create fabrication drawing** on the ribbon, context menu or contextual toolbar.

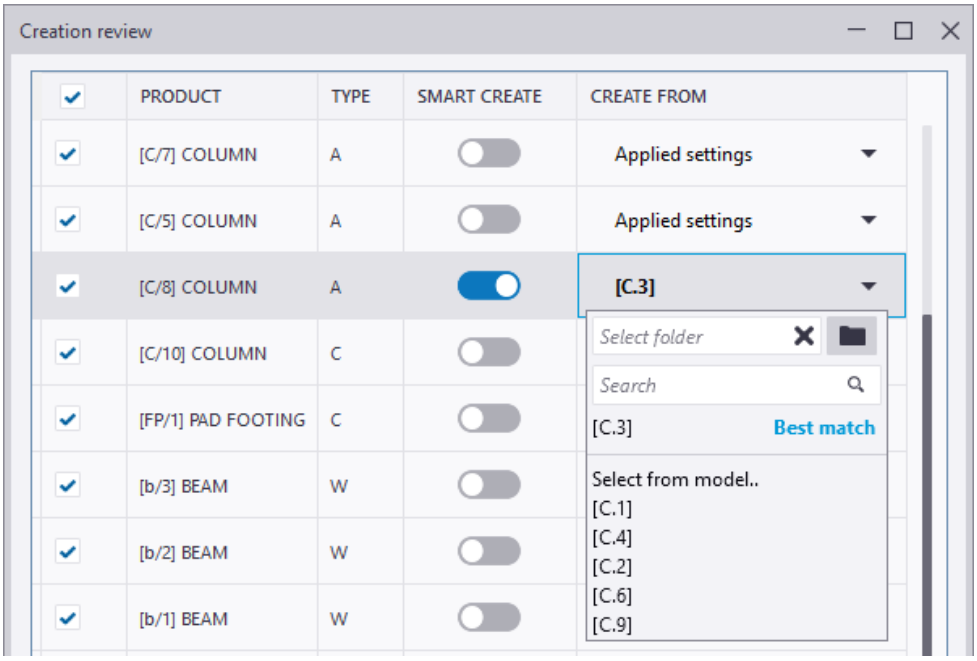
   After the tool has completed the evaluation of the model objects, the **Creation review** dialog box is displayed with a list of the selected objects. The best matching drawing templates for each individual object are shown in the **Create from** column.



4. Select the desired options and settings in the dialog box:

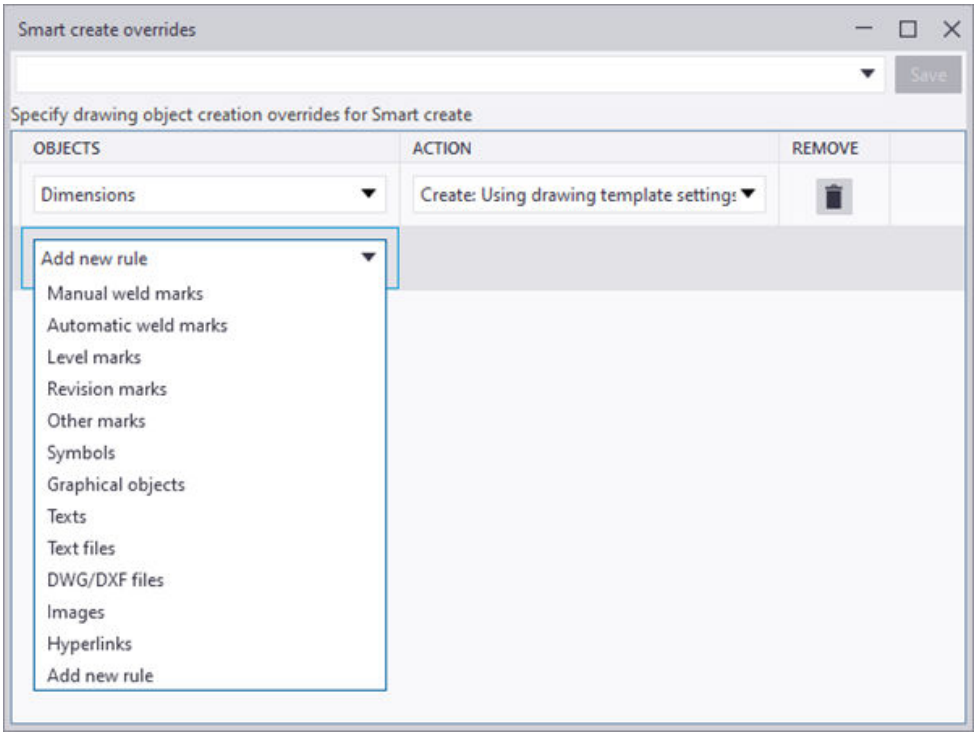
   - When you have **Smart create** enabled, you can select any other drawing template from the current or any other model in the drop-

down list. To use templates in another model, browse to the model folder. When no matching template is found, you will be notified.



- When you have **Smart create** enabled, you can select in the current model the drawing objects whose drawings you want to use as a template. To do this, select **Select from model...**, and select the object from the model among the objects that are available for selection.
- When you have **Smart create** enabled, define whether certain drawing objects should be recreated by using the settings of the selected

drawing template or applied settings. To do this, open the **Smart create overrides** dialog box by clicking the button with three dots.

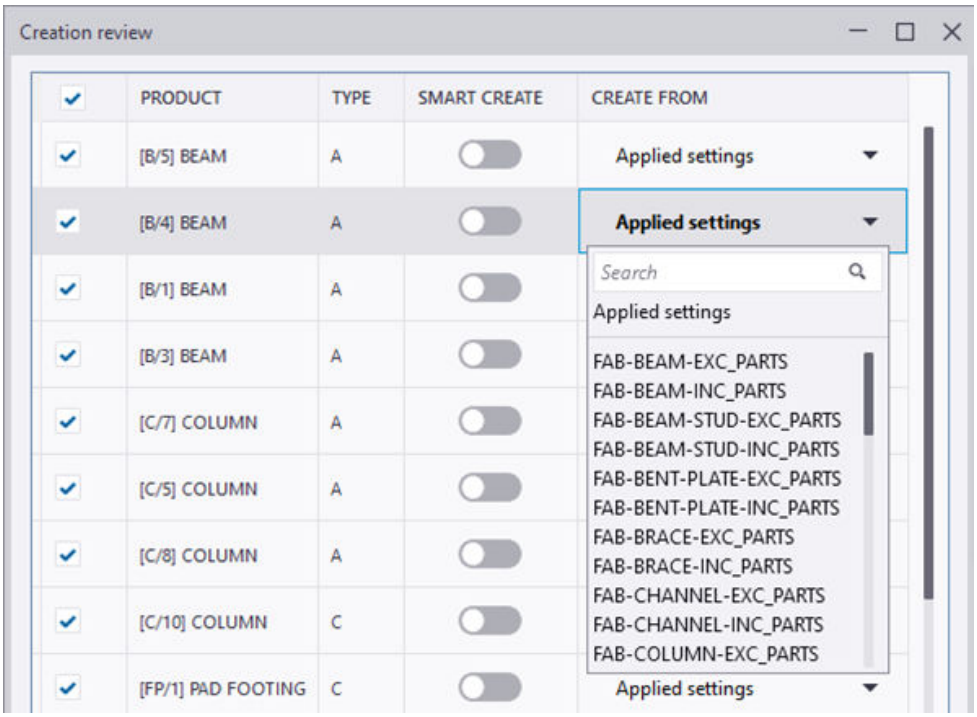


- To create a new override rule, select an object type from the **Objects** list, then select an action type in the **Action** list, give the override settings file a name, and click the **Save** button.
- Select the desired override settings file in the **Smart create overrides** list in the **Creation review** dialog box to use it.

  If you have added an override rule, but you have not saved it, the rule will not be used.
- If you do not want to use any override rules, select an empty string in the **Smart create overrides** list.

- To use applied settings only, disable **Smart create**. By default, if the best matching template has not been found, **Smart create** is disabled automatically and **Applied settings** becomes available. You can open the drop-down list to select any other saved settings.

  This method is handy especially when you now exactly which saved settings you want to use.



- Clear the check boxes in front of the model objects in the **Product** list to prevent the creation of drawings for these objects.
- To select the same best matching template or applied settings for several model objects, hold down **Shift** or **Ctrl**, select the model objects, and then select the desired best matching template or applied settings in the **Create from** column.
- Use search to find the desired drawing templates or applied settings.
- Enable or disable the following **Object visibility** options to control which objects are displayed in the **Creation review** dialog box:
  - **Assemblies**: Lists selected assemblies.
    - **Associated parts**: Lists selected parts associated to assemblies.
    - **Associated sub-assemblies**. Lists selected sub-assemblies associated to assemblies.
  - **Parts**: Lists all selected parts.
  - **Objects that have drawings**: Lists objects that already have drawings.

5. Click **Create**.

The drawings are created, based on drawing templates or applied settings, according to the settings defined, and for all the model objects listed in the **Creation review** dialog box. The created drawings are listed in **Document manager**.

Create drawings with Smart create

If you have created drawings in your model already, you can create drawings by using the best matching drawing template automatically detected in the current model. This method is very fast and efficient.

Note that **Smart create** does not look for the best matching drawing template for single-part and rebar assembly objects.

1. Select objects in the model.
2. Enable the **Smart create** switch only.
3. Select **Create fabrication drawing** on the ribbon, context menu or contextual toolbar.

The drawings are created based on the best matching drawing template in the current model. If a matching drawing is not found, drawings are created from the settings applied in the drawing properties dialog box. The created drawings are listed in **Document manager**.

3.4 Create single-part, assembly, or cast unit drawings through Quick launch

When you create single-part, assembly or cast-unit drawings through **Quick Launch**, first load the closest possible predefined properties in the related drawing properties dialog box first, then modify the properties if needed, and then create the drawing. Your environment contains some predefined properties and there are also some in the default environment.

Another way of creating single-part, assembly, and cast unit drawings is to use the **Create fabrication drawing** command. For details, see Create fabrication drawings (page 107).

NOTE For cast unit drawings, ensure that you are using the correct cast unit type, because some functionalities, for example numbering, are based on the cast unit type. Cast units have the cast unit type part property setting in the model - they are set either **Cast in place** or **Precast**. In the **Cast in Place** configuration, you can create cast unit drawings of cast-in-place cast units only.

To create drawings through **Quick Launch**:

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.

2. In the drawing properties dialog box, load the appropriate predefined drawing properties (saved settings).

   Always load predefined properties when you are creating a drawing. When you need to modify drawing properties, save the changes to a new properties file for future use.

   Ensure that for cast unit drawings, you use a property file that contains the desired **Cast unit definition method**:

   - **By cast unit position**: A drawing will be created from each cast unit. If there are identical cast units, one of them will serve as the base cast unit for the drawing. This is the most common method for creating cast unit drawings.
   - **By cast unit ID**: Each part in the model includes a unique GUID. You can create drawings by using cast unit GUIDs. The GUID determines the marking of the drawing. You can create several drawings from identical cast units.

3. Click **View creation** in the options tree on the left and do the following:
   - In **Common settings for all views**, define the Ignore bolt size, Weld size limit and Weld marks visible (page 906) settings, if necessary.
   - Add the necessary drawing views (page 689) and then select the view properties for the views in the **View properties** list.
4. Select the view and the properties that you want to change, click **View properties**, and modify the view properties including view, building object, dimensioning and mark settings and apply detailed object-level settings.
5. Click **Save** to save the view properties.
6. Click **Close** to return to drawing properties.
7. Save the drawing properties you loaded earlier.
8. Click **Apply** or **OK**.
9. Select the objects, or use an appropriate selection filter to select the objects you want to create drawings from and select the entire model.

   Activate only the **Select parts** switch on the **Selecting** toolbar when selecting parts. Otherwise selection could take a long time in large models.
10. Go to **Quick Launch** and type the appropriate drawing creation command: **Create single-part drawing**, **Create assembly drawing**, or **Create cast unit drawing**.
11. Number the model if you are prompted to do so.

Tekla Structures creates the drawings. The created drawings are listed in the **Document manager**. If you already have a drawing with the same type and mark, Tekla Structures will not create a new one.

TIP To automatically open any drawing after the drawing has been created, hold down **Ctrl+Shift** while you create the drawing.

See also

Define automatic drawing settings (page 650)

Single-part, assembly and cast unit drawing properties (page 906)

View properties in drawings (page 912)

3.5 Create AutoDrawings

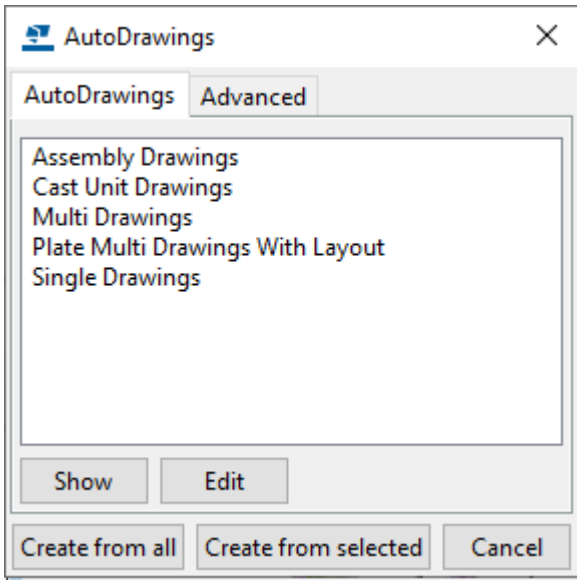
The AutoDrawings wizard combines a series of actions so that you can create drawings using a single command. You can use AutoDrawings to create single-part drawings, assembly drawings, cast-unit drawings and multi-drawings.

AutoDrawings wizards contain instructions telling Tekla Structures which drawing type, selection filter, and drawing properties to use. You can edit AutoDrawings wizard files or create your own ones. The AutoDrawings dialog box lists the available wizard files.

Create AutoDrawings

Before you can create AutoDrawings, the appropriate wizard files must exist and numbering must be up to date.

1. In the model, select the parts from which to create drawings. You can also select the entire model and use filters to fine-tune your selection. You can also use selection filters in the wizard file. If you do, your selection can include parts that will not be included in the drawings.
2. Type `AutoDrawing` in the **Quick Launch** box.
3. In the **AutoDrawings** dialog box, select a wizard file.



- To view the file, click **Show**.
- To edit the file, click **Edit**.

4. On the **Advanced** tab, select whether to create a log file, what information it should include and name the log file.
5. Click **Create from selected**.

AutoDrawings wizard files

You can create your own AutoDrawings wizard files using any standard text editor. Use the existing wizard files as examples to construct your own.

A typical AutoDrawings wizard file contains several sets of drawing requests containing drawing, attribute and part settings to apply to selected objects, as well as a selection filter. The order of sets is important, as Tekla Structures only creates one drawing for each object.

Place the newly created drawing wizard files in the attributes sub-folder in the model folder. The file name extension for wizard files is `.dproc`. You can also place them in firm and project folders. There might also be some predefined wizard files in your environment.

Example

The AutoDrawings wizard file set in this example creates an assembly drawing for an object that fulfills the selection filter criteria in the set. Tekla Structures will not create another assembly drawing for that object, even if it matches the criteria of the selection filter in later sets in the same wizard file.

The wizard file consist of the lines below. Note the use of parentheses.

```
set_drawing_type(assembly)
```

```
set_drawing_attributes(column)
set_template_drawing
set_filter(column_filter)
create_drawings()
```

Line	Description
`set_drawing_type(assembly)`	This line defines the type of drawing the wizard creates. The drawing type appears in parentheses. The options are: `single`: workshop drawings `assembly`: assembly drawings `multi_single`: workshop multi-drawings `multi_single_with_layout`: workshop multi-drawings with layout `multi_assembly`: assembly multi-drawings `multi_assembly_with_layout`: assembly multi-drawings with layout `cast_unit`: cast-unit drawings
`set_drawing_attributes(column)`	This line tells Tekla Structures which drawing properties to use when creating the drawings. The name of the saved drawing properties file appears in parentheses.
`set_template_drawing`	This line tells Tekla Structures to use the defined drawing as a template when creating a new drawing. This line is used instead of the line `set_drawing_attributes`. The path and the name of the drawing template are given in parenthesis like in the following example: `set_template_drawing("C:\TSModels\AngleModel":"[A.2]")`
`set_filter(column_filter)`	This line tells Tekla Structures which selection filter to use to select the parts from which to create drawings. The filter name appears in parentheses.

Line	Description
`create_drawings()`	This line starts the drawing creation. This line should always appear immediately after the lines `set_drawing_type`, `set_drawing_attributes` and `set_filter`.

Wizard log

Tekla Structures writes a log file when you run an AutoDrawings wizard. The log file contains information about errors, number of drawings created, commands used etc.

You can configure whether or not Tekla Structures creates a log file and how it is displayed using the **Advanced** tab in the AutoDrawings dialog box.

Create log options:

- **No**: Tekla Structures does not create a log file.
- **Create**: Tekla Structures creates a new log file and deletes the old one.
- **Append**: Tekla Structures adds a new entry to the existing log file.

Display log options:

- **No**: Tekla Structures does not display the log,
- **With associated viewer**: Tekla Structures displays the log file in an associated viewer (e.g. Notepad) when you run the wizard. You can edit the log file.
- **On dialog**: Tekla Structures displays the log file in a dialog box when you run the wizard. You can not edit the log file.

3.6 Create multidrawings

You can create multidrawings of selected parts and of selected drawings. You can also create empty multidrawings and copy or link views in them from other drawings.

If you create multidrawings from existing drawings, you can choose to include their individual drawing layouts. If you want to have separate lists, tables, and call-offs for each part or assembly, you should include the individual drawing

layouts. You can also include lists and tables for all parts or assemblies in the multidrawing.

Before creating a multidrawing, check the original assembly or single-part drawing, and clean it up, if necessary. Do not modify the multidrawing object that links to the original drawing.

Before you start, you can set to `TRUE` in the **Drawing properties** category of the **Advanced options** dialog box to remove extra space between drawing view labels and drawing views.

Also take a look at the following advanced options related to multidrawings:

XS_MULTI_DRAWING_VIEW_TITLE

XS_MULTIDRAWING_KEEP_OBSOLETE_DRAWINGS

XS_MULTI_DRAWING_VIEW_PLACING_TRIAL_NUMBER

Link or copy drawing views to empty multidrawings

You can create empty multidrawings in which you can link or copy views from other drawings as such or with the original drawing layout.

1. On the **Drawings & reports** tab, click **Multidrawing** --> **Empty drawing** .
2. On the **Drawings & reports** tab, click**Document manager** .
3. Open the empty multidrawing from **Document manager**.
4. Still on the **Document manager**, select the drawings that you want to copy or link to the multidrawing.
5. On the **Views** tab, click **From other drawing** and select one of the copying or linking commands:
   - **Copy**
   - **Copy with layout**
   - **Link**
   - **Link with layout**

   The drawing views are placed in the multidrawing from top to bottom starting from the top-left corner. The views are placed in the same order as they appear in **Document manager**. If you have sorted the list by the name, the drawings are created in the same order.
6. Arrange the views if they are on top of each other.

NOTE When you update multidrawings, the linked drawings are also updated.

Useful links

What does XS_CREATE_CONNECTION_WHEN_COPYING DRAWING_VIEWS do?

Create multidrawings of selected parts

Create single-part and assembly drawings of the parts that you select and place the drawings in a multidrawing. You can also keep the layout of the selected single-part or assembly drawings.

1. Select the parts from the model using appropriate selection switches or filters.
2. On the **Drawings & reports** tab, click **Multidrawing** and select one of the following commands depending on the desired result:
   - **New single-part drawings of selected parts**: Create single-part drawings of the selected parts and place the drawings in a multidrawing
   - **New single-part drawings of selected parts with layout**: Create single-part drawings of the selected part, keeping the layout of each drawing, and place the drawings in a multidrawing.
   - **New assembly drawings of selected parts**: Create assembly drawings of the selected parts and place the drawings in a multidrawing.
   - **New assembly drawings of selected parts with layout**: Create assembly drawings of the selected parts, keeping the layout of each drawing, and place the drawings in a multidrawing.

Create multidrawings of selected drawings

You can create multidrawings of the drawings you select. You can also keep the layout of the selected drawing in the new multidrawing.

1. On the **Drawings & reports** tab, click **Document manager**.
2. Select the drawings you want to add in the multidrawing from **Document manager**.
3. On the **Drawings & reports** tab, click **Multidrawing** and select one of the following commands:
   - **Selected drawings**: Create multidrawings of the selected drawings without the drawing layout.
   - **Selected drawings with layout**: Create multidrawings of the selected drawings, keeping the layout of each selected drawing.

   The drawing views are placed in the multidrawing from top to bottom starting from the top-left corner. The views are placed in the same order as they appear in **Document manager**. If you have sorted the list by the name, the drawings are created in the same order.

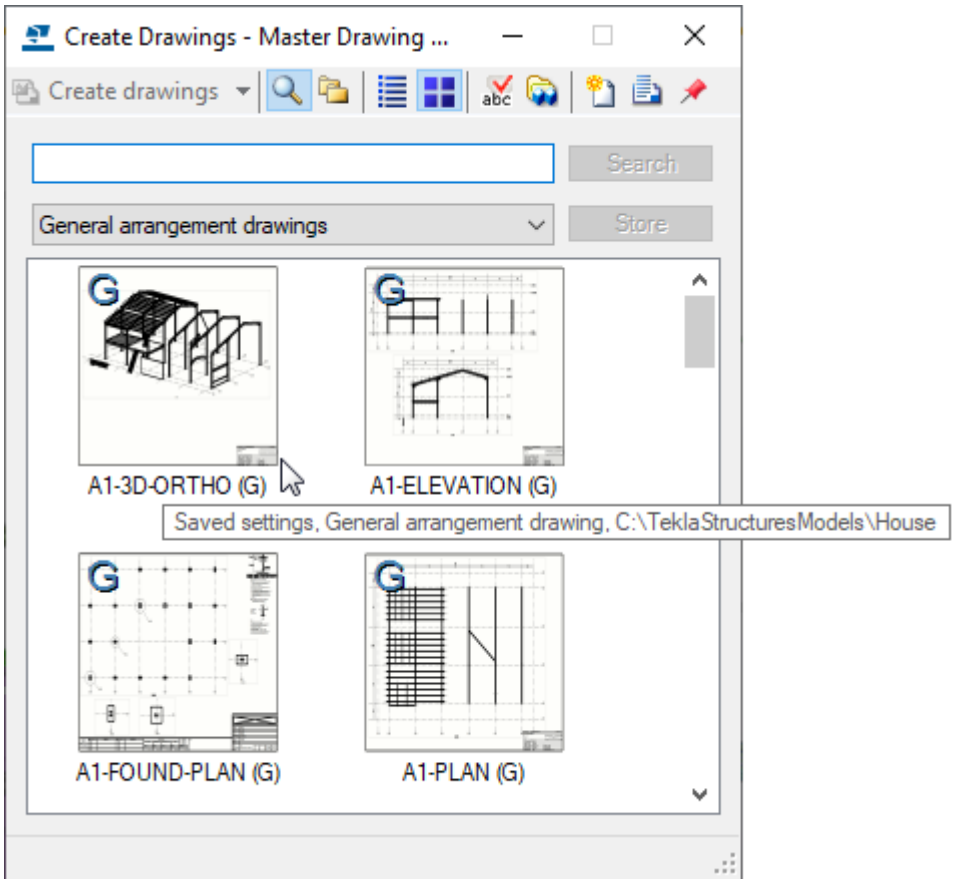
3.7 Create drawings in Master drawing catalog

The **Master drawing catalog** is a fast, efficient and controlled way of creating drawings using master drawings. In the **Master drawing catalog**, all drawing creation commands are available in one centralized location.

A master drawing is a Tekla Structures drawing or a set of drawing properties that is used for creating new drawings that look the same as the master drawing. There are several types of master drawings: cloning templates, saved settings and rule sets. You can also use the existing AutoDrawings wizard files as master drawings.

- To open the **Master drawing catalog**, on the **Drawings & reports** tab, click **Master drawing catalog**.

The image below shows the search view of the **Master drawing catalog** as a thumbnail list.



The **Master drawing catalog** has a toolbar containing commands for creating drawings, selecting the view type and the master drawing list type, displaying master drawing descriptions, selecting the models from which you want to use

cloning templates, creating rule sets, displaying **Document manager**, and for keeping the **Master drawing catalog** window always on top.

Tekla Structures populates the catalog by looking for items in the Tekla Structures default folder search order: first from the firm folder, then from the project folder, and then from the system folder and so on. Cloning templates in the folder defined for the advanced option `XS_CLONING_TEMPLATE_DIRECTORY` are shown in the catalog. You can define several folders for this advanced option.

See also

Master drawing types (page 122)

Create general arrangement drawings using saved settings in Master drawing catalog (page 126)

Create anchor bolt plans using saved settings (page 127)

Create single-part, assembly and cast unit drawings using saved settings in Master drawing catalog (page 130)

Create drawings using rule sets or wizards in Master drawing catalog (page 135)

Clone drawings (page 153)

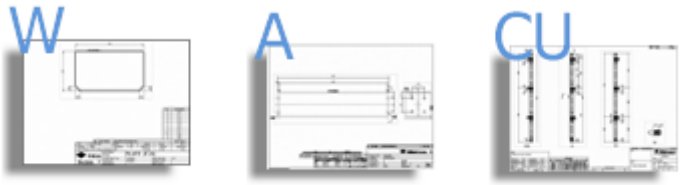
Search for master drawings and save the results in Master drawing catalog (page 141)

Customize Master drawing catalog (page 142)

Master drawing types

The master drawing type you should use depends on the type of the drawing that you want to create:

- You can create single-part drawings, assembly drawings or cast unit drawings using saved settings, rule sets, or cloning templates.



standard (W) standard (A) standard (C)

- You can create general arrangement drawings using saved settings.

standard (G)

- You can create multidrawings using wizards (old file-based rule sets)

Multi Drawings (R)

- For more information about adding different master drawing templates to **Master drawing catalog**, see Customize Master drawing catalog (page 142).

Cloning templates

Cloning templates are Tekla Structures drawings that are used as templates for creating new drawings. You can select a drawing from **Document manager** and add it to the **Master drawing catalog** to be used as a template.

You can also use cloning templates that are located in other models. When you have similar parts in several projects, you can maintain a set of cloning template models and then take the cloning templates in the cloning template models in use when necessary.

For more information about cloning and creating drawings using cloning templates, see the links below:

Clone drawings (page 153)

Create drawings using cloning templates in Master drawing catalog

Clone by using cloning templates located in other models

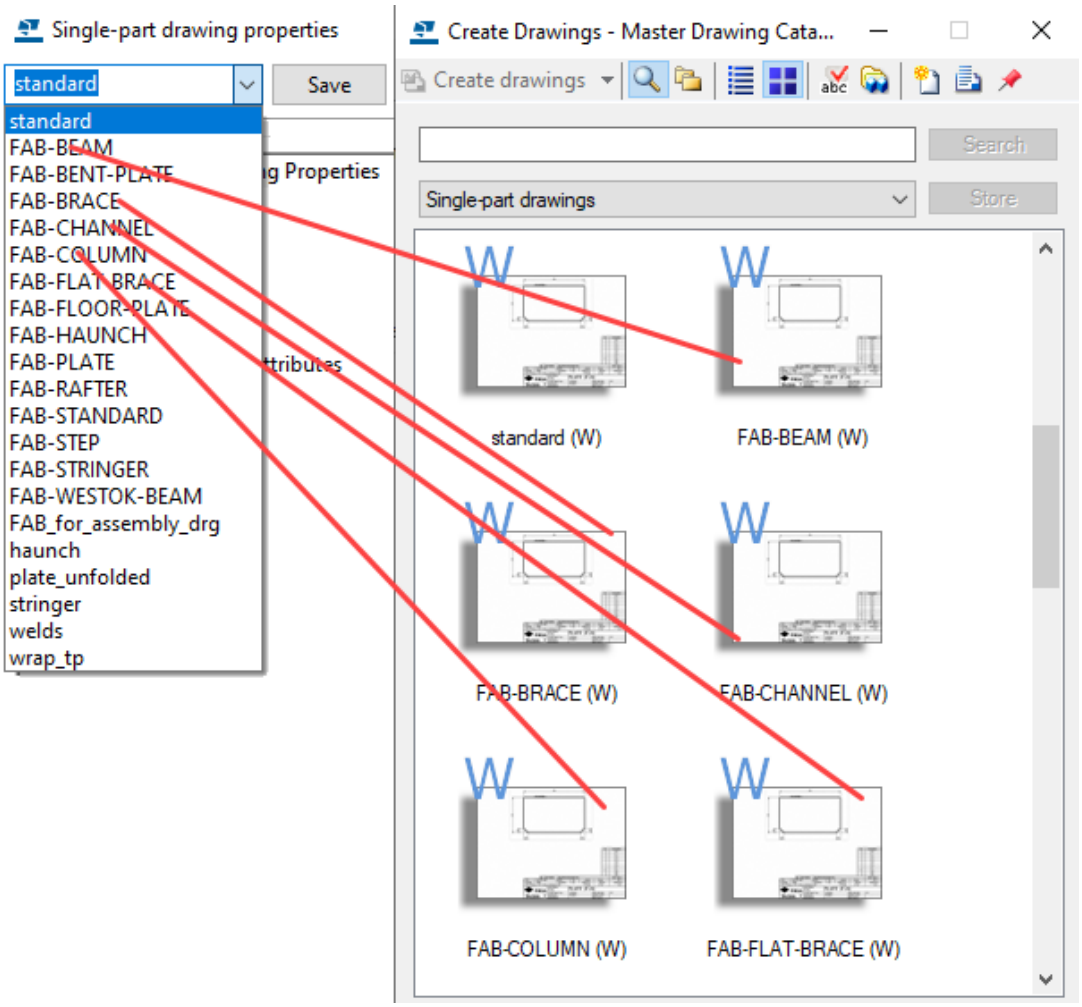
Saved settings

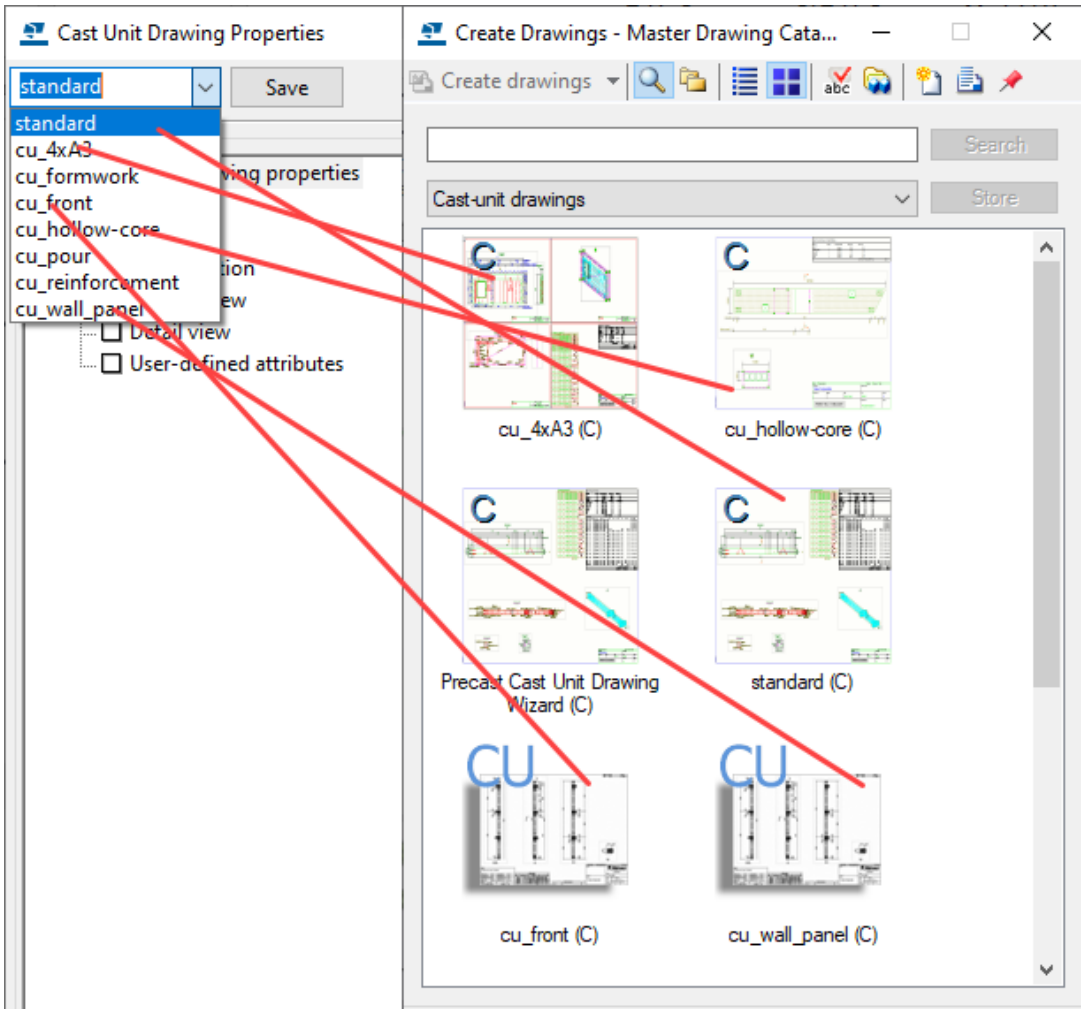
The *saved settings* in **Master drawing catalog** are *drawing property files* that have been created and saved in the drawing properties dialog boxes for different drawing types. There are also many predefined drawing property files.

Each drawing type has its own properties file. Default saved settings are located under the environment folder (`..\Tekla Structures\<version>\environments\`). The exact file location may vary depending on your

environment. When you save your own settings, they are saved under the current model directory.

Below are some examples how the saved settings are shown in the **Master drawing catalog** dialog box.





For more information about creating drawings using saved settings, see the links below:

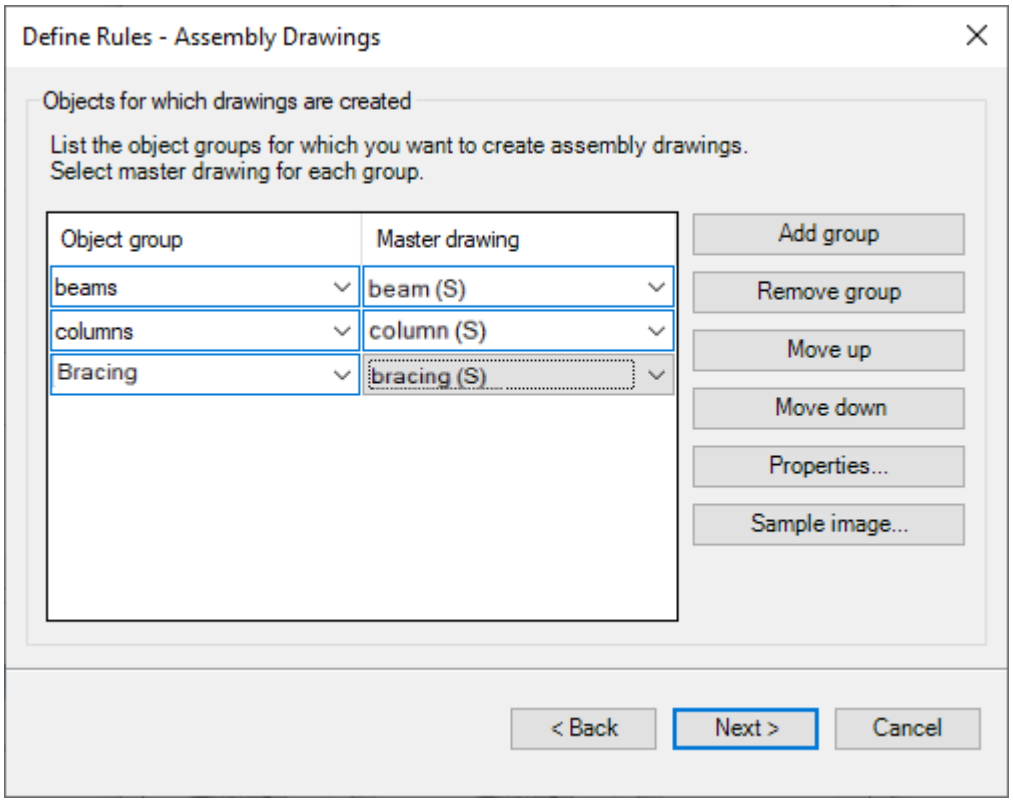
Create general arrangement drawings using saved settings in Master drawing catalog (page 126)

Create single-part, assembly and cast unit drawings using saved settings in Master drawing catalog (page 130)

Rule sets

Rule sets are sets of rules on how to create drawings for different object types. A rule set is a combination of object groups (model selection filters) and master drawing settings (cloning templates, saved settings) that define which objects to include in the drawing, and which drawing settings to use. You can use the existing AutoDrawings wizard files or create your own rule sets.

The order of sets is important, as Tekla Structures by default creates only one drawing for each object. For example, a rule set creates an assembly drawing for an object that fulfills the selection filter criteria in one set. Tekla Structures will not create another assembly drawing for that object, even if it matches the criteria of the selection filter in later sets in the same rule set file.



For more information about creating drawings using rule sets, see Create drawings using rule sets or wizards in Master drawing catalog (page 135).

Wizard files

A *wizard file* is a file-based wizard consisting of several sets of drawing requests containing drawing, attribute and part settings to apply to selected objects, as well as a selection filter. The order of sets is important, as Tekla Structures by default creates only one drawing for each object. You can edit the files in the **Master drawing catalog**. However, creating new wizard files is not possible in the **Master drawing catalog**. Instead, you can create a rule set, which is basically the same thing as a wizard: It applies drawing properties to objects selected by filters, but in a dialog box, not in a text file like a wizard.

NOTE In the **Master drawing catalog**, the only way to create multidrawings is to use the wizard files.

For more information about wizard files, see the following links:

Create drawings using rule sets or wizards in Master drawing catalog (page 135)

Customize Master drawing catalog (page 142)

Create general arrangement drawings using saved settings in Master drawing catalog

You can create GA drawings in **Master drawing catalog** on the basis of saved settings.

Before you create general arrangement drawings, create the model views that you need and ensure that the views are as you want them to be. The drawing views will have the same orientation and content as the model view you select. Fit work area using two points to select the area that you want to show in the general arrangement drawing.

1. On the **Drawings & reports** tab, click **Master drawing catalog**.
2. Select **Saved settings** from the list at the top, and then the desired set of GA saved settings.
3. If needed, modify the drawing properties by double-clicking the saved setting and clicking **Edit drawing properties...**.

   Remember to save the changes in the properties file.
4. Click **Create drawings** (**Alt+C**).
5. In the **Create general arrangement drawing** dialog box, select the views that you want to create and the desired option from the **Options** list.

   If you select **Empty drawing**, you can create an empty drawing and add drawing views in it afterwards.
6. Click **Create** to create the drawings.

Tekla Structures creates the drawings and adds them in **Document manager**.

Click in the **Master drawing catalog** toolbar to open **Document manager**.

For more information about creating cast in place general arrangement drawings, see Show pours in drawings (page 485).

See also

Master drawing types (page 122)

Create general arrangement drawings (page 105)

Create anchor bolt plans using saved settings

Anchor bolt plans are general arrangement drawings showing the anchor bolt layout. You can create anchor bolt plans in **Master Drawing Catalog** or using ribbon or pop-up menu commands.

Tekla Structures selects the objects that are included in the anchor bolt plan based on the following default rules:

- The part is a column or an almost vertical beam.

- The part is the main part of an assembly.
- A base plate is included in the assembly, and it is located lower than the main part. If there is more than one part in the column assembly that fulfills the rules, the lowest part of them is considered to be the base plate.
- Bolts are attached to the base plate.
- The assembly is cut by the view plane.
- The boundary box of the base plate in the xy-direction intersects the boundary box of the column.

Create anchor bolt plans

Below you will create an anchor bolt plan using the **Master drawing catalog**.

Before creating the anchor bolt plan, create a model view that is in the XY plane.

1. On the **Drawings & reports** tab, click **Master drawing catalog**.
2. Double-click a saved settings master drawing that has been created for anchor bolt plans, for example, `anchor_bolt_plan`.
3. Open the **General Arrangement Drawing Properties** dialog box by clicking **Edit drawing properties...**.
4. Click **View**
5. On the **Anchor bolt plan** tab, ensure that the option **Show as anchor bolt plan** is set to **Yes**.

   Define the drawing as an anchor bolt plan when you are creating the drawing. You cannot convert a general arrangement drawing to an anchor bolt plan afterwards.
6. Modify the enlarged part view scale, if necessary.
7. If needed, set the **Create detail views** option to **Yes** to create separate detail views.

   If you select **No**, Tekla Structures dimensions the anchor bolts in the enlarged view. Tekla Structures groups similar detail views so that similar details are drawn only once.
8. Set the **Detail view scale**.
9. Click **Save** to save the changes in the saved settings file.
10. Click **OK** to close the **Master Drawing Properties** dialog box.
11. Click **Create drawings**.
12. Select one view in the xy-plane.

    If you select some other type of view or several views, Tekla Structures displays a warning message and the anchor bolt plan is not created.

13. Click **Create**.

    Tekla Structures creates the anchor bolt plan.

    Tekla Structures creates bolt dimensions in the anchor bolt plan in the directions of the bolt group coordinate system. If the angle between the bolt group and the view coordinate systems is not orthogonal, Tekla Structures adds angle dimensions to the detail view. Use the advanced option XS_ANCHOR_BOLT_PLAN_USE_VIEW_COORDSYS_FOR_BOLT_DIMENSIONS to create the dimensions in the X and Y directions of the main view.

When details are compared, the following aspects are taken into account:

- bolt dimensions
- column profile
- column orientation (coordinate system)
- plate profile

TIP In the created drawing, go to the **File** menu, click **Settings** and check that **Ghost outline** is selected and **Drawing Color Mode** is set to **Color**. In color drawings with white background, hidden objects are shown as ghost outlines, if this setting is selected. In gray scale and black and white drawings, hidden objects are not shown even if **Ghost outline** is selected.

Define anchor bolt plan parts using drawing filters

1. Create the necessary general arrangement drawing filters in the **Filter Properties** subdialog box of the **General Arrangement Drawing Properties** dialog box.
2. On the **File** menu, click **Settings** --> **Advanced options** and go to the **Drawing properties** category.
3. Enter the following values for the following advanced options:
   - XS_ANCHOR_BOLT_PLAN_COLUMN_FILTER: `<the name of the drawing filter for columns>`
   - XS_ANCHOR_BOLT_PLAN_BOLT_FILTER: `<the name of the drawing filter for bolts>`
   - XS_ANCHOR_BOLT_PLAN_BASEPLATE_FILTER: `<the name of the drawing filter for base plates>`
   - XS_ANCHOR_BOLT_PLAN_ADDITIONAL_PARTS_FILTER: `<the name of the drawing filter for additional parts>`

If you do not use any of these advanced options, the default rules are used for determining the columns, base plates, and anchor bolts to be included in the anchor bolt plan.

NOTE Using these advanced options only affects the creation of new anchor bolt plan drawings, and has no affect on the existing ones.

Include assemblies in anchor bolt plans

If you have anchor bolts at varying elevations, you can create a general arrangement drawing at the level of the top-most base plate/anchor bolt.

The anchor bolt plan view extrema looks downwards. If the lowest level anchor bolt is not shown, adjust the advanced option `XS_ANCHOR_BOLT_PLAN_DRAWING_TOLERANCE`.

To specify the tolerance:

1. On the **File** menu, click **Settings** --> **Advanced options** --> **Drawing properties** .
2. Set a value for the advanced option XS_ANCHOR_BOLT_PLAN_DRAWING_TOLERANCE.

   Define the distance in millimeters. The default value is 200 mm.

Create single-part, assembly and cast unit drawings using saved settings in Master drawing catalog

You can create single-part, cast unit and assembly drawings in **Master Drawing Catalog** using saved settings.

NOTE • Cast units have the cast unit type part property setting in the model - they are set either **Cast in place** or **Precast**. It is important to use the correct cast unit type, because some features, for example numbering, are based on the cast unit type. In the Cast in Place configuration, you can create cast unit drawings of cast-in-place cast units only.

Create single-part, assembly and cast unit drawings

1. On the **Drawings & reports** tab, click **Master drawing catalog**.
2. Select the desired set of saved settings (drawing properties file) among the available saved settings.

   For cast unit drawings, select a file that contains the desired **Cast unit definition method**:

   - **By cast unit position**: A drawing will be created from each cast unit. If there are identical cast units, one of them will serve as the base cast unit for the drawing. This is the most common method for creating cast unit drawings.

- **By cast unit ID**: Each part in the model has a unique GUID. You can create drawings by using cast unit GUIDs. The GUID determines the marking of the drawing. You can create several drawings from identical cast units.

3. If needed, modify the drawing properties and apply object-level settings. To do this, double-click the saved setting and click **Edit drawing properties...**. Remember to save your changes in the property file.
4. If you are creating a drawing only for selected objects, select the objects.

   You can also use an appropriate selection filter and select the whole model. Activate only the **Select parts** selection switch when you select objects, otherwise the selection may take a long time.
5. Click **Create drawings** (**Alt+C**) or **Create drawings for all parts** (**Alt+A**).
6. Number the model if you are asked to do so.

Tekla Structures creates the drawings and adds them in **Document manager**.

Click the **Open Document manager** button on the **Master drawing catalog** toolbar to open **Document manager**.

NOTE If you already have a drawing with the same type and mark, Tekla Structures will not create a new one.

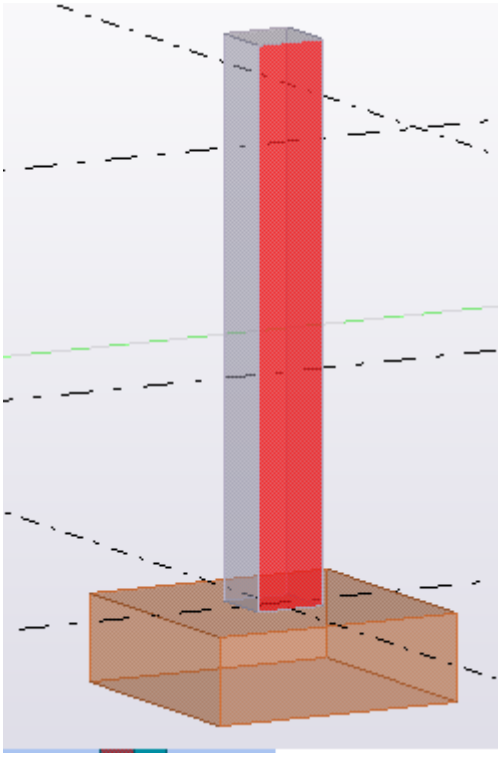
Example: Create cast unit drawings one by one

In this example, you will create a drawing of a cast unit using saved settings called **Column_with_BOM**.

1. In the model, right-click the concrete column and select **Cast unit** --> **Set top in form face** , and select the cast unit face that will face upwards in the form.

   Top-in-form face defines which side is pointing upwards in the front view of cast unit drawing. The **Coordinate System** of the cast unit drawing has to be **Fixed**.

   When you right-click the column and select **Cast unit** --> **Show top in form face** , the selected face is shown red, see the image below.

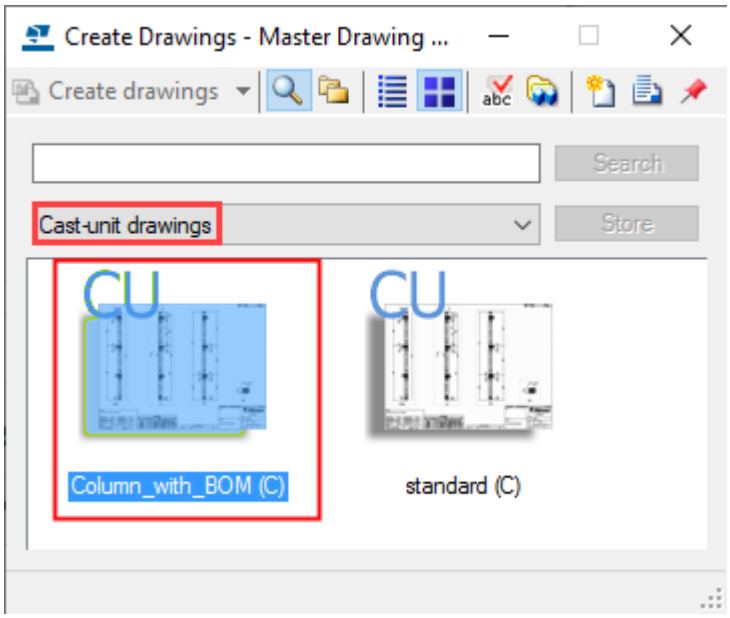


2. Still in the model, double-click the column and select **Precast** as the **Cast unit type**.

   It is important to use the correct cast unit type, because some features, for example, numbering, are based on the cast unit type.

3. On the **Drawings & reports** tab, click **Drawing properties** --> **Cast unit drawing** .
4. In the properties dialog box, load the appropriate predefined drawing properties (saved settings). In this example, **Column_with_BOM** is loaded.
5. Ensure that the **Column_with_BOM** settings contain the desired **Cast unit definition method**:
   - **By cast unit position**: A drawing will be created from each cast unit. If there are identical cast units, one of them will serve as the base cast unit for the drawing. This is the most common method for creating cast unit drawings.
   - **By cast unit ID**: Each part in the model has a unique GUID. You can create drawings by using cast unit GUIDs. The GUID determines the marking of the drawing. You can create several drawings from identical cast units.
6. Ensure that **Coordinate System** of the cast unit drawing is set to **Fixed**: Click **View creation** and go to the **Attributes** tab.
7. If you made any changes in **Column_with_BOM**, save the changes and close the dialog box.

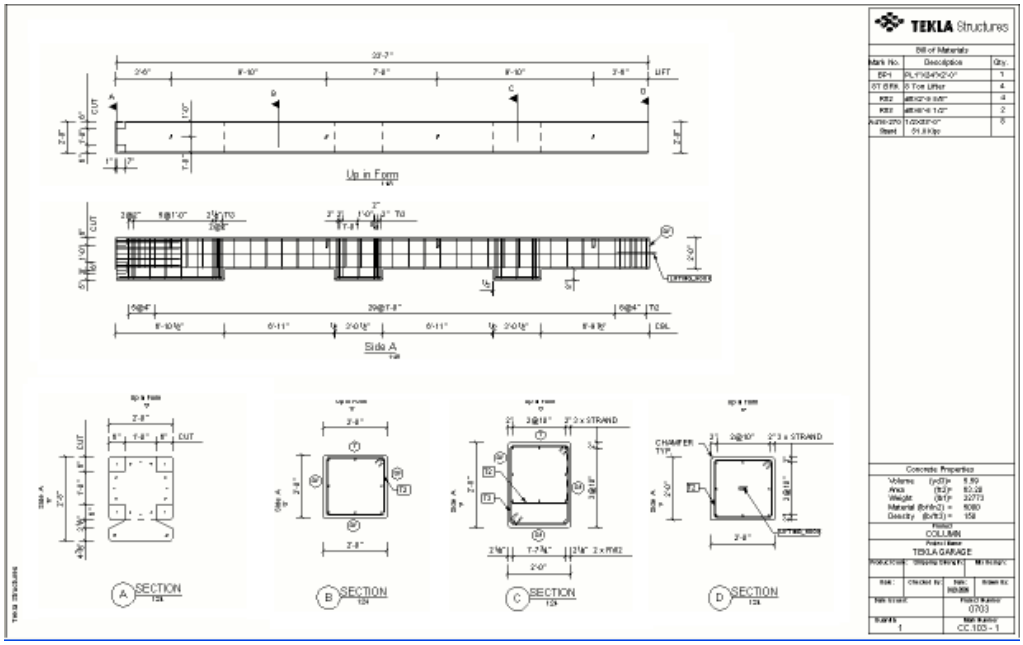
8. Select the column in the model.
9. On the **Drawings & reports** tab, click **Master drawing catalog**.
10. Set the type to **Cast-unit drawings** and select the saved settings **Column_with_BOM (C)**.



11. Click **Create drawings**.

    Tekla Structures creates the drawing.

    You can open the **Document manager** by clicking the **Open drawing list** button on the toolbar and then open the drawing to view it.



Example: Create assembly drawings from groups of similar parts

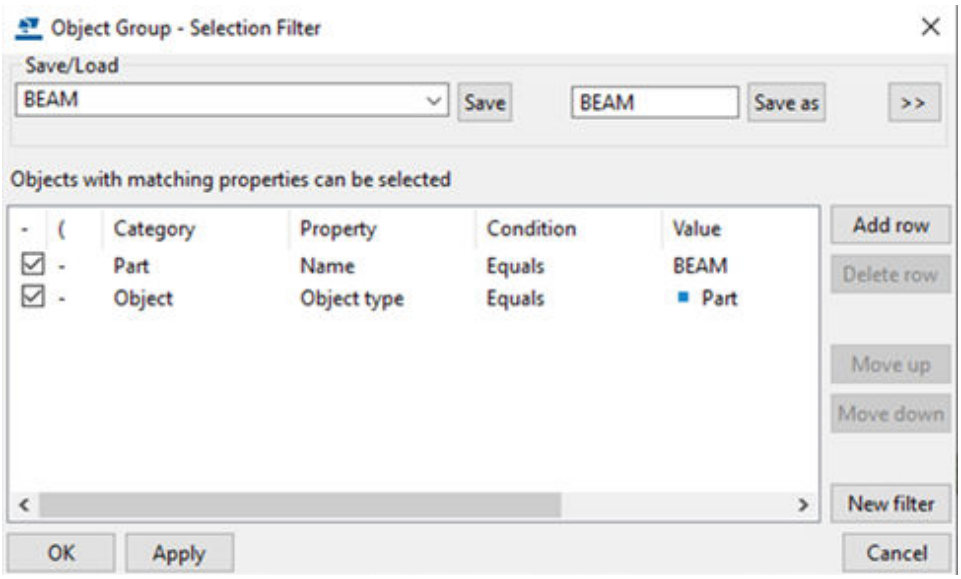
Creating drawings one by one may take a lot of time. You can automate the creation process a bit more by selecting groups of items and then create the drawings. For example, you can define the needed properties for the beams and then run all the beam drawings at once by using the Tekla Structures model selection filters.

In this example, you are going to use saved settings named **Beam_with_BOM** and create assembly drawings from all beams.

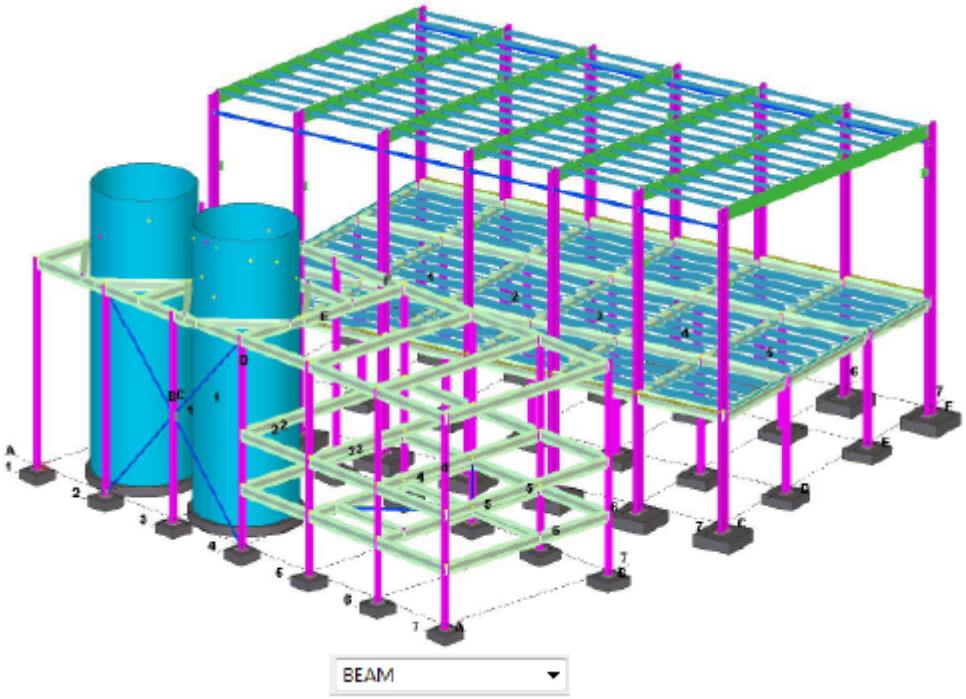
1. Define a selection filter that selects only one type of items, in this case beams.

   a. Click the selection filter button on the **Selecting** toolbar or press **Ctrl+G** to open the **Object Group - Selection Filter** dialog box.

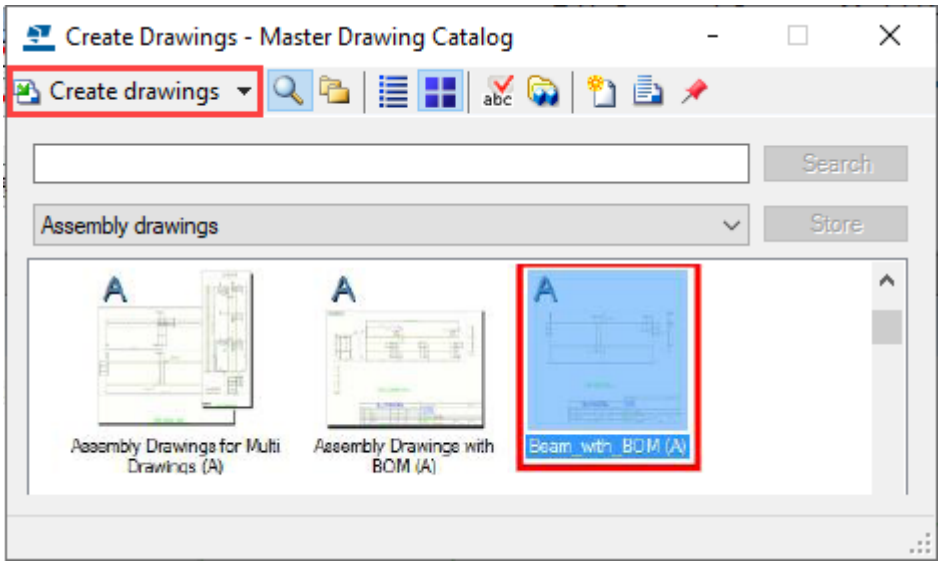
   b. Create a selection filter that selects all parts named BEAM and save it using the name **BEAM**.



2. Activate the **BEAM** filter from the list of selection filters and use area selection to select all beams in the model.



3. Open the **Master drawing catalog**: On the **Drawings & reports** tab, click **Master drawing catalog**.
4. Under **Assembly drawings**, select the saved settings master drawing. In this example, **Beam_with_BOM (A)** is selected.
5. Click **Create drawings**.



Tekla Structures creates the beam assembly drawings.

Create drawings using rule sets or wizards in Master drawing catalog

In the **Master drawing catalog**, you can create single-part, assembly and cast unit drawings using predefined rule sets, and multidrawings using predefined wizards. You can also create your own rule sets.

Limitation: In the **Master drawing catalog**, you can create multidrawings only by using file-based wizards and general arrangement drawings using saved settings. You cannot create rule sets for multidrawings or general arrangement drawings.

For more information about adding master drawings to **Master drawing catalog**, see Customize Master drawing catalog (page 142).

Create drawings using a rule set or wizard

1. On the **Drawings & reports** tab, click **Master drawing catalog** .
2. Select a rule set or a wizard file from the list.
3. If needed, modify rule set properties or wizard file contents by double-clicking the rule set or wizard.
4. If needed, edit the drawing properties of the master drawing connected to the rule set. To do this, double-click the rule set, select the master drawing from the list and click **Properties...** --> **Edit drawing properties...** . Remember to save your changes.
5. Select the whole model or desired parts.
6. Do one of the following:
   - If you selected parts, click **Create drawings** (**Alt+C**).
   - If you selected the whole model, click **Create drawings for all parts** (**Alt+A**).
7. Number the model if you are asked to do so.

Tekla Structures creates the drawings and adds them in the **Document manager**. Click the **Document manager** button on the **Master drawing catalog** toolbar to open the **Document manager**.

NOTE If you already have a drawing with the same type and mark, Tekla Structures will not create a new one.

Example - Create a new rule set and drawings for all parts

The following example will go through creating a basic rule set that will create assembly and single part drawings based on the names of the parts in the model.

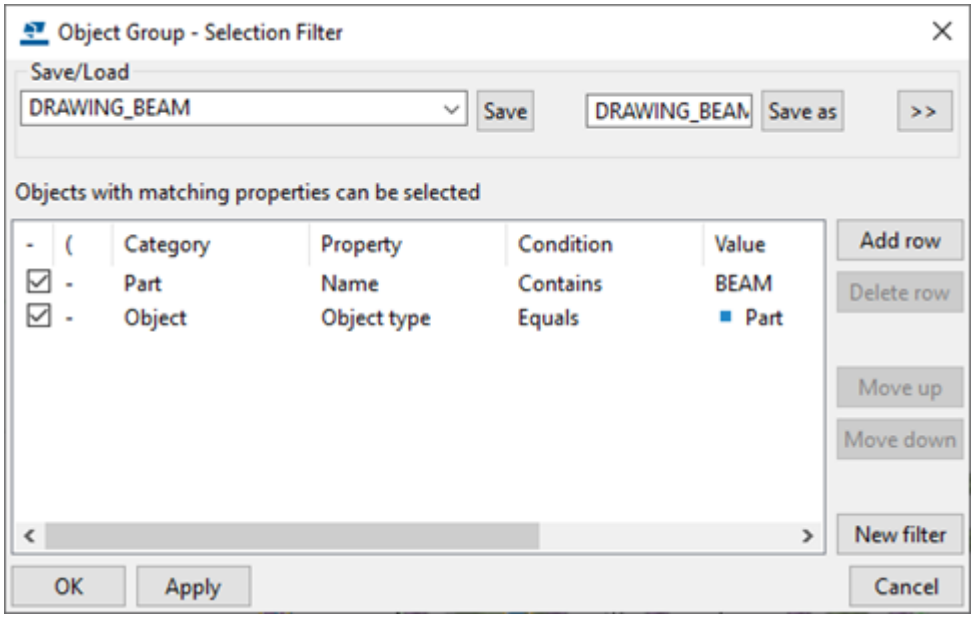
The model used here is a very simple model consisting of two columns with a beam in between, base plates and end plates. The parts are named **BEAM**, **COLUMN** and **PLATE**.

Before setting up a rule set in the **Master drawing catalog**, you need to have some saved settings (drawing properties) and some selection filters that link the drawing properties to the parts in the model.

Create selection filters

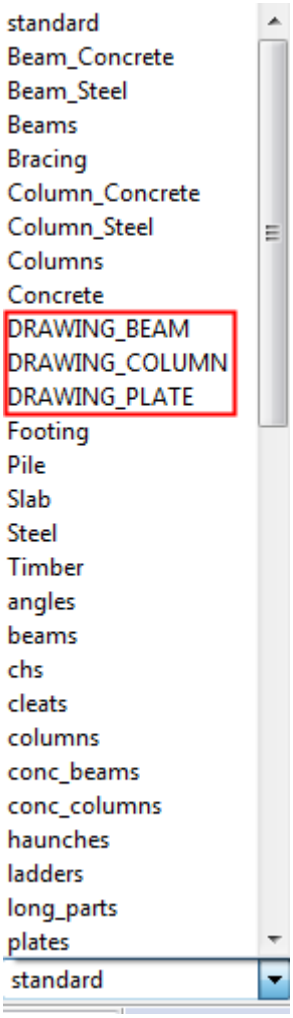
You need to have filters that select the beams, columns and plates.

1. Click the selection filter button on the **Selecting** toolbar or press **Ctrl +G** to open the **Object Group - Selection Filter** dialog box.
2. Click **Add row** and add a selection filter for the beams as shown below, saving it as **DRAWING_BEAM**.



3. In the same way, create selection filters for columns and plates, saving them as **DRAWING_COLUMN** and **DRAWING_PLATE**.

   The new filters are displayed in the selection filter list.



4. Close the **Object Group - Selection Filter** dialog box by clicking **Cancel**.

The new filters will be saved in the model folder. If necessary, copy them to the necessary folders, for example, to the firm or project folders.

Create saved settings (drawing properties)

Next you need to modify the drawing properties and save them so that they are displayed as saved settings in the **Master drawing catalog**. The saved settings are created separately for single-part drawings and assembly drawings.

1. On the **Drawings & reports** tab, click **Drawing properties** --> **Assembly drawing** and create properties for a beam and a column drawing.
2. Give a descriptive name for the drawing in the **Name** box, for example, **BEAM** for the beam properties and **COLUMN** for the column properties.
3. Save the properties as **DRAWING_BEAM** and **DRAWING_COLUMN**.

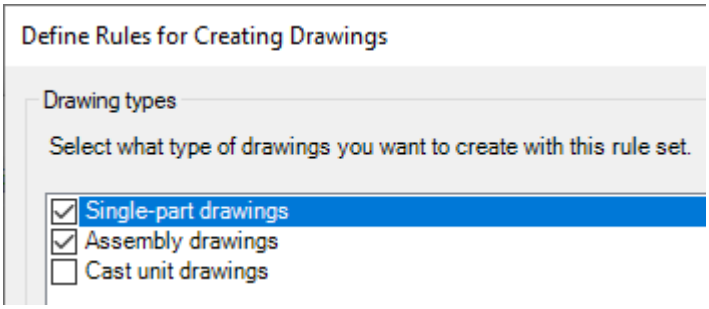
4. On the **Drawings & reports** tab, click **Drawing properties** --> **Single-part drawing** and create some single-part drawing settings for a beam shaft, column shaft and plates.
5. Give a descriptive name for the drawing in the **Name** box, for example, **BEAM SHAFT** for the beam shaft properties, **COLUMN SHAFT** for the column shaft properties and **PLATE** for the plate properties.
6. Save the settings as **DRAWING_BEAM_SHAFT**, **DRAWING_PLATES** and **DRAWING_COLUMN_SHAFT**.

Create a rule set

Now that you have created the necessary selection filters and saved settings, you can create a rule set that automatically creates single-part and assembly drawings for beams, columns and plates.

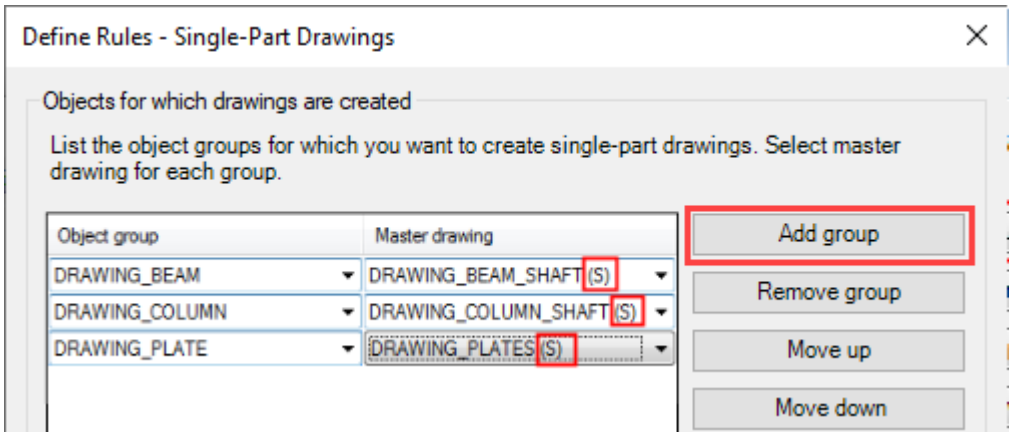
1. On the **Drawings & reports** tab, click **Master drawing catalog**.
2. Click the **Define rule set** button on the toolbar.
3. In the **Define Rules for Creating Drawings** dialog box, select the drawing types you want to create with the rule set and click **Next**.

   In this example, select **Single-part drawings** and **Assembly drawings**.

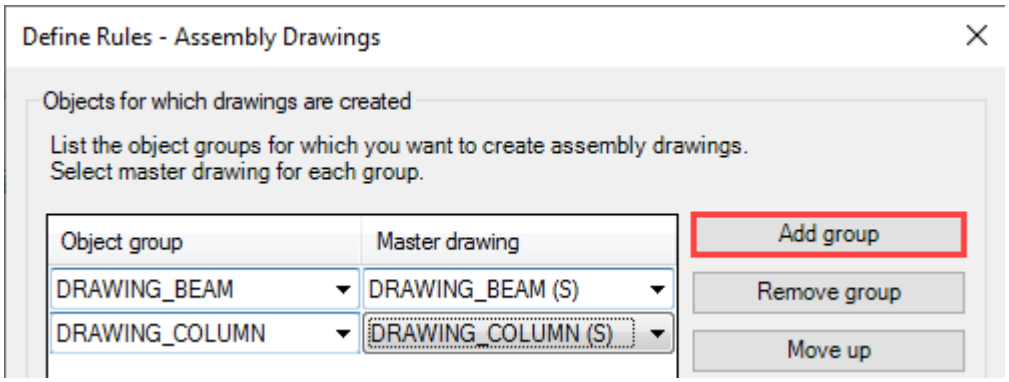


4. In the **Define Rules - Single-Part Drawings** dialog box, click **Add group** and select the filters and the corresponding single-part drawing saved settings.
5. Under **Object group** select the three new filters, and under **Master drawing**, select the three new saved settings.

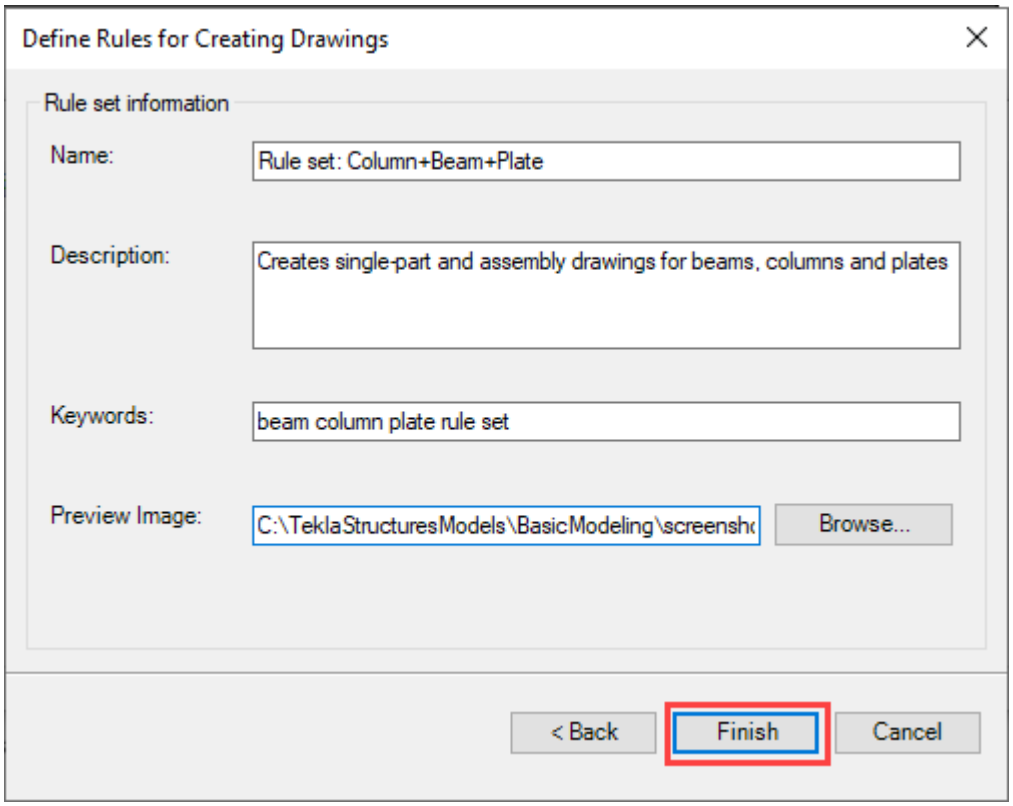
   The (S) after the master drawing name indicates that its type is saved setting.



6. Click **Next**.
7. In the **Define Rules - Assembly Drawings** dialog box, **Add group** and select the filters and the corresponding assembly drawing saved settings.
8. Under **Object group** select the two new filters, and under **Master drawing**, select the two new saved settings.



9. Click **Next**.
10. Give the rule set a name (**Rule Set: Column+Beam+Plate**), description and keywords.
11. Use **Browse** next to **Preview Image** and add a sample image for the master drawing.
12. Click **Finish** when you are done.



Create all drawings using the rule set

Now you can create drawings using the new rule set.

1. In the **Master drawing catalog Search** view, enter your search criteria in the box next to the **Search** button and click the button.

   In this example, `column beam plate` is entered. Tekla Structures finds the newly created rule set.

2. Right-click the rule set and select **Create drawings for all parts**.

   Tekla Structures starts creating the drawings. When the drawings have been created, a message is displayed indicating the number of created drawings, in this case six drawings were created.

3. To check which drawings have been created, click the **Open drawing list** toolbar icon to open the **Document manager**.

13.03.2017	00.00.0000	410* 287	A	[B.1]	BEAM
13.03.2017	00.00.0000	410* 287	A	[C.1]	COLUMN
13.03.2017	00.00.0000	410* 287	W	[F.1]	PLATE
13.03.2017	00.00.0000	287* 210	W	[F.2]	PLATE
13.03.2017	00.00.0000	410* 287	W	[M.1]	COLUMN SHAFT
13.03.2017	00.00.0000	287* 210	W	[M.2]	BEAM SHAFT

Search for master drawings and save the results in Master drawing catalog

In the **Master drawing catalog** search view, you can search for master drawings on the basis of the text that has been entered in the name, description and keyword boxes in the **Master Drawing Properties** dialog box and save your search results. In the folder view, you can copy master drawings from a search results folder to another folder to make it easier for you to locate the master drawings you need.

TIP Make sure to always add all important information for each master drawing in the **Master Drawing Properties** dialog box. This makes it a lot easier to find the master drawings you are looking for.

To search for master drawings and save the search results:

1. Enter the search criteria in the box next to the **Search** button.

   Tekla Structures remembers search strings. When you start typing a string, it suggests previous strings starting with the same characters.

2. Click **Search**. The results are listed.

3. To save the search results, click **Store**, type a name for the search, and click **OK**.

   The search results are added in the category list in the search view.

   The search results are also displayed as a folder in the folder view. There you can rename the saved search results, and copy the found drawings to the folders of your choice.

See also

Create drawings in Master drawing catalog (page 121)

Customize Master drawing catalog (page 142)

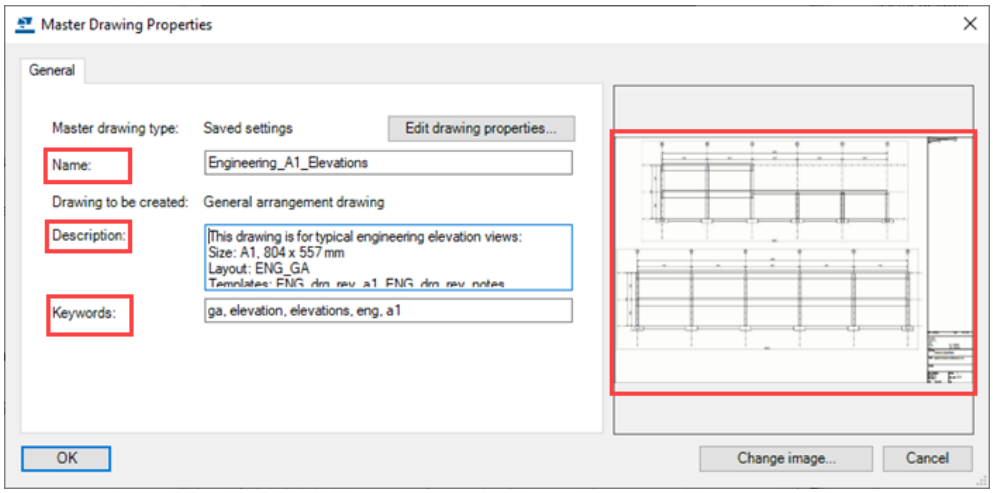
Customize Master drawing catalog

You can customize the **Master drawing catalog** according to your needs. Keeping the **Master drawing catalog** up to date and well organized is essential to create drawings fast and efficiently. Having useful master drawings saves time and effort.

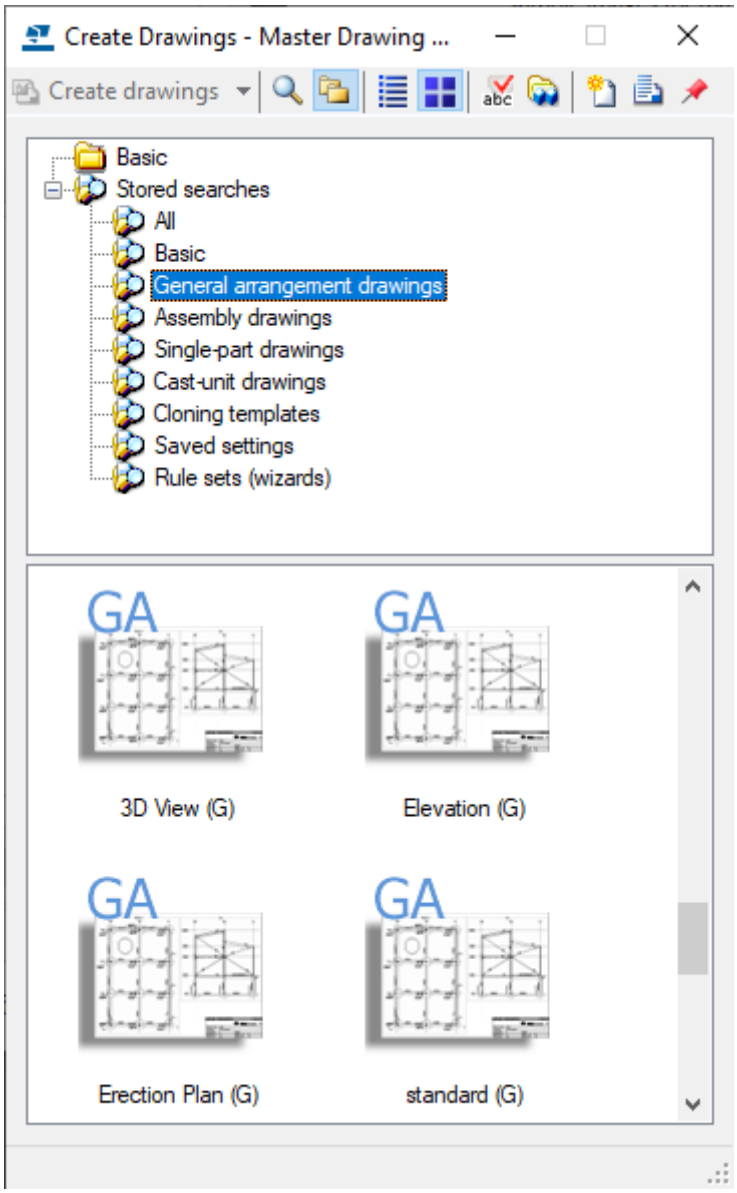
Depending on your environment, the **Master drawing catalog** usually contains some master drawings when you start using it, at least the

predefined saved settings (drawing property files) are there, and maybe some wizards.

TIP Make sure to always add all important information for each master drawing in the **Master Drawing Properties** dialog box. This makes it a lot easier to find the master drawings you are looking for.

- Add only the necessary master drawings (saved settings, rule sets and cloning templates), remove obsolete ones, organize the master drawings in folders, add proper sample images, and keep the properties, descriptions and keywords up to date.
- Each master drawing in the **Master drawing catalog** has its own properties:
  - You can search for master drawings based on the master drawing name, description and keywords that you add.
  - You can add a sample image for each master drawing (**Change image...** button).
  - The sample image is also shown as a thumbnail in the thumbnail view of the **Master drawing catalog**.
  - You can access drawing properties (**Edit drawing properties...** button) and you can, for example, apply object level settings.
  - 

- You can manage **Master drawing catalog** in the following ways:
  - Add new, rename, and move folders.
  - Copy master drawings to another folder.
  - Delete master drawings from folders.



NOTE Deleting master drawings from a folder or copying them to another folder in the folder view does not affect the catalog contents in any way. The folder view is just another way to help you to keep you master drawings well organized.

Add a rule set master drawing in Master drawing catalog

1. On the **Drawings & reports** tab, click **Master drawing catalog**.
2. Click the **Add rule set** toolbar button .

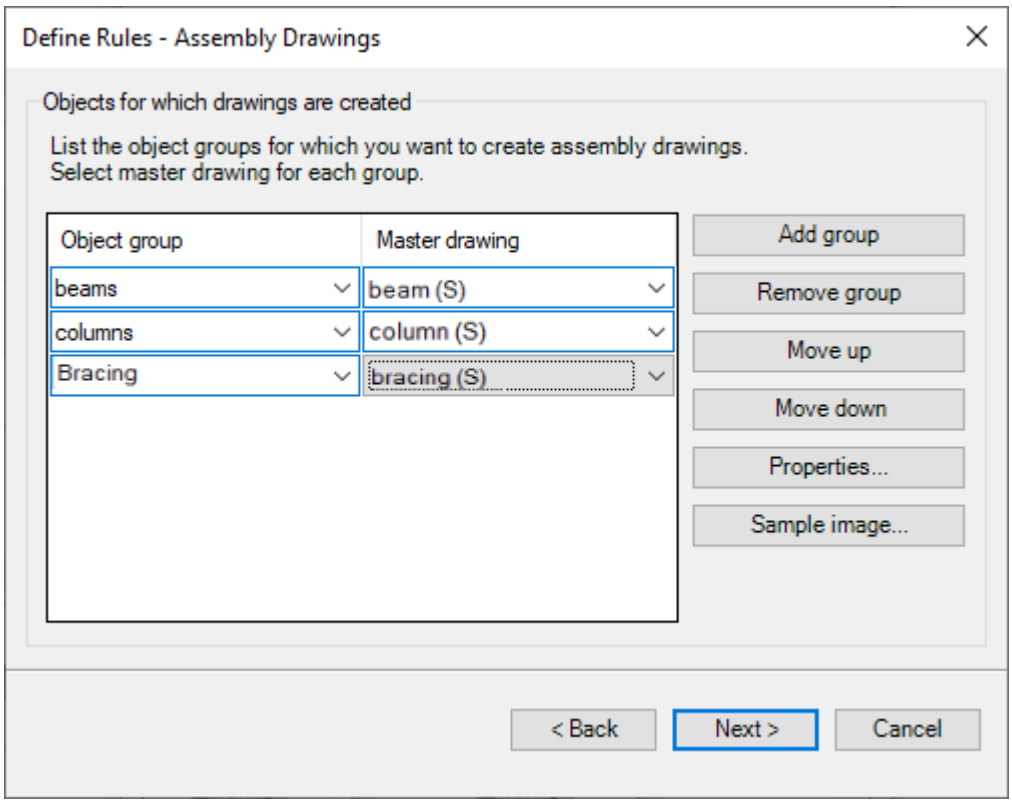
3. In the **Define Rules for Creating Drawings** dialog box, select the drawing types you want to create with the rule set.

   You can select several drawing types with the same rule set. For example, you can use the same rule set to create single-part drawings and assembly drawings.

4. Click **Next**.

5. In the next dialog box, click **Add group** and select the object groups for which you want to create drawings, and a master drawing for each group.

   Only those master drawing types are listed that can be used for the drawing types that you selected in the previous dialog box. Saved settings are marked with an S and cloning templates with T.



   You can use the buttons in the dialog box to do the following:

   - Use **Move up** and **Move down** to move a rule upwards and downwards in the list.

     If a rule that is higher in the list creates a drawing of an object, another rule lower in the list that would create a drawing of the same object will not do it again.

   - Use **Properties** to view or modify the properties of a master drawing that you have selected in the list.

   - Use **Preview** to display a sample image of the selected master drawing.

6. Click **Next**.
7. If you have selected several drawing types, repeat steps 4 and 5 for all drawing types that you have selected.
8. Modify master drawing properties of the rule set: give a name to the rule set, add a sample image, and enter a description and keywords.
9. Click **Finish**.

   The rule set is added in the **Master drawing catalog**.

Add a saved settings master drawing in Master drawing catalog

All predefined drawing property files are shown as saved settings in the **Master drawing catalog**. When you save a new drawing property file, Tekla Structures shows it in the **Master drawing catalog** automatically.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Modify the drawing properties in the **Single-part Drawing Properties**, **Cast Unit Drawing Properties**, **Assembly Drawing Properties** or **General Arrangement Drawing Properties** dialog box.
3. Load drawing properties that are as close to the ones you need as possible.
4. Modify the properties.
5. Give a name to the property file and save the file.

   The new property file is automatically added to the **Master drawing catalog**. If it is not displayed in the list, press **F5** to refresh the view.
6. Modify master drawing properties: give the master drawing a name, add a sample image, and enter a description and keywords.

Modify saved settings properties

You can modify the name, description and keywords of saved settings in **Master drawing catalog**, and change its sample image.

1. In the **Master drawing catalog**, double-click a master drawing of the desired type.
2. Modify the name of the drawing in the **Name** box.
3. Enter or modify the description of the master drawing. You can search master drawings on the basis of data that you enter here.
4. Enter or modify keywords for the master drawing. Use spaces between words. You can search master drawings on the basis of data that you enter here.

5. To add or change the sample image, click **Change image** and select the image. You can use `.bmp`, `.jpg`, `.jpeg`, or `.png` images. We recommend that you use `.png` images.
6. If needed, modify the drawing properties of the master drawing by clicking **Edit drawing properties...** to display the drawing properties dialog box. This also allows you to apply detailed object level settings.

   Save the modified properties. Ensure that you have the correct drawing property file name at the top.
7. Click **OK**.

Apply detailed object level settings in saved settings

You can apply drawing object level settings in the drawings that you create in **Master drawing catalog** for the saved settings type of master drawings. This way you can use the same saved settings for many drawings and only apply some specific object level settings at the point when you create the drawing, for example, use different kind of marks or change the reinforcement color.

To apply detailed object level settings in saved settings for a general arrangement drawing:

1. In the **Master drawing catalog**, double-click the saved setting you want to use for creating drawings.
2. In the **Master Drawing Properties** dialog box, click **Edit drawing properties...**
3. Click the toggle button at the bottom to empty the check boxes.
4. In the **Drawing Properties** dialog box, select only the **Use detailed object level settings** check box and ensure that the **Yes** button next to it is selected.
5. Click **Edit settings...**.
6. Select the filter to be used, the drawing object type and the drawing object property file to be used.
7. Click **OK**.
8. Click **Save** on the top of the dialog box to save the changes in the saved settings file (property file).
9. Click **OK** in the **Master Drawing Properties** dialog box.
10. Create the drawings.

Add a cloning template master drawing in Master drawing catalog

You can add drawings as cloning templates to the **Master drawing catalog** from **Document manager**.

Before you add a cloning template in the **Master drawing catalog**, create a drawing with the desired properties and contents and save it.

Limitations: You cannot add general arrangement drawings or multidrawings to the **Master drawing catalog** from the **Document manager**. You can clone general arrangement drawings using the **Clone** command in the **Document manager** dialog box.

1. On the **Drawings & reports** tab, click **Document manager**.
2. Select the drawing, right-click and select **Add to Master Drawing Catalog** (**Ctrl+M**).
3. Modify the master drawing properties of the cloning template. On the **General** tab, enter a name for the cloning template, and a short description and keywords.
4. Tekla Structures adds a preliminary thumbnail image for the cloning template. If necessary, click **Change image...**and browse for a new image. The same image will then be displayed when you right-click the thumbnail and select **Preview**.
5. Go to the **Drawing creation** tab and specify how dimensions and marks are created, and select the objects to be cloned from the master drawing.
6. Click **OK**.

   The new cloning template is added in the **Master drawing catalog**. The icon is displayed in the **Master** column of the **Document manager** next to the drawing you added to the **Master drawing catalog**.

Modifying cloning template properties

You can modify the name, description and keywords of cloning templates in **Master Drawing Catalog**, and change its sample image.

1. In the **Master drawing catalog**, double-click a master drawing of the desired type.
2. Modify the name of the drawing in the **Name** box.
3. Enter or modify the description of the master drawing.

   You can search for master drawings based on the data you enter here.
4. Enter of modify keywords of the master drawing.

   Use spaces between words. You can search for master drawings based on the data you enter here.
5. To add or change the sample image, click **Change image...** and select the image.

You can use `.bmp`, `.jpg`, `.jpeg`, or `.png` images. We recommend that you use `.png` images.

The sample image will also be used as the thumbnail image of the master drawing in the thumbnail view of the **Master drawing catalog**.

6. Go to the **Drawing creation** tab.
   a. For the **Dimensions**, **Marks**, and **Marks for welds in model**, define whether you want to **Clone** them, recreate them automatically when you clone a drawing (**Create**) or **Ignore** them in cloning.
   b. Select the objects to be cloned from the master drawing: **Drawing welds**, **Level marks**, **DWG/DXF**, **Text files**, **Texts**, **Symbols**, **Graphics**, and **Hyperlinks**).
7. Click **OK**.

Modify properties and contents of wizard files

You can modify wizard files in the **Master drawing catalog** using a text editor. You can change the name, description, and keywords, and add a sample image. The wizard files are by default located in your environment, but there might also be some wizard files in the firm or project folders.

1. In the **Master drawing catalog**, double-click a wizard file.
2. Modify the name of the drawing in the **Name** box. The file name extension for wizard files is `.dproc`.
3. Enter or modify the description of the master drawing.

   You can search for master drawings based on the data you enter here.
4. Enter of modify keywords of the master drawing.

   Use spaces between words. You can search for master drawings based on the data you enter here.
5. To add or change the sample image, click **Change image...** and select the image.

   You can use `.bmp`, `.jpg`, `.jpeg`, or `.png` images. We recommend that you use `.png` images.

   The sample image will also be used as the thumbnail image of the master drawing in the thumbnail view of the **Master drawing catalog**.
6. To edit the contents of the wizard file, click **Edit drawing properties...**

   The wizard file is opened in a text editor. Modify the contents of the file, and save and close it.
7. Click **OK**.

Wizard file contents

Wizard files consist of the following entries. Notice the use of parentheses.

```
set_drawing_type(assembly)
set_drawing_attributes(column)
set_template_drawing
set_filter(column_filter)
create_drawings()
```

Description of the entries:

`set_drawing_type(assembly)`

This line defines the type of drawing the wizard creates. The drawing type appears in parentheses. The options are:

Option	Creates
single	single-part drawings
assembly	assembly drawings
multi_single	single-part multidrawings
multi_single_with_layout	single-part multidrawings with layout
multi_assembly	assembly multidrawings
multi_assembly_with_layout	assembly multidrawings with layout
cast_unit	cast unit drawings

`set_drawing_attributes(column)`

This line tells Tekla Structures which drawing properties to use when creating the drawings. The name of the saved drawing properties appears in parentheses.

`set_template_drawing`

This line tells Tekla Structures to use the defined drawing as a template when creating a new drawing. This line is used instead of the line `set_drawing_attributes`. The path and the name of the drawing template are given in parenthesis like in the following example:
`set_template_drawing("C:\TSModels\AngleModel":"[A.2]")`

`set_filter(column_filter)`

This line tells Tekla Structures which selection filter to use to select the parts from which to create drawings. The filter name appears in parentheses.

`create_drawings()`

Tekla Structures starts creating the drawings. This line should always appear immediately after the lines `set_drawing_type`, `set_drawing_attributes` and `set_filter`.

Remove master drawings from Master drawing catalog

You can remove obsolete master drawings from the **Master drawing catalog**.

Limitations: In the folder view, you cannot remove master drawings from the **Master drawing catalog** through the folders that you have created yourself. Use the **Saved searches** folders to display the desired master drawings and then remove the extra ones.

1. On the **Drawings & reports** tab, click **Master drawing catalog**.
2. In the **Master drawing catalog**, select the master drawings you want to remove.
3. Right-click and select **Remove from Catalog**.
4. Confirm the removal.

   Tekla Structures removes the master drawings from the catalog. Any created drawings that are based on removed master drawings remain in **Document manager**. The catalog properties that you have entered for the master drawing are not deleted. They are still available for the master drawing if you decide to add the master drawing back to the catalog.

Add sample and thumbnail images to master drawings

You can create sample or preview images of the drawings. You can add sample images for master drawings in **Master drawing catalog**. The same image is used as a thumbnail in the **Master drawing catalog** thumbnail list.

Before you can add a sample image or a preview image to a master drawing, you must have it somewhere ready.

Sample images are just samples, they are not real-time drawings, and they do not update when the drawing changes.

First create the image:

1. Open the drawing you want to use.
2. Clean up the drawing from unnecessary objects, for example, associativity symbols.
3. On the **Views** tab, click **Screenshot** --> **Sample image for Master drawing catalog**.

The image is saved in the `\drawings` folder under the model folder in `.png` format with the same name as the drawing. You can see the name of the image on the status bar.

Then you can add the sample image in the master drawing properties:

1. On the **Drawings & reports** tab, click **Master drawing catalog**.
2. In the **Master drawing catalog**, locate the desired master drawing and double-click it to open the **Master Drawing Properties** dialog box.
3. Add the image you created:

- For the master drawings of the type saved settings, cloning templates, and wizard files, click **Change image...** and browse for the sample image. Click **OK** to save the image as a sample image.
- For rule sets, click **Next** until you get to the page where you can add the sample image. Click **Browse** and browse for the sample image. Click **Save** to save the image as a sample image.

Now you can display the sample image by selecting the master drawing on the catalog list, right-clicking and selecting **Preview**. The thumbnail image in the **Master drawing catalog** thumbnail list is a smaller version of the same sample image.

Add, rename, and move folders in Master drawing catalog

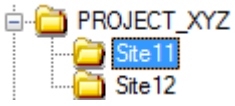
You can add, rename and move folders in **Master drawing catalog**:

1. In the **Master drawing catalog**, click the folder view button to go to the folder view.
2. **To add a folder**: Right-click the tree area in the upper part of the folder view, select **Create New Folder** and enter a name for the new folder.

   For example, enter `PROJECT_XYZ`.
3. **To add a subfolder**: Right-click folder, in this case `PROJECT_XYZ`, and select **Create New Sub-folder**.
4. Enter a name for the folder.

   For example, enter `Site12`.
5. Create another subfolder `Site10` following the steps 3 - 4.
6. **To rename a folder**: Right-click the folder, select **Rename** (**F2**) and enter a new name.

   For example, rename `Site10` to `Site11`.
7. **To move a folder upwards**: Right-click the folder and select **Move up**. In this case, move `Site11` one step up.



Copy master drawings to another folder in Master drawing catalog

1. In the **Master drawing catalog**, click the folder view button to go to the folder view.
2. Open the folder containing the master drawings that you want to copy to another folder, and select the drawings.
3. Right-click and select **Add to** and then select the desired folder.

The master drawings are copied. They are not removed from the original folder.

TIP Use the following keys and key combinations for selecting drawings:

- Select all displayed drawings: **Ctrl + A**
- Select consecutive drawings: Click the first drawing, hold down **Shift** and select the last drawing.
- Select several drawings: Click the first drawing, hold down **Ctrl** and select the rest of the drawings.

Remove master drawings from a Master drawing catalog folder

You may want to remove master drawings from a folder in the folder view, for example, when you have copied master drawings to another folder and no longer need them in the original folder.

1. In the **Master drawing catalog**, click the folder view button to go to the folder view.
2. In the upper part of the folder view, click the folder from which you want to remove drawings.

   The master drawings in the folder are shown in the lower part of the view.
3. Select the master drawings to be deleted, right-click and select **Remove from folder**.

The master drawings are removed from the folder. The master drawings are not removed from the catalog, just from the folder.

3.8 Clone drawings

You can clone drawings using the cloning templates added in the **Master Drawing Catalog** in the existing model and in other models, using a drawing in **Document manager** of the current model, and using the cloning templates in the template library.

You should consider cloning drawings when:

- There are several similar parts, assemblies, or cast units in the model.
- You want to ease editing work when there are several similar parts that have different position numbers. You may have separate drawings of these similar parts.
- The drawings need a lot of manual editing

The cloned drawing may contain more or less parts than the original drawing. Part properties, marks, associative notes and related text objects are cloned from a similar part in the original drawing.

If cloning does not produce a satisfactory result, you need to modify the cloned drawing manually. For example, you can create a drawing for one truss, modify the drawing, and then clone it for similar trusses. Sometimes you need to modify the cloned drawings where the trusses differ.

Cloned objects

The following objects can be cloned:

- Dimensions
- Marks for welds added in the drawing
- Marks for welds added in the model
- Level marks
- Revision marks
- Annotation objects
- All user-defined attributes of a drawing
- Texts
- Symbols
- Graphical drawing objects (shapes)
- Text files
- DWG/DXF files
- Hyperlinks
- Manually created section and detail views
- When you clone an assembly drawing that includes single-part drawings, Tekla Structures includes the single-part drawings in the cloned assembly drawing by default.

Create drawings using cloning templates in Master drawing catalog

In the **Master drawing catalog**, you can create single-part, assembly and cast unit drawings using cloning templates that you have added in the catalog from **Document manager**.

Note that only the cloning templates located in the folder that is set for the advanced option XS_CLONING_TEMPLATE_DIRECTORY are displayed in the **Master drawing catalog**.

Limitations:

- You cannot clone general arrangement drawings through the **Master drawing catalog**. You can clone general arrangement drawings only by using the **Clone** command in **Document manager**.
- You cannot clone multidrawings.
- The drawing properties of cloning templates cannot be modified through the **Master drawing catalog**.

1. On the **Drawings & reports** tab, click **Master drawing catalog**.
2. Double-click the cloning template that you want to use.
3. Go to the **Drawing creation** tab and specify how dimensions and marks are cloned.

   You can select to ignore the dimensions and marks, clone them, or recreate them.

   The **Marks** option controls revision marks and all building object marks.

   Selecting **Create** in the **Dimensions** and **Other marks** box is useful if cloning the dimensions or other marks does not produce satisfactory results. Using this option does not create any new views.
4. On the same tab, select which object types you want to clone.
5. If you want to create a drawing only for certain parts, select the parts from the model.

   You can also use an appropriate selection filter and select the whole model. Activate only the **Select objects** selection switch when you select objects, otherwise the selection may take a long time.
6. Click **Create drawings** or **Create drawings for all parts**.
7. Number the model if prompted to do so.

   Tekla Structures creates the drawings and adds them in **Document manager**. In the **Document manager**, cloned drawings are marked with the text **Drawing was cloned** in the **Changes** column.
8. Check the cloned drawing and modify it, if necessary.

TIP If you have cloning templates that you have created using an older Tekla Structures version and want to utilize the better associativity of a newer Tekla Structures version, refresh the drawing associativity by using the **Refresh associativity** command, which you can type in the **Quick Launch** box.

Clone by using cloning templates located in other models

You can take into use in the **Master drawing catalog** cloning templates located in other models, and create drawings using them.

Limitations: You can only use cloning templates from other models, not other types of master drawings, such as saved settings or rule sets.

1. On the **Drawings & reports** tab, click **Master drawing catalog**.
2. In the **Master drawing catalog**, click to open the **Models Containing Master Drawings** dialog box.
3. Click **Add model...** and browse to the desired model.
4. Click **OK**.

   Now the cloning templates in the defined folder are displayed in the **Master drawing catalog**.
5. Select the cloning template from the **Master drawing catalog** and create a drawing using the selected template.

TIP When you have similar parts in several projects, you can maintain a set of *cloning models* and then take the cloning templates in the cloning models in use when necessary.

Clone from Document manager

In addition to cloning drawings using cloning templates in **Master Drawing Catalog**, you can clone single-part, cast unit, assembly and general arrangement drawings from the **Document manager**.

NOTE
- When you clone drawings of assemblies or cast units, they must have the same type of main part as the assembly or cast unit from which the original drawing was created. For example, the upper chords of the original and the cloned truss could be main parts.
- In general arrangement drawings, one main view, and section and detail views are cloned. However, the cloning of the section and detail views may not be accurate.
- In general arrangement drawing cloning, annotations cannot be cloned properly for plan views if views are located in different XY coordinates. For example, if you have two GA drawings with plan views for different floors, the cloning result will be acceptable only if the floors are located on top of each other so the only difference between them is the Z coordinate. However, if the two plan views

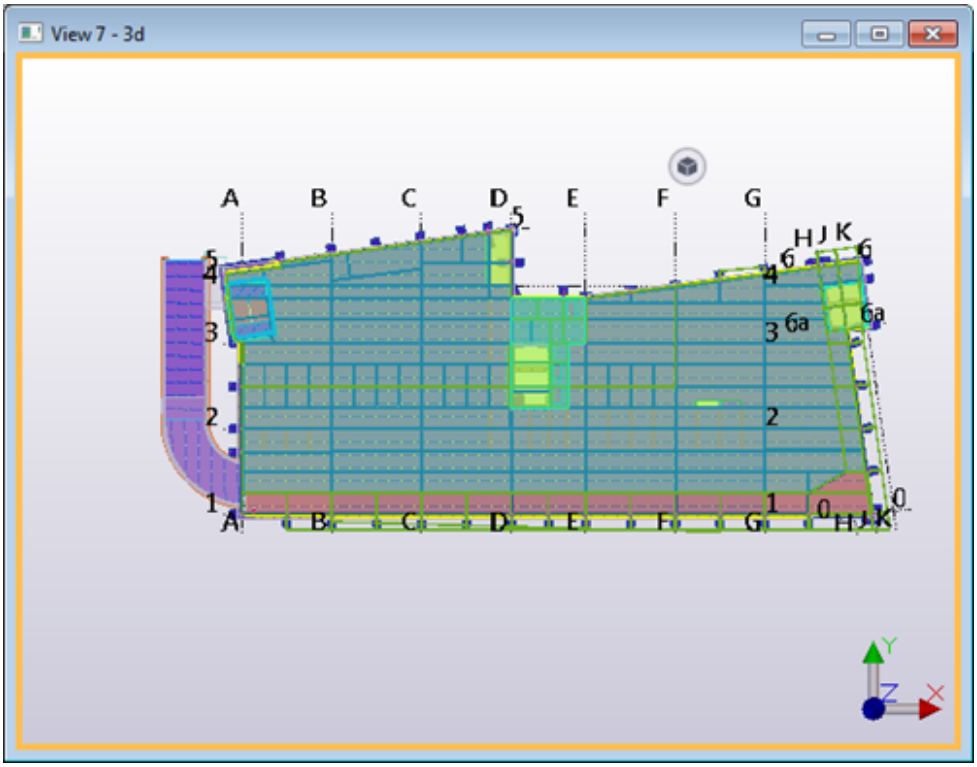
are located in two different buildings, the XY coordinates are different, and then the cloning result for annotations is not good.

Before cloning, finalize, save, and close the drawing you want to use as a cloning template.

Clone a drawing

To clone a drawing from **Document manager**:

1. In the model, select what to include in the drawing:
   - If you are cloning a single-part drawings, cast unit drawing or assembly drawing, select parts, assemblies or cast units.
   - If you are cloning a general arrangement drawing, select a model view. To do this, click the desired model view so that the view gets a yellow frame around it.



2. On the **Drawings & reports** tab, click **Document manager**.
3. Click **Clone** and do one of the following:
   - Select **Current model**, click **Select drawing** and, in **Document manager**, select the drawing that you want to use as a cloning template.
   - Select **Other model**, browse to the desired model and click **OK**. Then click **Select template** and select the drawing that you want to use as a cloning template. Leave the list open. For more detailed instructions

on this type of cloning, see section "Clone using drawing templates in a template library".

4. In the **Clone Drawing** dialog box, select the drawing object types you want to clone into the new drawing, and the actions for each object type.
   - For **Dimensions** and **Other marks** (all building object marks), select whether you want to **Clone** them, recreate them automatically when you clone a drawing (**Create**) or **Ignore** them in cloning.

     Selecting **Create** in the **Dimensions** and **Other marks** box is useful if cloning the dimensions or other marks does not produce satisfactory results. Using this option does not create any new views.
   - For other objects, select to **Clone** or **Ignore** them.
5. Click **Clone selected**.

Tekla Structures clones the drawing. In the **Document manager**, cloned drawings are marked with the text **Drawing was cloned** in the **Changes** column.

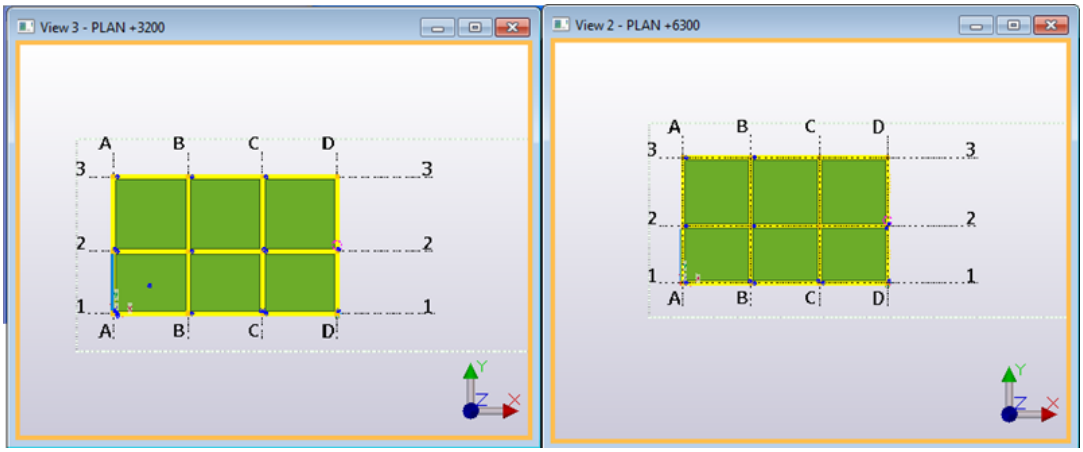
For an example of cloning a general arrangement drawing, see below.

Example: Cloning a general arrangement drawing

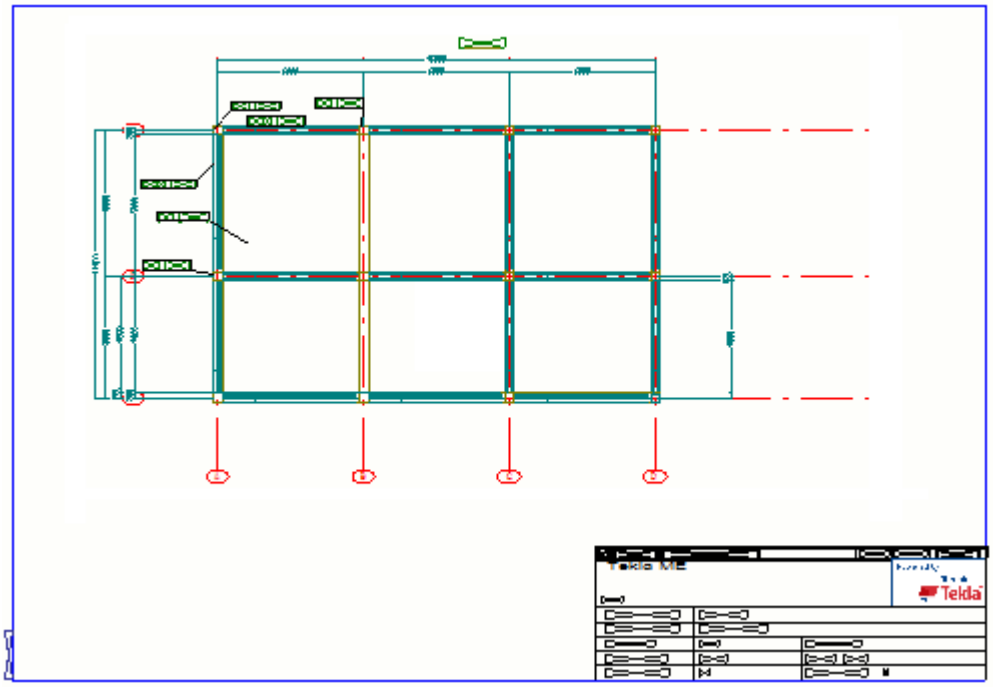
In this example, a general arrangement drawing has been created of the +3200 plan in a building, then the drawing is edited to get the desired result, and then a general arrangement drawing of the +6300 plan is cloned using the +3200 plan drawing as a cloning template.

For detailed instructions on cloning, see the section above.

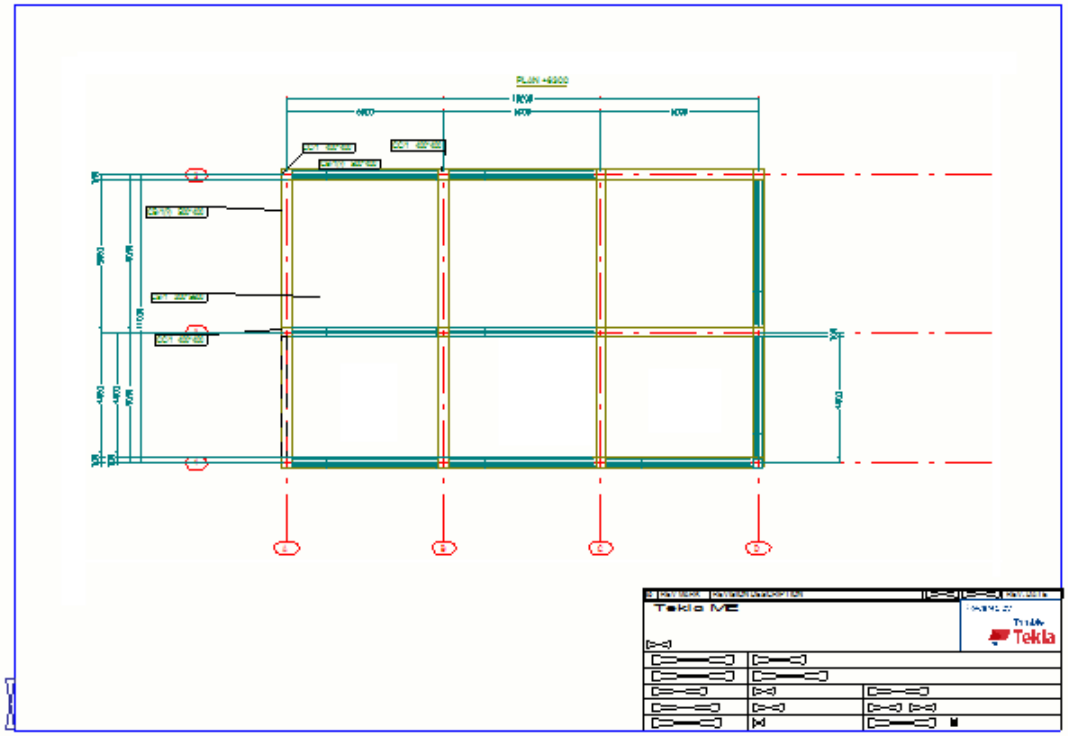
The first floor and second floor are quite similar:



The general arrangement drawing of the +3200 plan is edited view slightly, for example, some marks were deleted.



The model view that represents the +6300 plan was selected in the model and then the drawing is cloned by selecting the general arrangement drawing of the +3200 plan as a cloning template in the **Document manager**.



In the cloned drawing:

- The drawing view plane has been moved to match the model view +6300 plan.
- If there are parts in matching places, the marks have been cloned to the new location and the content has been updated.
- Dimensions are cloned.
- All building object marks are cloned.

Clone using drawing templates in template library

This is a more manual way of cloning drawings: you can create drawings using drawing templates located in the template library. The template library is in practice a model folder that contains the drawings that you want to use as drawing templates.

1. In a model that you use as a template model, create a drawing you want to use as a drawing template and save the drawing.
2. In another model where you want to create a drawing using a drawing template, select the objects to be included in the new drawing.
3. On the **Drawings & reports** tab, click **Document manager**.
4. In **Document manager**, select a drawing and click **Clone**.
5. Click **Other model**.

   Use **Browse for model...** to browse for another model folder containing the drawing templates that you want.
6. Click **Select template...**
7. In the **Drawing Templates** dialog box, select a drawing template and leave the list open.
8. In the **Clone Drawing** dialog box, use the **Objects and actions in cloning** options to define the drawing objects to be cloned and the actions for each cloned object.
9. Clone the drawing by clicking **Clone selected**.

Tekla Structures clones the drawing. In the **Document manager**, cloned drawings are marked with text **Drawing was cloned** in the **Changes** column.

TIP You can specify that a certain model folder is always used as a template library using the advanced option `XS_DRAWING_TEMPLATES_LIBRARY`.

Clone dimensions in selected views only

The dimension cloning options in the **Clone Drawing** dialog box affect all views, whereas the **Dimension creation method in this view** option sets the

dimension creation method for the selected view only. You can create automatic dimensions in the front view and clone the dimensions in section and end view, for example.

1. Double-click the drawing view frame to open the **View Properties** dialog box.
2. On the **Attributes 2** tab, set **Dimension creation method in this view** to **Clone**.

   Using this option affects the creation of the dimensions during cloning and re-dimensioning of existing drawings.
3. Click **Modify**.
4. Save and close the drawing.
5. Open the **Document manager**, select the drawing and click **Clone...**
6. In the **Clone Drawing** dialog box, select the objects to be cloned and the dimension cloning option (**Ignore**, **Clone**, **Create**).
   - If you select **Dimensions** > **Ignore**, the dimensions will be cloned only for those views that have **Dimension creation method in this view** set to **Clone**.
   - If you select **Dimensions** > **Clone**, the dimensions will be cloned for all views.
   - If you select **Dimensions** > **Create**, the dimensions will be created for all views except for those that have **Dimension creation method in this view** set to **Do not create**.
7. Click **Clone selected**.

Refresh drawing associativity after cloning

Often improvements in cloning and associativity require that the associative rules have to be recreated. This can be done by using the **Refresh associativity** command. When you use this command, you do not need to recreate the drawing.

For example, this command is very useful if you have a cloning template you have created using an older Tekla Structures version and want to utilize the better associativity of a newer Tekla Structures version.

1. On the **Drawings & reports** tab, click **Document manager**.
2. Open a cloning template whose associativity you want to refresh.
3. Go to **Quick Launch**, enter `Refresh associativity`, and click the **Refresh associativity** command on the displayed list.
4. Save the cloning template.

What to check in cloned drawings

Always check the cloned drawings to make sure that the drawing contents meet your needs, and marks, views and dimensions are correct.

You should go through the cloned drawings and check that everything is correct. Below is a checklist for this purpose.

Object	Check and change if necessary
Marks	• In general, mark contents are correct in cloned drawings, but sometimes you may need to modify the location of the marks. • Tekla Structures clones marks that can be mapped to the original drawing and creates new marks for parts that cannot be mapped to the template drawing.
Views	• Ensure that the view sizes and view orientation are correct, and that the views are placed correctly in the cloned drawing. The size of the views is updated according to the parts included in the views.
Dimensions	• If the cloned drawing contains less parts than the original drawing, the dimensions to the missing parts are automatically removed. • If the cloned drawing contains more parts than the original drawing, Tekla Structures dimensions the additional parts using automatic dimensioning, if you have set the advanced option `XS_INTELLIGENT_CLONING_ADD_DIMENSIONS` to `TRUE`. Because Tekla Structures uses automatic dimensioning for creating dimensions for new parts, you may have to check and correct the created dimensions. • Add missing dimensions and remove the incorrect ones.

3.9 Create multiple drawing sheets of the same part

You can create several drawings from the same part as multiple drawing sheets. The number of sheets is not limited, and each drawing can have its own properties.

This is useful when you need extra sheets for details and section views, and especially when you want to have several drawings of an assembly with different presentation options: one with a large 3D image of the assembly, one with additional section views, and one where only certain parts or sub-assemblies of the assembly are marked and dimensioned.

You can create multiple drawing sheets of the same part in the **Master Drawing Catalog** using rule sets or wizards, or using the drawing properties dialog box.

Create multiple drawing sheets with wizards

Before you start, ensure that you have created drawing property files (saved settings) for creating multiple sheets of the same part, each file containing the desired settings for a specific purpose and individual sheet numbers.

1. On the **Drawings & reports** tab, click **Master drawing catalog** to open the **Create Drawings - Master Drawing Catalog** dialog box.
2. Double-click the desired wizard file in the list to open the **Master Drawing Properties** dialog box.
3. Click **Edit drawing properties...** to open the wizard file (page 122).

   For an example of the wizard file contents, see Customize Master drawing catalog (page 142).
4. In the file, give the names of the drawing properties files that you want to use to create the multiple drawing sheets.

   Give the file names in parenthesis on the `set_drawing_attributes` row as shown below:

   ```
   set_drawing_type(cast_unit)
   set_drawing_attributes(sheet1, sheet2, sheet3)
   set_filter(column_filter)
   create_drawings()
   ```
5. Save the wizard file with a new name.
6. In the model, select the parts you want to create the drawings from.
7. Click the newly created wizard file in the **Master Drawing Catalog** and click **Create drawings**. Tekla Structures creates the drawings.

Create multiple drawing sheets using drawing properties

1. In the model, create a single-part, assembly, or cast unit drawing (page 107) of the desired model objects.
2. Open **Document manager**.
3. Right-click the created drawing and select **Properties**.
4. Adjust the drawing properties as needed, and change the drawing name so that it refers to the type of drawing you want to create, for example, `FRAME 3D`.

   Do not touch the **Sheet number** box.
5. Click **Modify** to apply the changes and leave the dialog box open.
6. In the open drawing properties dialog box, adjust the drawing properties again and change the name to match the type of the new drawing, for example, `FRAME FRONT`.
7. Change the sheet number in the **Sheet number** box to get another drawing of the same objects with a different sheet number.
8. Click **Apply** and create a new drawing of the same objects (page 107) with different properties.
9. Continue making all the sheets you need in the same way as in steps 6 - 8.

Example

An example how the **Document manager** shows the multiple drawings:

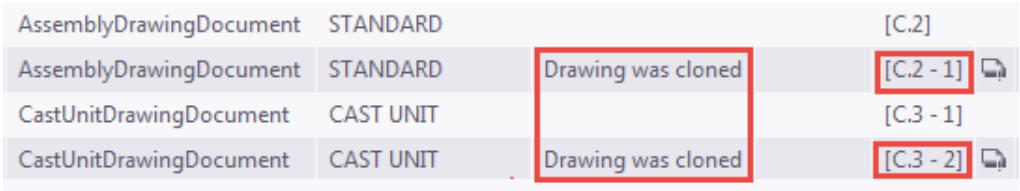
02.01.2012	02.01.2017	1179* 830	A	[FRAME.1 - 1]	FRAME FRONT
02.01.2012	02.01.2017	1179* 830	A	[FRAME.1 - 2]	FRAME DETAILS
02.01.2012	02.01.2017	1179* 830	A	[FRAME.1]	FRAME 3D

3.10 Copy a drawing to a new sheet

You can copy a drawing to a new sheet. This is useful, for example, if you want to have the same layout and views as in the original drawing, but highlight something else on the new drawing sheet.

1. On the **Drawings & reports** tab, click **Document manager**.
2. Select the drawing that you want to use as the original for copying.
3. Right-click and select **Create Drawings** --> **Copy to new sheet** .

Tekla Structures copies the original drawing to a new sheet and gives the copied drawing a new sheet number.



4. Open the new drawing sheet, make the necessary changes and save it.

NOTE If you use the **Recreate** (**Shift+R**) command for the copied drawing in **Document manager**, Tekla Structures asks if you want to copy the drawing again from the original drawing. If you answer yes, the previously copied drawing sheet will be replaced by a new copy of the previously selected original drawing.

See also

Create multiple drawing sheets of the same part (page 162)

4 Edit drawings

After you have created a drawing, you can modify properties of the drawing and views, building objects, and dimensions and marks already included in the drawing. You can add views, marks, notes, text, sketch objects and other objects. You can also change drawing colors, and use special custom presentations for some object types.

You can have a quick look at a snapshot of a drawing without opening the drawing itself:

Snapshots in drawings (page 170)

Open drawings (page 168)

You can edit the drawing name and titles:

Modify drawing names and titles (page 167)

You can change the drawing layout and the included table sets:

Select a new layout for your drawing (page 174)

Change table sets in drawings (page 176)

You can add and/or edit drawing views, dimensions, marks and other drawing objects:

Add views in drawings (page 179)

Modify, arrange and align drawing views (page 200)

Add dimensions in drawings (page 206)

Add marks, notes, and texts in drawings (page 289)

Add links in drawings (page 380)

Add and modify symbols in drawings (page 371)

Clone selected in drawings (page 422)

Add sketch objects in drawings (page 387)

Show building objects in drawings (page 426)

2D Library in drawings (page 529)

Custom presentations in drawings (page 543)

Moment connection symbols in Tekla Structures drawings (page 544)

Show welds in drawings (page 489)

Show reinforcement in drawings (page 448)

Show pours in drawings (page 485)

Show spiral beams in drawings (page 443)

Show reference objects in drawings (page 526)

Show grids in drawings (page 514)

User coordinate system (UCS) (page 548)

You can hide, move, reshape, resize, align, arrange, explode, and delete drawing objects and use cut lines to indicate parts partly outside the view border:

Show or hide drawing objects (page 408)

Hide part face areas and outlines with cover-up tools (page 413)

Align selected drawing objects (page 417)

Drag, reshape and resize drawing objects (page 419)

Arrange annotation objects (page 416)

Delete drawing objects (page 415)

Indicate cut lines in Tekla Structures drawings (page 541)

Explode drawing plug-ins and use ordinary drawing objects (page 542)

When you save and close the drawings, there are some things you need to keep in mind:

Save and close drawings (page 170)

4.1 Modify drawing names and titles

Tekla Structures names the drawings according to the name given in the drawing properties. The drawing name is displayed in **Document manager** and in drawing and report templates. You can change the default drawing name when you create the drawing and in an existing drawing. In addition to the drawing name, you can give titles for your drawings. Tekla Structures shows the titles in **Document manager** and in drawing and report templates, and also in print output file names. You can define up to three titles.

Rename drawings

1. In **Document manager**, right-click the drawing and select **Properties**.
2. Enter the new name in the **Name** box.

   The maximum number of characters in 32.
3. Click **Modify**.

   Close the **Document manager** and open it again to see the change.

Give titles to drawings

1. In **Document manager**, right-click the drawing and select **Properties**.
2. Enter the titles.
3. Click **Modify**.

   Close **Document manager** and open it again to see the change.

TIP If you want to customize your print file names and use title values instead of drawing names in them, you can define that the title entered in this dialog box is used in the print file name. To do this, enter the value `%TPL:TITLE1%` (or `%TPL:TITLE2%` or `%TPL:TITLE3%`) as the value for the advanced option `XS_DRAWING_PLOT_FILE_NAME_G` (or `_W`, `_A`, `_M` or `_C` depending on the drawing type) through **File** --> **Settings** --> **Advanced options** --> **Printing** .

For more information, see Customize print file names (page 622) and TITLE1..3.

4.2 Open drawings

You can open drawings both in the model and in the drawing mode. Only one drawing can be open at a time.

TIP To always open the drawings maximized, set the advanced option XS_OPEN_DRAWINGS_MAXIMIZED to `TRUE`.

Open a drawing in the model

Do one of the following:

- In the model mode, on the **Drawings & reports** ribbon tab, click **Document manager** (**Ctrl + L**), and then double-click a drawing in the list, or select a drawing in the list and click at the bottom.
- Select objects in the model, open the contextual toolbar and select the **Open or create drawings** button. Single-part drawings, assembly

drawings, cast unit drawings, and multidrawings containing the selected objects are listed in the menu. You can open a drawing just by clicking it in the list.

When you open a drawing, a message box is displayed showing the progress and what is happening, and a snapshot of the drawing is displayed. You can click **Cancel** to cancel the opening.

Open a new drawing when a drawing is already open

You can open another drawing while you already have a drawing open. The drawing that you already have open closes when you do this.

Do any of the following:

- In the drawing mode, on the **Drawing** ribbon tab, click **Document manager** (**Ctrl + O**), and then double-click a drawing in the list, or select a drawing in the list and click at the bottom.
- To open the next drawing in the **Document manager** list, press **Ctrl + Page Down**, or click the **Open next drawing** button at the bottom.
- To open the previous drawing in the **Document manager** list, press **Ctrl + Page Up**, or click the **Open previous drawing** button at the bottom.

If you have made some changes in the currently open drawing, Tekla Structures asks you if you want to save the changes before opening another drawing. You also have a chance to take a snapshot (page 170) of the current drawing or mark it ready for issuing (page 578). If you have set the advanced option XS_ALWAYS_CONFIRM_SAVE_WHEN_CLOSING_DRAWING to `TRUE` the confirmation message is always shown. The default value is FALSE. This means that if you close a drawing without making any changes, Tekla Structures does not by default prompt you to save the drawing.

When you open a drawing, a message box is displayed showing the progress and what is happening, and a snapshot of the drawing is displayed. You can click **Cancel** to cancel the opening.

Message: Cannot load the selected drawing

Sometimes, you cannot open the drawing because the drawing file cannot be found, or the drawing file is somehow incompatible. Tekla Structures then displays one of the following messages:

- "Cannot load the selected drawing. Could not find the drawing file. Check Tekla User Assistance for more information."

- "Cannot load the selected drawing. Incompatible drawing file. Check Tekla User Assistance for more information."

For more information, see Cannot load selected drawing.

4.3 Save and close drawings

Save a drawing

Tekla Structures automatically saves drawings at set intervals. You can also save your drawing when ever you want to.

- To save a drawing, in the open drawing, go to the **File** menu and click **Save drawing**.

  Both the drawing `*.dg` and the model `*.db1` and `*.db2` are saved. The drawings are saved in the `\drawings` folder under the model folder.

  A snapshot is by default created when you open and save a drawing. The snapshot represents the situation when the drawing was last saved. For more information, see Snapshots in drawings (page 170).

For more information about autosave and the autosave interval, see Autosave drawings.

Close drawings

You can only have one drawing open at a time. You always have to close an open drawing before you can open another one.

1. Do one of the following:
   - Go to the **File** menu and click **Close drawing mode**.
   - On the **Drawing** tab, click **Close**.
   - Click the **Close** button **X** in the upper-right corner of the drawing window.
2. If you have made changes in the drawing, Tekla Structures asks you to confirm if you want to save your changes.

   You can also select to mark the drawing ready for issuing. For more information, see Mark drawings ready for issuing (page 578).

4.4 Snapshots in drawings

With snapshots you can take a quick look at any drawings without opening the drawing. Use this tool when you just want to check but not edit a drawing, or take a look at several drawings while trying to find the one you are looking for, for example, a certain drawing revision. Using snapshot overlay, you can view the contents of drawings directly in model view without opening the actual drawing. You can also show drawing snapshots against the latest version of a drawing or against another drawing in the drawing mode. You can take snapshots of all types of drawings, and all drawing types can be overlaid with drawing snapshots.

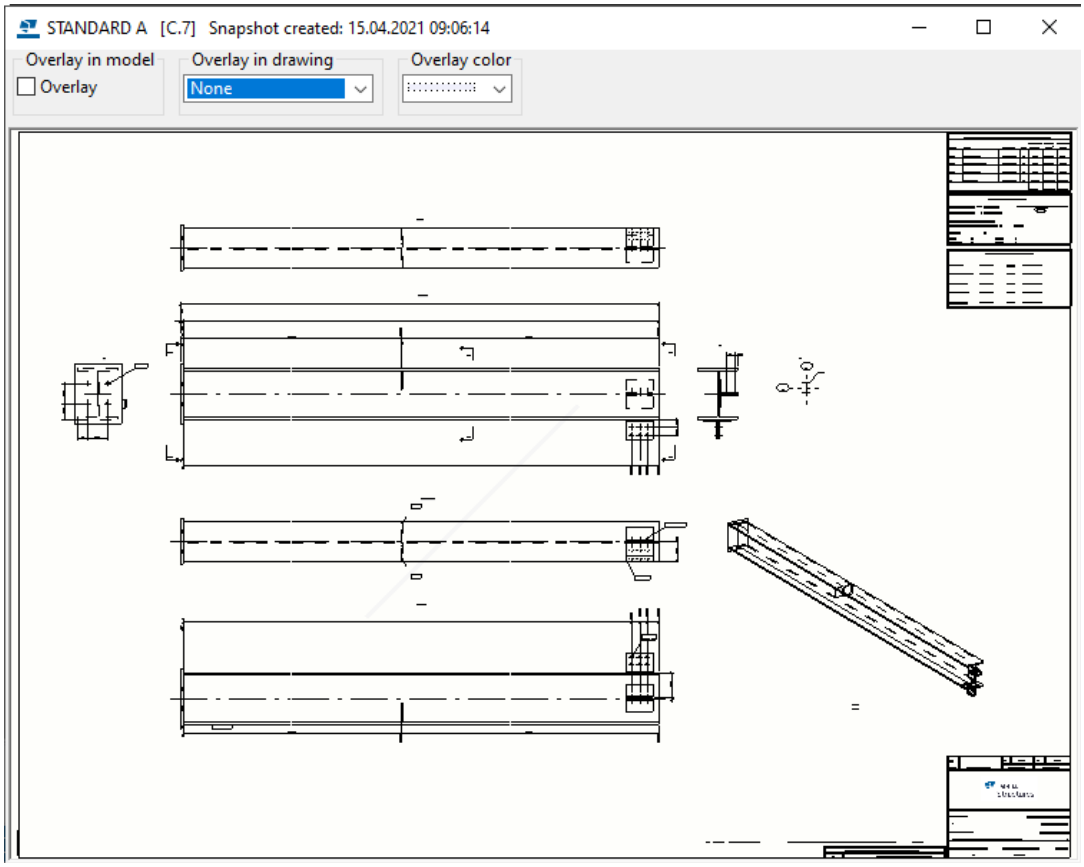
Create and view drawing snapshots

A snapshot is by default created when you open and save a drawing. The snapshot represents the situation when the drawing was last saved, so any newer changes to the model are not reflected in the snapshot.

1. Select a drawing and open it from the **Document manager**.
2. Save the drawing by going to the **File** menu and clicking **Save drawing**.

   The snapshot is saved in the `..\<model>\drawings\Snapshots` folder.
3. Select the same drawing from the **Document manager**.
4. Click the **Snapshots** button at the bottom of the **Document manager** to display the snapshot.

Below is an example of a snapshot.



If you have selected a drawing that does not have a snapshot and click **Snapshots**, an instructional view is displayed asking you to open the drawing and save it to create the snapshot.

To create a snapshot of a drawing automatically when the drawing is created, set XS_DRAWING_CREATE_SNAPSHOT_ON_DRAWING_CREATION to `TRUE` in the **Drawing properties** category in the **Advanced options** dialog box.

Snapshot overlay

You can view a snapshot of a drawing while you are editing a drawing. In the snapshot overlay, you can see the latest changes, and can align the drawing content, for example. You can see the changes in a drawing since the latest update, and switch between the snapshot and the actual drawing.

Finding the correct drawing by going through the snapshots is much faster than opening the actual drawings one by one.

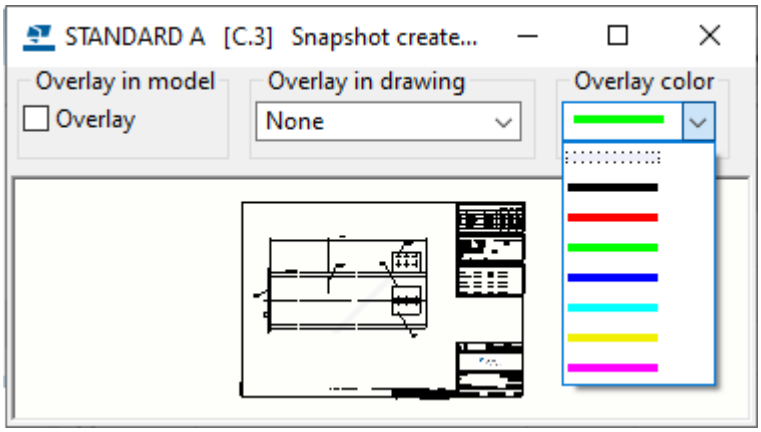
Limitations

- Texts are one sided when viewed from the model, and DX graphics do not support text in snapshots.

- Overlay in the model was not designed to work for the 3D drawing views .
- Views that are outside the printing area are displayed in the model overlay.
- Reading large GA drawing snapshots may take a long time.
- Loading the snapshot takes longer for the first snapshot than for the rest.

Before you can see a snapshot overlay of a drawing, you need to create a snapshot by opening a drawing, and saving the drawing.

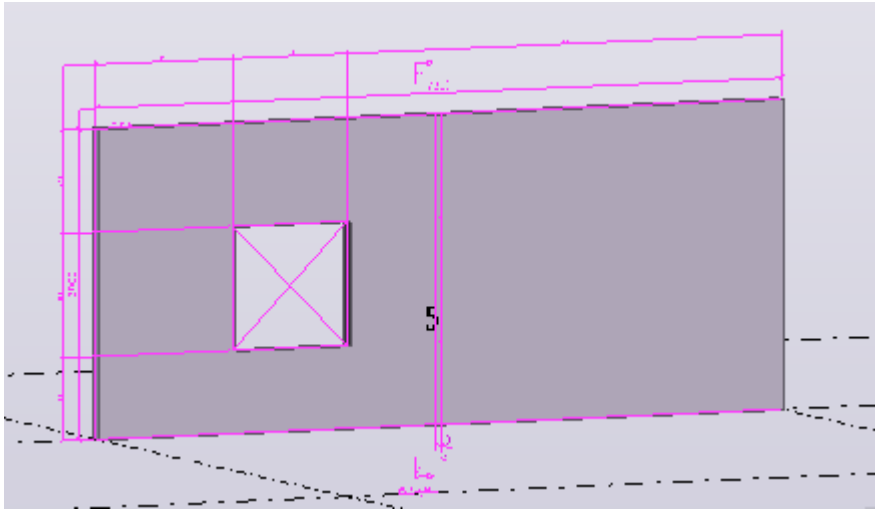
Below you can see the settings in the snapshot view:



Snapshot overlay in model

1. Open **Document manager** and select a drawing.
2. Click **Snapshots**.
3. At the top of the displayed snapshot dialog box, under **Overlay in model**, select the **Overlay** check box. You can also select a color for the snapshot.

A cast unit is overlaid with the corresponding cast unit drawing.



Snapshot overlay in drawings

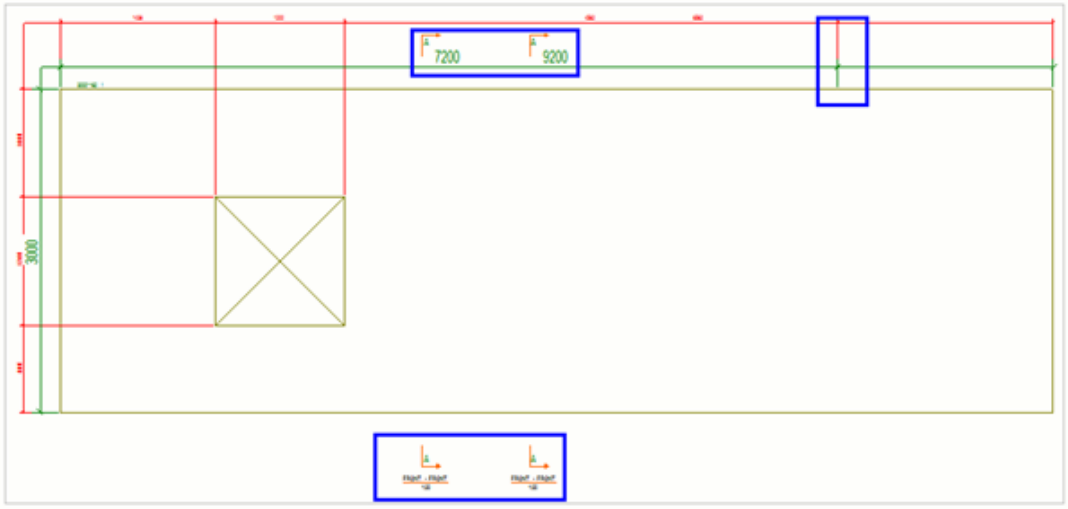
1. Click **Drawing** --> **Document manager**, and open a drawing that has changed after the latest snapshot.
2. Still in **Document manager**, select the same drawing and click **Snapshots**.
3. Select one of the options under **Overlay in drawing**:

   Overlay: Show the snapshot on the actual drawing so that both are visible.

   Show snapshot only: Show the snapshot only without the actual drawing.

   None: Quickly switch to the actual drawing.
4. Select a color for the snapshot.

In the example below, the size of the cast unit has changed since the snapshot was taken.



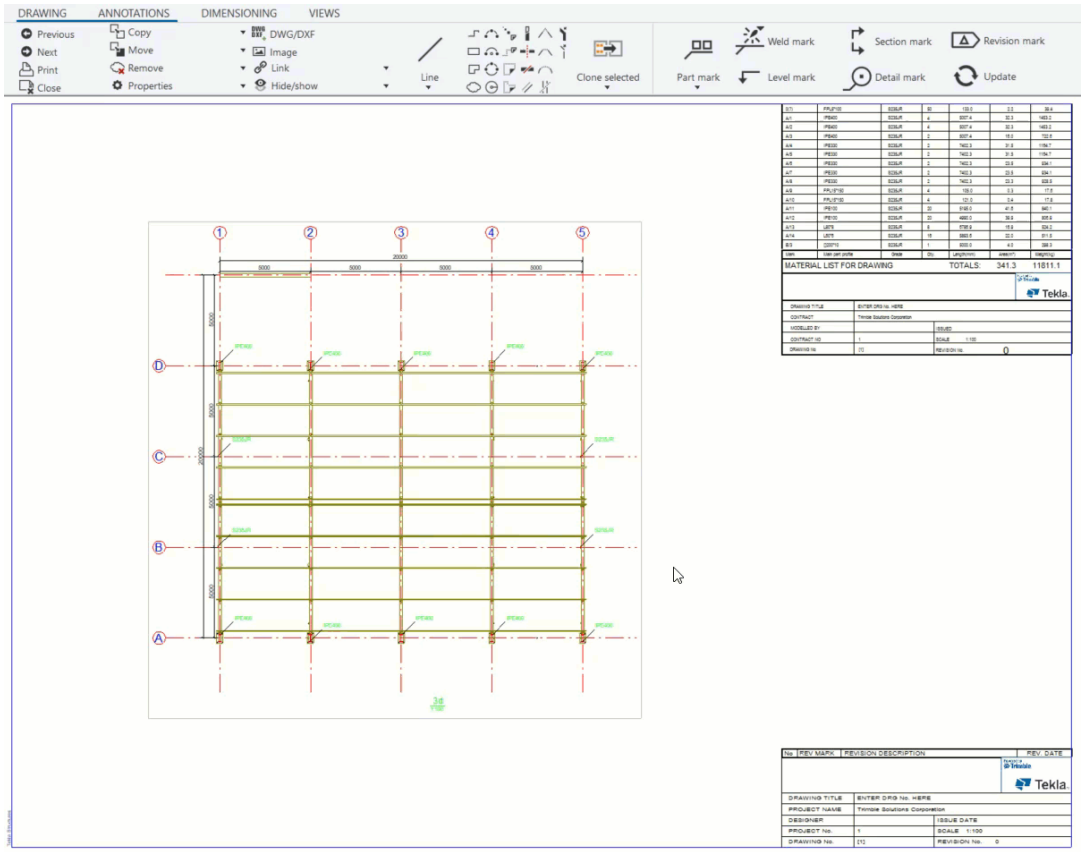
TIP If a plan view GA drawing is open, and you want to align content using another GA drawing, you can open a snapshot of the second drawing on the first one, and snap to the snapshot, when placing or moving objects in an open drawing. You can snap to drawing snapshot overlays with the **Snap to geometry lines / points** snap switch .

4.5 Select a new layout for your drawing

You can select which drawing layout you want to use in a drawing in the drawing properties dialog box. You can either use the new drawing layout for

the currently open drawing, or save the new drawing layout to drawing properties for future use.

1. Depending on where you are, do one of the following:
   - On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
   - In an open drawing, on the **Drawing** tab, select **Properties** --> **Drawing**.
2. Load drawing properties that are as close to the ones you need as possible.
3. Click **Layout**.
4. Select a new layout from the **Layout** list.
5. Do any of the following:
   - To apply and save the new layout to the open drawing, click **Modify**.
   - To save the changed drawing properties for future use, click **Save**.
6. Click **OK** to close the drawing properties dialog box.



See also

Define drawing layout (page 655)

Layout properties (page 910)

4.6 Change table sets in drawings

To make minor changes to the table sets in your drawings, you can exclude individual tables from the drawing layout. When you exclude a table, the table becomes a part of the current drawing, so any changes that you make to that table will not be saved to the drawing layout. This way, you can avoid and remove overlaps in drawing content and tables without creating new layouts or drawing sizes for each case.

Exclude tables from the drawing layout

1. In the open drawing, select the tables that you want to edit.
2. Right-click one of the tables, and select **Exclude template from layout**.

The tables are now part of the current drawing, so making changes to them does not affect the drawing layout.

NOTE If any table has been excluded from the drawing layout, you cannot change the drawing size.

According to your needs, see any of the following instructions.

Move excluded tables

1. In the open drawing, select the tables that you want to move.
2. Do any of the following:
   - Select the tables and hold down the left mouse button. Then, drag the tables to new positions, and release the mouse button.
   - Right-click one of the tables and select **Move**. Then, pick the origin for moving, and the destination point.

Rotate excluded tables

1. In the open drawing, select the tables that you want to rotate.
2. Right-click one of the tables.
3. Select **Rotate clockwise**.

The tables are rotated clockwise by 90 degrees.

Repeat steps 2 and 3 to achieve the right rotation angle.

Delete excluded tables

1. In the open drawing, select the tables that you want to delete.
2. Right-click one of the tables.
3. Select **Delete**.

You can undo deleting a table by pressing **Ctrl+Z** on the keyboard.

Edit excluded tables in Layout editor

1. In the open drawing, select the tables that you want to edit.
2. Do any of the following:
   - Double-click one of the tables, and select **Drawing layout editor**.
   - Right-click one of the tables, and select **Open Layout editor**.

   Note that when you open **Layout editor** like this, you can only modify layout properties related to tables.

   In the **Tables in use** list in **Layout editor**, excluded tables are shown as follows: `<<TABLE NAME>>`.
3. Do any of the following:

To	Do this
Scale a table	a. In the drawing layout, select the table. b. In the **Scale** box, type a new value. c. Press **Enter**.
Rotate a table	a. In the drawing layout, select the table. b. In the **Rotation** list, select a new rotation angle. c. Press **Enter**.
Allow overlapping with drawing views	a. In the drawing layout, select the table. b. Select the **Overlap with views** check box. If you do not select the **Overlap with views** check box, Tekla Structures keeps the drawing views outside the tables area when it recreates the drawing.

4. On the **Layout editor** ribbon, click **Save**.

The changes are saved to the current drawing only. The drawing layout is not updated to include the changes.

Add new tables to the current drawing

1. In the open drawing, select one of the excluded tables.
2. Do any of the following:
   - Double-click the table, and select **Drawing layout editor**.
   - Right-click the table, and select **Open Layout editor**.

   Note that when you open **Layout editor** like this, you can only modify layout properties related to tables.

   In the **Tables in use** list in **Layout editor**, excluded tables are shown as follows: `<<TABLE NAME>>`.

3. On the **Layout editor** ribbon, click **Add tables**.
4. In the **Available tables** dialog box, click the table that you want to add.
5. If you are adding a DWG or DXF file, or a keyplan, do one of the following:

To	Do this
Add a DWG or DXF File	a. Browse to find and select the file that you want to add to the drawing. b. Click **Open**.
Add a keyplan	• In the **Drawings** dialog box, select an existing key plan drawing and click **Select**. You can now close the **Available tables** dialog box.

6. Click the point in the drawing layout where you want to add the table.

   The tables are automatically anchored to the drawing view frame.
7. If you want to change the table scale or rotation, select the table and type a new **Scale** or **Rotation** value.

   Note that you can only edit the scale and rotation of template files (`.tpl`).

   Rotation and scale changes are applied to the drawing layout immediately. When you scale or rotate tables, the tables remain anchored to the same position.
8. If you want to allow the table to overlap with a drawing view, select the **Overlap with views** check box.

If you do not select the **Overlap with views** check box, Tekla Structures keeps the drawing views outside the tables area when it recreates the drawing.

Repeat steps 3 to 8 for all tables that you want to add.

9. On the **Layout editor** ribbon, click **Save**.

The new tables are saved to the current drawing only. The original drawing layout is not updated to include the tables.

Discard drawing-specific changes

- According to your needs, do one of the following:

To	Do this
Reinclude excluded tables in the drawing layout and restore their positions	1. Select the tables. 2. Right-click and select **Restore template position**.
Reinclude all excluded tables in the drawing layout and restore the original drawing layout	Do one of the following: • Right-click anywhere in the drawing, and select **Restore layout**. • In the **Layout editor** ribbon, click **Restore layout**.

4.7 Add views in drawings

Drawing views contain modeled Tekla Structures building objects and marks, dimensions, and other objects that you might have added. A drawing view is another way of looking at the model. Drawings may include several views. In addition to the views that Tekla Structures creates automatically based on your selection at drawing creation, you can also add new views and modify the existing views in an open drawing.

You can have different types of views in Tekla Structures drawings:

- Main views: front, top, back and bottom views
- Section views
- End views
- Single-part views
- 3D views
- Detail views
- Key plan views

- Views along grid lines
- Elevation views
- Plan views
- Views that you have created manually from model views, from areas in model views, or from areas in existing drawing views

Adding views manually

You can manually add more views in drawings:

Create views in drawings (page 180)

Copying, linking, and moving views

You can move, copy and link views from one drawing to another:

Copy, link, and move drawing views (page 188)

Modifying views, view location, and view labels:

You can modify views manually:

Modify, arrange, and align drawing views (page 200)

See also

Define drawing views (page 687)
Add and modify section marks in drawings (page 343)
Add and modify detail marks in drawings (page 347)
View properties in drawings (page 912)
Section view properties (page 929)

Create views in drawings

After creating a drawing, you can manually add more views.

You can create the following types of views in an existing drawing:

- Section views
- Curved section views
- Detail views
- Front, top, back, bottom views of parts
- 3 D views of parts
- Drawing views of an entire model view
- Drawing views of selected areas in a model view
- Drawing views of selected areas in a drawing view

TIP Before you create a new view, create a drawing view filter to adjust what is shown in a view.

Create a section view

You can create section views of the parts in a drawing view in an open drawing. The drawing must contain at least one view.

Note that you cannot create a section view from a 3D view. If you try to pick the points for the view, you will get an error message.

1. First, set the section mark properties in an open drawing:
   a. In the drawing property pane object list, select **Section mark**.
   b. Modify the section label content, and the section label, section mark symbol, and cutting line properties.

      For more information about the properties, see Add and modify section marks (page 343).
2. Next, set the section view properties in an open drawing:
   a. Hold down **Shift**, and on the **Views** ribbon tab, click **Section view**.
   b. Modify the view properties as required.

      For more information, see View properties in drawings (page 912).
3. Pick two points to define the position of the section plane.

   It is easier to pick the points if you activate orthogonal snapping: On the **File** menu, click **Settings** and select **Ortho** or press **O**.

   For beams, you can also try picking the top line of the beam and then the bottom line of the beam using the snap switch **Snap to perpendicular points**.
4. Pick two points to define the direction of the cut box and the section view depth.

   When you pick the cut box, exaggerate a little. You can adjust the view depth and the view boundary also later on.

   The direction of the section is the direction where the section mark arrows are pointing.
5. Pick the position for the section view.

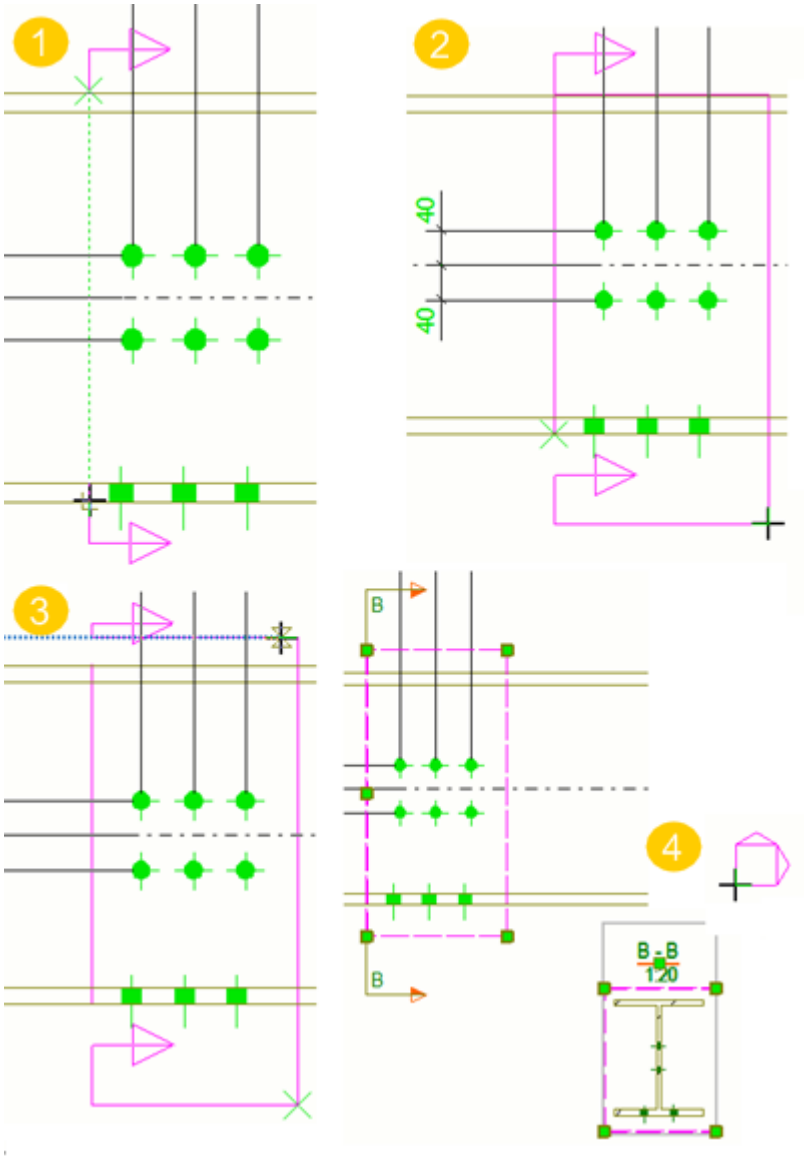
   A view symbol follows the mouse pointer allowing you to see where you are about to place the section view.

   The depth to the opposite direction is zero (0).

   Tekla Structures creates the section view using the current view properties and section mark properties, adds a section mark in the original view, and places the section view in the position you picked.

You can modify the created views. For more information, see Modify, arrange, and align drawing views (page 200).

TIP
- The view boundary of the created section view remains selected, and you can adjust the depth and the height of the view boundary by dragging the boundary.
- If necessary, change the view scale of the section view by adjusting the **Scale** setting.
- If you want to create another view, start the **Section view** command again.



(1) The first two picked points indicate the position of the section plane.

(2) The third picked point indicates the direction of the cut box and the depth of the section view. Here you can exaggerate a little.

(3) The fourth pick finalizes the cut box.

(4) A view symbol follows the mouse pointer while you are placing the section view. The section view is placed in the selected location. The section view remains selected and the view boundary highlighted right after the view creation.

The section mark is drawn in the original view. The view boundary of the section view is also highlighted in the original view right after the view creation.

Create a curved section view

You can create a curved section view of the parts in a drawing view in an open drawing. The drawing must contain at least one view. This tool is useful when you want to visualize an unfolded face of a building or to manage cladding.

Limitation: A curved section view does not work if it is done on a curved object that is vertical or in 3D.

1. First, set the section mark properties in an open drawing:
   a. In the drawing property pane object list, select **Section mark**.
   b. Modify the section label content, and the section label, section mark symbol, and cutting line properties.

      For more information about the properties, see Add and modify section marks (page 343).
2. Next, set the section view properties in an open drawing:
   a. Hold down **Shift**, and on the **Views** ribbon tab, click **Curved section view**.
   b. Modify the view properties as required.

      For more information, see View properties in drawings (page 912).
3. Pick three points on the cut plane.
4. Pick two points to indicate the cut box.
5. Pick a point to indicate the location of the curved section view.

   A symbol follows the cursor allowing you to see where you are about to place the curved section view.

Tekla Structures creates the curved section view using the current view properties and section mark properties, adds a section mark in the original view, and places the section view in the position you picked.

You can modify the created views. For more information, see Modify, arrange, and align drawing views (page 200).

Create a detail view

You can create a detail view from an area you select in an existing drawing view. By default, the detail view has the same scale as the main view, but in some environments the detail view is scaled up. The detail view has the same direction as the original view. Before creating the detail view label and detail mark, define a start number or letter in the drawing properties.

1. Open a drawing.
2. Check or change the start number or letter for the detail view and symbol:

   a. Double-click the drawing background and select **Detail view** from the options tree.

   b. Change the value when necessary, and click **Modify**.
3. Set the detail mark properties:

   a. In the drawing property pane object list, select **Detail mark**.

   b. Modify the detail name and detail label content, and the related properties, such as mark symbol and detail boundary.

      The shape of the detail boundary you select affects the way you select the area for your detail. For more information about detail marks and their properties, see Add and modify detail marks (page 347).

      To change the detail symbol to a custom symbol, use the advanced option XS_DETAIL_MARK_REFERENCE_SYMBOL.
4. Next, set the detail view properties:

   a. Hold down **Shift** and, on the **Views** tab, click **Detail view**.

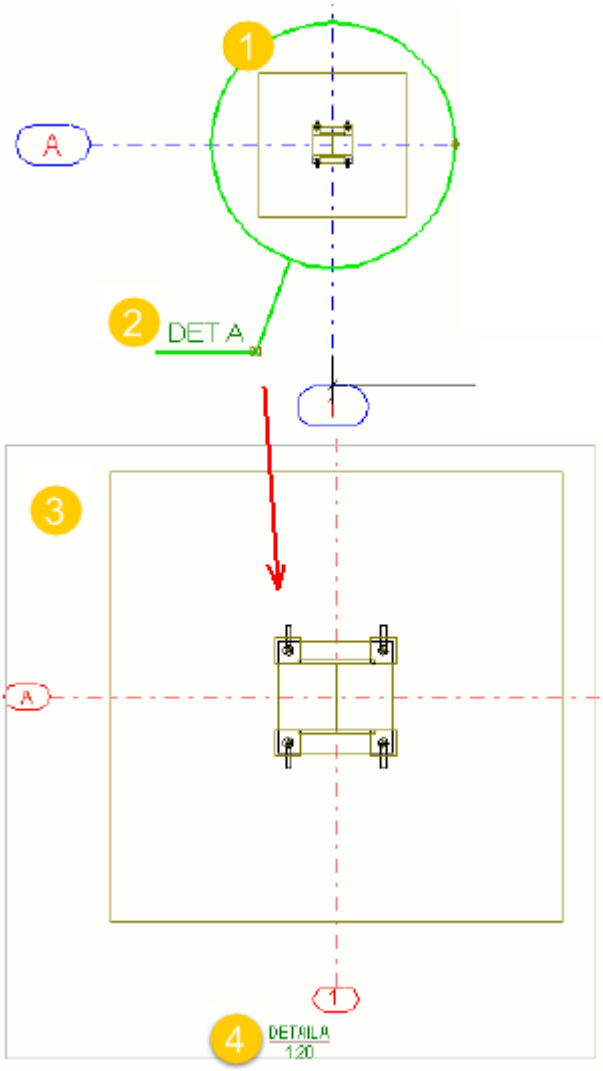
   b. Modify the view properties as required.

      For more information about view properties, see View properties in drawings (page 912).
5. Depending on the selected detail boundary shape, do one of the following:

   - If the shape is **Circle**, pick the center point of the circle and then pick a point on the circle.
   - If the shape is **Rectangle**, pick the corner points for the rectangle.
6. Pick a position for the detail mark.

7. Pick a position for the detail view.

Tekla Structures creates the detail view using the current detail mark and view properties. The detail view takes the view depth from the original view even if you try to change it. You can change the properties after creating the view.



(1) Detail boundary is set to **Circle**. You can increase or decrease the detail symbol size by dragging the handle on the detail boundary.

(2) Detail mark

(3) Detail view

(4) Detail view label

Create additional drawing views of parts

You can create additional views of a part within a single-part, cast unit or assembly drawing. You can select the part plane (front, top, back, or bottom) to use or create a 3 D view of the part.

1. Open a drawing.
2. On the **Views** tab, click **View of part** and select one of the following:
   - **Front**
   - **Top**
   - **Back**
   - **Bottom**
   - **3D view**

   Tekla Structures creates the view using the current view properties. If a plane already has a view in the drawing, a new one is not created.
3. Click the frame of the new view to open the view properties and modify the properties as required.

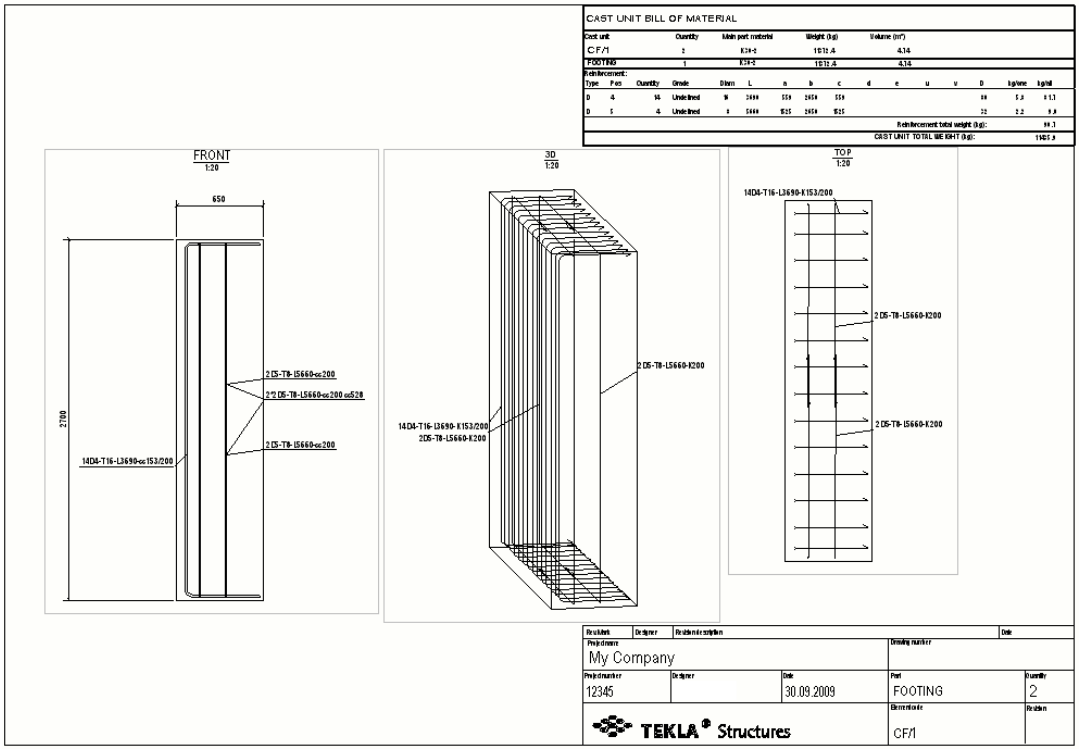
   If the drawing property pane is not open, double-click the view frame.

   For example, for 3D views, you may want to modify the view angle.

   For more information, see View properties in drawings (page 912).
4. Click **Modify**.

Example

In the example below, the drawing originally contained the front view only. A 3D view and a top view were added. The angle of the 3D view was modified in the view properties.



Create a drawing view of an entire model view

You can create a new drawing view of an entire model view.

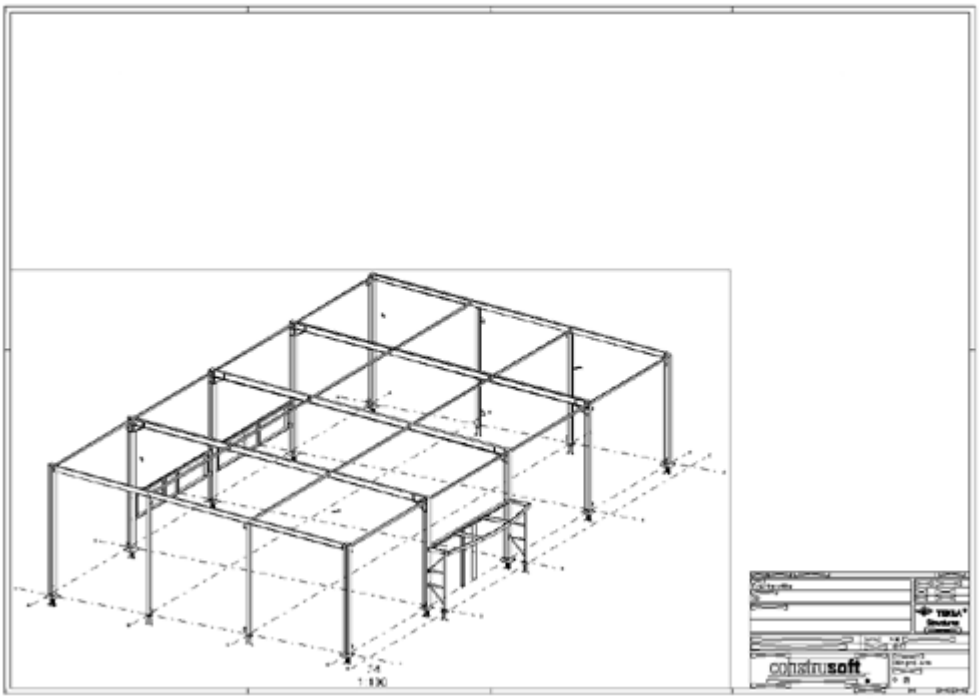
1. Open a drawing.
2. To open the model view list, on the **Views** tab, click **Model views** --> **Model view list**, and leave the list open.
3. To set the properties for the new view, select **View** in the drawing property pane object list.

   You might want to change the view scale, for example.

   You can also hold down **Shift**, and on the **Views** tab, click **Entire model view**.

   For more information about view properties, see View properties in drawings (page 912).
4. Open a model view from the model view list.
5. Click the open model view.

Tekla Structures creates a drawing view using the current drawing view properties. The drawing view has the same depth as the model view. Tekla Structures calculates the boundaries of the view to fit the entire model view into the drawing view, and places the view in the drawing.



Create a drawing view of a selected area in a model view

You can create a new drawing view of the area that you select in the model.

1. Open a drawing.
2. To open a model view list, on the **Views** tab, click **Model views** --> **Model view list**, and leave the list open.
3. To set the properties for the new view, select **View** in the drawing property pane object list.

   You might want to change the view scale, for example.

   Another way to open the viewe properties is to hold down **Shift**, and on the **Views** tab, click **Area in model view**.

   For more information about view properties, see View properties in drawings (page 912).
4. Open a model view from the model view list.
5. Pick two corners in the model view to define the X and Y dimensions of the drawing view.

   The X and Y directions use the coordinate system of the model view. The view depth is the same as defined in the drawing view properties.

Tekla Structures creates the drawing view.

Create a drawing view of a selected area in a drawing view

You can create a new drawing view of an area that you select in an existing drawing view.

1. Open a drawing.
2. On the **Views** tab, click **Area in drawing view**.
3. Select an area in the drawing view for the new drawing view.
4. Select a location for the new view. A symbol representing the view that you are about to place follows the mouse pointer, so it is easy for you to see where to place the view.

Tekla Structures creates a drawing view using the drawing view properties of the original drawing view. To modify the view properties after the view creation, click the view frame. If the property pane is not open, double-click the view frame. For more information about drawing view properties, see View properties in drawings (page 912).

Delete a drawing view

- To delete a drawing view, select the view in an open drawing and press **Delete** on the keyboard, or right-click and select **Delete**.

Copy, move and link drawing views

You can copy drawing views from other drawings in the current drawing as such or using the original layout. You can also move section and detail views to another drawing through **Document manager**. When you move views to another drawing, the source drawing and the target drawing will contain references to each other. It is also possible to link drawing views from other drawings as such or using the original layout. Linked drawing views update when the original views change.

Copy drawing views from other drawings

1. Open a drawing where you want to copy drawing views from another drawing.
2. Open the **Document manager** by pressing **Ctrl+O**.
3. Select the drawing containing the view you want to copy.

   You can select several drawings.
4. On the **Views** tab, click **From other drawing** and then click one of the following commands:
   - **Copy**: Copy the views from the selected drawings as such in the open drawing. Tekla Structures does not copy the drawing layout.
   - **Copy with layout**: Copy the views and layout from the selected drawings in the open drawing.

NOTE Copied drawing views do not update when the original views change.

Link drawing views from other drawings

1. Open a drawing where you want to link drawing views.
2. Open **Document manager** by pressing **Ctrl+O**.
3. Select the drawing containing the drawing view you want to link.

   You can select several drawings.
4. On the **Views** tab, click **From other drawing** and click one of the following commands:
   - **Link**: Display the views of the selected drawings in the open drawing. Tekla Structures does not copy the drawing layout.
   - **Link with layout**: Display the views and the layout of the selected drawings in the open drawing.

NOTE The changes of the original views are also shown in the linked views. If you modify the linked views, the changes will be lost, for example, when you save the model. Do all the modifications in the original view.

Move drawing views to another drawing

Note that if something changes in the detail in the source drawing, the detail view will be updated in the target drawing. If something changes in the section, the section view in the target drawing will not be updated.

TIP General arrangement drawings are often full of information. You may want to create empty general arrangement drawings and move detail or section views from the original general arrangement drawings to the empty drawing.

1. In an open drawing, select the frame of the drawing view you want to move.
2. Right-click and select **Move to drawing** from the context menu.
3. Select the target drawing from **Document manager**.

   You can select additional drawing views after you have started the **Move to drawing** command and move multiple drawing views in one go. If you have multiple drawing views selected before starting the **Move to drawing** command, the command will not be available.
4. Click **Move** in the **Move view to drawing** dialog box.
5. Close and save the source drawing.

   Tekla Structures moves the selected view to the target drawing and creates references between the source and target drawings.
6. Open the target drawing from the **Document manager**.

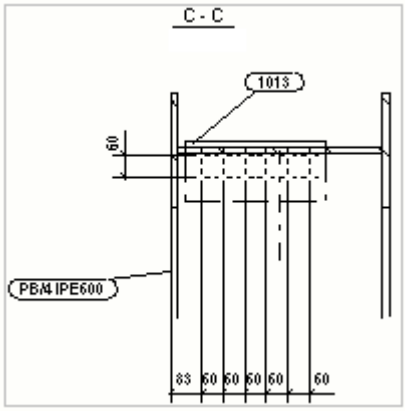
   The **Document manager** indicates that the drawing has been updated.
7. Arrange the drawing views, if necessary. For more information, see Modify, arrange and align drawing views (page 200).
8. Save the target drawing.

Example

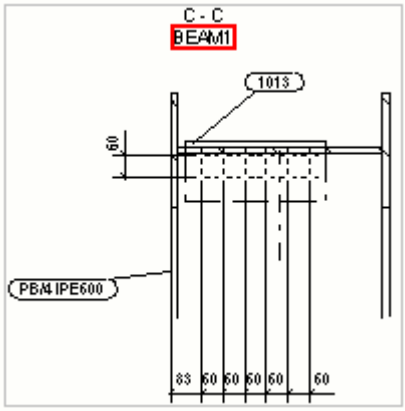
In this example, we have two drawings on the **Document manager**: `BEAM1` and `BEAM2`.

02.01.2018	06.01.2018	584* 410	A	[AB.5]	BEAM1
29.01.2018	06.02.2018	584* 410	A	[AB.6]	BEAM2

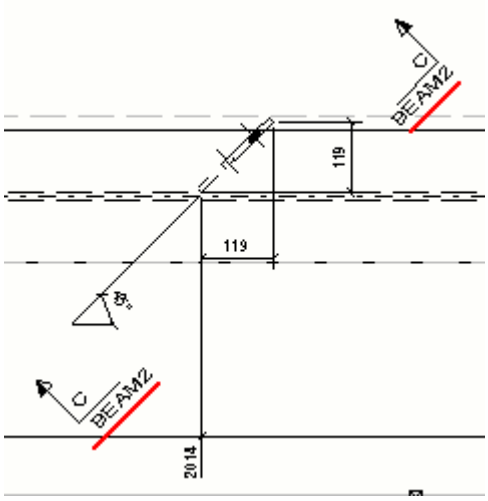
We move section view C-C from drawing `BEAM1` to `BEAM2`. Below is the original section view in the source drawing `BEAM1`.



We move this view to drawing `BEAM2` according to the instructions above. Below is the moved section view in the target drawing `BEAM2`. The view label name contains the name of the source drawing `BEAM1`.



In the source drawing `BEAM1`, the section mark of the moved section view contains a reference to the target drawing `BEAM2`.



TIP You can use the following advanced options to set the reference text for section or detail view labels:

XS_SECTION_VIEW_REFERENCE

XS_SECTION_SYMBOL_REFERENCE

XS_DETAIL_VIEW_REFERENCE

XS_DETAIL_SYMBOL_REFERENCE

Show and modify drawing views in the model

You can show the drawing views of the selected drawings in any of the open model views to get a clearer understanding of exactly where a drawing view is located in the 3D model, the view direction, and the 3D extents of that view. You can then modify the drawing views in the model and save your changes to the drawing.

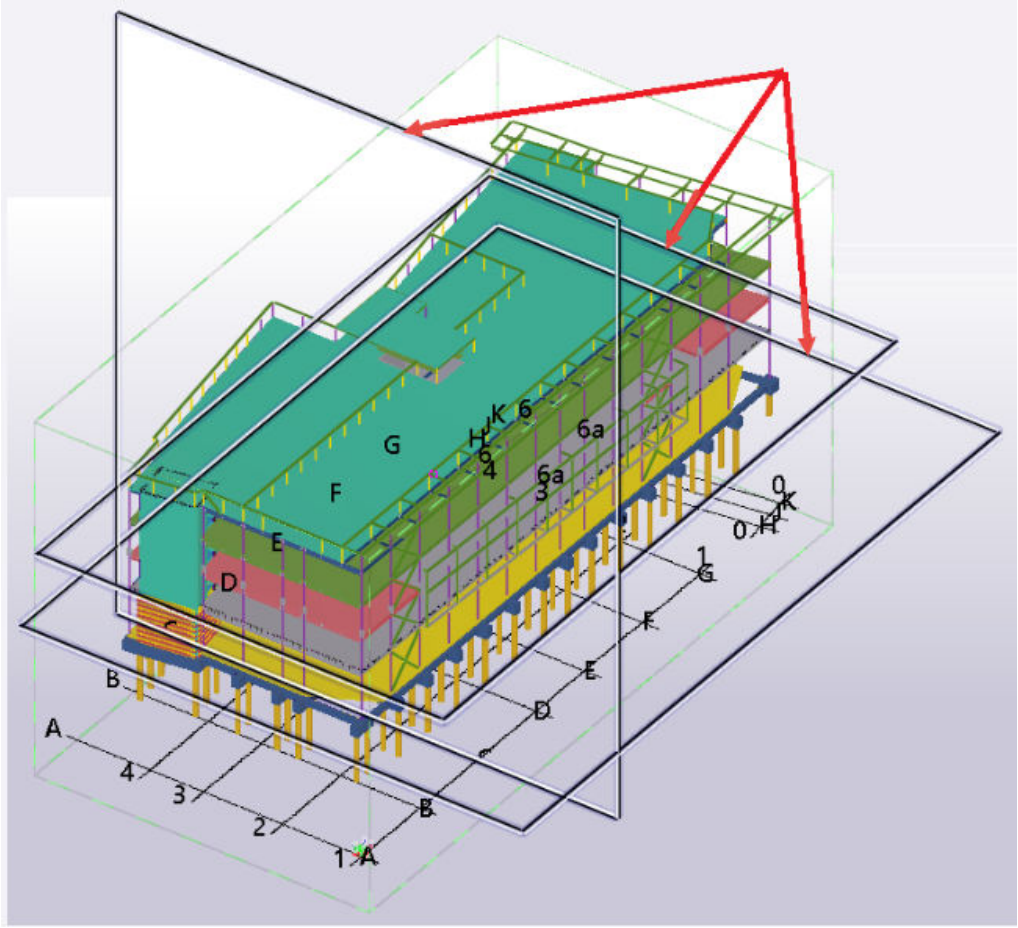
Show and modify drawing views in the model

1. In the modeling mode, click **Drawings & reports** --> **Document manager** to open **Document manager**.
2. Select one or more drawings in the document list.
3. Click the **Show drawing views in the model** button in the top-right corner of the **Document manager** window.

   The button changes color:

   Tekla Structures shows the views of the selected drawings in the model.

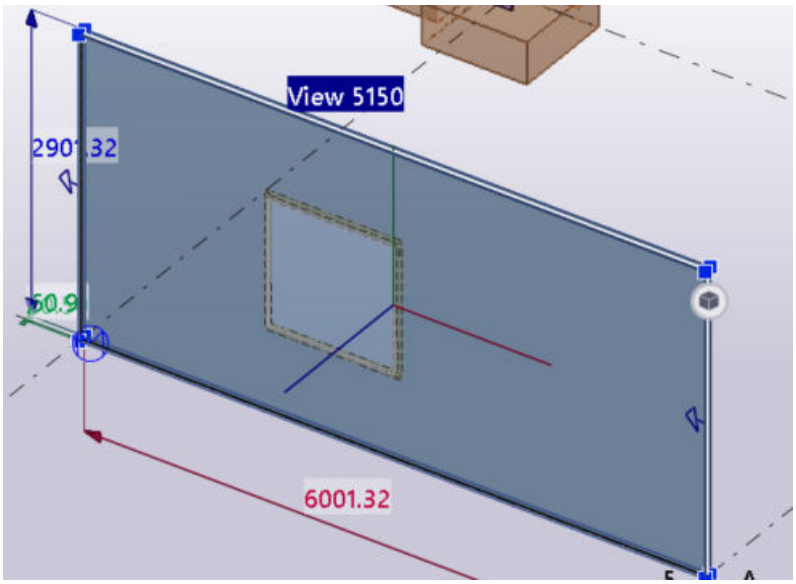
   You can drag the drawing view edges, corners, or faces, or the view plane edges, and modify the views using direct modification.



When **Show drawing views in the model** is active, and you select another drawing or set of drawings, Tekla Structures automatically updates the model view and the drawing views in it. This way you can easily walk through a list of drawings, selecting one drawing at a time, and see the views of that drawing in the model.

4. To show the name of a drawing view in the model, move the mouse pointer over an unselected view edge, or select the drawing view.

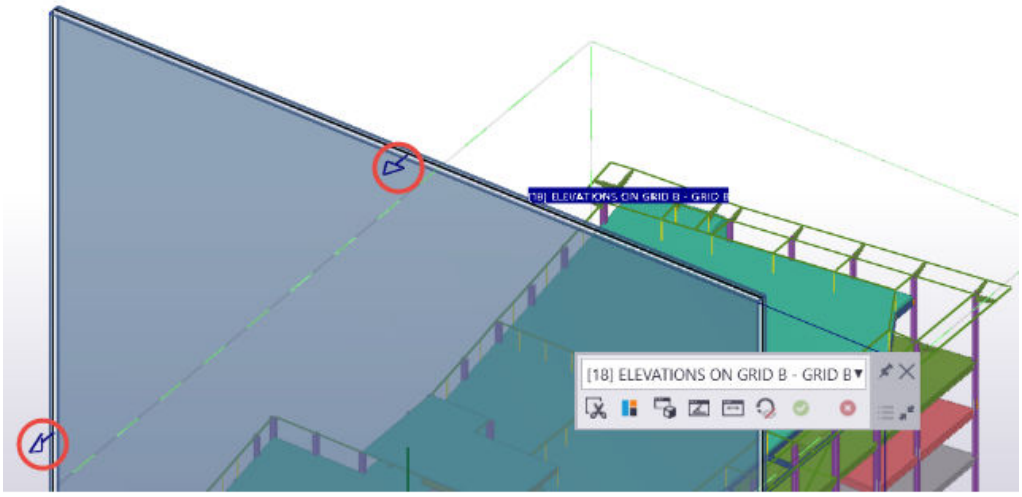
Tekla Structures displays a view label:



5. To show the direct modification handles and dimensions of a drawing view, select the drawing view.

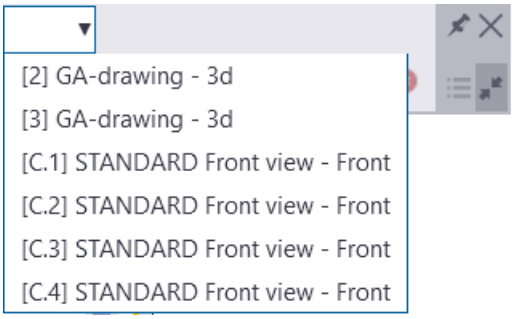
   Tekla Structures shows the view width and height, and depth above and below the view plane.

   The direction of the selected drawing view is shown by open arrows on the four edges of the view plane.

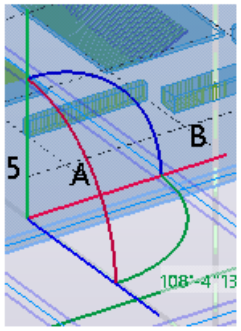


Work with drawing views in the model

Note that when working with the drawing views, you first need to select the desired drawing view in the model.

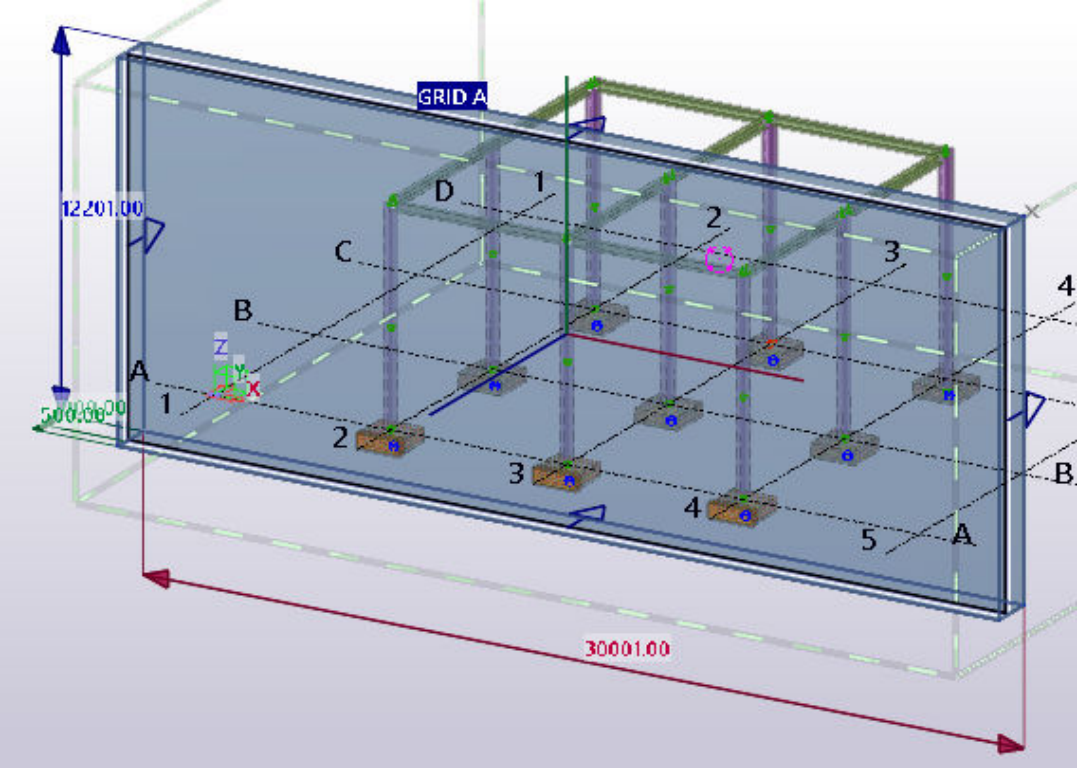
To	Do this
Select a drawing view in the model	Click the drawing view in the model view, or select it from the list on the contextual toolbar.  When you select one drawing view, all other drawing views are hidden in the model.
Deselect a drawing view	Hold down **Ctrl** and click an edge of the selected view.
Increase or decrease the view extents	• Drag the edges, corners, or faces of the drawing view, or the edges of the view plane. • Drag the direct modification handles, or view dimension arrows to change the view size. • You can also enter a new value in the dimension value box to change the view size. Note that you can override the current snap switches by holding down **Shift** while dragging.
Create clip planes	On the contextual toolbar, click **Clipping around selected view bounding box on/off**. Tekla Structures creates six clip planes that are aligned with the drawing view extents. The selected drawing view is isolated and highlighted and the rest of the model is hidden.

To	Do this
	When you modify the drawing view, the clip planes are automatically adjusted to keep their alignment to the view extents. You can also use drawing snapshot overlays (page 170) with the **Show drawing views in the model** command and the clip planes mode to isolate the view.
Emphasize objects intersecting the view bounding box	On the contextual toolbar, click **Emphasize objects intersecting the view bounding box on/off**. Every object that roughly intersects the drawing view is colored red, and every object that roughly does not intersect is made semi-transparent.
Create a temporary model view of the selected drawing view	On the contextual toolbar, click **Display selected view as a new model view**. This creates a new temporary model view based on the properties of the current model view, and sets the camera direction of the new model view to match the camera direction of the selected drawing view.
Set the selected drawing view's camera direction to match the camera direction of the current model view	On the contextual toolbar, click **Change view angle in selected views to match current model view**. The view angle of the drawing view changes to match the view angle of the current model view.
Reverse the drawing view camera direction	On the contextual toolbar, click **Reverse view direction**. The direction of the selected drawing view is shown by open arrows on the four edges of the view plane.
Move or rotate the drawing view	1. On the contextual toolbar, click **Show rotation handles** to display direct modification

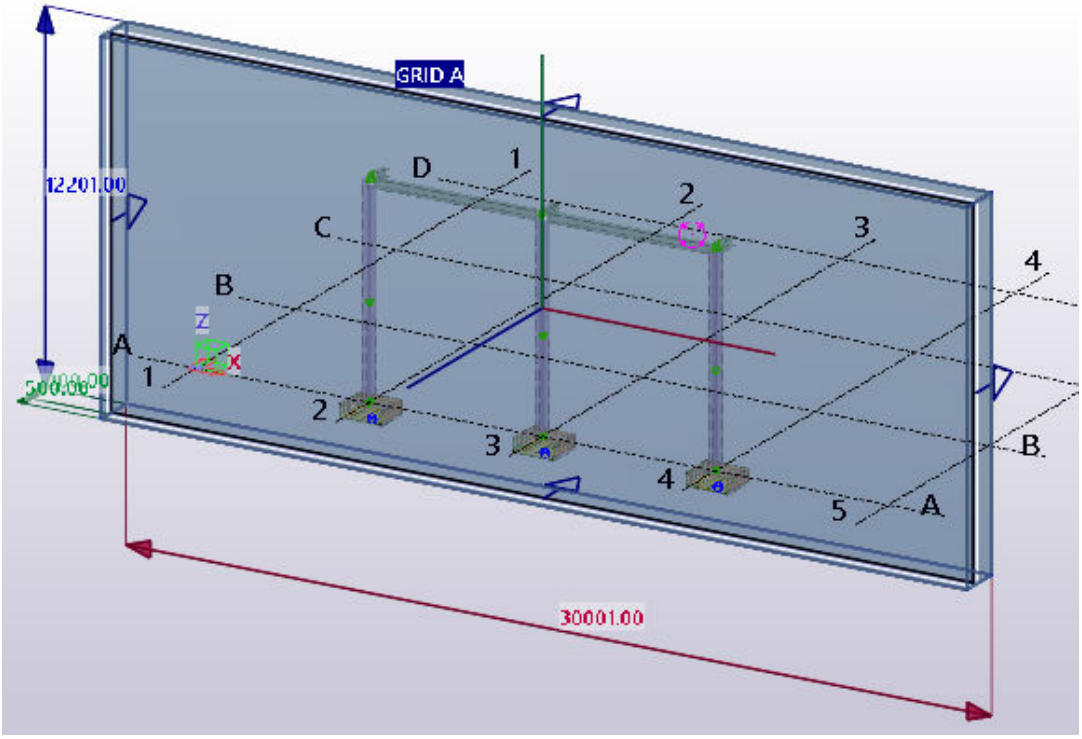
To	Do this
	rotation handles of the drawing view. The rotation handles are hidden by default. 2. Move or rotate the drawing view using the direct modification axis and rotation handles. 
Save any drawing view changes to the drawings	On the contextual toolbar, click **Update drawing**, or click the middle mouse button. Drawing view changes are only saved when you click the **Update drawing** button or the middle mouse button. You can edit a drawing view, then select a different drawing view in the model and edit that, and keep going and then click **Update drawing**.
Discard any drawing view changes	On the contextual toolbar, click **Discard changes**. Any drawing view changes that have not been saved are discarded.
Stop showing drawing views in the model	Press **Esc** or right-click and select **Interrupt**. If you have made changes but have not saved them, you are asked to save the changes or discard them.

Examples of drawing views in the model

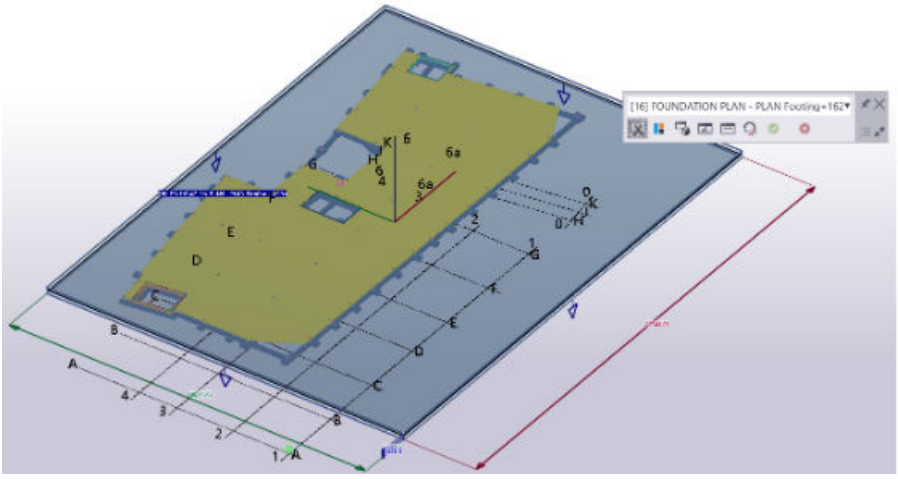
A drawing view is selected in the model:



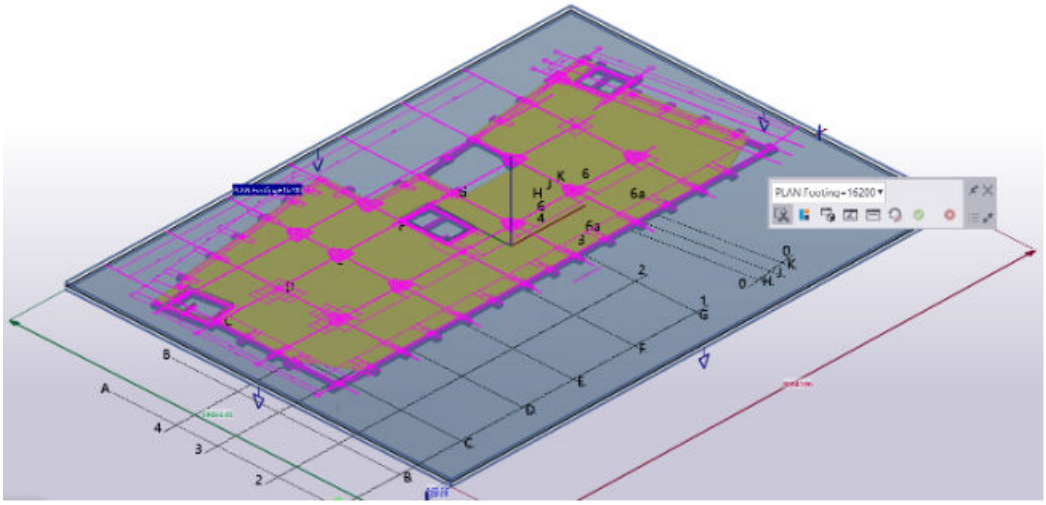
The **Clipping around selected view bounding box on/off** command has been used to isolate the view:



Clip planes have been created in the drawing view of a plan drawing:



Drawing snapshot overlays are used with the **Show drawing views in the model** command and the clip planes mode to isolate the view:



Modify, arrange and align drawing views

Note that you need to set the XS_VIEW_FRAMES_VISIBLE to `TRUE` if you want to show drawing view frames all the time. If you set this advanced option to `FALSE` (default), the drawing view frames are hidden until you move the mouse pointer over the view.

TIP If you want to select several views, hold down **Shift** and click the view frames.

Resize the drawing view boundary

The *view boundary* or *view restriction box* is the dashed frame around the contents of a drawing view, which becomes visible when you click the view frame. You can resize the view boundary, for example, to show just a specific part of the view contents.

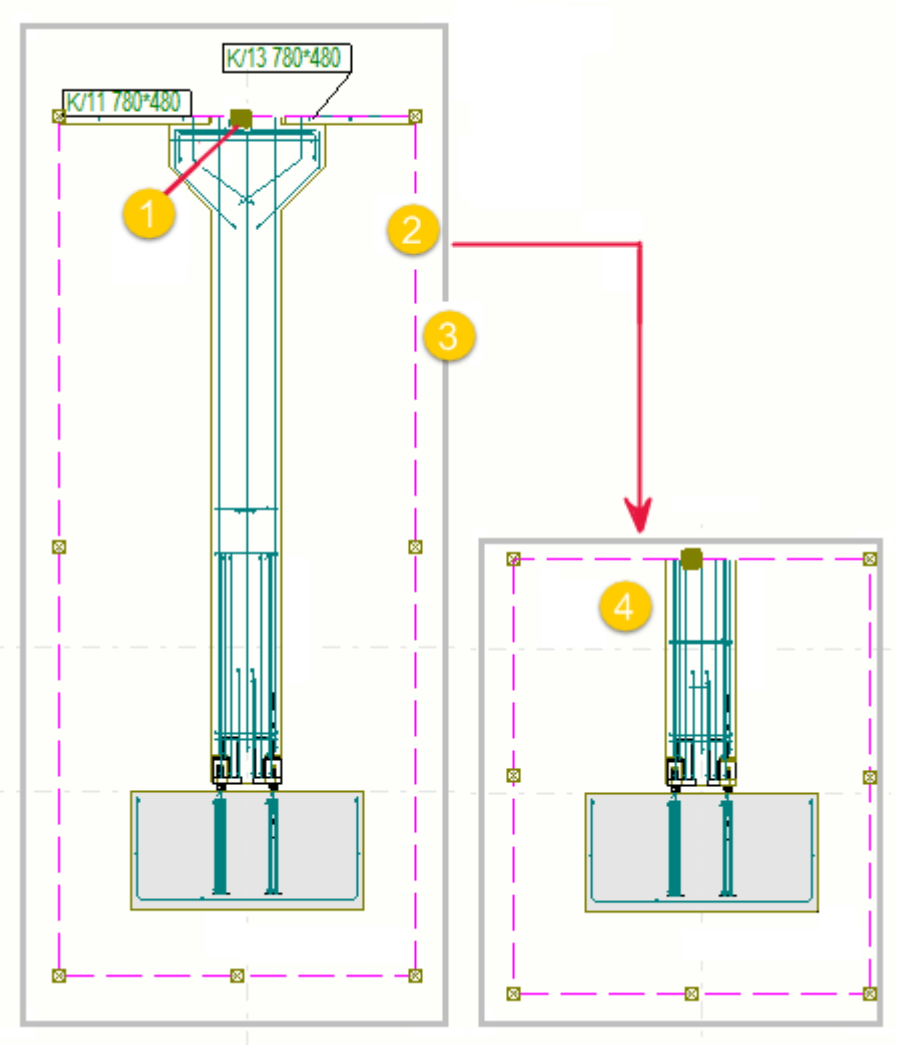
When you select a view, the view boundary is highlighted also in other views. When you change the selected view's boundary in the other views using view boundary handles, the change can be seen in the view boundary of the selected view. This way you can easily adjust the section view plane and view depth, and you do not need to go to the view properties to do this.

1. Click the view frame.
2. Click one of the handles on the view boundary.
3. Drag the handles along the x or y axis of the view.

   When the view boundary changes size, the view frame adjusts with the new boundary size.

   For more information about visualizing views in other views, see the following advanced options:

- XS_VISUALIZE_VIEW_IN_ANOTHER_VIEWS
- XS_VISUALIZE_VIEW_IN_FATHER_VIEW_ONLY



(1) View boundary handle

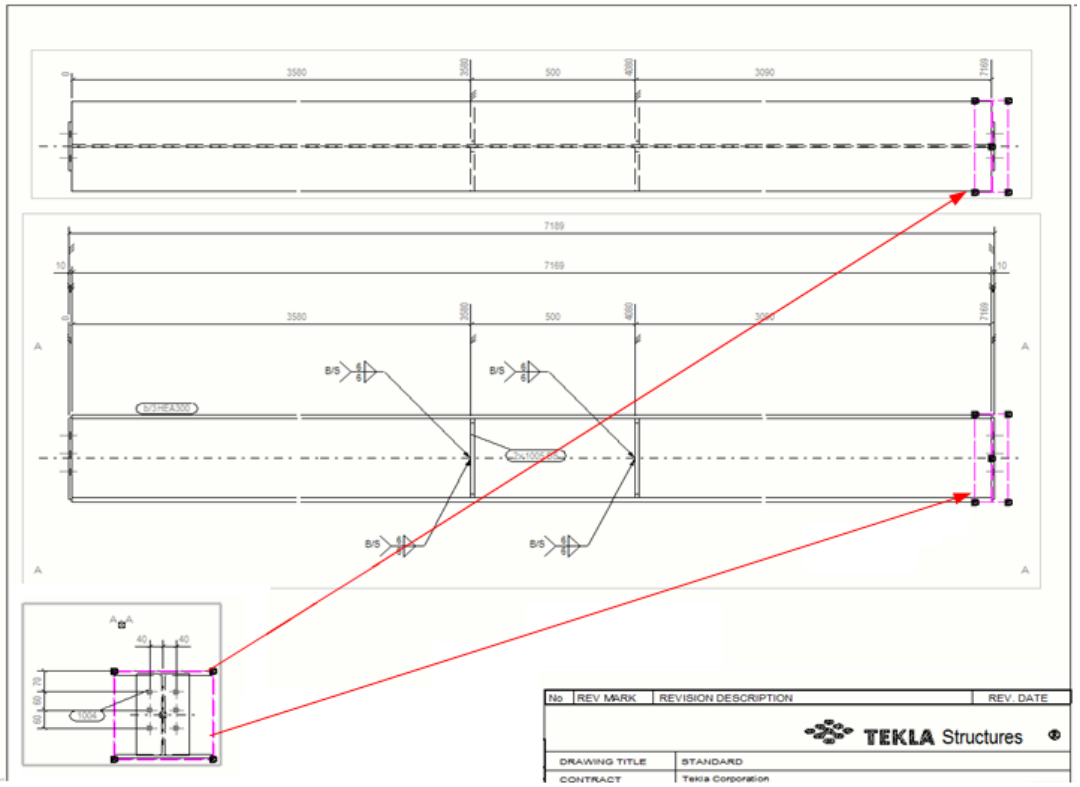
(2) View boundary

(3) View frame

(4) Resized view boundary and view

Example

In the example below, the section view in the bottom-left corner has been selected, and the view boundary is highlighted in two other views. You can change the view boundary of the selected view in the other views by dragging the view boundary handles, to change the section view depth, for example.

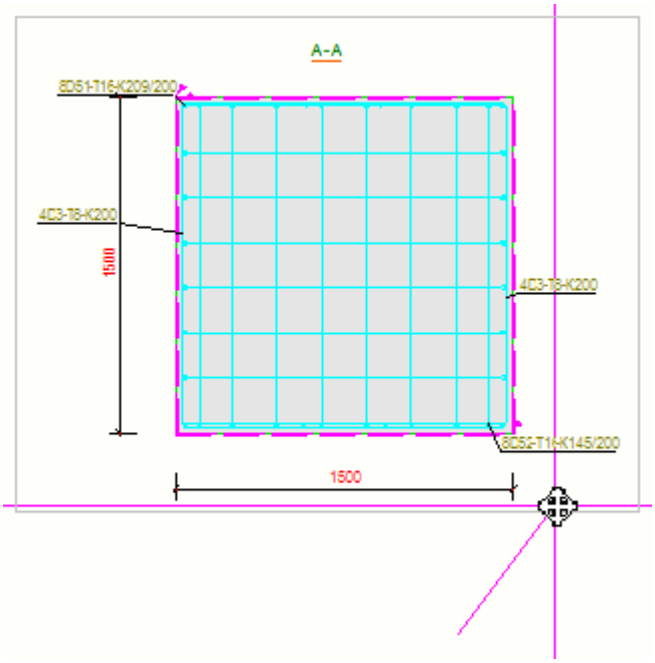


Drag drawing views

You can move drawing views by dragging.

1. Click the view frame of the view you want to drag.

To select several views, hold down **Shift** and click the view frames.



2. Hold down the left mouse button and drag the view to a new location.

   The view follows the cursor while you drag the view, which makes it easier to follow the view movement.

NOTE When you drag a view, its placing setting may change to **Fixed** depending on what is set for the advanced option XS_CHANGE_DRAGGED_VIEWS_TO_FIXED . This advanced option is by default set to `TRUE`, which means that the placing setting changes to **Fixed** when the view is dragged.

Rotate drawing views

You can rotate the drawing views in open drawings.

1. In an open drawing, click the frame of the drawing view you want to rotate.
2. On the **Views** tab on the ribbon, click **Rotate view**.
3. Enter the angle in degrees, for example, `90` or `-90`, in the displayed dialog box and click **Rotate**.

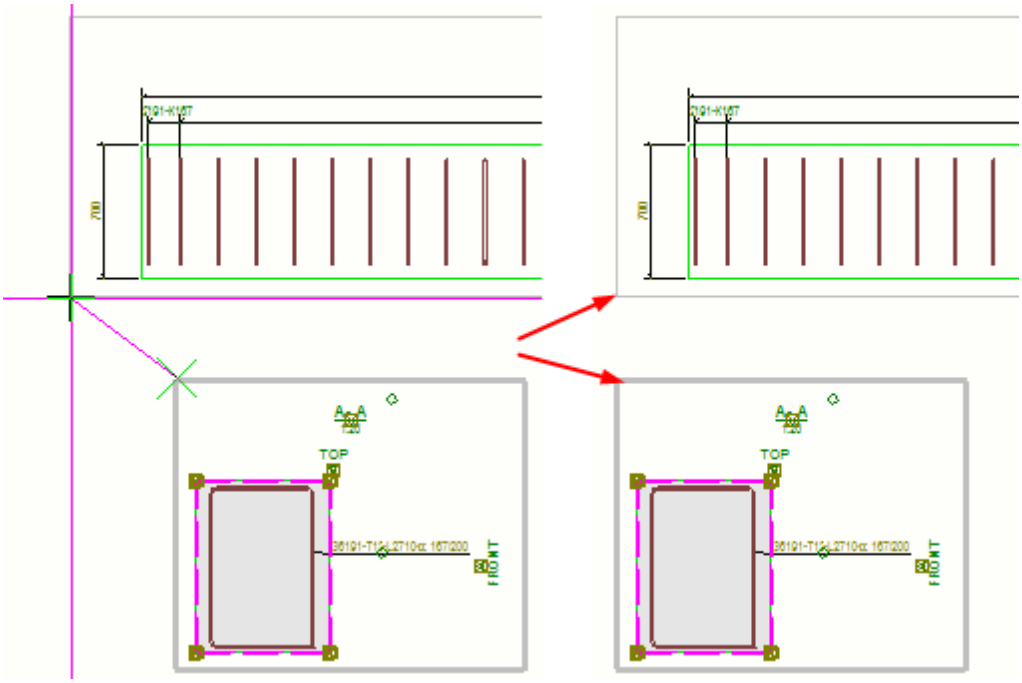
Align drawing views

You can align views vertically or horizontally.

1. Select a drawing view in an open drawing, then right-click to display the pop-up menu.
2. Select **Align vertically** from the pop-up menu.

3. Pick a point in the selected view.
4. Pick a point in the view with which you want to align the selected view.

   Tekla Structures moves the views so that the picked points line up vertically.



You can align the selected drawing view horizontally by selecting the **Align horizontally** command from the pop-up menu.

Arrange drawing views

You can fit the drawing views so that they include all connected objects, and rearrange the drawing views using the drawing's current layout properties.

- On the **Views** tab, click **Arrange**.

NOTE

- **Arrange views** only affects views where the **Place** option is set to **Free** in the view properties. **Fixed** views are not moved.
- If you have set XS_CHANGE_DRAGGED_VIEWS_TO_FIXED to `TRUE` (default) and drag a view in a drawing, the view becomes fixed, and the **Arrange views** command does not work
- **Arrange views** may change the drawing size if you have used automatic scaling of drawing views.

Modify drawing view properties

You can modify view properties in an open drawing for one or more views at a time.

1. Do one of the following:
   - To modify the properties of one view only, click the view frame.
   - To modify the properties of several views, hold down **Ctrl** and click the view frames of the views you want to modify.
   - If the property pane is not open, double-click the view frame.

   The view properties (page 912) are displayed in the property pane.
2. Modify the view properties as required.

   For example, change the view **Scale**.
3. If you selected several views for modifying, and you have different values for a property in the property pane, the text *Varies* is displayed as the value. If you want to have a same value for all views, select a value from the list.
4. If there are some detailed object level settings that you want to apply:
   a. Click the **Object groups** button at the bottom.
   b. Load the detailed object level settings that you want to use and click **Modify**.
   c. Select the **Object group settings** check box.
5. Click **Modify**.

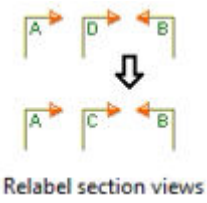
The view properties change according to the changes you made. For more information about drawing view properties, see View properties in drawings (page 912).

Relabel section views or detail views

When you have removed some of the section views or detail views from a drawing, you can use macros to relabel all the section views or detail views to have continuous labeling again (A-A, B-B, etc.).

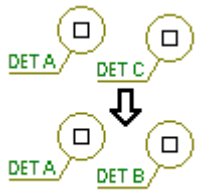
1. In an open drawing where you have removed section views or detail views, click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
2. Click the arrow next to **Applications** to open the applications list. You can also search for **Relabel section views** or **Relabel section views**.
3. Do one of the following depending on whether you want to relabel section views or detail views:

- Double-click **Relabel section views**.



Relabel section views

- Double-click **Relabel detail views**.



Relabel detail views

Section views or detail views are relabeled so that they have continuous labeling again. You will get a message, when the relabeling is completed.

Add single-part views in assembly drawings

In addition to automatically including single-part views in assembly drawings through drawing properties, you can also add single-part drawing views in an open assembly drawing.

1. Open the assembly drawing.
2. In the drawing view, select the parts of which you want to create a single-part view.
3. Right-click and select **Create single-part views**.

Tekla Structures adds the single-part views to the assembly drawing using the current properties in **Assembly Drawing Properties** (**Drawing properties** --> **Assembly drawing** --> **Layout** --> **Other** --> **Single-part attributes**).

See also

XS_NO_END_VIEWS_TO_INCLUDED_SINGLE_DRAWINGS

Include single-part drawings in assembly drawings (page 695)

4.8 Add dimensions in drawings

Tekla Structures contains several tools for modifying dimensions and for adding new dimensions in an open drawing. Some dimensions are created automatically in the drawings, as defined in the drawing properties.

Add manual dimensions

You can manually add dimensions to drawings:

Add dimensions manually (page 207)

Add dimensions manually to general arrangement drawings (page 217)

Add dimensions to reinforcement manually (page 219)

Add closing dimensions to rebar dimensions (page 227)

Dimension rebars with Rebar group dimensioning application (page 228)

Add manual dimensions using User Coordinate System (page 217)

Recreate dimensions for all parts (page 270)

Dimension center of gravity (COG) (page 271)

Add or remove dimension points (page 278)

Modify dimensions

You can modify dimensions manually:

Modify dimensions (page 260)

Add dual dimensions manually (page 260)

Control dimension tag content (page 260)

Exaggerate selected dimensions (page 260)

Show plate side marks on dimension leader lines (page 260)

Change the location of short outside dimension texts (page 260)

Set the dimension extension line lenght (page 260)

Drag dimension marks (page 287)

Move the end of the dimension line (page 288)

Add dimension points in anchor bolt plans (page 285)

Set a new dimension start point (page 277)

Link perpendicular dimension lines (page 286)

Combine dimension lines (page 286)

Customize dimension line arrows (page 275)

See also

Dimension and dimensioning properties in drawings (page 931)

Define dimensioning (page 725)

Show spiral beams in drawings (page 443)

Add dimensions manually

In addition to automatic dimensions defined for the drawing at drawing creation, you can add dimensions manually in an open drawing. Manual dimensioning is not recommended in 3D views.

1. Hold down **Shift** and, on the **Dimensioning** tab, click one of the dimensioning commands:

   Horizontal: Create a dimension in the x direction by picking the points to be dimensioned. X depends on the current UCS.

   Vertical: Create a dimension in the y direction by picking the points to be dimensioned. Y depends on the current UCS.

   Perpendicular: Create a dimension perpendicular to a line. Define the line by picking two points to set the direction of the dimension line. Then pick the points to be dimensioned.

   Orthogonal: Create a dimension in either the x or y direction by picking the points to be dimensioned. Tekla Structures uses the direction of the larger overall distance. X and y depend on the current UCS.

   Curved dimension, orthogonal reference lines: Create a curved dimension with orthogonal reference lines. Pick three points to define the arc, and then pick the points to be dimensioned. The dimension text on the line can be either a distance or an angle value.

   Curved dimension, radial reference lines: Create a curved dimension with radial reference lines. Pick three points to define the arc, and then pick the points to be dimensioned. The dimension text on the line can be either a distance or an angle value.

   Free: Create a dimension parallel to a line between any two points you pick.

   COG: Indicate the location of the center of gravity (COG) in single-part, assembly, and cast unit drawings by creating COG dimensions and adding a COG symbol at the center of gravity. You can also create COG dimensions in section views.

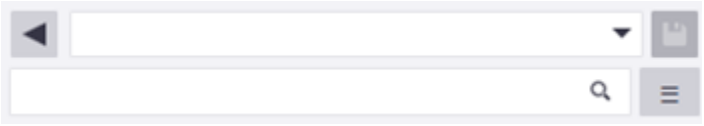
   Parallel: Create a dimension parallel to a line. First pick two points to define the direction of the dimension line, and then pick the points to be dimensioned.

Radial: Create a radial dimension. Pick three points to define the arch, and then pick a position for the dimension.

Angular: Create an angular dimension. Pick the vertex point and two points to define the angle, and then pick the side to place the dimension.

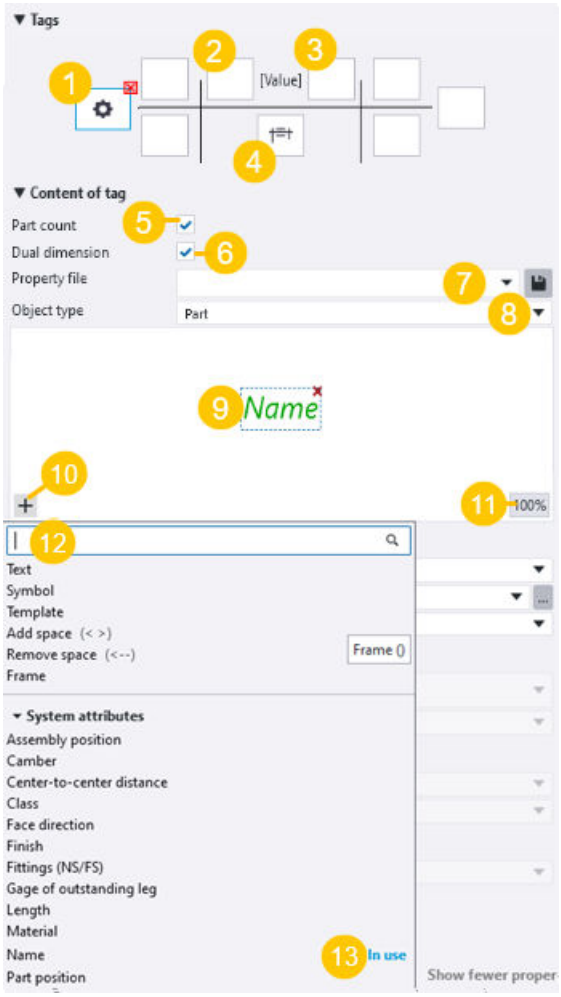
You can also create a dimension by selecting the dimension type from the drawing property pane object list.

2. You can load predefined dimension properties and search for properties:



3. Modify the dimension line and dimension value properties (page 932) as required. For example, define the dimension line and arrow properties, adjust dimension value font, position, background, frame, units, format, and precision, and set the dimension placing properties. You can also select to exaggerate the dimensions, or show plate side marks (page 260).
4. Add dimension tags with the desired contents.

The **Content of tag** section is displayed when you click a tag container.



(1) Available tag containers around the dimension line. Click a tag container to select it. Use the red cross button in the upper-right corner of the container to remove the contents. You can enter text or select the desired elements from the element list.

In tags, you cannot use template attributes such as `MODEL_TOTAL` that refer to the whole model. Tags only check the information from the object in the drawing, and not from the whole model.

(2) Prefix tag

(3) Postfix tag

(4) Dual dimension tag

(5) Part count check box: When you click a tag container, the **Part count** check box becomes available. When you select the check box, the selected tag shows the part count.

(6) Dual dimension check box: When you click the dual dimension tag container **(4)**, the **Dual dimension** check box becomes available. When you select the check box, dual dimensions (page 260) will be applied.

(7) Property file: Load a set of predefined dimension tag contents.

(8) Object type: Define the type of the building object that you are dimensioning. The options are **Part**, **Bolt**, **Rebar**, **Surface treatment**, and **Pour object**. This setting is not available when you have selected a dimension in an open drawing.

(9) Tag editor for adding content in dimension tags. You can drag elements to another location inside one tag container. Click the red cross button in the upper-right corner to delete the element.

(10) Element list. Use the plus button to open the element list, and then select the elements to add in the current tag container.

(11) The % button shows the current zoom level in the preview. You can zoom in or out by scrolling the middle mouse button. Click the button to zoom to best fit.

(12) Search box for searching elements. Enter the element that you want to locate.

(13) If you have already added an element in a tag, the element is marked with text **In use** in the element list.

5. Define the dimension tag properties:
   - Select the tag rotation. Each tag has an independent rotation setting, so you can rotate some tags and leave others unrotated. **Parallel to dimension line** does not rotate the tag, whereas **Perpendicular to dimension line** rotates the tag.
   - Select the tag frame type and color and the text color, font, and height. You can add a frame around the whole tag or around single elements.
   - If you want to show the part count in the tag, select the **Part count** check box.
   - To filter out some of the automatically created tag content (page 260), use **Excluded tags** and select a predefined drawing view filter.
6. Add the dimensions.

   Follow the instructions on the status bar while adding the dimensions.

   It is very important to use correct snap switches (page 53) to get the dimensions correct. You can change the switch temporarily by right-clicking and selecting the switch that best suits for your purpose.

   In dimension types where you click the middle mouse button to place the dimension line, the placement setting affects the result. If you have set **Placing method** to **Fixed**, the position you click with the middle mouse button will be the location of the dimension line. If you have set **Placing** to

Free, the middle mouse button click defines which side of the object the dimension line is located on, and Tekla Structures places the dimension line.

Tekla Structures adds the dimensions by using the current dimension properties.

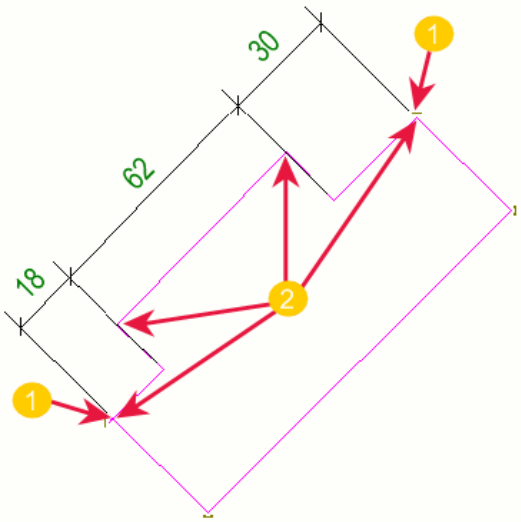
7. To stop adding dimensions, press **Esc**.

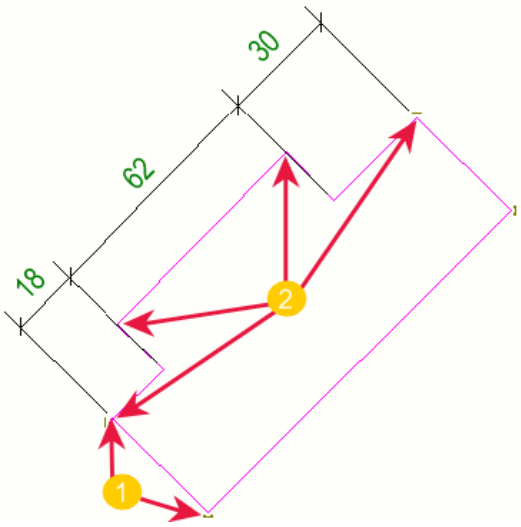
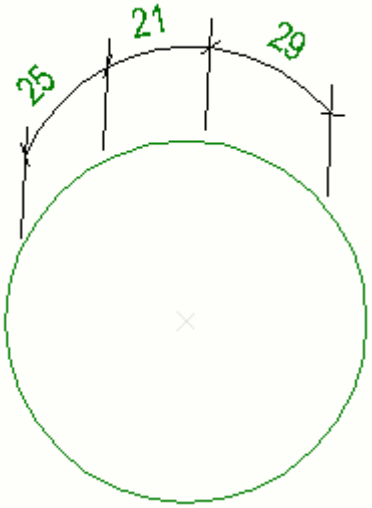
   If you are not happy with the dimensions, you can modify both automatic and manual dimensions (page 260) afterwards. You can also change the dimension location by dragging. When you drag the dimensions, the **Placing** setting changes to fixed by default. You can also drag the dimension mark (page 287), and when you drag it, a leader line is automatically created.

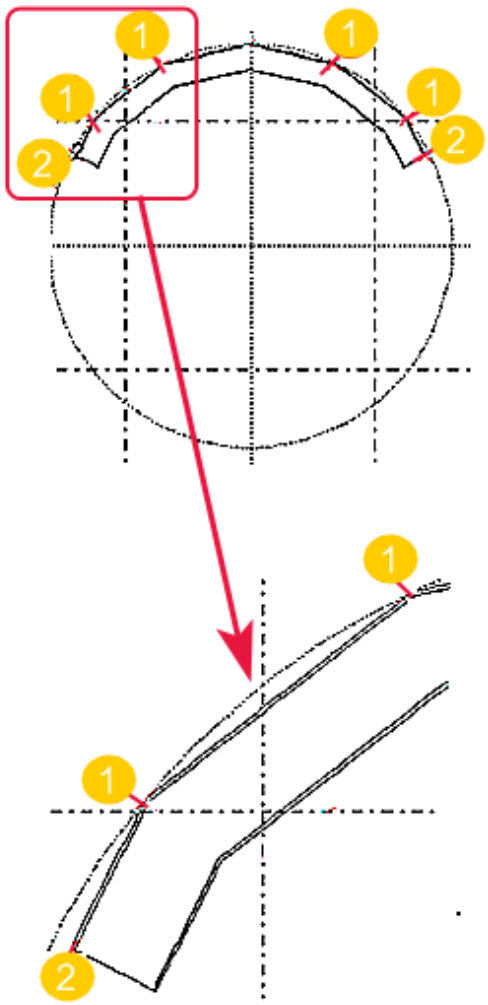
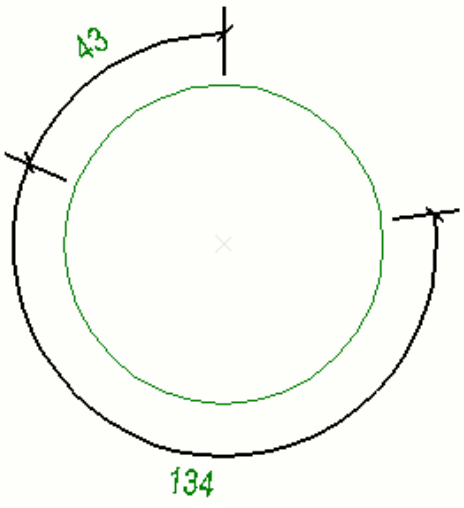
Delete unnecessary dimensions

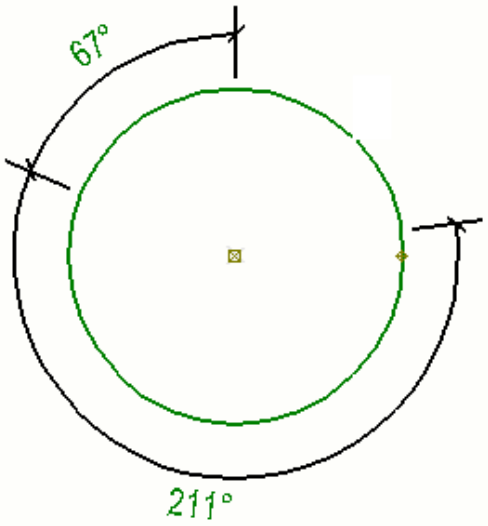
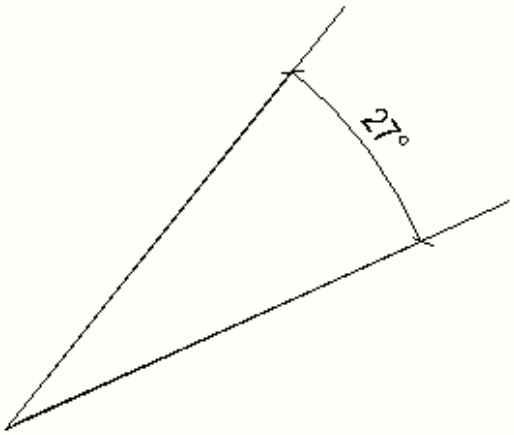
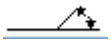
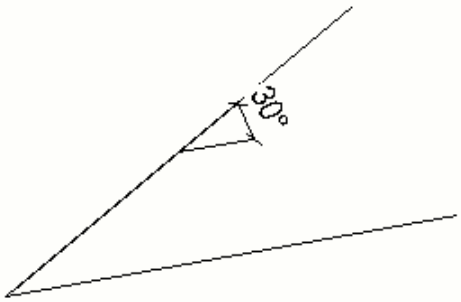
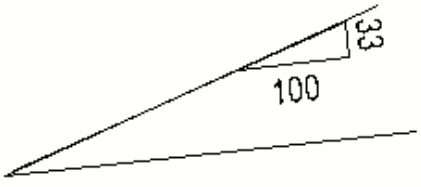
- Select the dimensions and press **Delete** on the keyboard, or right-click and select **Delete**.

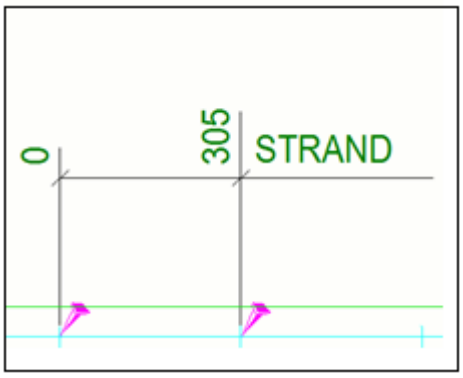
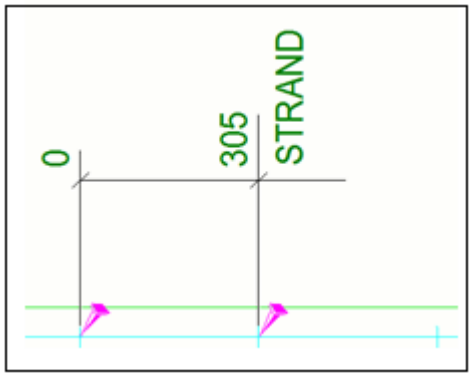
Examples of manual dimensions

Commands and settings	Example
Add parallel dimension **(1)** Shows the edge points picked for defining the direction for the parallel dimension line. **(2)** Shows the picked dimension points picked.	

Commands and settings	Example
Same part as before, now dimensioned with **Add perpendicular dimension** **(1)** Shows the edge points picked for defining the direction for the perpendicular dimension line. **(2)** Shows the picked dimension points.	
Add curved dimension with orthogonal reference lines The dimension mark on the line is a distance value.	

Commands and settings	Example
If the ends of a curved beam or polybeam have been cut or fitted, the points at the ends of the beam do not necessarily lie on the true curve of the beam. This is because curved beams are created with straight segments. The points are indicated with **(2)**. To avoid creating incorrect curved dimensions, pick the three points that define the arc using three of the points indicated with **(1)** . Use end point snap.	
Add curved dimension with radial reference lines **Curved dimension** set to **Distance**. The dimension mark on the line is a distance value.	

Commands and settings	Example
Add curved dimension with radial reference lines **Curved dimension** set to **Angle**. The dimension tag on the line is an angle value.	
Add angular dimension **Angle dimension** set to **Degrees at angle vertex**. 	
Add angular dimension **Angle dimension** set to **Degrees on side**. 	
Add angular dimension **Angle dimension** set to **Triangle**. **Triangle base length** set to 100.	

Commands and settings	Example
Add radial dimension	
Unrotated dimension tag	
Rotated dimension tag	

See also

Modify dimensions (page 260)

Dimension properties (page 932)

Mark elements (page 981)

Define object protection and placement settings in drawings (page 677)

Change the prefix in radial dimensions (page 785)

Add dimensions manually to general arrangement drawings

You can dimension parts to grids along their X, Y, or X and Y axis and along grid lines in plan views in open general arrangement drawings.

Limitation: Parts like beams that are not located along the grids are not dimensioned.

1. Open a general arrangement drawing.
2. To check and change the dimension properties, select **Straight dimension** in the property pane object list and make the necessary changes.
3. On the **Dimensioning** tab, click **GA dimensions** and then click one of the following commands:
   - To dimension along grid lines, click **Add dimensions along grid lines** and pick the view in the location where you want to create the dimensions.
   - To dimension along part X or Y axis or both, select the part, and click **Along part X axis**, **Along part Y axis** or **Along part X and Y axis**. The parts are dimensioned to the two nearest grids.

     Along part X axis:

     Along part Y axis:

     Along X and Y axis:

   Tekla Structures creates the dimensions according to the properties you defined in the dimension properties.

For instructions on how to dimension reinforcement in GA drawings, see Add dimensions to reinforcement manually (page 219).

See also

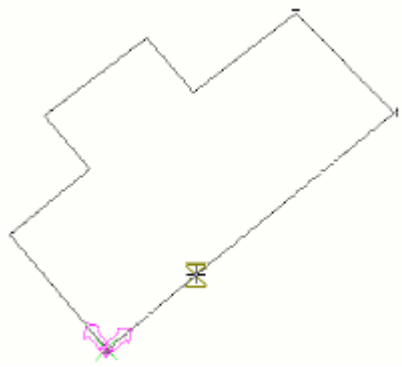
Dimension properties (page 932)

Add manual dimensions using User Coordinate System

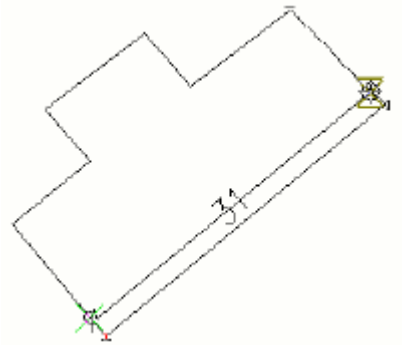
You can pick UCS (user coordinate system) points to define the current coordinate system in the drawing view. If you do this, the dimensioning commands **Horizontal** , **Vertical** and **Orthogonal** follow the defined coordinate system.

1. Open a drawing.
2. On the **Views** tab, click **User coordsys** --> **Set origin by two points**.

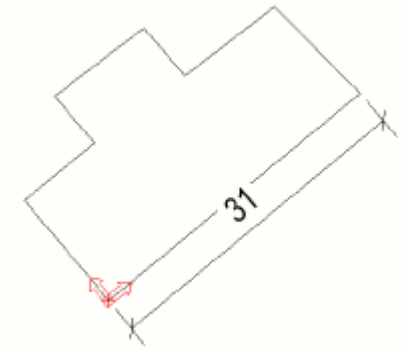
3. Pick the UCS origin and the UCS X direction.



4. Click **Dimensioning** --> **Horizontal** .
5. Pick the start and end point for the dimension.



6. Point to the location where you want to add the dimension line and click the middle mouse button.



NOTE When you use the **Orthogonal** command , it automatically finds out whether the dimension line is in the X or Y direction depending on which one is the closest.

See also

User coordinate system (UCS) (page 548)

Add dimensions manually (page 207)

Add dimensions to reinforcement manually

You can manually add dimensions to reinforcing bar groups.

Each reinforcing bar group may have dimension marks with or without tags, distributed dimension lines, or variants of these styles. The dimensions are created based on the dimension properties defined in the rebar dimension mark properties. The **Rebar** command for creating the rebar dimension marks is available in GA drawings and cast unit drawings.

- You might want to add rebar dimension marks especially in concrete cast unit drawings, where there is only one reinforced cast unit visible.
- The dimension lines or distribution lines show the distribution of the rebars in the group, and typically draw dashed lines from the dimension lines to the rebars when you drag the dimension outside the rebar group. You might want to use distribution lines especially in GA reinforcement drawings, because they may contain a lot of parts with rebar groups, and you often need to show only one rebar from the group and drag the dimension line to a proper place to see everything clearly.
- When adding dimensions, start by using the predefined dimension settings defined in your own environment. You can modify these settings for individual dimensions, if necessary.

The Default environment contains the following rebar dimension settings files you can load in the rebar dimension mark properties:

- `rebar_dimension_line` for creating distribution lines
- `rebar_dimension_mark` for creating dimension marks
- `rebar_tagged_dimension_mark` for creating tagged dimension marks

For more information about typical settings, see section "Typical dimension mark, tagged dimension mark, and dimension line settings" below.

In addition to the rebar dimensioning methods described below, you can also dimension rebars by using the **Rebar group dimensioning** application, see Dimension rebars with Rebar group dimensioning application (page 228).

Add dimensions to rebar groups

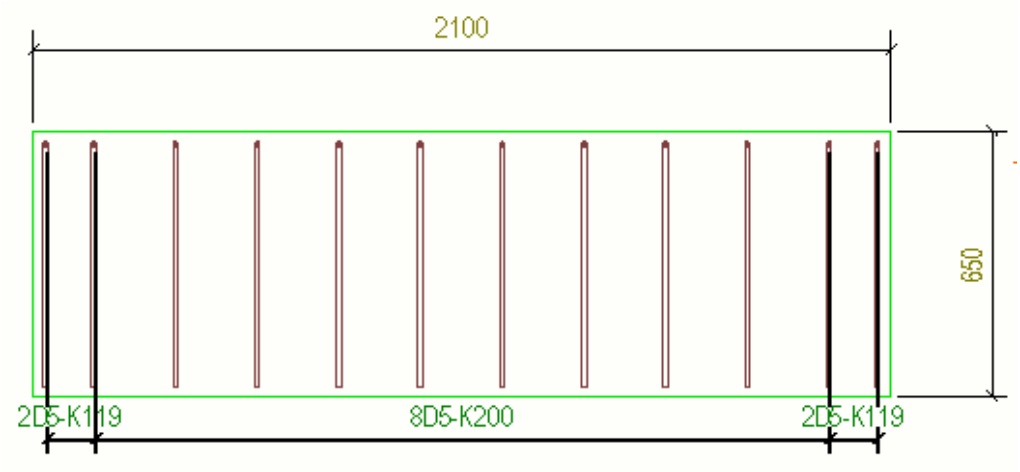
1. To open the rebar dimension mark properties, hold down **Shift** and, on the **Dimensioning** ribbon tab, click the **Rebar** command.

2. Load one of the predefined rebar dimension settings files, go through the rebar dimension properties (page 932), and see if you want to change something.
3. Select a rebar group in the drawing.
4. If you have selected **Fixed** (default) as the **Placing method** option in the rebar dimension properties, you will be asked to pick a position.

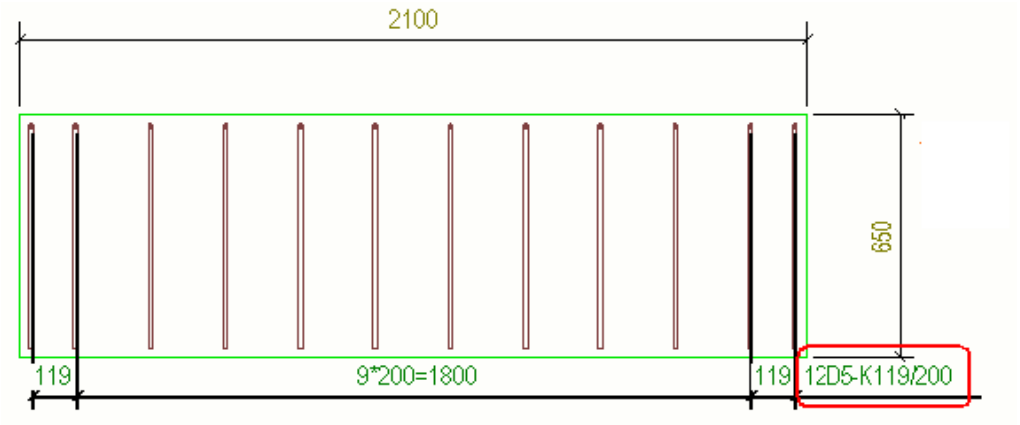
   The rebar dimensions are created. You can continue picking more reinforcing bar groups, or stop creating rebar dimensions by pressing **Esc** on the keyboard.

TIP You can also add rebar dimension marks with the **Add mark** --> **Dimension mark** command on the context menu, or with the **Add rebar dimension mark** command through **Quick Launch**. You can also create a keyboard shortcut for the command.

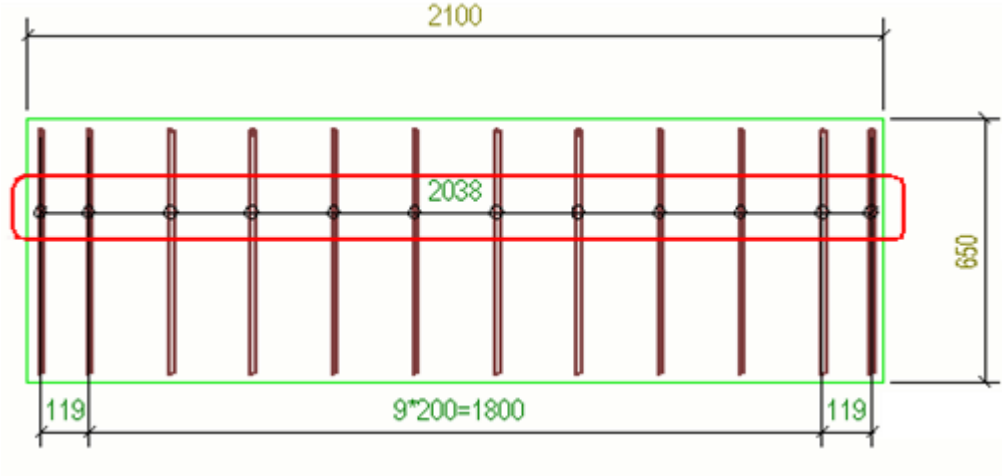
Below is an example of rebar dimension marks:



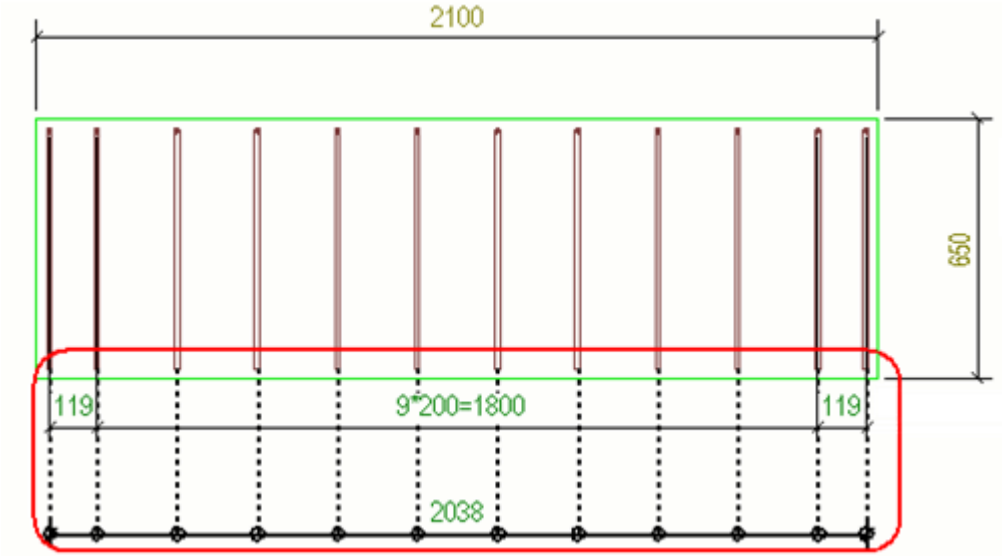
Below is an example of tagged dimension marks:



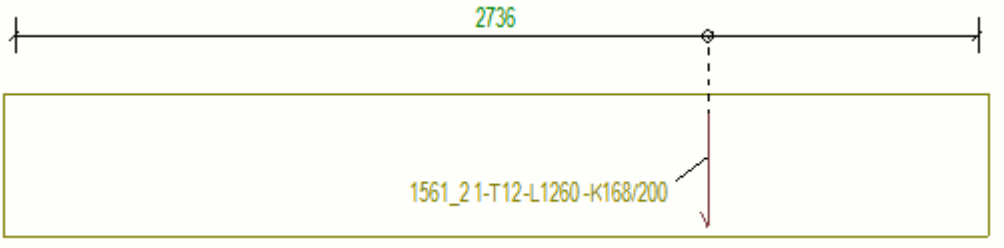
Below is an example of dimension lines:



Below is an example of the dimension line when it has been dragged outside the reinforcing bar group:



Below is an example where only one rebar of the group is visible, and the dimension line has been dragged outside the group.



You can change the representation of the current rebar dimension after creating it by clicking the rebar dimension in an open drawing, and modifying

the dimension properties as required. For example, you might want to add more tags, or select how to align tags in curved dimensions.

Note that you cannot change the following properties when you have a drawing open and you are editing a rebar dimension mark:

Dimension to

Part edge

Tapered dimension type

Curved dimension type

Placing rebar dimensions

During manual dimension creation the **Placing method** setting is considered as follows:

- If you have selected more than one rebar object, the **Dimension mark** command ignores the **Fixed** option and automatically places the rebar dimension mark for each selected rebar object.
- If you have selected only one rebar object, and **Placing method** is set to **Fixed**, you will be prompted to pick a position for the dimension.
- If you have selected only one rebar object, and **Placing method** is set to **Free**, the dimension will be placed automatically.
- With the **Rebar** command on the ribbon, you can create rebar dimension marks without selecting a rebar object first. In this case, you will be prompted to pick a rebar object:
  - If **Placing method** is set to **Fixed**, you will be prompted to pick a position for the dimension, and if **Placing method** is set to **Free**, the dimension will be placed automatically.
  - After the dimension has been placed either manually or automatically, you will be prompted to pick another rebar object.

Typical dimension mark, tagged dimension mark, and dimension line settings

`rebar_dimension_mark.rdim`

Dimension to: **All rebars**

Subgrouping: selected

Extension line to visible rebar: not selected

Combine equal dimensions: **3*60=180**

Minimum number to combine: **1**

Prefix set in dimension tags.

Show numeric value: not selected.

`rebar_tagged_dimension_mark.rdim`

Dimension to: **All rebars**

Subgrouping: selected

Extension line to visible rebar: not selected

Combine equal dimensions: **3*60=180**

Minimum number to combine: **1**

Set the appropriate tag contents.

`rebar_dimension_line.rdim`

Dimension to: **Start and end rebars**

Subgrouping: not selected

Extension line to visible rebar: selected

Combine equal dimensions: **Off**

Create dimension marks to all rebars

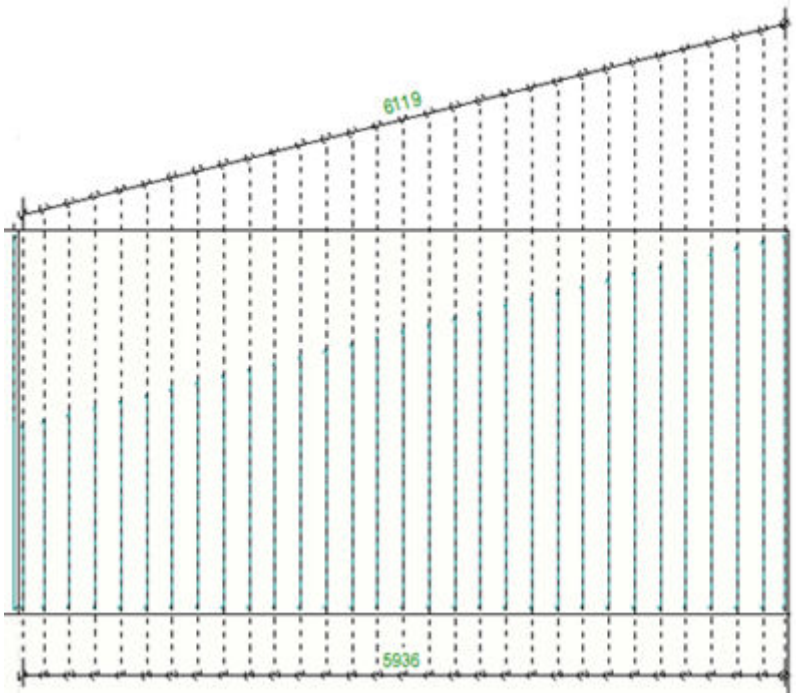
You can add dimension marks to all rebars at one go with the **Create dimension marks to all rebars** application. This application creates any types of rebar dimension marks.

1. Open a rebar drawing.
2. To adjust the rebar dimension mark properties (page 932), open the drawing property pane object list and select **Rebar dimension mark**.
3. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
4. Search for the **Create dimension marks to all rebars** application, and double-click the application.

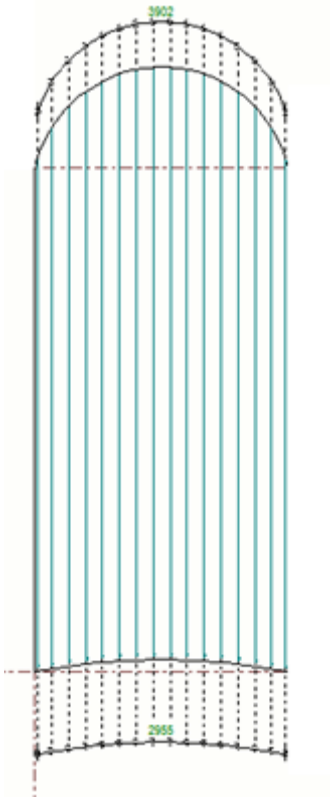
   Rebar dimension marks are created for all rebars according to the defined rebar dimension mark properties.

Examples of rebar dimensions

- Below is a tapered skewed part, and the dimension line follows the shape of the edge that is closest to where you pick. The dimensions have **Skewed** representation.

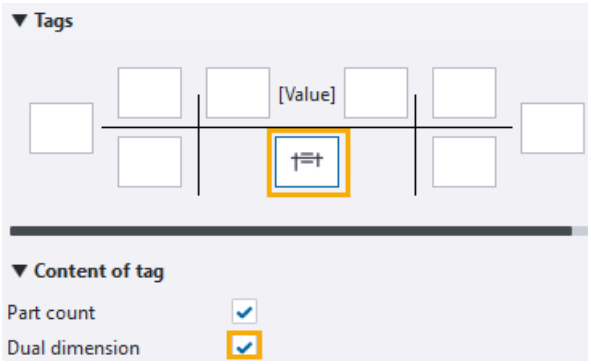


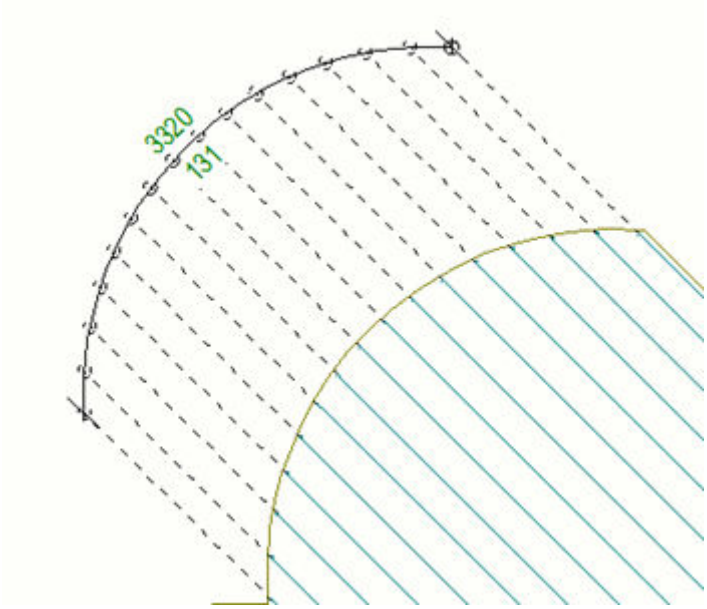
- Below is a tapered curved part, and the dimensions have **Curved** representation.



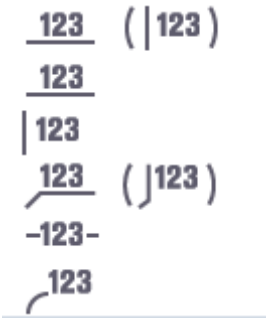
- Below is an example of curved orthogonal dimensions of a tapered curved rebar group with a dimension tag:

- You can also add middle tags in rebar dimensions. Here dual dimensions (page 260) have been applied:



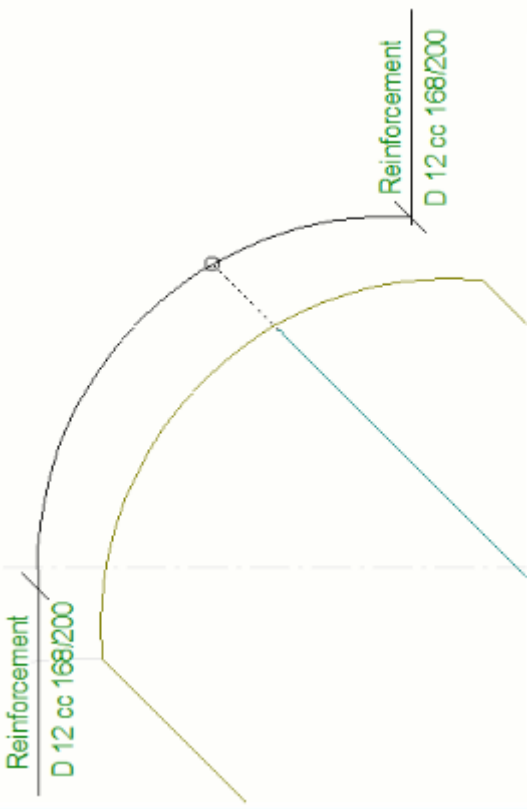


- Curved dimension tags can be aligned by selecting one of the options in the **Curved dimension tag type** list in the dimension properties:



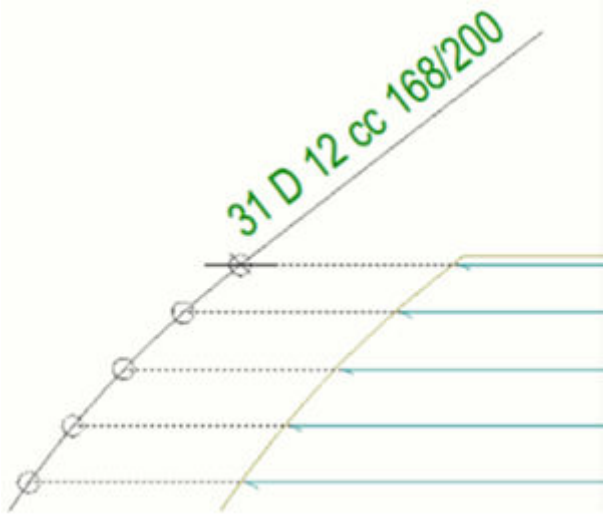
In the example below, only one rebar is visible, and rebar tags are aligned vertically

:

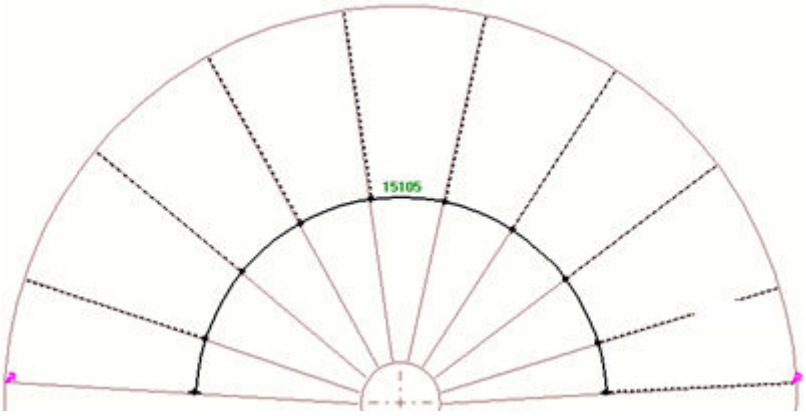


In the example below, the dimension tag follows along the dimension curve

:



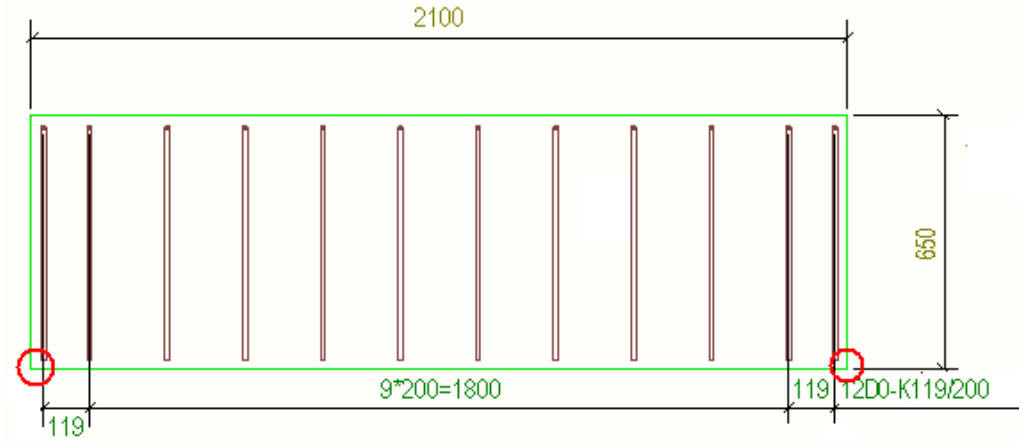
- Below is an example of curved orthogonal dimensions in a radial reinforcing bar group.



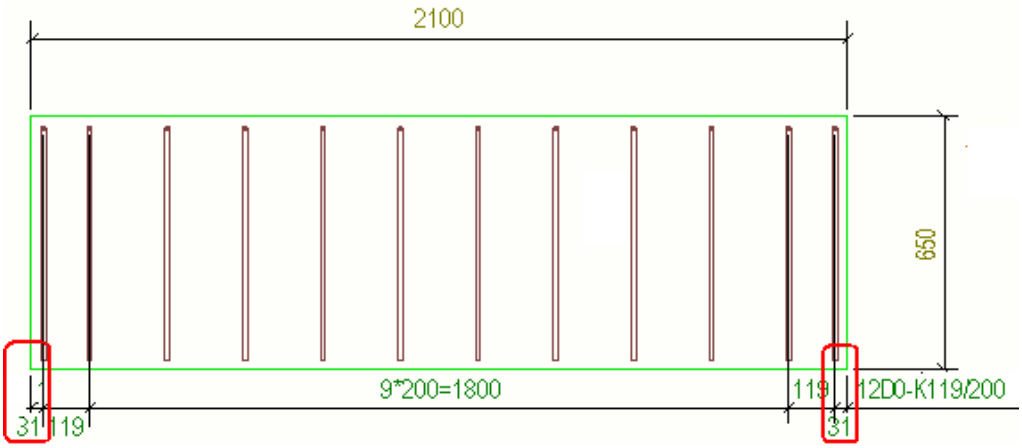
Add closing dimensions to rebar dimensions

In a drawing containing a reinforcing bar group, you can manually add closing dimensions to the edge of the part.

1. Open a cast unit drawing.
2. Select a reinforcing bar group dimension line.
3. On the **Dimensioning** tab, click **Add point**.
4. Select the points where you want to add the closing dimensions.



Tekla Structures creates the closing dimensions.



You can also define closing dimensions in rebar dimension mark properties by selecting the **Part edge** option.

Dimension rebars with Rebar group dimensioning application

The **Rebar group dimensioning** application offers different styles for presenting rebar group dimension lines and dimension marks flexibly. For example, you can mark and dimension multiple stirrups at one go. The **Rebar group dimensioning** helps you to efficiently create quality reinforcement drawings according to the market requirements.

Add dimensions to rebar groups

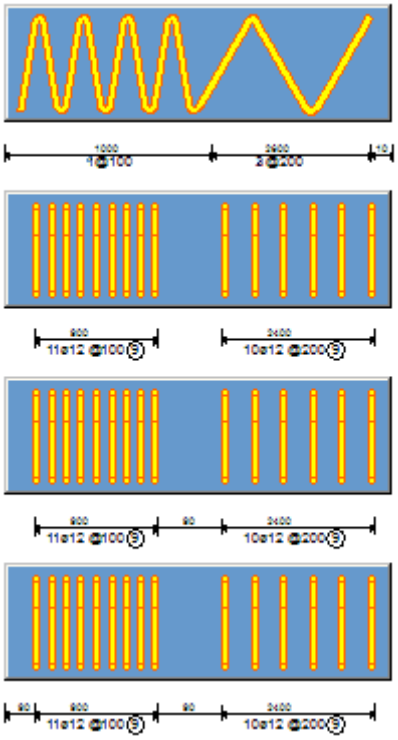
1. In a drawing, select the rebar groups to be dimensioned.
2. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.

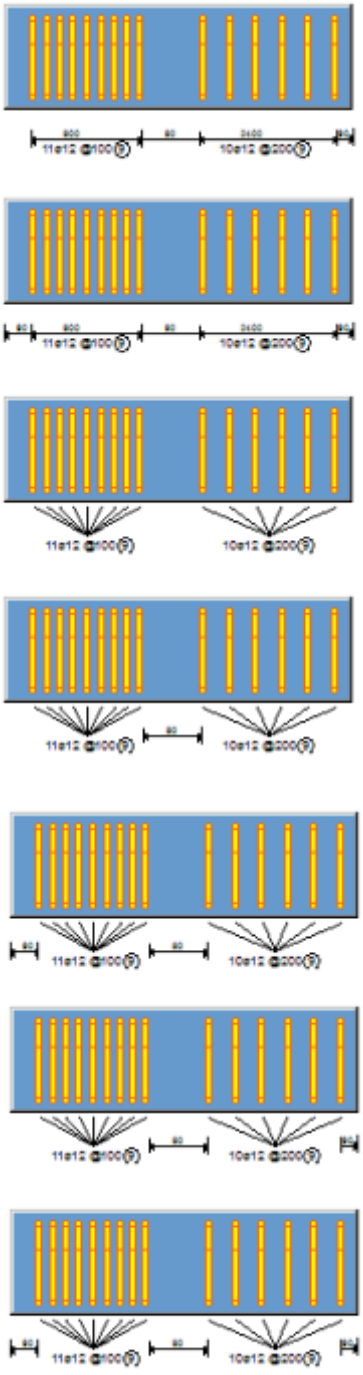
3. Click the arrow next to **Applications** to open the applications list.
4. In the **Applications** list, select **Rebar group dimensioning**.
5. Pick a location for the dimension line in the model.
6. Double-click the dimension to adjust the settings:
   - On the **Parameters** tab, define what the dimensions look like and what it contains. You can also define the content of the stirrup dimension marks.
   - On the **Extra marks in front** and **Extra marks behind** tabs, create an extra mark in front of or behind the dimension line.
   - On the **Advanced settings** tab, define the offsets, spaces and cc distances for the stirrup dimension marks.
   - On the **Rebar lines** tab, define the generation and the appearance of the leader lines belonging to the stirrup dimensions.

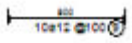
   For more information about the settings, see ***Rebar group dimensioning settings*** below.
7. Click **Modify**.

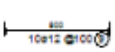
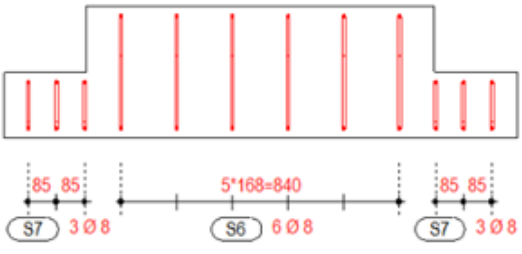
Rebar group dimensioning settings

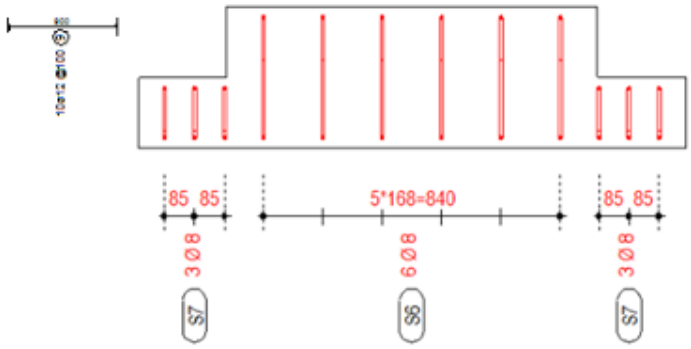
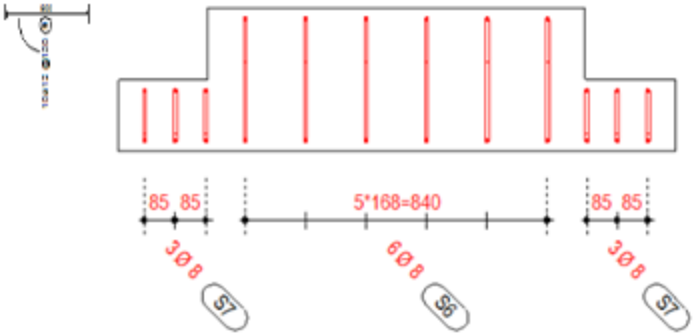
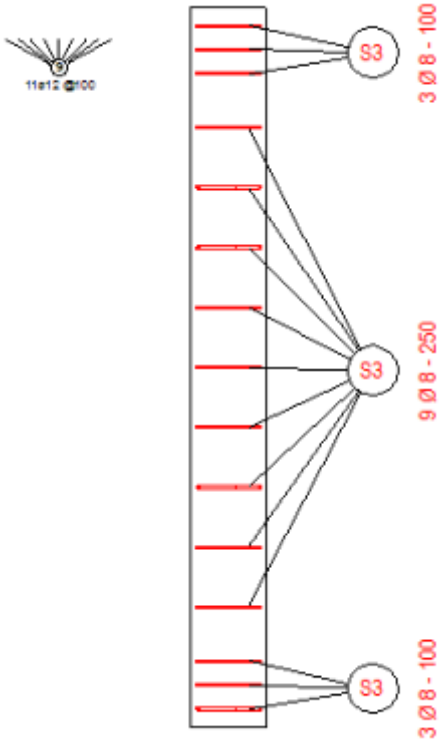
Parameters tab

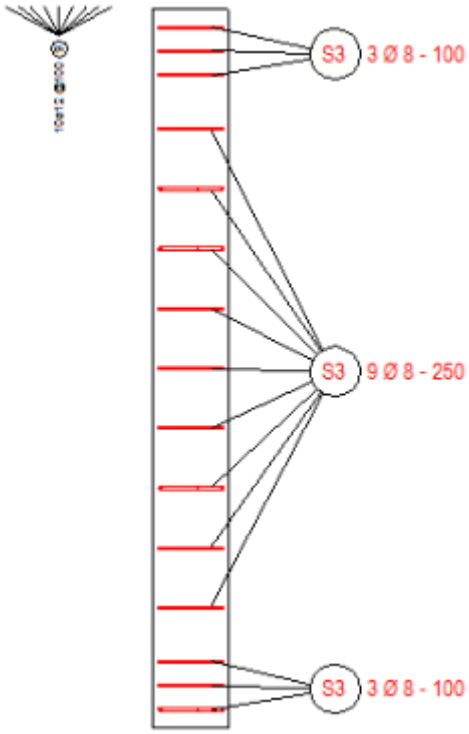
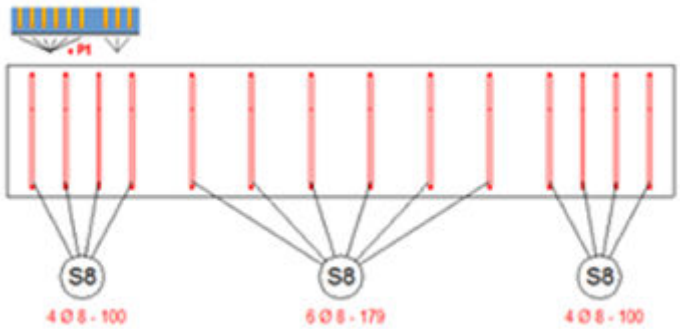
Setting	Options and descriptions
Annotation type	Select the annotation type. The options are: 

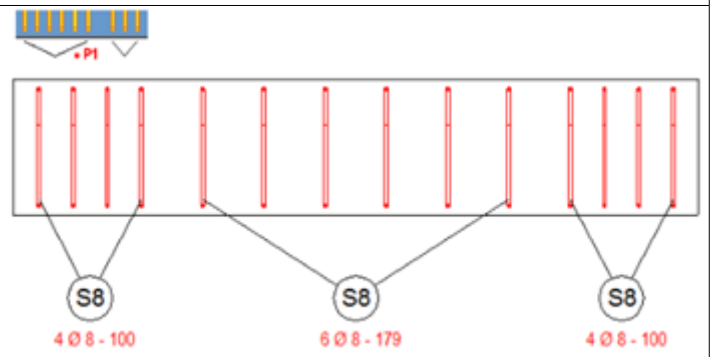
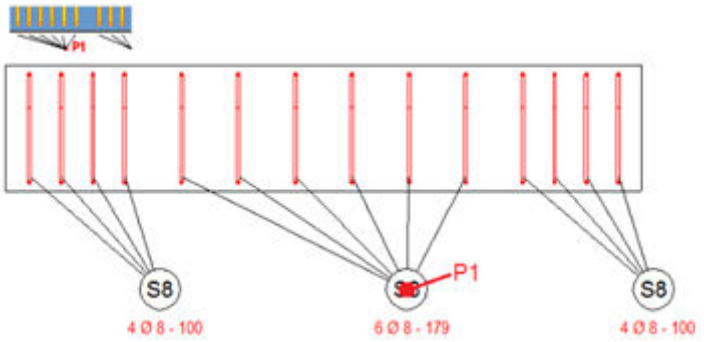
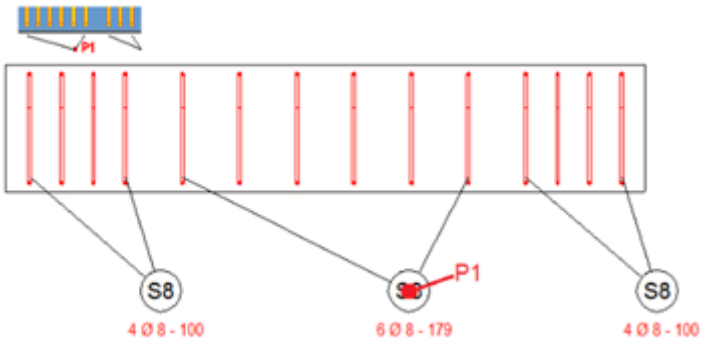
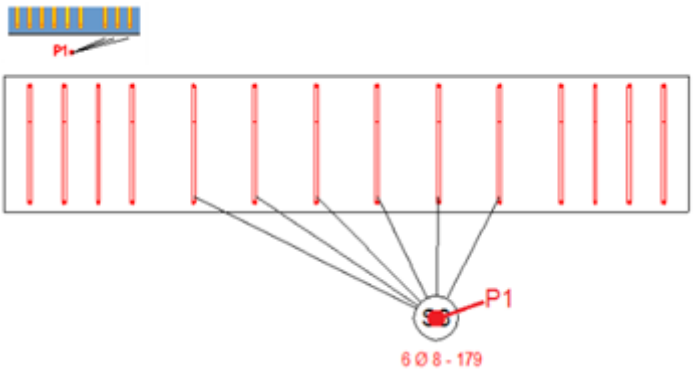
Setting	Options and descriptions
	
	Define the positioning of the mark. The available options depend on the selected **Annotation type**

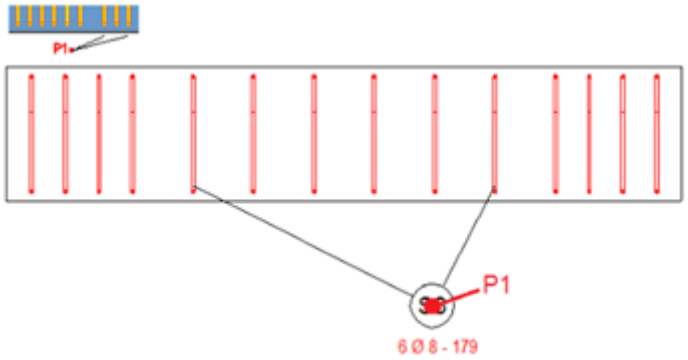
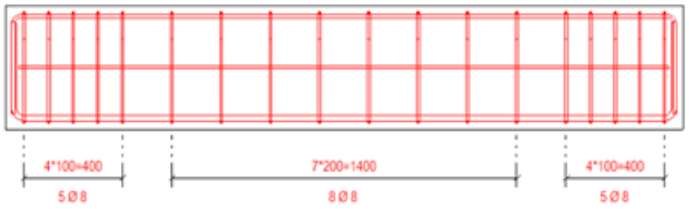
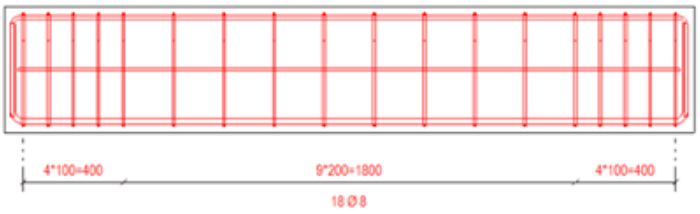
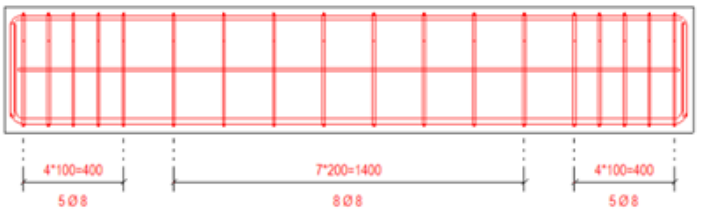
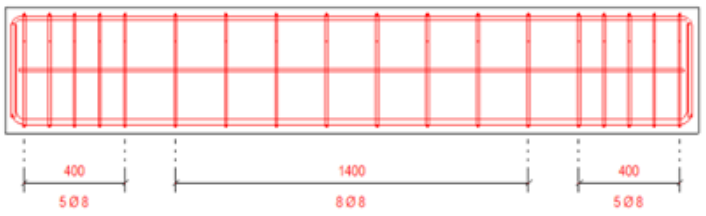
Setting	Options and descriptions
Angle ☑ 90	option. You can also define the mark angle in the **Angle** box. The options are: 

Setting	Options and descriptions
	               Mark position examples:  

Setting	Options and descriptions
	 In the example below, the mark angle has been defined.  

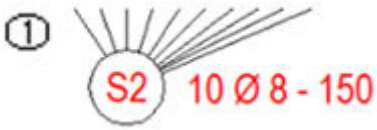
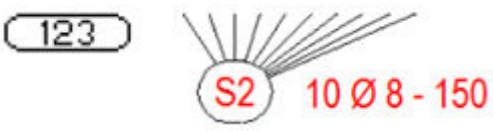
Setting	Options and descriptions
	
 Group number 	Define the number and the location of the mark leader lines. You can also indicate which group you are working with by entering the group number in the **Group number** box. This option is available for certain annotation types only. The options are: 

Setting	Options and descriptions
	   In the following example, group number 2 is defined.  In the following example, group number 2 is defined.

Setting	Options and descriptions
	
Consider irregular spaces as separate groups	**Yes**  **No** 
Dimension each CC distance	**Yes**  **No** 
Combine dim. between rebar - dim.	Allows you to combine the dimensions of the distance between two rebar groups with the

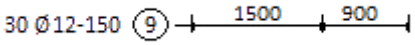
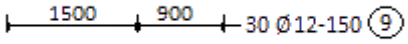
Setting	Options and descriptions
	dimensions of the rebar group where the spacing is the same as the distance between the groups. It also combines the first/last group dimension line with the distance to the part end if the distance to the border is the same as the rebar group spacing. For an example, see the section "More examples" below.
Distribution line properties	Select the desired dimension properties for the displayed dimension line by selecting a dimension properties file. The available properties are the ones that have been defined and saved in Dimension Properties (page 931).
Available elements	Select the information to be displayed in the mark such as grade, diameter and cc distances for mark 1 and mark 2.
Elements in mark	List of the information that you have selected to display in mark 1 and mark 2.
Text properties	Define the text properties. The available properties files are the ones that have been defined and saved in Text properties (page 364).
Position	Select where you want to place the mark. The options are: • **Automatic**: Mark 1 is positioned above the dimension text, when the dimension is above the part, and under the dimension text when the dimension is under the part. • **Above dim. text**: Mark 1 is always positioned above the dimension text. • **Below dim. line**: Mark 1 is always positioned below the dimension text. When placing the mark, the dimension text font size for the above text position and the spacing values defined on the **Advanced settings** tab for both positions are considered. This setting is only available for the non-radial annotation types.
Units	Define the units: • **Automatic** • **mm** • **cm** • **m** • **foot - inch**

Setting	Options and descriptions
	• **inch** Only available for the following mark content: • **Length** • **Cc** • **cc min** • **cc max** • **cc exact** • **cc target** • **Length itemized**
Format	Define the format: • ### • ###[.#] • ###[.##] • ###[.###] • ###.# • ### #/# • ###.## • ###.### Only available for the following mark content: • **Length** • **Cc** • **cc min** • **cc max** • **cc exact** • **cc target** • **Length itemized**
Precision	Define the precision: • 0.00 • 0.50 • 0.33 • 0.25 • 1/8 • 1/16

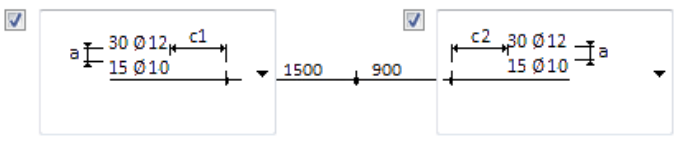
Setting	Options and descriptions
	• 1/32 • 1/10 • 1/100 • 1/1000 For example, for precision 0.33, the actual value 50.40 is displayed as 50.33. 1/8, 1/16, and 1/32 are for imperial units. 1/10, 1/100, and 1/1000 are used for defining precision without rounding.
Sum values A B C **Sum segm rebar axis** **Length TplEd**	These options are only available for the following mark content: • **Length** • **Cc** • **cc min** • **cc max** • **cc exact** • **cc target**
Mark 2 position	Define whether and how the rebar position is displayed in mark 2. The options are: **Not** **Before main mark** **Behind main mark** **Above main mark** **Below main mark**
Mark 2 frame	Select the frame type and color for the mark 2. This option is available for certain annotation types only. The options are:  

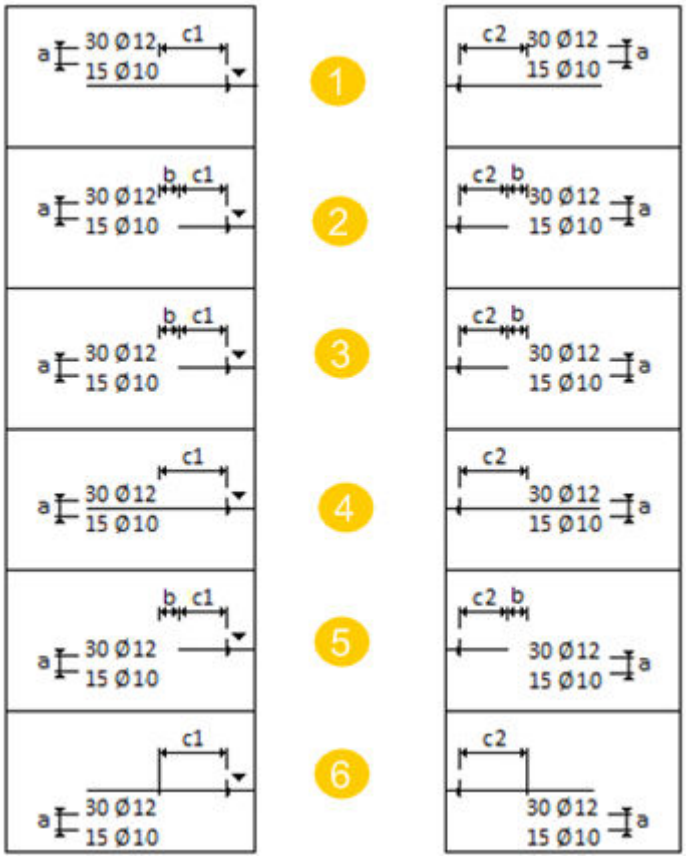
Setting	Options and descriptions

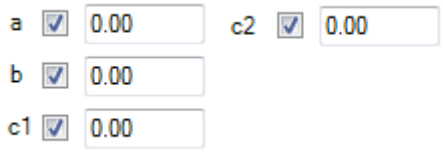
Extra marks in front and Extra marks behind tabs

Setting	Options and descriptions
Marks in front of the dimension line	To create marks in front of the dimension line, select **Yes**. **No** is the default value. 
Marks behind the dimension line	To create marks behind the dimension line, select **Yes**. **No** is the default value. 
Available elements	Select the information to be displayed in the mark in front of or behind the dimensions line.
Elements in mark	List of the information that you have selected to display in the mark in front of or behind the dimensions line.
Text properties	Define the text properties for the marks. The available properties files are the ones that have been defined and saved in Text properties (page 364).
Position	Define whether and how the rebar position is displayed in marks. The options are: **No** **Before main mark** **Behind main mark**
Text properties	Define the text properties for the rebar position. The available properties files are the ones that have been defined and saved in Text properties (page 364).
Number	Select how to show the number of rebars. The options are: **Total number in rebar group** **Number displayed in view** **Total number in drawing**

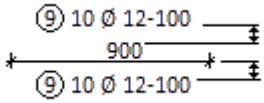
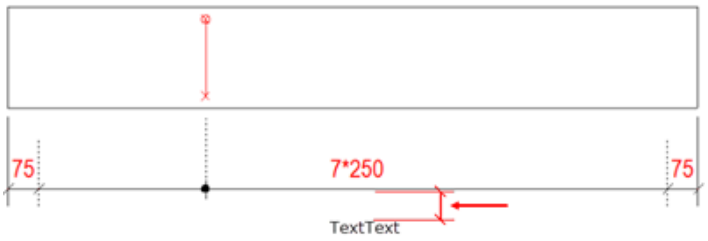
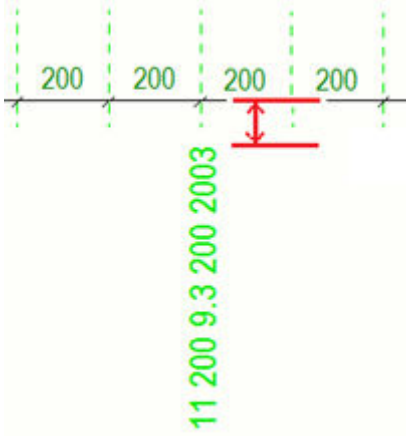
Setting	Options and descriptions
	Total number in cast unit These options are only available for the **Number** element.
Units	Define the units: • **Automatic** • **mm** • **cm** • **m** • **foot - inch** • **inch** Only available for the following content: • **Length** • **Cc** • **cc min** • **cc max** • **cc exact** • **cc target** • **Length itemized**
Format	Define the format: • ### • ###[.#] • ###[.##] • ###[.###] • ###.# • ### #/# • ###.## • ###.### Only available for the following mark content: • **Length** • **Cc** • **cc min** • **cc max**

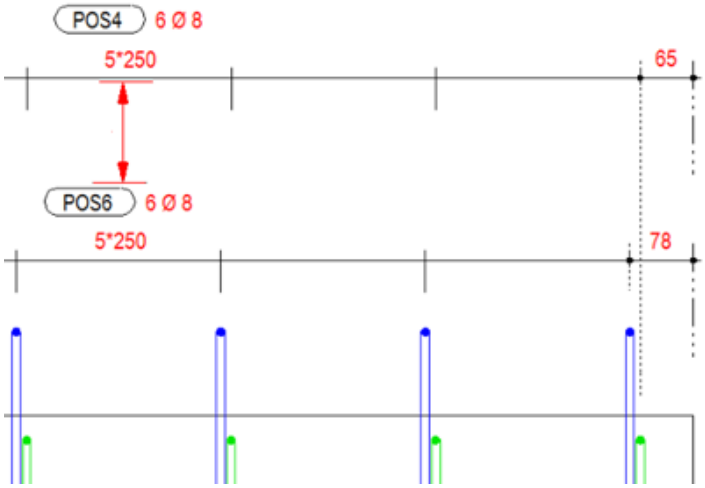
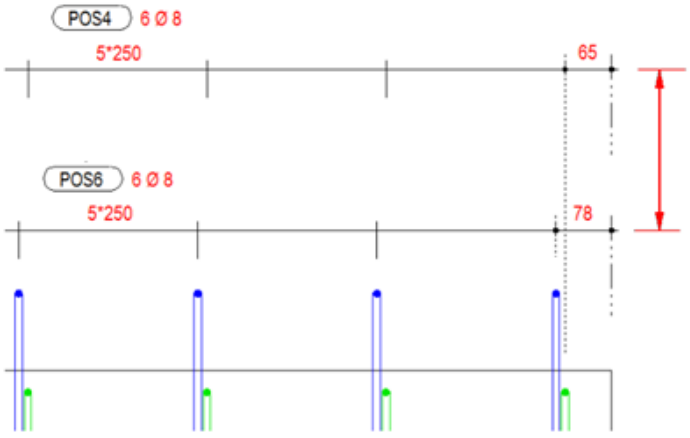
Setting	Options and descriptions
	• **cc exact** • **cc target** • **Length itemized**
Precision	Define the precision: • 0.00 • 0.50 • 0.33 • 0.25 • 1/8 • 1/16 • 1/32 • 1/10 • 1/100 • 1/1000 For example, for precision 0.33, the actual value 50.40 is displayed as 50.33. 1/8, 1/16, and 1/32 are for imperial units. 1/10, 1/100, and 1/1000 are used for defining precision without rounding.
Sum values A B C **Sum segm rebar axis** **Length TplEd**	These options are only available for the following mark content: • **Length** • **Cc** • **cc min** • **cc max** • **cc exact** • **cc target**
Mark placement settings	 The options 3 is the default option.

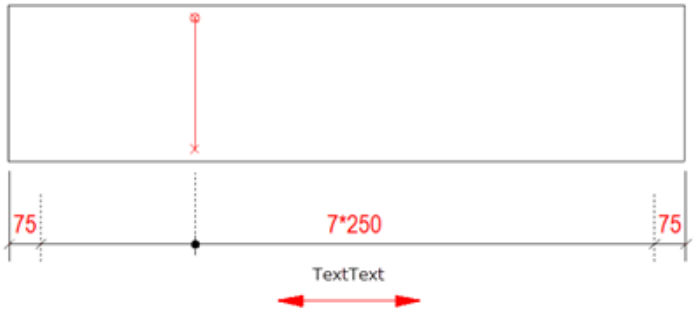
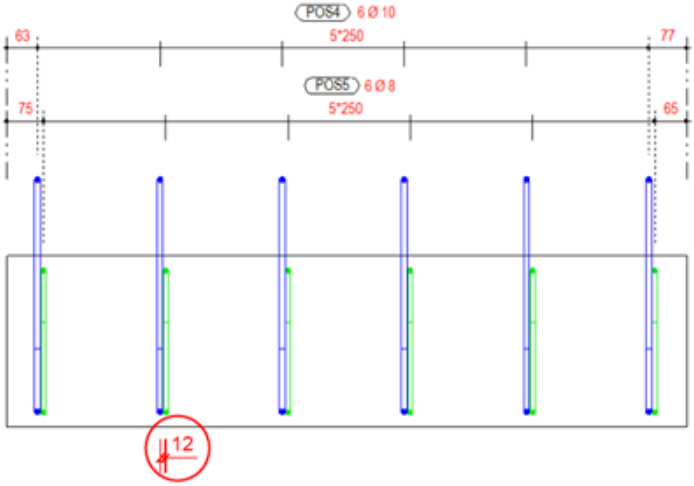
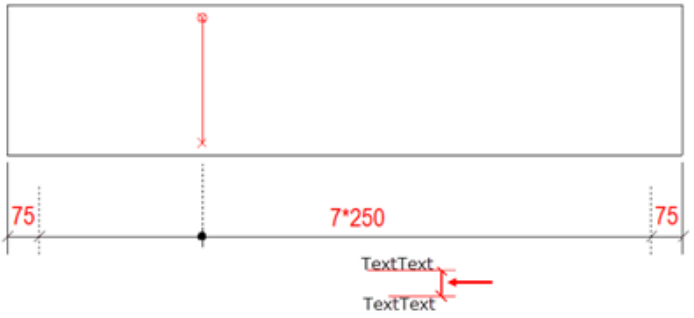
Setting	Options and descriptions
	 (1) All the marks are placed above the dimension line. (2) The middle of the last mark is on the the dimension line. (3) The middle (calculated in the dir. - to the dimension line) of the mark group is on the dimension line. This is the default option. (4) The dimension line is extended between the marks. • If there is only one mark, it is placed above the line. • If there are two marks, one mark is placed above and the other under the line. • If there are three marks, two marks are placed above and one mark is placed under the line. (5) The middle of the first mark is on the dimension line. (6) All marks are placed under the dimension line.

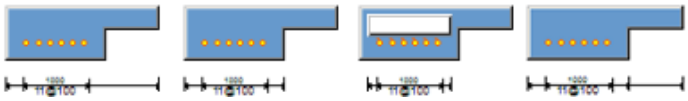
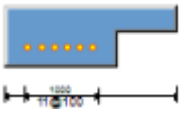
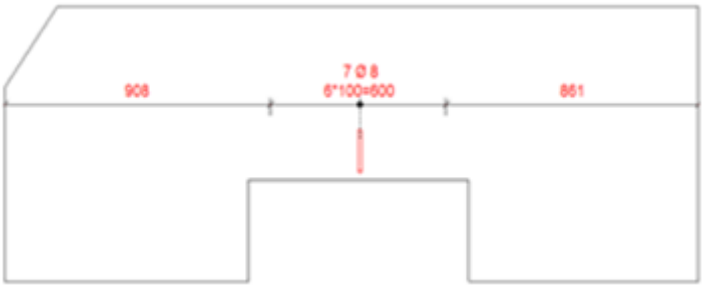
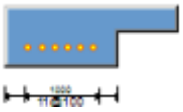
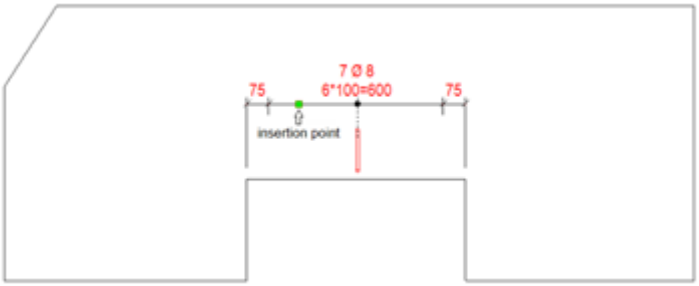
Setting	Options and descriptions
	The parameters a, b, c1 and c2 are needed to get the desired distances between the marks and the dimension line.  The default values are: a = 1 b = 1 c1 = 5 c2 = 5

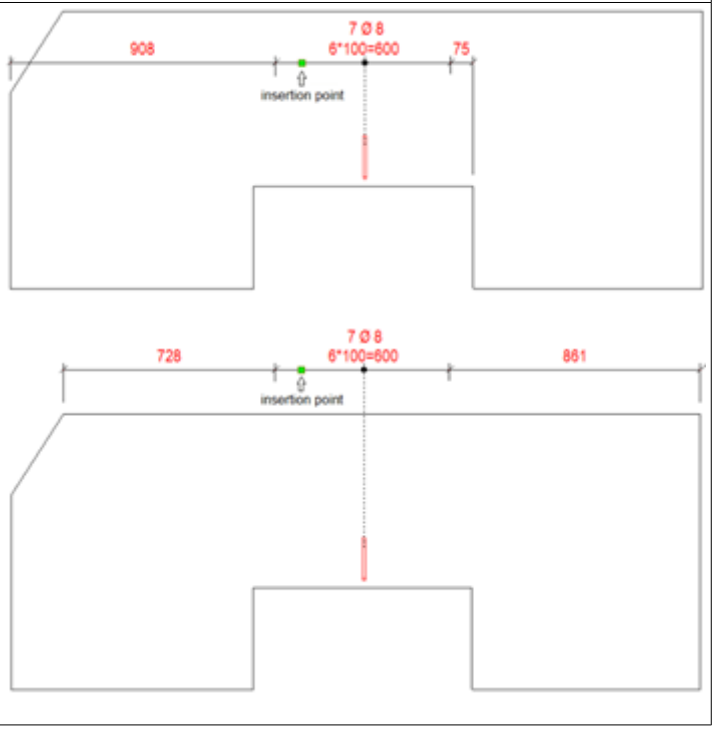
Advanced settings tab

Setting	Options and descriptions
First mark spacing 	Enter a millimeter value to indicate the space between the dimension line and the first line of dimension mark text.  You can also define the first mark distance when the label is below the dimension line. 

Setting	Options and descriptions
Space between Mark1 and Mark2	Define the space between mark 1 and mark 2
Free space below text / **Dimension line spacing**	If you select **Free space below text**, enter a millimeter value to indicate the space between the last line of dimension mark text and the next dimension line.  If you select **Dimension line spacing** and enter a millimeter value to indicate the space between two or more dimension lines. 

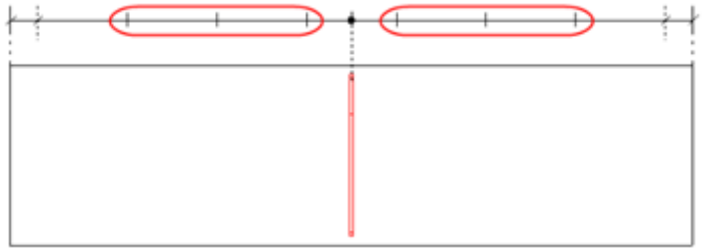
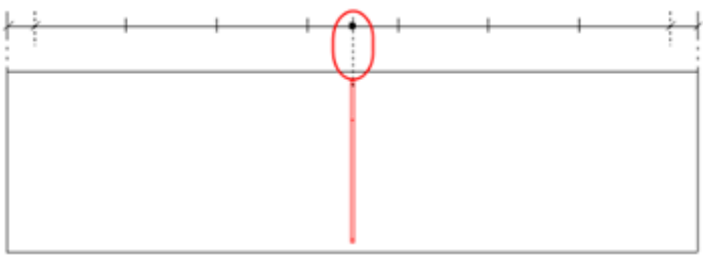
Setting	Options and descriptions
Text offset dimension line	
Group dimensions	Control if dimensions are grouped or not. Grouping also works when the distance between the groups is zero.
Grouping tolerance	Define whether double stirrup groups are combined into one dimension line. If the distance between the stirrups (in the image below 12 mm) is greater than the entered value (=10 mm), two dimension lines are created: 
Group mark spacing	Enter a millimeter value to indicate the space between several lines of dimension mark text. 

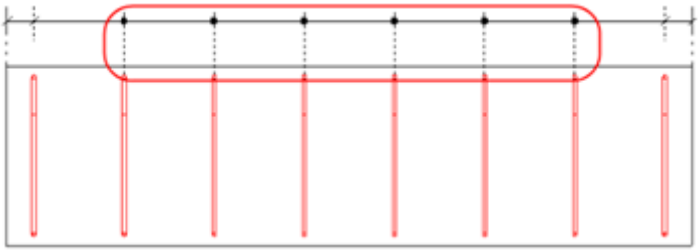
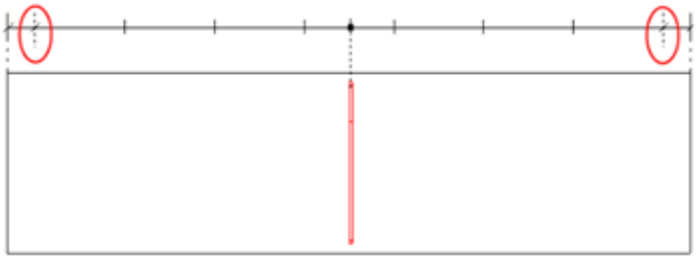
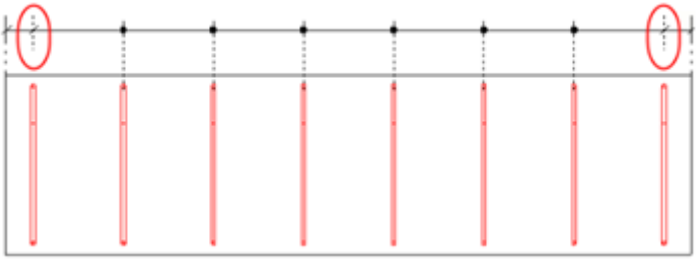
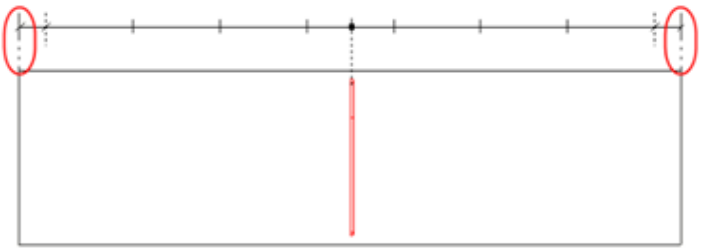
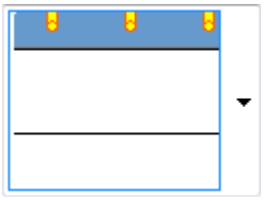
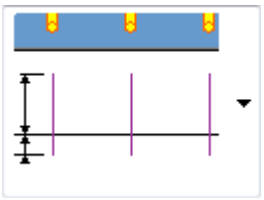
Setting	Options and descriptions
Part extrema	Define how to close the dimension lines on the contour of the concrete part. The following options are available:  Examples:  The dimension line is always positioned on the outermost lines of the concrete part.   The dimension line is positioned to the nearest side/geometry point of the concrete part relative to the picked insertion point of the dimension line (plugin). See the examples below. 

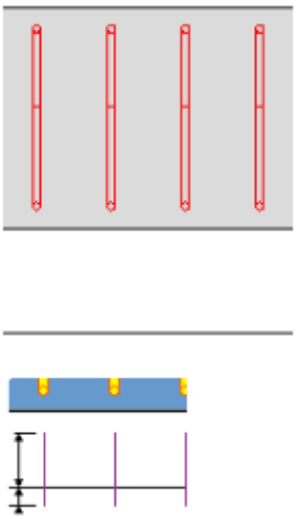
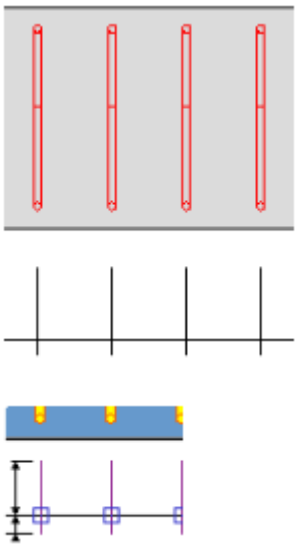
Setting	Options and descriptions
	

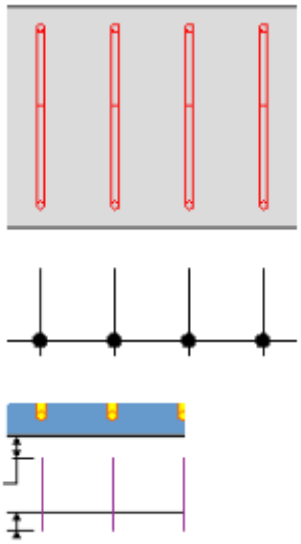
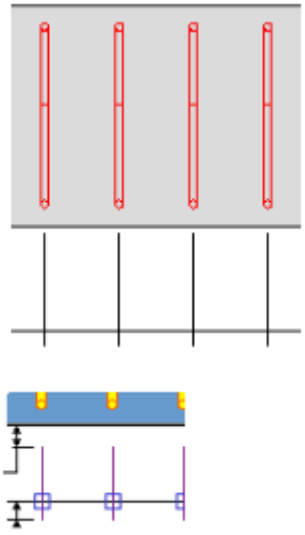
Rebar lines tab

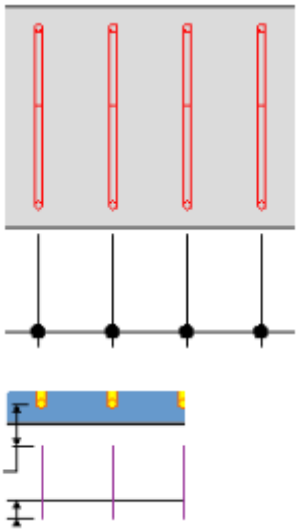
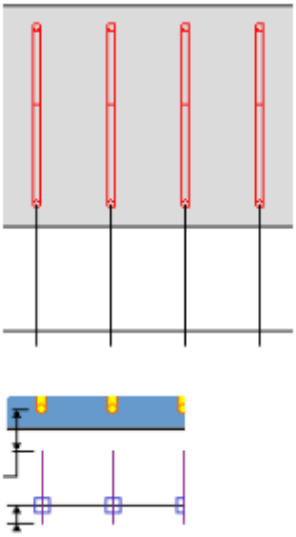
On the **Rebar lines** tab, you can define the generation and the appearance of the leader lines and symbols of the dimension line. You adjust the settings of **Not visualized rebars**, **Visualized rebars**, **Group end rebars** and **Part edges**.

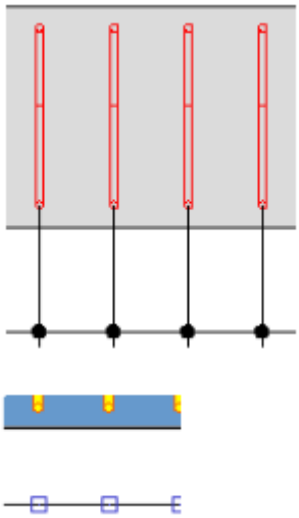
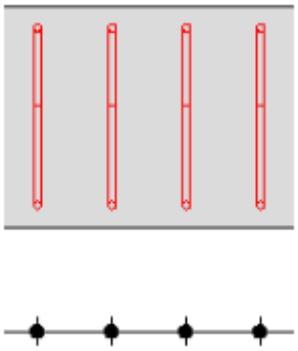
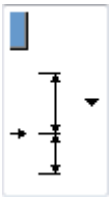
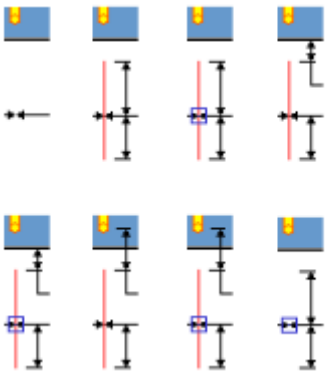
Setting	Options and descriptions
Not visualized rebars	
Visualized rebars	

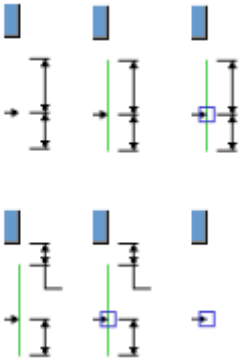
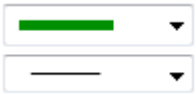
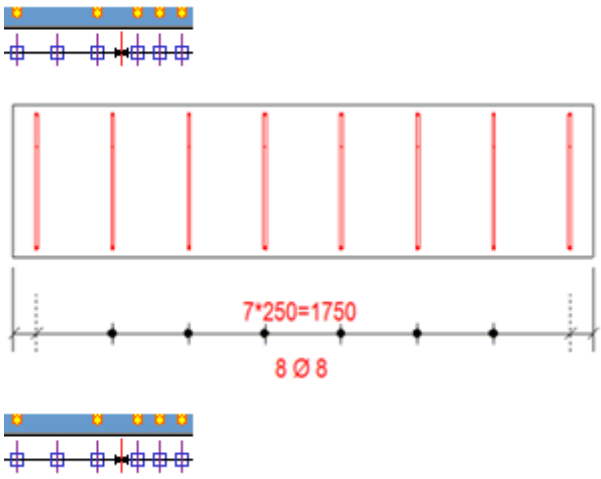
Setting	Options and descriptions
	
Group end rebars	Applied to the first and last bar in the group.  
Part edges	
 	Define the generation of the leader lines and symbols for the **Not visualized rebars** or **Visualized rebars**. The options are: No leader lines or symbols.

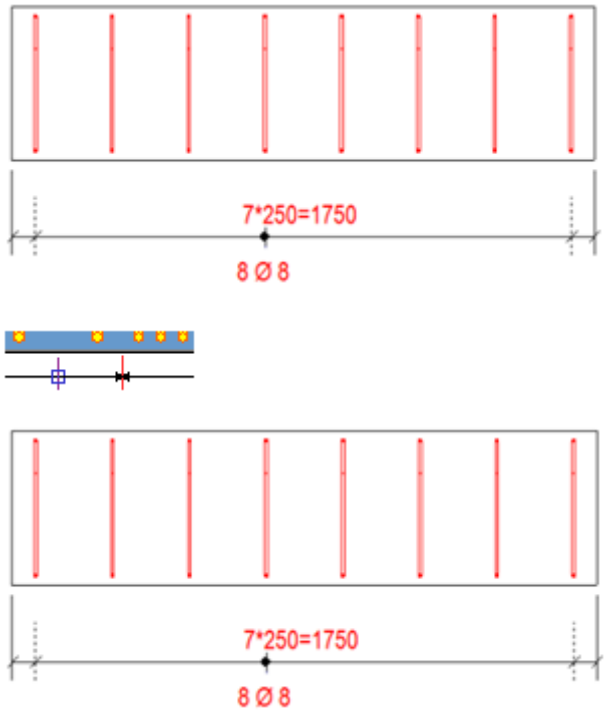
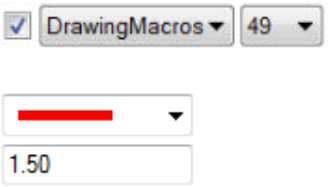
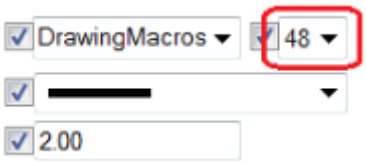
Setting	Options and descriptions
	 Leader lines. Define the length of the leader lines relative to the dimension line.  Leader lines and symbols. Define the length of the leader lines relative to the dimension line.

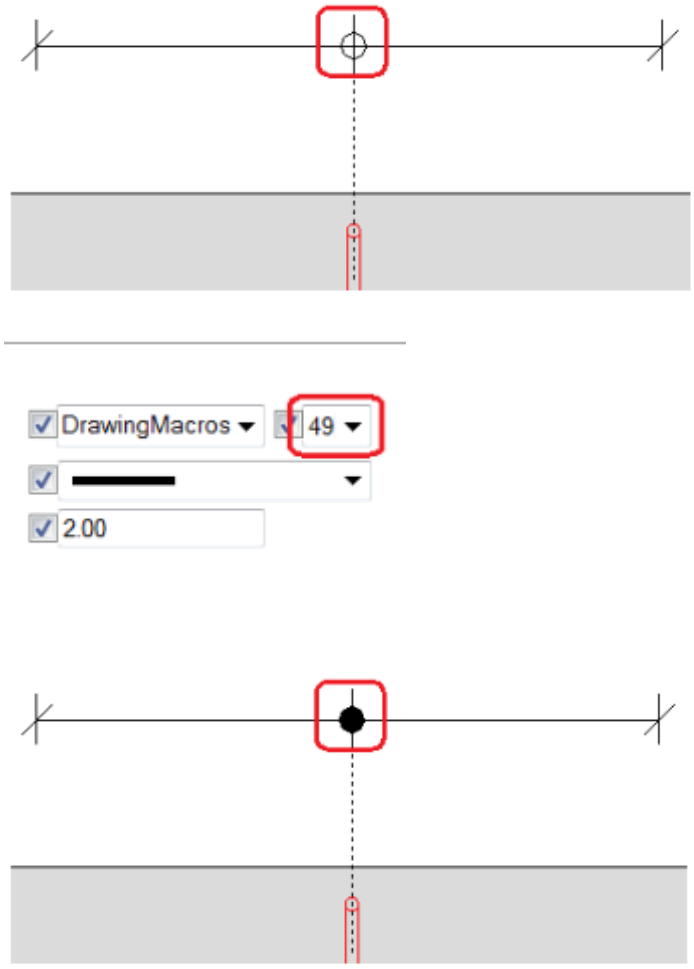
Setting	Options and descriptions
	 Leader lines. Define the length of the leader lines relative to the contour of the concrete part.  Leader lines and symbols. Define the length of the leader lines relative to the contour of the concrete part.

Setting	Options and descriptions
	 Leader lines. Define the length of the leader lines relative to the rebar.  Leader lines and symbols. Define the length of the leader lines relative to the rebar.

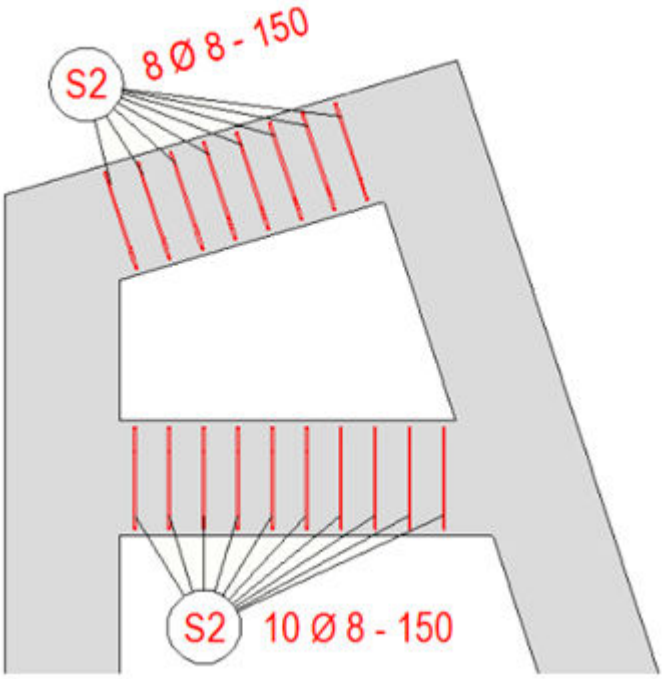
Setting	Options and descriptions
	 Symbols only 
	Define the generation of the leader lines and symbols for the **Part edges**. See the examples for the option **Not visualized rebars** above. The following options are available: 

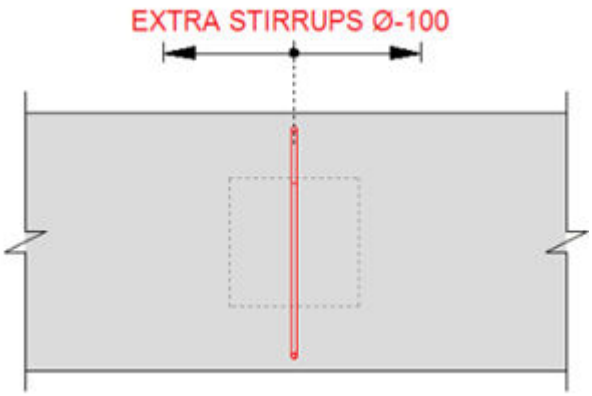
Setting	Options and descriptions
	Define the generation of the leader lines and symbols for the contour of the concrete part See the examples for the option **Not visualized rebars** above. The following options are available: 
	Define the color and the line type for the various leader line types.
	Define whether all leader lines and symbols are shown for **Visualized rebars** and if this should be done for one single rebar only. The following options are available: 

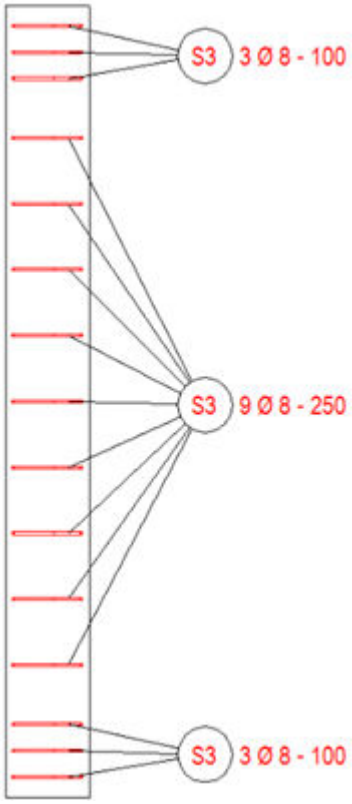
Setting	Options and descriptions
	
	Define the symbol file and symbol number to be used. You can use existing symbols in Tekla Structures by selecting a symbol file and a symbol number. You can also define the color and the size of the symbol. Symbols are defined separately for **Not visualized rebars**, **Visualized rebars**, **Group end rebars** and **Part edges** Examples: 

Setting	Options and descriptions
	

More examples

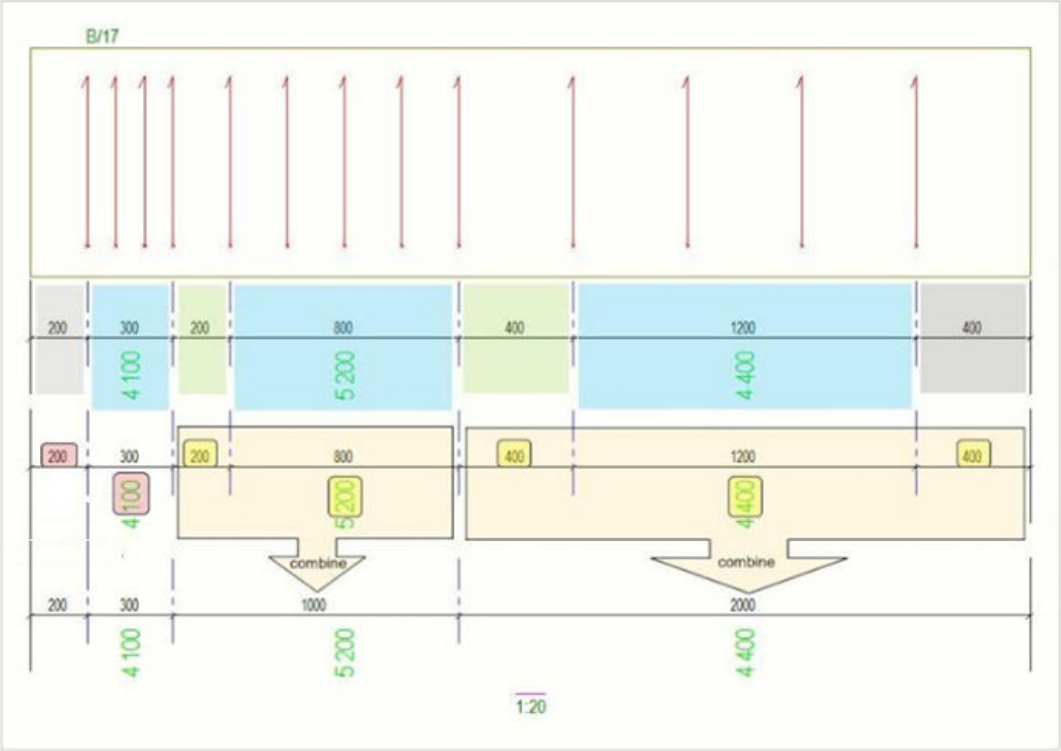






Example of the setting Combine dim. between rebar - dim. on the Parameters tab

- In the example below, the first dimension from the top shows the rebar groups (blue color) with the distances between the groups (green color) plus the distances to the part ends (grey color). Dimensions are not combined.
- The second dimension shows the same situation with marked equalities between the drawing rebar group spacing (cc) and the distances between the groups.
- In the third dimension, the new combining type **Consider same spacings** has been applied. The green distances between the two blue rebar groups have been combined with the group that has the same spacing (cc) as the distance between the groups.
- The combining has also been applied to the gray distance to the part end because the distance to the part end is the same as the spacing (cc) of the adjacent rebar group.



Modify dimensions

You can modify the created dimensions in several ways: modify the dimension appearance, add dual dimensions, filter dimension tag content, exaggerate dimensions, select to show plate side marks, change the location of short outside dimension texts, or change the dimension extension line length.

Modify dimension properties

You can modify the properties of the dimensions (page 932) in an open drawing.

1. Click a dimension, which can be either automatically or manually added.

   If the drawing property pane is not open, double-click the dimension.

   If you want to modify several dimensions, hold down **Ctrl** and click the dimensions.

   If you have selected several dimensions, the properties that have different values have the text *Varies*. You can select another value from the list.

2. Modify the dimension line properties, such as type, color, and arrow.
3. Modify the dimension value properties, such as font, place on line, background, frame, units, format, and precision. You can group the digits, combine equal dimensions, and define if you want to show the numeric value of the dimensions.

4. Modify the exaggeration settings as required.
5. Select if you want to show plate side marks, and modify the plate side mark settings.
6. Add dimension tags as required by using the tag editor (page 932), modify font and frame properties, and set the dimension tag rotation. Here you can set the dimension prefix and postfix.

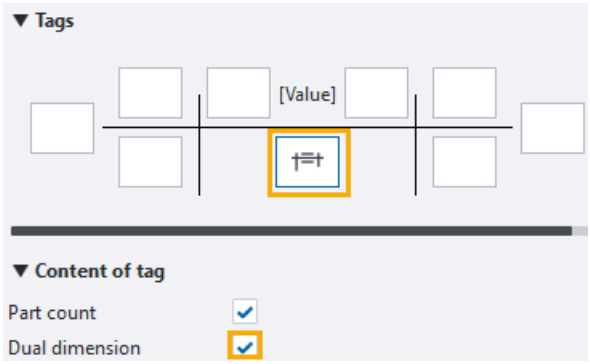
   Here you can include part count to dimension tags and select a filter that removes the desired default content from the tag. You can also select the curved dimension tag type to control how the tags are aligned to the dimension.
7. Modify the dimension placing settings.

   For example, here you can set the dimension to be free or fixed. **Free** lets Tekla Structures to decide the location and direction of the dimension. **Fixed** allows you to place the dimension at any point.
8. Click **Modify**. The selected dimensions are modified.

Add dual dimensions manually

In dual dimensions, you can show dimensions in different units and format, above and below the dimension line.

1. On the **File menu**, click **Settings** --> **Options** , and go to the **Drawing dimensions** settings.
2. Set the unit, formats and precision, and activate the dual dimensions for the necessart drawing types in **Dimensions in tags**.
3. Click **OK**.
4. Click a dimension in your drawing. If the drawing property pane is not open, double-click the dimension.
5. Go to the **Tags** section, open the **Content of tag** area, click the middle tag box, and select the **Dual dimension** check box. Add the desired elements in the middle dimension tag.

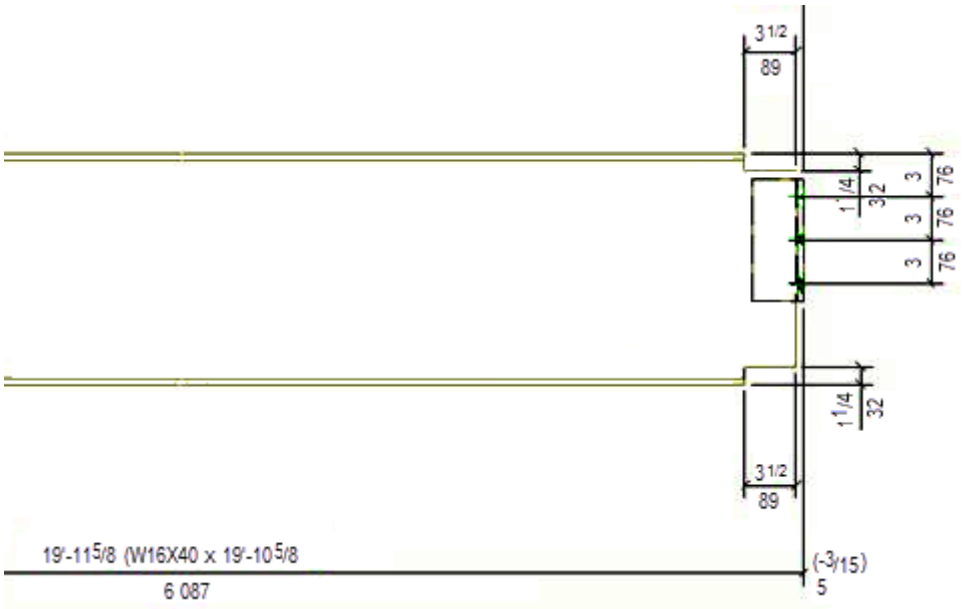


6. Click **Modify**.

7. If you do not want to show dual dimensions in all dimensions, you can remove the contents of the middle tag by clicking the red cross in the tag corner or by clearing the selection from the **Dual dimension** check box.

Example

Below is an example of dual dimensions that use the unit mm and format ###.

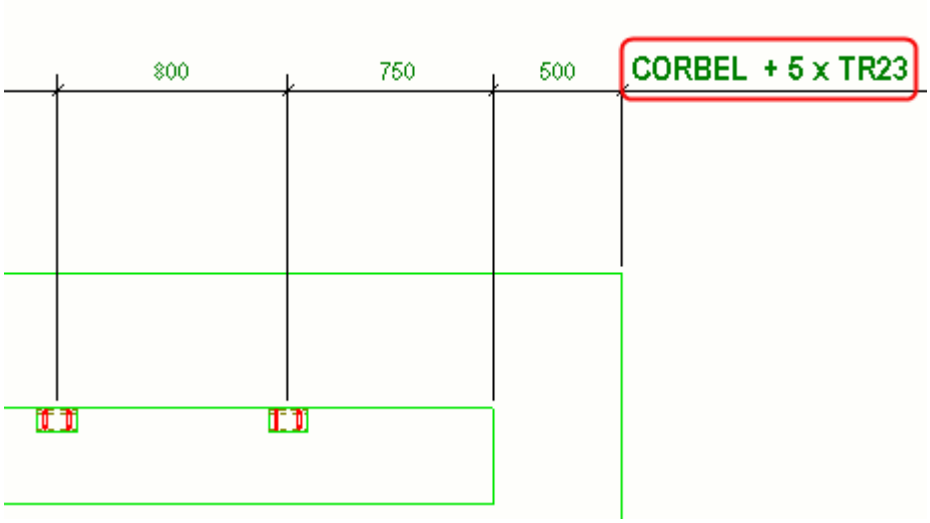


You can also add dual dimension automatically (page 779).

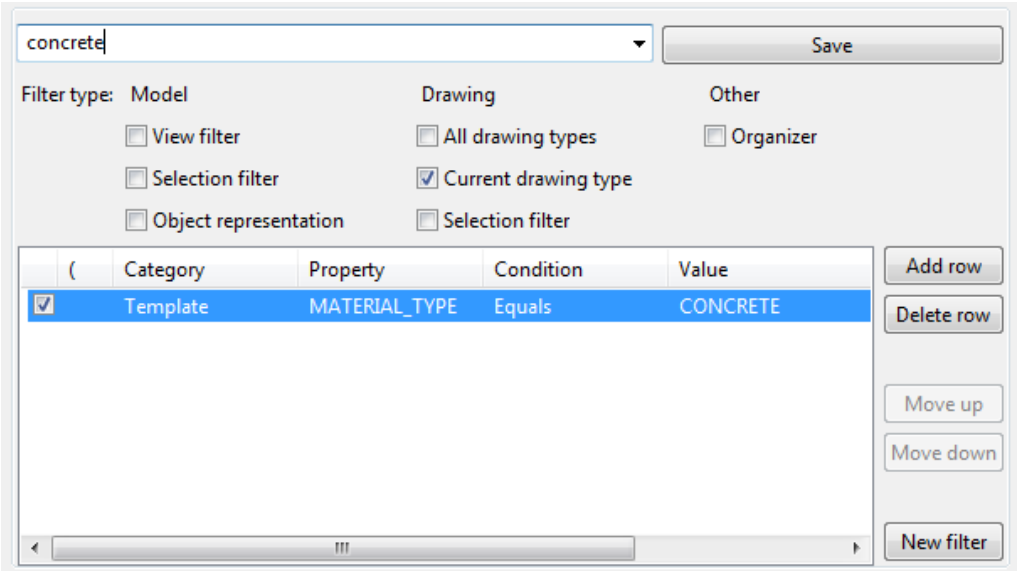
Filter out dimension tag content

You can remove from a dimension tag some content that has been added in the tag automatically based on the dimension end point locations. First you need to create a drawing view filter that you will use for selecting the content that needs to be removed.

The example below shows a tag that automatically contains the text "CORBEL". You will now remove this text.



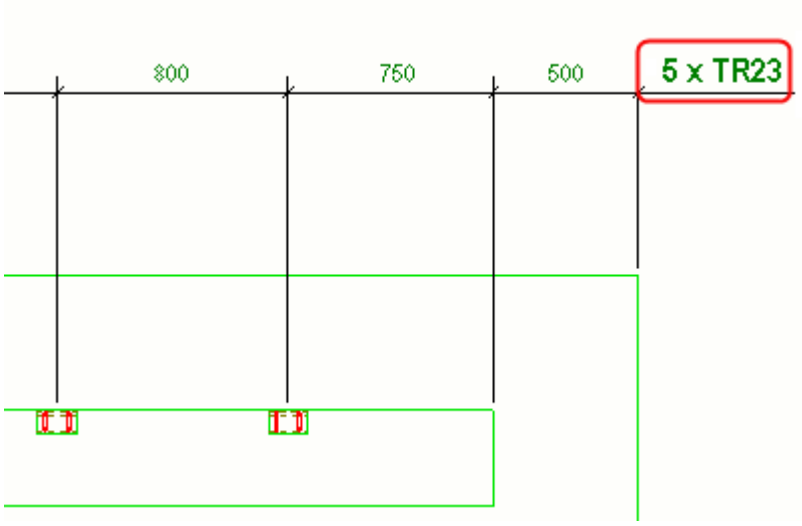
1. Open a drawing.
2. Double-click the view frame to open the **View Properties** dialog box.
3. To create a filter, click **Filter** in the options tree.
4. Click **Add row** and create a filter according to the example below.



5. Enter the name of the filter in the box next to the **Save** button and click the button to save the filter. In this example, enter `concrete`.
6. To close the dialog box, click **Cancel**.
7. In the drawing, click the dimension that contains the content that you want to remove. If the property pane is not open, double-click the dimension.
8. Go to the **Excluded tags** section in the property pane, select the filter you just created. Here, select the `concrete` filter.

9. Click **Modify**.

   Tekla Structures removes the text "CORBEL" from the dimension tag. The material type of corbel is concrete, and the `concrete` filter removes all concrete parts from the tag.



Control dimension tag content with associativity rules

Dimension tag content can also be controlled using associativity rules. Selecting the right associativity rules helps you to easily select objects that need to be shown in the tags and to set up dimension tags, without having to create complex exclusion filters. Only attributes that correspond to the selected objects will be shown in the tags.

This is especially helpful in conflict situations where dimension points might not be associated with the desired objects, because there are other objects in the same dimension point location.

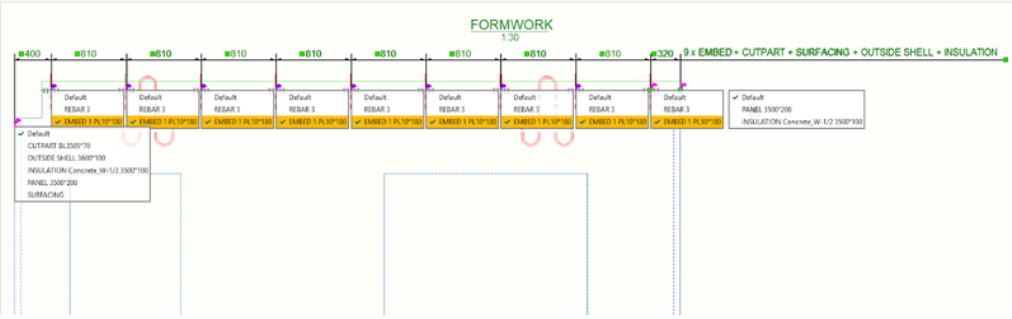
To adjust dimension tag content:

1. In an open drawing, right-click a dimension whose tags you want to modify.
2. In the context menu, select **Show dimension associativity rules**, and then select the objects that you want to show in the dimension tag.

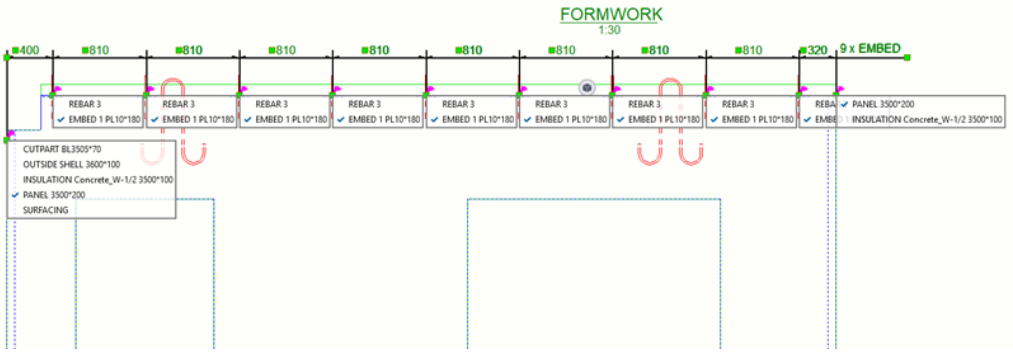
   The objects that are not selected from the associativity rule list are immediately filtered out from the tag.

   Example:

   The properties that correspond to all dimensioned objects are shown in the tag:



After selecting the appropriate objects from the associativity rule list, the objects that are not selected are immediately filtered out from the tag.



Limitations

There are some hard-coded filtering rules in the dimension tag code that you should keep in mind:

- The main part properties are always filtered out if the dimension is associated to the main part and secondary parts.

  Example: You create a dimension that is connected to a main part and a number of embeds (secondary parts). The marks are displayed from the embeds (secondary parts), and the main part is filtered out automatically.

- The attributes of sub-assembly secondary parts are always filtered out. Instead, the attributes of sub-assembly main parts are shown.

  Example: You create a dimension that is connected to embeds. All embed sub-parts need to be filtered out. In addition, if the embed main parts are not in the array, they are added to this array.

Exaggerate selected dimensions

You can exaggerate narrow dimensions to make them easier to read. When exaggerated, a dimension that is narrower than the limit defined in the **Options** dialog box is enlarged using the defined scale. If there are many exaggerated dimensions, Tekla Structures arranges them automatically.

1. On the **File menu**, click **Settings** --> **Options** and go to the **Drawing dimensions** page.

2. Set the **Exaggeration limit** and **Exaggeration scaling**.

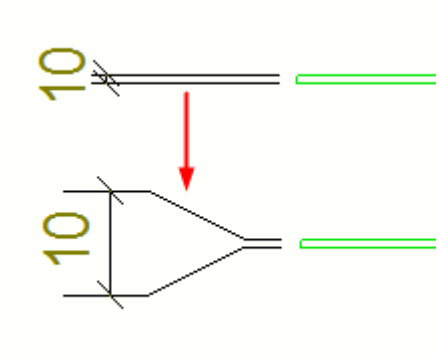
   Exaggeration scaling defines whether you are using **Paper** or **Model** as the exaggeration scaling method.

   - If you select **Paper**, the exaggeration limit is multiplied by the view scale.
   - If you select **Model**, and the scale is 1:10, all the dimensions smaller than 10 mm are exaggerated regardless of the drawing scale.
3. Click **OK** to save the settings and close the **Options** dialog box.
4. In an open drawing, click the dimension that you want to exaggerate.
5. In dimension properties, go to the **Exaggeration** section, and do the following:
   - In **Exaggerate short dimensions**, select **Specified**.
   - Select the **Direction** of the exaggeration: **Left / Down**, **Right / Up** or **Both**.
   - Set the **Origin**:

     : The origin is near the dimension line.

     : The origin near the measured object.
   - Set the desired exaggeration width, and the position and height for the exaggerated dimension.
6. Click **Modify**.

   The selected dimensions are exaggerated.



You can also define dimension exaggeration in the view level properties (page 783).

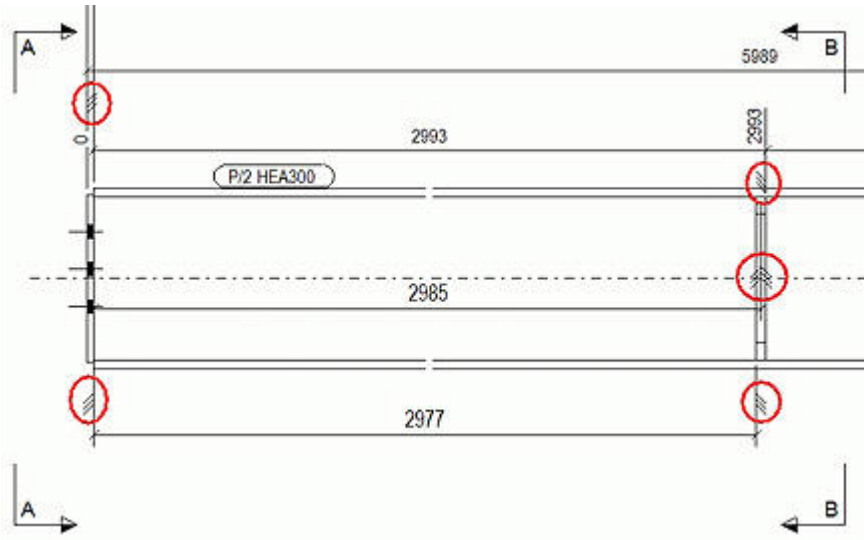
Show plate side marks on dimension leader lines

You can show plate side marks on dimension leader lines. The plate side marks indicate whether the dimension point is to the face or center of the part, such as a plate, web, or flange.

1. In an open drawing, click the dimension where you want to show plate side marks.
2. To control the plate side mark symbol and insert the symbols, in the dimension properties, go to the **Plate side marks** section and set the **Side mark type** to **Specified** .

   The option **Automatic** is available only in intelligent drawings, which is the case when the advanced option `XS_INTELLIGENT_DRAWING_ALLOWED` is set to `TRUE`.
3. Modify the other properties of the plate side marks as required:
   - Select the left and right plate side mark.

   - Set the mark size.
   - Set an offset for the mark from the dimension line.
   - Adjust the mark color.
4. Click **Modify**. The plate side marks are created.



Also check the following topics:

XS_DIMENSION_PLATE_SIDE_MARK_SYMBOL_LEFT

XS_DIMENSION_PLATE_SIDE_MARK_SYMBOL_CENTER

XS_DIMENSION_PLATE_SIDE_MARK_SYMBOL_RIGHT

XS_INTELLIGENT_DRAWING_ALLOWED

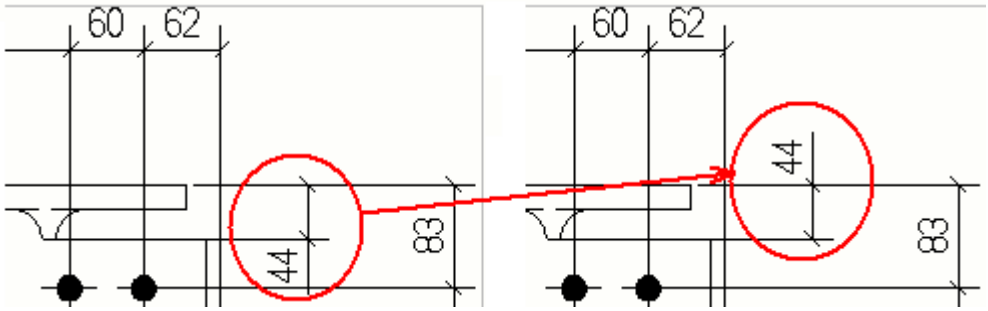
Change the location of short outside dimension texts

You can select to place the texts of short dimensions outside the dimensions. In the dimension properties, set the **Short dimensions** setting to **Outside**. You can also select on which side of the extension line the dimension text is placed.

Limitations:

- You can flip only start or end dimensions in a dimension set.
- You can place the dimension text outside the dimensions if there is enough space for the dimension text.

1. On the **Dimensioning** tab, click **Flip outside dimension**.
2. Click the dimension whose location you want to change.

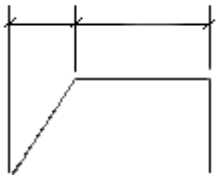
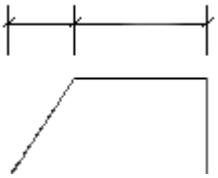
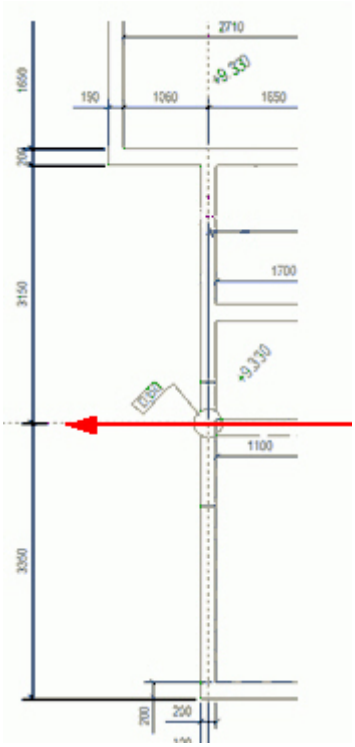


You can change the dimension location by dragging the dimensions (page 287).

Set the dimension extension line length

You can adjust the length of the dimension extension lines in the dimension properties in an open drawing. You can also adjust the extension line length using advanced options.

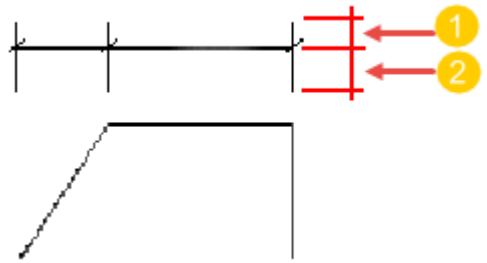
1. Open a drawing and click a dimension.
2. Select an option from the **Short extension line** list:

Options	Description
No	Use the short extension line only if the dimension line falls on a grid line:  If you want to exaggerate (page 783) dimensions, you need to set this option to **No**.
Yes	Create extension lines all of the same length: 
On grid lines only	Use the short extension line automatically if a dimension line falls on a grid line. Elsewhere the extension line will be as usual. 

3. Click **Modify**.

The dimension extension lines are shortened according to your selections.

To adjust the extension line lengths, go to **File** --> **Settings** --> **Advanced options** --> **Dimensioning - general** and set the following advanced options. These advanced options adjust the extension line length in relation to the text size:



(1) Text height * 1.0 (default). Use XS_DIMENSION_EXTENSION_LINE_AWAY_FACTOR to adjust the length of the dimension extension lines that are facing away from the dimension points.

(2) Text height * 1.5 (default). Use XS_DIMENSION_EXTENSION_LINE_TOWARD_FACTOR to adjust the length of the dimension extension lines that are facing towards the dimension points.

Recreate dimensions for all parts

You can recreate the dimensions that were originally created for the parts in the drawing.

Redimensioning removes all manually created dimensions. Tekla Structures does not redimension linked, 3D, or key plan views.

1. In an open drawing, click a drawing view frame to open the view properties, and check that you have set the **Dimension creation** setting to **Automatically**.

   Tekla Structures only recreates the dimensions in the views where this setting is set to **Automatically**.

2. On the **Dimensioning** tab on the ribbon, click **Recreate all dimensions**.

   Tekla Structures redimensions the parts in the view and removes all manually created dimensions.

See also

Add dimensions manually (page 207)

View properties in drawings (page 912)

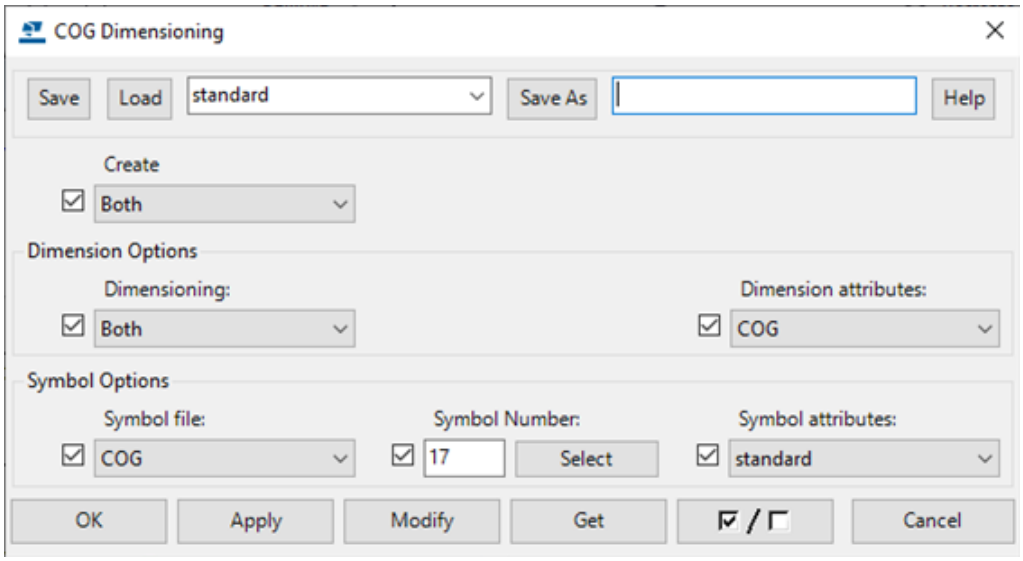
Dimension center of gravity (COG)

You can indicate the location of the center of gravity (COG) in single-part, assembly, and cast unit drawings by creating COG dimensions and a COG symbol at the center of gravity. You can also create COG dimensions in section views. COG dimensions will be automatically updated if the single part, assembly, or cast unit changes. The COG dimensions can also be cloned.

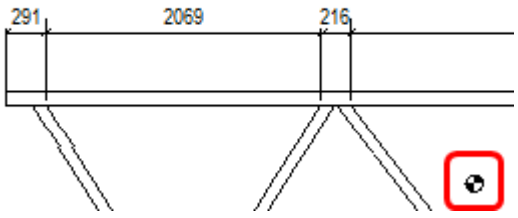
Limitations:

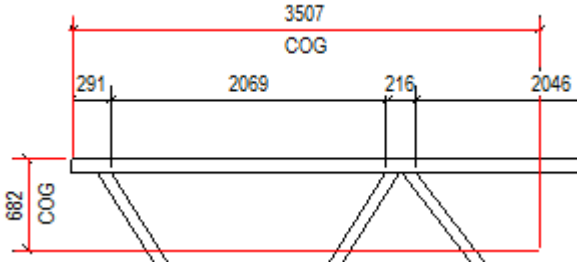
- If you copy or link a drawing containing COG dimensions to another drawing, such as a multidrawing, the COG dimensions will not be copied.
- You cannot create COG dimensions in general arrangement drawings or multidrawings.

1. On the **Dimensioning** tab in an open drawing, click **COG** .
2. Modify the options as required:



- In **Create**, select **Symbol** to see only the COG symbol, or **Dimensions** to see only COG dimensions. To see both, select **Both**.



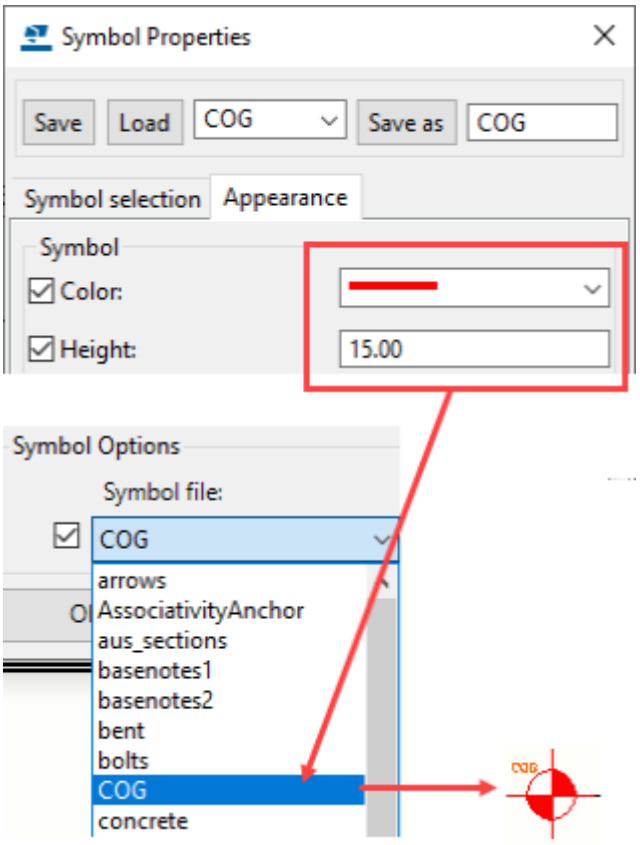


- In **Dimensioning**, select to create **Horizontal** or **Vertical** dimensions, or **Both**.
- In **Dimension attributes**, you can load predefined dimension settings.

  The appearance settings (size, color, etc.) of the COG dimensions are read from the dimension settings file you load in **Dimension attributes**. The dimension unit, format and accuracy also come from the settings file. To create and save dimension settings files, on the **Drawing** tab, click **Properties** --> **Dimension**. For example, you may create a special COG dimension settings file to change the color or arrow type, and load the saved settings in **Dimension attributes**.
- In **Symbol Options**, you can change the **Symbol file** in use by selecting another symbol file in the list. To change the COG symbol, click **Select**, and select another symbol. To load other symbol settings from another symbol settings file, select another file from the **Symbol attributes** list.

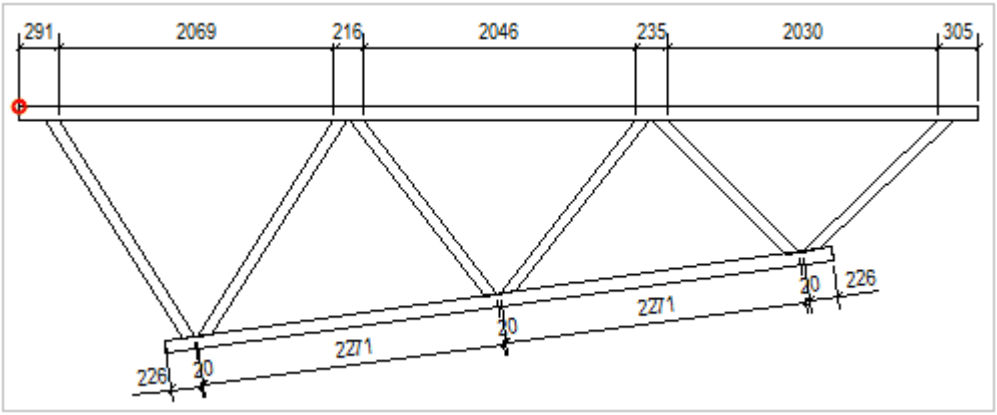
  You can access the symbol options only if you have selected **Both** or **Symbol** for **Create**. The appearance settings (height, color, etc.) of the symbol are read from the symbol properties file you load in **Symbol attributes**.

  To create and save symbol settings files, on the **Drawing** tab, click **Properties** --> **Symbol** . For example, you may create a special COG symbol settings file to change the color and height of the symbol, and load the saved settings in **Symbol attributes**.



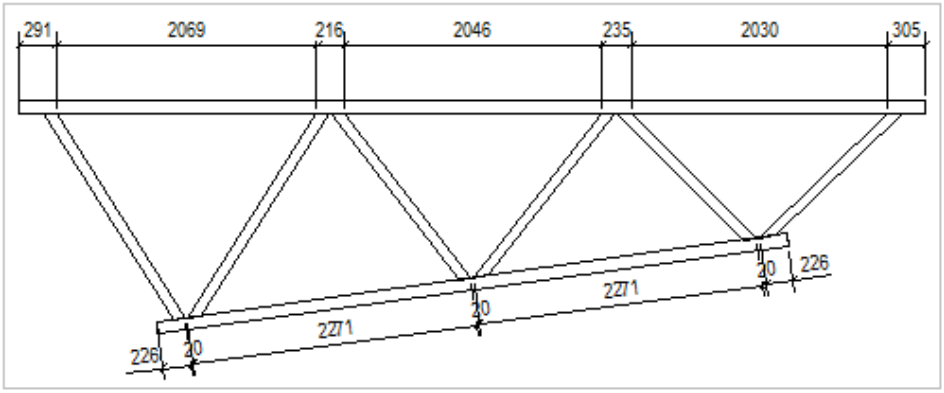
3. Click **OK**.
4. Pick the first point to specify the origin of the dimensions.

   The origin is the point from which you want to measure the location of the center of gravity. This point must be located within the view frame.

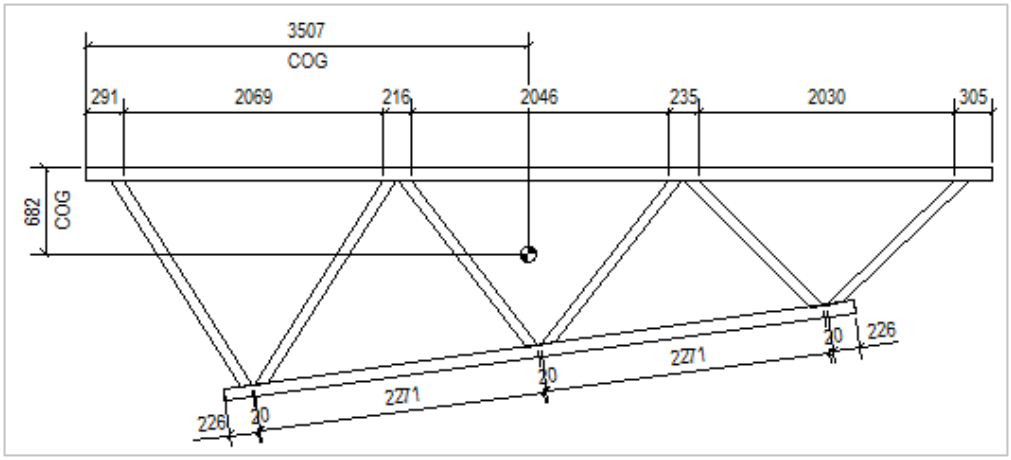


5. Pick the second point to place the dimensions.

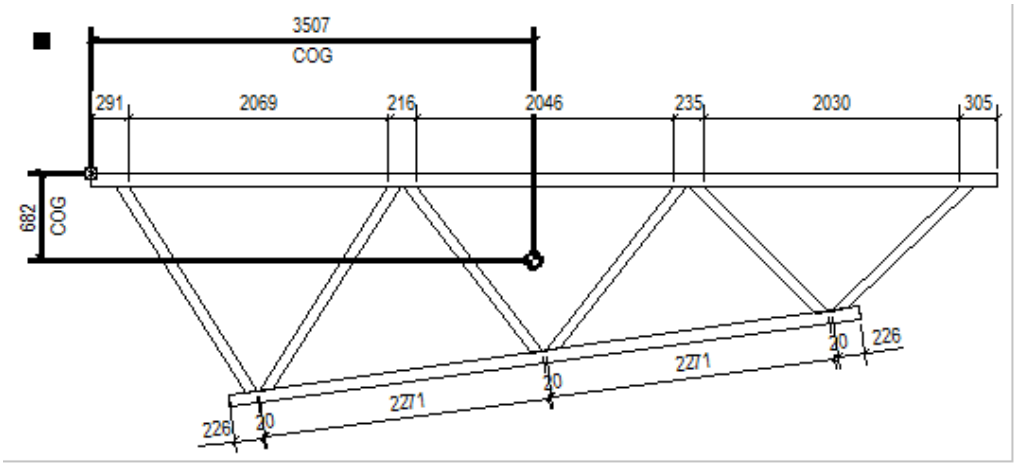
This point may fall outside the view frame.



The example below shows the created dimensions.



6. When the dimension is selected, handles are shown at the dimension origin and the dimension location. You can drag these handles to adjust the origin or the location, or move them using the standard editing commands.



You can inquire the coordinates of the center of gravity of assemblies, parts, or welds with the template attributes COG X, COG Y, and COG Z.

What does the COG calculation take into account?

COG calculation takes the following into account:

- Material
- Surface treatment
- Part cuts
- Fittings
- Edge chamfers

COG calculation does not take the following into account:

- Rebars
- Bolts
- Bolt holes
- Welds

Customize dimension line arrows

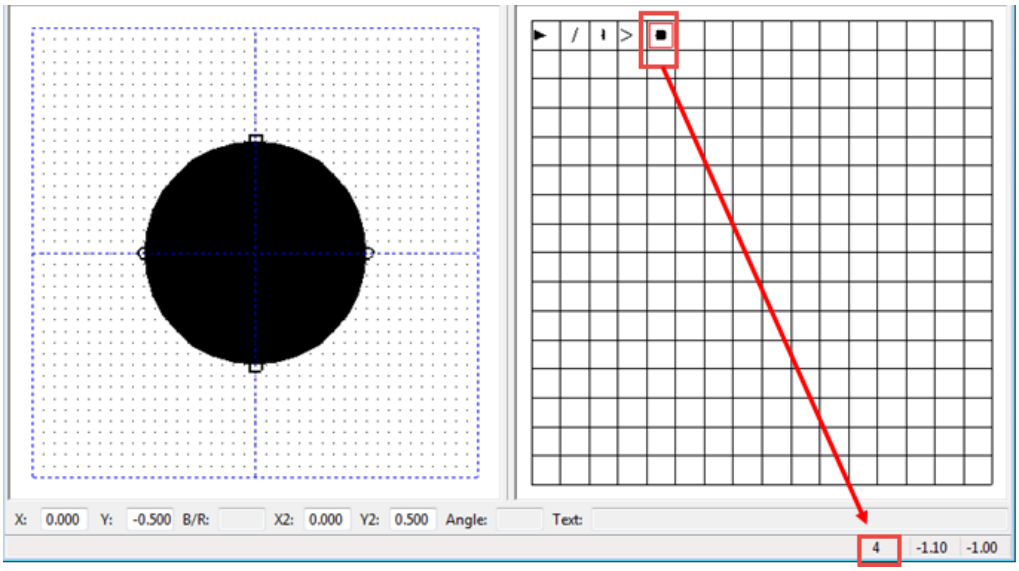
If you do not find a suitable dimension line arrow in the **Arrow** list in the dimension properties, you can create and use an arrow of your own.

First you need to create the arrow symbol in the Symbol Editor, and save the created symbol in the `dimension_arrows.sym` file. Then add the position number of the new symbol in the `dimension_arrows.sym` file to the configuration file `dimension_arrows.txt` file. This file tells which arrows are available for use in your environment. Finally, you need to create a bitmap for the new arrow to be used in dimension properties.

1. To open Symbol Editor, on the **File** menu, click **Editors** --> **Symbol editor** .

2. Open the `dimension_arrows.sym` file located in the Common environment (`...ProgramData\Trimble\TeklaStructures\<version>\Environments\common\symbols`) or in your environment under the `\symbols` folder.
3. Click an empty symbol slot and sketch your symbol with Symbol Editor drawing tools.

   You can also import AutoCAD or MicroStation files through **File** --> **Import**.
4. When the symbol is completed, point the symbol slot to check the number of the new symbol at the bottom of the window.



5. Save the `dimension_arrows.sym` file by clicking **File** --> **Save** .
6. To close the Symbol Editor, click **File** --> **Exit** .
7. Open the `dimension_arrows.txt` file located in the same `\symbols` folder as the `dimension_arrows.sym` file.

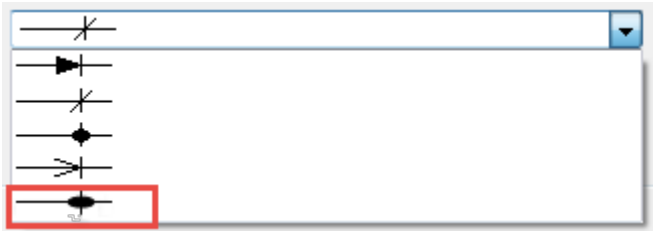
   Open the file in Microsoft Notepad, for example. The file contains a list of symbol numbers.
8. Add the number of your symbol preceded by two zeros (00) in a correct position and separate it with a comma:

   000,001,002,003,**004**
9. Click **File** --> **Save** to save your change.
10. Create a bitmap of the new arrow and save it in the `..\ProgramData\Trimble\Tekla Structures\<version>\Bitmaps` folder.

    Use the following naming convention in the file name: `dr_dialog_dim_arrow_type_004.bmp`.
11. Finally, restart Tekla Structures.

12. Check that the new arrow is displayed in the **Arrow** list in the dimension properties:
    - Click a dimension in a drawing to open dimension properties (page 932), and then open the **Arrow** list. If the property pane is not open, double-click the dimension.

      The new arrow symbol is now available:



NOTE We recommend that you define a firm folder (page 371) for symbols, because the default folders are overwritten when you upgrade to a newer version of Tekla Structures. Add the firm folder to the advanced option DXK_SYMBOLPATH.

See also

Dimension properties (page 932)

Set a new dimension start point

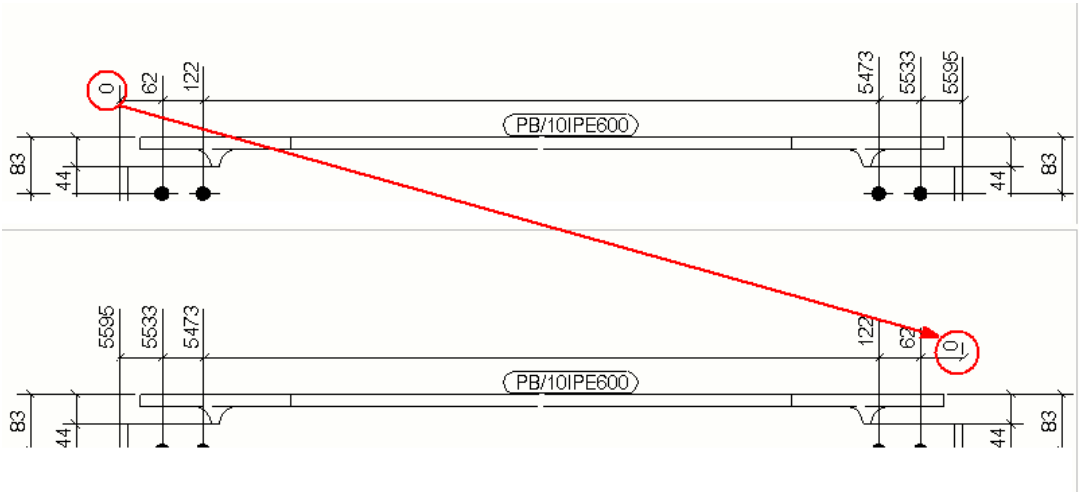
You can select a new start point for running dimensions (dimensions that start from a common start point).

1. Select an existing dimension in a drawing.
2. On the **Dimensioning** tab, click **Set start point**.
3. Select the new start point.

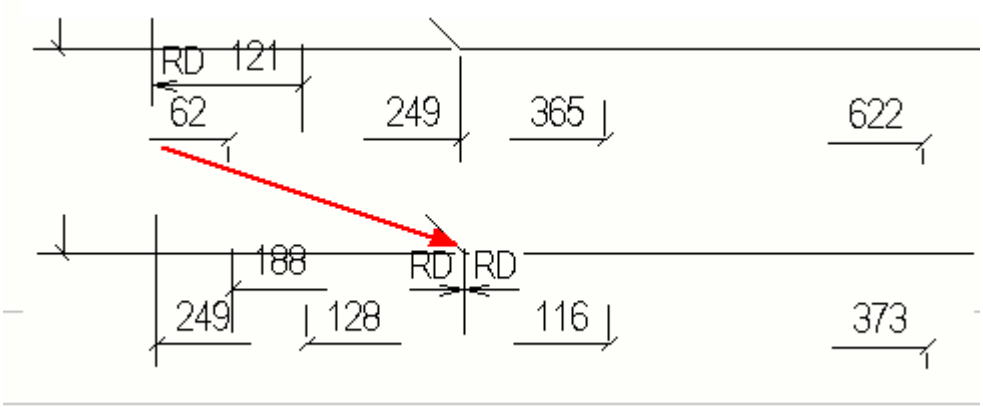
   Tekla Structures automatically updates the dimensions.

Example

You can use this command to swap the running dimensions start point to the opposite end of the member.



When you use the US absolute dimension type, Tekla Structures draws a new RD symbol (Running Dimension) at the new zero point and updates the dimensions according to the new start point.



See also

Dimension properties (page 932)

Add or remove dimension points

You can add new dimension points to, or remove existing dimension points from the selected dimension set. You can only add points to one dimension line at a time.

1. Select the dimension line.
2. On the **Dimensioning** tab, click **Add point**.
3. Click a position on the part where you want to add the dimension point.

You can add several points.

To remove a point, on the **Dimensioning** tab, click **Remove point** and click the point you want to remove. You can remove several points in a row.

TIP A quick way to add **and** remove dimension points is to first click a positions on the parts where you want to add the dimension points and, if some of them seem unnecessary, remove those by holding down **Shift** and then clicking the points you want to remove.

See also

Dimension properties (page 932)

Modify dimensions (page 260)

Add closing dimensions to rebar dimensions (page 227)

Display and change dimension point associativity

You can change the dimension point associativity to avoid incorrect associativity, or to simply make the associativity selection distinct. Each dimension point associativity anchor shows a list of objects that have locations available for association. You can change the dimension point associativity in straight dimensions only.

Dimensioning points created in Tekla Structures are associated to the objects where the dimension points have been added. This allows the dimensions to update automatically when the objects are changed during modeling.

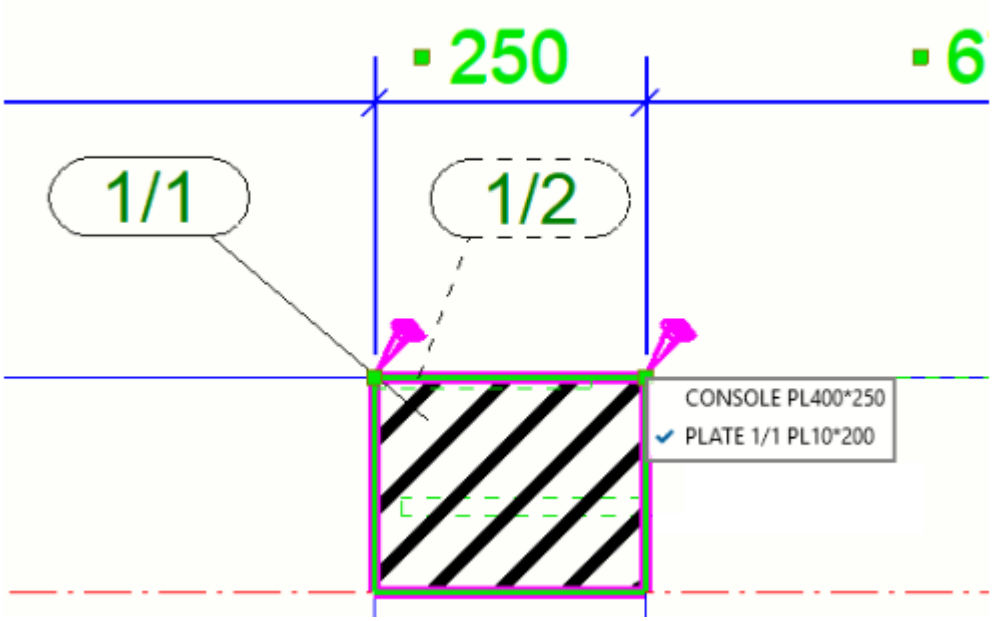
Sometimes the dimension point locations may be unclear due to orthogonal projection. A dimension point might be not be associated to the desired object because there are various other objects in the same point location. In drawing update or cloning, this may lead to unwanted dimensioning values or associativity.

It is also possible to associate dimensions to object center lines outside the object area.

Display and change dimension point associativity

1. In an open drawing, select the desired dimension or dimension set.
2. Click a dimension point associativity anchor.

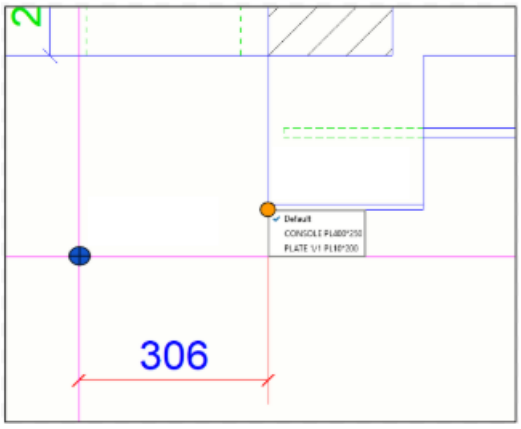
This displays the dimension associativity rule list, which shows the objects where the dimension point can be associated.



Note that the feature intelligently selects the rules based on the data collected on previously selected rules in the range of the dimension that is being created. When you continue picking dimension points you may notice that the associativity rules similar to the rules picked earlier are selected automatically.

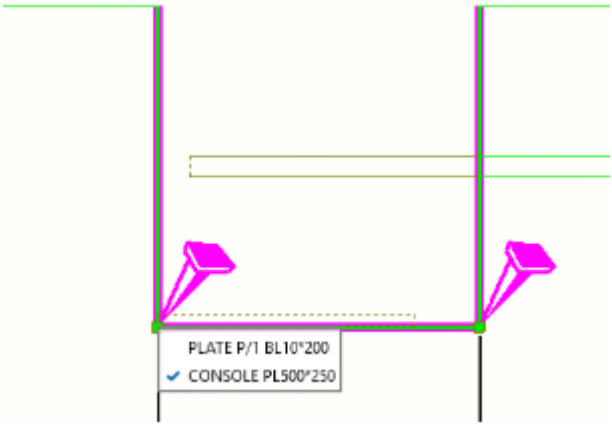
The associativity rule list is located so that it avoids overlapping with the building objects and mouse pointer. The feature detects the current position of the mouse pointer and keeps the associativity rule list on the opposite side relative to the picked dimension point.

In the following image, the mouse pointer position is indicated with a blue circle and the picked dimension point with an orange circle:

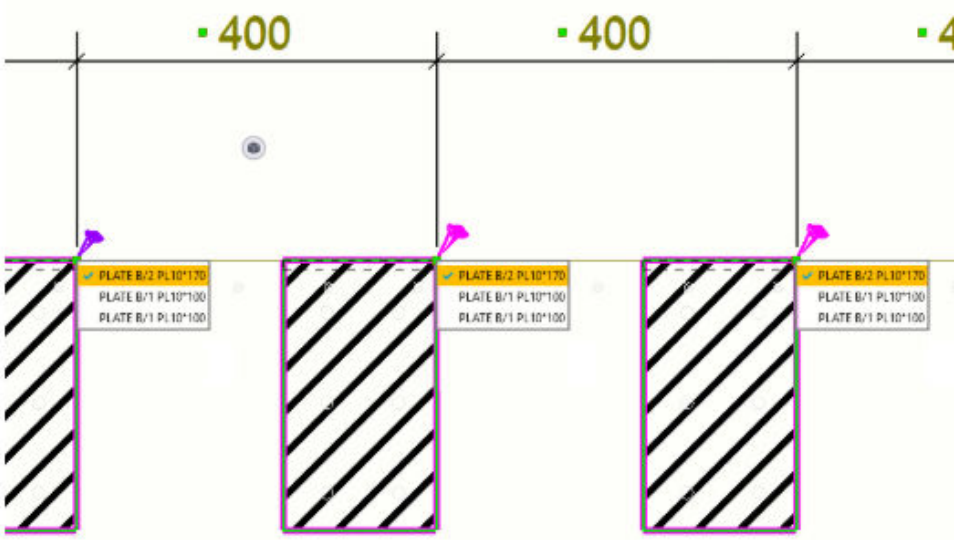


3. Click a rule in the list to select it and associate the dimension point to a new object.

When you click the rule, the corresponding object is highlighted in the drawing. This makes it easy for you to to check if the selected rule refers to the object you want.



4. You can also do the following:
   - Open a several dimension associativity rule lists at a time by holding down **Ctrl** or **Shift** and clicking dimension point associativity anchors, or holding down **Alt** and using area selection.
   - Open all associativity rules related to the selected dimension, click a dimension and select **Show dimension associativity rules** from the context menu.
   - You can alter associativity rules by scrolling the mouse wheel while mouse cursor hovers over a rule list.
   - To select associativity rules related to same object types in all opened rule lists, double-click the desired rule. Selected rules will be highlighted with yellow color in all opened rule lists.



- You can control the maximum number of shown rules in the associativity rule list with the following advanced options:

  XS_INTELLIGENCE_MAX_RULE_COUNT

  XS_INTELLIGENCE_MAX_PART_COUNT

  XS_INTELLIGENCE_MAX_PLANE_COUNT

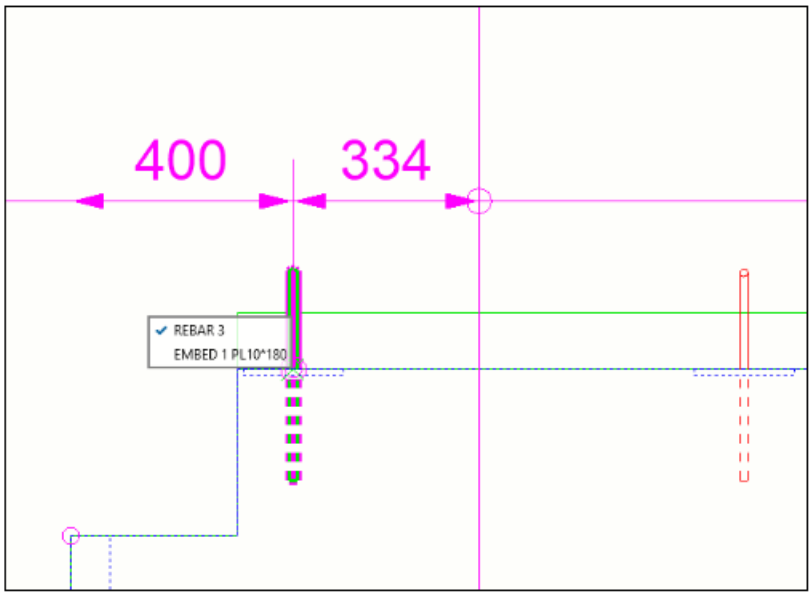
  If the needed associativity option is missing from the rule list, increase the number of the shown rules by changing the advanced option values.

5. Click the drawing background to close the rule list.

Select dimension point associativity in manual dimensioning

You can select the associativity rules for each dimension point during manual dimensioning of the drawing objects. This way you can properly associate a dimension during its creation, and the dimension locations are preserved better during updates and cloning.

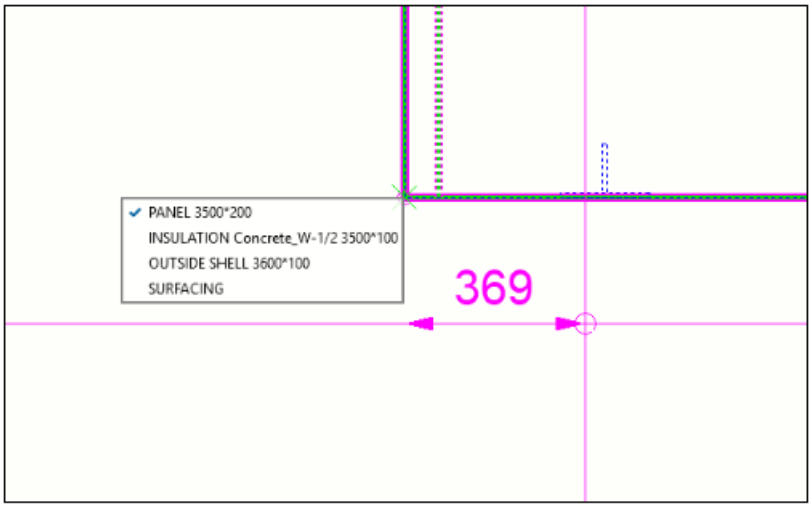
1. In an open drawing, activate the dimension creation associativity functionality: Go to **File** --> **Settings**, and select the **Dimension creation associativity** check box.
2. Start creating a dimension with any of the following dimensioning commands: **Horizontal**, **Vertical**, **Orthogonal**, **Perpendicular**, **Parallel** or **Free**.
3. To display the associativity rule list, pick a dimension point position.



4. Select the desired associativity rule in one of the following ways:

- By scrolling the mouse wheel when the mouse pointer is on the associativity rule list.
- By directly selecting the desired associativity rule from the list by clicking the rule with the left mouse button.
- By entering the **Select next associativity option** command in **Quick Launch**.
- By using a keyboard shortcut that you have defined for **Select next associativity option**.

When you select a rule, the related object is highlighted in the drawing, which allows you to check if the selected rule refers to an appropriate object.



When you have selected the associativity rule, you can continue creating the dimension by picking the following dimension points. The associativity rule list is displayed each time when a point with available associativity rules is detected.

Limitations

- Selecting dimension associativity rule during manual dimension creation works only with straight type of dimensions, only with the following commands: **Horizontal**, **Vertical**, **Orthogonal**, **Perpendicular**, **Parallel** and **Free**.
- Some rules can be missing in the associativity rule list. To increase the number of rules listed in associativity rule list, increase the value of the following advanced options:

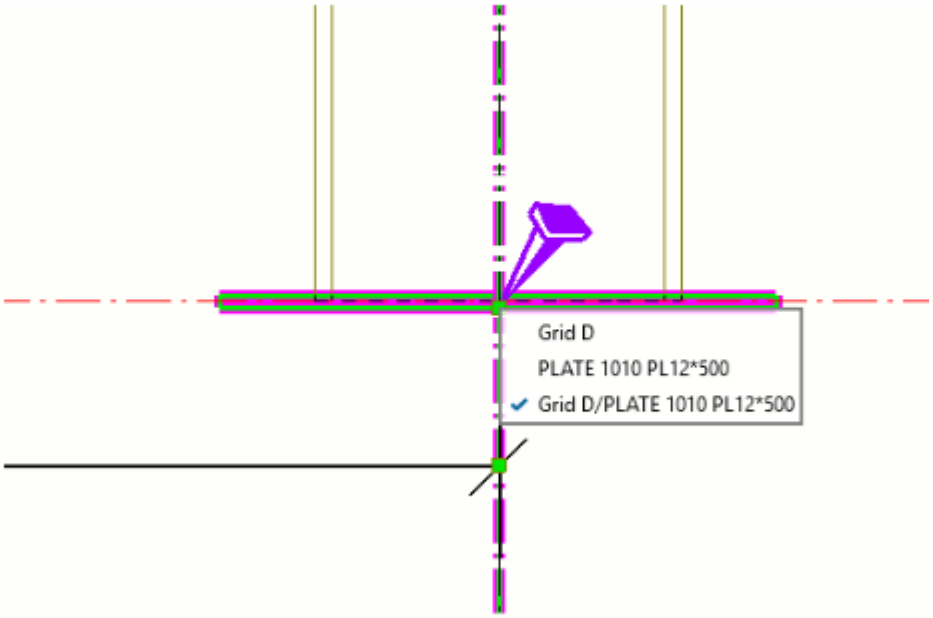
  XS_INTELLIGENCE_MAX_RULE_COUNT

  XS_INTELLIGENCE_MAX_PART_COUNT

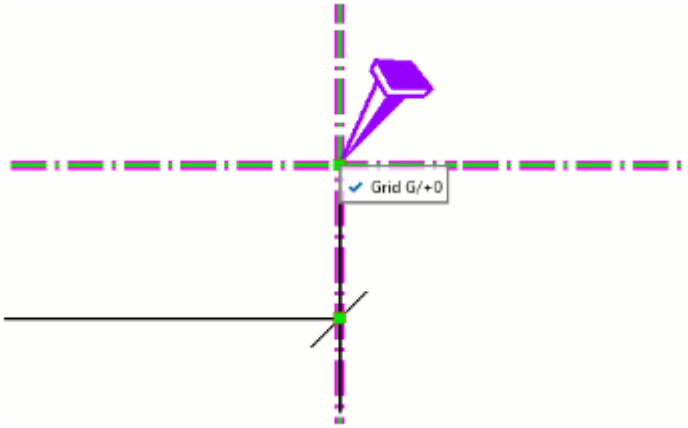
  XS_INTELLIGENCE_MAX_PLANE_COUNT

Associativity rule list content

- The following attributes are shown in the rule list for the following building objects:
  - Steel parts: "NAME", "PART_POS", "PROFILE"
  - Concrete objects: "NAME", "PROFILE"
  - Polygon cuts: "NAME", "PROFILE"
  - Reinforcement: "NAME", "REBAR_POS"
  - Bolts: "NAME", “DIAMETER”, “LENGTH”
  - Grid: Grid "NAME"
  - Intersection of grid line and building object side: Grid “NAME” or the label of the building object:



  - Intersection of grid lines: Grid “NAME”/”NAME”:



- Building object reference line or centerline: “Reference point” or “Center” accordingly
- Pour units: Pour object "POUR_TYPE", # "POUR_NUMBER"
- The **Default** associativity rule is not always shown in the associativity rule list, unless it is the only rule available, or if it has been previously selected as the active rule for the chosen dimension point. If **Default** is selected, the dimension point will automatically select which object to follow during the dimension update based on the new position of the objects. Note that there might not be any rule in the associativity rule list that would correspond the behavior of the **Default** option. Selecting another option than the **Default** option makes it possible to guarantee which object will be followed during the dimension update.
- Display dimension associativity with the **Clone** and **Clone selected** commands:
  - Changing associativity rules in selected dimension points allows you to achieve more accurate results in cloning dimensions in drawings. An associativity option selected in a rule list will be preserved in the cloned dimension if the selected rule corresponds to a similar object in a cloned drawing or in a target selection in a GA drawing. Objects are not considered to be similar if they have been modeled by using different commands, such as steel beam, steel plate, reinforcement, or bolt.
  - Note that when the drawing is cloned automatically due to renumbering, the associativity rules are recomputed from scratch and any preferred rules set by the user are lost.

Add dimension points in anchor bolt plans

You can add dimension points to dimensions inside the enlarged views in an open anchor bolt plan.

Limitations: You cannot create new dimension lines that have dimension points inside both the enlarged views and the plan view.

To add dimension points to the dimensions inside the enlarged views:

1. Select the enlarged view frame.
2. Select the dimension to modify.
3. Right-click and select **Add dimension point**.

See also

Create anchor bolt plans using saved settings (page 127)

Add automatic dimensions to general arrangement drawings (page 794)

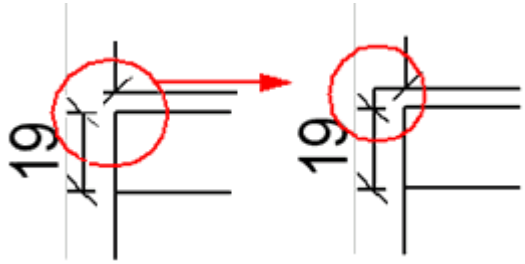
Link perpendicular dimension lines

You can connect two perpendicular dimension lines. Connecting dimension lines makes your drawings clearer and easier to read. For example, you might want to link dimension lines of embedded objects in a cast unit, floor beams in a floor plan, or anchor bolts in an anchor bolt plan.

1. In an open drawing, hold down **Ctrl** and select two perpendicular dimension lines to connect.
2. On the **Dimensioning** tab, click **Link dimensions**.
3. If you want to unlink the dimension lines you linked, select the linked dimension line and, on the **Dimensioning** tab, click **Unlink dimensions**.

Example

Below is an example of linked dimension lines:



See also

Modify dimensions (page 260)

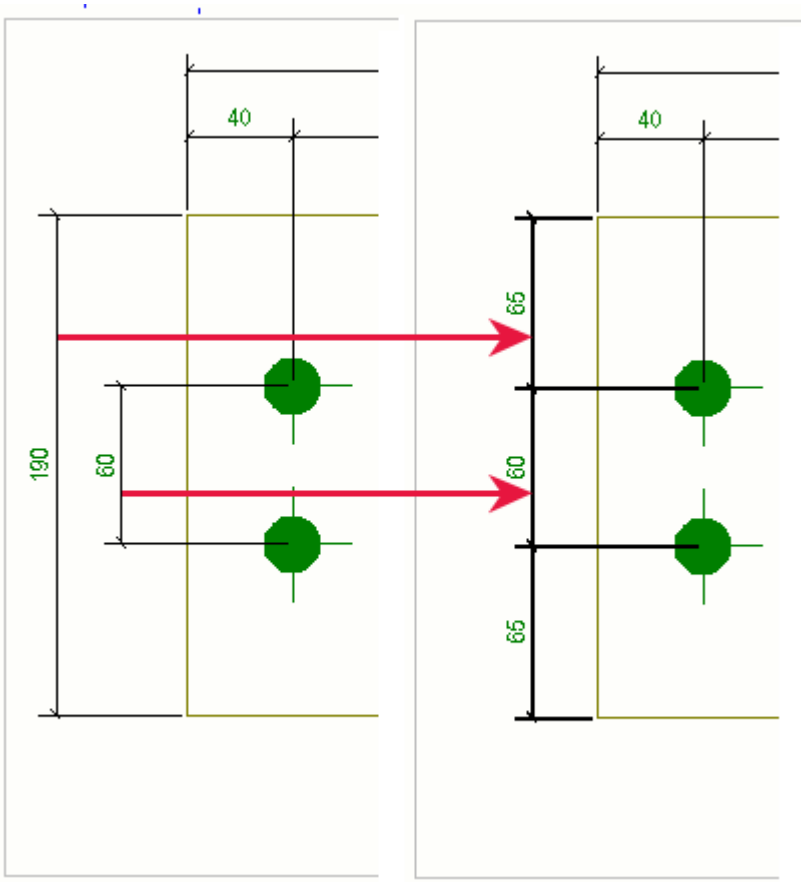
Combine dimension lines

You can manually combine a group of two or more parallel dimension lines into one line.

Note that you cannot uncombine the combined dimension lines, you need to create the dimensions again from the start.

1. Hold down **Ctrl** and click the dimension lines you want to combine.
2. On the **Dimensioning** tab, click **Combine lines**.

   You can also right-click and select **Combine dimension lines**.



See also

Modify dimensions (page 260)

Drag dimension marks

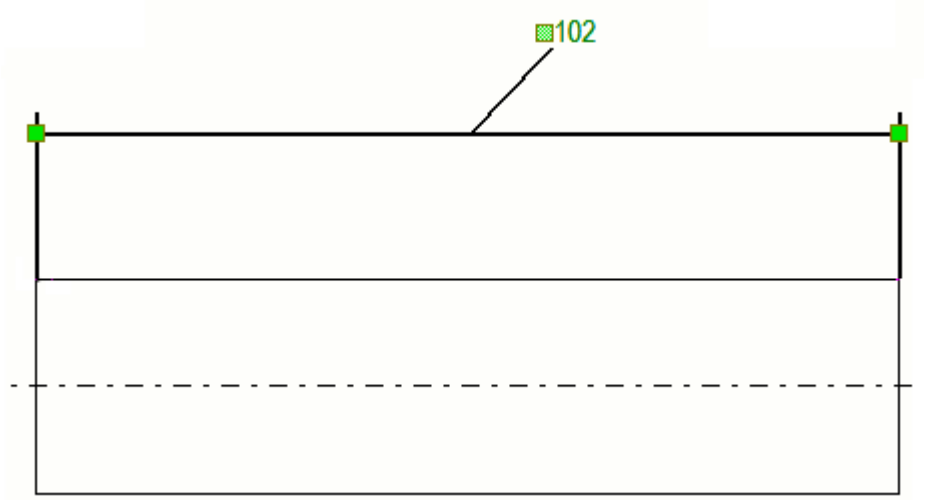
Dimension marks can be freely dragged to avoid overlapping dimensions and marks.

Note that you can only drag relative dimensions, not absolute dimensions.

Go to **File** --> **Settings** and ensure that **Drawing drag & drop** is selected .

1. Point the handle, press down the left mouse button, and drag the dimension mark to the desired location.

Tekla Structures automatically adds a leader line to the dimension mark that is dragged from its original position.



Note that if you do not want to draw a leader line when a dimension text is dragged away from the dimension line, set the advanced option `XS_LEADER_LINE_TO_DRAGGED_DIMENSION_TEXT` to `FALSE` in the category **Dimensioning - general** in the **Advanced options** dialog box. The default value is `TRUE`.

See also

Drag, reshape and resize drawing objects (page 419)

Modify dimensions (page 260)

Move the end of the dimension line

You can move the end of the dimension line orthogonal to itself outside the part. The dimension tags are moved together with the end of the line. This is useful when the tags would otherwise cover part geometry or other objects, such as dimensions or marks.

Limitations: You can move the end of the dimension line in all straight dimensions, except elevation dimensions.

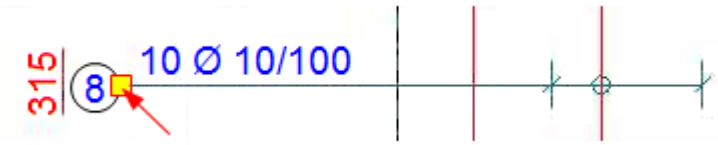
1. In an open drawing, click the dimension.

   The dimension line handle becomes visible.

2. Select the handle and drag the end of the line to the desired position.

   It is easier to select the handle when your hold down **Alt** and then click the handle.

   If the dimension contains dimension tags, the tags are visible while you drag.



4.9 Add marks, notes, and texts in drawings

You can add many kinds of annotation objects in your drawings, such as marks, notes, and texts. Some of these are associative and some independent.

Associative annotation objects

Associative annotation objects are updated according to the changes you make in a building object in the model. Dimensions, marks, and notes are associative.

In addition to the automatic marks and dimensions that you already may have in a drawing, you can add new ones and modify their properties:

Drawing content manager (page 290)

Add part marks manually in drawings (page 301)

Add reinforcement marks manually in drawings (page 303)

Add reinforcement marks with Rebar group marking application (page 305)

Adjust mark visibility in an existing drawing (page 326)

Add reinforcement ticks or reinforcement tags in drawings (page 339)

Show welds in drawings (page 489)

Add marks to spiral beams (page 443)

Add level marks in drawings (page 342)

Add associative notes in drawings (page 329)

Add dimensions manually to general arrangement drawings (page 217)

Modify marks or notes (page 335)

Drag leader line associativity point (page 364)

Update part and weld marks in drawings (page 336)

Delete marks for selected parts (page 337)

Check changed marks, notes and dimensions and remove change symbols (page 351)

Merge marks (page 356)

Add dimensions manually (page 207)

Independent annotation objects

Independent annotation objects are not linked or connected to the Tekla Structures model in any way. For example, texts, DWG/DXF files, symbols, section marks, detail marks, revision marks, various types of sketch objects, links and hyperlinks are all independent annotation objects.

Independent annotation objects will become associative if you snap to a building object when you create them, because then the objects will get associativity points. For example, in texts, the content does not change when the model changes but the associativity point may change. Another example is a line: if you snap to a part edge when you create a line in a drawing, the line will get an associativity point and move with the part in the drawing if the part changes in the model.

You can add and modify texts, shapes, section marks, detail marks, DWG/DXF files, links, hyperlinks, revision marks, sketch objects, and symbols in an open drawing only.

Add and modify section marks in drawings (page 343)

Add and modify detail marks in drawings (page 347)

Add and modify text in drawings (page 364)

Add revision marks in drawings (page 370)

Add and modify symbols in drawings (page 371)

Add links in drawings (page 380)

Add sketch objects in drawings (page 387)

See also

Drawing objects, views and layouts (page 68)

Add dimensions in drawings (page 206)

Drawing content manager

Drawing content manager is an interactive tool for checking and editing building objects and drawing content, especially marks, dimension marks, and associative notes, in the current drawing.

Use **Drawing content manager** to quickly and easily:

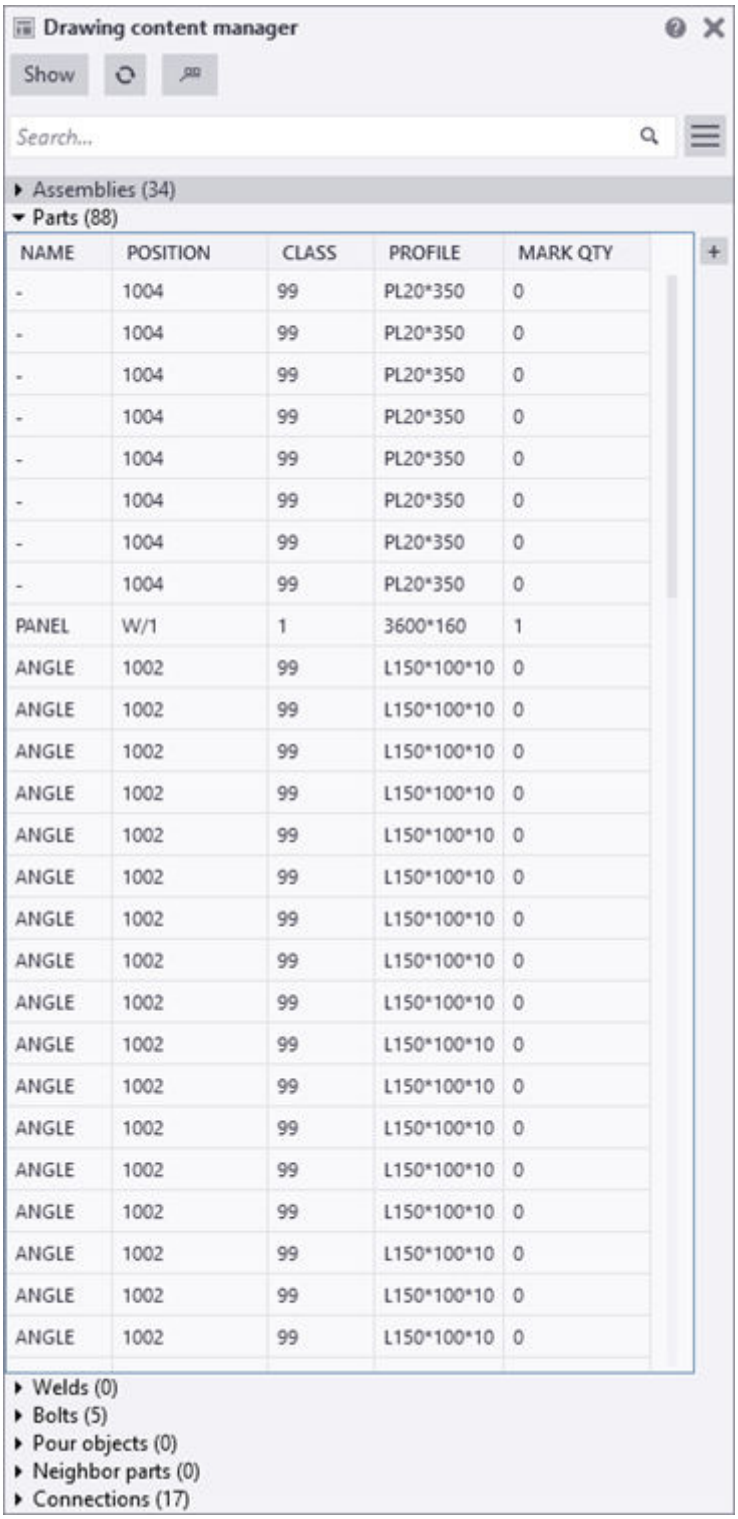
- Select relevant drawing content for checking and editing purposes, and target actions on single or multiple building objects or marks simultaneously.
- Add, modify or remove building object marks, dimension marks, and associative notes.
- Add weld marks to model welds in drawings.
- Highlight either objects or marks.

- Check how many marks building objects have in one view or in the entire drawing.
- Add other content to building objects depending on the object type, such as rebar dimension lines.
- Modify building object or mark properties.
- Hide/show building objects.

With this tool you can avoid duplicate, missing, or false information, and efficiently check if all necessary content exists, without having to manually go through all drawing objects separately.

- To open **Drawing content manager**, first open a drawing and click the **Drawing content manager** button in the side pane. Then click **Show** to populate the **Drawing content manager** list. If you do not want to list all building objects, narrow your selection by selecting certain drawing views, drawing areas or multiple objects, and then click **Show**.

  TIP If you suspect that some objects are missing from the list, go to the drawing view properties by double-clicking the drawing view frame and set the needed building objects visible.



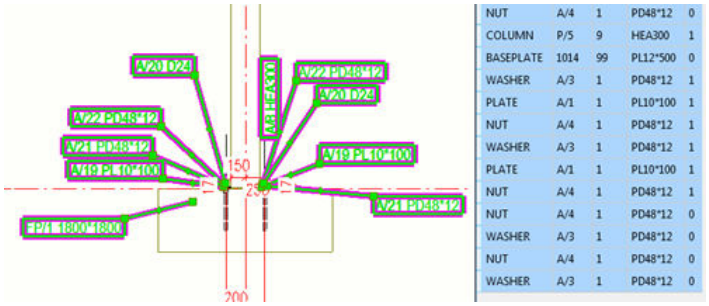
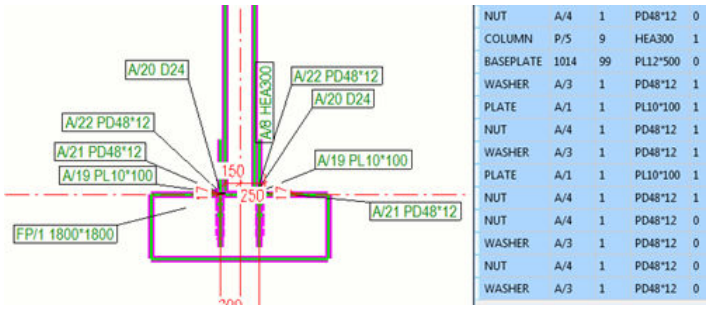
NAME	POSITION	CLASS	PROFILE	MARK QTY
-	1004	99	PL20*350	0
-	1004	99	PL20*350	0
-	1004	99	PL20*350	0
-	1004	99	PL20*350	0
-	1004	99	PL20*350	0
-	1004	99	PL20*350	0
-	1004	99	PL20*350	0
-	1004	99	PL20*350	0
PANEL	W/1	1	3600*160	1
ANGLE	1002	99	L150*100*10	0
ANGLE	1002	99	L150*100*10	0
ANGLE	1002	99	L150*100*10	0
ANGLE	1002	99	L150*100*10	0
ANGLE	1002	99	L150*100*10	0
ANGLE	1002	99	L150*100*10	0
ANGLE	1002	99	L150*100*10	0
ANGLE	1002	99	L150*100*10	0
ANGLE	1002	99	L150*100*10	0
ANGLE	1002	99	L150*100*10	0
ANGLE	1002	99	L150*100*10	0
ANGLE	1002	99	L150*100*10	0
ANGLE	1002	99	L150*100*10	0
ANGLE	1002	99	L150*100*10	0
ANGLE	1002	99	L150*100*10	0
ANGLE	1002	99	L150*100*10	0
ANGLE	1002	99	L150*100*10	0
ANGLE	1002	99	L150*100*10	0

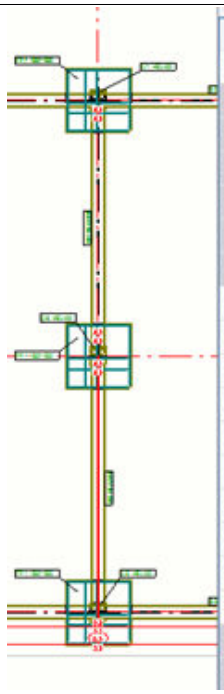
- **Drawing content manager** lists building objects in the drawing, and by default the related mark content and quantity. You can add and change property columns. Associative notes and dimension marks are also considered and counted as marks.

- You can only see the contents of one category at the time. If you have a lot of objects in a category, and you need to scroll, the objects are loaded as you scroll.
- The following building object categories are available in **Drawing content manager** if they exist in the drawing, and have been set visible in the drawing **View properties**:
  - **Assemblies**
  - **Parts**
  - **Welds**
  - **Bolts**
  - **Reinforcement**
  - **Pour objects**
  - **Neighbor parts**
  - **Neighbor reinforcement**
  - **Connections** - Connection objects have a symbol representation in drawings. By default, the connection representation symbol is the symbol number 142 in the `xsteel.sym` file in the `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\symbols\` folder. You can modify the connection representation symbol in Symbol Editor. Connection symbols are only highlighted in a drawing when you have selected the objects from the **Drawing content manager** list, otherwise connection symbols are hidden.

How to use the building object list

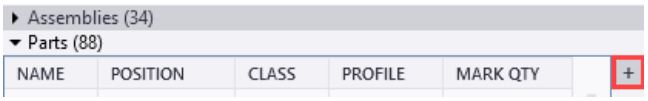
To	Do this
Show in the **Drawing content manager** list the building objects that are hidden in the drawing	Click **Show hidden items** on the **Options** menu.
Select and highlight the selected building objects or marks in all drawing views where they are visible	Click **Show in all views**on the **Options** menu.
Refresh the **Drawing content manager** list with correct mark quantity information	Click **Update**. Do this every time you hide some building objects, or if the number of marks changes.

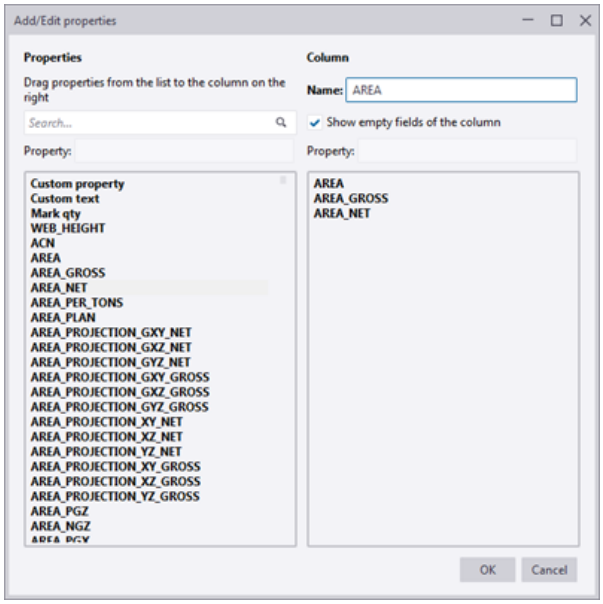
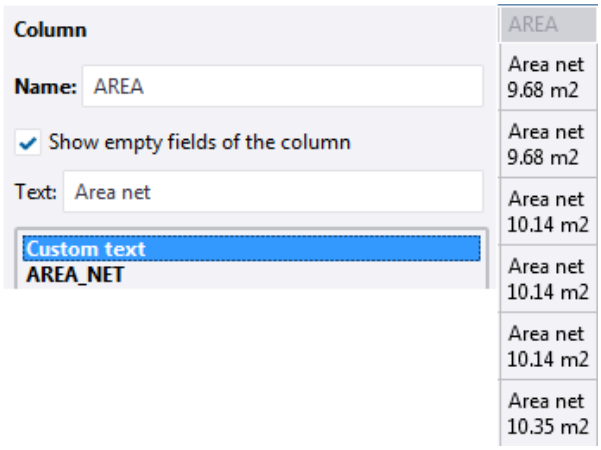
To	Do this
Select and highlight either building objects or marks in the drawing	Select the objects in **Drawing content manager** and change the selection from object to mark using the switch . When building object selection is active, the switch is gray , and when mark selection is active, the switch is blue . Marks selected:  Parts selected: 
Highlight objects in the **Drawing content manager** list	Select and highlight objects in the drawing. Parts selected in the drawing are highlighted in the list:

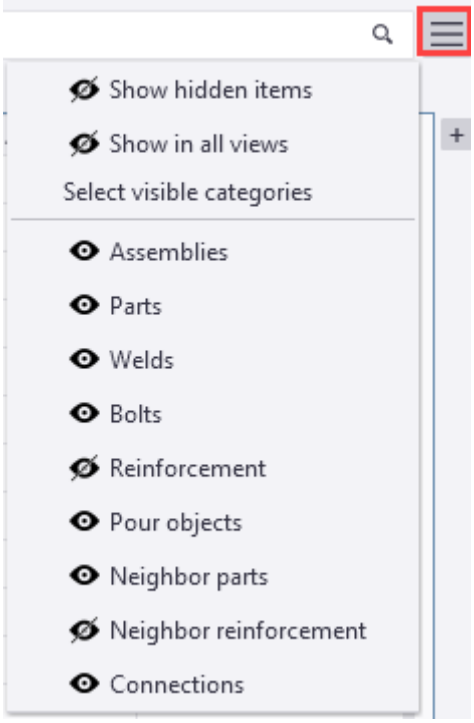
To	Do this
	

NAME	POSITION	CLASS	PROFILE	MARK QTY
PAD FOO	Concrete_FP-	8	1500*1500	0
PAD FOO	Concrete_FP-	8	1500*1500	0
PAD FOO	Concrete_FP-	8	1500*1500	0
PAD FOO	Concrete_FP-	8	1500*1500	0
COLUMN	c/1	7	HEA400	0
COLUMN	c/1	7	HEA400	0
COLUMN	c/2	7	HEA400	0
COLUMN	c/1	7	HEA400	0
COLUMN	c/1	7	HEA400	0
COLUMN	c/1	7	HEA400	0
COLUMN	c/1	7	HEA400	0
COLUMN	c/1	7	HEA400	0
COLUMN	c/1	7	HEA400	0
BEAM	b/1	3	HEA300	0
BEAM	b/1	3	HEA300	0
BEAM	b/1	3	HEA300	0
BEAM	b/2	3	HEA300	0
BEAM	b/2	3	HEA300	0
BEAM	b/2	3	HEA300	0

Collapse or expand building object categories	Use the arrow buttons ▶◢.
Search for and list specific building objects	Use the search box. You can use operators OR and AND. AND is the default, you do not need to type it. For example, to find all ground beams and footings, enter `FOOTING OR GROUND_BEAM`. To find only ground beams with position number GB/1, enter `GROUND_BEAM GB/1`.
Sort mark properties in alphabetical order	Click a column header.

To	Do this
Sort columns cumulatively	Click a column to sort by that column first and then hold down **Shift** and click another column to secondarily sort by that column.
Add a new property column in **Drawing content manager**	1. Click the arrow button next to a category to expand the category. 2. Click the + button nex to the the property column title row. The + button is displayed when a category is open.  3. 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

To	Do this
	properties. You can add several properties to the same column.  • If you want to add custom text in a column cell, select **Custom text** and type the required text in the displayed **Text** box. The press **Enter** to 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. 4. Enter the name for the property column and click **OK**.
Change a property column	Right-click the column name and select **Edit**.

To	Do this
Remove a property column	Right-click the column name and select **Remove**.
Change the order of the columns	Drag the columns to the new position.
Hide category types from the list of visible categories	Click the **Options** button to open the **Options** menu and click the categories on the displayed list. 
Save category properties	All generic category properties are saved in the `DrawingContentManagerCategories.xml` file located by default in the system folder defined for the advanced option `XS_SYSTEM`. All property column modifications (new columns, edited columns, column order) are saved in the `DrawingContentManagerCategories_<user>.xml` file in the `\attributes` folder under the current model folder. First, Tekla Structures tries to find the user-specific setting file `DrawingContentManagerCategories_<user>.xml`. If it cannot be found, Tekla Structures will next look for `DrawingContentManagerCategories.xml`.

To	Do this
	You can convert the user-specific file to a generic one just by removing `_<user>` from the file name. The folder search order for the generic settings file is the following: • `\attribute` folder under the model folder • Project folder (`XS_PROJECT`) • Firm folder (`XS_FIRM`) • System folder (`XS_SYSTEM`) • Folder defined by the advanced option `XS_USER_SETTINGS_DIRECTORY`
Copy selected rows to clipboard to be used in Microsoft Excel	Select rows in the **Drawing content manager** list and use **Ctrl+C** to copy the selected rows to the clipboard in the `.csv` format. You can then paste the rows to a Microsoft Excel sheet. Hold down **Crtl** or **Shift** to select several rows.

Add marks to building objects in a drawing

You can add marks and notes to all visible building objects in a drawing.

1. Click **Show** to populate the **Drawing content manager** list. If you do not want to list all building objects, narrow your selection by selecting certain drawing views, areas or multiple objects, and then click **Show**.
2. Select the building objects that need marks from the **Drawing content manager** list.

   You can only select multiple objects inside one category. Activate the switch to select only objects in the drawing. The selected objects are highlighted in the drawing.

   To select the whole object category, click the category name, for example, **Parts** or **Welds**.
3. Right-click in the drawing and select one of the following options:

   Add associative note: Notes are created using the current associative note properties.

   Add mark --> **Using view properties**: Marks are created using the view level mark properties.

   Add mark --> **Using applied mark properties**: Marks are created using the current mark properties.

   Add weld mark: Marks are created using the view level mark properties.

   Add mark --> **Dimension mark**: Marks are created using current applied dimension properties.

Tekla Structures creates the marks or notes. The added marks are shown in **Drawing content manager**. To update the mark quantity, click **Update**.

Note that if you select several building objects in the drawing at the same time, and if some of the selected objects already have marks, the content of the existing marks does not change.

Check mark count

1. Click **Show** to populate the **Drawing content manager** list. If you do not want to list all building objects, narrow your selection by selecting certain drawing views, areas or multiple objects, and then click **Show**.
2. If you want to know the mark count in the whole drawing, click **Show in all views** on the **Options** menu.
3. To include hidden objects in the list and show their mark count as well, click **Show hidden items** on the **Options** menu.
4. To make sure that the mark information is up to date, click **Update**.

   Note that **Drawing content manager** supports counting all kinds of building object marks, dimension marks, and associative notes.

Hide building objects from drawing or drawing view

1. Click **Show** to populate the **Drawing content manager** list. If you do not want to list all building objects, narrow your selection by selecting certain drawing views, areas or multiple objects, and then click **Show**.
2. Select the building objects from the list, right-click in the drawing and select **Hide/Show** --> **Hide from drawing view** or **Hide from drawing**.

Delete marks

If you are not happy with the existing marks, you can delete them.

1. Click **Show** to populate the **Drawing content manager** list. If you do not want to list all building objects, narrow your selection by selecting certain drawing views, areas or multiple objects, and then click **Show**.
2. Click **Show in all views** and **Show hidden items** on the **Options** menu to show in the list building objects in the whole drawing, including the hidden building objects.
3. If you are looking for certain building objects, use the **Search...** box.

4. Activate the mark selection .
5. In **Drawing content manager**, select the building objects whose marks you want to delete.
6. Press **Delete** on the keyboard, or right-click in the drawing and select **Delete**.

   If you have activated **Show in all views** and **Show hidden items**, marks are deleted from all views, including marks for hidden building objects.

7. Click **Update** to update the mark information in the list.

Modify building object or mark drawing properties

- To modify the properties of building objects in the drawing, select the objects from the **Drawing content manager** list, right-click in the drawing and select **Properties...**. If you are looking for certain building objects, use the **Search...** box.
- To modify the properties of the marks, select the objects from the list, activate mark selection , hold down **Shift** and double-click a mark.
- If you wish to list some of the marks, such as weld marks, but not show the marks in the drawing, use the first color option in weld mark properties, it will show the marks invisible. First select the objects from the list, activate mark selection , hold down **Shift** and double-click a weld mark. Then change the weld mark text and line color and click **Modify**.

Add part marks manually in drawings

In an open drawing, you can add part marks for all building objects, such as parts, bolts, surface treatment and connections, or for the selected building objects only.

TIP Use orthogonal snapping when you create marks and associative notes with leader lines. Orthogonal snapping allows you to place your marks and notes in a more consistent way in exact locations. When you start dragging a mark or note from a leader line end handle, and you have orthogonal snapping on, the leader line end handle locks to the closest orthogonal point in the drawing (0, 45, 90, 135, 180 degrees). To activate orthogonal snapping, press **O**.

Before creating marks, ensure that you have the suitable part mark properties defined. To adjust the mark properties, do one of the following:

- If you create marks using applied properties: In an open drawing, select **Part mark** or any other mark type from the property pane object list.

Ensure that the displayed properties are the ones that you want to use, and modify the properties if needed. You can also load predefined properties.

- If you create marks using view level mark properties: Double-click the view frame of the view where you want to add the part marks, and select **Part mark** or another mark type in the options tree. Ensure that the displayed properties are the ones that you want to use, and modify the properties if needed. You can also load predefined properties.

NOTE When you create a mark manually, and the mark content is not defined for the object type you have selected in the drawing, a mark with a text element set to "Content undefined" is created, and you can then define the mark content and properties in the property pane.

For a detailed list of mark settings and their options, and instructions on using the mark editor, see Mark properties (page 972). For a list of elements that you can add in marks, see Mark elements (page 981). For instructions on modifying existing marks, see Modify marks or notes (page 335).

To	Do this
Add part marks for all building objects in a drawing	On the **Annotations** tab, click **Part mark** --> **For all parts** . Tekla Structures adds the part marks by using the current view level part mark properties. If you have deleted marks manually from the drawing, this command will not create the marks, and you will have to create them manually part by part.
Add part marks for selected building objects in a drawing	1. Select the building objects in the drawing: • Activate the **Select parts in drawings** selection switch and select the parts using area selection, for example. • Open the **Drawing content manager** from the side pane, and click **Show** to populate the **Drawing content manager** list. You can also narrow down the selection by selecting areas, single or multiple views, or single or multiple building objects. Then ensure that building object selection is active in **Drawing content manager**, and select the desired building objects from the list.

To	Do this
	2. Create the marks by doing one of the following: • On the **Annotations** tab, click **Part mark** --> **For selected parts** . Tekla Structures adds the part marks by using the current view level part mark properties. • Right-click and select **Add mark** and then select **Using view properties** or **Using applied mark properties**. Note that if you select several building objects at the same time, and some of the selected objects already have marks, the content of the existing marks does not change.

Modify mark properties

1. To modify the mark properties, click the mark. If the property pane is not open, double-click the mark. If you want to modify several marks, hold down **Ctrl** and click the marks.

   Click the **Properties** or **Values** button to show either the property name or the property value in the mark editor view. These buttons are only available when you edit an existing mark.
2. Modify the properties.

   If you have selected several marks, the properties that have different values have the text *Varies*. You can select another value from the list.
3. Click **Modify** to apply the changes in all selected marks.

Delete unnecessary marks

- To delete a mark, select the mark and press **Delete** on the keyboard, or right-click and select **Delete**. You can also delete marks in (page 290), where you can narrow your selection by selecting certain drawing views, areas or multiple marks, and then delete those.

See also

Modify marks or notes (page 335)

Mark elements (page 981)

Mark properties in drawings (page 972)

Add reinforcement marks manually in drawings

You can add marks manually to reinforcement and meshes in drawings.

Reinforcement marks may already exist in the drawing if you have selected to create the marks at drawing creation (page 811). If there are no marks, you can add them manually.

NOTE If you have created rebar marks automatically but you cannot see the marks, recreate the marks or modify the view or drawing level settings for rebar or neighbor rebar marks.

Before creating marks, ensure that you have the suitable reinforcement mark properties available. To adjust the mark properties, do one of the following:

- If you create reinforcement marks using applied properties: In an open drawing, select **Rebar mark** in the property pane object list. Ensure that the displayed properties are the ones that you want to use, and modify the properties if needed. You can also load predefined properties.
- If you create reinforcement marks using view properties: Double-click the view frame of the view where you want to add the part marks, and select **Reinforcement mark** in the options tree. Ensure that the displayed properties are the ones that you want to use, and modify the properties if needed. You can also load predefined properties.

NOTE When you create a reinforcement mark manually, and the mark content in the mark properties is not defined for the object that you have selected in the drawing, a mark with a text element set to "Content undefined" is created, and you can then select the mark and modify the mark content in the property pane.

For a detailed list of mark settings and their options, and instructions on using the mark editor, see Mark properties (page 972). For a list of elements that you can add in reinforcement marks, see Mark elements (page 981). For instructions on modifying existing marks, see Modify marks or notes (page 335).

1. Open a drawing that contains reinforcement.
2. Select the reinforcement by doing one of the following:
   - Hold down **Shift** and select the desired reinforcement.
   - Open the **Drawing content manager**, and click **Show** to populate the **Drawing content manager** list. You can select areas, single or multiple views, or individual reinforcement. Then ensure that building object selection is active , and select the desired reinforcement from the list.
3. Right-click and select **Add mark**, and then select **Using applied mark properties** to use the applied reinforcement mark properties, or **Using view properties** to use the view level reinforcement mark properties.

The reinforcement marks are created.

You can drag the reinforcement marks and the leader line associativity points to a new location.

Tekla Structures optimizes the leader line associativity point automatically. You can switch off the optimization and place the leader line associativity point where you want to by setting the advanced option XS_ENABLE_REBAR_MARK_LEADER_LINE_BASE_POINT_OPTIMIZATION to `FALSE`.

Modify mark properties

- To modify a reinforcement mark, in an open drawing, click a reinforcement mark. If the property pane is not open, double-click the mark. Then change the properties and click **Modify**.

Delete unnecessary rebar marks

- To delete a rebar mark, select the mark and press **Delete** on the keyboard, or right-click and select **Delete**. You can also delete marks in (page 290), where you can narrow your object selection by selecting certain drawing views, areas or multiple marks, and then delete those.

See also

Mark properties in drawings (page 972)

Mark elements (page 981)

Add reinforcement marks with Rebar group marking application

The **Rebar group marking** application offers different styles to create marks to rebar groups and distribution areas flexibly. With the **Rebar group marking** application you can efficiently create quality reinforcement drawings according to the market requirements.

Mark rebar groups

1. In a drawing, select the rebar groups to be marked.
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. In the **Applications** list, click **Rebar group marking**.
5. Pick the position for the leader line.
6. Pick the position for the mark.

7. To adjust the mark settings, double-click the mark, and make the necessary changes:
   - On the **Geometry** tab, define the shape and position of the rebar mark, and mark line and leader line settings.

     On the **Mark 1**–**Mark 3** tabs, define the content for the rebar mark, such as the diameter, cc distances and the number.

     On the **Lines and symbol** tab, define the generation of distribution and leader lines, the color of the lines, and the line type. You can also define the symbol created at the intersection of distribution lines and leader lines.
   - On the **Symbols on rebars** tab, define the lines and symbols for the rebar mark.

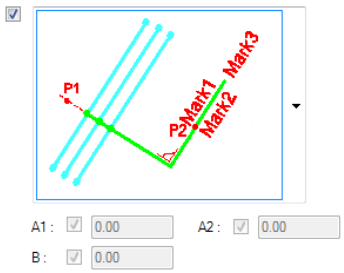
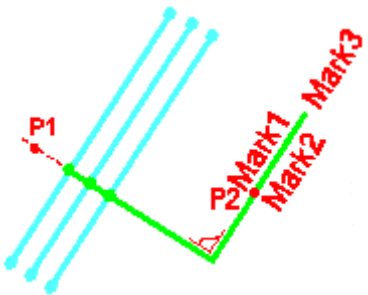
   For more information about the settings, see ***Rebar group marking settings*** below.
8. Click **Modify**.

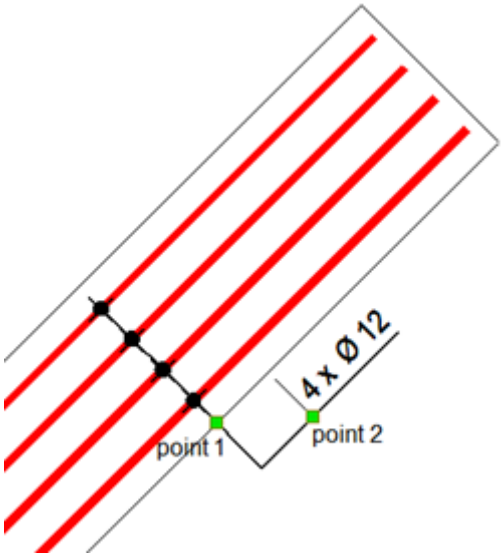
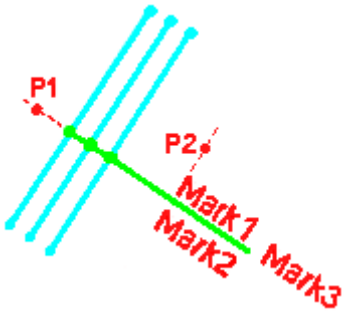
Tekla Structures creates the rebar marks according to the defined settings.

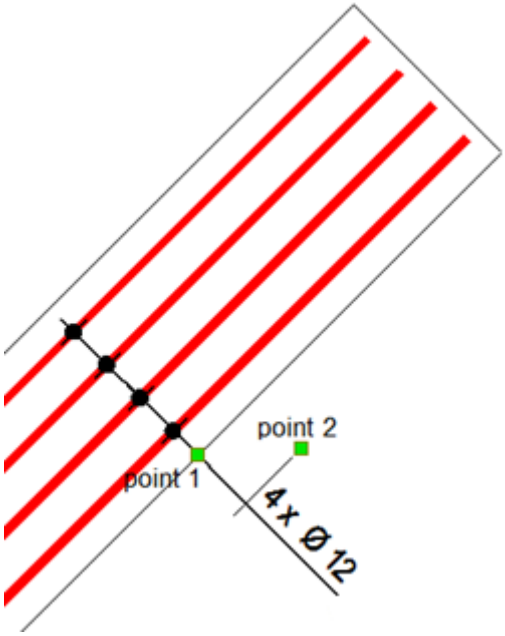
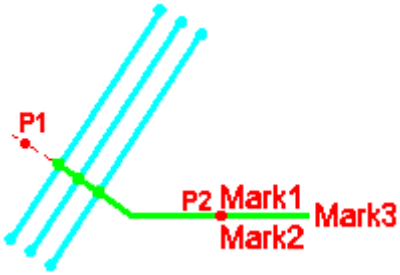
Rebar group marking settings

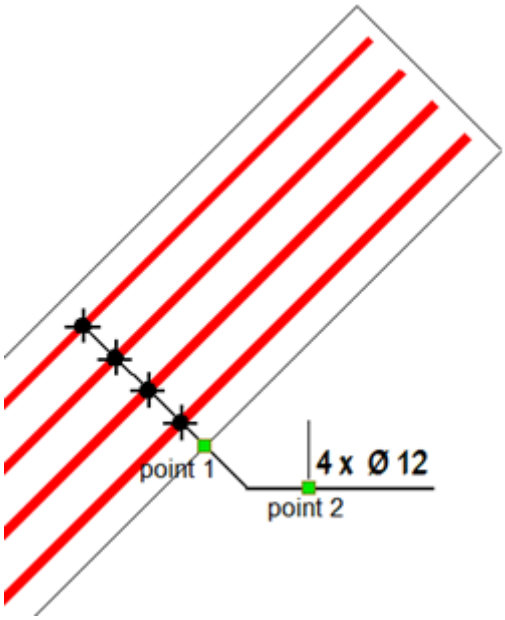
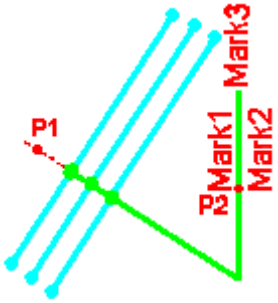
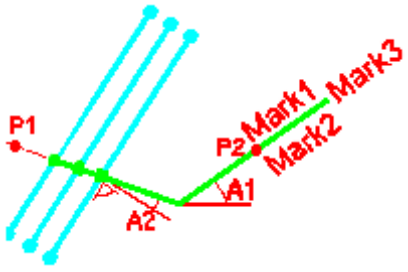
Geometry tab

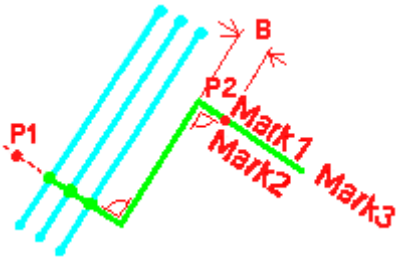
On the **Geometry** tab, define the shape and position of the rebar mark, and the mark line and leader line settings.

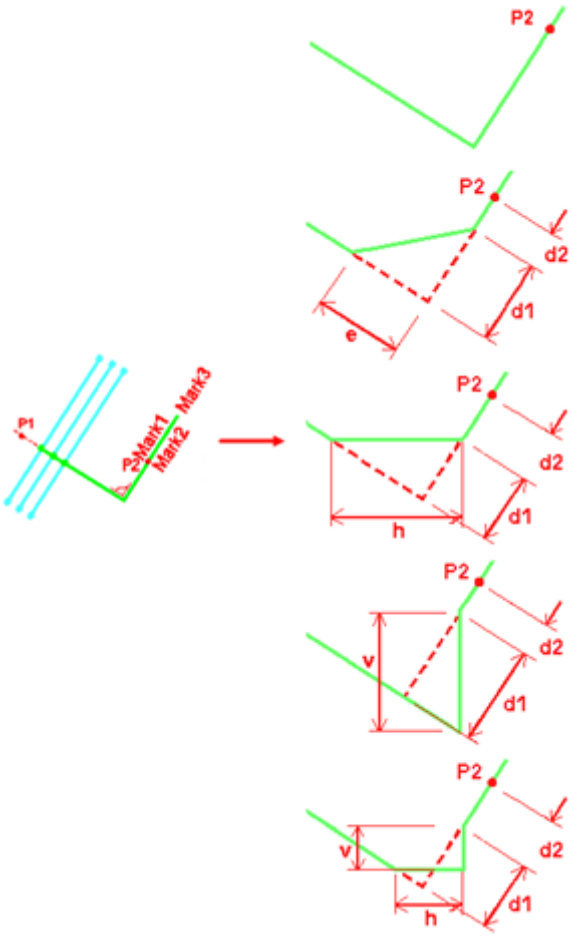
Setting	Options and descriptions
	Define the shape of the rebar mark. In the descriptive option images, P1 is the first point that you pick and P2 the second one. **Mark x** indicates the locations of the marks defined on the **Mark 1**–**Mark 3** tabs. The options are: Option 1:  • The leader line is perpendicular to the rebars. • The mark line is parallel to the rebars. Example:

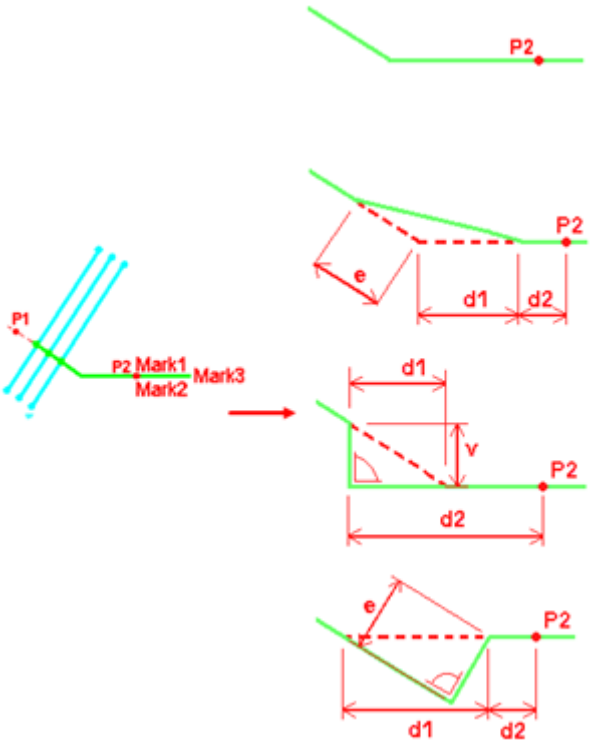
Setting	Options and descriptions
	 Option 2:  • The leader line is perpendicular to the rebars. • The mark line is also perpendicular to the rebars. Example:

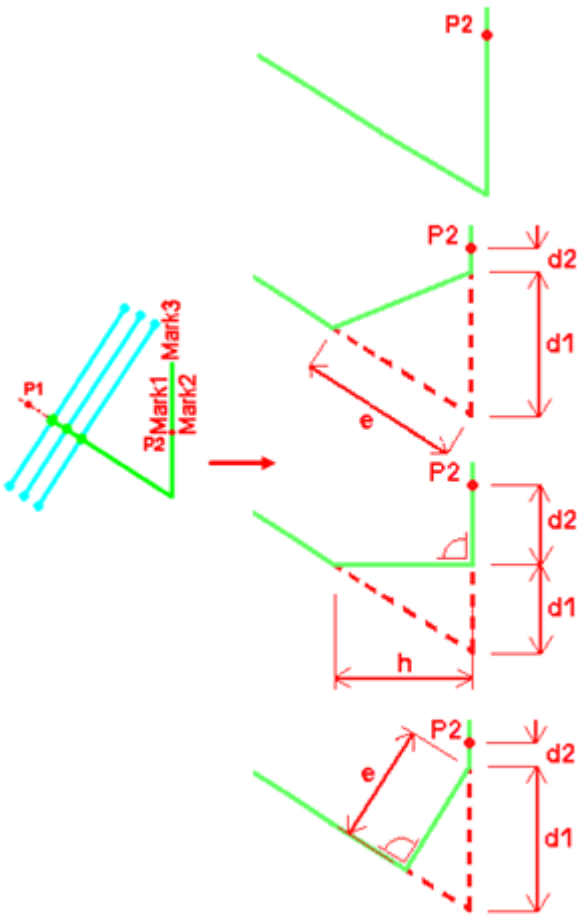
Setting	Options and descriptions
	 Option 3:  • The leader line is perpendicular to the rebars. • The mark line is horizontal. Example:

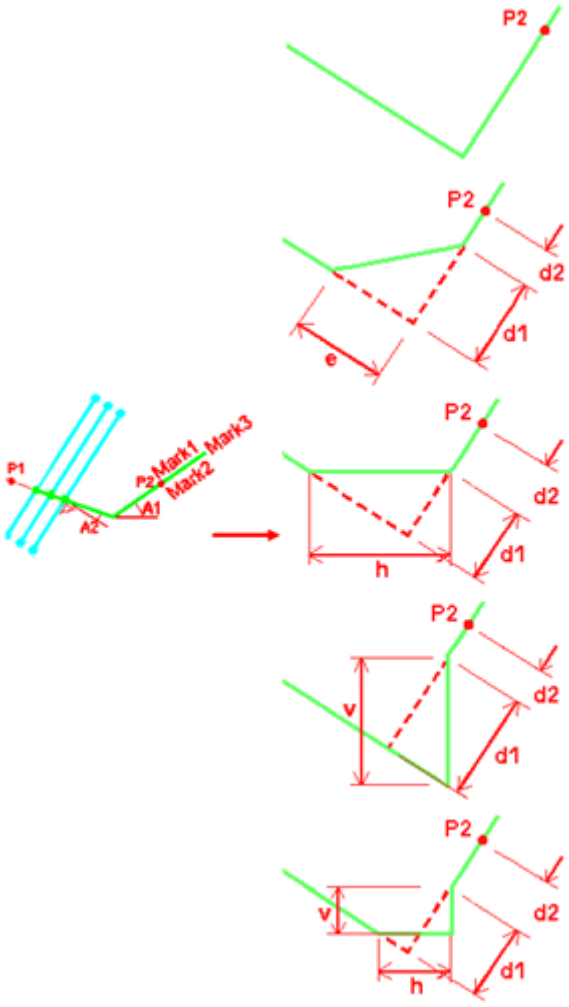
Setting	Options and descriptions
	 Option 4:  • The leader line is perpendicular to the rebars. • The mark line is vertical. Option 5:  • The leader line direction is defined by an angle relative to the rebars. The angle can be entered for the option **A2**.

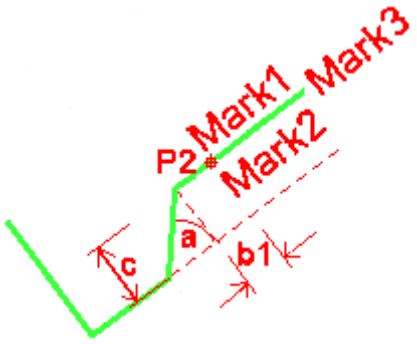
Setting	Options and descriptions
	• The mark line direction is defined by an angle relative to the horizontal line. The angle can be entered for the option **A1**. Option 6:  • See the image above to see how to pick the points P1 and P2. Use the option **B** to define the distance of the marks.

Setting	Options and descriptions
	Define the shape of the leader line. The available options depend on the selected mark line option. 

Setting	Options and descriptions
	

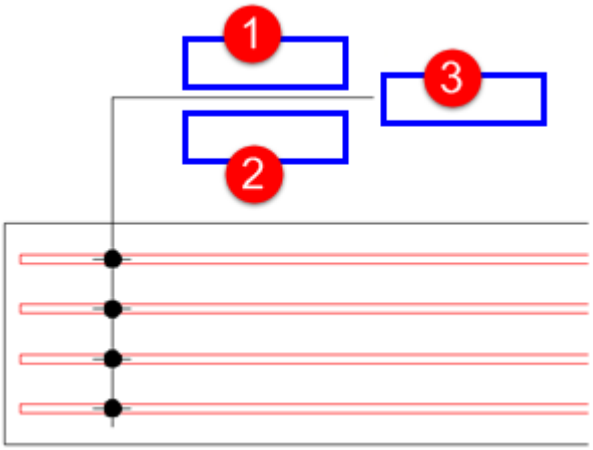
Setting	Options and descriptions
	

Setting	Options and descriptions
	 You can also affect the shape of the leader line for many of the options with the settings **d1**, **d2**, **h**, **v**, and **e**.
	Define the shape of the mark line. The options are

Setting	Options and descriptions
	 You can also affect the shape of the mark line for some of the options with settings **b1**, **b2**, **a**, and**c**, and **e**
	Define an offset for **Mark 3** from the mark line.

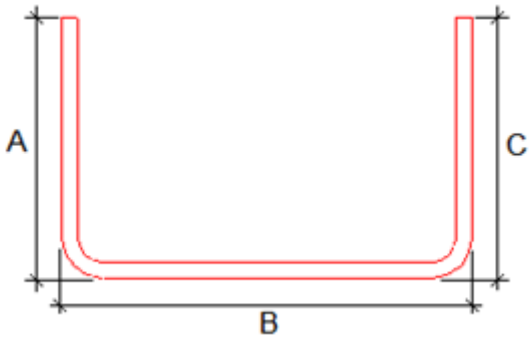
Mark 1–Mark 3 tabs

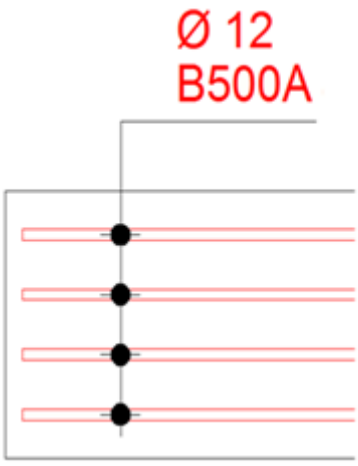
On the **Mark 1**–**Mark 3** tabs, you can define the content of the rebar marks. You can have three separate marks with the desired content within one rebar mark: **Mark 1**, **Mark 2** and **Mark 3**. Each of the marks may have extra marks. The following image shows the position of each mark within the entire rebar mark:

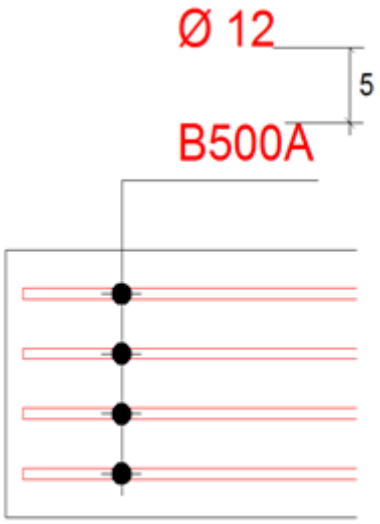
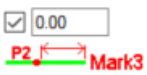


Setting	Options and descriptions
Available elements	Select the rebar information to be displayed in the mark such as grade, diameter and cc distances.
Elements in mark	List of the selected information that will be displayed in the mark.
Text properties	Define the text properties. The available properties files are the ones that have been defined and saved in Text properties (page 364).
Add frame	Add frame around mark elements. You can select the frame type and color. Note that symbol, new line, and pull-out picture cannot have a frame.
Create	Define whether the mark should be created or not. The options are **Yes** and **No**.
Extra mark: **Create**	Define whether and how the extra marks are placed in the mark. The options are: **No** **Before main mark** **Behind main mark**
Units	Define the units: • **Automatic** • **mm** • **cm** • **m** • **foot - inch** • **inch** Only available for the following mark content:

Setting	Options and descriptions
	• **Length** • **Cc** • **cc min** • **cc max** • **cc exact** • **cc target** • **Length itemized**
Format	Define the format: • ### • ###[.#] • ###[.##] • ###[.###] • ###.# • ### #/# • ###.## • ###.### Only available for the following mark content: • **Length** • **Cc** • **cc min** • **cc max** • **cc exact** • **cc target** • **Length itemized**
Precision	Define the precision: • 0.00 • 0.50 • 0.33 • 0.25 • 1/8 • 1/16 • 1/32 • 1/10

Setting	Options and descriptions
	• 1/100 • 1/1000 For example, for precision 0.33, the actual value 50.40 is displayed as 50.33. 1/8, 1/16, and 1/32 are for imperial units. 1/10, 1/100, and 1/1000 are used for defining precision without rounding.
Sum of exact lengths **Length along rebar axes** **Sum segm rebar axis** **Length TplEd**	**Sum of exact lengths**:  If you select **Length along rebar axes**, the length is calculated along the center line of the rebar: **Sum segm rebar axis** counts rebar lengths from rebar geometry. This option does not take into account bending radius lengths. **Length TplEd** takes rebar length from template attribute (user attribute property LENGTH). These options are only available for some of the following mark content: • **Length** • **Cc** • **cc min** • **cc max**

Setting	Options and descriptions
	• **cc exact** • **cc target**
Total number in rebar group **Number displayed in view** **Total number in cast unit**	These options are only available for mark content **Number**. **Total number in rebar group**: Displays the total number of rebars in the rebar group, regardless of the number of the physically visible rebars in the drawing. **Number displayed in view**: Only displays the number of the visible rebars in the drawing view. **Total number in cast unit**: Displays the total number of rebars in the cast unit.
Row distance	Define the distance between the mark rows. Example: **Row distance** = 0  Example: **Row distance** = 5

Setting	Options and descriptions
	
	Define the distance of the mark text from the leader line.
Mark rotation	Define how to rotate the mark text. You can rotate the mark text horizontally and vertically or to the mark line direction (default).
Center position number to line	Centers the rebar position number to the end line. This will automatically place the position number as the first element in the mark and align this element with the leader line.
Extra line length 	When **Mark 1** and **Mark 2** have no text, the extra line length setting on the **Mark 3** tab is activated. Specify the length of the line.

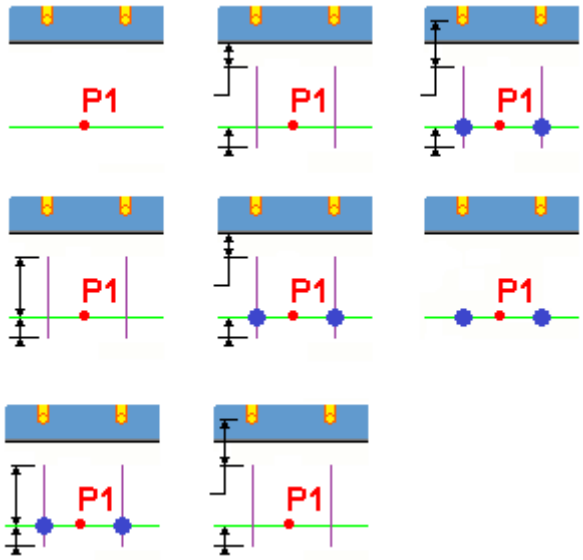
Lines and symbol tab

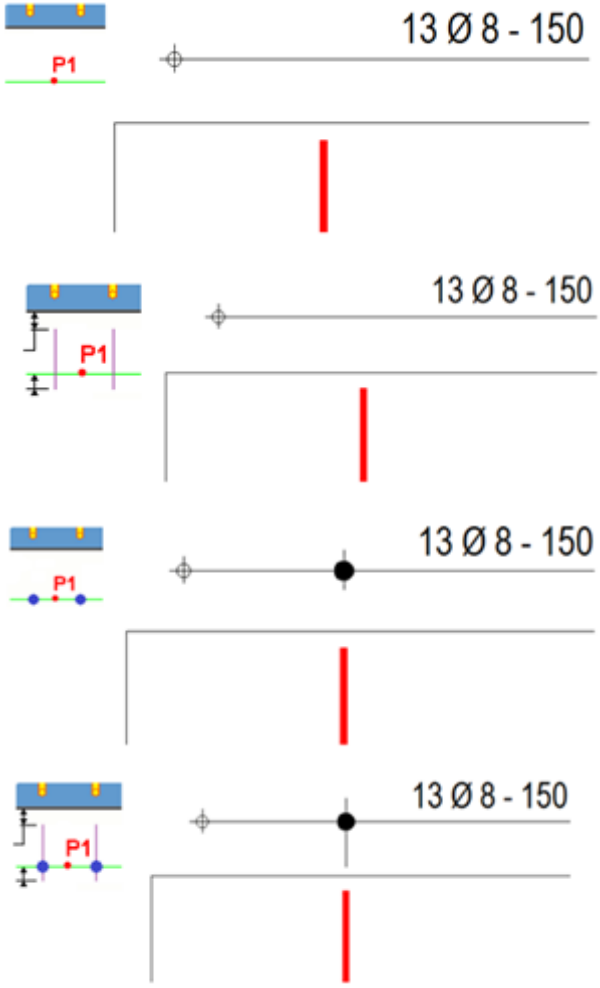
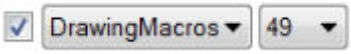
On the **Lines and symbol** tab, you can define the generation of distribution and leader lines, the color of the lines, and the line type. You can also define the symbol created at the intersection of distribution lines and leader lines.

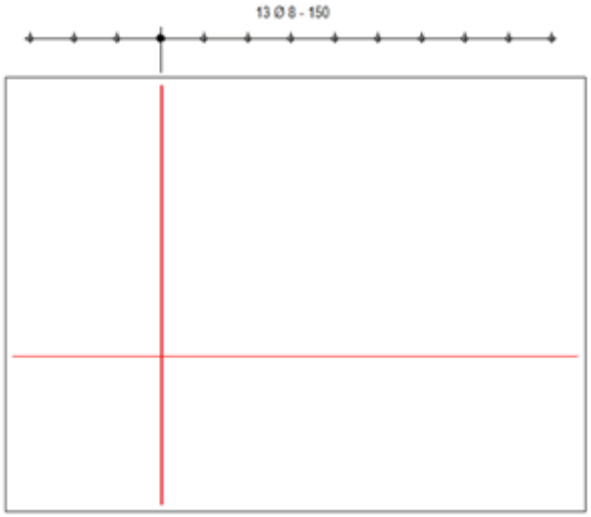
Setting	Options and descriptions
Distribution line: **Color** **Distribution line**: **Line type** **Leader line**: **Color** **Leader line**: **Line type**	Define the **Color** and **Line type** for the **Distribution line** and the **Leader line**.
Symbol between distribution and leader line	Set **Show** to **Yes** to show a symbol between distribution lines and leader lines. Then select the symbol file that you want to use and the symbol number. You can also change the symbol color and size.

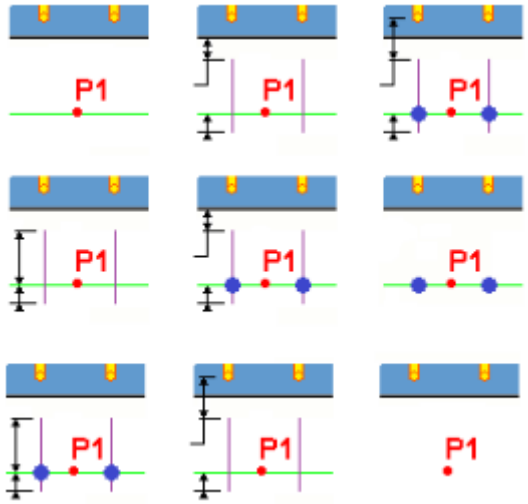
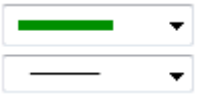
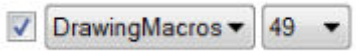
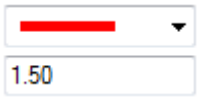
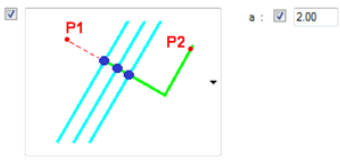
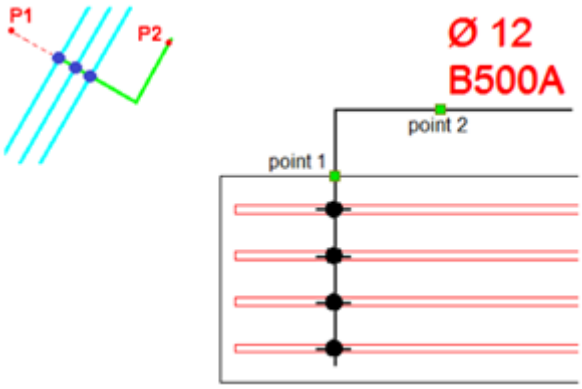
Symbols on rebar tab

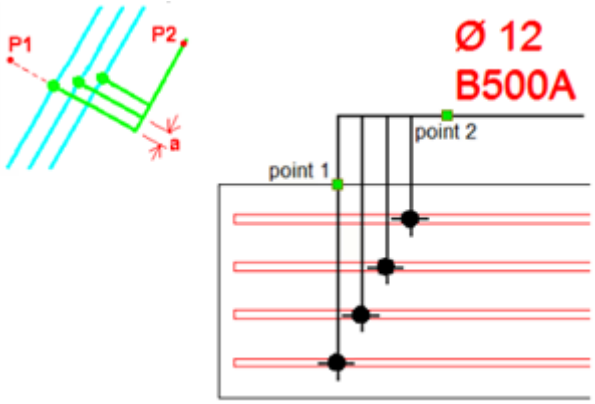
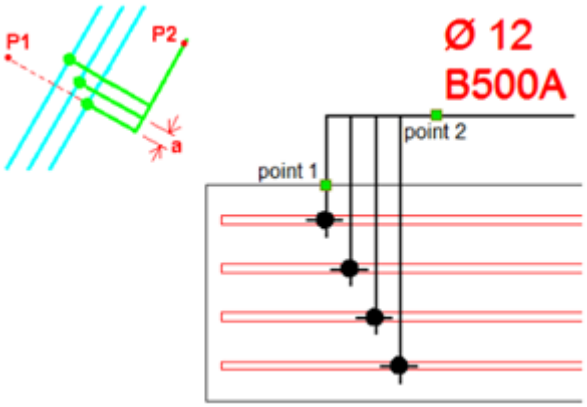
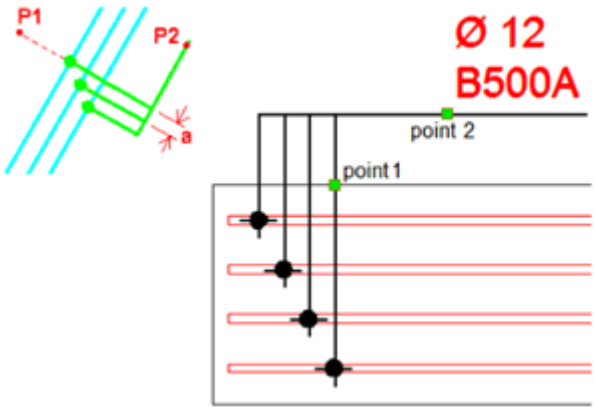
On the **Symbols on rebars** tab, you can define the lines and symbols for the rebar mark.

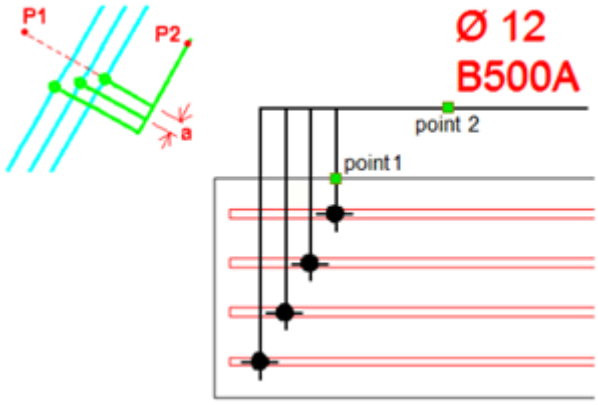
Setting	Options and descriptions
Visualized rebar s: **Small lines** **Visualized rebar s**: **Symbols**	Define the generation of the symbols and lines on the dimension lines of the visible rebars. The options are:  Examples:

Setting	Options and descriptions
	 : Define the color and the line type for the lines.  : Define the symbol file and symbol number to be used. You can use existing symbols in Tekla Structures by selecting a symbol file and a symbol number. : Define the color and the size of the symbol.
Not visualized rebars **First and last rebar**	The settings listed below only apply for **Not visualized rebars** and **Not visualized rebars**. The settings are only applicable if the option **bar in the**

Setting	Options and descriptions
	middle of group is defined (page 1016) for **Visibility of reinforcing bars in group** in reinforcement properties. Define if symbols/lines should be applied for **Not visualized rebars** or **Not visualized rebars**. Example of **Not visualized rebars**:  Example of **Not visualized rebars**: Define the generation of the symbols and lines for the dimension line. The options are:

Setting	Options and descriptions
	  : Define the color and the line type for the lines.  : Define the symbol file and symbol number to be used. You can use existing symbols in Tekla Structures by selecting a symbol file and a symbol number.  : Define the color and the size of the symbol.
 	Define how to place the symbols. The options and their examples are listed below: 

Setting	Options and descriptions
	  

Setting	Options and descriptions
	 You can define the distance between the lines by using the setting **a** if that is available for the option.

Adjust mark visibility in an existing drawing

In addition to defining the mark visibility settings in drawing properties before creating a drawing, you can also modify the mark visibility settings in an open drawing.

1. Open a drawing.
2. Double-click the drawing background to open drawing properties.
3. Do one of the following depending on the drawing type. Note that all settings are not available for all mark types.

Drawing type	Adjust mark visibility settings
Single-part, assembly and cast unit drawings:	1. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**. 2. Click a mark option in the options tree. For example, click **Part mark**. 3. Go to the **General** tab and select whether you want to display marks by selecting one of the **Visibility in view** options. The available options depend on the mark type: • **distributed**: Distributes the marks in the view. Tekla Structures only creates marks that are not visible in the other views. • **always**: Always creates marks in the view, irrespective the settings in other views.

Drawing type	Adjust mark visibility settings
	• **preferred**: Acts as distributed, but the preferred view has a higher priority. Select **preferred** to one view only in a drawing. If you set other views to **distributed**, the marks are located only in the view that has the setting **Visibility in view** set to **preferred**. • **none**: Does not create marks. Note that if you want to create your own marks manually, always use the **none** option. Selecting another option, for example **always**, might slow down the drawing update during opening, even if you had deleted the marks manually. 4. In **Parts out of view plane**, select whether you want to display marks for parts that are out of view plane: • **Visible**: Displays marks for parts outside the view in the drawing. • **Not visible**: Displays no marks for parts outside the view in the drawing. 5. For bolt marks, select if you want to show bolt marks in main parts, secondary parts, sub-assembly main parts, or in sub-assembly secondary parts. For bolt marks, you can also define the **Ignore size**, which filters standard-sized bolt marks out of drawings. Tekla Structures will not display bolt marks of the size you enter here. 6. For weld marks, set the following visibility options: • **Welds**, **Welds in sub-assemblies** and **Welds in hidden parts**: **None** displays no welds in the drawing. **Site** displays only site welds in the drawing. **Workshop** displays only workshop welds in the drawing. **Both** displays both site welds and workshop welds in the drawing. • **Weld size limit**: Enter a weld size to filter welds of that size out of the drawing. This

Drawing type	Adjust mark visibility settings
	is useful when you only want to show non-typical welds in a drawing. 7. Click to save the changes in view properties, and then click **Close** to return to drawing properties. 8. Define the following settings on the drawing level under **Common settings for all views**: • **Weld mark visible** controls the visibility of model weld marks in assembly drawings. The options are: • **In one view**: Tekla Structures automatically finds the most relevant view for showing the model weld marks. Every weld mark is shown in a drawing in one view only. • **In all views**: Tekla Structures adds the model weld marks in all drawing views that contain the part with the weld. • **Weld size limit**: Enter a weld size to filter welds of that size out of the drawing. This is useful when you only want to show non-typical welds in a drawing. This setting is available for single-part and assembly drawings. • **Ignore size** filters standard-sized bolt marks out of drawings. Tekla Structures will not display bolt marks of the size you enter here. This setting is available for single-part and assembly drawings. 9. Click **Modify**.
General arrangement drawings:	1. Click a mark button in drawing properties. For example, click **Part mark...**. 2. Go to the **General** tab and select whether you want to display marks by selecting one of the **Visibility in view** options. The available options depend on the mark type: • **distributed**: Distributes the marks in the view. Tekla Structures only creates marks that are not visible in the other views. • **always**: Always creates marks in the view, irrespective the settings in other views.

Drawing type	Adjust mark visibility settings
	• **preferred**: Acts as distributed, but the preferred view has a higher priority. Select **preferred** to one view only in a drawing. If you set other views to **distributed**, the marks are located only in the view that has the setting **Visibility in view** set to **preferred**. • **none**: Does not create marks. Note that if you want to create your own marks, always use the **none** option. Selecting another option, for example **always**, might slow down the drawing update during opening, even if you had deleted the marks manually. 3. In **Parts out of view plane**, select whether you want to display marks for parts that are out of view plane: • **Visible**: Displays marks for parts outside the view in the drawing. • **Not visible**: Displays no marks for parts outside the view in the drawing. 4. For bolt marks, select if you want to show bolt marks in main parts, secondary parts, sub-assembly main parts, or in sub-assembly secondary parts. For bolt marks, you can also define the **Ignore size**, which filters standard-sized bolt marks out of drawings. Tekla Structures will not display bolt marks of the size you enter here. 5. Click **OK**. 6. Click **Modify**.

See also

Mark properties in drawings (page 972)

Add associative notes in drawings

You can add associative notes in an open drawing. Associative notes are extra marks that contain additional information about the object they are attached to. Associative notes are intelligent, and updated according to the changes made in the related object in the model. You can add associative notes to

building objects in drawings, such as parts, bolts, reinforcement, surface treatments, edge chamfers, reference objects, pour breaks, and pour objects. You can add multiple notes to one object, whereas you can only add one mark to one object.

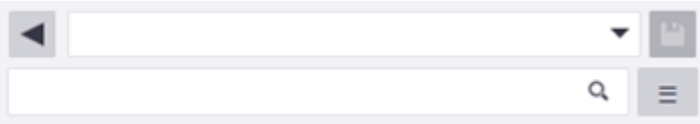
NOTE When you create a note, and the note content in the note properties is not defined for the object type you have selected in the drawing, a note with a text element set to "Content undefined" is created, and you can then select the note and modify the note content and properties in the property pane.

---

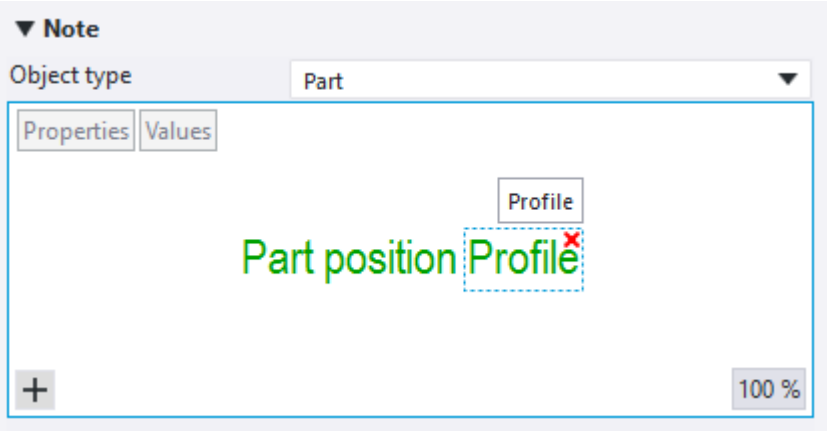
1. In an open drawing, on the **Annotations** tab, click **Note**. Then hold down **Shift** and select one of the following commands:
   - **With leader line**: Add an associative note with a leader line at the position that you specify.
   - **Without leader line**: Create an associative note without a leader line at the position that you specify.
   - **Along line**: Add an associative note along the line that you specify.

   Alternatively, you can select **Associative note** in the drawing property pane object list.

   You can also right-click a building object and select **Add associative note**. Then you can change the properties after adding the note.
2. You can load predefined note properties and search for properties:



3. In **Object type**, select the building object type. This defines the building object to which you attach the note.
4. In the note editor:

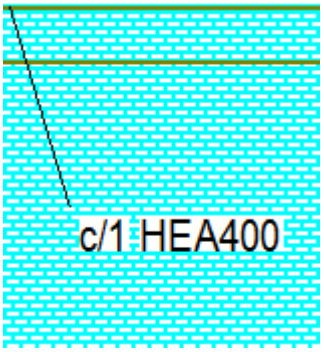
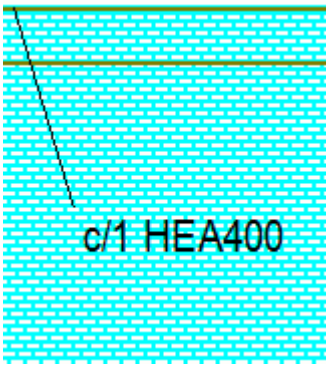


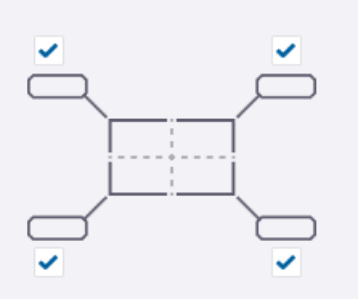
- Select a note container, click the plus button to open the element list, and select the element to add in your note container. You can also click the editor background first and then click the plus button.

  The available elements (page 981) vary according to the selected building object type.
- To delete an element, click the red delete button in the upper-right corner of the element. You can also click the element, and press **Delete** on the keyboard.
- You can drag elements and element containers to another location.
- The **%** button shows the current zoom level in the preview. You can zoom in or out by scrolling the middle mouse button. Click the button to zoom to best fit.

5. Modify the note properties.

Setting	Description
Font color, **Font**, **Font height**	Modify the font color, type, and height. To modify the font properties for the whole note, click the editor background. To modify the font properties for a single element, click the element in the editor. Clicking **...** gives more font choices.
Frame, **Frame color**	Add frame around the whole note or its elements, and define the frame type and color. You can also add a frame from the element list by clicking and selecting **Frame**.
Units, **Format**	If appropriate, change the unit and format. This property is only available for elements for which you can change the unit and format.

Setting	Description
Background	Select **Opaque** to hide the part of the drawing that is covered by the note.  Select **Transparent** to show the part of the drawing that is covered by the note so that the line work is visible, for example. 
Leader line, **Arrow**, **Arrow height**, **Arrow length**	Modify leader line properties. The **Leader line** options are: **No leader line** **Leader line** **Along line** You can also modify the arrow type, arrow height, and arrow length of the leader line.
Alignment	Select one of the alignment options: **Left** **Center**

Setting	Description
	Right **Leader line**
Rotation	Modify the rotation of the note by entering a new rotation angle.
Placing method	To place the note exactly in the position that you pick and keep it there, select **Fixed**. **Free** allows Tekla Structures to search for the first suitable location for the note. When you use the option **Fixed**, the mark stays where it is even though you update the drawing, whereas with **Free**, Tekla Structures tries to find the optimal place for the note. For more information about placing, see "Placement properties for marks, dimensions, notes, texts and symbols" (page 1033).
Position	Define the areas where Tekla Structures searches for a position to place the note. 
Distances	Define the distances and the search margin. **Distance s** (search margin) is the empty margin that you want to leave around the note. **Distance d min** (minimum distance) is the minimum distance of the note from the part.

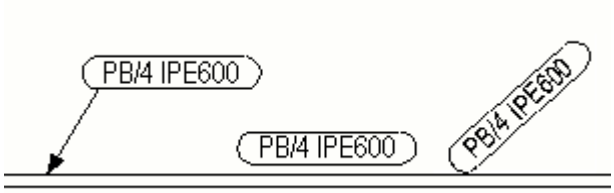
Setting	Description
	Distance d max (maximum distance) is the maximum distance of the note from the part. Note that if you use high values for **Distance s** and **Distance d min**, the note placement does not work properly.

6. Select the building object to which you want to add the note.

   Tekla Structures adds the associative note.

   To add the same note to another object, continue selecting. To stop adding notes, press **Esc**.

See below for some examples of associative notes. The one on the left is **With leader line**, the one in the middle **Without leader line**, and the one on the right **Along line**.



TIP For edge chamfers and other hard-to-see items, it is easier to use the context menu command **Add associative note** because then you do not have to select the object again after selecting the command.

Modify note properties

1. To modify the note properties, click the note. If the property pane is not open, double-click the note. If you want to modify several notes, hold down **Ctrl** and click the notes.

   Click the **Properties** or **Values** button to show either the property name or the property value in the note editor view. These buttons are only available when you edit an existing note.

2. Modify the properties.

   If you have selected several notes, the properties that have different values have the text *Varies*. You can select another value from the list.

3. Click **Modify** to apply the changes in all selected notes.

Delete unnecessary notes

- To delete a note, select the note and press **Delete** on the keyboard, or right-click and select **Delete**. You can also delete notes in (page 290), where you can narrow your selection by selecting certain drawing views, areas or multiple notes, and then delete those.

See also

Mark elements (page 981)

Mark properties in drawings (page 972)

Modify marks or notes

You can modify the properties of the marks and notes in an open drawing. You can modify the marks that were created automatically when you created the drawing as well as the marks and notes that you have manually added in existing drawings.

1. To modify the properties of a single mark or note, click it in an open drawing. If the property pane is not open, double-click the mark or note.

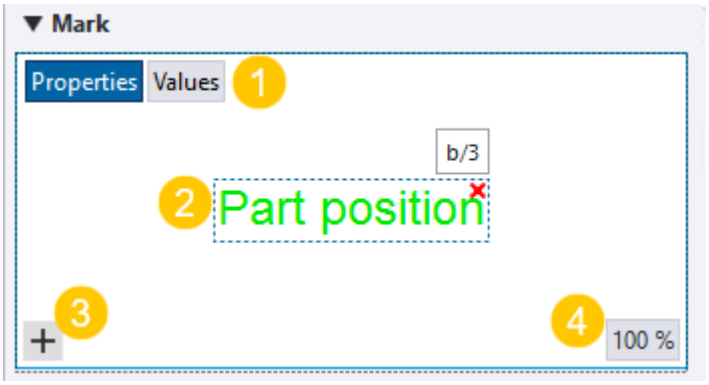
   To modify the properties of several marks or associative notes, do one of the following:

   - Hold down **Shift** and click the marks or notes that you want to modify.
   - Activate the **Select marks and notes in drawings** selection switch, right-click and select **Select part marks** and **From current drawing view** or **From all drawing views**.
   - Activate the **Select marks and notes in drawings** selection switch and use area selection to select the needed marks or notes.

2. Modify the mark or note contents and properties as required:

   If you have selected several marks or notes, the properties that have different values have the text *Varies*. You can select another value from the list.

   -

   (1) Click the **Properties** or **Values** button to show either the property name or the property value in the editor view. These buttons are available when you edit an existing mark or note.

   (2) The preview of the mark or note shows the elements that you have added, the selected representation, and the possible frames around the whole mark or around one or several mark elements.

(3) Use the plus button to open the element list and select the elements to add in your mark or note container. The available elements depend on the type of the selected object.

You can drag elements and containers in another position. To delete an element, click the red delete button ✖ in the upper-right corner of the element. You can also click the element, and press **Delete** on the keyboard.

If your mark or note already contains the element, the element has a text **In use**.

For more information about the available elements, see "Mark elements in drawing marks" (page 981).

(4) The **%** button shows the current zoom level in the preview. You can zoom in or out by scrolling the middle mouse button. If you have zoomed in or out, you can return to the best fit by clicking the **%** button.

For a detailed list of properties, see Mark properties in drawings (page 972).

3. Click **Modify** to apply the changes in all selected marks or notes.

Define marks (page 809)

Add part marks manually in drawings (page 301)

Mark properties in drawings (page 972)

Mark elements (page 981)

Update part and weld marks in drawings

You can update part marks and weld marks in an open drawing. Normally part marks and weld marks are up to date when you open the drawing. Updating is needed in frozen drawings.

To update marks in an open drawing, do any of the following:

To	Do this
Update all part marks	On the **Annotations** tab, click **Update** --> **All part marks** .
Update selected part marks	1. Select the part marks you want to update. 2. On the **Annotations** tab, click **Update** --> **Selected part marks** .

To	Do this
Update all weld marks	On the **Annotations** tab, click **Update** --> **All weld marks** .

Tekla Structures updates the marks according to your selection.

See also

Freeze drawings (page 577)

Delete marks for selected parts

You can easily select and delete marks for the parts that you have selected, even in big models.

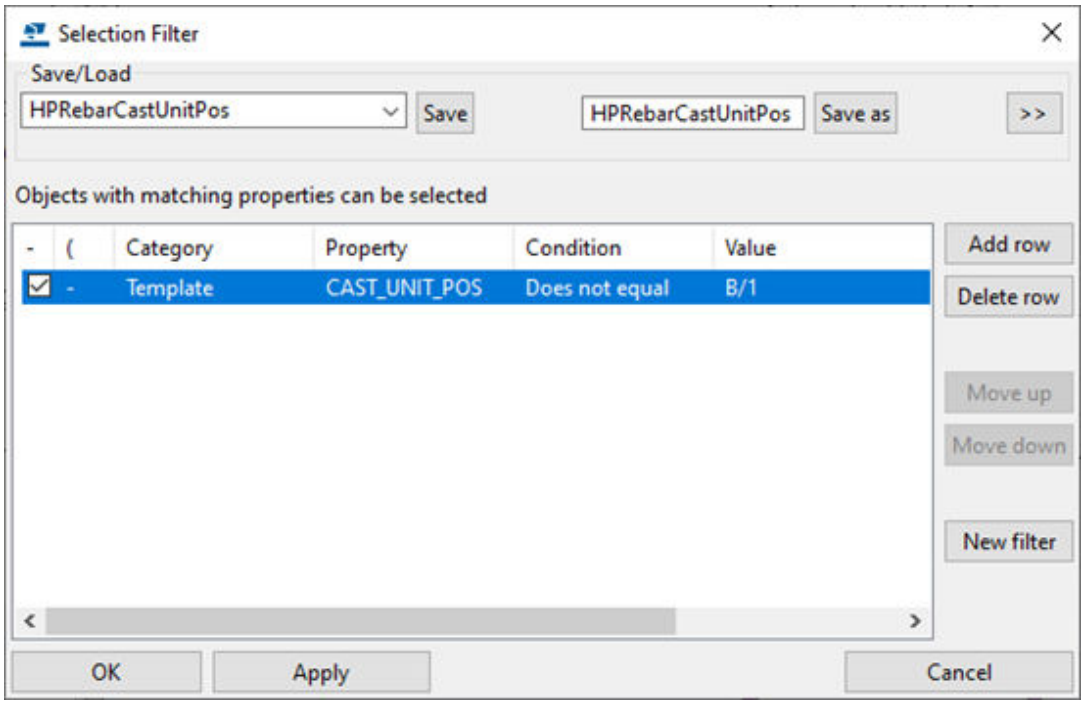
With the **Select part marks** command you can select the part marks that you want to delete in the current window or in all drawing windows. You might want to create a selection filter first, then use area selection to select the parts. Another way to select the marks to delete is to use the **Drawing content manager**.

To	Do this:
Delete marks for selected parts using selection filter	1. In an open drawing, click **Selection filter** and create a selection filter that filters out parts that you do not want to select. For an example of the selection filter, see **Example filter** below. 2. When you are ready, click **Apply**. 3. Only activate the **Select parts in drawings** selection switch. 4. Select the parts using area selection. 5. Right-click and select **Select part marks** and **From current drawing view** or **From all drawing views** . 6. To delete the marks, press **Delete** on the keyboard, or right-click in the drawing and select **Delete**.

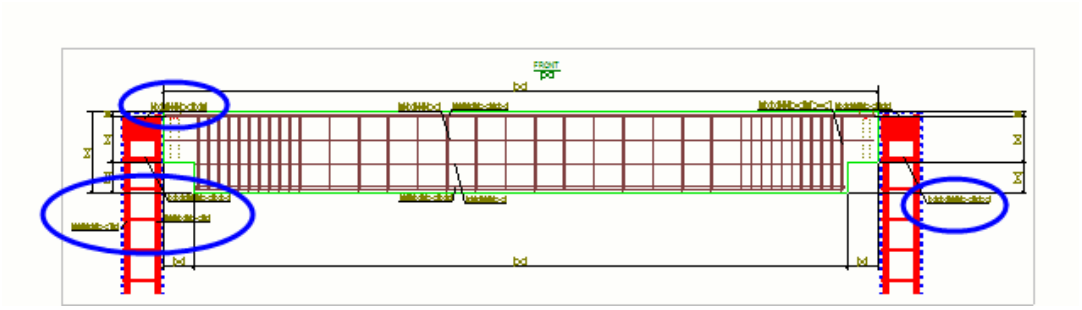
To	Do this:
Delete marks for selected parts using **Drawing content manager**	1. In **Drawing content manager**, click **Show** to populate the **Drawing content manager** list. You can narrow your selection by selecting certain drawing views, areas or multiple objects, and then click **Show** 2. Click the eye buttons next to **Show in all views** and **Show hidden items** to include in the list all building objects in the whole drawing, including the hidden building objects. 3. Click the toggle button to activate mark selection. When you do this, the button changes to yellow . 4. From **Drawing content manager**, select the building objects whose marks you want to delete. 5. Press **Delete** on the keyboard, or right-click in the drawing and select **Delete**.

Example filter

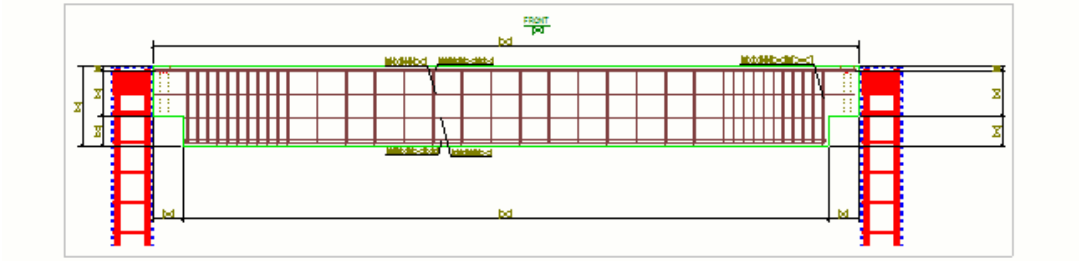
In the following example, you do not want to delete marks from cast units with a position number B/1. This means that when you use this filter, activate the **Select parts in drawings** selection switch, and use area selection to select the parts, all other parts get selected.



The image below shows the selected parts and the part marks that you want to delete.



The marks have been deleted.



Add reinforcement ticks or reinforcement tags in drawings

Reinforcement ticks and reinforcement tags are types of reinforcement marks that you can add for selected reinforcement in drawings. You can add

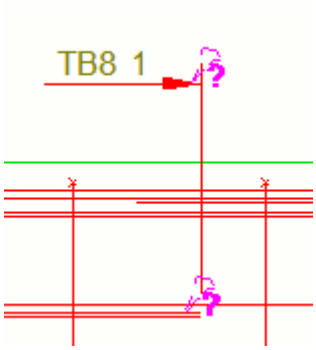
reinforcement ticks and reinforcement tags for individual reinforcement bars, reinforcement bar groups, and reinforcement bar sets.

You can add reinforcement ticks and reinforcement tags manually from the **Applications & components** catalog when you have a drawing open.

Add reinforcement ticks

1. Open a drawing that contains reinforcement.
2. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
3. In the catalog, double-click **Reinforcement tick**.
4. In the **Reinforcement tick** dialog box, adjust the used reinforcement mark attribute, mark alignment, arrow orientation, arrow head type, line color, arrow head dimensions, and arrow offsets according to your needs.
5. Click **Apply** or **OK**.
6. In the drawing, select the reinforcement.
7. Select a point near the end of the reinforcement to place the reinforcement tick.

See an example of a reinforcement tick below.



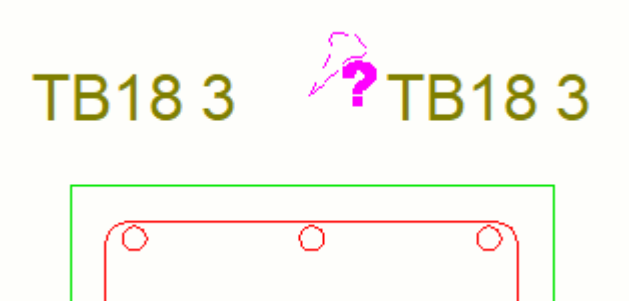
Add reinforcement tags

You can add reinforcement tags for reinforcing bars that are perpendicular to the drawing view.

1. Open a drawing that contains reinforcement.
2. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
3. In the catalog, double-click **Reinforcement tag**.

4. In the **Reinforcement tag** dialog box, adjust the used reinforcement mark attribute, tag pattern, tag orientation, and rotation angle according to your needs.
5. In the drawing, select the reinforcement.
6. Select a point near the end of the reinforcement to place the reinforcement tag.

See an example of a reinforcement tag below.

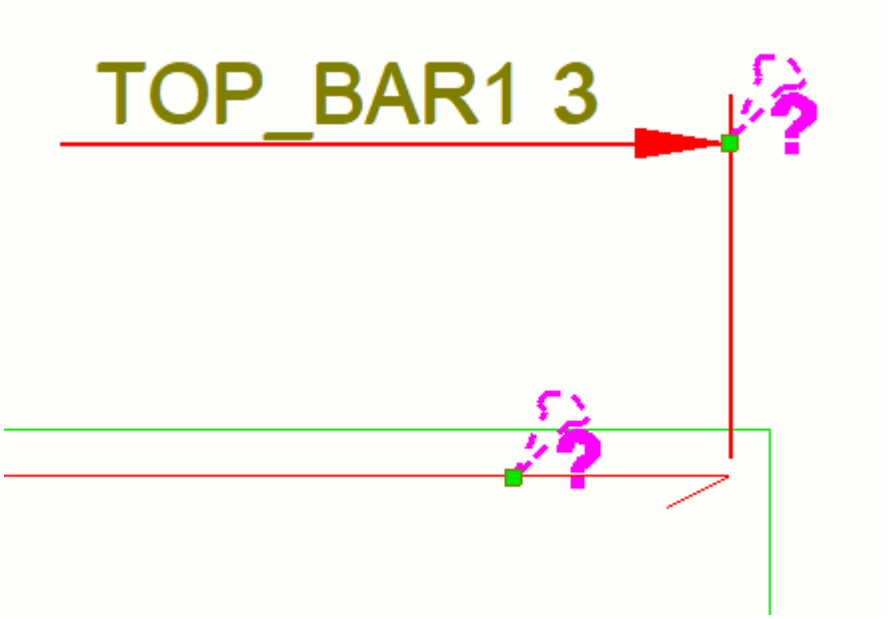


Move reinforcement ticks or reinforcement tags

You can change the position of previously created reinforcement ticks and reinforcement tags by dragging their handles.

1. Select the reinforcement tick or reinforcement tag.

   Two green handles appear.



2. Do any of the following:

To	Do this
Move the reinforcement tick or reinforcement tag	a. Select the handle that is closer to the reinforcement tick or reinforcement tag. b. Drag the handle to a new location, and release the mouse.
Mark another reinforcing bar in a reinforcing bar group or reinforcing bar set	a. Select the handle that is further from the reinforcement tick or reinforcement tag. b. Drag the handle to a new location, and release the mouse. Note that you cannot mark another reinforcing bar if you have added the reinforcement tick or the reinforcement tag for an individual reinforcing bar.

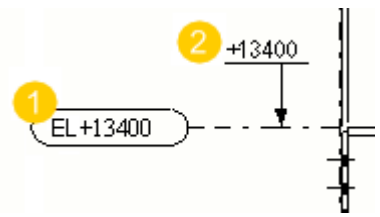
Add level marks in drawings

A *level mark* represents the elevation of a point. In addition to the automatic elevation dimensions that you can define in the drawing properties before creating the drawing, and the elevation information in the grid labels added in the model, you can also add manual level marks in your drawing to ensure that the dimensions are correct.

NOTE The shortening value added in the user-defined properties of a part also affects level marks.

1. Open a drawing.
2. On the **Annotations** tab, click **Level mark**. If you need to modify the properties first, hold down **Shift** and click the command button.

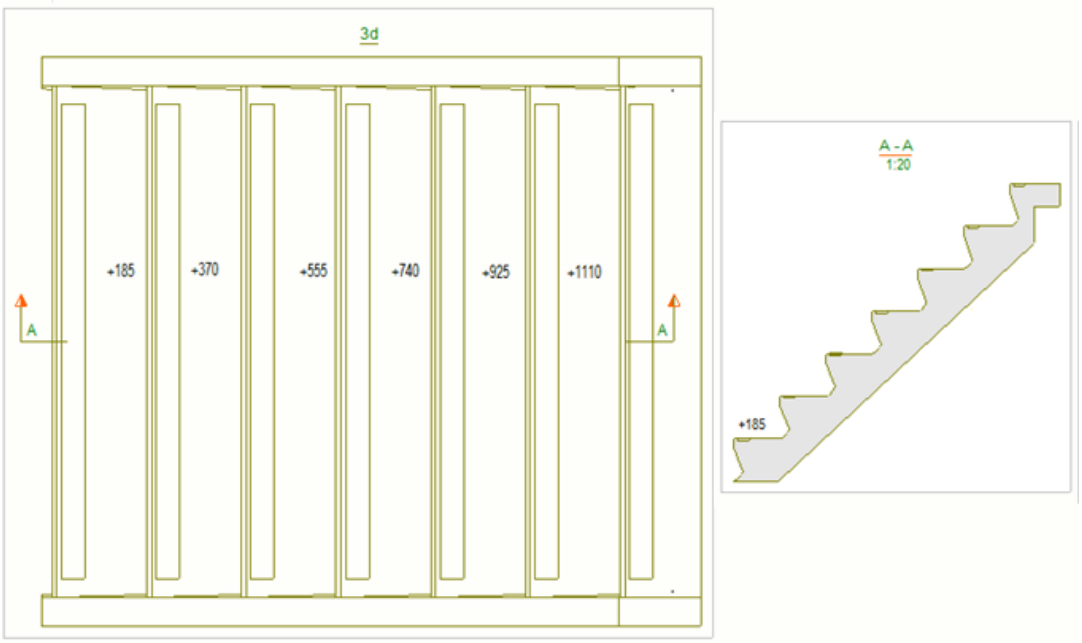
   For more information about the level mark properties, see Level mark properties (page 1001).
3. Pick an associativity point for the leader line and a position for the mark. The point you can pick depends on the active snap switches.



(1) Elevation information in the grid label

(2) Elevation mark created with **Add level mark** in the drawing

You can also add level marks in GA drawing plan views. In this case, you might want to leave the leader line out.



See also

Level mark properties in drawings (page 1001)

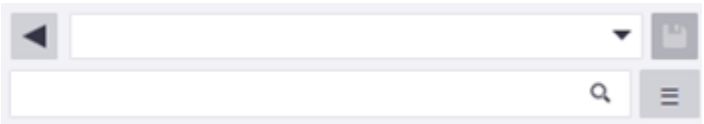
Add and modify section marks in drawings

You can add section marks in an open drawing, and modify the existing section marks.

You can add section marks only, or section marks together with section views. One example of a situation where you might want to add a section mark without a section view is when you have a similar section that already has a section view. For more information about creating section views, see Create drawing views (page 180).

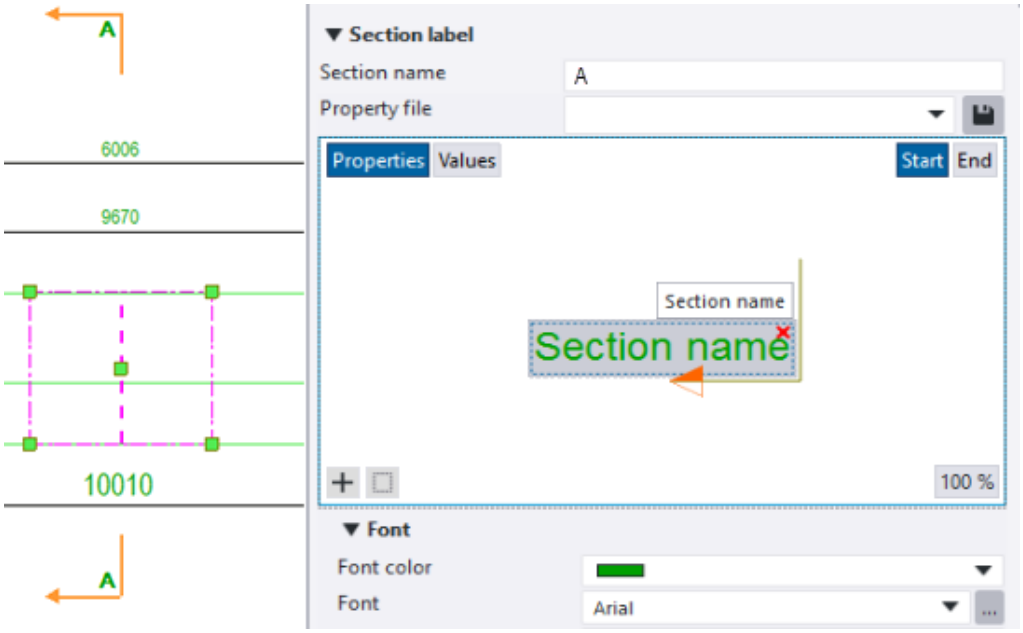
Add section marks

1. In an open drawing, on the **Annotations** tab, hold down **Shift** and click **Section mark**. Alternatively, you can double-click the command, or select **Section mark** from the drawing property pane object list.
2. You can load predefined section mark properties and search for properties:



3. Enter a name for the section.

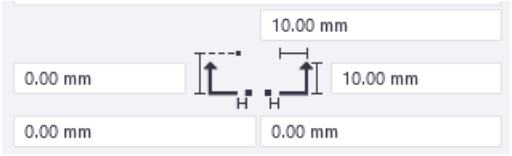
4. To load a set of predefined section mark contents, click the mark editor area and, in **Property file**, select the property file.
5. In the mark editor:



- Click the **Properties** or **Values** button to show either the property name or the property value in the preview. These buttons are only available when you are modifying a mark.
- To indicate which end of the cutting line you want to work with, click **Start** or **End**.
- To add more containers in the mark, click the container button . You may add up to five containers.
- To add elements in an empty mark container, click the container button in the preview and select the desired element.
- To add more elements in the current container, click the plus button to open the element list, and then select the desired element.
- You can drag elements inside and between containers. Containers cannot be dragged.
- To delete an element in a container, click the red delete button in the upper-right corner of the element.
- To delete a container, click the red delete button in the upper-right corner of the container.
- To add a frame, click a container or an element, click the plus button and select **Frame** from the element list.

6. Modify section mark properties:

Setting	Description
Font color, **Font**, **Font height**	Define the label font color, type, and height. Before changing the font, select the object that you want to modify: • To change the font in an element, click the element. • To change the font in a container, click the container. • To change the font in all containers, click the background. Clicking **...** gives more font choices.
Frame, **Frame color**	Add a frame around the selected elements and define the frame type and color. You can also add a frame from the element list.
Units, **Format**	Change the unit and format. This property is only available for elements for which you can define a unit and format.
Show on	Define if the section marks are shown at both ends of the cutting line or only at one end.
Text position	Define the position of the container in relation to the line or in relation to the symbol or symbol center line.
Text rotation	Define the rotation of the container.
X offset for text	Set the horizontal offset of the section mark container from the cutting line.
Y offset for text	Set the vertical offset of the mark container from the cutting line.
Symbol **Symbol color** **Symbol size** **X offset for symbol** **Y offset for symbol**	Define the cutting line symbol properties separately for the start and end symbols. If you want to use custom symbols, define the advanced options XS_SECTION_SYMBOL_RIGHT_ARROW_

Setting	Description
	SYMBOL and XS_SECTION_SYMBOL_ LEFT_ARROW_SYMBOL. **X offset for symbol** defines the horizontal offset of the symbol from the cutting line. **Y offset for symbol** defines the vertical offset of the symbol from the cutting line.
Line **Distances**	Define section line color and dimensions. 

7. Pick the first and the second point on the cut plane.
8. Create as many section marks as you need, and press **Esc** to stop.

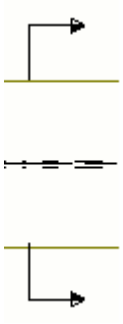
Move or copy section marks

- Copy section marks using **Ctrl**+drag.
- Copy section marks inside a drawing view with the context menu commands **Copy** --> **Linear** and **Copy** --> **Linear...**, the ribbon command **Copy**, or by entering **Copy - Linear** in **Quick Launch**. You can also use the keyboard shortcut **Ctrl+C**.
- Move section marks using **Shift**+drag.
- Move section marks inside a drawing view with the context menu command **Move** --> **Linear**, the ribbon command **Move**, or by entering **Move - Linear** in **Quick Launch**. You can also use the keyboard shortcut **Ctrl+M**.

Section marks can only be copied or moved within a view. When moved beyond the view frame, the view frame will adjust so that the section mark will always be within the frame.

Modify section mark properties

1. To modify the section mark properties, click the section mark. If the property pane is not open, double-click the section mark. If you want to modify several section marks, hold down **Ctrl** and click the marks.



Click the **Properties** or **Values** button to show either the property name or the property value in the mark editor view. These buttons are only available when you edit an existing mark.

If you have a section view created for the section mark, and you click the section mark in the drawing, the property pane title changes to **Section mark with view** and you can also modify the section view label properties in the same property pane.

2. Modify the properties.

   If you have selected several section marks, the properties that have different values have the text *Varies*. You can select another value from the list.

3. Click **Modify** to apply the changes in all selected section marks.

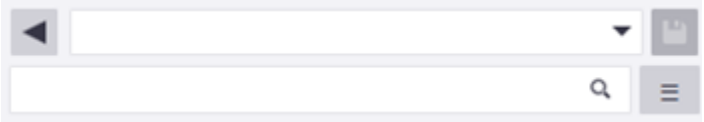
Add and modify detail marks in drawings

You can mark details in the drawing with detail marks, and modify the existing detail marks.

You can add detail marks only, or detail marks together with detail views. One example of a situation where you might want to add a detail mark without a detail view is when you have a similar detail that already has a detail view. For more information about creating detail views, see Create drawing views (page 180).

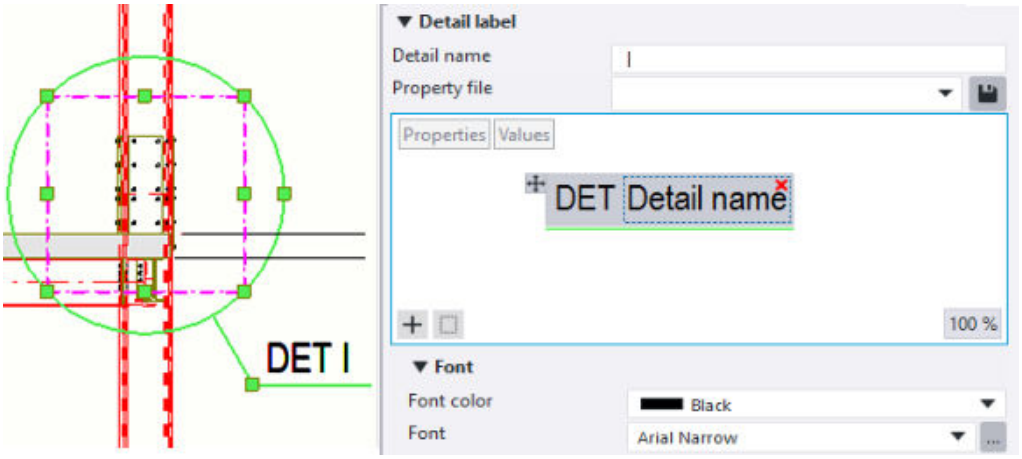
Add detail marks

1. In an open drawing, on the **Annotations** tab, hold down **Shift** and click **Detail mark**. Alternatively, you can double-click the command, or select **Section mark** in the drawing property pane object list.
2. You can load predefined detail mark properties and search for properties:

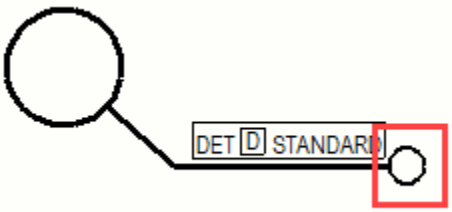
3. Enter a name for the detail.

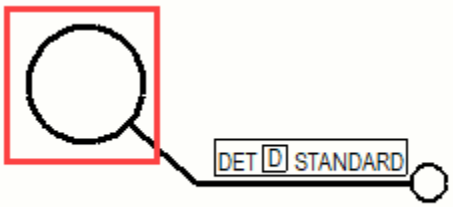
4. To load a set of predefined detail mark contents, click the mark editor area and, in **Property file**, select the property file.
5. In the mark editor:



   - Click the **Properties** or **Values** button to show either the property name or the property value in the preview. These buttons are only available when you are modifying a mark.
   - To add more containers in the mark, click the container button , and select an element. You may add up to five separate containers.
   - To add elements in an empty mark container, click the container button in the preview and select the desired element.
   - To add elements in the current mark container, click the container, click the element button to open the element list, and then select the desired element.
   - You can drag elements inside and between containers. You can also drag containers to another location by the handle .
   - To delete an element in a container, click the red delete button in the upper-right corner of the element.
   - To delete a container, click the red delete button in the upper-right corner of the container.
   - To add a frame, click a container or an element, click and select **Frame** in the element list.
6. Modify detail mark properties:

Setting	Description
Font color, **Font**, **Font height**	Define the label font color, type, and height. Before changing the font, select the object that you want to modify:

Setting	Description
	• To change the font in an element, click the element. • To change the font in a container, click the container. • To change the font in all containers, click the background. Clicking **...** gives more font choices.
Frame, **Frame color**	Add a frame around the selected elements and define the frame type and color. You can also add a frame through the element list.
Units, **Format**	Change the unit and format. This property is only available for elements for which you can define a unit and format.
Text position	Define the position of the detail mark container in relation to the line, in relation to the symbol, or in relation to the symbol center line.
X offset for text	Set the horizontal offset of the detail mark container from the label start point.
Y offset for text	Set the vertical offset of the detail mark container from the bottom line of the label.
Symbol **Symbol color** **Symbol size**	Define the detail symbol, symbol color, and symbol size. Detail symbol appears at the end of the detail label line. 
Detail boundary **Line color** **Line type**	Define the detail boundary shape, and the detail line color and type.

Setting	Description
	

7. Create the detail mark. Depending on the selected detail boundary shape, do one of the following:
   - If the shape is **Circle**, pick the center point of the circle and then pick a point on the circle.
   - If the shape is **Rectangle**, pick corner points for the rectangle.
8. Pick a position for the label.
9. Continue adding detail marks, or press **Esc** to interrupt.

Move or copy detail marks

- Copy detail marks using **Ctrl**+drag.
- Copy detail marks inside a drawing view with the context menu commands **Copy** --> **Linear** and **Copy** --> **Linear...**, the ribbon command **Copy**, or by entering **Copy - Linear** in **Quick Launch**. You can also use the keyboard shortcut **Ctrl+C**.
- Move detail marks using **Shift**+drag.
- Move detail marks inside a drawing view with the context menu command **Move** --> **Linear**, the ribbon command **Move**, or by entering **Move - Linear** in **Quick Launch**. You can also use the keyboard shortcut **Ctrl+M**.

Note that detail marks can only be moved or copied within a view. When moved beyond the view frame, the view frame will adjust so that the detail mark will always be within the frame.

Modify detail mark properties

1. To modify the detail mark properties, click the detail mark. If the property pane is not open, double-click the detail mark. If you want to modify several detail marks, hold down **Ctrl** and click the marks.

   Click the **Properties** or **Values** button to show either the property name or the property value in the mark editor view. These buttons are only available when you edit an existing mark.

   If you have a detail view created for the detail mark, and you click the detail mark in the drawing, the property pane title changes to **Detail mark with view**. You can also modify the detail view label properties in the same property pane.

2. Modify the properties.

   If you have selected several detail marks, the properties that have different values have the text *Varies*. You can select another value from the list.

3. Click **Modify** to apply the changes in all selected detail marks.

See also

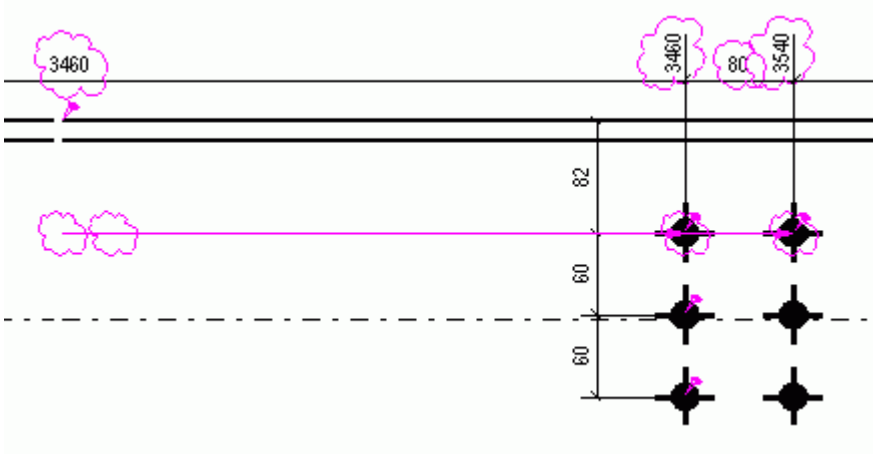
Create views in drawings (page 180)

Check changed marks, notes and dimensions and remove change symbols

Tekla Structures highlights the marks and dimension marks that have changed due to changes in the model, and dimension points that have been moved. Tekla Structures also highlights the changed angle dimensions, level marks, and associative notes.

Tekla Structures highlights the changes in the following way:

- A change symbol (by default, a cloud) is drawn around the old point, the new point and the dimension values, or around the changed mark or note.
- An arrow is drawn from the old dimension point to the new one.



Remove change symbols

After you have checked all the change symbols that Tekla Structures has created in your drawing, you can remove all of them or just the selected ones.

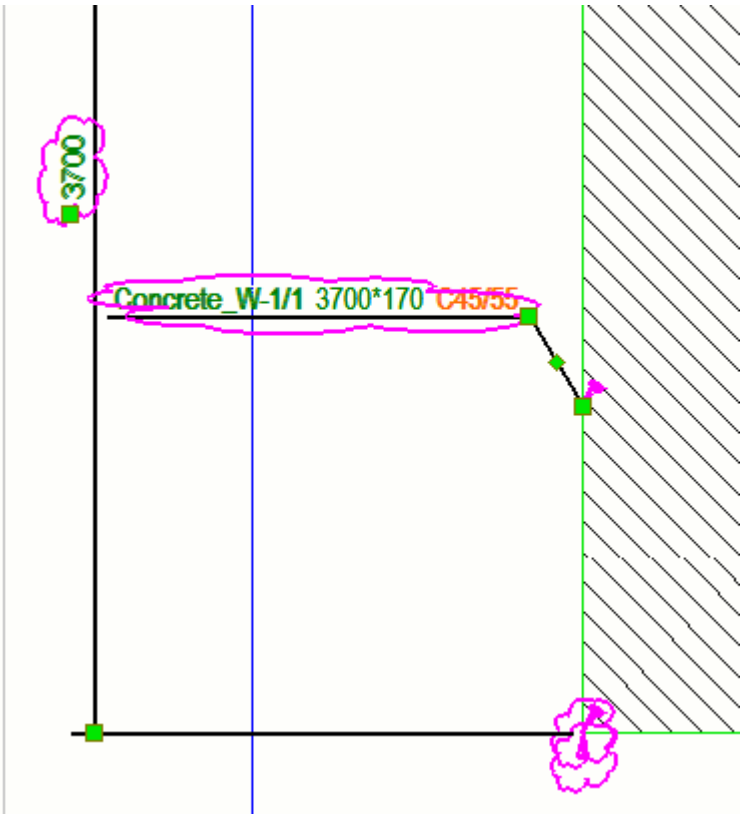
To remove the change symbols, do one of the following in the open drawing:

To	Do this
Remove all change symbols at one go	• On the **Drawing** tab, click **Remove** --> **All change symbols**.

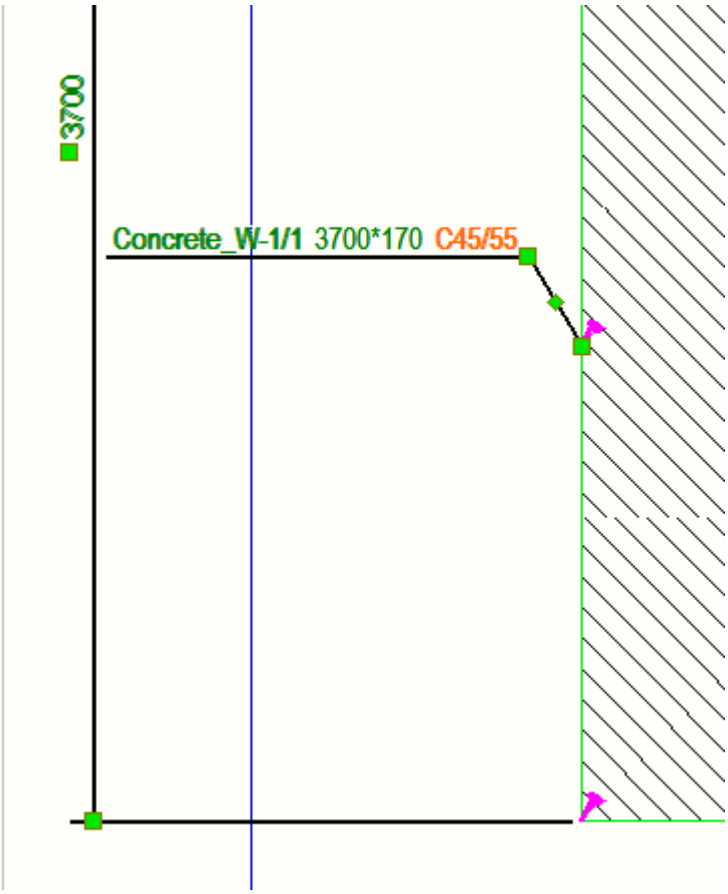
To	Do this
Remove all dimension change symbols	• On the **Drawing** tab, click **Remove** --> **All dimension change symbols** .
Remove the selected dimension change symbols	1. Select the dimension change symbols you want to remove. 2. On the **Drawing** tab, click **Remove** --> **Selected dimension change symbol** .
Remove all mark change symbols	• On the **Drawing** tab, click **Remove** --> **All mark change symbols** .
Remove the selected mark change symbols	1. Select the mark change symbols you want to remove. 2. On the **Drawing** tab, click **Remove** --> **Selected mark change symbol** .
Remove all associative note change symbols	• On the **Drawing** tab, click **Remove** --> **All associative note change symbols** .
Remove the selected associative note change symbols	1. Select the associative note change symbols you want to remove. 2. On the **Drawing** tab, click **Remove** --> **Selected associative note change symbol** .

Examples

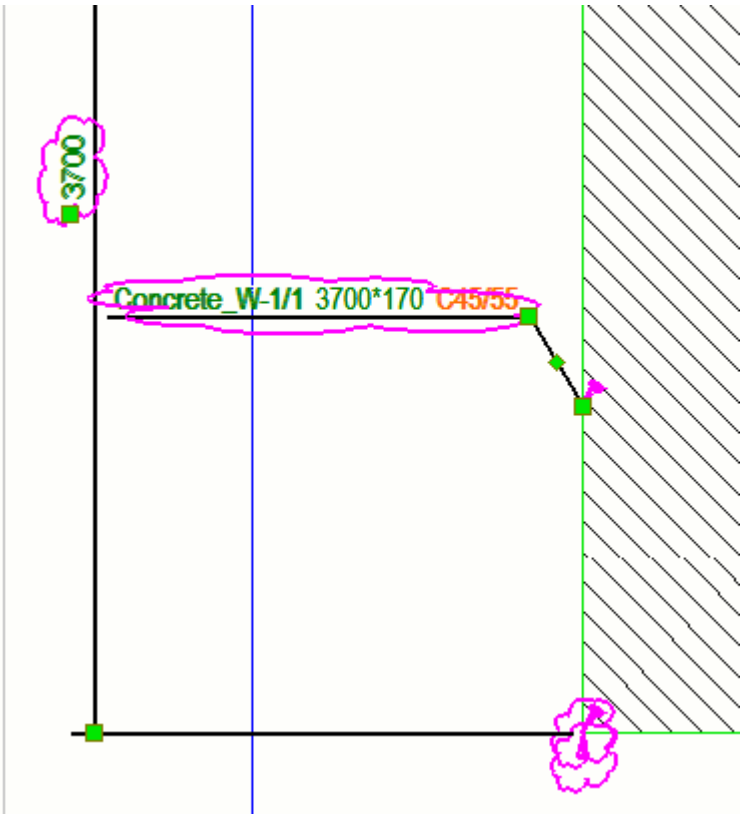
The following image shows an example of a mark change symbol after a material change, and of a dimension change symbol after a change in the size of the part.



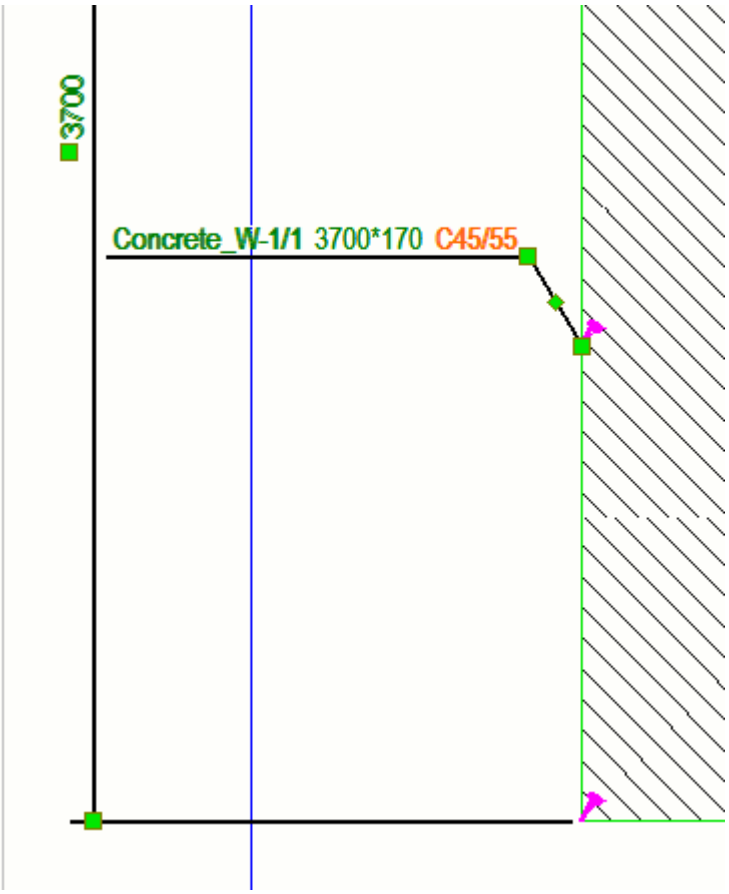
The following image shows the dimension text and the mark after you have selected **Remove** --> **All change symbols**.



The following image shows an example of a mark change symbol after a material change, and of a dimension change symbol after a change in the size of the part.



The follwoing image shows the dimension text and the mark after you have selected **Remove** --> **All change symbols**.



Related advanced options

There are some advanced options related to change symbols that you may want to use:

- XS_HIGHLIGHT_ASSOCIATIVE_DIMENSION_CHANGES
- XS_HIGHLIGHT_MARK_CONTENT_CHANGES
- XS_ASSOCIATIVE_CHANGE_HIGHLIGHT_SYMBOL
- XS_ASSOCIATIVE_CHANGE_HIGHLIGHT_SIZE

Merge marks

You can merge marks to reduce the number of marks in the drawing, and to make the drawing cleaner. You can merge marks if their content matches. You can merge marks automatically before creating a drawing, through drawing properties in an open drawing, and manually in an open drawing.

For more information about automatic merging, see Merge marks automatically (page 828).

For more information about mark properties, including merging settings, see Mark properties (page 972).

Merged part marks

A merged part mark means that you have only one part mark for similar parts in a drawing, instead of a separate mark for each of the parts. Merged part marks indicate the number of included parts, and contain the defined part mark contents, and the near side and far side information. The marks are merged only in X direction of the main part.

Tekla Structures merges marks for visible parts in drawings if:

- The secondary parts are welded or bolted to the same main part.
- The parts are on the same line.
- Distances between the parts are equal.
- The parts have the same part position.
- Distance between parts is not more than what is set for the advanced option XS_PART_MERGE_MAX_DISTANCE.
- There are at least as many parts in the array as it has been set for the advanced option XS_MIN_MERGE_PART_COUNT.

Limitations

- You cannot merge part marks (assembly marks) that are not part of the same assembly.
- Tekla Structures does not merge neighbor part marks.

Advanced options in merging marks

In merging part marks, you may find the following advanced options useful:

XS_MULTIPLIER_SEPARATOR_FOR_MERGED_PART_MARK

XS_NSFS_POSTIX_FOR_MERGED_PART_MARK

XS_NS_POSTIX_FOR_MERGED_PART_MARK

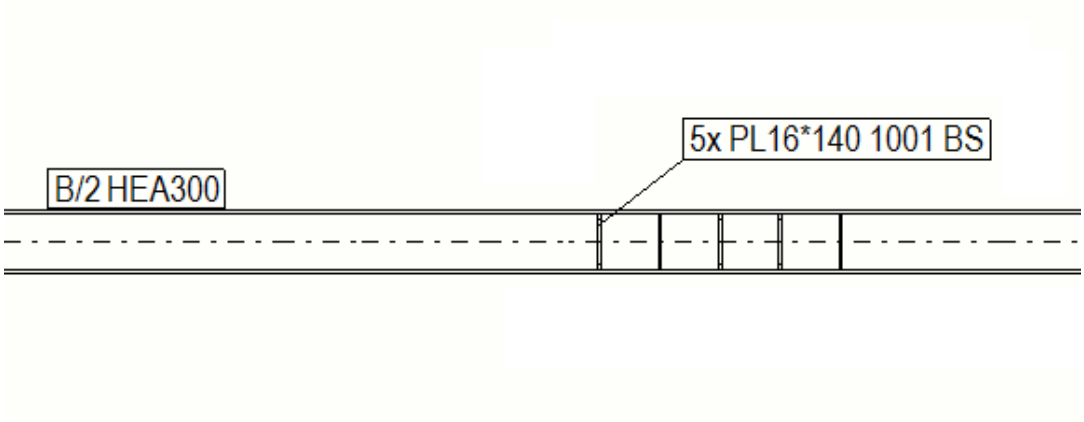
XS_FS_POSTIX_FOR_MERGED_PART_MARK

XS_PART_MERGE_MAX_DISTANCE

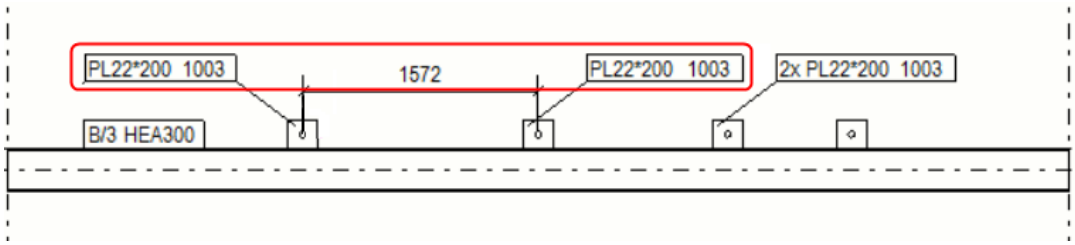
XS_MIN_MERGE_PART_COUNT

Example

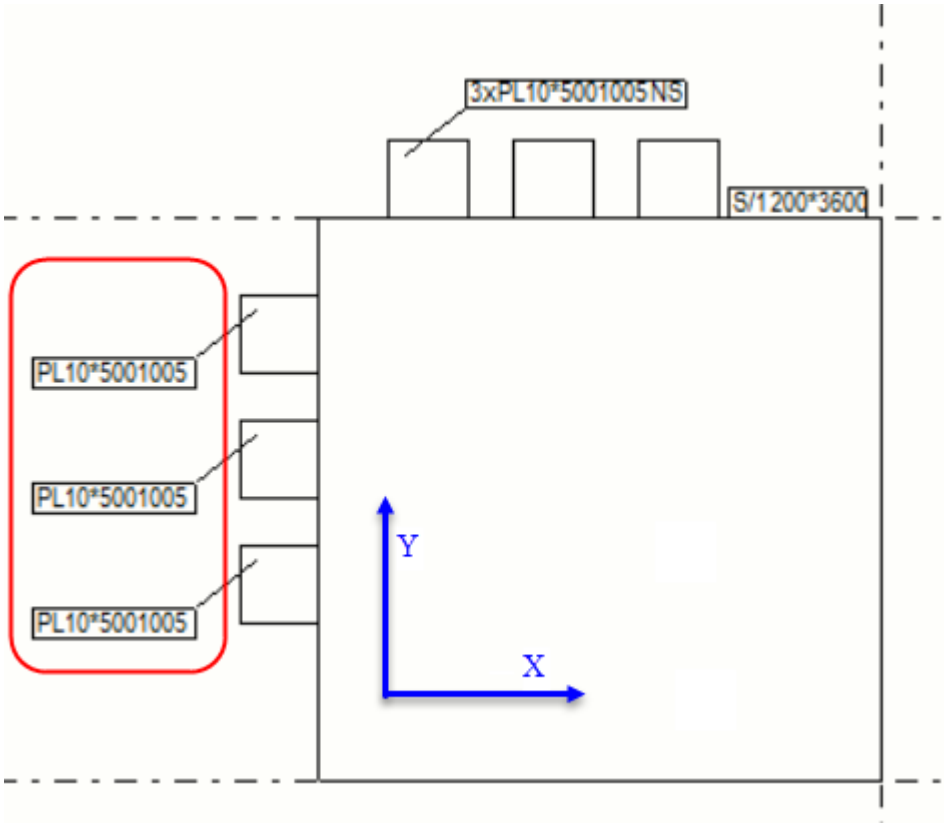
In the example below, the part marks are merged in X direction of the HEA300 beam (main part).



In the example below, the leftmost part marks are not merged, because they are too far from each other.



In the example below, the marks in the Y direction are not merged, because the marks are merged only in X direction (which in this example is horizontal).



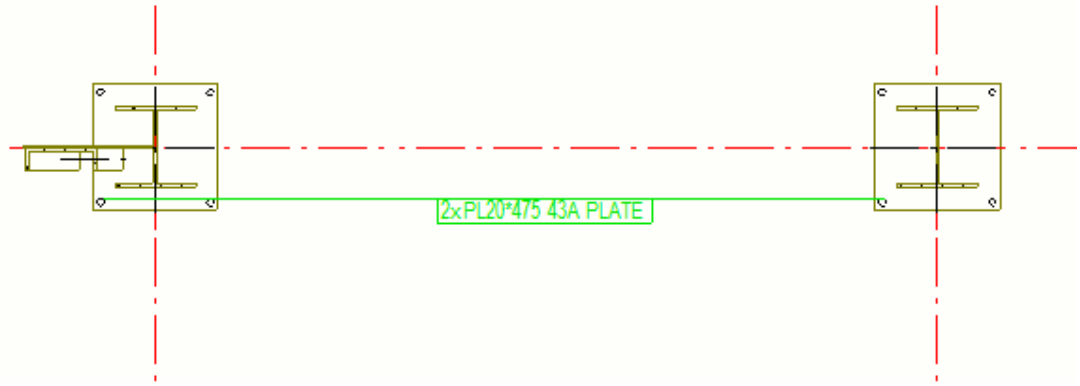
Merge part marks or bolt marks manually

NOTE
- You can only merge marks that have the same content.
- You can only merge bolt marks that have not been merged already.

1. Open a drawing in which you want to merge part or bolt marks, and ensure that the marks are visible.
2. Select the marks that you want to merge.
3. Right-click and select **Merge**.
4. To change the properties of the merged mark, click the merged mark and modify the settings. If the property pane is not open, double-click the mark.
5. Select one of the options in the **Merged marks** list:
   - **One leader line to group**: Creates one leader line for a group of parts.
   - **One leader line per row**: Merges the marks and creates one leader line for a row of parts.
   - **Parallel leader lines**: Merges the marks and creates parallel leader lines.
   - **Leader lines to one point**: Merges the marks and draws all leader lines to one point.
6. Select the leader line type and the arrow to be used.
7. When you are done, click **Modify**
8. If needed, you can split the merged marks by selecting the marks to split, right-clicking and selecting **Split**.

If merging part marks fails for some reason, an error message is displayed in the status bar: "Mark properties do not match, could not merge all marks".

In the following example, **Leader lines to one point** has been selected, and a leader line is going from the mark frame to each part it is related to:



Merged reinforcement marks

Tekla Structures can automatically merge similar reinforcement marks, and you can also merge reinforcement marks manually. Merged reinforcement marks may include several blocks, and additional information. Blocks combine similar single marks.

To merge reinforcement marks in drawings, the reinforcement must be attached to a concrete part or cast unit in the model.

Tekla Structures merges marks for visible reinforcing bars in drawings if:

- The bars belong to the same concrete part or cast unit.
- The direction of the bars is the same.
- The bar marks are identical.
- The bars are close to each other.
- A straight line can be drawn through all the bars.

Merge reinforcement marks manually

1. Open a drawing in which you want to merge reinforcement marks, and ensure that the marks are visible.
2. Select the reinforcement marks to merge in the drawing.
3. Right-click and select **Merge**.
4. To change the properties of the merged reinforcement mark, such as the leader line settings, click the merged mark, and modify the settings. If the property pane is not open, double-click the mark.
5. When you are done, click **Modify**.
6. If needed, you can split the merged marks by selecting the marks to split, right-clicking and selecting **Split**.

NOTE
- If merging of rebar marks fails due to settings that are not compatible with merging, or to some other errors in settings, a new error message is displayed: "Could not merge all marks, please check the applied merged rebar mark settings".
- When you merge several reinforcement marks, and the mark text is very long, an error message is displayed, and the marks will not be merged.

Merge marks by adjusting drawing properties

In a final drawing, you can merge marks also by adjusting the drawing properties.

To activate merging through drawing properties:

1. Open a drawing.

2. Double-click the drawing background to open drawing properties.
3. Adjust mark visibility settings according to the examples below:

Drawing type	Adjust mark visibility settings
Cast unit drawings	1. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**. The merging needs to be set view by view separately. 2. Click a mark type in the options tree. For example, click **Part mark**. 3. Go to the **General** tab, and set **Merge marks** to **On**. 4. In **Merged marks**, select one of the following: • **One leader line to group**: creates one leader line for a group of parts. • **One leader line per row**: Merges the marks and creates one leader line for a row of parts. • **Parallel leader lines**: Merges the marks and creates parallel leader lines. • **Leader lines to one point**: Merges the marks and draws all leader lines to one point. 5. Select the leader line type and the arrow to be used. 6. To merge reinforcement marks, click **Reinforcement mark** in the options tree, and go to the **Merging** tab. • In **Identical marks in same cast unit**, select one of the following: • **One leader line to group** creates one leader line for a group of reinforcing bars. • **One leader line per row**: Merges the marks and creates one leader line for a row of reinforcing bars. • **Parallel leader lines**: Merges the marks and creates parallel leader lines.

Drawing type	Adjust mark visibility settings
	• **Leader lines to one point**: Merges the marks and draws all leader lines to one point. • **No merge**: Marks are not merged, an individual leader line is created for each mark. • If you select **No merge**, you still need to define the mark content for the marks that Tekla Structures automatically merges on the **Merging** tab. • In **Preferred direction of merge**, if there are several possible merge directions, select **Merge horizontal** or **Merge vertical**. • In **Available elements**/**Elements in mark**, Select the contents to be included in the merged reinforcement marks (page 981). To ensure that merged reinforcement marks appear in the drawing, always include **Symbol separating blocks in mark** as the last element in the reinforcement mark. To omit the separating symbol, leave this box blank, but still include the element in the mark. 7. Select the leader line type and the arrow to be used. 8. Click **Save** to save the changes in view properties, and then click **Close** to return to drawing properties. 9. Click **Modify**.
General arrangement drawings	1. Click a mark type button in drawing properties. For example, click **Part mark...**. 2. Go to the **General** tab, and set **Merge marks** to **On**. 3. In **Merged marks**, select one of the following: • **One leader line to group** creates one leader line for a group of parts. • **One leader line per row**: Merges the marks and creates one leader line for a row of parts.

Drawing type	Adjust mark visibility settings
	• **Parallel leader lines**: Merges the marks and creates parallel leader lines. • **Leader lines to one point**: Merges the marks and draws all leader lines to one point. 4. Select the leader line type and the arrow to be used. 5. To merge reinforcement marks, click **Reinforcement marks...** in drawing properties and go to the **Merging** tab. • In **Identical marks in same cast unit**, select one of the following: • **One leader line to group** creates one leader line for a group of reinforcing bars. • **One leader line per row**: Merges the marks and creates one leader line for a row of reinforcing bars. • **Parallel leader lines**: Merges the marks and creates parallel leader lines. • **Leader lines to one point**: Merges the marks and draws all leader lines to one point. • **No merge**: Marks are not merged, an individual leader line is created for each mark. • If you select **No merge**, you still need to define the mark content for the marks that Tekla Structures automatically merges on the **Merging** tab. • In **Preferred direction of merge**, if there are several possible merge directions, select **Merge horizontal** or **Merge vertical**. • In **Available elements**/**Elements in mark**, select the contents to be included in the merged reinforcement marks (page 981). To ensure that merged reinforcement marks appear in the drawing, always include **Symbol separating blocks in mark** as the last element in the reinforcement mark. To omit the

Drawing type	Adjust mark visibility settings
	separating symbol, leave this box blank, but still include the element in the mark. 6. Select the leader line type and the arrow to be used. 7. Click **OK**. 8. Click **Modify**.

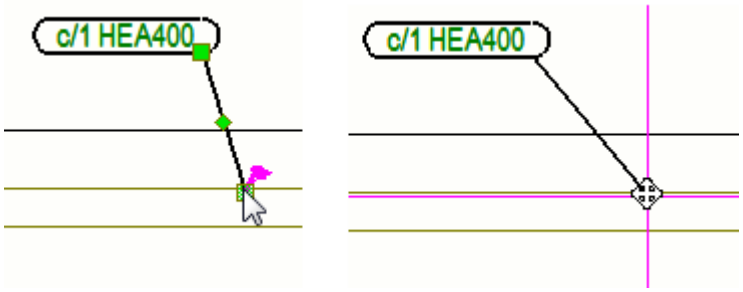
Drag leader line associativity point

You can move the associativity point of a leader line by dragging it in marks and notes, for example.

Ensure that you have selected **Drawing drag & drop** in **File** --> **Settings** .

- Hold down the left mouse button and drag the associativity point of a leader line to a new location.

If the associativity point is originally on a line, you can drag it along that line. If the associativity point is originally inside a part, you can drag the point inside that part.



Add and modify text in drawings

You can add several lines of text in a text object, and use word wrapping if you like. For the text, you can select the desired color, height, font, angle, frame, and leader line arrow type, and align the text as required. You can also modify the text and its properties after adding the text.

Add text in an open drawing

1. Open a drawing.
2. To create a single line or multiple lines of text, hold down **Shift**, and on the **Annotations** tab, click **Text**, and select one of the following commands:

   Text: Add text without a leader line at the position you pick.

   With leader line: Add text with a leader line at the position you pick.

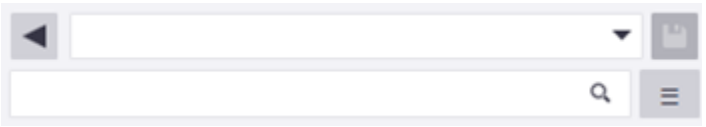
Along line: Add text along a line at the position you pick.

Along line, arrow at end: Add text along a line at the position you pick. An arrow is inserted at the second position you pick.

Along line, arrow at start: Add text along a line at the position you pick. An arrow is inserted at the first position you pick.

You can also select **Text** in the property pane object list and modify the leader line setting in text properties.

3. You can load predefined text properties and search for properties:



4. Type the text in the **Text** box. You can add a line break by pressing **Enter**.
5. Modify the font color, type, height, frame type and frame color as required.
6. To wrap the words, set **Word wrapping** to **Yes**. You can then define the line length using the **Ruler width** option. When the word wrapping is activated, the text object gets a handle, which you can drag to make the text wider or narrower.
7. Add a frame around the text and select the frame color. To add a frame, select one of the following frame options:

Rectangular

Line

Round

Circle

Diamond

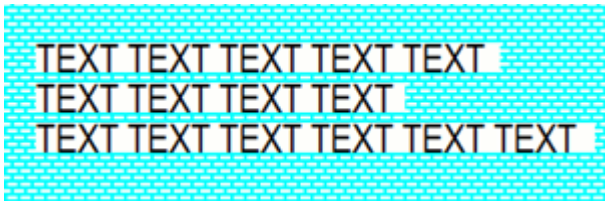
Hexagon

Triangle

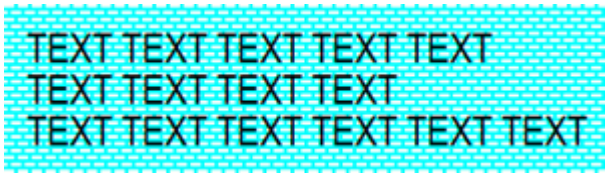
Sharpened

8. In **Background**, select one of the following:

Opaque hides the area in the drawing that is covered by the text:



Transparent shows the area in the drawing that is covered by the text so that the linework is shown:



9. Select the leader line type and the leader line arrow type, height, and length. The leader line options are:

   No leader line

   Leader line

   Along line or leader line

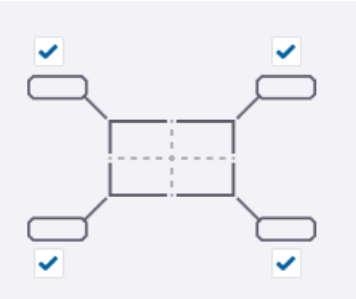
   Along line

   Inside part along part

10. Adjust the placing settings:
    - Select one of the **Alignment** options: **Left**, **Center**, **Right** or **Leader line**.
    - In **Rotation**, change the rotation of the text.
    - To exactly place the text in the position you pick, and keep it there, set the **Placing method** button to **Fixed**.

      Free allows Tekla Structures to search for the first suitable location for the text.

- In **Position**, define the areas where Tekla Structures searches for a position to place the text. This option can only be adjusted when **Placing method** is set to **Free**.



-

  Distance s (search margin) is the empty margin that you want to leave around the text.

  Distance d min (minimum distance) is the minimum distance of the text from the part.

11. Pick the point where you want to place the text. Depending on the command, you have to pick one to three times.

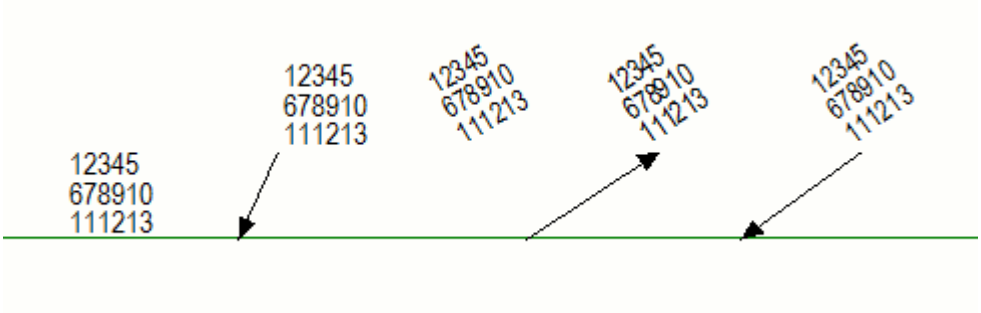
    To add the same text again, continue picking. To stop adding texts, press **Esc**.

    By default, the text is aligned to the left. The line spacing is automatically adjusted by the font size that you select. You can also affect the line space by using the advanced option XS_MARK_LINE_SPACE_FACTOR.

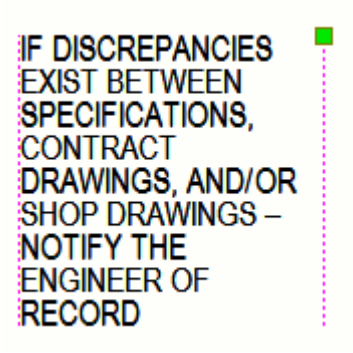
    You can drag the associativity point of the text leader line freely after adding the text.

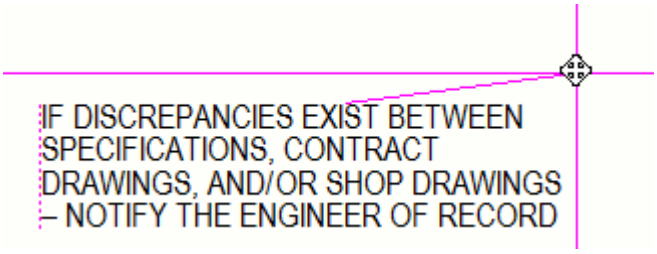
Examples

See below for examples of the different text options. From left: **Text**; **With leader line**; **Along line**; **Along line, arrow at end**; and **Along line, arrow at start**.

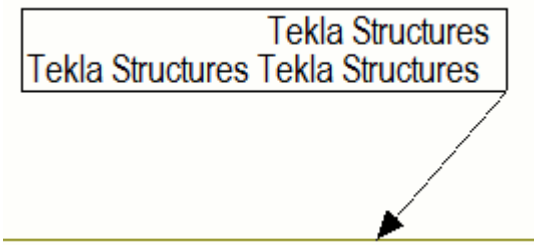


The text object gets a handle when word wrapping is activated, and you can drag the handle to change the text object size:





In the following example, the text is aligned to the leader line start:



TIP In addition to using the text tool, you can add texts as links from .txt or .rtf files. This allows you to use more sophisticated formatting of the text, and add tables, for example. For more information, see Add links to rich text files (page 380).

Modify text properties

1. To modify the text properties, click the text object in the drawing. If the property pane is not open, double-click the text. If you want to modify several texts, hold down **Ctrl** and click the text objects.
2. Modify the properties.

   If you have selected several texts, the properties that have different values have the text *Varies*. You can select another value from the list.
3. Click **Modify** to apply the changes in all selected texts.

Delete unnecessary texts

- To delete a text object, select the object and press **Delete** on the keyboard, or right-click and select **Delete**.

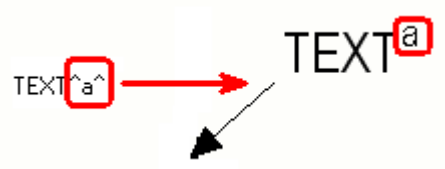
Add text in superscript

You can use superscript in texts included in all your text objects, part marks, dimension marks, and associative notes.

1. On the **File** menu, click **Settings** --> **Advanced options** and go to the **Dimensioning - general** category.
2. Ensure that the advanced option XS_SUPERSCRIPT_USED_IN_DRAWING_TEXTS is set to `TRUE`.
3. Open the properties of a text object, mark or associative note by holding down **Shift** and clicking a text, mark or note command on the ribbon, or by selecting the command in the property pane object list.

   For example, on the **Annotations** tab, click **Text** --> **With leader line** .
4. Do one of the following:
   - For text objects, type the desired text in the **Text** box.
   - For marks and associative notes, add a **Text** element in the editor and type the desired text in the element box.
5. Enter circumflexes (^) around the characters that you want to have in superscript.
6. Add the text, mark or note in the drawing.

The following example shows how the superscript is typed in the text and what it looks like in the existing drawing.



NOTE You can affect the height of the superscript text with the XS_SUPERSCRIPT_HEIGHT_FACTOR advanced option.

See also

Add and modify text in drawings (page 364)

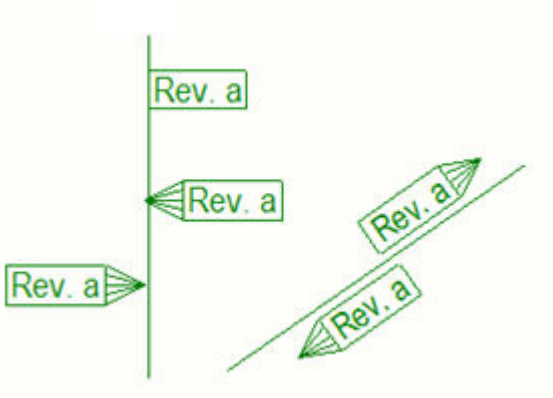
Add marks, notes, and texts in drawings (page 289)

Add dimensions in drawings (page 206)

Add revision marks in drawings

Revision marks are used for indicating a change in the Tekla Structures model or drawing, and the objects that have changed. Tekla Structures creates the revision mark using the properties in the **Revision Mark Properties** dialog box, and the information entered in the **Revision Handling** dialog box.

Before you add a revision mark, create a drawing revision (page 580) in **Document manager** for a particular drawing so that the revision information is available for the revision mark.



1. Open a drawing that has a revision created in **Document manager**.
2. On the **Annotations** tab, click **Revision mark** and select one of the following commands:
   - **Add revision mark**
   - **Pointing to left**
   - **Pointing to right**
   - **Along line, pointing to left**
   - **Along line, pointing to right**

   The **Revision Mark Properties** dialog box is displayed.

3. In the revision **Mark** list on the right, select another revision number first and then switch to the revision you are documenting. For example, if you are documenting revision 1 of the drawing, first switch to revision 2 in the list, and then back to 1. The revision data is updated in the dialog box when you do this.
4. To place the revision mark exactly in the position you pick and keep it there, click the **Place...** button and select **fixed** in the **Placing** list.
5. Go to the **Appearance** tab and set the text color, height, font and angle, the frame color, leader line and type, and the leader line arrow type and size.
6. Click **OK** or **Apply**.
7. Pick a point or points to place the mark.

   Tekla Structures creates the revision marks. You can also see the revisions in **Document manager**.

   To delete any unnecessary revision marks, select the marks and press **Delete** on the keyboard.

4.10 Add and modify symbols in drawings

You can add symbols in drawings as such and in marks, notes, object representations, and line arrows, for example. Symbols can be created and edited in Symbol Editor (SymEd). The file extension for symbols is `.sym`.

Add symbols in drawings

You can add symbols in an open drawing as such.

The symbols can be represented in three different ways: without leader line, with leader line, and along line. Tekla Structures adds symbols using the properties defined in symbol properties.

1. In the drawing, hold down **Shift** and, on the **Annotations** tab, click **Symbol** and one of the following symbol commands:
   - **Symbol**: Add a symbol in the current drawing without a leader line.
   - **Along line**: Create a symbol along the line you define by picking two points. Then pick an insertion point for the symbol.
   - **With leader line**: Add a symbol with a leader line pointing to the point you pick first.
2. Modify the symbol properties:
   - **File**: Change the symbol file in use. Click **Select** to show more files.

- **Number**: Change the symbol in use. Click **Select** to show more symbols.
- **Place...**: Modify the placing settings:

  Free allows Tekla Structures to search for the first suitable location for the symbol.

  Fixed allows you to place the symbol in any location.

  When you use the option **Fixed**, the symbol stays where it is even though you update the drawing, whereas with **Free**, Tekla Structures tries to find the optimal place for the symbol.

  Search margin is the empty margin that you want to leave around the symbol.

  Minimum distance is the minimum distance of the symbol from the object.

  In **Quarter**, define the areas where Tekla Structures searches for a position to place the symbol.

  For more information about placing various annotation objects, see Placement properties for marks, dimensions, notes, texts and symbols (page 1033)
- **Symbol**: Change the color, height, and angle of the symbol.
- **Frame**: Change the frame type, leader line, and color.

3. Click **OK**.
4. Pick one to three points in the drawing to place the symbol. The symbol command you selected affects the number of points to pick.

Add symbols in marks

You can add symbols in marks. You can select the symbol file to be used and the symbol to be added in the mark.

1. In an open drawing, click a mark. If the property pane is not open, double-click the mark.
2. In mark properties, in the editor view, click the location where you want to add the symbol.
3. To add the symbol, click to open the element list and select **Symbol**.
4. In the **Mark content - symbol** dialog box, select the desired symbol file in the **Symbol file** list.
5. Click the symbol you want to use. You can also enter the number of the symbol.

6. Click **OK**.

   Tekla Structures adds the symbol in the selected mark location.

7. Click **Modify** in mark properties.

   Tekla Structures adds the symbol in the mark.

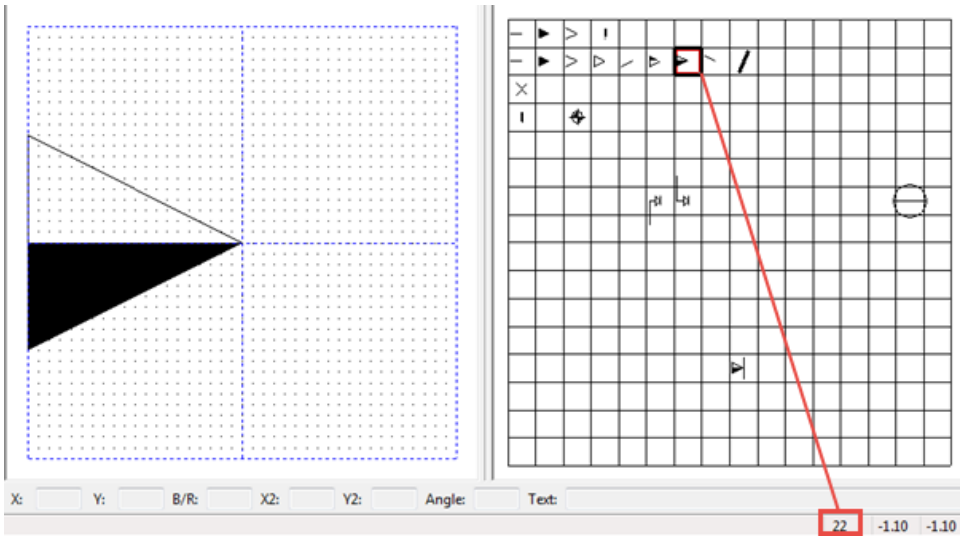
Change a symbol in a symbol file

You can manually modify a symbol in a symbol file. We recommend that you save the modified symbol file with another name in another folder, for example, in the model, firm or project folder.

1. You can open the symbol file that contains the symbol that you want to modify in Symbol Editor in several ways:
   - On the **File** menu, click **Editors** --> **Symbol editor** . Open a symbol file using **File** --> **Open** .
   - Double-click a symbol that you have added in an open drawing. In the **Symbol Properties** dialog box, then click **Select...** next to the **File** box, select a file from the **Symbol files** browser and click **Edit...**.
2. Modify the symbol in Symbol Editor:
   a. Click the symbol and sketch your new symbol with the Symbol Editor drawing tools.

      You can also import AutoCAD or MicroStation files on the **Import** tab.

   b. When you are happy with the symbol, point the symbol to check the number of the symbol at the bottom of the window.



3. Click **File** --> **Save as...** and give a new name.

   We recommend that you save the file with a new name in a new location, for example, in the model, firm or project folder. Do not modify the

original symbol files delivered with your Tekla Structures software. If you need to modify any symbols, copy the original symbol file and work on the copy, keeping the original symbol file intact.

Tekla Structures reads the symbol files in certain search order, section "Symbol file search order" below.

4. Click **OK**.

TIP In Symbol Editor, you can copy symbols between symbol files (`*.sym`). Press **Ctrl + C** and select the symbol you would like to copy, then open the symbol file you want to copy to (or a new symbol file), select the location for the symbol and press **Ctrl + V**.

Change the symbol file in use

You can change the currently used symbol file if the current symbol file does not contain the symbol that you need.

Do one of the following, depending on what you are working with:

- If you want to change the symbol file for a symbol that you have added in a mark, double-click a mark in an open drawing. Double-click the symbol in the mark editor. In the **Mark content - symbol** dialog box, select a new symbol file from the list of symbol files. Then select the desired symbol.
- Open the **Symbol Properties** dialog box by double-clicking a symbol that you have added in a drawing. Click **Select...** next to the **File** box. Select a new file from the **Symbol files** list and click **OK** or double-click the file. Then select the desired symbol.

Create a new symbol file

In addition to the default symbol files that are delivered with Tekla Structures, you can create symbol files of your own, and save them in model, firm, or project folder, for example.

If you decide to use your own files, you can add `DXK_SYMBOLPATH` in the `options.ini` file under the model folder and define your own symbol file folder paths there. The symbol files are read in a certain search order, see section "Symbol file search order" below.

1. On the **File** menu, click **Editors** --> **Symbol editor**.
2. Click **File** --> **New**.

   You can also open an existing symbol file, edit it and save with a new name.
3. Create the symbol in the Symbol Editor.

4. Click **File** --> **Save** and save the symbol file in the folder that you use for keeping your symbol files.

   If you opened an existing symbol file, use **File** --> **Save as** and give the symbol file another name.

Modify symbol properties

You can modify the properties of symbols in an open drawing.

To modify the symbol properties:

1. Double-click a symbol.
2. Switch all the check boxes off in the dialog box by clicking the on/off switch at the bottom of the dialog box, and select only the check boxes next to the properties that you want to modify..
3. If necessary, change the symbol file in use and select the symbol to use.
4. To modify symbol placement settings, click **Place...**

   Here you can set the placing to free or fixed, specify the search margin, minimal distance and select the desired quarter to place the symbol.

   For more information, see Placement properties for marks, notes, dimensions, texts and symbols in drawings (page 1033).
5. Go to the **Appearance** tab and set the symbol color, height and angle, and the frame type, leader line and color.
6. Click **Modify**.

Customize leader line arrow symbols

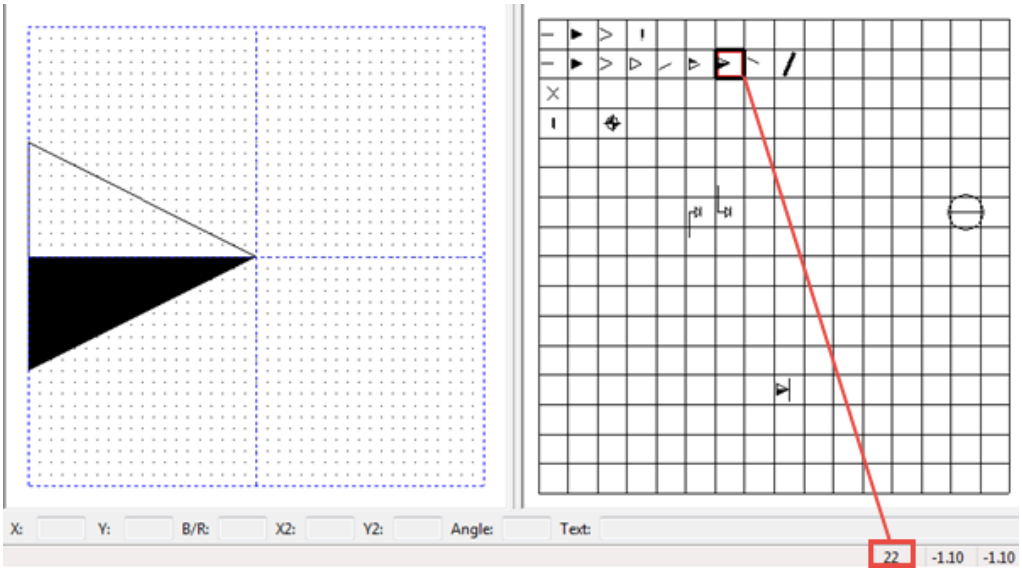
If you do not find a suitable leader line arrow in the **Arrow** list in mark properties, for example, you can add an arrow of your own in the list.

First you need to create a new arrow symbol in the Symbol Editor, and save the created symbol in the `arrow.sym` file. Then you need to add the position of the new symbol in the `arrow.sym` to the `arrow.txt` configuration file, which tells which arrows are available for use in your environment.

1. On the **File** menu, click **Editors** --> **Symbol editor** to open the Symbol Editor.
2. Open the `arrow.sym` file located in the common environment or in your own environment under the `\symbols` folder.
3. Click an empty symbol slot and sketch your symbol with the Symbol Editor drawing tools.

   You can also import AutoCAD or MicroStation files through **File** --> **Import** .

4. When the symbol is completed, point the symbol slot to check the number of the new symbol at the bottom of the window.

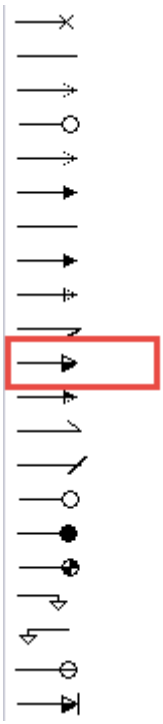


5. Save the `arrow.sym` file by clicking **File** --> **Save**.
6. Click **File** --> **Exit** to close the Symbol Editor.
7. Open the `arrow.txt` file located in the same `\symbols` folder as the `arrow.sym` file.

   The file contains a list of symbol numbers.
8. Add the number of your symbol preceded by zero (0) in the correct position and separate it with a comma:

   016,017,018,019,020,021,**022**,023,024,032,048,049,101,102,110,200
9. Click **File** --> **Save** to save your change.
10. Add an image of the arrow in the `.bmp` format in the `..\ProgramData\Trimble\Tekla Structures\<version>\Bitmaps` folder on your computer.

    Use the following format in the file name:
    `dr_dialog_arrow_type_022.bmp`.
11. Double-click a mark in a drawing to open mark properties.
12. Open the **Arrow** list, and you should see that the new arrow symbol is now available for use.



NOTE We recommend that you define a firm folder for symbols, because the default folders are overwritten when you upgrade to a newer version of Tekla Structures. Add the firm folder to the advanced optionDXK_SYMBOLPATH. For instructions about defining a firm folder, see the related section below.

Add surfacing symbols in drawings

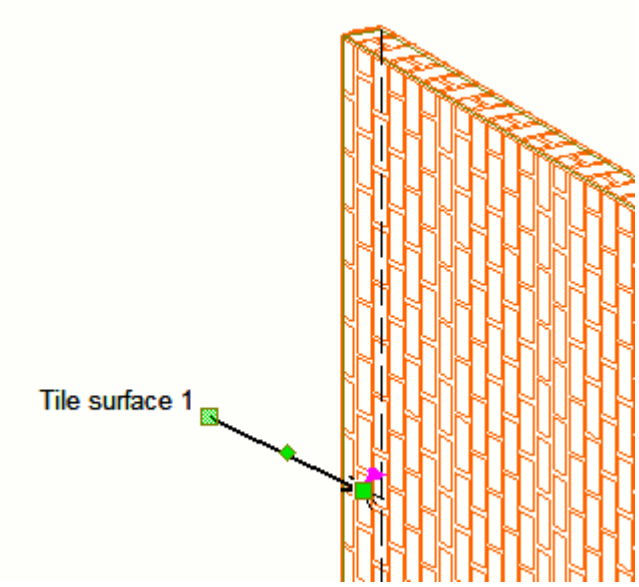
You can add surfacing symbols in cast unit drawings using the **Add surfacing symbols** macro.

Before you start, ensure that you have an object that has surface treatment in the model, and that you have created a cast unit drawing of that object. Also check in the view properties that surface treatment **Visibility** is set to **Visible**.

1. Open a cast unit drawing that has a part containing surface treatment.
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 **Add surfacing symbols**.
5. In the **Create surface symbols** dialog box, select the texts that you want to include in the surface treatment symbol from the **Available elements** list and add the texts to the **Elements in mark** by clicking **Add >**.

6. Select **All Views** to include the symbols in all drawing views, or **Selected Views** to include the symbols only in the views you select.
7. Change the font settings, if necessary.
8. If you selected **Selected Views**, select the views where you wish to have surface treatment symbols.
9. Click **Create**.

Tekla Structures creates the surface treatment symbols according to the defined settings. You can change the symbol properties and the text afterwards in the text properties that are displayed when you double-click the symbol. For example, you can change the symbol file and number.



Symbol Editor

You can open Symbol Editor by clicking **File** --> **Editors** --> **Symbol editor** . In Symbol Editor, you can create new symbol files, and create and modify symbols.

The **Symbol files** dialog box in drawing mode allows you to change the symbol file in use. It also provides access to Symbol Editor.

We recommend that you have a look at the Symbol Editor User's Guide so that you get familiar with creating new symbols and modifying the existing ones. You will find a link to the guide on the PDF documentation page.

We strongly recommend that you do not modify the original symbol files delivered with your Tekla Structures application. If you need to modify any

symbols, copy the original symbol file and work on the copy, keeping the original symbol file intact.

If your symbols are located in a protected folder, the symbols are read-only, because you cannot save a modified symbol in a protected folder if you are not an administrator. In this case, run Tekla Structures as administrator.

Symbol file search order

The symbol files (`.sym`) are searched from the following folders in the following order:

1. All `DXK_SYMBOLPATH` folders

   The advanced option DXK_SYMBOLPATH is defined in the environment initialization file `<your_environment>.ini` and in the Tekla Structures initialization file `teklastructures.ini`.

   Alternatively, you can add your own DXK_SYMBOLPATH in `options.ini` under the model folder and define your own symbol folder paths there. Note that also the path settings in the `<your_environment>.ini` file need to be added there.

   The reading order of the `.ini` files containing the `DXK_SYMBOLPATH` definition:

   1. `teklastructures.ini`
   2. `<your_environment> .ini`
   3. `options.ini`

2. Current model folder

   Add here any additional symbol files that you might need.

All symbol files that are found are available to be used in Tekla Structures. If there are duplicate file names, the first one found is used and the rest are discarded. If the model folder contains a symbol file that has the same name as another symbol file in `DXK_SYMBOLPATH`, the one in the model folder is discarded.

Example of overriding a symbol temporarily

If there is no permanent need to add your own symbol paths in the `options.ini` file in the model folder, you can temporarily override symbols. In this example, you will temporarily override the weld symbol with a customized symbol located in your model folder:

1. Customize the symbol (page 489) in `ts_welds.sym` first.
2. Copy the edited `ts_welds.sym` in your model folder `C:\TeklaStructuresModels\ <mymodel>`

3. Open the `options.ini` file located in your model folder, and add the line `DXK_SYMBOLPATH=C:\ TeklaStructuresModels\<mymodel>; C:\TeklaStructures\2017\Environments\common\symbols;` + possible other symbol folders in your local environment.

Define a firm folder for images and symbols

You can define a firm folder where Tekla Structures always searches for the images and symbols. When you store the images and symbols in this folder, you do not have to move them from folder to folder when you install a new version of Tekla Structures. Installing a new version does not replace the files in the firm folder.

Define the firm folder in the `options.ini` file under the model folder or `user.ini` using the advanced option `XS_FIRM`.

To define the firm folder for images and symbols in the `options.ini` file:

1. Modify the `options.ini` file to include the advanced option `DXK_SYMBOLPATH` that points to the firm folder.

   The advanced option `DXK_SYMBOLPATH` may contain multiple paths separated by semicolons.

   Since Tekla Structures version 19.0, the folder path definitions like `%DATADIR%` or `%XS_FIRM%` in `DXK_SYMBOLPATH` have not converted the paths correctly when used in `options.ini` located in the firm folder. Currently in `options.ini` in the firm folder you need to write the absolute paths for `DXK_SYMBOLPATH`, like in the example below:

   ```
   DXK_SYMBOLPATH=C:\ProgramData\Trimble\Tekla Structures
   \2020\Environments\common\symbols\;C:\firm\Symbols\;
   ```

2. In Template Editor, click **Options** --> **Preferences** --> **File Locations** and enter the path to the firm folder also for **Symbols, pictures**.

4.11 Add links in drawings

You can add the following kinds of links in drawings:

- Links to text files
- Links to other drawings
- Hyperlinks to internet (url) addresses
- Links to DWG/DXF files
- Links to images

Add links to rich text files

You can insert a link to a text inside a frame in a drawing. First you create a `.txt` or `.rtf` file in WordPad, and then add a link to it in a Tekla Structures drawing. Tekla Structures adds the text using some basic formatting settings you can set in the file itself, and some of the properties in **Text File Properties**.

NOTE If you change the text in the text file, it will change in all drawings containing a link to the text file.

Limitation: The `.rtf` texts need to be created with WordPad.

1. Create the text file.

   You can add basic formatting to the file, such as bold, italics, underline, strikethrough, subscript, and superscript, use indent and tabs, and set a specific font. You can add bulleted lists with hyphens and circular bullets, and numbered lists, and nest the lists. Simple tables are also supported.

   When you add the text link, Tekla Structures preserves the formatting settings.

   Note that the text links do not support hyperlinks or images.

   Tekla Structures colors will always be shown correctly. If you use in the `.rtf` file a color that is not supported in Tekla Structures, then the closest Tekla Structures color will be used.

2. Open the drawing where you want to add the text file.

3. On the **Annotations** tab, click **Rich text**.

4. Browse for the file.

5. For `.txt` files, you can set the text color, height and font.

6. For `.rtf` files, set the desired scale.

   The **Scale** tells how much the `.rtf` content should be scaled. If you define 1 as the scale, the letters will keep the same size as the letters in the original `.rtf` file.

   Color, height and font settings are not adjustable.

7. Set the frame line type and color.

8. Select if you want to adjust how the text is placed:

   - **Scale to fit** : You only need to pick the upper left corner of the frame when placing the text. Tekla Structures inserts the object in its original size. In this case, when you change the text frame size by dragging from the handles, the text does not wrap, and the font scales automatically.

- **No scaling**: You only need to pick the upper left corner of the frame when placing the text. Tekla Structures adjusts the object size to fit the frame. The text wraps when you change the size of the frame by dragging the handles. The minimum width of the box is defined by the longest word.

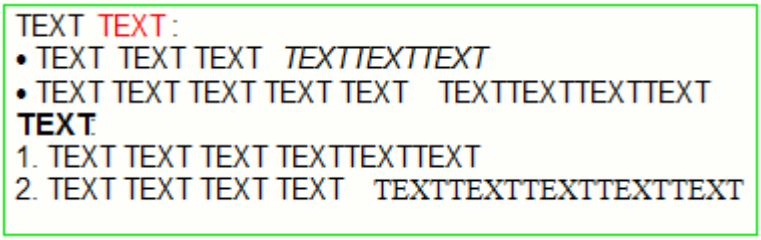
9. Click **OK** or **Apply**.
10. Pick a location for the upper left corner of the text frame in the drawing.

    Tekla Structures adds the link to the text file.
11. You can edit the text and modify the text file properties:
    - To edit the text file, double-click the text inside the frame. Tekla Structures opens the original text file.
    - To modify the text file properties, double-click the frame around the text. For `.rtf` files, you cannot modify the text color or font here, you need to changes these in the `.rft` file itself. For plain text files, you can also change the font color.

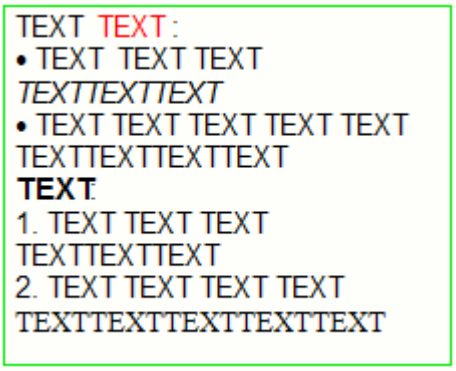
      You can also open the text file properties from **Quick Launch**: start typing `text file properties`, and select **Text File Properties** from the list.

Examples

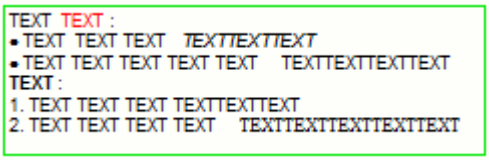
In the following example you can see that the text file may contain bulleted lists, numbered lists, colors, italics, and bold, and you can change the font of the desired text parts.



In the following example, the scaling type is **No scaling**. The text wraps when you change the size of the frame by dragging the handles, so that the text always fits the frame. The font size does not change.



In the following example, the scaling type is **Scale to fit**. If you change the text frame size by dragging from the handles, the text does not wrap, and the font size changes automatically so that the text fits the frame.



Add links to other drawings

You can insert a link to another drawing in a frame in a drawing. Tekla Structures adds the link to the drawing using the properties in **Drawing Link Properties**.

1. Open the drawing.
2. On the **Drawing** tab, click **Link** --> **To another drawing** .
3. Modify the text color, height, font and effect.
4. Modify the frame line type and color.
5. Select if you want to scale the link. If you select **No scaling**, Tekla Structures inserts the link in its original size.

   If you select **Scale to fit**, Tekla Structures adjusts the object size to fit the frame.
6. In the **Drawing** list, select the drawing to link to.

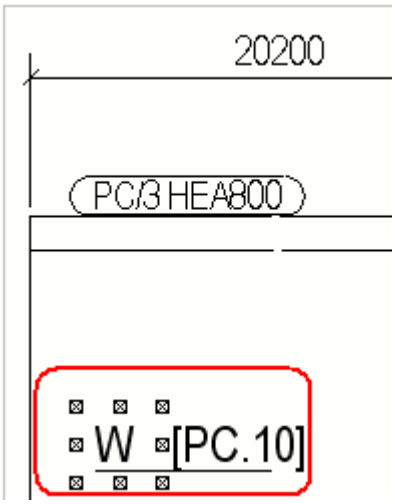
   The drawings on the list are the drawings in the current model.
7. If you want to display text for the link instead of the drawing name, enter the text in the **Text** box.
8. Pick two points to define the frame and add the link.
9. Click **OK** or **Apply**.

You can open the linked drawing by double-clicking the link.

Note that you can also open drawing link properties from **Quick Launch**: start typing `drawing link` and select **Drawing Link Properties** from the displayed list.

Example

In the example below, **Scale to fit** has been selected, and the link contains the name of the drawing.



Add hyperlinks

You can add hyperlinks to internet addresses (URLs) or files within a frame in a drawing.

1. Open a drawing where you want to add a hyperlink.
2. On the **Drawing** tab, click **Link** --> **Hyperlink** .
3. Modify the text color, height, font and effect.
4. Modify the frame line type and color.
5. Select if you want to scale the link.

   If you select **No scaling**, you only need pick the upper left corner of the frame when you insert the link. Tekla Structures inserts the link in its original size. If you select **Scale to fit**, you need to pick two points to define the frame. Tekla Structures adjusts the link size to fit the frame.
6. In the **File or URL** text box, type or paste an internet address or a file name and path.

   To locate the file, click **Browse...**. Tekla Structures inserts an active hyperlink to the location you specify.
7. If you want to display text for the hyperlink instead of the hyperlink, type the text in the **Text** box.

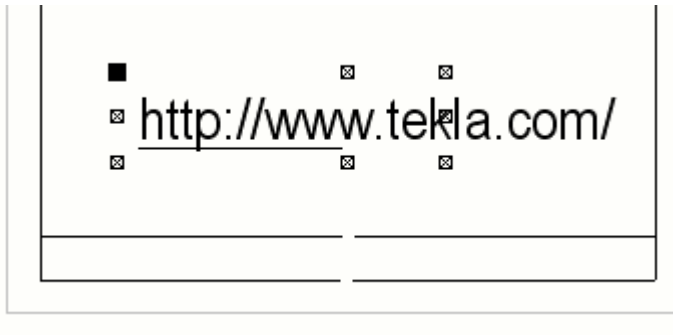
8. Click **OK** or **Apply**.
9. Pick one or two points in the drawing to indicate the corners of the hyperlink frame.

Double-click the hyperlink text in the drawing to go to the internet address in a browser.

Note that you can also open hyperlink properties through **Quick Launch**: start typing `hyperlink properties`, and select **Drawing hyperlink properties** from the displayed list.

Example

In the example below, **Scale to fit** has been selected. The internet address for the hyperlink is shown.



Add links to DWG and DXF files

You can insert a DWG or DXF file inside a frame in a drawing as a link. Tekla Structures adds the DWG or DXF file link using the properties in **DWG/DXF Properties**. When you modify the original file, Tekla Structures also modifies all the linked instances in drawings.

Tekla Structures supports AutoCAD version 2018 and earlier in DWG/DXF links. The added dwg/dxf files also support AutoCAD control codes, such as `%%u` for underscore, or `%%c` for a circle diameter dimensioning symbol ⌀.

1. Open the drawing where you want to insert a link to a DWG/DXF file.
2. On the **Drawing** tab, click **DWG/DXF**.
3. Select the scaling options:
   - **Scaling**:
     - **X**: When you use this setting, you need to pick the left upper corner of the frame to insert the file. You can only set the drawing scale in the X direction.
     - **XY**: When you use this setting, you need to pick the left upper corner of the frame to insert the file. You can set the drawing scale in both the X and Y direction.

    - **Scale to fit**: When you use this setting, you need to pick the left upper and bottom right corners of the frame to size and create the frame. Tekla Structures scales the file to fit the frame.
    - **Best fit**: When you use this setting, you need to pick the left upper and bottom right corners of the frame to size and create the frame. Tekla Structures scales the file to fit, maintaining its original aspect ratio.
  - **Scale in X**
    - Scales the file in the X direction. Enter a coefficient to indicate the scale, for example, 1.0 for 100%, 1.5 for 150%, etc. The scaling type must be set to **X** or **XY**.
  - **Scale in Y**
    - Scales the file in the Y direction. Enter a coefficient to indicate the scale, for example, 1.0 for 100%, 1.5 for 150%, etc. The scaling type must be set to **XY**.
4. Select the link frame line type and color.
5. In **Name**, browse for the DWG or DXF file you want to use.
6. Pick one or two points in the drawing to place the frame.
7. Click **OK** or **Apply**.

   Tekla Structures adds a link to the DWG or DXF file inside a frame in the drawing.

   Note that you can also open DWG/DXF properties through **Quick Launch**: start typing `DWG/DXF`, and select **Drawing DWG/DXF properties** from the displayed list.

TIP If you want to insert DWG files as reference models in your model, see Insert a reference model.

Add links to image files

You can insert a images inside a frame in a drawing. Tekla Structures adds the image link using the properties in **Image properties**. When you modify the original file, Tekla Structures also modifies all the linked instances in drawings.

The supported file types are `.bmp`, `.jpg`, `.jpeg`, `.gif`, `.tif`, `.tiff`, and `.png`.

1. Open the drawing where you want to insert a link to an image.
2. On the **Drawing** tab, click **Image** .
3. Select the scaling options:

- **Type**
  - **X**: When you use this setting, you need to pick the left upper corner of the frame to insert the image. You can only set the drawing scale in the X direction.
  - **XY**: When you use this setting, you need to pick the left upper corner of the frame to insert the image. You can set the scale in both the X and Y directions.
  - **Scale to fit**: When you use this setting, you need to pick the left upper and bottom right corners of the frame to size and create the frame. Tekla Structures scales the image to fit the frame.
  - **Best fit**: When you use this setting, you need to pick the left upper and bottom right corners of the frame to size and create the frame. Tekla Structures scales the image to fit, maintaining its original aspect ratio.
- **Scale in X**
  - Scales the file in the X direction. Enter a coefficient to indicate the scale, for example, 1.0 for 100%, 1.5 for 150%, etc. The scaling type must be set to **X** or **XY**.
- **Scale in Y**
  - Scales the file in the Y direction. Enter a coefficient to indicate the scale, for example, 1.0 for 100%, 1.5 for 150%, etc. The scaling type must be set to **XY**.

4. Select the link frame line type and color.
5. In **Name**, browse for the image file you want to use.
6. Pick the points in the drawing to place the frame.
7. Click **OK** or **Apply**.

   Tekla Structures adds a link to the image file inside a frame in the drawing.

4.12 Add sketch objects in drawings

Sketch objects are objects that you might want to add in a drawing to highlight important information, or to indicate the positions of openings, windows, doors, and such. In Tekla Structures drawings, you can draw clouds, lines, circles, rectangles, arcs, polylines, and polygons, for example.

You can manually explode and combine many types of sketch objects, cover part lines and faces, and create fillets and chamfers. You can re-order sketch

objects, or sketch objects and building objects. You can also trim, extend, split and divide sketch objects, and copy objects using offset.

Sketch objects are associative if they have associativity points, which means that they are attached to a building object through a point, for example, a leader line start point.

Click the following links to find out more about sketching tools and sketch objects:

Draw and modify sketch objects in drawings (page 388)

Combine and explode sketch objects in drawings (page 391)

Create and add pattern lines in drawings (page 394)

Re-order sketch objects in drawings (page 393)

Trim drawing lines (page 401)

Split sketch objects (page 403)

Divide sketch objects (page 403)

Create fillets in drawings (page 405)

Create chamfers in drawings (page 406)

Copy sketch objects with offset (page 404)

Hide part face areas and outlines with cover-up tools (page 413)

See also

Drawing objects, views and layouts (page 68)

Drawing sketch object properties (page 1038)

Draw and modify sketch objects in drawings

You can draw lines, polylines, polygons, rectangles, circles, arcs, and clouds in an open drawing with drawing sketching tools.

1. To open the properties first and then draw the sketch object, hold down **Shift**, and on the **Drawing** tab, click one of the following commands:

	Line --> **Draw line**: Sketch a single line between two points that you pick.
	Line --> **Draw single lines continuously**: Create multiple single lines at one go using the current line properties. Tekla Structures creates a continuous line as separate line sections.

	You can combine (page 391) the sections into one by using the **Combine** command. Note that you can open the properties for the continuous single lines only after you have created the line.
	Line --> **Pattern line**: Sketch a line containing a pattern that you select using the current pattern line properties. **Tip:** If you need a new pattern, use the **Pattern line editor** to create it (**File** --> **Editors** --> **Pattern line editor**).
	Draw rectangle: Sketch a rectangle between the points that you pick. You can create rectangles with horizontal and vertical sides.
	Draw circle by center and radius: Sketch a circle by picking the center point first and then a point on the circle that specifies the radius.
	Draw circle by three points: Sketch a circle that traverses the three points you pick, either clockwise or counter-clockwise.
	Draw arc by end points and center: Sketch an arc by specifying two end points first and then a center point of the circle to which the arc belongs.
	Draw arc by three points: Sketch an arc that traverses the three points that you pick, either clockwise or counter-clockwise.
	Draw polyline: Sketch a line with straight segments using the current polyline properties. Remember to set the **Bulge for all lines** factor if you are creating curved polylines.
	Draw polygon: Sketch a polygon by picking the corner points using the current polygon properties. To close the polygon, pick the start point again or click the middle mouse button. To stop, press **Esc**.

	Draw cloud: Sketch a cloud that traverses the points that you pick. To close the cloud, click the middle mouse button. Remember to set the **Bulge for all lines** factor for the clouds, otherwise the cloud shape is not drawn properly.

2. Modify the sketch properties as required. For example, you can add hatch patterns in rectangles, polygons, and circles. For details, see Drawing sketch object properties (page 1038).
3. Sketch the objects by following the instructions below and on the status bar. To stop sketching, press **Esc**.

While using sketching tools

- While sketching a polyline or a polygon, you can:
  - Snap to temporary graphical points and set them as reference points by holding down **Ctrl** and clicking a point on a temporary line.
  - Set multiple reference points using DWG objects and Tekla Structures building objects.
  - Delete a reference point by holding down **Ctrl** and clicking the point.
  - Create a polygon by clicking the start point of the polyline.
  - Delete the selected corner point by right-clicking and selecting **Delete**.
  - Add handles by dragging a handle point on the polyline or polygon. You can then change the shape by dragging the handles.
  - You can explode (page 391) rectangles, polylines, and polygons into separate straight lines, and combine (page 391) polylines into closed polygons.
- To move and rotate shapes, click **Move** --> **Rotate** on the **Drawing** tab.
- To resize shapes, such as rectangles, drag the shape handles.
- Object dimensions are shown while you draw the objects, and also when you select the objects. To hide the dimensions of the objects, use the **Toggle graphical object dimensions** command in drawings: Go to **Quick Launch**, start typing `toggle graphical object dimensions`, and select the **Toggle graphical object dimensions** command from the list that appears, or use the **Ctrl + D** shortcut. When the dimensions are visible, a small **d** is displayed on the status bar.
- You might want to activate orthogonal snapping by pressing **O** while sketching to show ortho angles. You can define snapping steps (page 53) and custom angles in **Drawing snap settings** (**File** --> **Settings** --> **Snap settings**).

- The view scale is taken into account when you copy sketch objects from one view to another, which means that the scale of the object and the related numerical information adjusts to the scale of the target view.
- Sketch object dimensions take into account dimension settings in drawings in general. This means, for example, that units can be controlled in the same way as for the other dimensions. For more information, see Units and decimals (page 893).
- You can also change the mutual order of sketch objects (graphical objects), and the mutual order of sketch objects and building objects. You can bring sketch objects forward or in front of other sketch objects, or send them backward or behind other sketch objects. You can also send sketch objects behind building objects or bring them in front of building objects. You can also re-order DWGs and images in the same way. For more information, see Re-order sketch objects in drawings (page 393).

Modify sketch objects

1. Click the sketch object that you want to modify. If the property pane is not open, double-click the object.

   To select several sketch objects, hold down **Ctrl** and click the objects.
2. Modify the properties.

   If you selected several sketch objects, only properties common to all objects are displayed. The properties that have different values have the text *Varies*. You can select another value from the list.
3. Click **Modify** to apply the changes in all selected sketch objects.

See also

Drawing sketch object properties (page 1038)

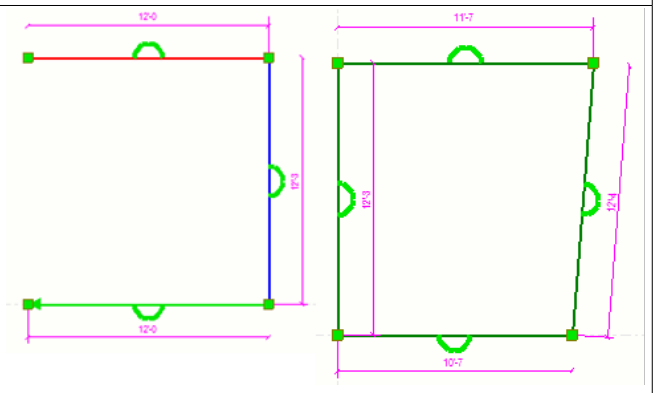
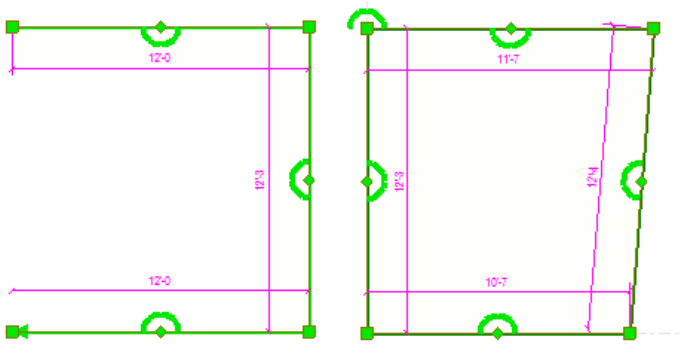
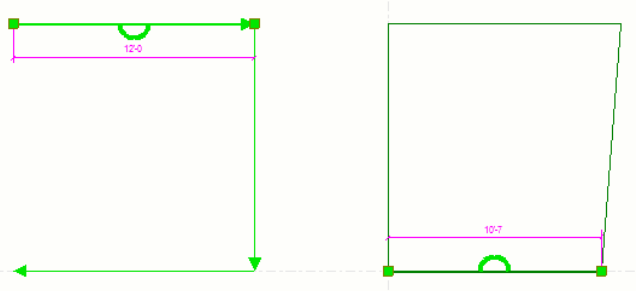
Hide part face areas and outlines with cover-up tools (page 413)

Define customized line types in TeklaStructures.lin (page 899)

Combine and explode sketch objects in drawings

You can explode sketch objects like polylines, rectangles, or polygons into separate straight lines. You can also combine separate lines, continuous lines, and polylines into a single polyline or a closed polygon.

To	Do this
Combine separate lines, continuous lines or polylines into a single polyline or a closed polygon	1. Select the separate lines or polylines.

To	Do this
	 2. Go to the **Drawing** tab and click **Combine** . Separate lines and polyline are combined into a single polyline or a closed polygon. When you select the combined shape, you can see that Tekla Structures has combined the lines and created more handles in the shape: 
Explode a polyline, rectangle, or polygon into separate straight lines	1. Select polylines, rectangles or polygons. 2. Go to the **Drawing** tab and click **Explode** . The polylines, rectangles and polygons are exploded into single lines: 

See also

Draw and modify sketch objects in drawings (page 388)

Re-order sketch objects in drawings

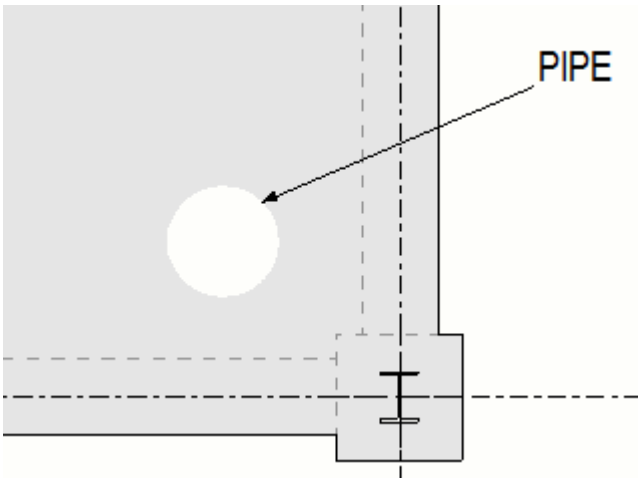
In drawings, you may want to indicate positions of openings, windows, doors, and such with sketch objects. You can change the mutual order of sketch objects (graphical objects), and the mutual order of sketch objects and building objects. You can bring sketch objects forward or in front of other sketch objects, or send them backward or behind other sketch objects. You can also send sketch objects behind building objects or bring them in front of building objects. You can also re-order sketch objects with DWGs and images in the same way.

If you have several sketch objects (objects on several layers), the drawing order affects on which layer the forward and backward commands place the objects.

Newly created sketch objects are placed on their own layer in the drawing so that newer ones are on top of the older ones.

1. Right-click a sketch object, DWG or an image and click **Re-order**.
2. Select one of the following commands:
   - **Send backward**: Moves the selected object one step closer to the back of all other sketch objects.
   - **Send to back**: Places the selected object behind all other sketch objects.
   - **Bring forward**: Moves the selected object one step closer to the front of all other sketch objects.
   - **Bring to front**: Places the selected object in front all other sketch objects.
   - **Send behind model objects**: Places the object behind all building objects. You can also set this option in sketch object properties (**Behind model objects** --> **Yes**).
   - **Bring in front of model objects**: Places the object in front of all building objects. You can also set this option in sketch object properties (**Behind model objects** --> **No**).

     In the example below, a circle has been drawn with white filling, and the circle is has been brought in front of the grey building object.



NOTE
- Sketch objects that are placed behind building objects cannot be rearranged together with sketch objects that are placed in front of the building objects.
- You cannot change the mutual order of the building objects.

Create and add pattern lines in drawings

You can easily apply special and complex lines to drawings by using pattern lines. You can create new pattern lines and modify existing pattern lines in **Pattern line editor**, and add the created pattern lines in drawings using the **Pattern line** command. Pattern lines may include symbols, lines and spacings. You can create pattern lines to indicate cut lines, foundations, weld paths, or electrical wiring, for example.

Create a pattern line

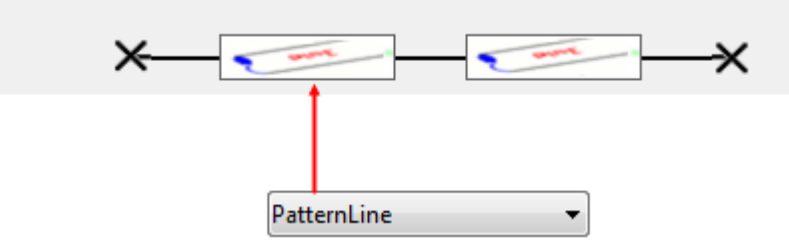
1. In the model or when a drawing is open, on the **File** menu, click **Editors** --> **Pattern line editor**.

   If you have not defined a pattern line earlier, all **Pattern line editor** fields and containers are empty.

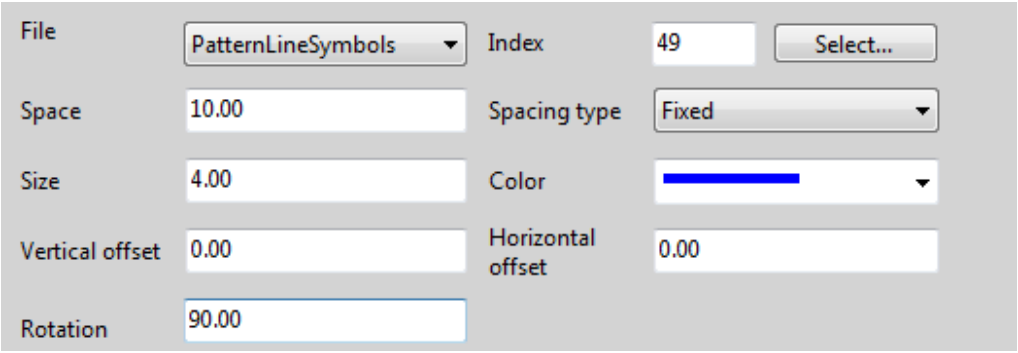
2. Click **Browse...** to browse for an image to be used as a sample for the pattern line.

   If you do not select an image, the default image will be used.

   The selected image is also shown later on when you select a pattern line in the **Pattern line properties** dialog box.



3. Enter a name for the pattern line in **Create new**. If you want to modify a pattern line, select the desired pattern line from the list first.
4. In the **Start element** area, click **New** on the right to create a new start element. The start element starts the pattern line (marked with red in the dialog box example below).



If you select **Symbol** as the **Element type**:

- Select the symbol file from the **File** list.
- In the **Index** list, enter the number of the symbol. To select a symbol in a separate window, click **Select...** and double-click the desired symbol.
- In **Space**, enter the space allocated for the element in millimeters.
- In **Size**, enter the size of the element in millimeters.
- In **Spacing type**, select the spacing type for the elements on the pattern line:

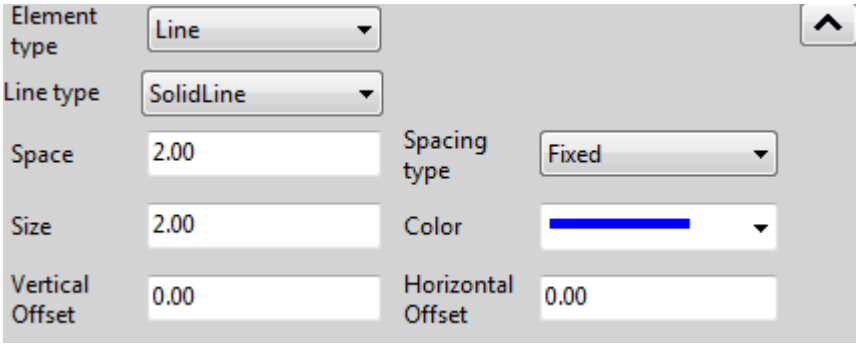
  Fixed: The unoccupied space will not be distributed.

  Variable: The unoccupied space will be equally distributed to the elements with this spacing type.

  Infinite: The blocks will occupy all the space between the two points. The space will be distributed to fixed elements, and the rest will be equally distributed to the infinite elements.
- In **Color**, select the element color.

- In **Vertical offset** and **Horizontal offset**, enter offsets for the elements in millimeters.
- In **Rotate**, define a rotation angle for the element.

If you select **Line** as the **Element type**:



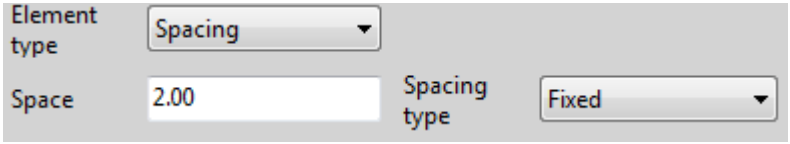
- Select a line type from the **Line type** list.
- In **Space**, enter the space allocated for the element in millimeters.
- In **Size**, enter the size of the element in millimeters.
- In **Spacing type**, select the spacing type for the elements:

  Fixed: The unoccupied space will not be distributed.

  Variable: The unoccupied space will be equally distributed to the elements with this spacing type.

  Infinite: The blocks will occupy all the space between the two points. The space will be distributed to fixed elements, and the rest will be equally distributed to the infinite elements.
- In **Color**, select the element color.
- In **Vertical offset** and **Horizontal offset**, enter offsets for the elements in millimeters.

If you select **Spacing** as the **Element type**:



- In **Space**, enter the space allocated for the element in millimeters.
- In **Spacing type**, select the spacing type for the elements:

  Fixed: The unoccupied space will not be distributed.

Variable: The unoccupied space will be equally distributed to the elements with this spacing type.

Infinite: The blocks will occupy all the space between the two points. The space will be distributed to fixed elements, and the rest will be equally distributed to the infinite elements.

5. In the **End element** area, click **New** to create a new end element in the same way as you did for the start element.

   The end element is marked with red in the editor dialog box:

6. On the **Repeating and continuous elements** tab, define the elements that are repeating on the pattern line (marked with red in the editor dialog box):

7. On the **Repeating and continuous elements** tab, define the elements that are drawn continuously on the pattern line (marked with red in the editor dialog box):

8. You can check your pattern line definitions and how the line looks by clicking the **Preview** button :

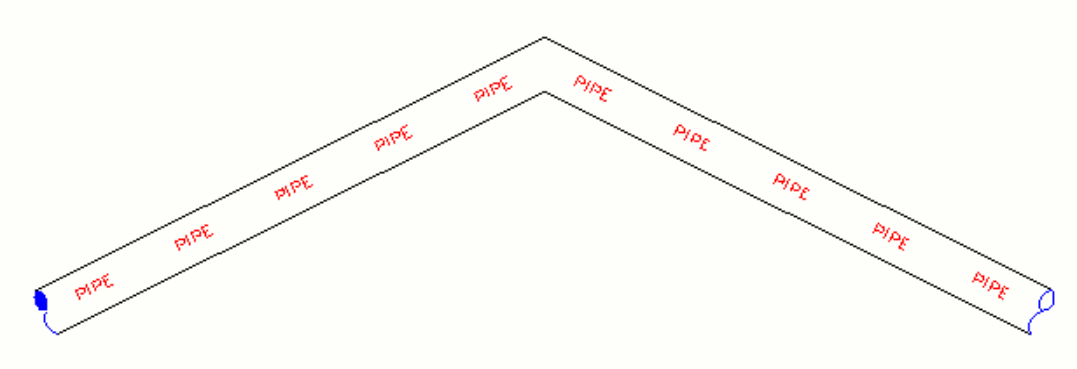
   - If you have the **Automatic** preview option active, Tekla Structures automatically opens a drawing preview that shows the created pattern line with different spacing options taken from the **Pattern line properties** dialog box. If you already have a drawing open, Tekla Structures will ask if you want to save and close the drawing before the preview is opened.
   - To update the preview by clicking the button , select the preview option.
   - You can also see previews of other saved pattern lines by selecting a pattern line from the list at the top.
   - To close the preview, click the **Preview** button again.

9. Save your pattern line by clicking the **Save** button . The pattern line is saved in a `.clp` file in the `\attributes` folder under the model folder.

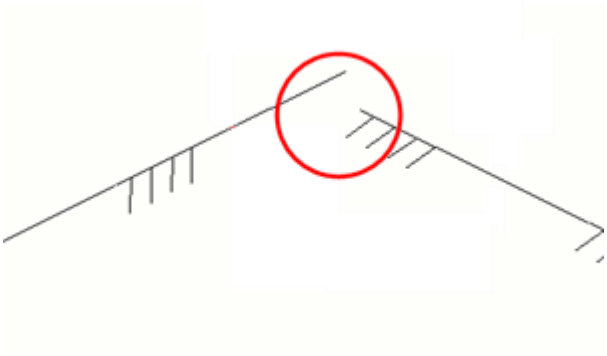
Now you can open a drawing and create a pattern line using the pattern line that you defined.

The symbols in the following example are taken from the `PatternLineSymbols.sym` symbol file.

- The symbol at the left end is the start element, symbol #49.
- The symbol at the right end is the end element, symbol #48.
- Blue color is defined for the start and end elements.
- Symbol #51 in red is used as a repeating element.
- Two SolidLines are used as continuous elements with vertical offsets 2.0 and -2.0.



Tip: If your pattern lines are jumping like in the example below, try using continuous elements instead of repeating elements.



Add a pattern line in a drawing

You may select among predefined pattern lines, or create your own in **Pattern line editor**.

Limitations

- Pattern lines are not included in a **2D Library** detail unless the pattern line has been exploded.
- You cannot rotate or mirror created pattern lines.

- You cannot clone pattern lines.

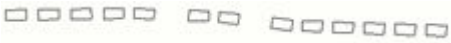
1. In an open drawing, hold down **Shift** and click **Drawing** --> **Line** --> **Pattern line**.
2. In the **Pattern line properties** dialog box, select the pattern line that you want to use from the **Pattern line** list. You can also load other pattern line properties by selecting a properties file from the list at the top and clicking **Load**.
3. Select whether to show the **Start element** and/or the **End element** in the pattern line. The start and end elements are defined in the **Pattern line editor**.
4. Select how the elements are aligned and spaced in **Spacing options**:

   Left aligns the elements to the left.

   Right aligns the elements to the right.

   Centered centers the elements.

   Distributed uses the same space in between the elements.

   Best fit tries to fit as many elements as possible with minimal distortion. This setting affects both element size and space.

5. Set the **Size** of the pattern line.
6. Select the desired **Scaling** option:

   View draws the line and elements scaled down to the view.

   Paper draws the line and elements scaled down to the scale of the paper.
7. Enter the **Vertical offset** in millimeters if necessary. A positive value moves the whole pattern line upwards and a negative value downwards.
8. Select the **Color** of the pattern line.

   The color depends on how you set the colors in **Pattern line editor** and **Pattern line properties**:

   - If you select **Default** as the color both in **Pattern line editor** and in **Pattern line properties**, all elements will be black.

- If you select the **Default** color in **Pattern line editor**, and you select another color in **Pattern line properties**, the color selected in **Pattern line properties** will be used.
- If you select some other color than **Default** in **Pattern line editor**, and you select another color in **Pattern line properties**, the color selected in **Pattern line editor** properties will be used.

9. If you want to save the pattern line properties in a properties file, enter a name for the file and click **Save as**.
10. Click **OK**, pick the points for a polyline, and end the line by clicking the middle-mouse button. Note that you can select the handles in the added pattern line and move the sections of the polyline by dragging.

Pattern line elements

A pattern line consists of one or several blocks of elements. These element blocks are arranged between two given points.

The illustration below shows a pattern line consisting of two repeating symbol elements in three blocks.

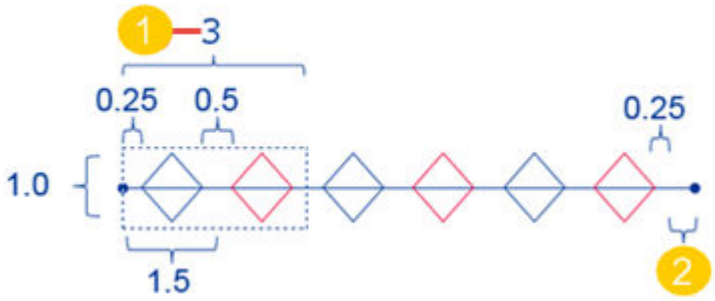
The symbol elements have the following settings:

Space = 1.5

Spacing type = Fixed

Size = 1

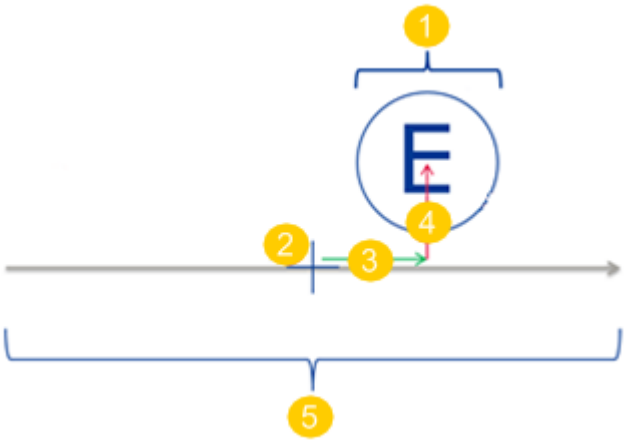
Color = 1st element is blue, and the 2nd element is red



1. Block size

2. Unoccupied space, which is the space that is left when the blocks are arranged between two points. This space is redistributed to the elements which have the spacing type **Variable**.

The following illustration describes the different element settings. The letter E inside a circle is an element:

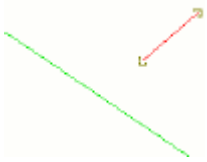


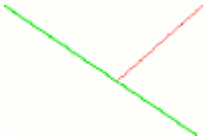
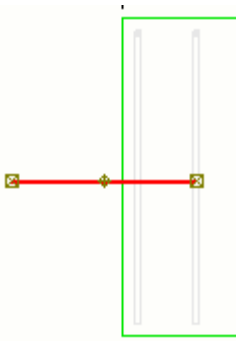
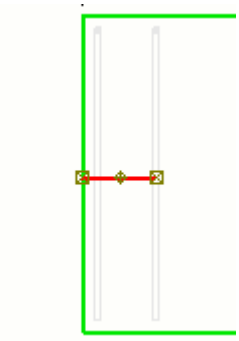
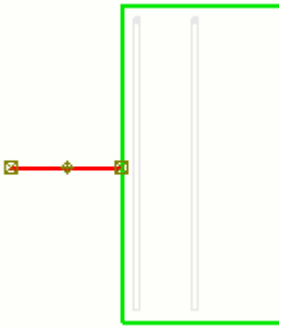
1. Size of the element
2. Origin
3. Horizontal offset from the origin
4. Vertical offset from the origin
5. Space

Trim drawing lines

You can shorten or extend a line relative to the boundary you select in an open drawing. The boundary can be a line, part, arc, or rectangle, for example.

1. On the **Drawing** tab, click **Trim** .
2. Select the object you want to use as a boundary.
3. Click the middle mouse button.
4. Do one of the following:

To	Do this
Extend the line	Click the end of the line to extend it to the boundary line. Original lines:  The extended line:

To	Do this
	
Shorten the line from the left or right end	Click the line at the end from which you want to shorten it. Original line:  Line clicked at the left end:  Line clicked at the right end: 

See also

Add sketch objects in drawings (page 387)

Split sketch objects

You can cut a sketch object into two pieces at a point you select in an open drawing. You can split lines, polylines, circles, and arcs.

1. Select the line.

2. On the **Drawing** tab, click **Split**.
3. Pick a point on the line to indicate the location for splitting.

4. Tekla Structures splits the line into two.

See also

Add sketch objects in drawings (page 387)

Divide sketch objects

You can divide a sketch object into a number of segments that you specify in an open drawing. You can divide lines and arcs.

1. Select the line.

2. On the **Drawing** tab, click **Divide**.
3. In the displayed **Segments** dialog box, enter the number of segments, for example 4, and click **OK**.

Tekla Structures divides the line into four lines.

See also

Add sketch objects in drawings (page 387)

Copy sketch objects with offset

You can copy lines, circles, polylines, polygons and rectangles in the direction you point using the offset that you specify. You can also create new circles centered in the same location as the original circle and adjust the radius by the offset that you specify.

1. Select the object that you want to copy in a drawing, for example a line or a circle.
2. On the **Drawing** tab, click **Copy with offset**.
3. Click the drawing view in the direction where you want to copy the object.
4. Type the offset in the displayed box , and press **Enter**.

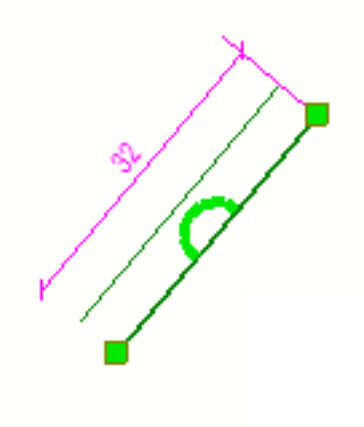
   Tekla Structures copies the selected shape to the specified direction. For example, if you are copying a line, Tekla Structures makes a new copy of the line in the specified location. If you are copying a circle, Tekla Structures creates a new circle that is centered in the same location as the original circle, and adjusts the radius by the offset that you specified.

   When you copy a line that has a bulge, the created line has the same bulge as the original one.

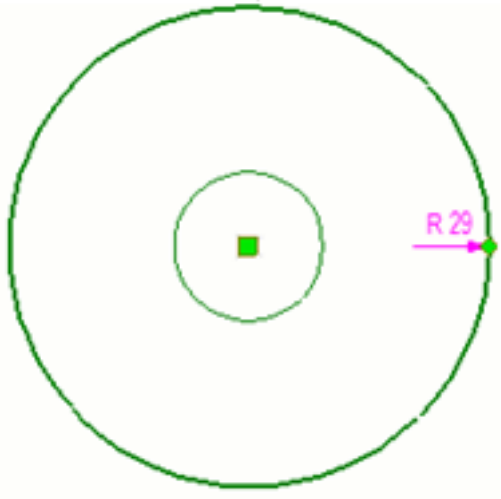
   If the rectangle has no bulges, the copied object will still be a rectangle after you use Segments. Note that if the original rectangle has a bulge, it is not possible to draw a new rectangle using **Copy with offset**. When you use segments, the new copied object will become a polygon.

Example

Example of a copied line:



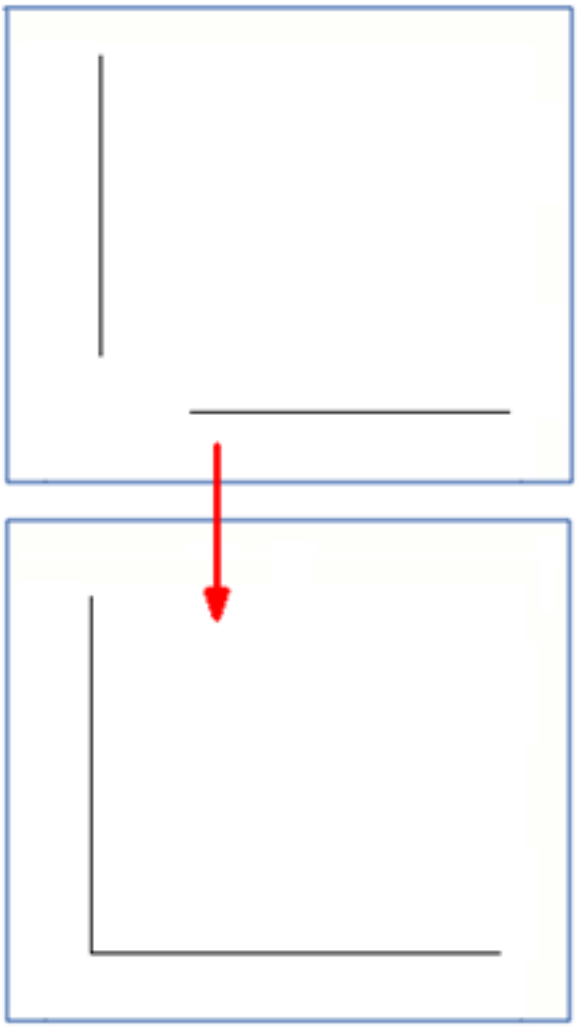
Example of a copied circle:



Create fillets in drawings

You can connect two intersecting lines by extending the two selected lines to their intersection point by creating a fillet. If no intersection point is found or if it is outside the drawing, nothing will be done.

1. Open a drawing.
2. Select two intersecting lines.
3. On the **Drawing** tab, click **Create fillet**.



Create chamfers in drawings

You can create chamfers between two lines using a distance that you specify. You can create both straight and round chamfers.

1. Select two intersecting lines.
2. Do one of the following:

   - To create a straight chamfer, on the **Drawing** tab, click **Create straight chamfer**.

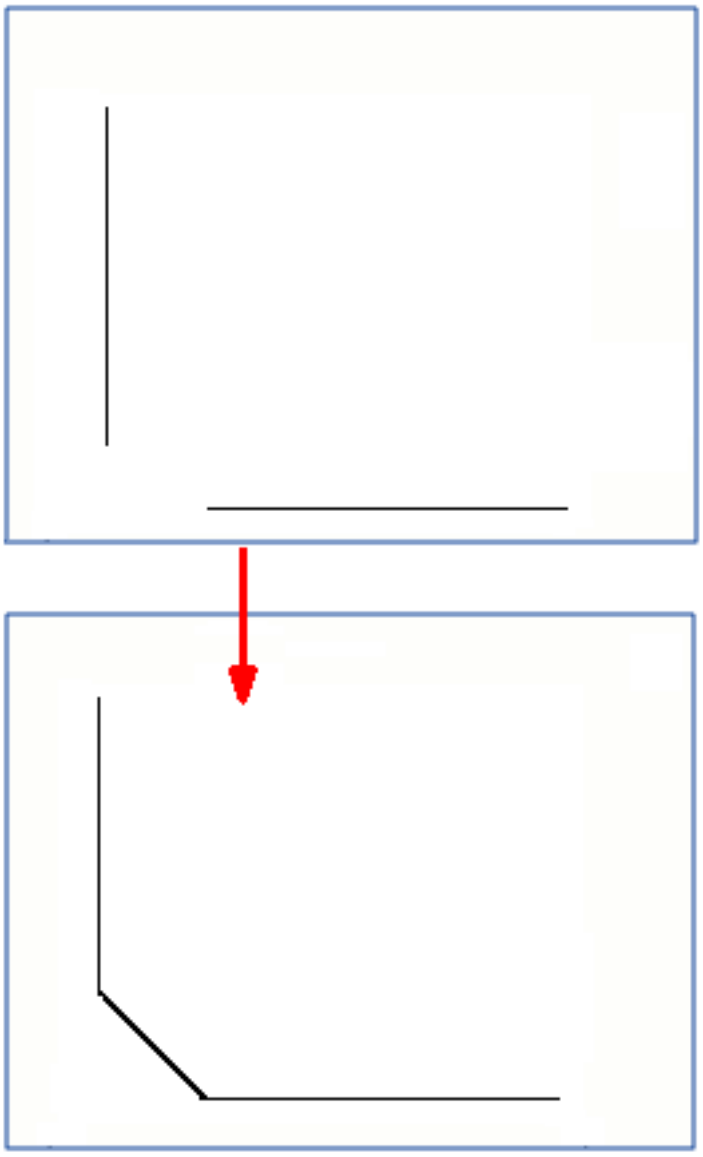
     Enter the distance that you want to have between the two lines (the length of the chamfer line) in the displayed dialog box.

- To create a round chamfer, on the **Drawing** tab, click **Create round chamfer**.

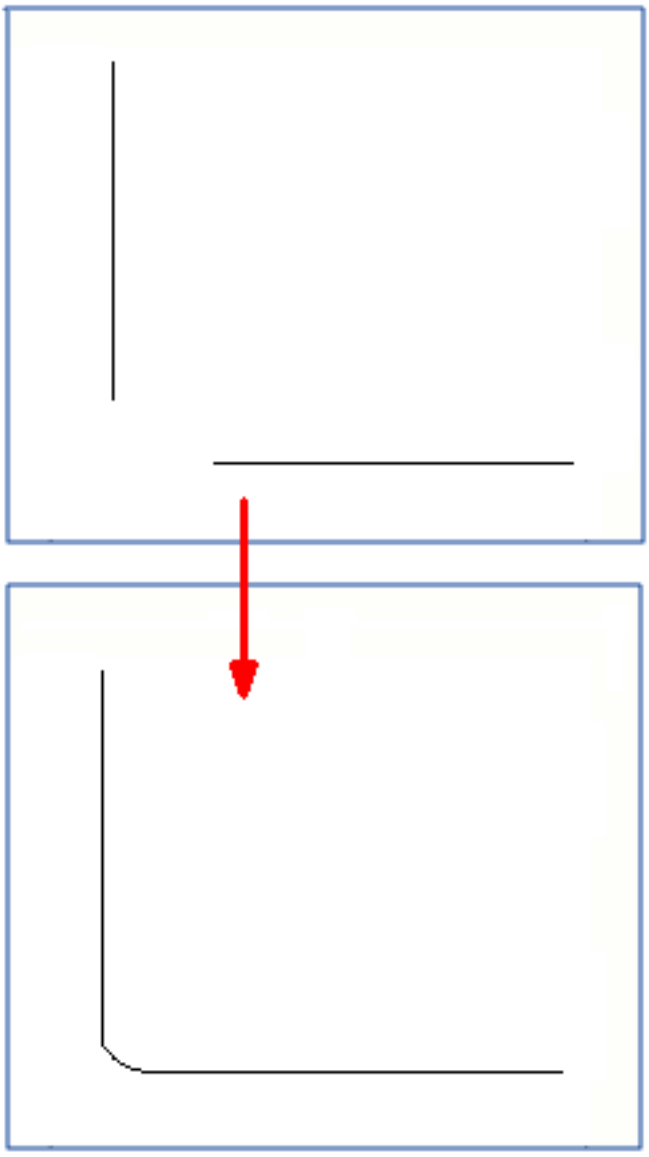
  Enter the desired radius in the displayed dialog box.

Example

Example of a straight chamfer:



Example of a round chamfer:



4.13 Show or hide drawing objects

You can hide selected drawing objects in either drawings or drawing views. You can also list which objects you have hidden. You can also hide dimensions of graphical drawing objects.

Hide objects in drawings and drawing views

You can hide selected objects in drawings and drawing views, for example, parts, marks, dimensions, or grids. When you hide a part, Tekla Structures will

hide all its related objects. Note that if an object is hidden in drawing views, it will not be printed.

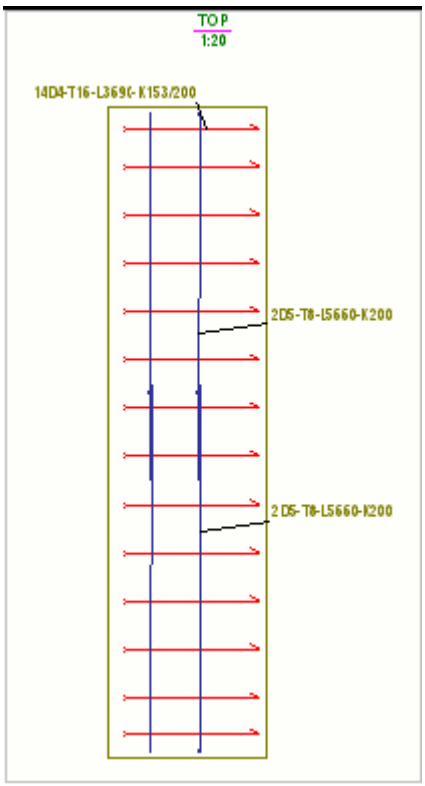
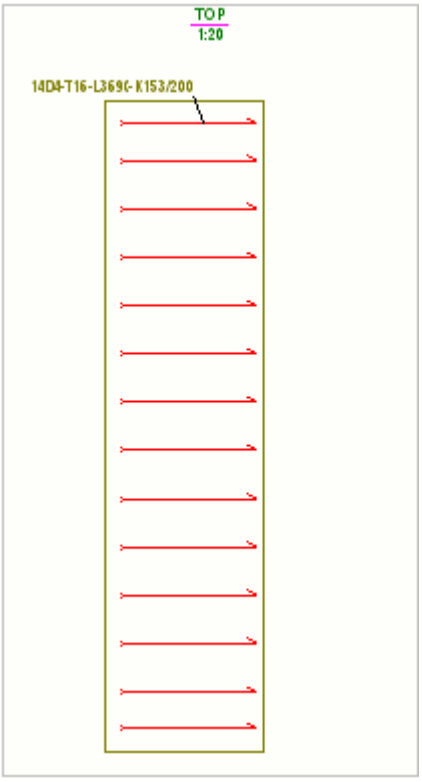
1. Open a drawing.
2. Check that the ghost outline option is selected in **File** --> **Settings** --> **Ghost outline** .
3. Select the **Color** mode by pressing **B**.

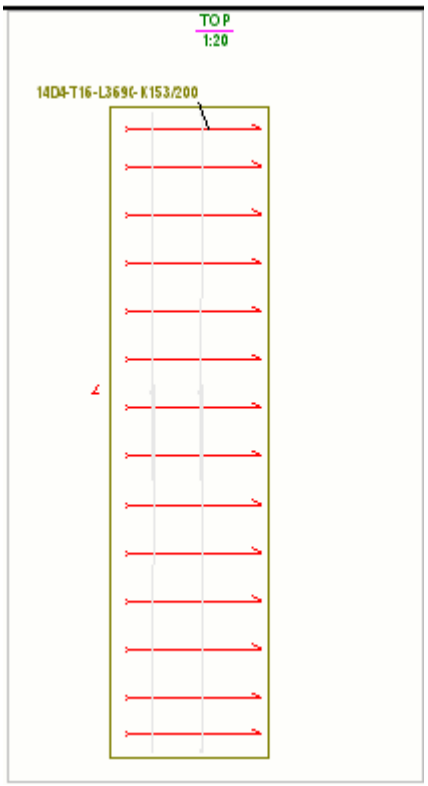
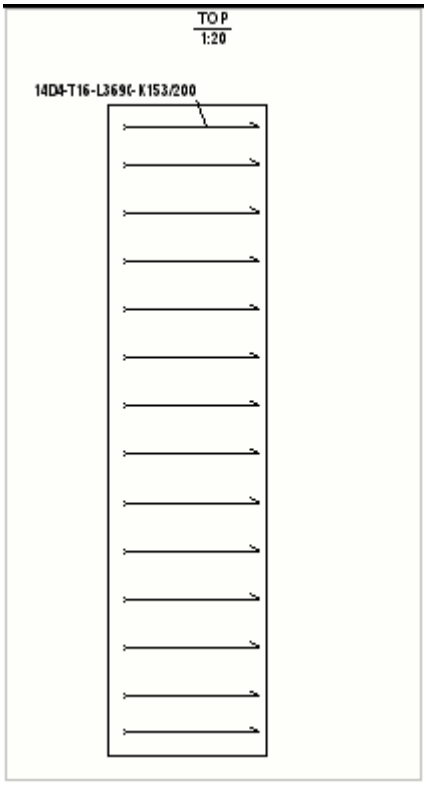
   Hidden objects are shown as ghost outlines in color drawings. In grayscale and black and white drawings, hidden objects are not shown even if **Ghost outline** is selected.
4. Select the objects you want to hide in one of the following ways:
   - Activate the appropriate selection switch and select the objects using area selection, for example. For example, to select parts, use **Select parts in drawings**.
   - Open the **Drawing content manager** from the side pane, and click **Show** to populate the **Drawing content manager** list. You can also select areas, single or multiple views, or single or multiple building objects to narrow down your selection. To select parts, ensure that building object selection is active , and to select marks, ensure that mark selection is active . Then select the desired building objects from the list.
5. On the **Drawing** tab, click **Hide/show** and one of the following commands:
   - **Hide from view**: Tekla Structures hides the selected object in the view where the selected objects are located.
   - **Hide from drawing**: Tekla Structures hides the selected object in all drawing views.

   You can also right-click and select the commands from the pop-up menu.
6. When you want to show the hidden objects again, go to the **Drawing** tab and click **Hide/show** and select **Show in view** or **Show in drawing**.

Example

Below there are some examples of the results with different combinations of selections.

Setting	Example
Color mode is set to **Color** in **File** --> **Settings** . No objects have been hidden. Marks are shown.	
Color mode is set to **Color** and **Ghost outline** is **not** selected in **File** --> **Settings** . Selected reinforcing bars are hidden and related marks are not shown.	

Setting	Example
Color mode is set to **Color** and **Ghost outline** is selected in **File** --> **Settings** Hidden rebars are shown as ghost outlines and related part marks are not shown.	
Color mode is set to **Black and white** and **Ghost outline** is selected in **File** --> **Settings** . The ghost outline has no effect on black and white drawings. Rebars are hidden and related marks are not shown.	

List hidden parts in drawings

You can select whether to list the hidden parts in drawings, for example, in the material list.

Note that listing hidden objects does not work in all templates. It works in templates containing PART rows, but not in hierarchical templates. For example, if the template is of type ASSEMBLY - PART, and the Assembly is included in the drawing, all of its parts will also be included.

1. Open a drawing containing hidden parts.
2. Double-click the drawing to open the drawing properties dialog box.
3. Click **Layout...**
4. Select whether you want to list the hidden parts in **List hidden objects in templates**. **No** removes all information about the hidden parts, also from the total weight.
5. Click **Modify**.

   In the example below, the hidden reinforcing bars are listed.

CAST UNIT BILL OF MATERIAL

Cast unit	Quantity	Main part material	Weight (kg)	Volume (m³)
CF/3	8	K30-2	6879.6	2.87
FOOTING	1	K30-2	6879.6	2.87

Reinforcement:

Type	Pos	Quantity	Grade	Diam	L	a	b	c	d	e	u	v	D	kg/one	kg/all
D	5	12	Undefined	12	3190	600	2050	600					60	2.8	34.0
												Reinforcement total weight (kg):			34.0
												CAST UNIT TOTAL WEIGHT (kg):			6903.4

   In the example below, the hidden reinforcing bars are not listed.

CAST UNIT BILL OF MATERIAL

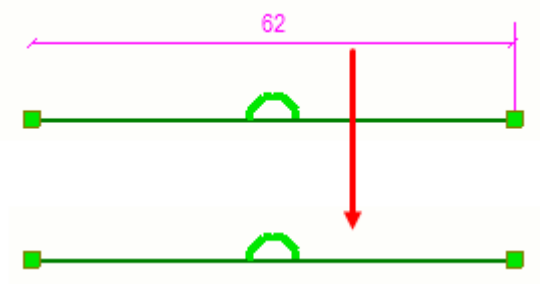
Cast unit	Quantity	Main part material	Weight (kg)	Volume (m³)	
CF/3	8	K30-2	6879.6	2.87	
FOOTING	1	K30-2	6879.6	2.87	
				CAST UNIT TOTAL WEIGHT (kg):	6879.6

Hide or show dimensions of drawing sketch objects

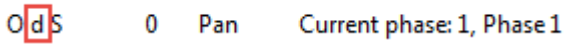
You can hide and show the dimensions of the sketch objects in drawings, such as lines, or arcs, from **Quick Launch** or by using a keyboard shortcut. By default, the dimensions are visible.

In an open drawing where you want to hide or show sketch object dimensions, do one of the following:

- Press **Ctrl+d** to hide the dimensions. To show the dimensions, press **Ctrl+d** again. For more information about shortcuts, see Customize keyboard shortcuts.
- Go to **Quick Launch**, start typing `toggle graphical object dimensions`, and select the **Toggle graphical object dimensions** command from the list that appears. To show the dimensions again, do the same.



When the dimensions are visible, a small **d** is displayed on the status bar:



See also

Hide part face areas and outlines with cover-up tools (page 413)

Modify grids and grid lines in drawings (page 514)

4.14 Hide part face areas and outlines with cover-up tools

Use cover-up line, rectangle, polyline or polygon tool for quickly hiding areas of building object faces or outlines in drawings. Cover-up objects are also shown in printouts.

1. Open a drawing.
2. On the **Drawing** tab, click one of the cover-up tools:

   Draw cover-up rectangle

   Draw cover-up line

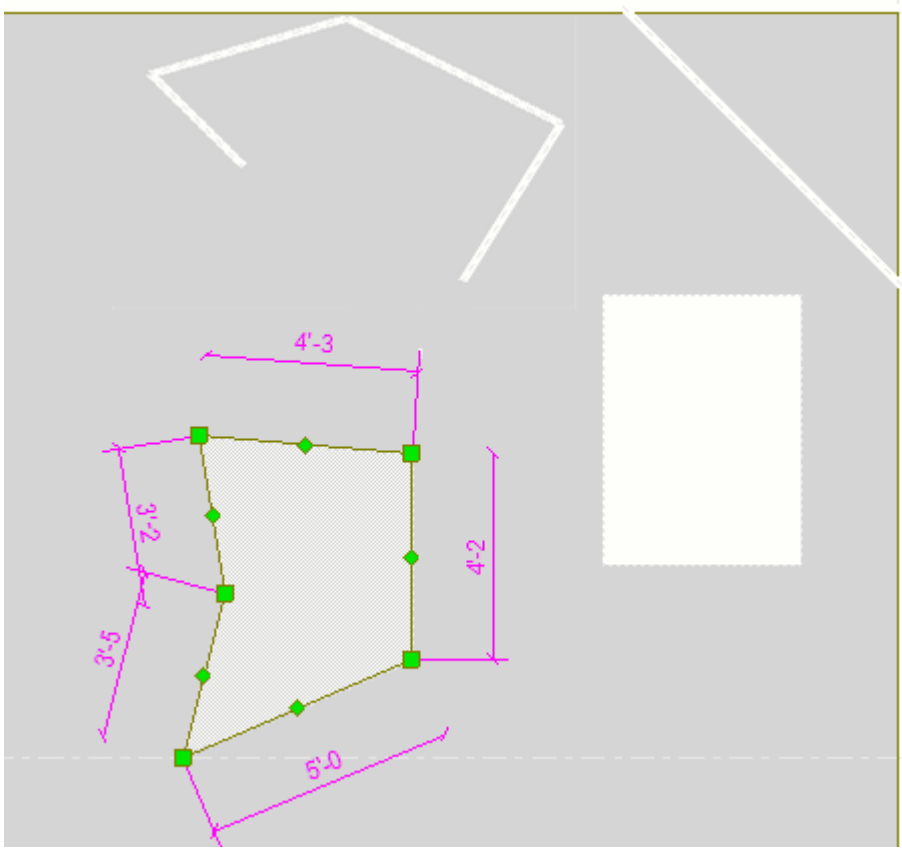
Draw cover-up polyline

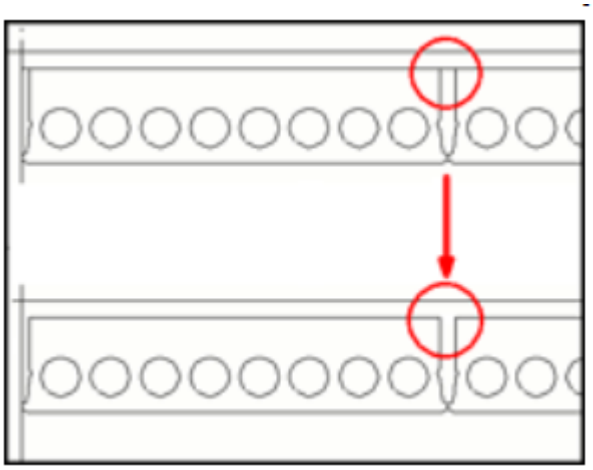
Draw cover-up polygon

3. Pick points and draw a non-transparent area or line over the area that you want to hide.

   The cover-up object dimensions are shown while you draw it, and also when the object is selected. You can do the following with cover-up objects:

   - Drag cover-up objects to another location.
   - Resize cover-up objects by dragging the handles.
   - Reshape polyline and polygon objects by dragging the handles.





Limitation: The **Re-order** command in the pop-up menu has no effect on the cover-up object. To have a cover-up area hiding sketch objects, such as polygons and polylines, use **Re-order** for the sketch object and send it behind model object.

See also

Add sketch objects in drawings (page 387)

4.15 Delete drawing objects

You can delete drawings, and some of the drawing objects inside the drawings.

NOTE You cannot delete building objects in a drawing, such as parts, bolts, welds, chamfers, reinforcing bars, or surface treatment; all changes to building objects are made in the model.

For more information about various drawing object types, see Drawing objects, views, and layouts (page 68). For details about building objects in drawings, see Show building objects in drawings (page 426).

- You can **delete** the following, for example:
  - **Annotations**: To delete dimensions, marks, notes, and other annotation objects (text objects, sketch objects, .rtf files, symbols, links, hyperlinks, images, DWG/DXF files), select the object and press **Delete** on the keyboard, or right-click and select **Delete**. You can also delete marks and notes in Drawing content manager (page 290), where you can narrow your selection by selecting certain drawing views, areas or multiple objects and then delete those. You can also delete marks for the parts you specify by using selection filters (page 337).
  - **Views**: To delete drawing views: Select the view and press **Delete** on the keyboard, or right-click and select **Delete**.

- **Tables in layout**: You can also delete tables in the drawing layout, such as a material list or revision table. For detailed instructions, see Change table sets in drawings (page 176).
- **Change symbols**: To delete the change symbols (change clouds) for dimensions, marks, and notes, see instructions in Check changed marks, notes and dimensions and remove change symbols (page 351)
- **Drawings**: You can also delete unnecessary drawings through **Document manager**. For more information, see Delete drawings and file documents (page 590).

- You can also **hide objects** from drawing views or from drawings, where the objects remain in the drawing but are hidden. For detailed instructions, see Show and hide drawing objects (page 408). The selected object and all its related objects are hidden, and shown as ghost outlines in color drawings.

  Note that if an object is hidden in drawing views, the object will not be printed.
- You might also want to **change the visibility** of some of the objects so that they are not visible in the drawing. Take a look at the drawing and view level properties for various objects, such as parts (page 1004), surface treatment (page 1012), reinforcement (page 1016), model welds (page 1036), and reference objects (page 526), and check the available visibility settings.

See also

Drawings in Tekla Structures (page 46)

4.16 Arrange annotation objects

You can reposition annotation objects such as marks, dimensions or texts on the basis of the protection and placing properties of each drawing object type.

Before arranging annotation objects, check and change the **Protection** settings in view properties. With protection properties you can define protected areas in drawings to prevent text, marks or dimensions from being placed in that area. Also, double-click annotation objects in the drawing, and check the placing settings, and change those as required. Placing properties define where and how far the annotation objects are placed, and the empty margin around the mark. If the object is set to fixed, the **Arrange objects** commands will reposition the objects and keep the fixed state.

1. In the open drawing, select the annotation objects you want to arrange:
   - Select the individual objects such as marks, dimensions and texts by pressing down **Shift** and clicking the objects.
   - Use selection switches to select only texts, marks, and associative notes, for example, and use area selection to select the objects that you want.

- To select marks or associative notes only, first activate the **Select parts in drawings** selection switch and select the parts using area selection, for example, and then right-click and select **Select part marks**.
- Another way to select marks or associative notes only is to open the **Drawing content manager** from the side pane, click **Show** to populate the **Drawing content manager** list, ensure that marks selection is active , and select the desired building objects from the list.

2. On the **Annotations** tab:
   - To arrange marks and texts near current location, click **Arrange objects** --> **Near current location** .

     Tekla Structures positions the selected drawing objects so that they do not overlap other objects. Objects located in a free location are not moved, and overlapping objects are moved as close to the current location as possible.
   - To arrange marks and texts ignoring the current location, click **Arrange objects** --> **Ignore current locations** .

     Tekla Structures positions the selected drawing objects so that they do not overlap with other objects without checking the current location of the objects.
   - To arrange the selected marks and notes so that you avoid crossing leader lines, click **Arrange marks** on the **Annotations** tab. Remember that you first have to select the marks or notes.

See also

Define object protection and placement settings in drawings (page 677)

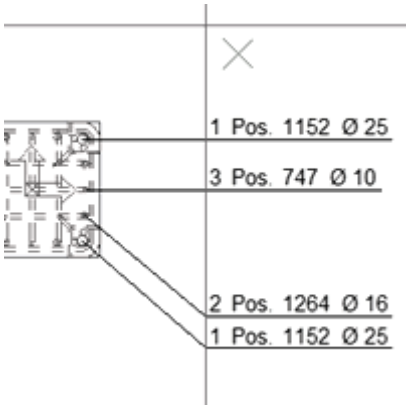
4.17 Align selected drawing objects

You can align objects (texts, marks, notes and views) above, below, to the left, to the horizontal center, to the vertical center and to the right of the point you pick. You can also position objects horizontally or vertically at equal distances from each other.

In an open drawing, do any of the following:

To	Do this
Align selected views, texts, marks, and notes	1. Select the objects that you want to align. You can select one or several objects using appropriate selection switches or filters and area selection, for example. If you want to

To	Do this
	align marks and notes only, you can use **Drawing content manager** for first selecting the building objects and then the related marks or notes. For more information about **Drawing content manager**, see Drawing content manager (page 290). 2. Select the aligning command from the toolbar: Click to align objects above the point you pick. Click to align objects below the point you pick. Click to align objects to the right of the point you pick. Click to align objects to the left of the point you pick. Click to align objects to the vertical center of the point you pick. Click to align objects to the horizontal center of the point you pick. Click to position objects to the vertical center of the point you pick at equal distances. Click to position objects to the horizontal center of the point you pick at equal distances. 3. Pick a location for the alignment. If you have mixed objects selected, such as views and text objects, you will be asked to select which objects you want to align. 4. If you selected one of the two commands that position objects at equal distances, type the

To	Do this
	distance in the displayed dialog box and press **Enter**. **TIP** To create a row of objects, align them first to the top and then position them horizontally at equal distances from each other. You do not need to reselect the objects between the two commands.
Align selected marks and notes automatically	• Click **Align marks** --> **Auto-align marks** on the **Annotations** tab. This aligns the selected marks and notes by stacking the marks or notes and placing them evenly. This command applies to all marks and associative notes, except weld marks.
Align selected marks and notes around a point	• Click **Align marks** --> **Align to a point** on the **Annotations** tab and pick a point. This command tries to avoid crossing leader lines when placing the marks.  This command applies to all marks and associative notes, except weld marks.

4.18 Drag, reshape and resize drawing objects

Many drawing objects, dimension lines and leader lines of many drawing objects have handles. You can use these handles to drag, reshape, and resize objects. If you have activated the **Drawing drag & drop** switch, you do not

need to select the handles or objects separately to drag, reshape or resize the objects.

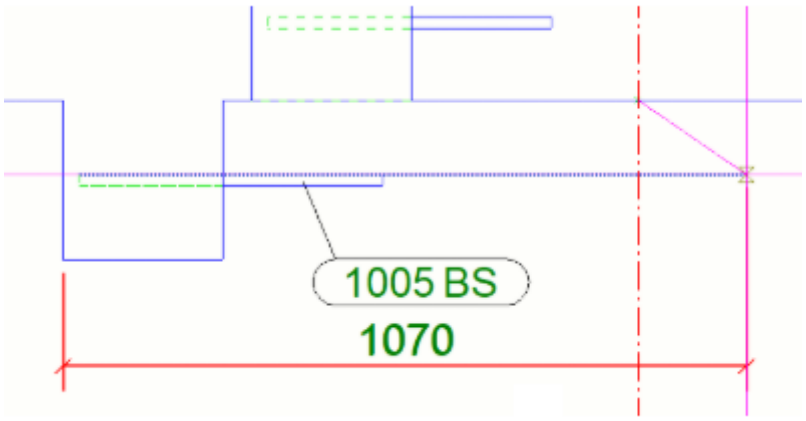
Drag, resize or reshape annotation objects, sketch objects and grid labels

- In an open drawing, go to the **File** menu, click **Settings** and select the **Drawing drag & drop** check box. After this, you can move, drag, resize or reshape drawing objects such as annotations, sketch objects and grid labels without selecting the objects or handles first.
- To move the handle points of sketch objects to resize or reshape, press down the left-mouse button at the handle and drag the handle to the new position.
- To drag the handles of grid labels, you need to select the grid line first.

  Ensure that you have activated the **Select grid line** selection switch .
- You can also drag the associativity point of the leader line (page 364) of the annotations objects.

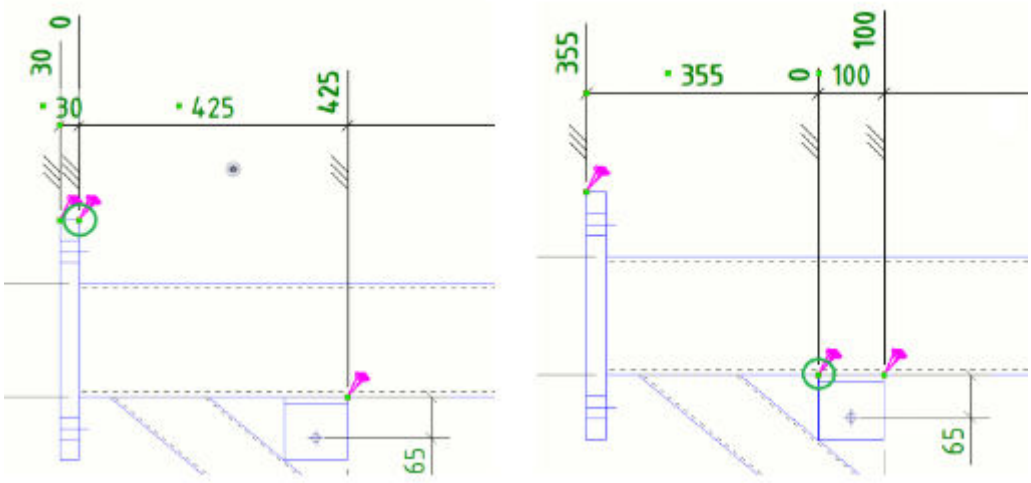
Drag dimensions

- Activate the **Drawing drag & drop** switch to drag without selecting the handle first.
- Dimension marks can be freely dragged (page 287) to avoid overlapping dimensions and marks.
- You can only drag relative dimensions, not absolute dimensions.
- Straight (linear) dimensions in drawings can be modified by dragging the dimensions from dimension handles.



- Dragging in single dimension lines: If you try to drag a dimension point to a new position that is located behind an existing dimension point, the selected dimension point will be deleted from the old position and recreated in the new, preferred position. If you try to drag the point to the same coordinate position where the existing point is already located, the dragged dimension point is dropped back automatically to the initial position.

- Dragging in dimension sets: If you try to drag the point into the area between two points of neighbor dimension line, the selected dimension point will be deleted from the old position and recreated in the new, preferred position. If you try to drag the point to the same coordinate position where the existing point is already located, the dragged dimension point will be deleted from the old location and will be combined with the existing one, after dropping to the new location.
- Dragging a dimension point with a zero coordinate in running dimensions allows you to change the coordinate of the start dimension point without running the **Set dimension start point** command.



NOTE When you drag a mark, note, text, or dimension, its placing setting may be set to **fixed** depending what its set for the advanced options listed below. The mark, note, text or dimension stays where it is, even though you update the drawing. You may want to check the settings defined for the following advanced options:

XS_CHANGE_DRAGGED_DIMENSIONS_TO_FIXED

XS_CHANGE_DRAGGED_MARKS_TO_FIXED

XS_CHANGE_DRAGGED_NOTES_TO_FIXED

XS_CHANGE_DRAGGED_TEXTS_TO_FIXED

See also

Define object protection and placement settings in drawings (page 677)

Adjust part mark leader lines with advanced options (page 821)

Place reinforcement mark leader line associativity point automatically (page 821)

Drag leader line associativity point (page 364)

Modify, arrange and align drawing views (page 200)

Show welds in drawings (page 489)

4.19 Clone selected in drawings

The **Clone selected** command allows you to clone previously created annotation objects and drawing object representations among the assemblies or cast units with the same type and similar shape in GA drawings.

When editing GA drawings, it is often required to add annotations, dimensioning and styles for building objects as repetitive task. The new **Clone selected** feature clones existing annotation objects,drawing object representations and styles from selected source objects to selected target objects in GA drawings. With this feature, you can reduce manual repetition considerably.

You can clone annotation objects and drawing object representations inside one drawing view or among different drawing views.

Clone selected recognizes the following types of drawing content:

- Associative and independent annotation objects: dimensions, marks, texts, symbols, text files, and DWG/DXF files
- Sketch objects, such as circles, rectangles, and polygons
- Object representations and styles: line colors, line types, hatches

Adjust cloning settings

Before you clone, you may want to define how and what to clone in **Cloning settings**. You can also clone using the default settings.

1. On the **Drawing** tab, click **Clone selected** --> **Cloning settings** .
2. Define the cloning settings:

Merge	Override all annotation objects and building object properties in the cloning target by cloning matching content from the cloning source. This is the default mode.
Keep all	Keep the existing annotation objects and building object properties in the cloning target and only clone the missing ones from the cloning source.
Discard all	Discard all annotation objects and building object properties in the cloning target and clone all selected content from the cloning source.
Clone building object properties and styles	Clone drawing object properties such as line colors and types, hatching styles and other

	representation properties in the drawing object properties for parts, reinforcement, bolts and welds.
Place marks and associative notes according to protection settings	Automatically run the **Arrange drawing objects near current location** command for all associative annotation objects except for level marks, weld marks, dimensions and independent annotation objects. This option positions the annotation objects in the cloning target so that they do not overlap other objects. Objects located in a free location are not moved, and overlapping objects are moved as close to the current location as possible.

3. Click **OK**.

Clone selected annotations and object properties

Before you clone, first modify the source object so that it contains the desired dimensions and annotations, and adjust the source object properties.

When cloning dimensions, remember to set the dimension associativity rules before cloning to avoid the situation where it is unclear which object a dimension point is associated to. For more information, see Display and change dimension point associativity (page 279).

1. In an open GA drawing, click **Clone selected** on the **Drawing** tab.
2. Select the cloning source objects by clicking the objects or using area selection.

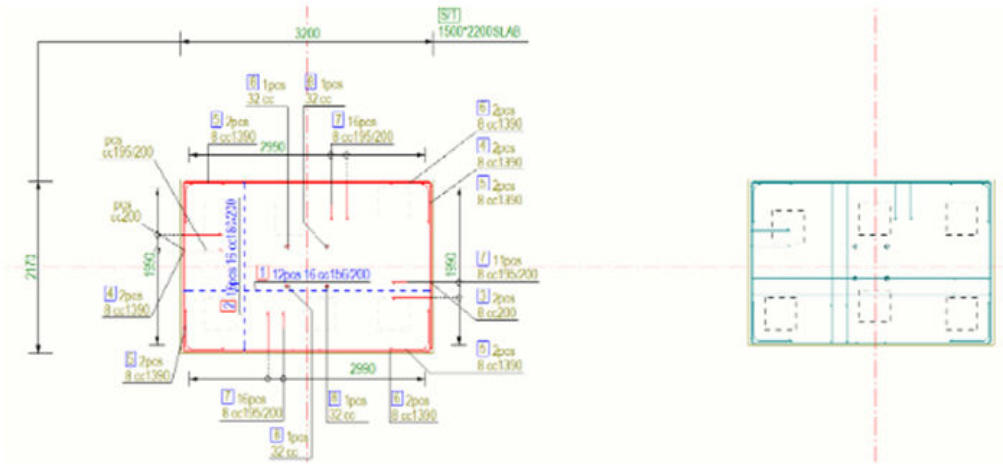
   You can also select the objects before activating the **Clone selected** command.

   You can also select the objects using **Drawing content manager** and selection filters.

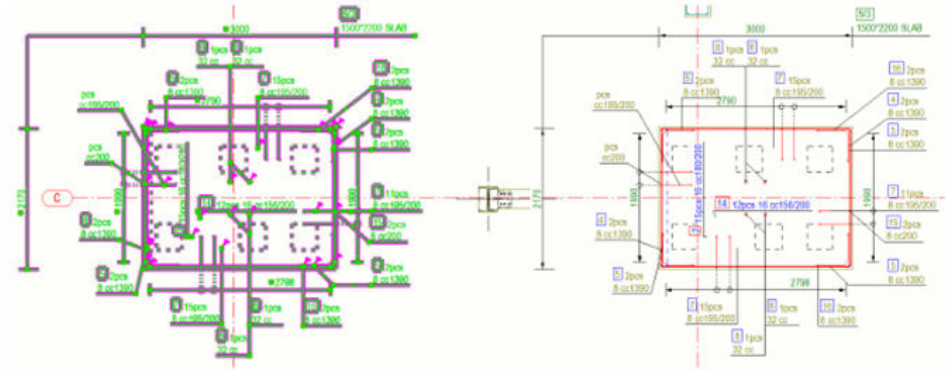
   You can deselect objects by holding down **Ctrl** and clicking the selected object.
3. Press the middle mouse button to complete the selection.
4. Select the cloning target objects by clicking the objects or using area selection.
5. Press **Esc** to stop the cloning.

TIP You can also clone from the source that you selected last. To do this, click **Clone from last selected source** on the **Drawing** tab.

Below on the left a detailed footing and on the right a similar footing that will be the target of cloning:



Below, the detailed footing on the left has been used as source for the target on the right. The details have been cloned to the target.



NOTE Cloning of annotations to multiple assemblies or cast units produces best results only when annotations and dimensions are associated to single-assemblies or cast units in source selection.

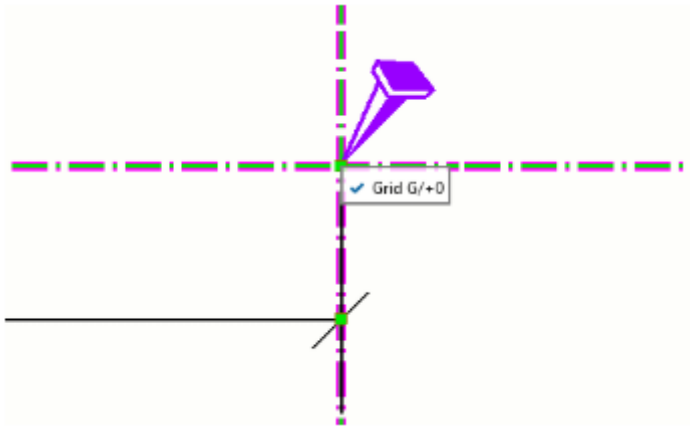
When the source selection contains annotations and dimensions associated to multiple assemblies, Clone selected will produce accurate results only if corresponding groups of assemblies or cast units are selected in the target one by one, not all at once.

Limitations

- Some drawing objects cannot be used as source objects, such as grid lines, section view symbols, drawing view names, neighbor parts and neighbor reinforcement. These objects types will not be highlighted during/after area or single selection. Messages on the status bar indicate if an object cannot be selected.

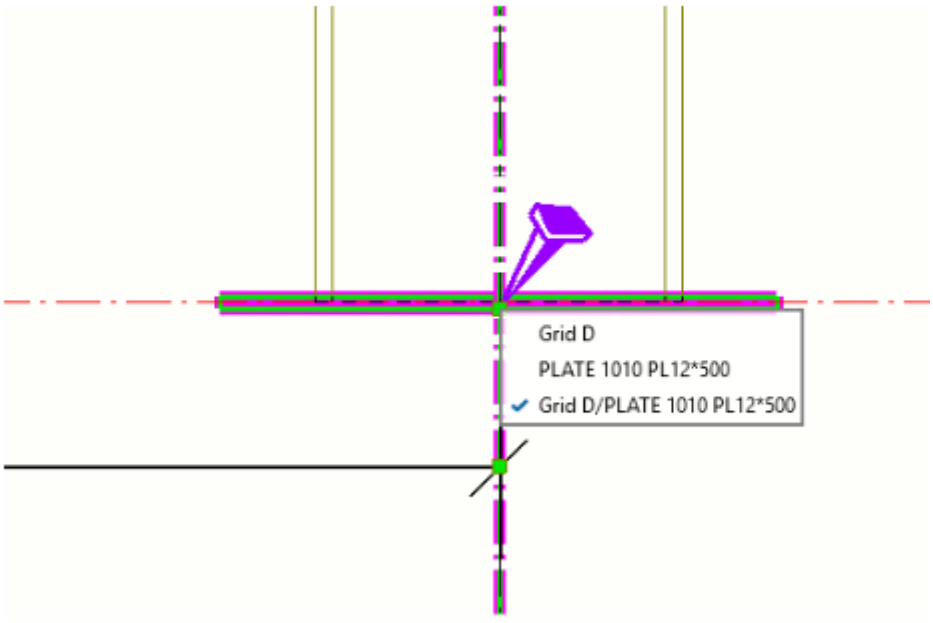
- **Clone selected** cannot be used with pour units.
- To achieve accurate cloning results all dimensions should be associated either to grid line intersection points or to intersections of building objects and grid lines.

  Dimension point associated to intersection of two perpendicular grid lines:



  Note that all dimension points that are located in arbitrary position along the grid lines in the source selection will be cloned to wrong coordinates in the target.

  Dimension point associated to intersection of part side and grid line:



- **Clone selected** cannot be used for cloning radial or curved dimensions.
- Cloning of annotations to mirrored objects created with the **Mirror** command in the model does not produce accurate results.

- Cloning modes do not affect cloning of independent annotation objects or sketch objects. Listed object types will be copied to target as many times as clone selected is applied to the target objects.
- Dimensions will not be cloned if the viewing directions of the source and target drawing views do not match. In this case, a message is shown on the status bar.

4.20 Show building objects in drawings

Building objects are 2D representations of the 3D objects in the model, such as parts, bolts, welds, pour objects, surface treatment, and reinforcement. Building objects in drawings get their appearance from the automatic drawing settings when you create a drawing. After creating the drawing, you can change the appearance in an open drawing.

You can modify building object representation, adjust the visibility settings, and add annotations to the objects. You cannot delete building objects.

Click the following links for more information:

Modify building object properties (page 427)

Shorten parts in selected views (page 427)

Show edge chamfers in drawings (page 429)

Show fillet edges in drawings (page 433)

Show neighbor parts and neighbor reinforcement in general arrangement drawings (page 438)

Hide part face areas and outlines with cover-up tools (page 413)

Show welds in drawings (page 489)

Show pours in drawings (page 485)

Show reinforcement in drawings (page 448)

Show spiral beams in drawings (page 443)

See also

Part and neighbor part properties in drawings (page 1004)

Bolt properties in drawings (page 1010)

Model weld properties in drawings (page 1036)

Surface treatment properties in drawings (page 1012)

Pour object and pour break properties in drawings (page 1029)

Reinforcement object properties in drawings (page 1016)

Modify building object properties

You can modify the drawing properties of the building objects (parts, bolts, reinforcement, surface treatment, welds, pour breaks, pour objects) in an open drawing. For example, you can change the color and how different kinds of lines are shown, or add hatch patterns in parts and cross sections.

Limitations: You can change the color of the part center lines in the drawing and view level part properties. You can change the line type of the part center lines with the advanced option XS_CENTER_LINE_TYPE.

1. Click a building object, for example a part, reinforcement, surface treatment, or bolt. If the property pane is not open, double-click the building object.
2. Change the building object visibility and representation.

   For example, change which lines and markings are shown, and modify the color and type of the lines. You can also add fills to building objects.

   For details about the properties of various building objects, click the links at the bottom of this page.

   Note that the center line is only shown for assembly main parts but not for secondary parts when looking from the cross section direction. If the part is viewed from the side, the center line is also shown for secondary parts.
3. Click **Modify**.

You can also change the part reference line type using the advanced option XS_DRAWING_PART_REFERENCE_LINE_TYPE, and use the advanced XS_DRAWING_POINT_SCALE to scale the points Tekla Structures uses to create reference lines.

See also

Part and neighbor part properties in drawings (page 1004)

Define automatic drawing part properties (page 852)

Bolt properties in drawings (page 1010)

Surface treatment properties in drawings (page 1012)

Reinforcement object properties in drawings (page 1016)

Pour object and pour break properties in drawings (page 1029)

Model weld properties in drawings (page 1036)

Add hatches (fills) to parts in drawings (page 868)

Shorten parts in selected views

You can shorten parts in the selected view in an open drawing. By default, there is an empty area shown in the middle of the shortened part, but you can use a shortening symbol instead, and adjust the shortening symbol color and line type.

1. Click a drawing view frame in an open drawing. If the property pane is not open, double-click the drawing view frame.
2. In the property pane, in **Cut parts**, select **Yes**, **Only in x direction**, or **Only in y direction**.
3. To cut skewed parts, select the **Cut skew parts** check box.
4. In **Minimum cut part length**, set the minimum length of the shortened part.

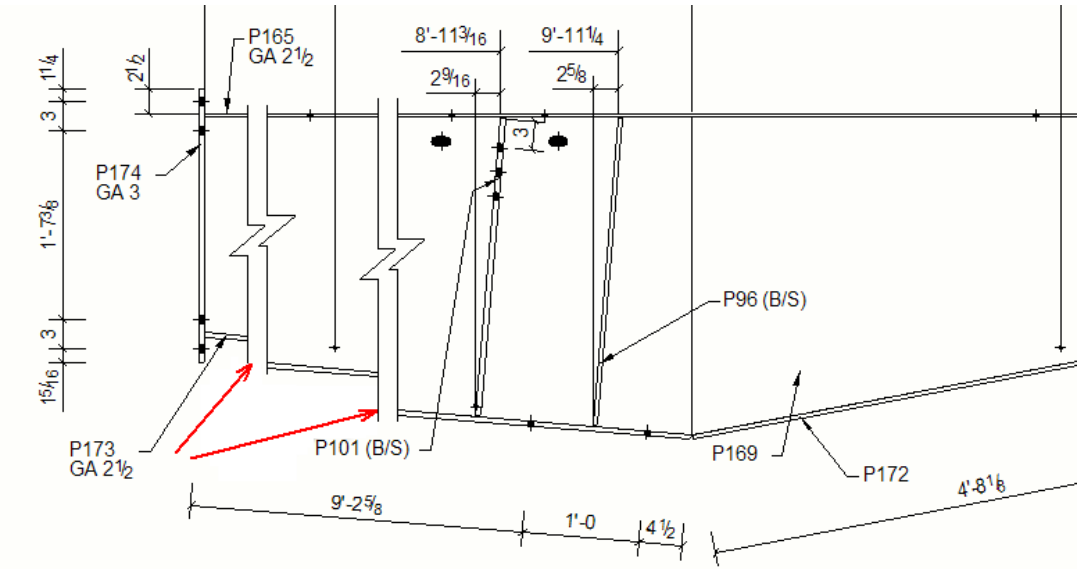
   This option defines how long the part must at least be to get shortened. The length of the part must be at least twice the entered value.
5. In **Space between cut parts**, enter the distance between the cut parts on paper.
6. Click **Modify**.

TIP To use a view shortening symbol instead of the empty area in the middle of the shortened part, set the advanced options XS_DRAW_VERTICAL_VIEW_SHORTENING_SYMBOLS_TO_PARTS and XS_DRAW_HORIZONTAL_VIEW_SHORTENING_SYMBOLS_TO_PARTS to `TRUE`.

You can control the appearance of the view shortening symbol with the advanced options XS_SHORTENING_SYMBOL_COLOR, XS_SHORTENING_SYMBOL_LINE_TYPE, and XS_SHORTENING_SYMBOL_WITH_ZIGZAG.

Example

Below is an example where the zig zag shortening symbol is used.



See also

Shorten or lengthen parts (page 708)

View properties in drawings (page 912)

Show edge chamfers in drawings

You can show edge chamfers in drawings, and control the way they are shown by modifying the part properties and the edge chamfer properties. You can also associative notes to edge chamfers.

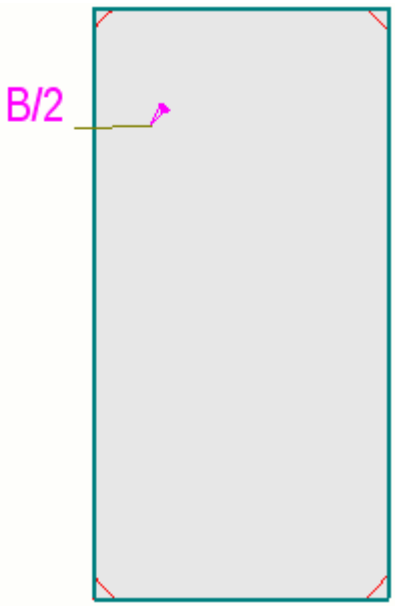
Show edge chamfers in a drawing

You can select whether to show edge chamfers in your drawing, and the way they are shown.

1. Open a drawing of a part where you have edge chamfers, and double-click the drawing background to open the drawing properties.
2. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
3. In the **Additional marks** area in part properties, select the **Edge chamfers on/off** check box.
4. In the **Representation** list, depending on the desired result, select **Outline** or **Exact**.

Exact

Outline



5. Save the view properties and click **Close**.
6. Click **Modify**.
7. If needed, double-click the edge chamfer in the drawing and modify the line color and type in edge chamfer properties.

Define default line color and type for edge chamfers

You can define a default color and line type for edge chamfers in drawings.

1. On the **File** menu, click **Settings** --> **Options** and go to the **Drawing objects** settings.
2. Define the default line color for the edge chamfer.
3. Define the default line type for the edge chamfer.
4. To save the default settings and close the dialog box, click **OK**.

TIP You can override the default settings manually in a drawing by double-clicking the edge chamfer and changing the edge chamfer line color and type in edge chamfer properties.

Change edge chamfer line color and type manually

You can change the line type and color of the edge chamfers manually in an open drawing. This overrides the default color and type defined in the **Options** dialog box.

1. To display the edge chamfer properties, double-click an edge chamfer in an open drawing.

2. Select the desired color and line type.
3. Click **Modify**.

   The line color and type changes.

   Background color is often used for edge chamfer lines for the reason that you may not want to print edge chamfers, or see them in small scale drawings, but you want to be able see and select them, for example, to add associative notes.

Example

The following examples show how the edge chamfers are displayed with different part representation settings:

Part representation **Exact**.

Part representation **Outline**, edge chamfer not selected.

Part representation **Outline**, edge chamfer selected.

Add associative notes to edge chamfers

You can add associative notes to edge chamfers.

1. Open a drawing of a part that has edge chamfers.
2. Double-click the parts and edge chamfers and modify the part properties and edge chamfer properties so that edge chamfers are visible and you can easily select them.
3. To open associative notes properties, hold down **Shift** and on the **Annotations** tab in the drawing, click one of the commands in the **Note** drop down. You can also select **Note** in the property pane object list.
4. Select **Edge chamfer** in the **Object type** list.
5. In the associative note editor, click the plus button to open the element list and select the elements to be added in the note. You can delete the unnecessary elements from the note by clicking the red delete button
6. Modify other properties as required.

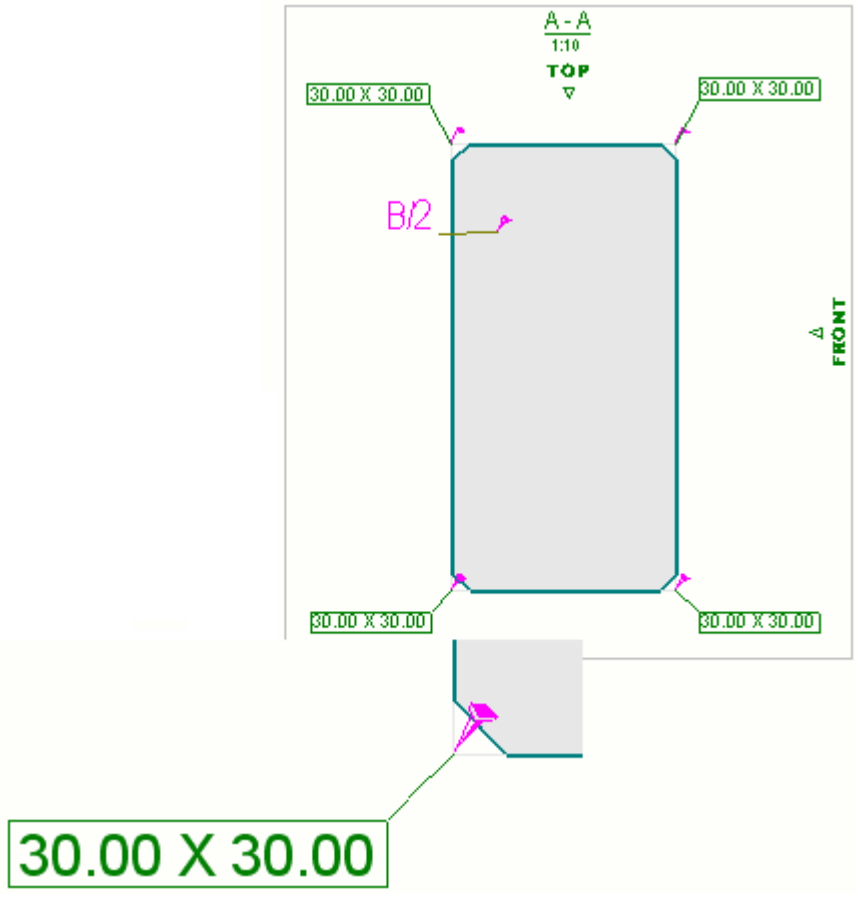
7. To place the note, click the edge chamfer in the drawing.

   If you use a leader line, you need to pick a position for the note.

Example: Edge chamfers

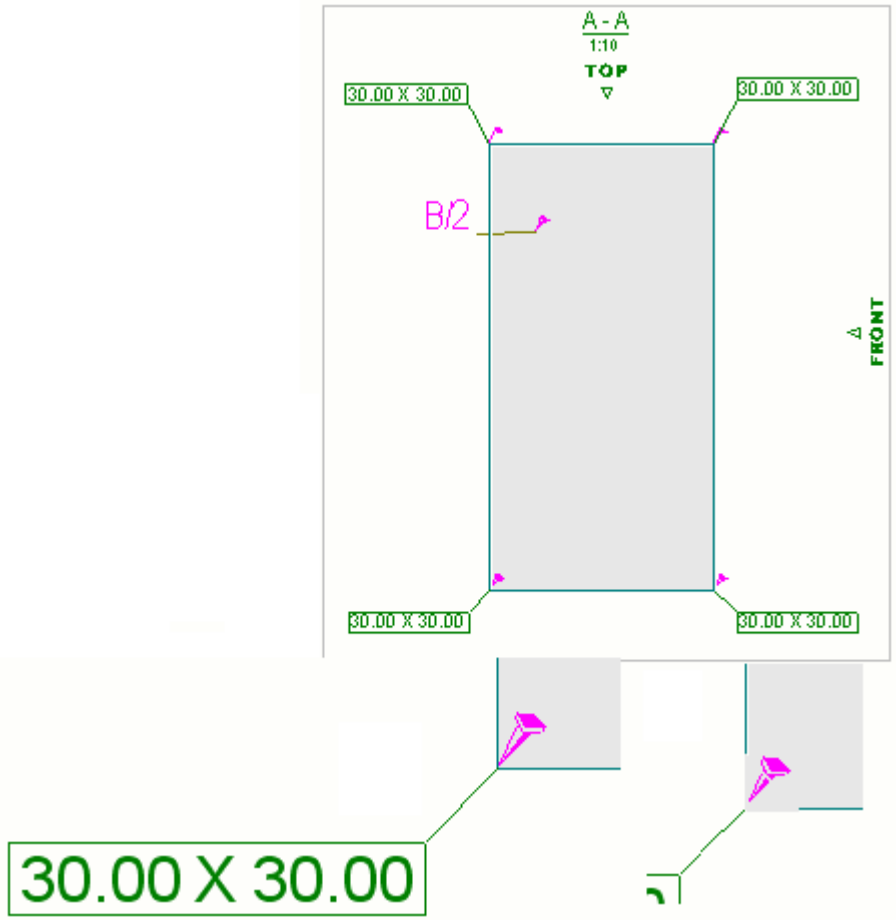
See below for typical examples of the ways you can show edge chamfers in drawings.

In the first example, the part **Representation** has been set to **Exact** and **Edge chamfers** have been set visible. Background color is used for edge chamfer lines, because you usually do not want to show edge chamfers in printouts, but may want to see and select them in the drawing, for example, to add associative notes.



In the second example, **Representation** has been set to **Outline** and **Edge chamfers** have been set visible. Background color is used for edge chamfer lines, because you may want to see and select chamfers in the drawing, for example, to add associative notes. This representation type is often used when

the scale is small and you do not need to see the small chamfers clearly. The edge chamfer presented in the bottom right corner of the image shows what the edge chamfer looks like when it is selected.



Show fillet edges in drawings

Fillet edges are lines that define the boundary between straight faces and curved faces in the model. Some examples of building objects containing fillet edges are profiles with curved fillets, slabs or contour plates with curved chamfers, and curved polybeams. The fillet edges can be shown in all drawing types for parts and pours. Fillet edges are visible by default for new drawings, and invisible for drawings that are created in a Tekla Structures version earlier than 2016.

Show fillet edges in GA drawings

To show the fillet edges, you need to set **Fillet edges** visible in part properties. For profiles, the **Exact** representation might be required to show the fillet

edges, depending on the profile. You can control the fillet edge visibility in part properties and in pour object properties on the drawing, view and object level.

To set the fillet edges visible on the drawing level in a general arrangement drawing:

1. Open a general arrangement drawing that contains steel or concrete parts with curved faces, for example, profiles with curved fillets, slabs or contour plates with curved chamfers, or curved polybeams.
2. Double-click the drawing background to display the **General arrangement drawing properties** dialog box.
3. Click **Part** or **Pour object.**
4. If you need to change part representation (page 1004) for profiles, select **Exact** from the **Representation** list.
5. Under **Additional marks**, select the **Fillet edges on/off** check box.
6. If needed, set the hidden lines options:
   - Select the **Hidden lines on/off** check box to show the hidden lines of other parts.
   - Select the **Own hidden lines on/off** check box to show the part's own hidden lines.
7. Click **Modify**. Fillet edges are now visible in the drawing.

Show fillet edges in individual parts

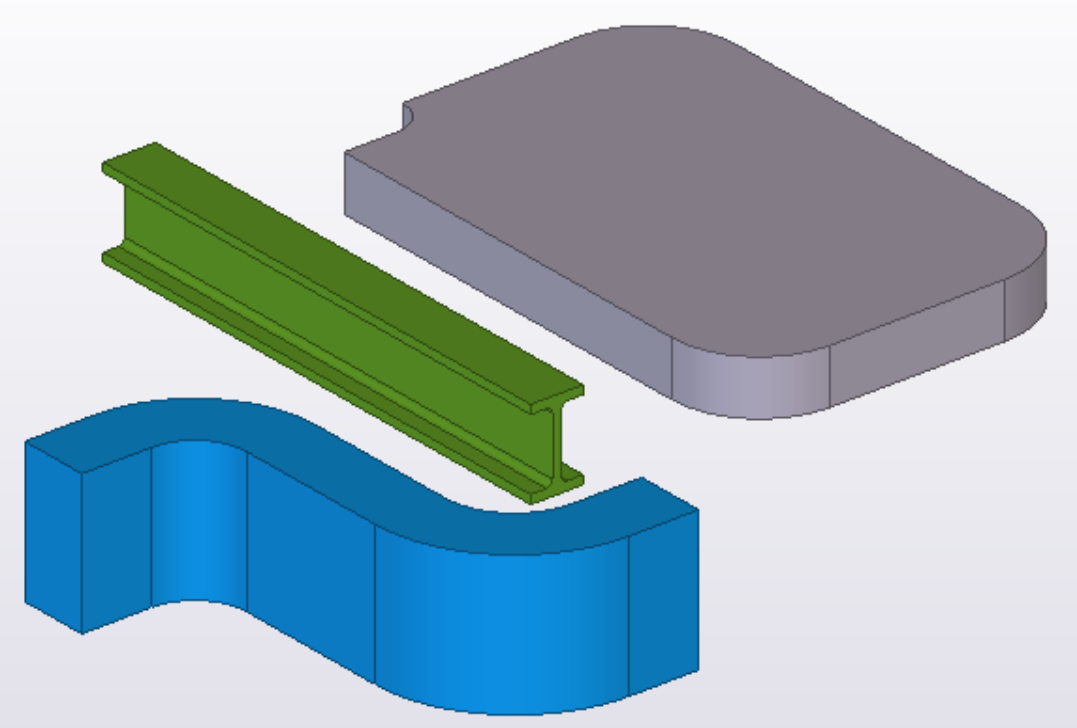
You can also modify individual parts in drawings so that the fillet edges are shown.

1. Open a drawing that contains steel or concrete parts with curved faces, for example, profiles with curved fillets, slabs or contour plates with curved chamfers, or curved polybeams.
2. To open the properties, click the desired part. If the property pane is not open, double-click the part.
3. In the **Visibility** section, activate the **Fillet edge** option .
4. If needed, set the hidden lines options:
   - Activate the **Hidden lines** option to show the hidden lines of other parts.
   - Activate the **Own hidden lines** option to show the part's own hidden lines.
5. Ensure that **Exact** is selected in the **Representation type** list.
6. Click **Modify**.

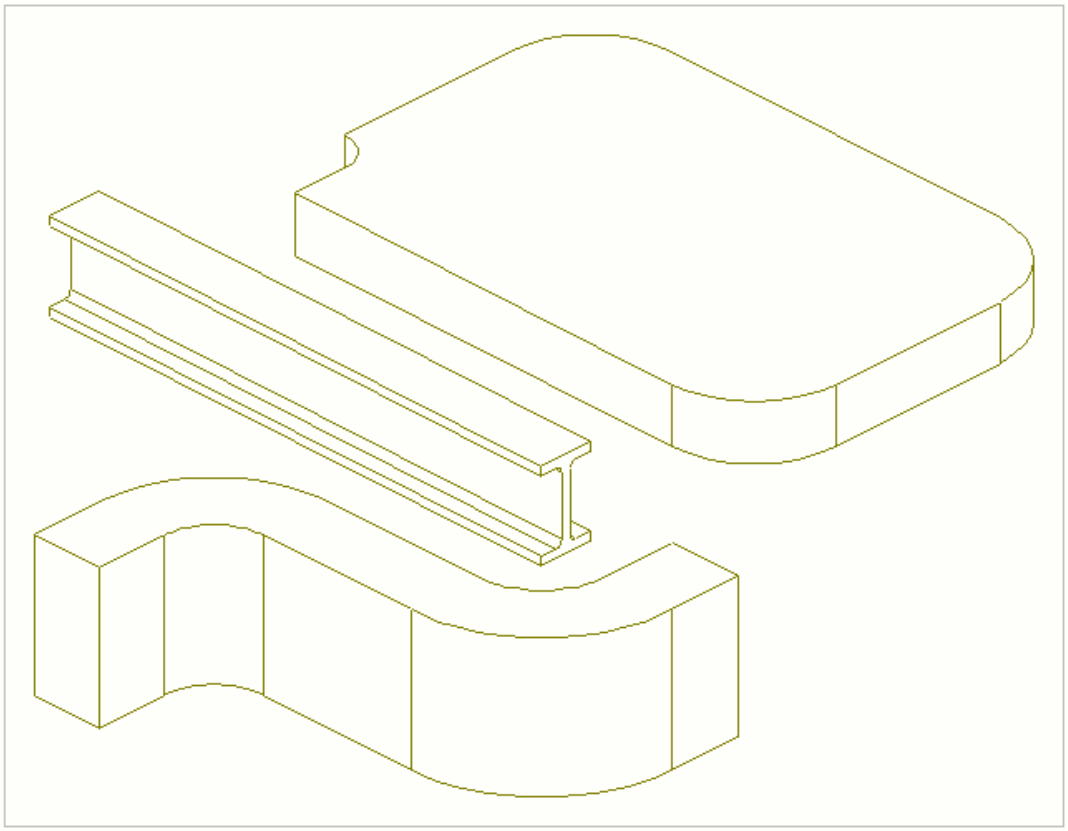
Examples

Example 1

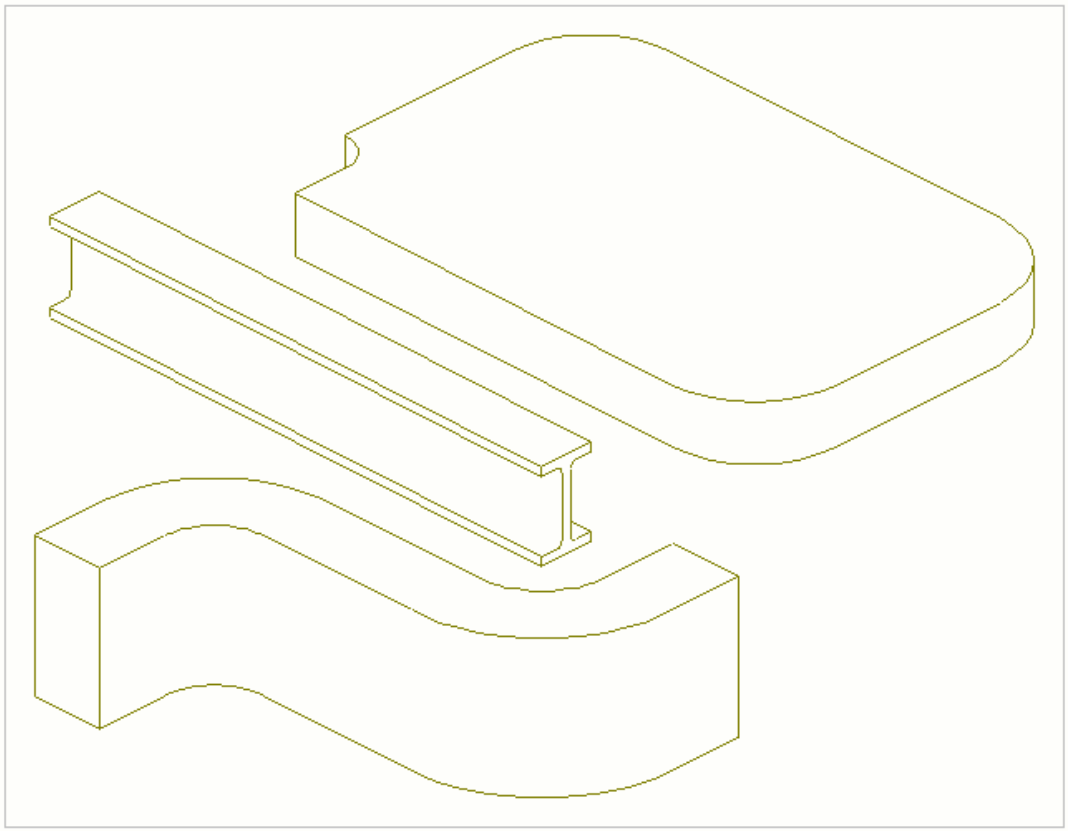
Below is an example of fillet edges in the model:



Below is an example of parts in a drawing showing the fillet edges (**Fillet edges on/off** selected in the drawing level part properties or **Fillet edge** selected in part properties in the property pane):

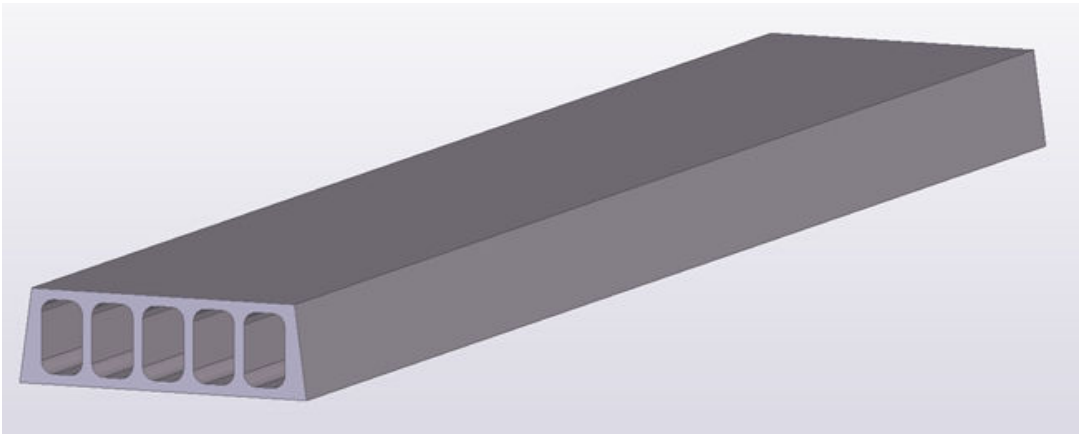


Below is an example of parts in a drawing not showing the fillet edges (**Fillet edges on/off** not selected in the drawing level part properties or **Fillet edge** set in part properties in the property pane):

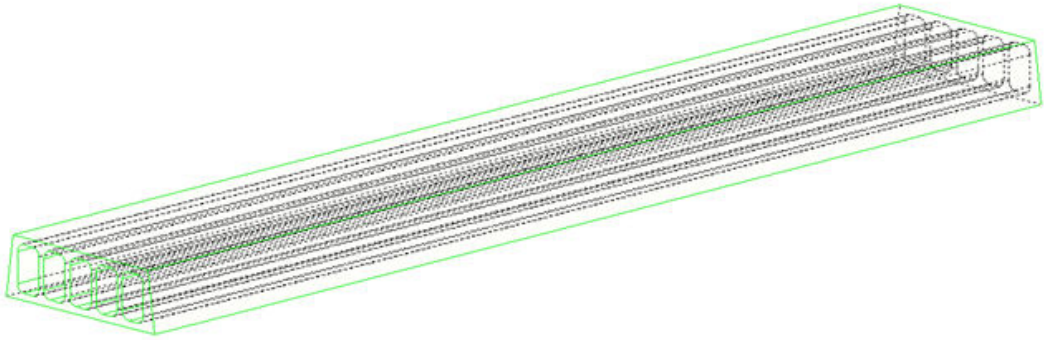


Example 2

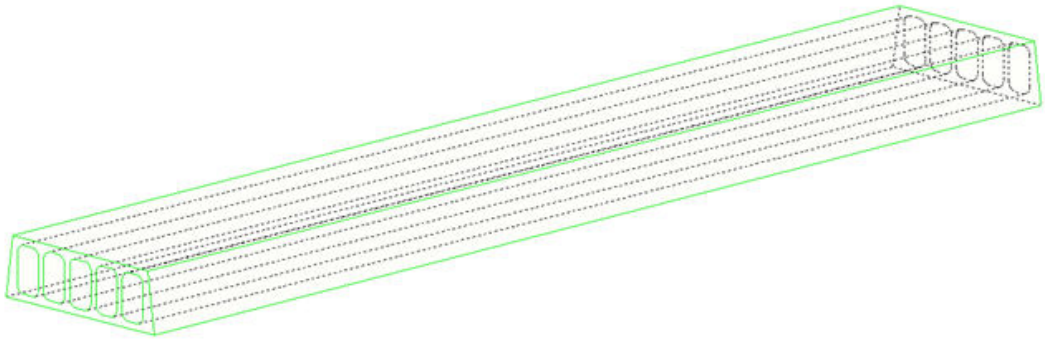
Below is an example of a hollow-core slab in the model:



Below is an example of the hollow-core slab in a drawing showing the fillet edges (**Fillet edges on/off** selected in the drawing level part properties or **Fillet edge** set in part properties in the property pane). As you can see, the representation is not very clear when the fillet edges are visible:



Below is an example of the hollow-core slab in a drawing not showing the fillet edges (**Fillet edges on/off** not selected in the drawing level part properties or **Fillet edge** set in part properties in the property pane):



Show neighbor parts and neighbor reinforcement in general arrangement drawings

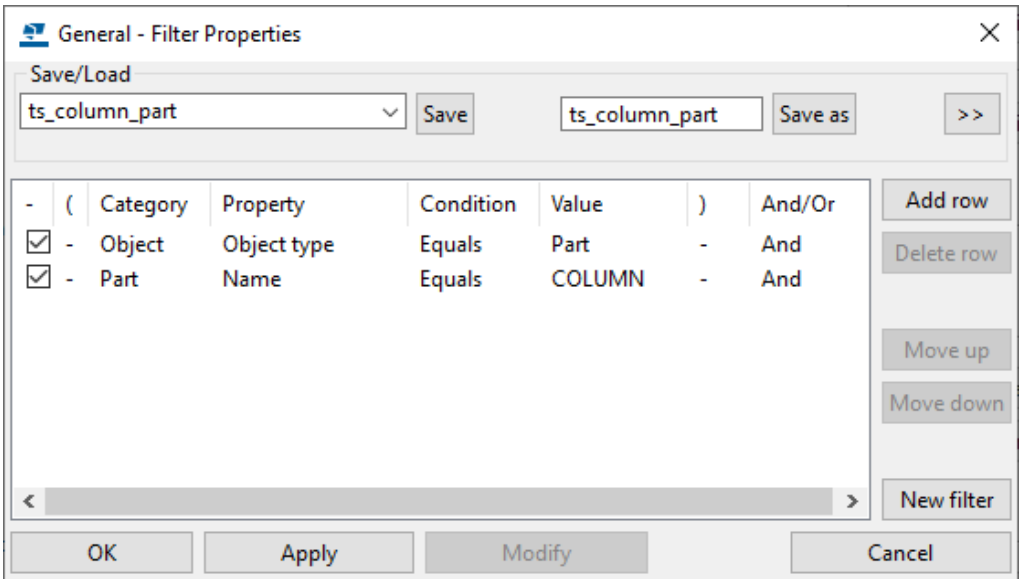
In GA drawings, you can define which parts are neighbor parts using neighbor part filters in the drawing or view level properties. The parts that fulfill the filtering criteria will be treated as neighbor parts. Note that you must also define a filter for normal parts to get the neighbor parts working.

The functionality described below for neighbor parts works in a similar way also for neighbor reinforcement. At the end of this page, there is an example filter for neighbor reinforcement.

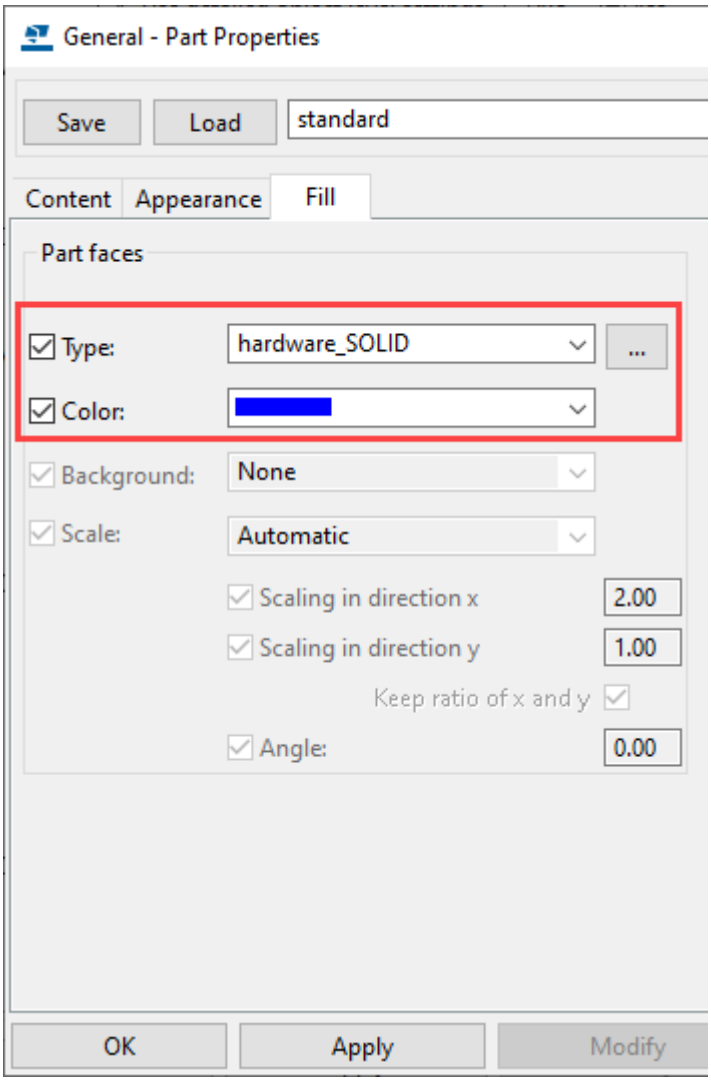
When you have defined which parts are normal parts and which are neighbor parts using the filters, you can define the way to show the parts and neighbor parts by modifying part and neighbor part properties.

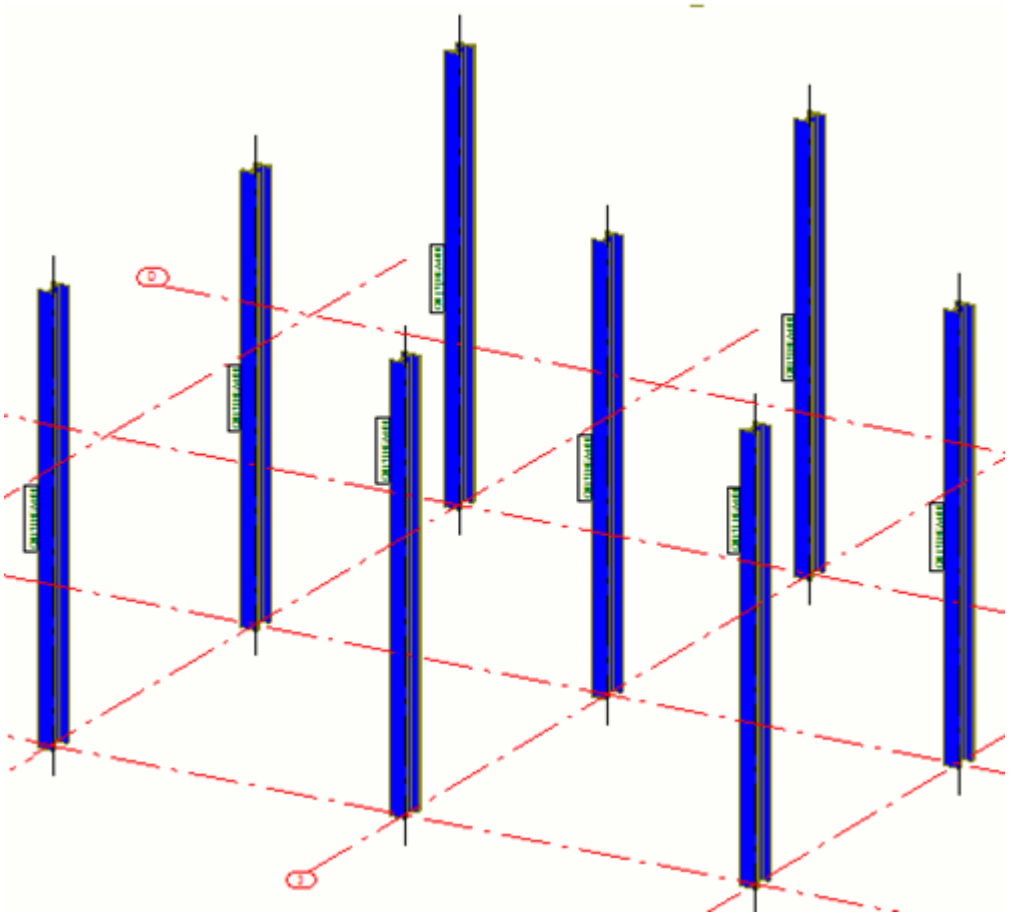
This simple example describes how you can show the normal parts (columns) in blue, and the neighbor parts (beams) in red. You need to create the necessary filters, and modify part and neighbor part properties.

1. In an open GA drawing, double-click the drawing background to open the **General arrangement drawing properties**.
2. To open the **General - Filter Properties** dialog box, click **Filter**, create a filter by **Part - Name** and **Object - Object type** for columns, and click **Modify**.

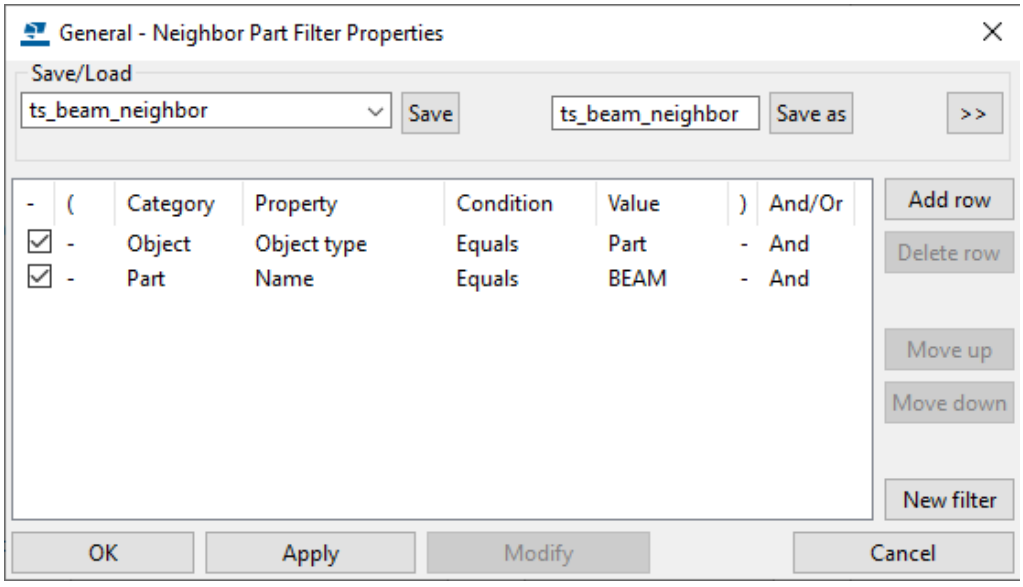


3. To go the **General - Part Properties** dialog box, click **Part** and on the **Fill** tab, select a fill, set the fill color to blue, and click **Modify**.



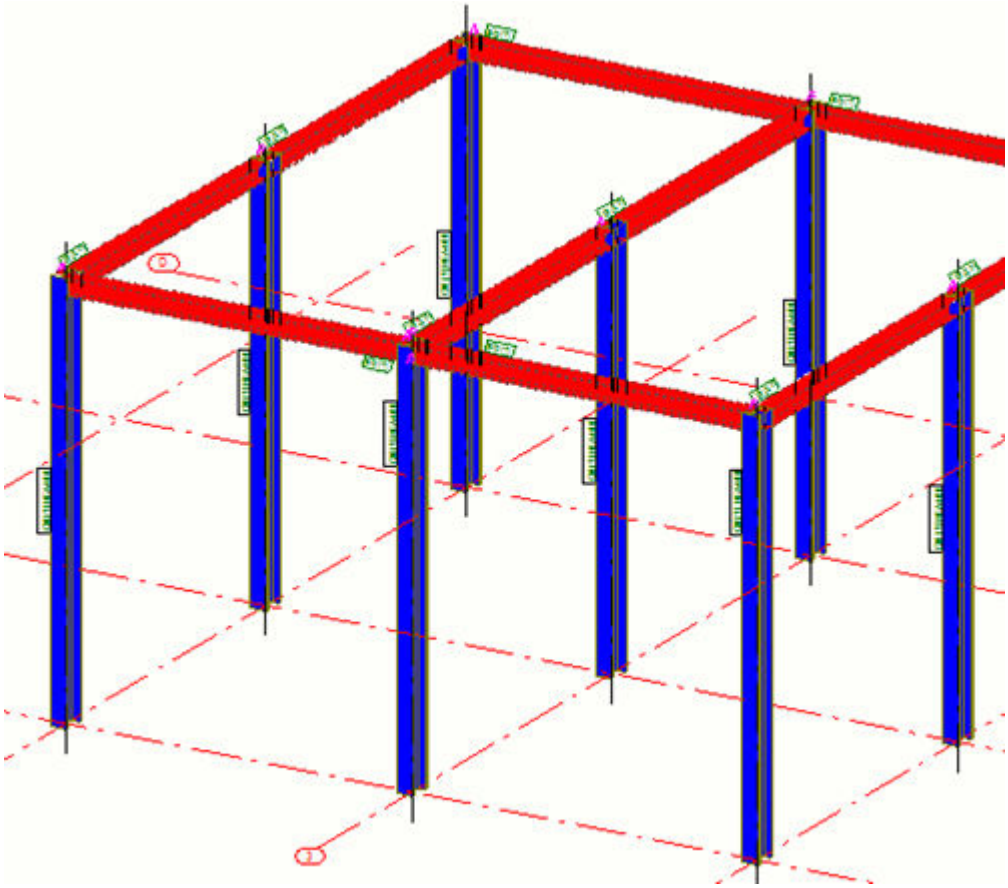


4. To open the **General - Neighbor Part Filter Properties** dialog box, click **Neighbor part filter**, create a filter by **Part - Name** and **Object - Object type** for beams, and click **Modify**.



5. To open the **General - Neighbor Part Properties** dialog box, click **Neighbor part**and do the following:
   - On the **Visibility** tab, set **Neighbor parts** to **By extreme**.
   - On the **Fill** tab, select a fill type, and set the fill color to red.
6. Click **Modify**.

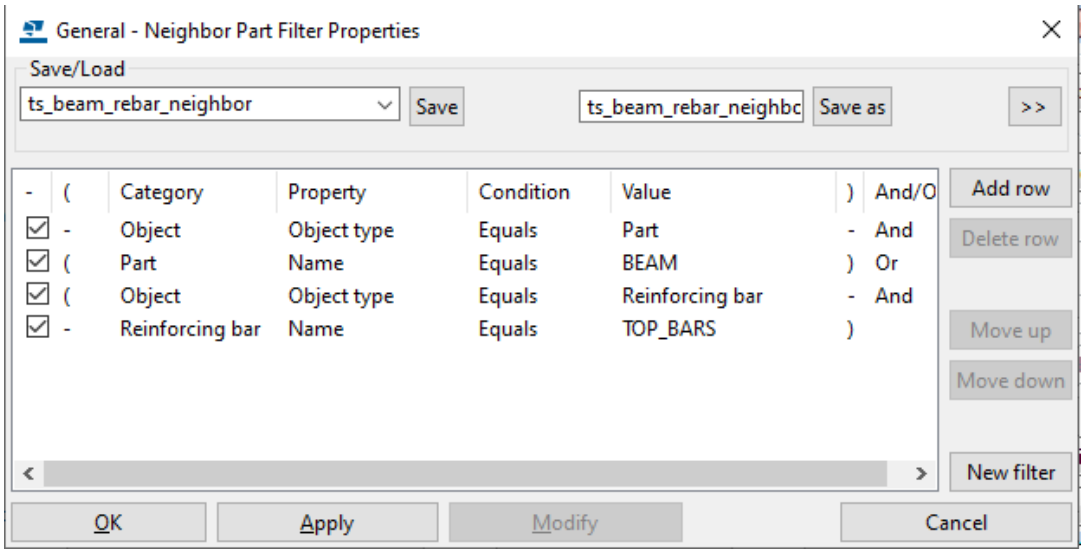
Now the parts are shown in blue and the neighbor parts in red.



TIP
- If you do not want to show the neighbor parts in the GA drawing, use a part filter for defining and selecting the normal parts, then go to **General - Neighbor Part Properties**, and on the **Visibility** tab, set **Neighbor parts** to **None**.
- Object level settings is another alternative for defining neighbor parts.

Example of a neighbor part and neighbor reinforcement filter

Below is an example of a filter that filters both neighbor parts and neighbor reinforcement.



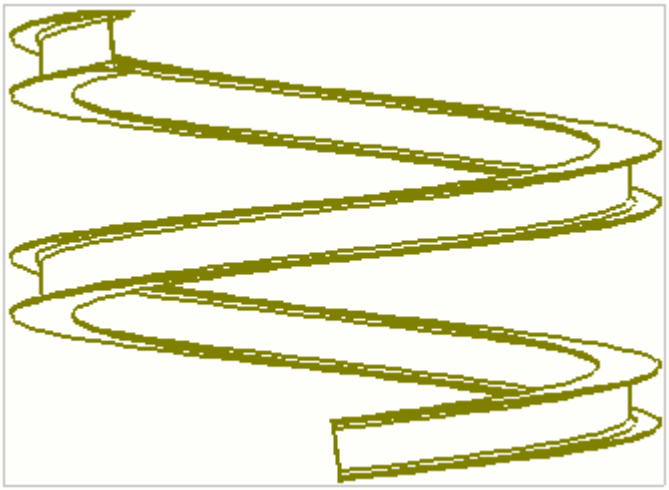
Show spiral beams in drawings

You can show spiral beams in drawings either folded or unfolded. In unfolded views, spiral beams are unfolded straight.

For details about creating concrete and steel spiral beams, see Create a concrete spiral beam and Create a steel spiral beam.

In the example below, the **Unfolded** option is set to **Yes** in view properties on the **Attributes 2** tab. Note that the part is cut in the view.

In the following example, the **Unfolded** option is set to **No**.



Dimension spiral beams

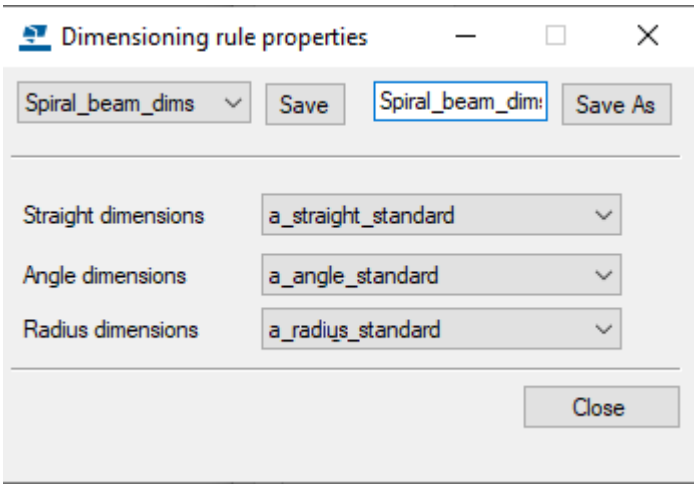
You can add straight dimensions, angle dimensions and radius dimensions to spiral beams.

In the following, the automatic creation of the dimensions in the defined views is explained.

For more information about dimension properties, see Dimension properties (page 932).

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
3. Click **Dimensioning** on the options tree.
4. Add a new row and select **Spiral beam dimensions** as the **Dimensioning type**, select the desired dimensioning rule properties and click **Edit rule**.
5. In the **Dimensioning rule properties** dialog box, select the predefined dimension properties that you want to use in **Straight dimensions**, **Angle dimensions**, and **Angle and radius dimensions**.

   If none of the available properties suit your needs, open a drawing, open the property pane, and then open straight dimensions, angle dimensions, and radius dimensions through the object list, and edit and save new settings for these dimension types. The saved settings will be available for selecting in the spiral beam **Dimensioning rule properties** dialog box for the three dimension types.
6. Save the dimension rule properties by entering a unique name and clicking **Save as**.



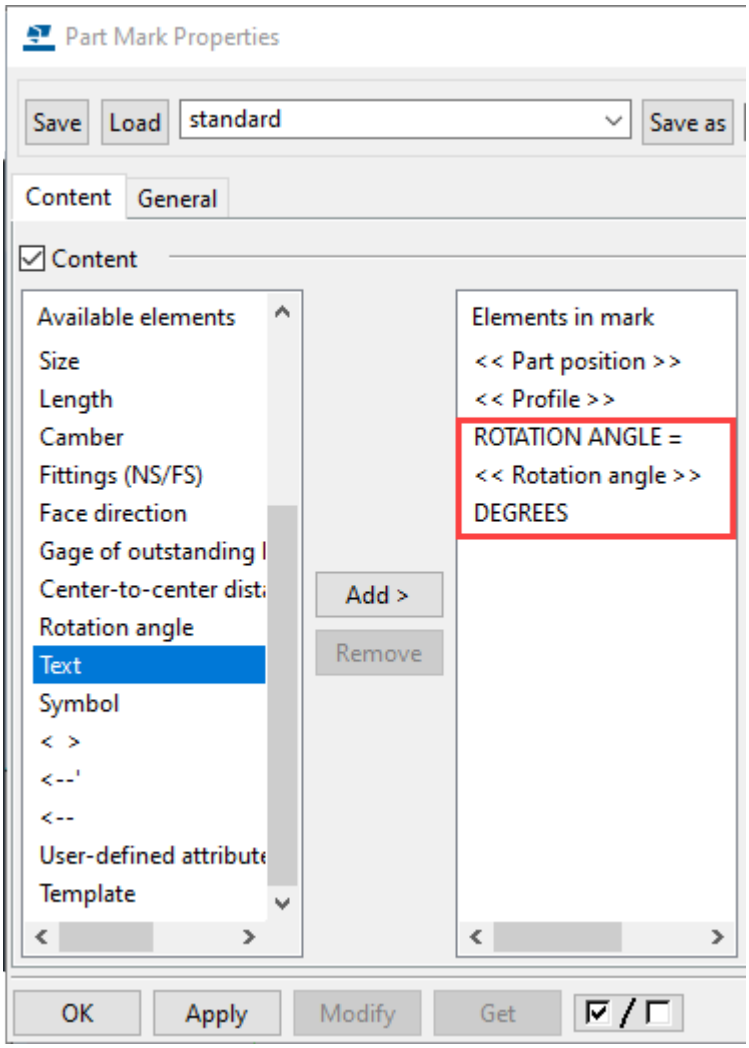
7. Save the view properties by clicking **Save** and return to the drawing properties dialog box by clicking **Close**.
8. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

Spiral beam part marks

You can show the rotation of the spiral beam in part marks. The rotation angle is defined in the spiral beam properties in the model.

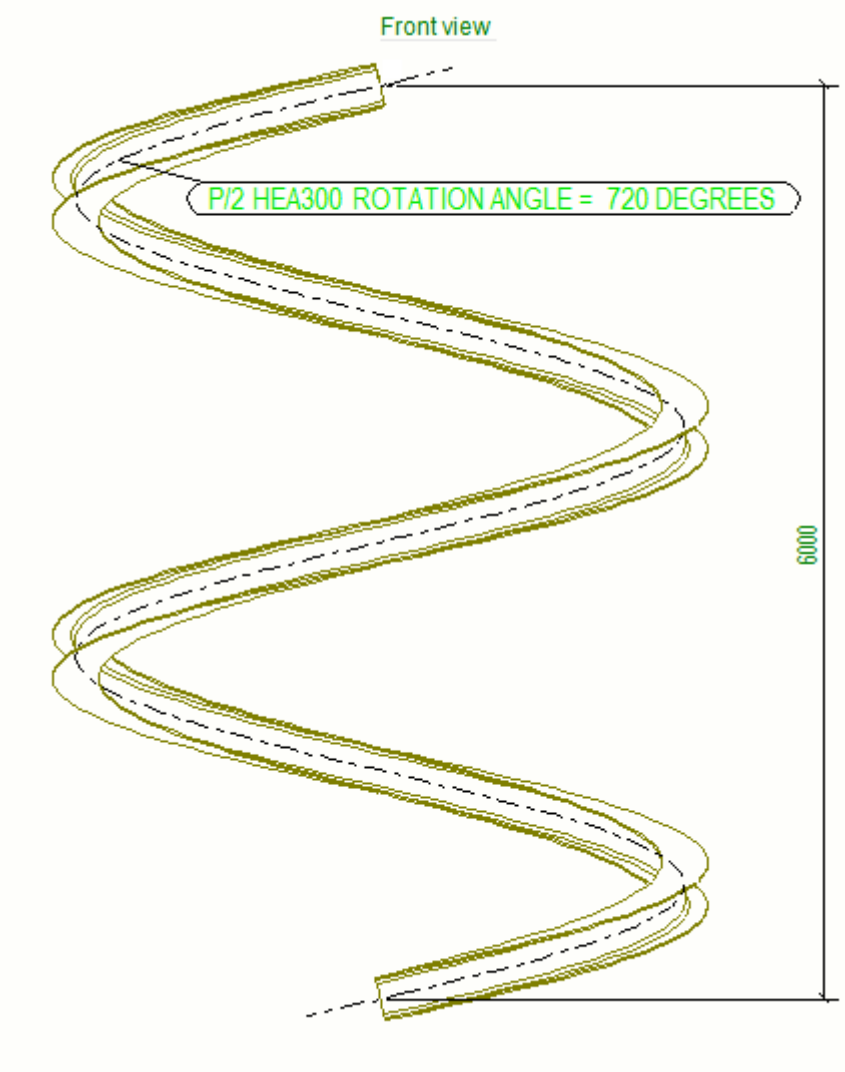
For more information about part mark properties and elements, see Mark properties (page 972) and Mark elements.

- You can add part marks (page 301) to spiral beams in the same way as for other parts using one of the following ways:
  - In an open drawing, select a spiral beam, right-click and select **Add mark** and then select **Using applied mark properties** to use the current mark properties, or **Using view properties** to use view level mark properties.
  - In an open drawing, select a spiral beam, and on the **Annotations** tab, click **Part mark** and select **For selected parts**.
  - You can also define automatic marks (page 811) in drawing properties before creating the drawing.

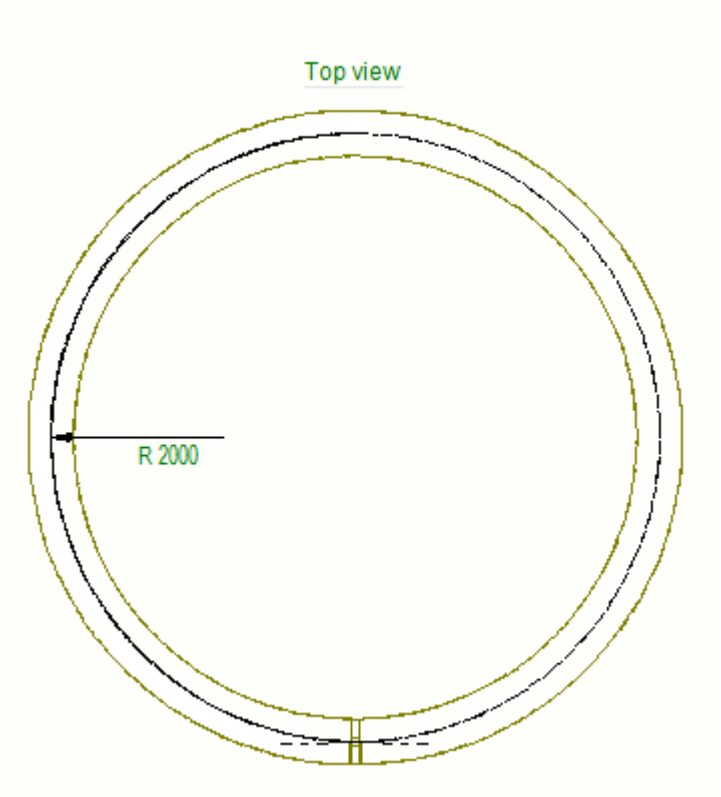

Part Mark Properties
Save
Load
standard
Save as
Content
General
Content
Available elements
Size
Length
Camber
Fittings (NS/FS)
Face direction
Gage of outstanding
Center-to-center dist
Rotation angle
Text
Symbol
< >
<--'
<--
User-defined attribute
Template
Add >
Remove
Elements in mark
<< Part position >>
<< Profile >>
ROTATION ANGLE =
<< Rotation angle >>
DEGREES
OK
Apply
Modify
Get

Examples of spiral beam dimensions and marks

The example below shows a straight dimension and a part mark that contains the rotation angle information:



The following example shows a radius dimension of a spiral beam:



Show reinforcement in drawings

There are several ways in Tekla Structures that you can use to affect how the reinforcement is shown in drawings.

Click the links below to find out more:

Show a single rebar in a group (page 449)

Show layer information on reinforcing bars in drawings (page 449)

Create a drawing view for a reinforcement mesh (page 474)

Define automatic reinforcement and mesh properties (page 885)

Reinforcement/Neighbor reinforcement and mesh properties in drawings (page 1016)

Show rebar assemblies in drawings (page 481)

You can dimension reinforcement:

Add dimensions to reinforcement manually in drawings (page 219)

Dimension rebars with Rebar group dimensioning application (page 228)

You can add marks and pull-out pictures to reinforcement:

Add reinforcement marks manually in drawings (page 303)

Add reinforcement marks with Rebar group marking application (page 305)

Draw rebar pull-out pictures with Draw rebar pull-outs application (page 451)

Draw rebar pull-out pictures with Rebar pull-out picture and marking application (page 454)

Show a single rebar in a group

You can show one rebar in a group or in a mesh and hide the rest.

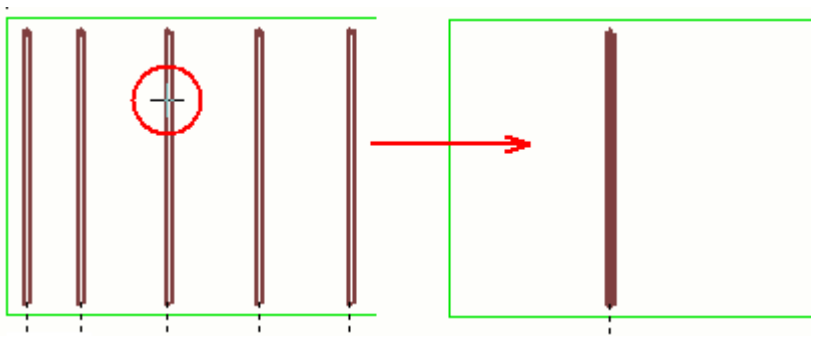
1. In an open drawing, select a rebar group or mesh.
2. Go to **Quick Launch**, type `Adjust Reinforcing Bars`, and click the **Adjust reinforcing bars** command on the displayed list.
3. Click the rebar you want to remain visible.

   Only one bar is displayed, and the others are hidden.

   When you use the `Adjust Reinforcing Bars` command to select the visible reinforcing bar, the **customized** option becomes available in the **Bars in groups** option list in rebar properties in the property pane. Note that this option is only available after you have used the **Adjust reinforcing bars** command and not when you define the drawing settings before you create the drawing, for example.
4. If you want to adjust the location of a single reinforcing bar, right-click the reinforcing bar group, and click **Adjust location**.

   You can only adjust the bar location in one direction inline with other group members.
5. Click the location where you want to place the bar.
6. If needed, change the number of visible bars again by clicking the bar and changing the setting to **all** in rebar properties, for example.

Example



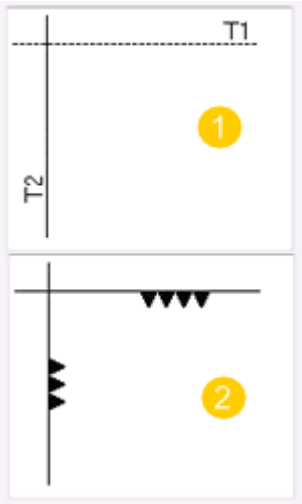
Show layer information on reinforcing bars in drawings

You can mark reinforcing bar layers with different marking styles and line types in a drawing by using the **Rebar layering marker** macro.

Before you can show layer information in a drawing, you first need to run the **Rebar classification** macro in the model. The **Rebar classification** classifies the meshes and reinforcing bars in the selected walls or slabs by their position. All reinforcing bars and meshes get an attribute indicating the layer where they are placed inside the concrete element.

To show reinforcing bar layer information in a drawing:

1. Open the drawing.
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 **Rebar layering marker** to display the **Rebar layering marker** dialog box.
5. Select symbol style or level prefix style from the list on the left showing the preview.
6. Select the marker line type.
7. Do one of the following depending on the selected marking style:
   - For symbol style markers, select the symbol you want to use, and the symbol height.
   - For level prefix style markers, select the level prefix.
8. Click **All objects** to show layering markers on all reinforcing bars, or select the individual reinforcing bars and click **Selected objects** to show the markers on the selected bars only.



1. Level prefix style layering marker. The number, for example number 1 in T1, indicates the layer number. The letter, for example T in T1, indicates whether the reinforcing bar is on the top, bottom, near side or far side layer.

2. Symbol style layering marker. The number of triangles indicates the layer number from the face. Triangle orientation indicates whether the reinforcing bar is on the top, bottom, near side or far side layer. For example for top bars, the triangle head points downwards, and for bottom bars upwards.

Draw rebar pull-out pictures with Draw rebar pull-outs application

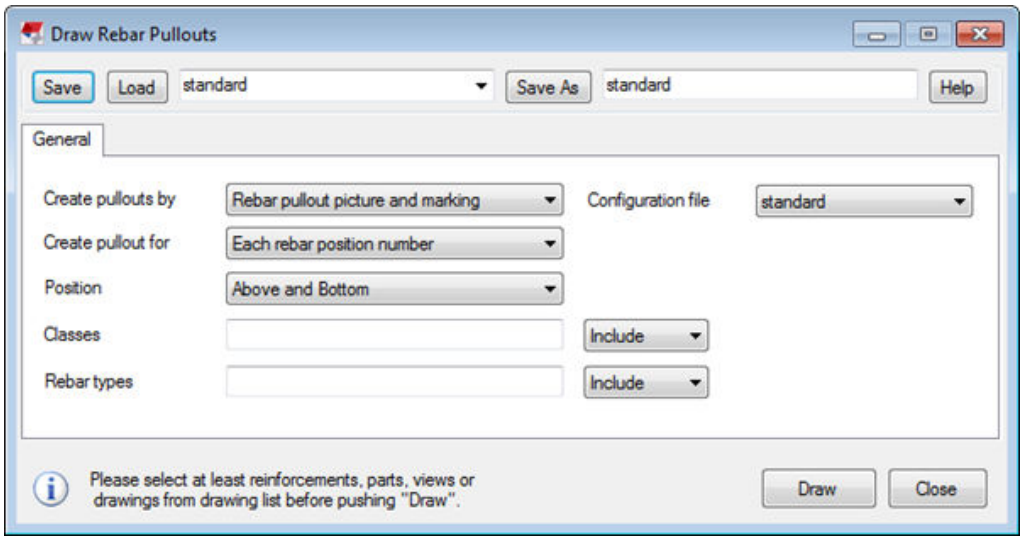
The **Draw rebar pull-outs** application creates exploded drawings of reinforcement using the settings defined in **Rebar pull-out picture and marking** or in associative notes. The application can be used for creating pull-out pictures automatically for multiple rebars at one go. To optimize flexible working with different types of drawings, use **Draw rebar pull-outs** together with **Rebar pull-out picture and marking**.

The application creates rebar pull-out pictures based on the objects that you select. You can select:

- Reinforcement: Bending shapes are only created for the selected rebars.
- Parts: Bending shapes are created for rebars in the selected concrete part.
- Views: Bending shapes are created for rebars in the selected drawing view.
- A drawing from the **Document manager**: Bending shapes are created for rebars in the selected drawings.

Draw pull-out pictures

1. In the drawing, select the object for which you want to create pull-out pictures: reinforcement, parts, views, or a drawing from the **Document manager**.
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. In the **Applications** list, double-click **Draw rebar pull-outs**.



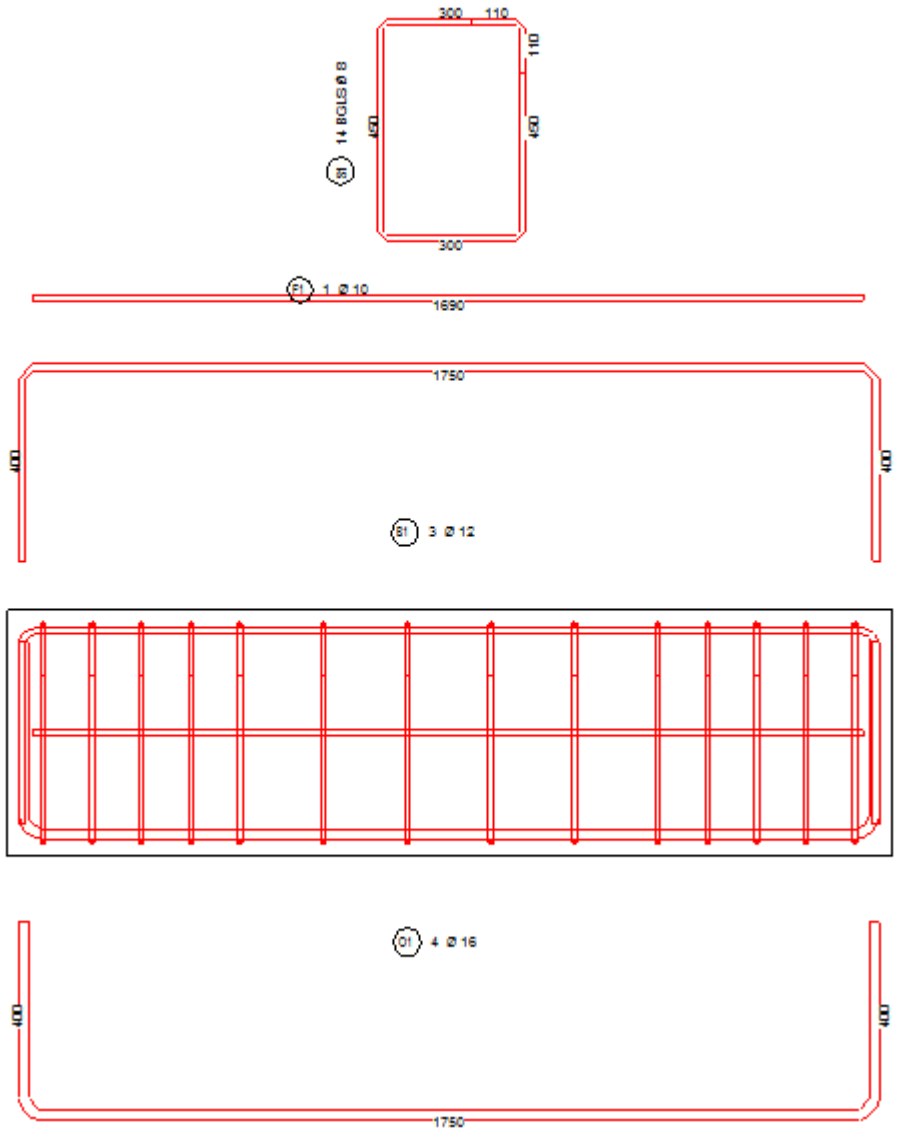
5. Define the type of method, the positions of the bending shapes, and other necessary settings:

Create pull-outs by	**Rebar pull-out picture and marking**: • Create the bending shapes on the basis of a saved **Rebar pull-out picture and marking** properties file. Define and save the properties in the **Rebar pull-out picture and marking** application, which can be started from the **Applications & components** catalog. • In the **Configuration file** list, select the desired settings file. **Associative note**: • Create the bending shapes on the basis of a saved associative notes properties file. • Define and save the properties in associative note properties. It is important to define at least one pull-out properties file for associative notes.
Create pull-out for	**Each rebar position number**: A bending shape is created for each rebar position number. **Each rebar group**: A bending shape is created for each rebar group.
Position	Defines the position of the bending shapes relative to the concrete part. The options are: **Above and bottom**: Positions the bending shapes above and at the bottom of the concrete part.

	Bottom: Positions the bending shapes at the bottom of the concrete part. **Above**: Positions the bending shapes above the concrete part. **Left**: Positions the bending shapes on the left side of the concrete part. **Right**: Positions the bending shapes on the right side of the concrete part.
Classes	Include or exclude the defined rebar classes. Define the class of the rebars. Use the **Exclude** or **Include** to define whether to create a bending shape for those rebars.
Rebar types	Include or exclude the defined bending shapes. Here you can define the shape codes of the rebars. Use **Exclude** or **Include** to define whether to create a bending shape for those rebars. You can inquire the shape code of a rebar by clicking **Inquire object** on the ribbon.

6. Click **Draw**.

Examples:



Draw rebar pull-out pictures with Rebar pull-out picture and marking application

With the **Rebar pull-out picture and marking** application you can visualize rebars by using bending shapes. Bending shapes can be positioned both inside and outside the concrete shape. Bending shapes are provided with marks containing information about the rebar.

Create rebar pull-out pictures and marks

1. In a drawing, select the rebar group for which you want to create a bending shape mark.

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. In the **Applications** list, select **Rebar pull-out picture and marking**.
5. Pick a point for the position of the bending shape mark.

   Note that you change the alignment setting for placement in the settings.
6. Double-click the mark to adjust the settings:
   - On the **Rebar** tab, define the geometry, line properties, scale, and location of the bending shape. Here you can also define the hook directions, and the presentation options.
   - On the **Mark 1** and **Mark 2** tabs, define the content, appearance and the position of the marks associated with the bending shape.
   - On the **Dimensions** tab, define how the dimensions are displayed, and how the rounding is performed.

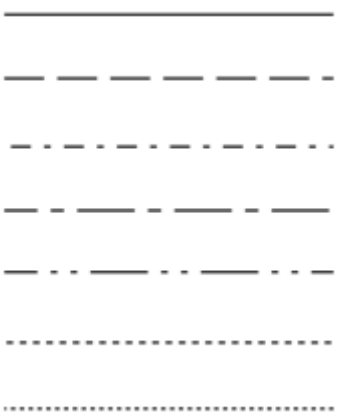
   For more information about the settings, see ***Rebar pull-out picture and marking*** *settings* below.
7. Click **Modify**.

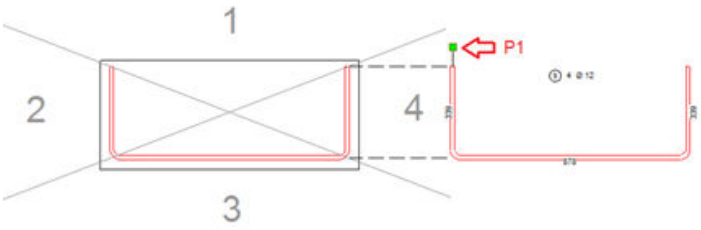
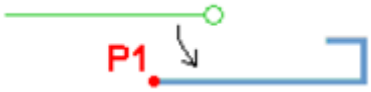
Rebar pull-out picture and marking settings

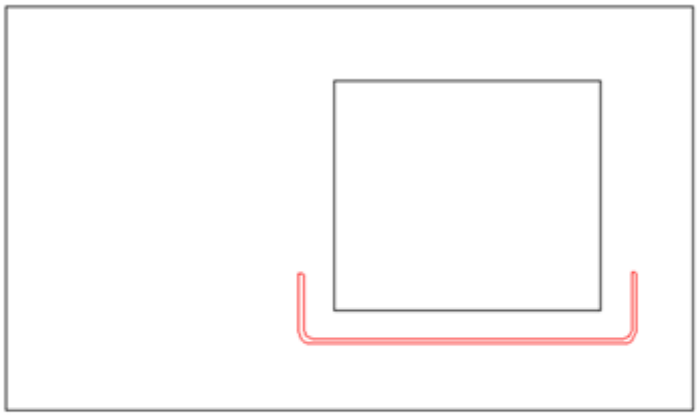
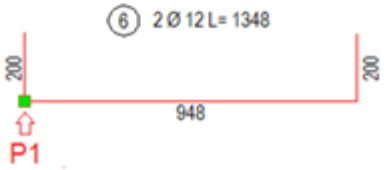
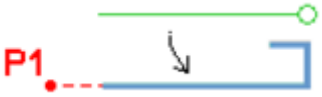
Rebar tab

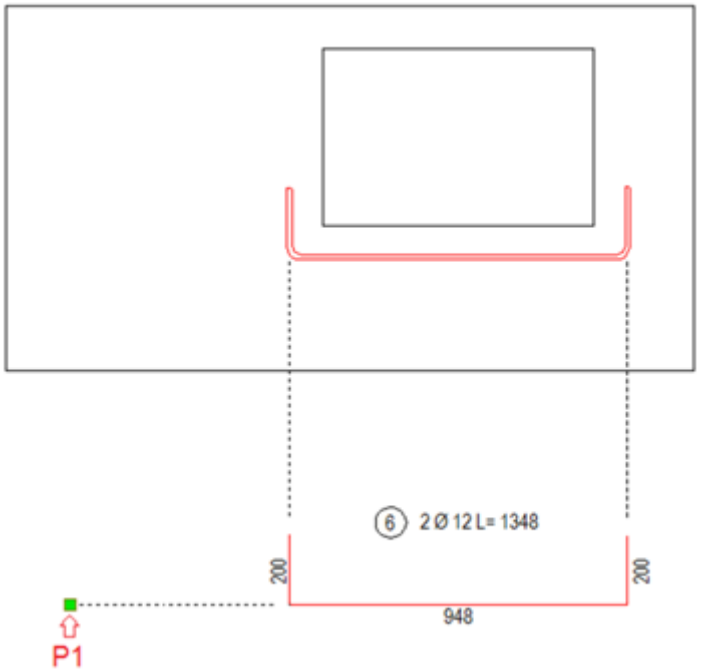
On the **Rebar** tab, define the representation and the location of the bending shape. Here you can also define the hook directions.

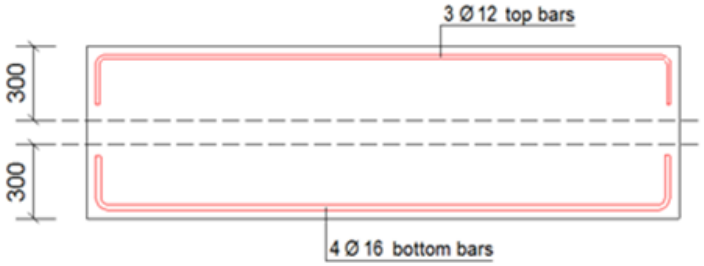
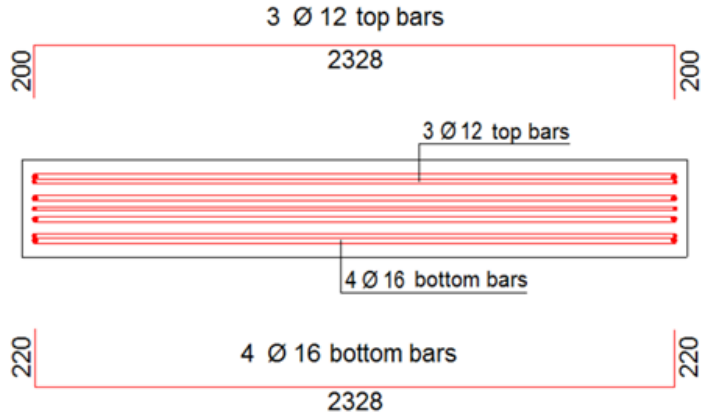
Setting	Options and descriptions
Rebar geometry	Select one of the following options: This is a schematic representation of the bending shape, without bending radius in the corners of the bending shape. In this option, the bending shape is represented with bending radius.
Use same line as pattern	• **Yes**: The color and line type are defined based on what is set in the drawing properties.

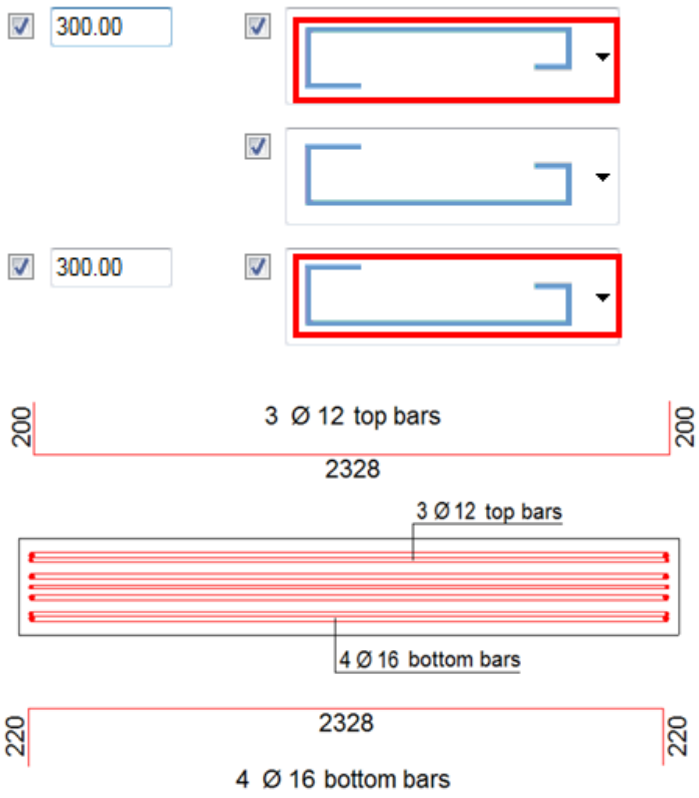
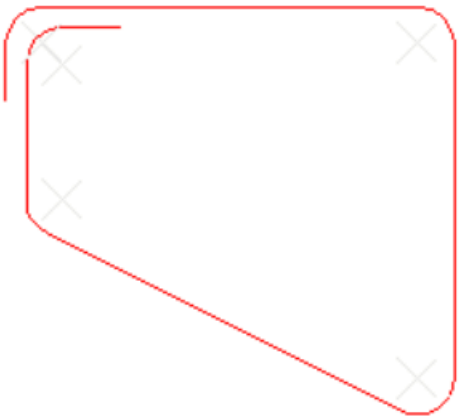
Setting	Options and descriptions
	• **No**: The color and line type are defined based on the **Color** and **Line type** settings in this dialog box.
Scale	In **Type**, select: • **Auto** to use the current view scale for the pull-out. • **Custom** to define a scale that is different from the view scale. For example, if the view scale is 1:50 and you want the pull-out to be created using the 1:25 scale, type `25` in the **Scale value** box.
Color	
Line type	
Reinforcing bar representation	Select the rebar representation: • **Single line**: The bending shape is represented with a single line. • **Double lines nominal diameter**: The bending shape is represented with double lines. The nominal diameter of the bar is considered. • **Double lines actual diameter**: The bending shape is represented with double lines. The actual diameter of the bar is considered, including the ribs on the bar.

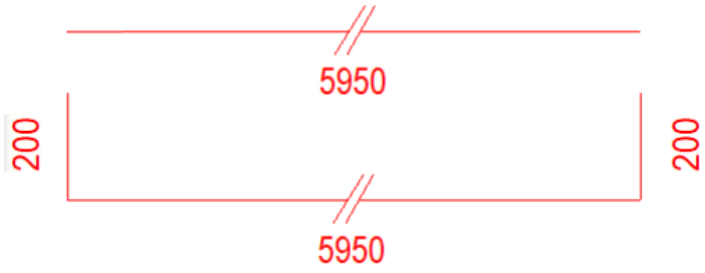
Setting	Options and descriptions
Rebar location	Define the position of the bending shape using any of the options described below: Option 1: In this option, the area around the rebar is divided in four quadrants. The insertion point can be picked in any random quadrant. The position of the bending shape is based on the picked quadrant: • Insertion point in quadrant 1: the bending shape is centered on top of the rebar. • Insertion point in quadrant 2: the bending shape is centered to the left of the rebar. • Insertion point in quadrant 3: the bending shape is centered under the rebar. • Insertion point in quadrant 4: the bending shape is centered to the right of the rebar. Example:  Option 2:  In this option, the insertion point P1 is the start point of the bending shape. Example:

Setting	Options and descriptions
	  Option 3:  In this option, the bending shape is located on the line of the insertion point P1. The bending shape will be positioned centered relative to the position of the rebar in the concrete shape. Example:

Setting	Options and descriptions
	 Option 4: This option creates marks and hooks for all rebars in the view.
Rotation of bars perpendicular to view	Rotate rebar pull-out pictures vertically or horizontally. This setting affects single bars perpendicular to the view only.
Custom rotation angle	Rotate all pull-out pictures using the same rotation angle.
Hook directions	Define the direction of the hooks of the bars. Seen from a top view from a beam, for example, the direction of the hooks may be defined as **Top**, **Middle** and **Bottom** of the the beam. For all of the directions, you can select between two options: Use the **Thickness** box to define an area for the bars on the top and bottom side of the beam.

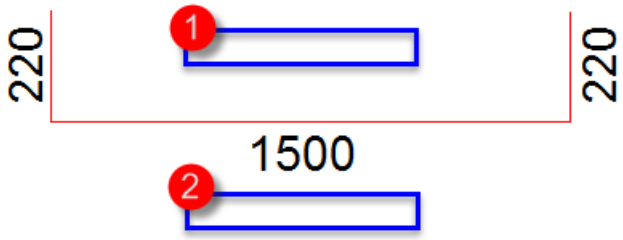
Setting	Options and descriptions
	Example 1: In the front view of a concrete beam below, both the **Top** and the **Bottom** thicknesses are set to `300`:  Example 2: Top view of a concrete beam  Example 3: Top view of a concrete beam with other hook directions

Setting	Options and descriptions
	
Presentation options	When set to **Yes**, **Exaggeration** exaggerates a shape when the reinforcing bar parts are overlapping. This setting works in a way similar to the pull-out pictures in rebar marks. In the following example, **Exaggeration** is set to **Yes**.  **Shorten bars longer than** allows you to set a maximum output length for long bars. If the

Setting	Options and descriptions
	boundary of the shape exceeds the maximum length, the pull-out picture is shortened with cut lines. Use this setting especially for straight bars or straight bars with hooks. In the following example, **Shorten bars longer than** is set to 1000.00. 

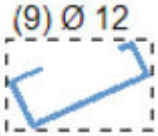
Mark 1 and Mark 2 tabs

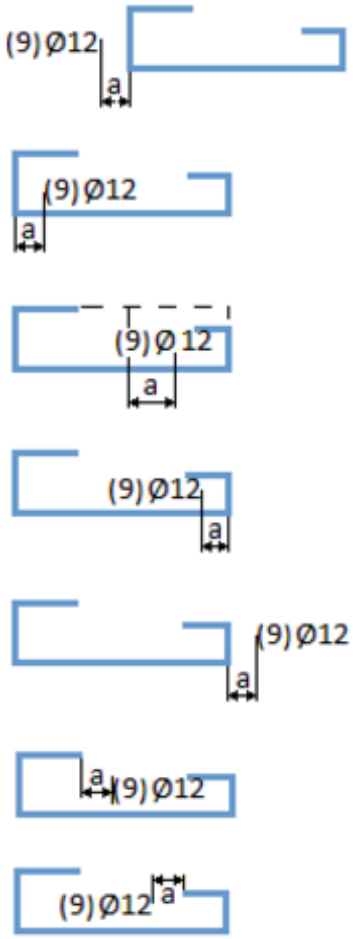
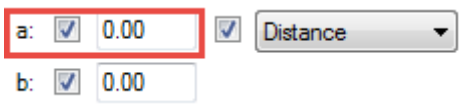
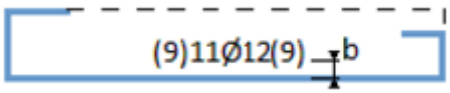
On these tabs, you can define the generation of **Mark 1** and **Mark 2**. For example:

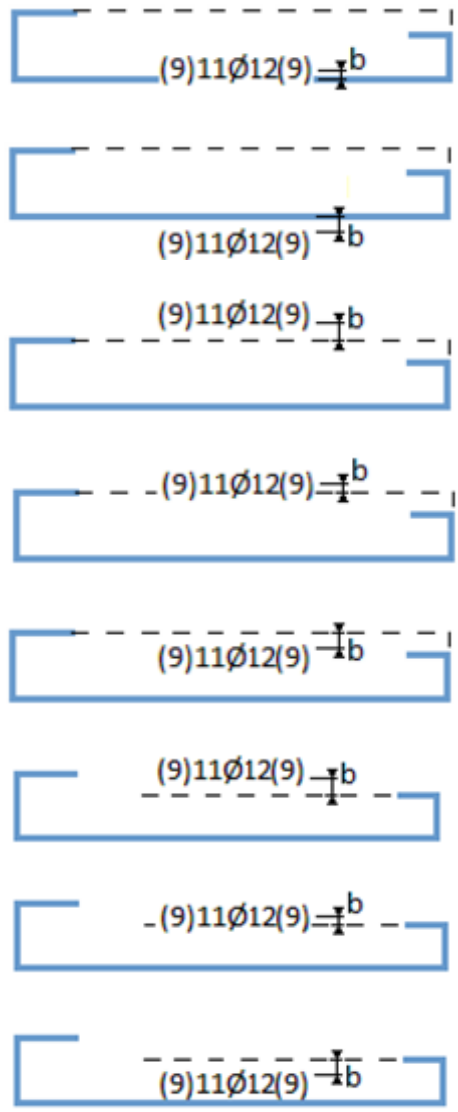
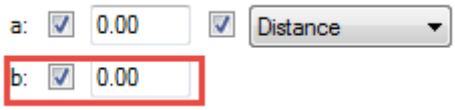


Settings	Options and descriptions
Available elements	Select the rebar information to be displayed in the mark, such as grade, diameter, and cc distances.
Elements in mark	A list of the selected information or elements that will be displayed in the mark.
Text properties	Define the text properties. The available properties files are the ones that have been defined and saved in Text properties (page 364).
Rebar text properties	Define the text properties for the rebar position. The available properties files are the ones that have been defined and saved in Text properties (page 364).
Number of rebars	**Total number in rebar group**: Displays the total number of rebars in the rebar group, regardless of

Settings	Options and descriptions
	the number of the physically visible rebars in the drawing. **Number displayed in view**: Only displays the number of the visible rebars in the drawing view. **Total number in cast unit**: Displays the total number of rebars in the cast unit. These options are only available when you select **Number** as the mark content.
Units	Define the units: • **Automatic** • **mm** • **cm** • **m** • **foot - inch** • **inch** Only available for the following mark content: • **Length** • **Cc** • **cc min** • **cc max** • **cc exact** • **cc target** • **Length itemized**
Format	Define the format: • ### • ###[.##] • ###[.###] • ###.# • ### #/# • ###.## • ###.### Only available for the following mark content: • **Length** • **Cc**

Settings	Options and descriptions
	• **cc min** • **cc max** • **cc exact** • **cc target** • **Length itemized**
Precision	Define the precision: • 0.00 • 0.50 • 0.33 • 0.25 • 1/8 • 1/16 • 1/32 • 1/10 • 1/100 • 1/1000 For example, for the precision 0.33, the actual value 50.40 is displayed as 50.33. 1/8, 1/16, and 1/32 are for imperial units. 1/10, 1/100, and 1/1000 are used for defining precision without rounding.
Mark positioning	Define the positioning of the mark. There are three options available: Option 1: Mark text parallel is to the bending shape. Option 2:  Mark text is placed horizontally. Option 3:

Settings	Options and descriptions
	 Mark text is placed vertically.
Horizontal text position	Define the position of the text in the horizontal direction. The options are:  Use the setting **a** to define a distance: 
Vertical text position	Define the position of the text in the vertical direction. The options are: 

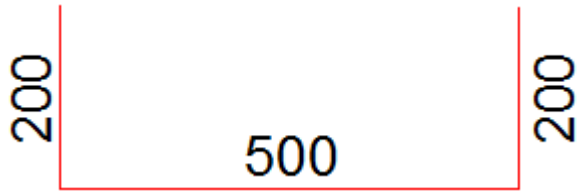
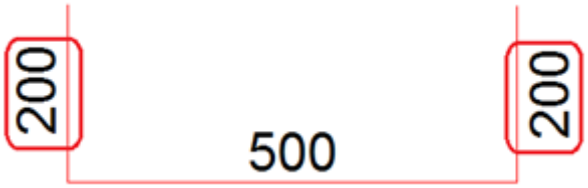
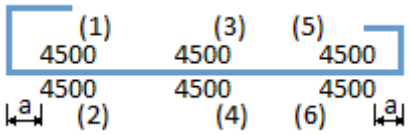
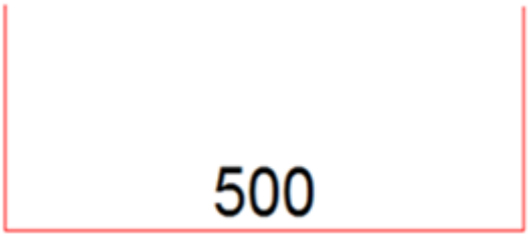
Settings	Options and descriptions
	 Use box **b** to define a distance: 

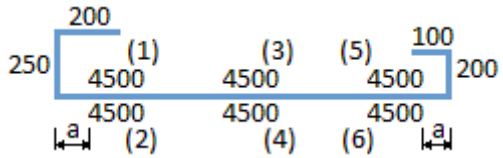
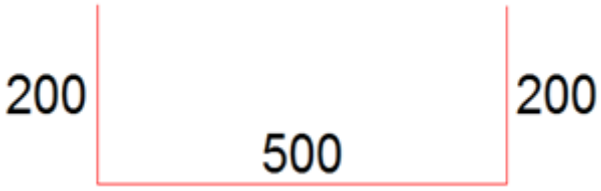
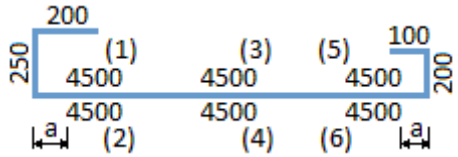
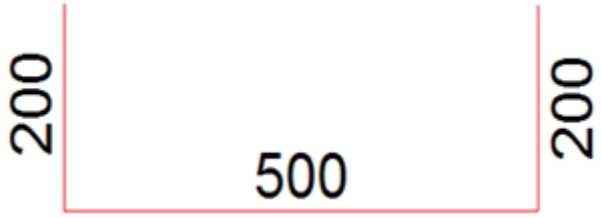
Dimensions tab

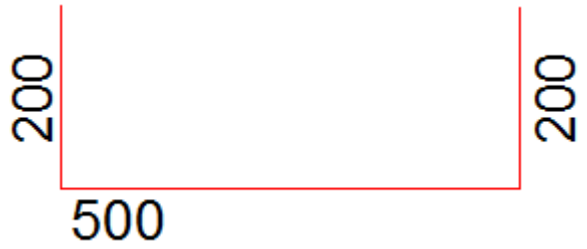
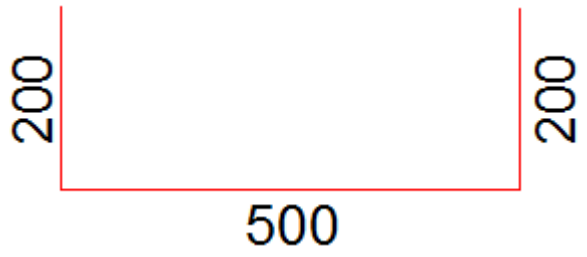
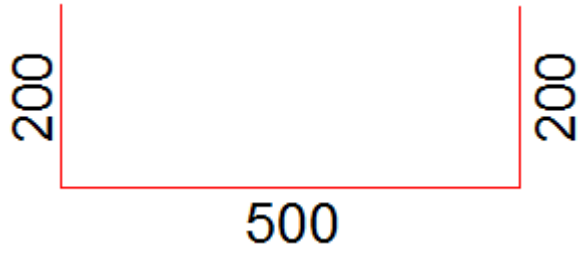
On the **Dimensions** tab, you can define how the dimensions are displayed, and how the rounding is performed.

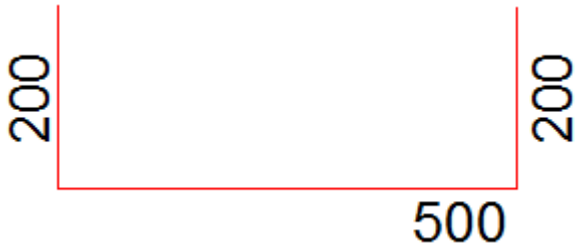
Note that to allow duplicate dimensioning, set `PullOutShowDuplicateDims` to the desired value in the `rebar_config.inp` file. For example, `PullOutShowDuplicateDims=3` shows all dimensioning, and `PullOutShowDuplicateDims=0` does not show duplicate dimensions. The

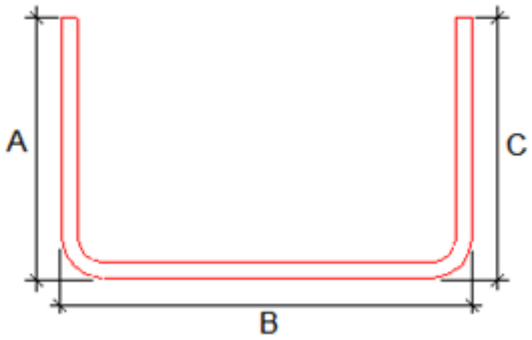
default value is `0`. For more informaton, see Reinforcement settings for drawings (rebar_config.inp) (page 1023)

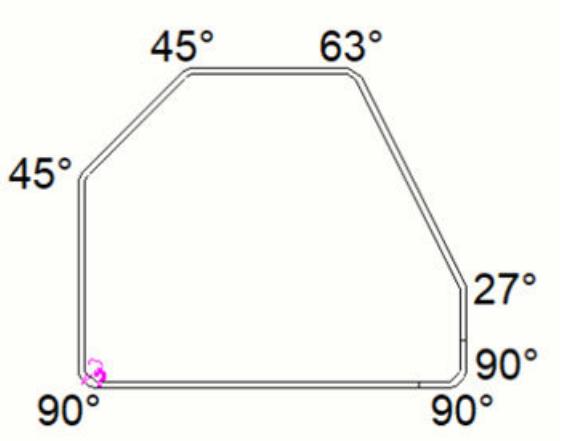
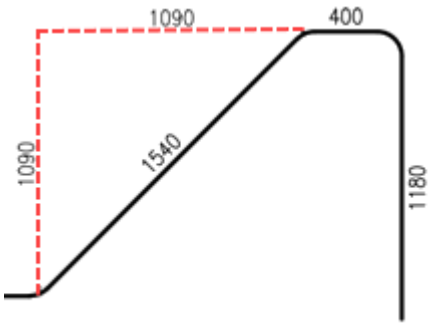
Settings	Options and descriptions
Create	Define whether all bending dimensions are displayed with the bending shape or not. The options are: • **Yes**  • **No**
Generation of leg lengths	Define the generation of the length of the legs, marked in the image below:  Option 1:   Option 2:

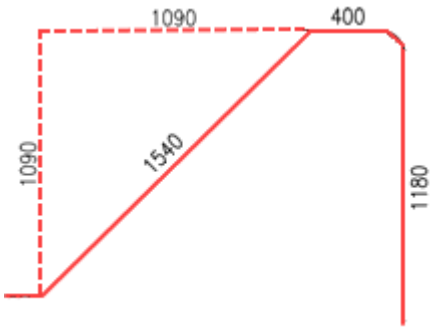
Settings	Options and descriptions
	  Option 3:  
Tapered lengths	To dimension rebars when the rebars are of different lengths, select **Yes**.
Location	Define the location of the length of the leg, marked in the image below: The options are:

Settings	Options and descriptions
	• **None** • **(1)** • **(2)**  • **(3)**  • **(4)** 

Settings	Options and descriptions
	• **(5)** • **(6)** 
Round settings	Define whether the rounding is performed by user settings or by `rebar_config.inp`. If you select to use `rebar_config.inp`, you cannot define the units, precision, and format.
Units	Define the units. The options are: • **Automatic** • **mm** • **cm** • **m** • **foot - inch** • **inch**
Format	Define the format. The options are: • ### • ###[.##] • ###[.###] • ###.# • ### #/# • ###.## • ###.###
Precision	Define the precision. The options are: • 0.00

Settings	Options and descriptions
	• 0.50 • 0.33 • 0.25 • 1/8 • 1/16 • 1/32 • 1/10 • 1/100 • 1/1000 For example, for the precision 0.33, the actual value 50.40 is displayed as 50.33. 1/8, 1/16, and 1/32 are for imperial units. 1/10, 1/100, and 1/1000 are used for defining precision without rounding.
Text properties	Define the properties for the text that will be displayed for the bending shape (length of the legs). The available properties files are the ones that have been defined and saved in Text properties (page 364) .
Length calculation	**Exact length**:  If you select **Along rebar axis**, the length is calculated along the center line of the rebar:

Settings	Options and descriptions
Angle dimensions	In **Create**, define whether you want to show the bending angles of the rebar pull-outs. The options are: • **Yes**: Shows the bending angles.  • **Dimension lengths**: Creates perpendicular dimensions when the bar shape gets a not 90° angle that returns to parallel or perpendicular to the original bar leg. This option is useful for complex bends that will normally be done on-site. 

Settings	Options and descriptions
	• **Both dimensions**: Creates both **Dimension lengths** type of dimensions and basic angle dimensioning.  You can also define the **Text properties** for the bending angles. The available properties files are the ones that have been defined and saved in Text properties (page 364) . Define the **Format** for angle dimensions. The options are: • ### • ###[.##] • ###[.###] • ###.# • ### #/# • ###.## • ###.### Define the **Precision** for angle dimensions. The options are: • 0.00 • 0.50 • 0.33 • 0.25 • 1/8 • 1/16 • 1/32 • 1/10 • 1/100

Settings	Options and descriptions
	• 1/1000 You can also define a **Color** and **Line type** for the angle dimensions.

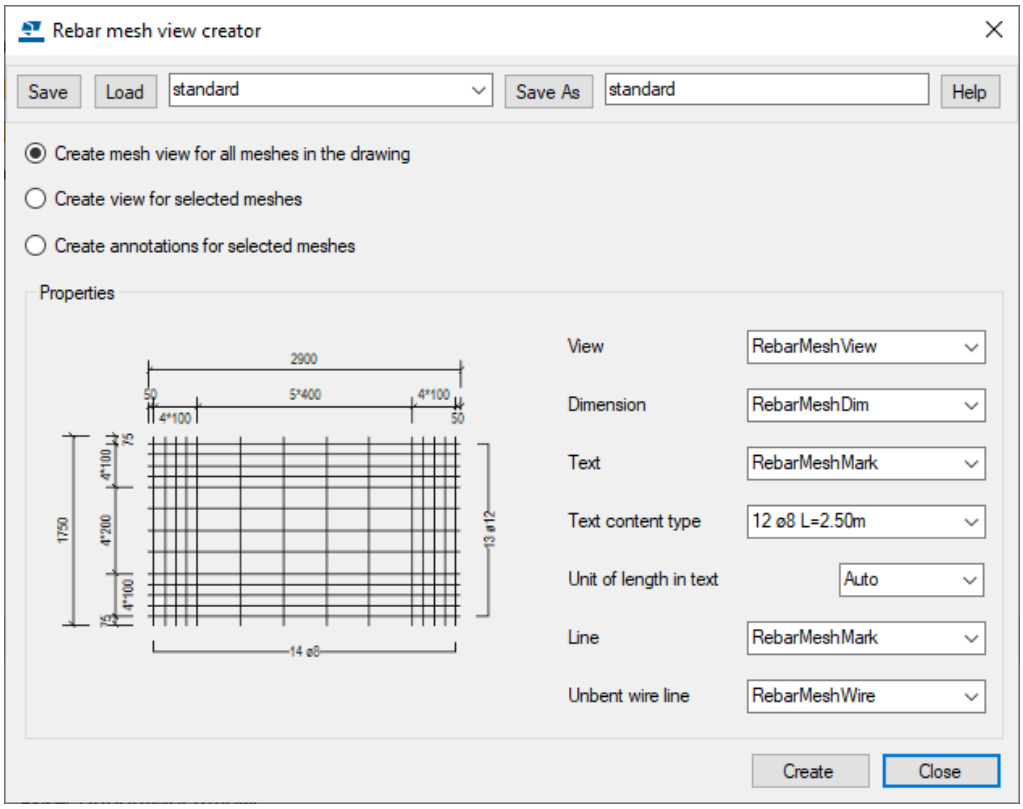
Create a drawing view for a reinforcement mesh

You can create drawing views each containing one reinforcement mesh by using the **Rebar mesh view creator** macro. The mesh view contains the overall mesh length and width dimensions, and dimension lines for wire spacing in horizontal and vertical directions. Also the wire sizes are displayed. You can create mesh views in general arrangement and cast unit drawings.

1. Open a drawing containing meshes.
2. Select the meshes.

   If you want to create views for all meshes in the drawing, you do not need to select separately.

3. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
4. Click the arrow next to **Applications** to open the applications list.
5. Double-click **Rebar mesh view creator**.



6. You have the following options for creating mesh views:
   - If you want to create a mesh view for each separate mesh in the current drawing, select **Create mesh view for all meshes in the drawing**.
   - If you want to create a view for the selected meshes only, select **Create view for selected meshes**. Here you must select the meshes before running the macro.
   - If you have already created the mesh views and just want to add dimension lines and diameters in the views, select **Create annotations for selected meshes**. Here you must select the meshes before running the macro.
7. Select the view, dimension, text, line, and unbent wire line property files that you want to use in the new the mesh view.
8. Select the wire annotation type in **Text content type** .

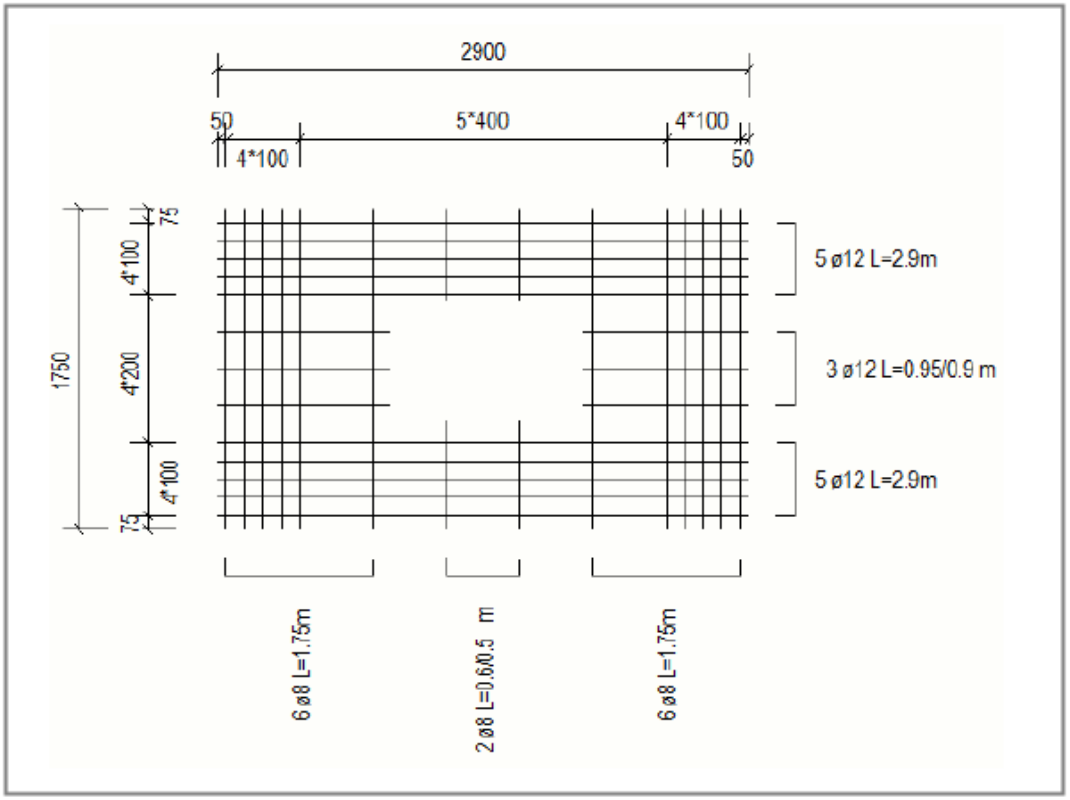
   The choices are and .
9. Select the wire length unit in **Unit of length in text**.

   When you select **Auto**, with imperial units the text will be using the current units, and otherwise the text will use **m**.
10. Click **Create**.

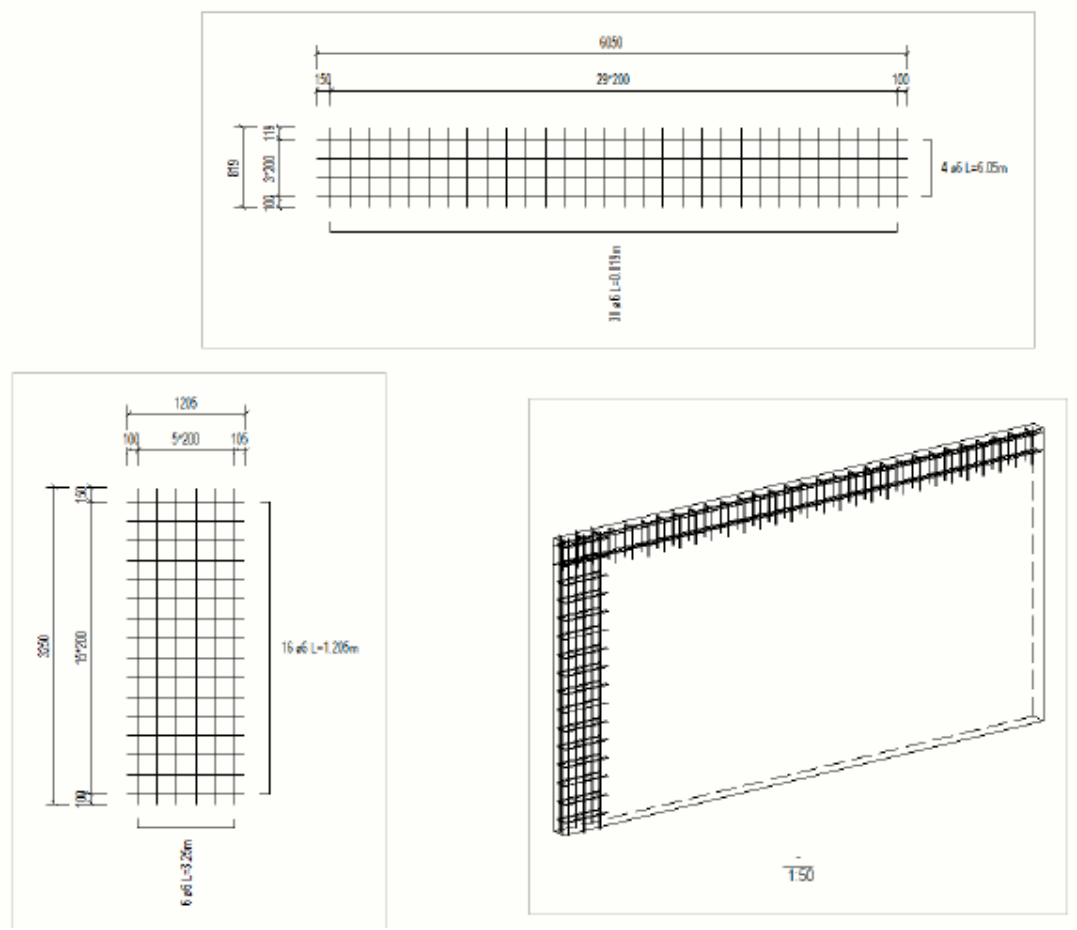
    Tekla Structures creates or updates the views based on your selections and the property files.

Example

Below is an example of a mesh view.



The drawing below contains a 3D view of a wall with two bent meshes, and a separate unbent mesh view of both of the meshes.



Show rebar coupler and end anchor symbols in drawings

You can show graphical symbols representing the rebar couplers and end anchors in bending schedules and in rebar mark pull-out pictures using the Pullout graphical attribute.

You need a precast concrete or cast-in-place structure model with generated rebar detailing drawings (GA or cast unit drawings), and the reinforcement details need to be modeled by using any of the rebar coupler or rebar end anchor tools available in the **Applications & components** catalog. The rebar coupler and end anchor symbols work based on rebar UDAs, and these UDAs are controlled by the rebar coupler and rebar end anchor tools.

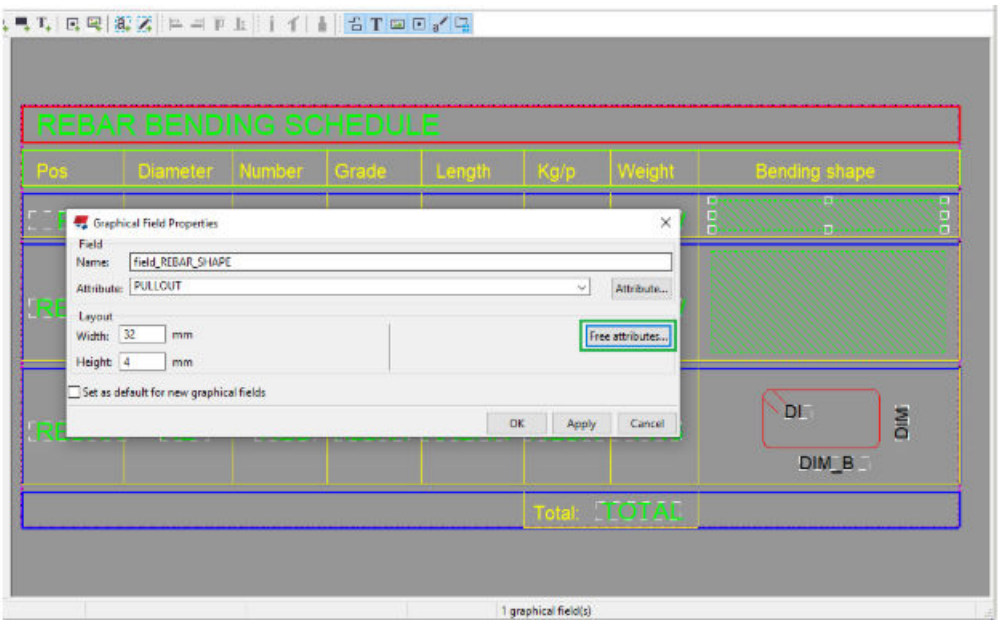
The symbols are read from the file defined in `RebarCoupler.Symbols.dat`, by default `CouplerSymbols.sym` located in `..\ProgramData\Tekla Structures\<version>\environments\common\symbols`.

You can adjust the font name, font size, font color, end mark, bending radius visibility and a number of other bending schedule attributes individually for the Pullout graphical attribute in Template Editor, according to your company or country standards.

Display rebar coupler and end anchor symbols in rebar bending schedules

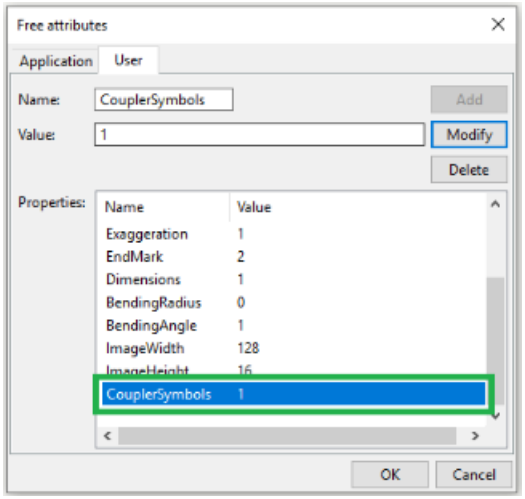
Note that the PULLOUT attribute shows in pull-out pictures graphical symbols representing the couplers at the bar ends only if PullOutRepresentation=0 in the `rebar_config.inp` file.

1. In a drawing that contains rebar couplers or end anchors shown in a rebar bending schedule, open the bending schedule template in Template Editor by double-clicking a rebar shape in the bending schedule.
2. Double-click the graphical field containing the bending shape.
3. In the **Graphical Field Properties** dialog box, ensure that the PULLOUT attribute is displayed in the **Attribute** field, and open the **Free attributes** dialog box.

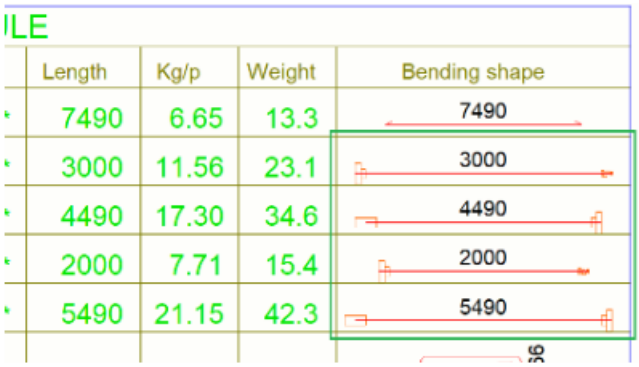


4. Select the CouplerSymbols attribute, and enter 1 in the **Value** field. Adjust other properties as required.

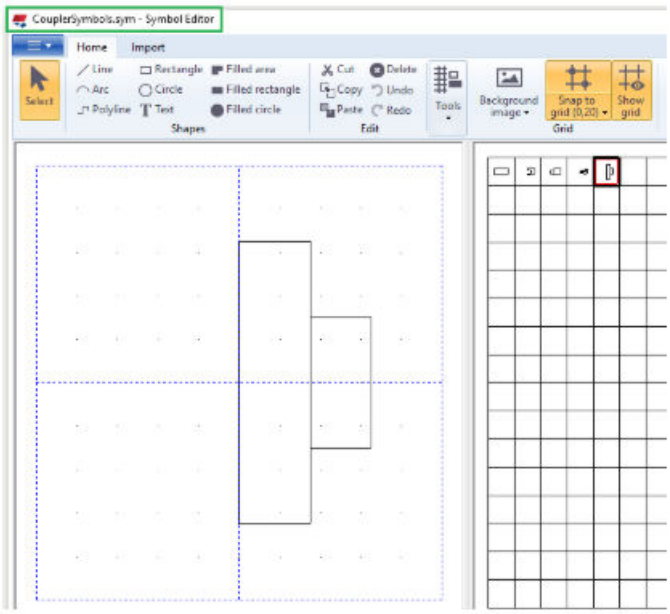
Rebar coupler symbols will be shown if CouplerSymbols property value is set to 1 and disabled if 0 is entered. The default value is 1.



5. Save the changes in the template and close Template Editor.
6. Refresh the template by right-clicking the template in the drawing and selecting **Refresh template** from the context menu.

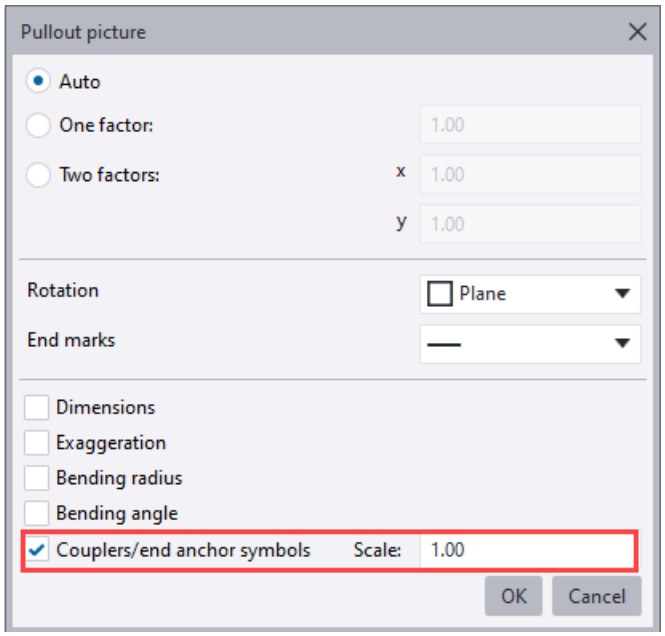


TIP You can customize rebar coupler symbols in Symbol Editor by modifying the `*.sym` file defined in the `RebarCoupler.Symbols.dat` file. By default, `CouplerSymbols.sym` is used.

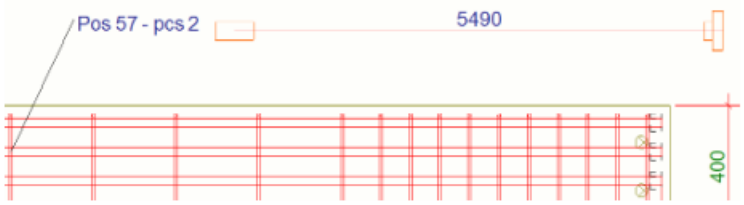


Display rebar coupler and end anchor symbols in rebar mark pull-out pictures

1. In an open drawing that contains reinforcement with couplers or end anchors, click a rebar mark in which you want add a pull-out picture, and add the **Pullout picture** element in the mark from the element list. The **Pullout picture** dialog box is displayed.
2. Select the **Couplers/end anchor symbols** check box, and set the desired scale.



3. Click **OK**.



Show rebar assemblies in drawings

You can create fabrication drawings of rebar assemblies, and show rebar assemblies with various representations. You can define view-specific rebar assembly settings on the **Rebar assembly** tab in view-level reinforcement properties, and add assembly position in reinforcement marks and associative notes.

You can create a rebar assembly in the model by using the **Create rebar assembly** command. To create rebar assembly drawings, select the rebar assembly and click the **Create fabrication drawing** command, which you can find on the ribbon, context menu, and contextual toolbar.

Adjust rebar assembly visibility and representation

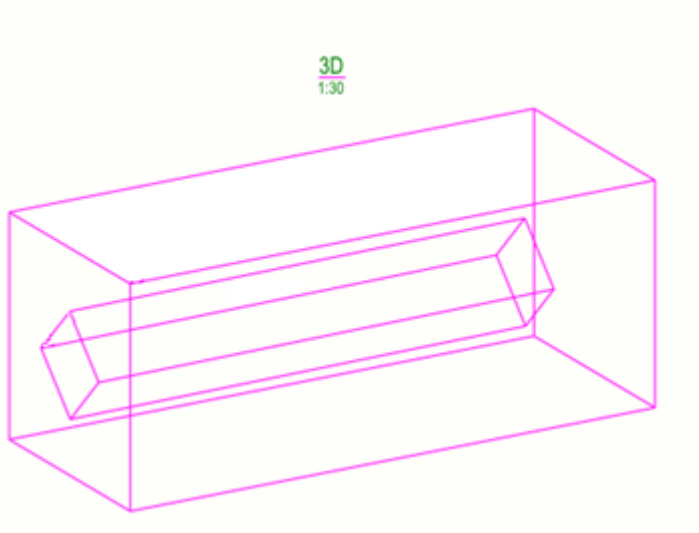
1. Open the desired rebar assembly drawing.
2. To open the view-level properties dialog box, double-click the frame of the drawing view that contains a rebar assembly.

3. To adjust the rebar assembly visibility and representation, in the options tree on the left, select **Reinforcement**, and go to the **Rebar assembly** tab.

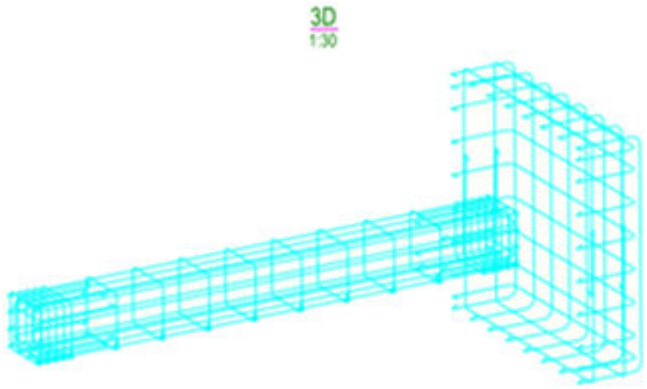
   The **Rebar assembly** tab exists in drawing view level reinforcement properties for cast unit, assembly, and general arrangement drawings.

   - **Visibility of rebar assemblies**: Select **Visible** or **Not visible**.
   - **Representation**:

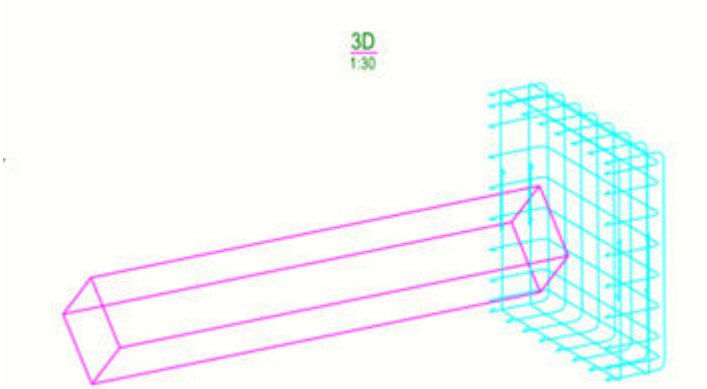
     Outline shows the rebar assembly and all its sub-assemblies as outline boxes.



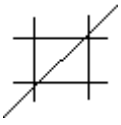
     As individual rebar objects shows all rebar objects in the rebar assembly and sub-assemblies.

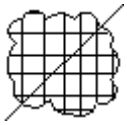


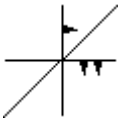
     Sub-assemblies as outline, highest level as individual rebar objects shows all sub-assemblies as outline boxes and the highest level of the rebar assembly as individual rebar objects.



- **Color**, **Type**: Set the rebar assembly line color and type.
- **Mesh symbol**, **Mesh symbol size**: You can choose to display a mesh symbol for the rebar assembly. Type a number of the symbol in the **Mesh symbol** box. The mesh symbol appears in the middle of the diagonal line. You can also change the mesh symbol size.

Symbol 1: 

Symbol 2: 

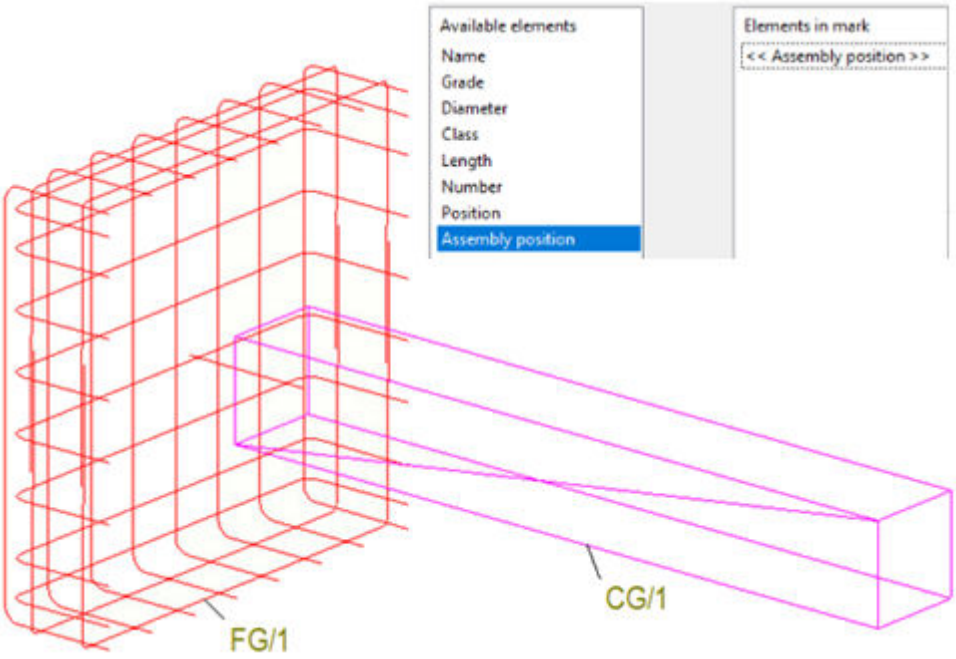
Symbol 3: 

4. Click **Modify**.

Add assembly position to reinforcement marks

The **Assembly position** element is available in reinforcement mark and associative note properties. It gives the position number of the associated rebar assembly.

1. To open the view-level properties dialog box, in an open rebar assembly drawing, double-click the frame of the drawing view that contains a rebar assembly.
2. In the options tree on the left, select **Reinforcement mark**.
3. To add the assembly position in the mark, in the **Available elements** list, double-click **Assembly position**.



4. Click **Modify**.

Filter view contents by using assembly template attributes

You can use the assembly template attributes `ASSEMBLY.HIERARCHY_LEVEL` and `ASSEMBLY.ASSEMBLY_POS` for filtering the rebar assembly drawing view contents.

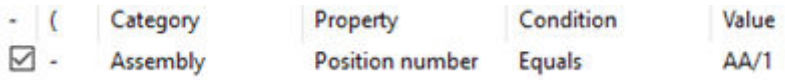
To create filters, in drawing view properties, click **Filter** in the options tree.

Examples:

- To show the desired assembly level, select **Template** as **Category**, type `ASSEMBLY.HIERARCHY_LEVEL` as **Property**, select **Equal**, and define the desired assembly level as the value.
- To show several levels, use the `ASSEMBLY.HIERARCHY_LEVEL` attribute, select **Greater or equal** and type the number of the lowest level that you want to show as the value.
- To show or hide the rebar assemblies you do not want to see, use `ASSEMBLY.ASSEMBLY_POS`, select **Equal** or **Does not equal** and type the assembly position number.
- The following example filter shows the hierarchy level 3 assemblies that have the position number AA/1:

-	(	Category	Property	Condition	Value
☑	-	Template	ASSEMBLY.HIERARCHY_LEVEL	Equals	3
☑	-	Template	ASSEMBLY.ASSEMBLY_POS	Equals	AA/1

- You can also create a drawing view filter by selecting **Assembly** as **Category** and **Position number** as **Property** and then define the assembly position number as the value. With this filter the view shows assembly AA/1 (including all its sub-assemblies).



-	(	Category	Property	Condition	Value
☑	-	Assembly	Position number	Equals	AA/1

Show pours in drawings

You can show pour object geometry and pour breaks in general arrangement drawings. Pour drawings can be used for communicating the sequencing of the pours, and the properties of the pours and pour breaks. You can add marks and associative notes to pour objects, and associative notes to pour breaks.

How to enable pours

Options and functionality related to pours are only available if pours are enabled in the model. You can enable pours in a model by setting the XS_ENABLE_POUR_MANAGEMENT advanced option to `TRUE`. In the default environment, pours are only enabled in the (Concrete) Contractor role.

WARNING If the pours are enabled in the model, do not disable the pours using `XS_ENABLE_POUR_MANAGEMENT`, especially in the middle of the project. This may cause problems if you have drawings containing pours, and if you are sharing your model. The pours and pour breaks in the model and in the drawings may get invalid, and you may lose all pour-related modeling work.

Pour objects

The geometry of the pour objects is presented as monolithic concrete. The pour objects can be presented in plan, section and 3D drawings. Tekla Structures shows the pour object geometry in general arrangement drawings exactly like it has been modeled: overlaps and extra outlines vanish if the parts collide, have the same concrete material grade, have **Cast in place** as the **Cast unit type**, and have the same pour phase.

You can select whether you want to show pour objects or not. You can use different colors, line types and fills in different pour objects. You can also modify the pour object properties in an open drawing on object, view and drawing level. For more information about showing the pour objects, see Show pour objects, pour marks, and pour breaks in drawings (page 883).

Pour object marks

In pour object marks, you can show information related to pour objects, for example, pour number, pour type, material grade, planned pour start date and the name of the pour team. You can add automatic pour marks (page 809)

before creating the pour drawing, and add (page 301) and modify (page 335) pour object marks in an open drawing. You can also add associative notes (page 329) to pour objects.

Pour breaks

Pour breaks are shown in general arrangement drawings as they have been modeled. The pour breaks are represented by a symbol, which you can change by using the advanced option XS_POUR_BREAK_SYMBOL. The symbol scale and the spacing between the symbols follows the drawing view scale automatically.

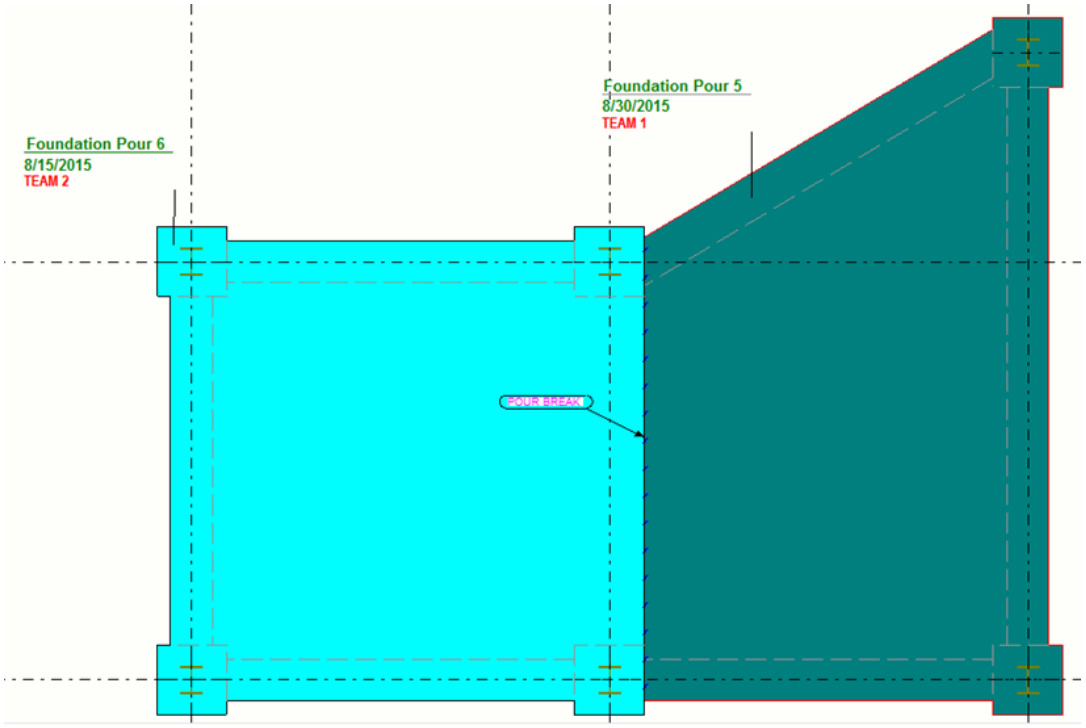
You can modify the pour break properties, and add associative notes (page 329) to the pour breaks in an open drawing.

Predefined pour drawing properties and templates

In the default environment, the Concrete Contractor role contains some predefined drawing properties, a couple of traditional report templates, and an organizer report template for pours. Your own environment may also contain some predefined drawing properties, and templates for pours. For examples of pour drawings and pour reports, see examples beloww.

Examples

In the image below, you can see two pour objects colored differently, each having a pour object mark. The pour break can be seen between the pour objects, it is represented by a symbol. The pour break is marked with an associative note.



Modify pour objects, pour marks and pour breaks in a drawing

After you have created a pour drawing, you can open it and modify the pour objects, pour marks and pour breaks.

Modify properties for the whole drawing

1. Open a pour drawing and double-click the drawing background to go to the drawing properties.
2. If you want so show the pours in the drawing, click the **View** button, check that **Show pours in drawing** is set to **Yes**, and click **OK**.
3. Click the **Pour object...** button to modify the properties:
   - **Content** tab: Select whether you want to see hidden lines, own hidden lines, edge chamfers, or fillet edges by selecting **On** or **Off**.
   - **Appearance** tab: Set the color and type for visible lines and hidden lines.
   - **Fill** tab: Select the fill for the pour object face and/or pour object section face.
4. Click **OK**.
5. Click the **Pour object mark...** button, select the contents and appearance of the mark, and click **OK**.
6. Click the **Pour breaks...** button, check that **Visibility** is set to **Visible**, and click **OK**. You can also select whether you want to show pour break hidden lines on the **Content** tab. On the **Appearance** tab, you can change the color and type of the visible and hidden lines in pour breaks.
7. Modify the other properties as required. For example, click **Reinforcement...** and set the **Visibility of all reinforcing bars** to **Visible** to show the reinforcement in the pour drawing.
8. Click **Modify** to apply the changes in your pour drawing. The changes are applied to all pour objects, pour marks and pour breaks in the drawing.

   Modify properties for individual objects

   - To modify individual pour objects, pour breaks or pour marks in a drawing, click the individual pour object, pour break, or pour mark in the drawing. This opens the properties in the property pane. If you do not have the property pane open, you need to double-click the object, break, or mark. You can apply your changes by clicking **Modify**.

Change the pour break symbol

If you want to use another pour break symbol, you change the symbol by using the advanced option `XS_POUR_BREAK_SYMBOL`.

1. On the **File menu**, click **Settings** --> **Advanced options** and go to the **Drawing properties** category.

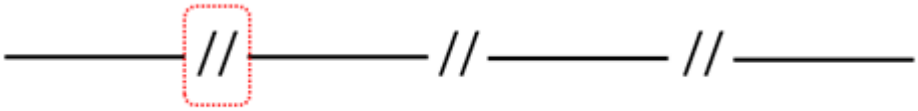
2. Set a new value for the advanced option `XS_POUR_BREAK_SYMBOL`.

   The default value is `PourBreaks@0`. The value refers to the `PourBreaks.sym` file where the symbol is defined. The symbol value starts with the symbol library file name (`PourBreaks`) and ends with the number of the symbol (`0`). The default symbol file is located in `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\symbols`.

   You can also create a new symbol file containing a new symbol, and save it. Then define the new `.sym` file for `XS_POUR_BREAK_SYMBOL`. If you wish to use a symbol file that is not located under your environment folders, enter the complete path to the symbol file location, the symbol file name and symbol number as the value for this advanced option.

3. Click **OK**.

Example

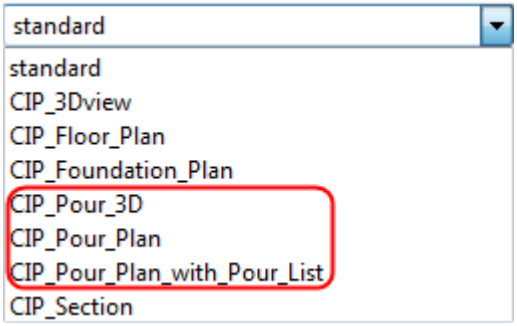


Examples of pour drawings and pour reports

The Concrete Contractor role in the default environment contains some predefined templates and settings for pour drawings and reports.

Examples of drawing properties predefined for pours

The Concrete Contractor role in the default environment contains predefined general arrangement drawing properties `CIP_Pour_3D`, `CIP_Pour_Plan`, and `CIP_Pour_Plan_with_Pour_List`. These drawing properties use detailed object level settings to color pour objects with different colors based on pour number. In addition, the drawing properties file `CIP_Pour_3D` does not render hidden lines, and `CIP_Pour_Plan_with_Pour_List` adds a pour list to the drawing.



Examples of reports predefined for pours

The Concrete Contractor role in the default environment contains predefined report templates for pour information:

- There are two traditional types or report templates that can be created using the Create reports command: `Pour_List` and `Pour_Schedule.csv`.
- The `Pour Organizer` report can be exported to excel format.

Show welds in drawings

Tekla Structures shows the welds that you have created in a model as weld seams and weld marks in drawings. In drawings, you can add intelligent marks to model welds both automatically and manually. You can also add drawing weld marks manually, but these marks do not necessarily have an associated physical weld in the model.

How welds are displayed in drawings

Tekla Structures shows the welds that you have added in a model as weld seams and weld marks in drawings. You can also add weld marks manually in an open drawing.

Weld concepts

Model welds are displayed as *weld marks* and *welds* or *weld seams* in drawings. Welds and weld marks can be controlled separately. For example, you may want to show the welds in one drawing view and the weld marks in another.

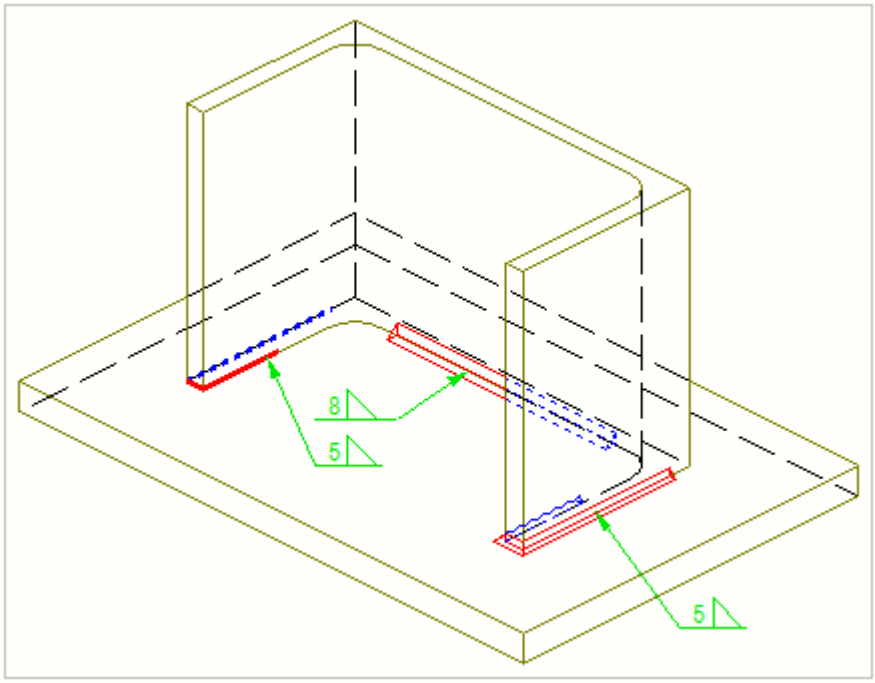
3D model welds are representations of welds in the real world. *Weld solids* in drawings are representations of the welds in models. *Weld seam* is the part of the weld path where the weld solid is drawn. *Weld path* may consist of several weld seams.

Weld solids are shown in drawings for the following welds:

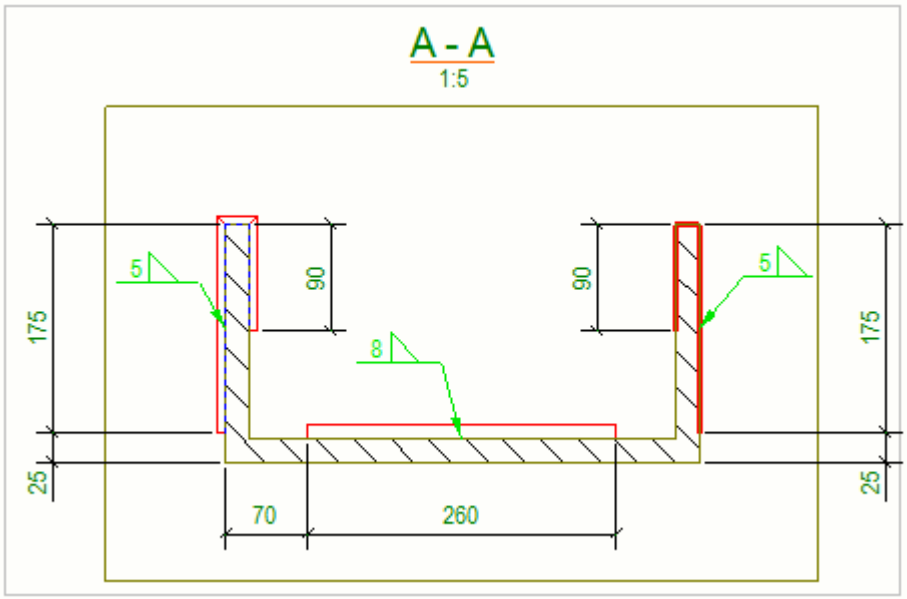
- Weld solids are shown in drawings for those weld types that have real solid support. Welds that have no real solid support are shown in the model with a hexagonal placeholder, and in drawings weld solids are not shown.
- Custom cross-section welds are also supported.

Weld solids can be shown as outlines or paths, with or without hidden lines.

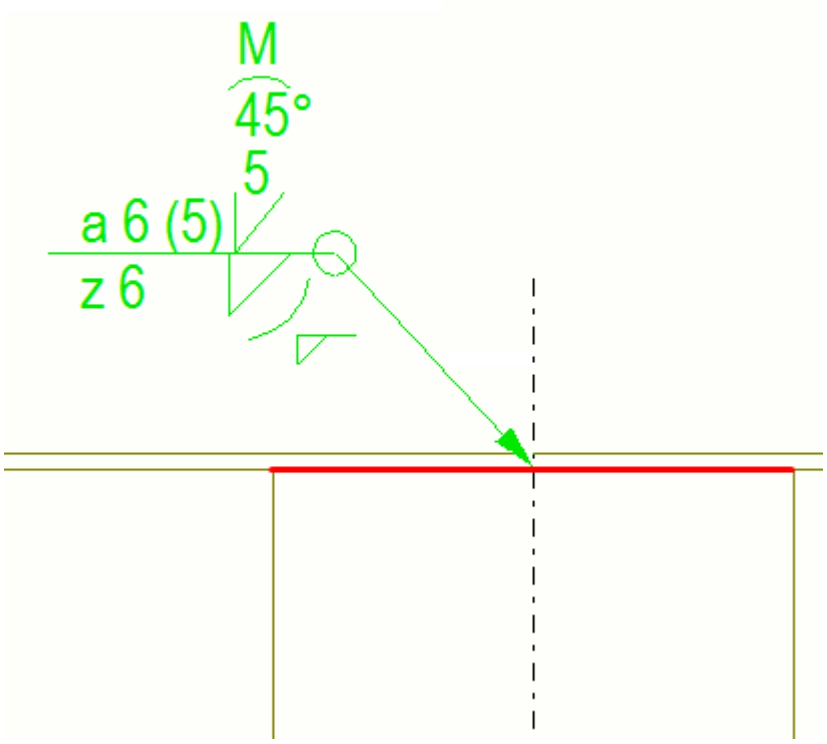
In the first example, the welds on the right and in the middle are drawn with outlines and own hidden lines. The weld on the left shows the weld path and hidden lines:



The second example is a cross-section of the structure. The welds on the left and in the middle are drawn with outlines, and the weld on the right shows the path. The weld dimensions have been added manually.



The *weld symbols* inside the weld marks indicate the weld properties defined for the model weld in the model or for the drawing weld mark in the drawing. Below is an example of a model weld seam (in red) and a model weld mark (in green) in a drawing.



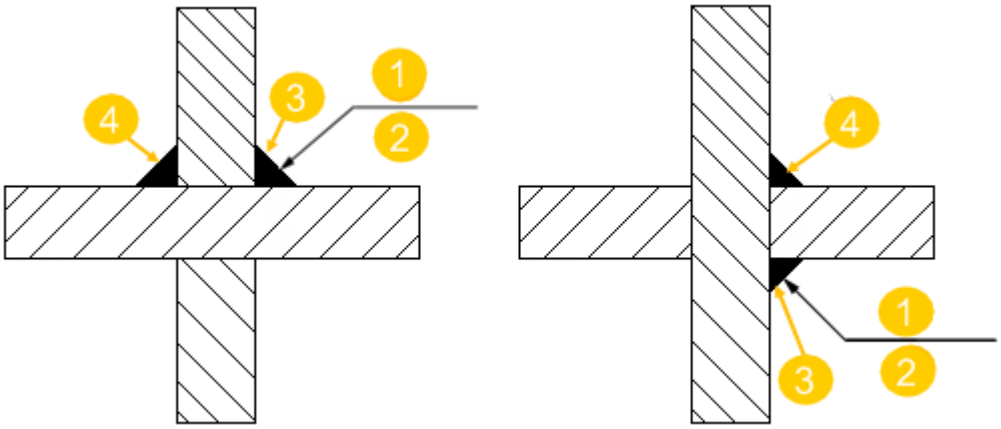
In addition to the weld symbols, the weld mark contains a reference line and an arrow. The arrow connects the reference line to the *arrow side* of a connection. The welds on the arrow and *other sides* of a part can have different weld properties.

Weld placement

When parts are welded together, you can place welds on:

- The arrow sides only
- The other sides only
- Both the arrow and other sides

The following images describe the basic placement principles of welds.



(1) Above line

(2) Below line

(3) Arrow side for weld

(4) Other side for weld

By default Tekla Structures places the welds above line 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`.

Model weld properties

To change model weld properties, you need to modify the weld in the model. When you update the model, the weld objects and weld marks are updated in the drawing according to the model changes. In drawings, you can modify the contents and the appearance of the model weld marks and the visibility, representation and appearance of model weld objects.

Weld solids can be shown in single-part, assembly, and general arrangement drawings.

Add drawing weld marks manually

You can add weld marks manually in an open drawing. Tekla Structures creates manual drawing weld marks using the current drawing weld mark properties. To add manual drawing weld marks, you do not need to have a model weld present in the drawing.

1. In an open drawing, go to the **Annotations** tab, hold down **Shift** and click **Weld mark**.
2. Modify the content and the appearance of the drawing weld mark.
   - In **Edge/Around**, indicate whether only one edge of a face should be welded, or the entire perimeter.
   - In **Workshop/Site**, indicate where the weld should be made.
   - Set **Stitch weld** to **Yes** to create a staggered, intermittent weld. **No** creates a non-staggered intermittent weld.
   - Enter or select the drawing weld mark properties that you want to include in the weld mark under **Above line** and **Below line**.

     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.

     The middle button is blue when the values are linked. This means that if you change a value in either of the columns, the corresponding value in the other column also changes.

- Set the font and mark background properties and adjust the leader line type and color.
- To exactly place the weld mark in the position that you pick and keep it there, set **Placing method** to **Fixed**. **Free** allows Tekla Structures to search for the first suitable location for the weld mark.
- In **Position**, define the areas where Tekla Structures searches for a position to place the weld mark.
- In **Distance s**, define the empty margin that you want to leave around the mark.
- In **Distance d min**, define the minimum distance of the mark from the part.

For more information about drawing weld mark properties, see Drawing weld mark properties (page 991).

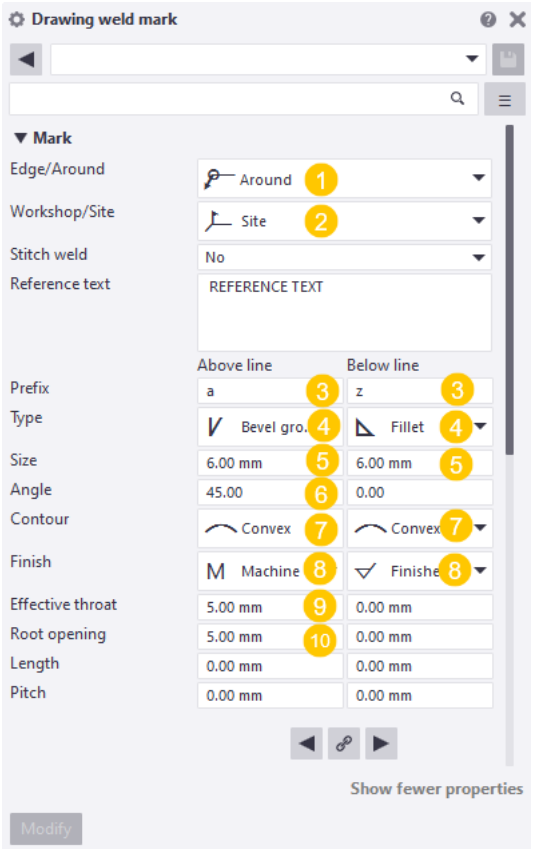
3. Pick a position for the drawing weld mark.

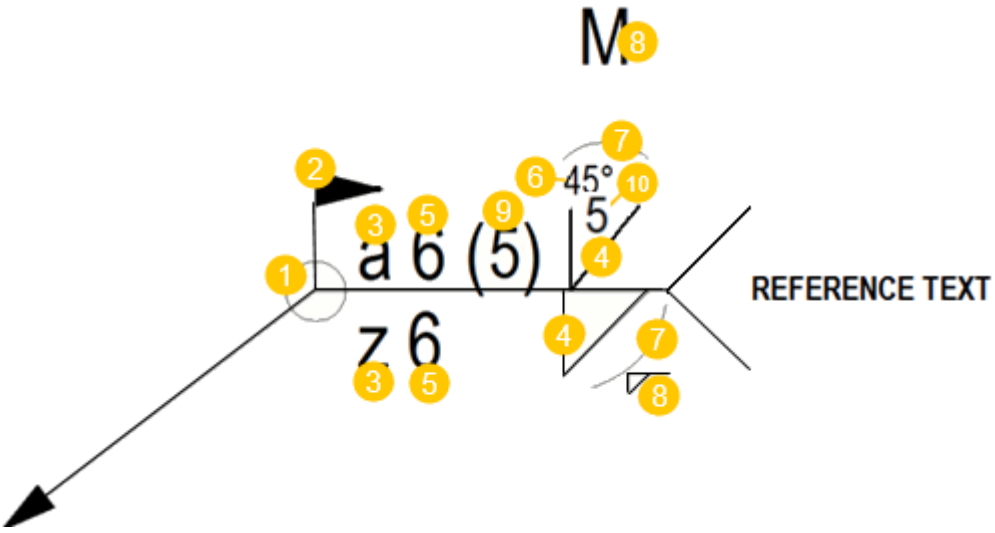
   The drawing weld mark is created. You can drag the mark to a more suitable location by the leader base point handle.

Example: Drawing weld marks added in drawings

In this example, the first image below shows the drawing weld mark properties in the drawing property pane. The properties are numbered in the image. The

second image shows a drawing weld mark in a drawing, and how and where the properties are displayed in the mark.





(1) Edge/Around, here a weld around symbol is used

(2) Workshop/Site, here a site weld symbol is used

(3) Weld prefix

(4) Weld type

(5) Weld size

(6) Weld angle

(7) Weld contour symbol

(8) Weld finishing symbol

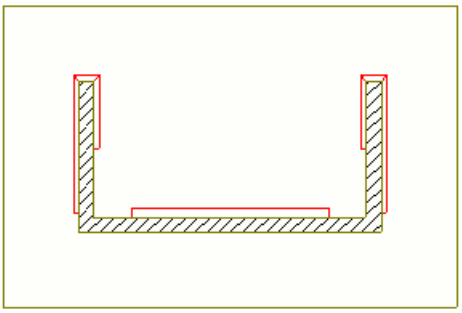
(9) Effective throat

(10) Root opening

Add model weld marks manually

You can add marks to model welds in an open drawing, for example, if you have not created those already at the drawing creation. To add model weld marks manually, you need to have a model weld present in the drawing. Tekla Structures creates model weld marks using the properties defined for the model weld in the model. You can also adjust the visibility and appearance of the model weld marks in the drawing.

1. Open a drawing that contains welds created in the model.
2. Do one of the following:
   - Select a model weld in the drawing.

   - Select several model welds in the drawing using the **Select welds** selection switch and area selection, or the selection functionality in **Drawing content manager**.

   If you cannot see any welds in the drawing, adjust the weld visibility settings in drawing view properties (page 1036). Also check the section "What if welds or weld marks are not visible in a drawing" below.
3. Right-click and select **Add weld mark**.

   The weld mark is created.

   Note that when a weld mark is added through the context menu, it is visible even if its size is below the minimum weld size limit defined in the drawing view properties.
4. To modify the model weld mark properties, do one of the following:
   - Click a model weld mark in the drawing. If the property pane is not open, double-click the weld mark.

- Select several model welds using the **Select welds** selection switch and area selection, or **Drawing content manager**. When the welds are selected, right-click and select **Select weld marks** and **From current drawing view** or **From all drawing views**. Activate all selection switches again . Then press down **Shift** and double-click a weld mark.

5. Modify the content and the appearance of the weld marks:
   - Activate the eye buttons next to weld properties to show the properties in model weld marks.

     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.

     The middle button is blue when the values are linked. This means that if you change a value in either of the columns, the corresponding value in the other column also changes.
   - Set the font properties, adjust the leader line type and color, and select if you want to use an opaque or transparent background in the mark.
   - In **Distance s**, define the empty margin that you want to leave around the mark.
   - In **Distance d min**, define the minimum distance of the mark from the part.

   For more information about model weld mark properties, see Model weld mark properties in drawings (page 996).
6. Click **Modify**.

The changes are applied.

You can drag the weld marks to another location. Note that you cannot drag the mark away from the associated weld, but you can only drag it along the weld seam. For more information, see section "Drag weld marks" below.

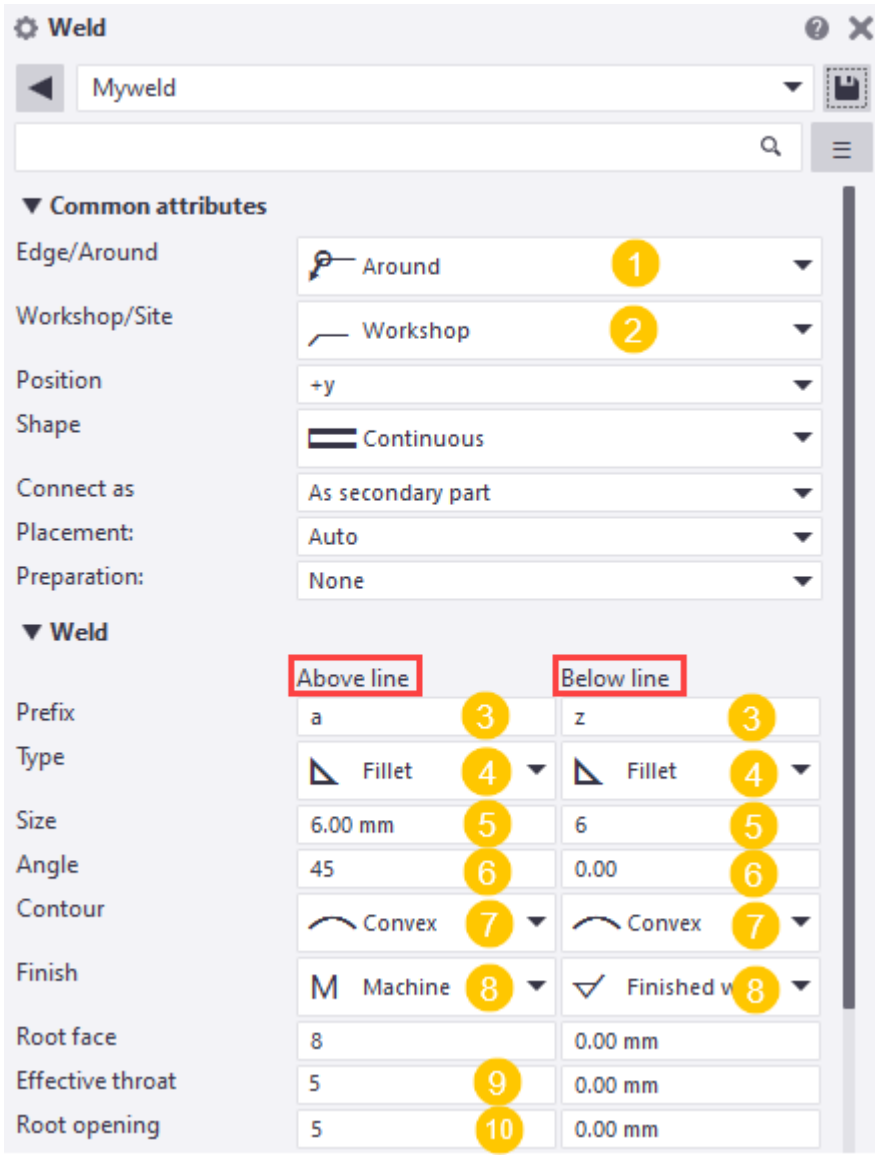
Examples: Model weld marks in drawings

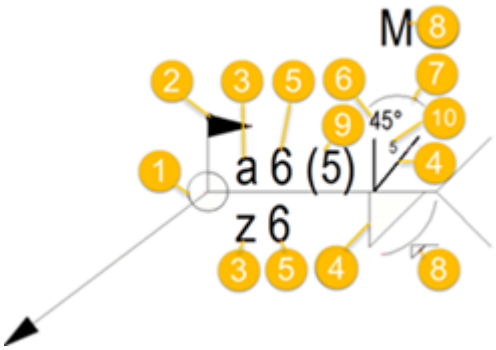
Model welds are welds that you have added in the model. In drawings, they are represented by weld seams and weld marks. See some examples below representing the weld properties in the model and what the weld marks look like in drawings.

Example 1

In the first example, the first image below shows an example of weld properties in the model. You can add welds in the model by selecting one of

the welding commands on the ribbon **Steel** tab. Some of the weld properties are numbered in the image, and the second image shows how these properties are shown in a weld mark in a drawing.





(1) Edge/Around, weld around symbol is used

(2) Workshop/Site, site weld symbol is used

(3) Weld prefix

(4) Weld type

(5) Weld size

(6) Weld angle

(7) Weld contour symbol

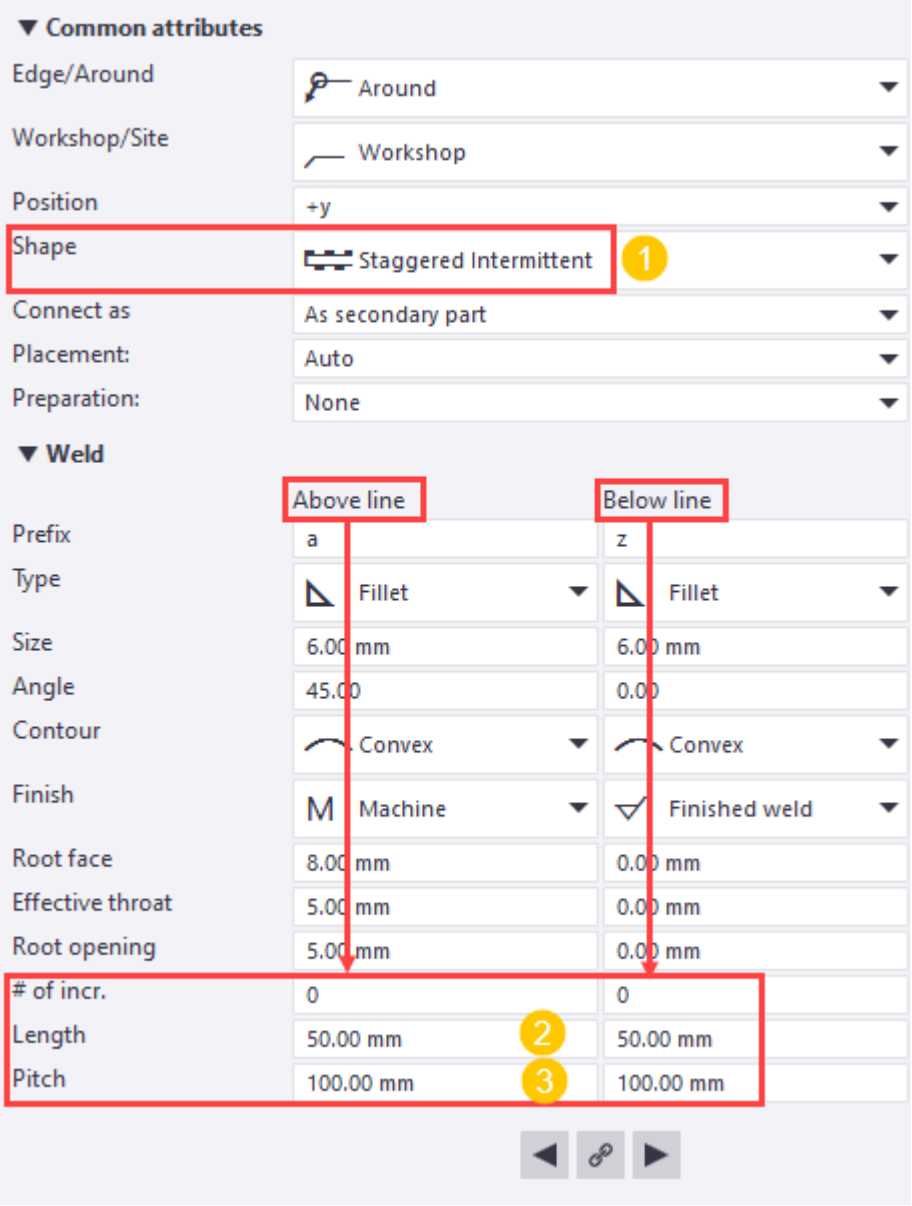
(8) Weld finishing symbol

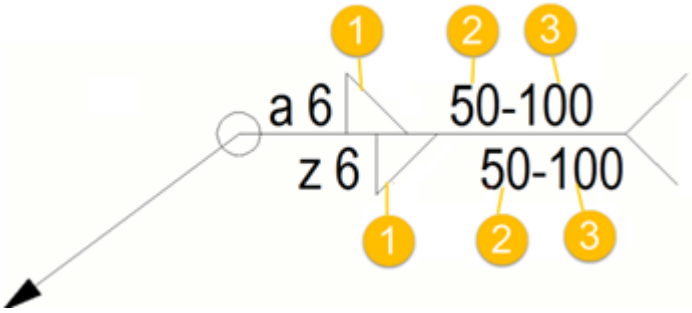
(9) Effective throat

(10) Root opening

Example 2

The example below shows a staggered, intermittent weld. The length is set to 50 and the pitch to 100.





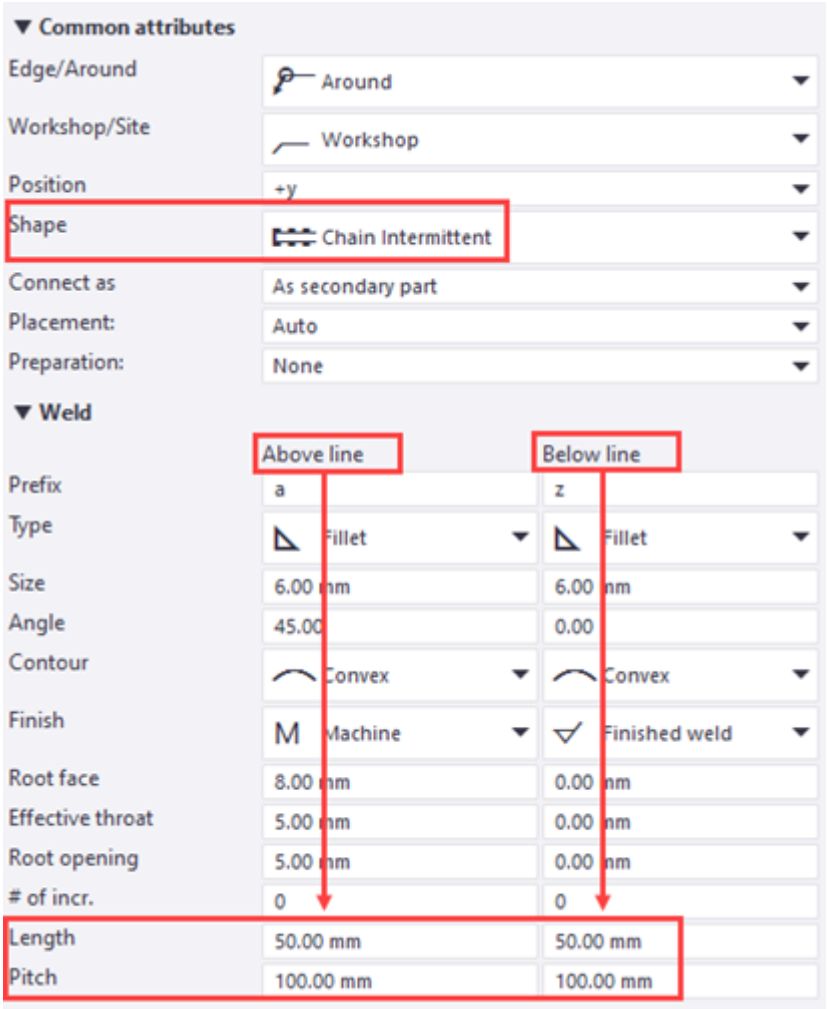
(1) Staggered, intermittent weld

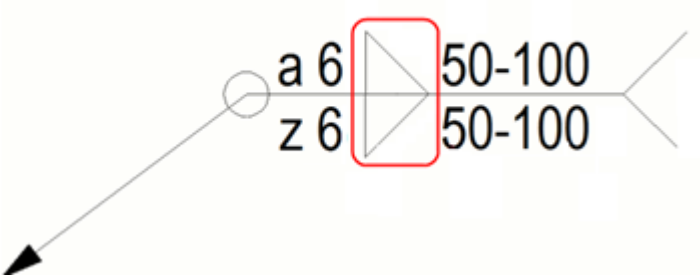
(2) Length of weld segment

(3) Pitch (center-to-center spacing) of weld segments

Example 3

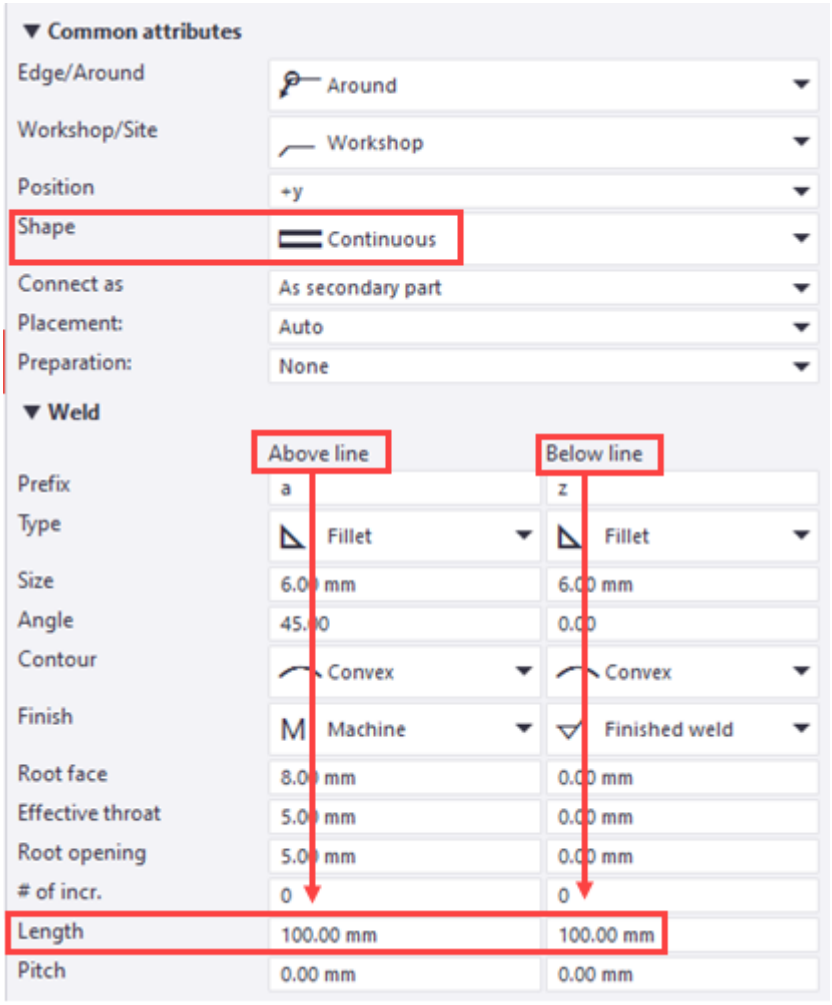
The example below shows a non-staggered, intermittent weld. The length is set to 50 and the pitch to 100. The pitch is shown in the weld mark when the pitch value is greater than 0.0.

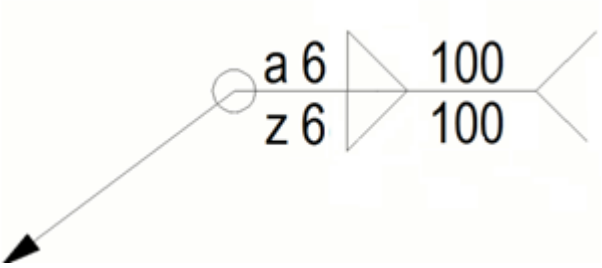




Example 4

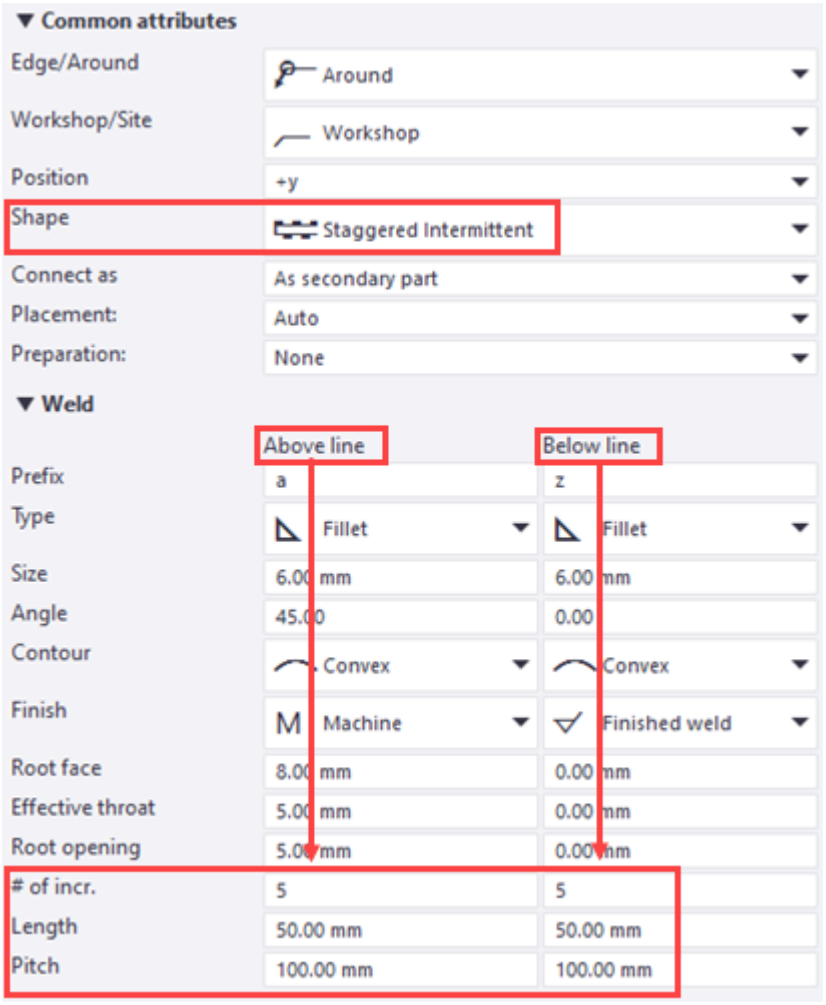
Below is an example of a continuous weld.

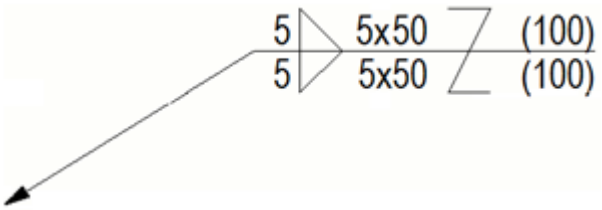




Example 5

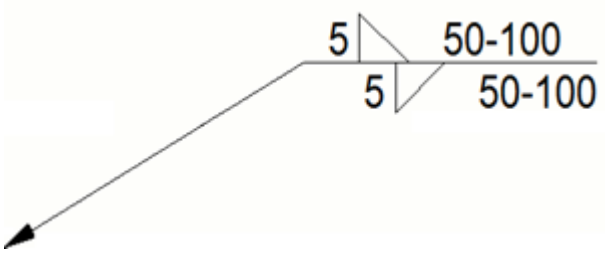
In this example, the staggered, intermittent weld option is selected, and the advanced option `XS_AISC_WELD_MARK` is set to `FALSE` to produce an ISO-compliant weld mark.





Example 6

In this example, the staggered, intermittent weld option is selected like in the previous example, but the advanced option `XS_AISC_WELD_MARK` is set to `TRUE` to produce an AISC-compliant weld mark.



TIP For instructions on how to customize weld symbols, see section "Customize weld type symbols" later on this page.

Modify model weld mark properties on view level in a drawing

Model weld properties are set in the model. At drawing creation, you can define the weld mark properties, including the mark visibility and appearance. You can also modify which welding properties you want to show in model weld marks, and adjust the appearance of the model weld marks in weld mark properties on the drawing level (GA drawings) and on the drawing view level.

1. To modify the weld marks on the view level, in an open drawing, double-click a view frame to open the **View Properties** dialog box.
2. To open weld mark properties, click **Weld mark** in the options tree on the left.
3. Select whether to show the weld number in **Weld number** (**No**/**Yes**).
4. Under **Visibility**:
   - In **Welds** and **Welds in sub-assemblies**, select what kind of weld marks to show, or whether to hide all welds (**None**, **Site**, **Workshop**, **Both**).
   - In **Welds in hidden parts**, select what kind of weld marks to show for hidden parts (**None**, **Site**, **Workshop**, **Both**).
   - In **Weld size limit**, enter a weld size limit to filter weld marks of that weld size out of the drawing.

     Even though you set the weld size limit, weld marks are always shown if they have reference texts.

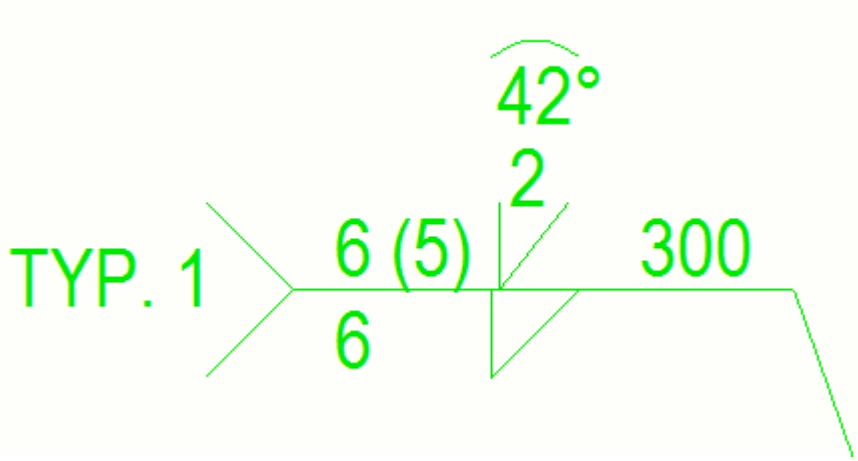
     Note that you can also set the **Weld size limit** for all views at one go at drawing creation at the bottom of the **View creation** page in single-part and assembly drawing properties (page 906).
   - Note that the **Weld mark visible** setting controls the visibility of model weld marks in assembly drawings at drawing creation. This setting is located at the bottom of the **View creation** page in assembly drawing properties (page 906). You can set **Weld mark visible** to **In one view** or **In all views**.

5. Under **Above line**, **Below line** and **Other**, clear a check box in the **Visible** column next to a weld mark property that you do not want to show in the weld mark. Note that if you hide **Size**, **Prefix** is also hidden, and when you hide **Length**, **Pitch** is also hidden.
6. To adjust the placing properties, click **Place...**.
7. Go to the **Appearance** tab and modify the weld mark text and line appearance.
8. Click **Modify**.

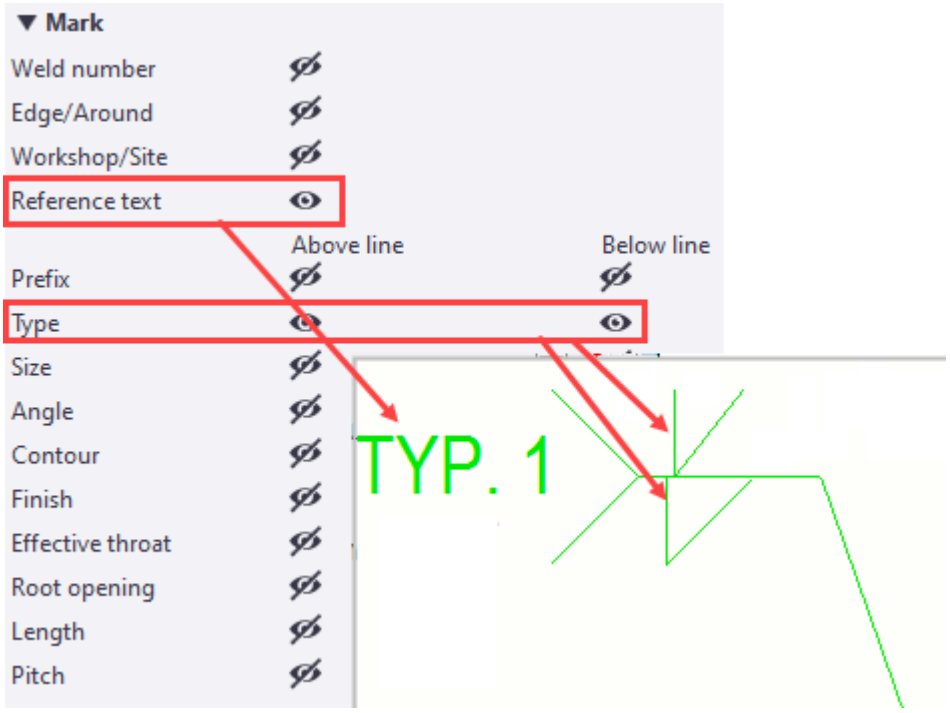
TIP To modify the properties of individual model weld marks in the property pane, in an open drawing, click a model weld mark. If the property pane is not open, double-click the model weld mark. To select several marks, for example for deleting, right-click the weld mark and select **Select weld marks** and **From current drawing view** or **From all drawing views**.

Examples

The first example shows a model weld mark where a lot of properties are visible:



In the second example, all other model weld mark properties have been hidden, except **Type** and , and **Reference text**:



TIP
- You can omit welds from drawings by weld type using the advanced option XS_OMITTED_WELD_TYPE. Then you first need to set a value for the **Weld size limit**.
- You can also customize some weld type symbols (page 489).
- For more information about defining automatic weld mark properties to be used at drawing creation, see Define automatic model weld properties in drawings (page 882).

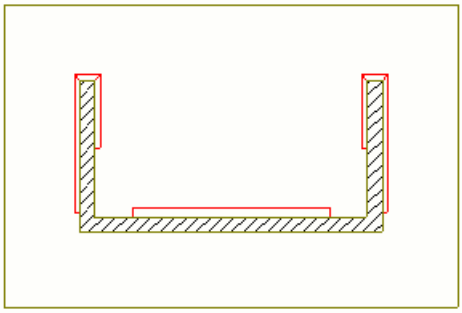
Modify model weld object representation and appearance in a drawing

You can modify the model weld object representation and appearance manually on object level.

1. In an open drawing, click a weld seam. If the property pane is not open, double-click the weld seam.

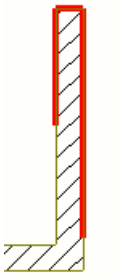
   It is easier to select the model weld if you only activate the **Select drawing welds** selection switch .

   In the following image, weld seems are shown in red:

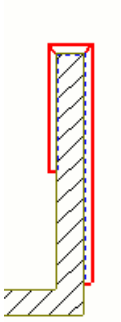


2. Select whether you want to show **Hidden lines** and **Own hidden lines**.
3. Select the desired **Presentation type**. The options are:

   Path



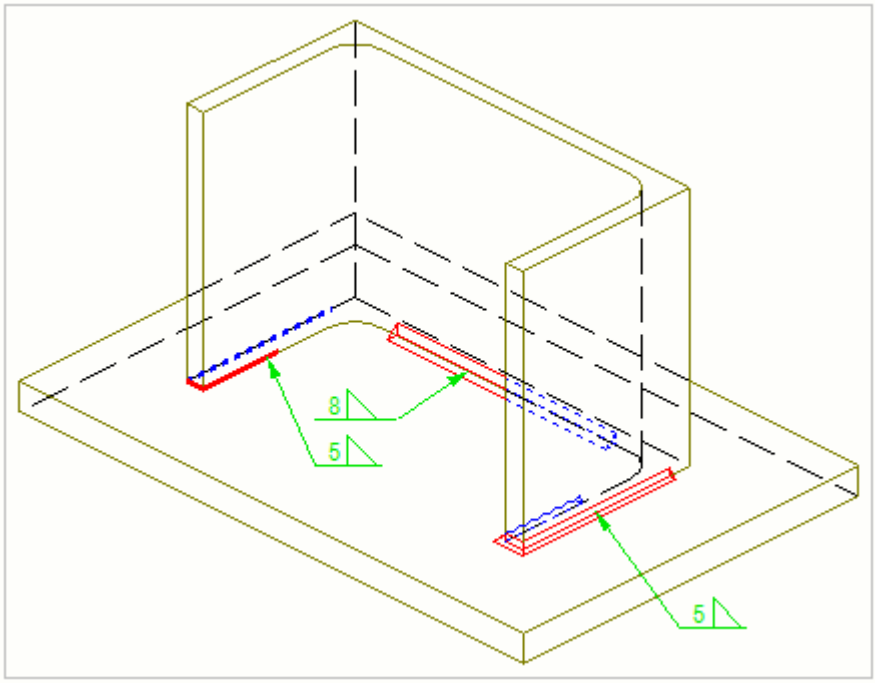
   Outline



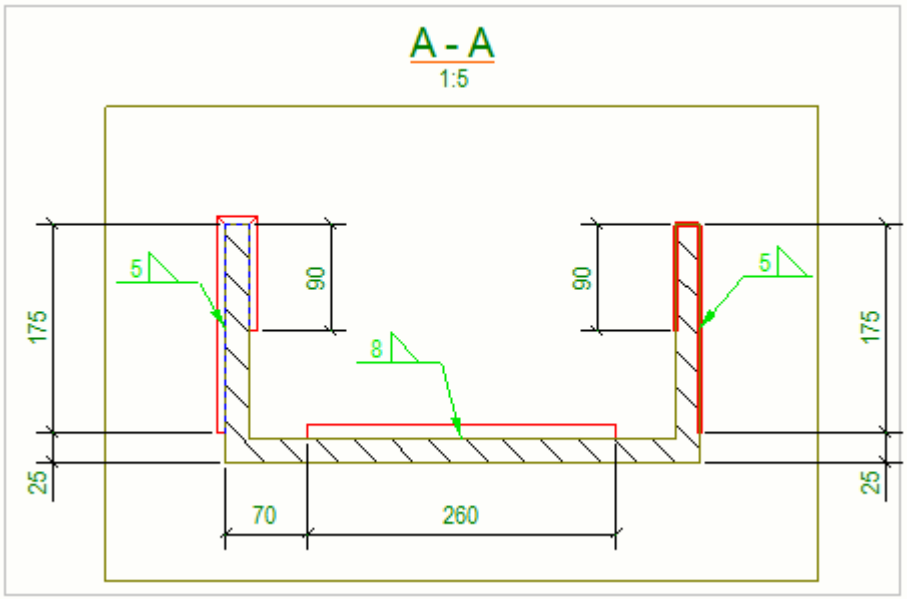
4. Modify the color and line type of the **Visible lines** and **Hidden lines**.
5. Click **Modify**.

Examples

In the first example below, the welds on the right and in the middle are drawn with outlines and hidden lines. The weld on the left shows the path and hidden lines.



The second example is a cross-section of the structure. The welds on the left and in the middle are drawn with outlines, and the weld on the right shows the path. The welds on the right and left go around the part corner. The welds have manual dimensions.



TIP You can define automatic weld properties (page 882) for model welds before creating a drawing. You can also modify the weld properties on view level in an open drawing by double-clicking frame of the drawing view containing the weld objects and selecting **Weld** in the options tree.

On drawing and view level, you can also modify the visibility settings (page 1036).

Drag weld marks

You can drag model weld marks by the associativity point of the weld mark leader line along the weld seam. This way you can position the weld marks more optimally for increased clarity in the drawings. Manual drawing weld marks not associated to model welds can be dragged freely.

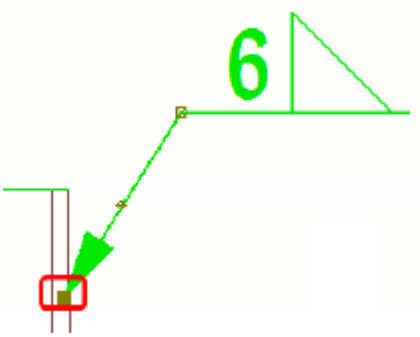
TIP Having **Drawing drag & drop**(**File menu** --> **Settings**) selected makes selecting the leader line base point and dragging it much easier.

Limitation: You cannot drag the weld mark leader base point to the back-side of a double-sided weld.

1. Click the weld mark near the leader line associativity point.

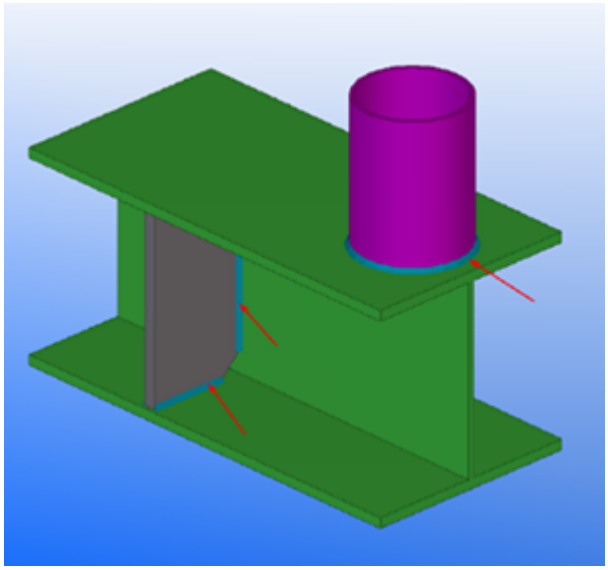
   If you have **Drawing drag & drop** selected, you do not need to click the leader line, pointing is enough.

2. Hold down the left mouse button and drag the associativity point to a new location by the leader line associativity point handle located in the tip of the arrow.

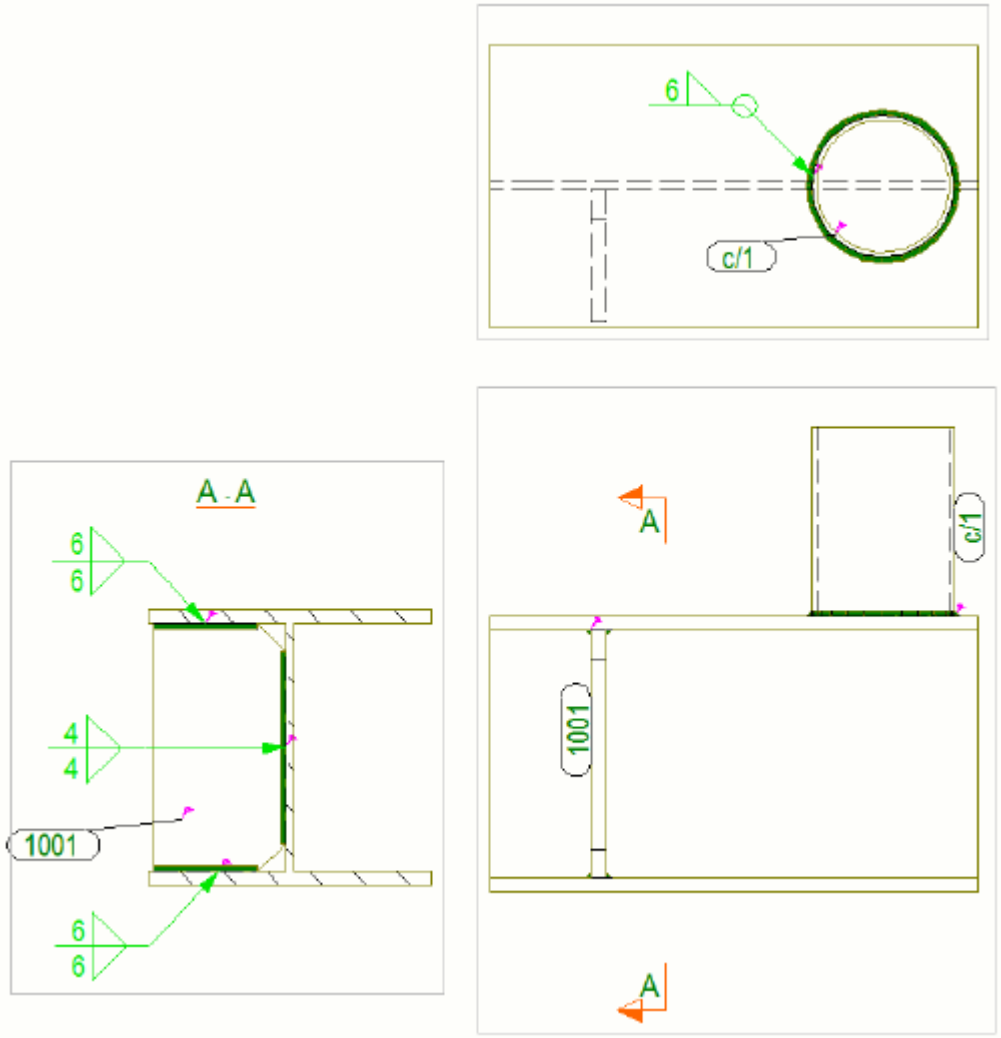


Example

The first image below shows the welds in the model.



The second image shows the model weld marks in a drawing. The area within which the weld mark leader line associativity point can be dragged is indicated with dark green.

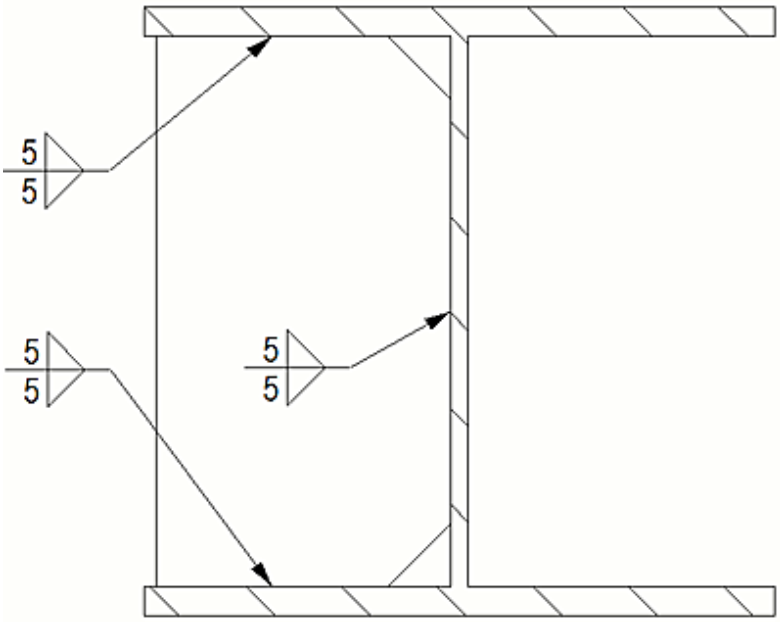
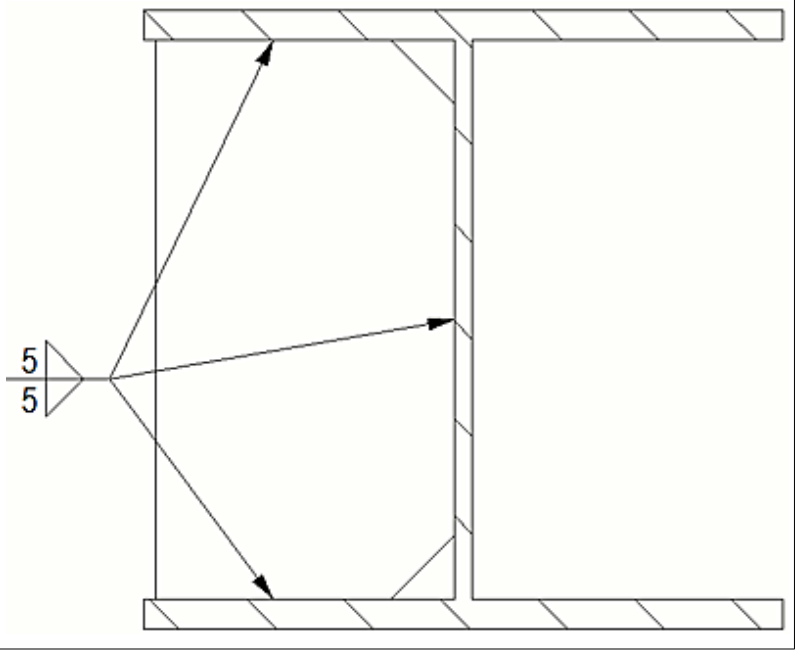


Merge weld marks

You can force Tekla Structures to use the same mark for identical welds in a drawing by merging the weld marks.

1. Open a drawing.
2. Hold down **Ctrl** and select the weld marks to merge.
3. Right-click and select **Merge** in the context menu.

   Tekla Structures combines the marks.
4. If needed, you can split merged weld marks by right-clicking the merged mark and selecting **Split** in the context menu.

Description	Example
Original drawing	
Merged weld marks.	

Customize weld type symbols

Most of the weld type symbols are hard coded, but you can edit some of them in Symbol Editor.

The bottom seven symbols (symbols 20 - 26) in the **Type** list in model weld properties and drawing weld mark properties (see image below) are taken from the `TS_Welds.sym` file. You can edit any of these seven symbols in Symbol Editor to create a custom weld symbol. The rest of the weld symbols

are hard coded. Note that the symbol in the **Type** list does not change when you update the symbol.

Melt through

Steep flanked single V butt

Steep flanked single bevel butt

Edge

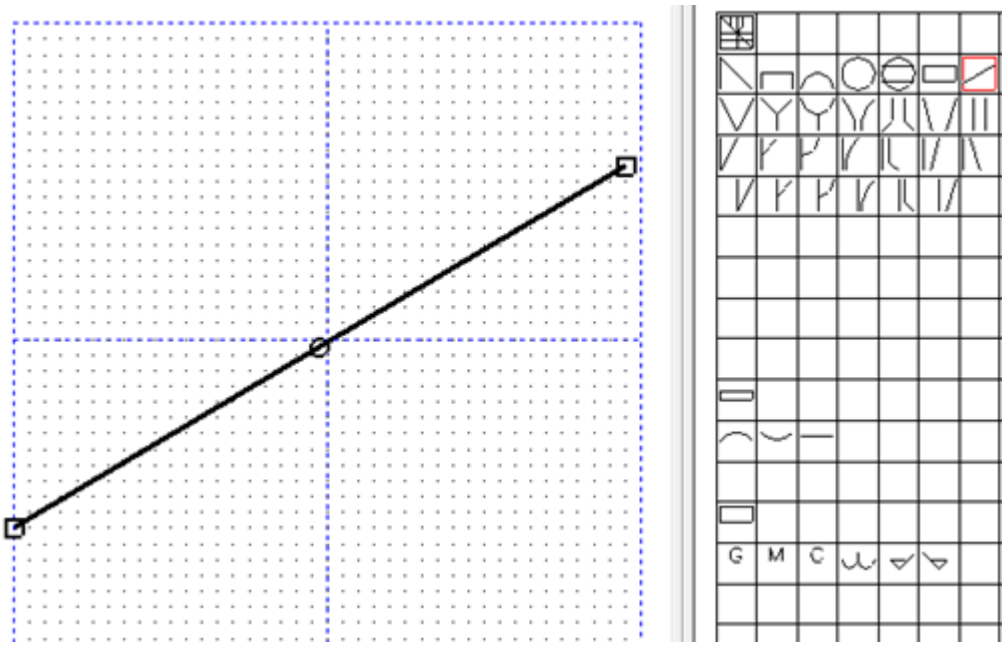
 ISO Surfacing

Fold

Inclined

1. Open Symbol Editor as administrator.
2. Click **File** --> **Open** and browse to the folder were you have the `TS_Welds.sym` file.
3. Select the file and click **OK**.
4. Modify the desired symbol.

   When doing this, you need to keep the symbol in the same scale as the other symbols. If your symbol is too large to fit in the box, you can let it extend beyond the borders:



5. Save the symbol by selecting **File** --> **Save**.

For more information about Symbol Editor, click the Symbol Editor User's Guide link on the PDF documentation page.

What if welds or weld marks are not visible in a drawing

If you cannot see model welds or weld marks in the drawing, check the following settings:

- Check the settings related to weld and weld mark visibility under **Common settings for all views** on the **View creation** page in drawing properties. To do this, open the drawing properties by double-clicking the drawing background and clicking **View creation** in the options tree on the left:
  - **Weld size limit**: Filters welds and weld marks of the defined weld size and smaller out of all views in a drawing. This setting is available in single-part and assembly drawings.
  - **Weld marks visible**: Controls the visibility of model weld marks in assembly drawings. The options are:
    - **In one view**: Tekla Structures automatically finds the most relevant view for showing the model weld marks. Every weld mark is shown in a drawing in one view only.
    - **In all views**: Tekla Structures adds the model weld marks in all drawing views that contain the part with the weld.
- Check the weld mark visibility settings in the view. To do this, double-click the view frame and click **Weld mark** in the options tree on the left:
  - Check the model weld mark visibility check boxes on the **Content** tab.
  - Check the values for the following: **Welds**, **Welds in sub-assemblies**, **Welds in hidden parts**, **Weld size limit**.
  - Check the line color settings on the **Appearance** tab.
- Check the weld visibility settings in the view. Double-click the view frame and click **Weld** in the options tree on the left:
  - Check the values for the following: **Welds**, **Welds in sub-assemblies**, **Weld size limit**
  - Check the line color settings on the **Appearance** tab
- Check the **Custom presentation** tab in weld and weld mark view properties.
- In drawing view properties, check that you have not set any view filters that prevent the welds or weld marks from showing. To do this, double-click the view frame and click **Filter** in the options tree on the left, and check if there are any active filters.

4.21 Show grids in drawings

You can show grids and grid line labels in single-part, cast unit, assembly, and general arrangement drawings. You can set automatic grid properties and also manually modify properties in an open drawing.

To	Click below
Modify grid properties and the properties of individual grid lines manually	Modify grids and grid lines in drawings (page 514) Drawing grid and grid line properties (page 1041)
Set automatic grid properties before you create the drawing	Define automatic grid properties (page 892)
Customize drawing grid labels by including extra text and symbols	Customize drawing grid labels (page 516)

Modify grids and grid lines in drawings

You can modify the properties of individual grids or grid lines in an open drawing. You can move grid labels in drawings by dragging them. This is useful if the grid label is covering an important area in a drawing. You can also hide grids and grid lines.

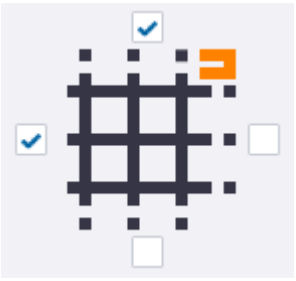
The grid and grid line selection switches are located at the bottom of the window.

Modify grid and grid line properties in drawings

You can modify properties of individual grids or grid lines in an open drawing, for example, hide the grid lines, change grid line and grid label settings, and select which labels are displayed.

1. Ensure that you have the correct selection switch selected. To modify grids, use the **Select grid** selection switch, and to modify grid lines, use the **Select grid line** selection switch.
2. Click the grid or a grid line. If the property pane is not open, double-click the grid or grid line. Tekla Structures displays grid properties or grid line properties, according to your selection.

3. The **Visibility** option **Visible** shows both grid lines and grid labels in the drawing. If you want to see the grid labels only, select **Only grid labels visible**.
4. Modify the grid line and grid label properties as required. For example, modify the grid line extension length, grid line color and type, grid label font properties and grid label frame type.
5. In **Placing**, select which grid labels you want to show.



6. Click **Modify**.

NOTE You can also set a fixed width for the grid label frames, a width factor for the grid label frames, or the label text font:

XS_GRID_TEXT_FONT

XS_DRAWING_GRID_LABEL_FRAME_FIXED_WIDTH

XS_DRAWING_GRID_LABEL_FRAME_LINE_WIDTH_FACTOR

Drag grid labels

Dragging grid labels is useful if the grid label is covering an important area in a drawing.

1. Ensure that you have the **Select grid line** selection switch selected.
2. Click a grid.
3. Hold down the left mouse button and drag the label by the handle to a new location.

Hide grids or grid lines

You can hide grids and grid lines if you do not want to have them visible in a drawing.

1. Ensure that you have the correct selection switch selected:

   To hide grids, use the **Select grid** selection switch, and to hide grid lines, use the **Select grid line** selection switch.

2. Right-click the grid or the grid line and select **Hide/Show** --> **Hide from drawing view** .

TIP If you want to show the hidden grids or grid lines, in an open drawing, press **B** until the color mode (page 61) is **Color**, right-click the grid or the grid line, and select **Hide/Show** --> **Show in drawing view**. You can see and select the hidden grids and grid lines only in **Color** mode.

You can also change the color mode in an open drawing through **File** --> **Settings** --> **Color mode**.

See also

Drawing grid and grid line properties (page 1041)

Define drawing grids (page 891)

Colors in drawings (page 61)

Customize drawing grid labels

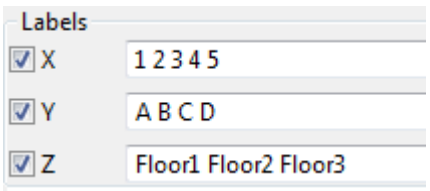
In GA drawings, you can customize drawing grid labels by including extra text and symbols in the labels. You can define the label text, offsets and offset prefixes in the user-defined grid attributes in the model and show the grid label text in drawings. You can also defined the prefixes in an open drawing. You can use a combination of traditional grid labels and custom labels, or only use custom labels.

Before you can customize grid labels, you need to modify the user-defined grid properties in the model, and also other grid properties, depending on your needs. You can also choose to modify the prefix text in the drawing.

Modify model grid properties

1. In the model, click the grid to open grid properties. If the property pane is not open, you need to double-click the grid.
2. Modify the grid properties as required.

   In this example, you will create grid labels for elevations, so you need to modify the **Labels** for Z direction.

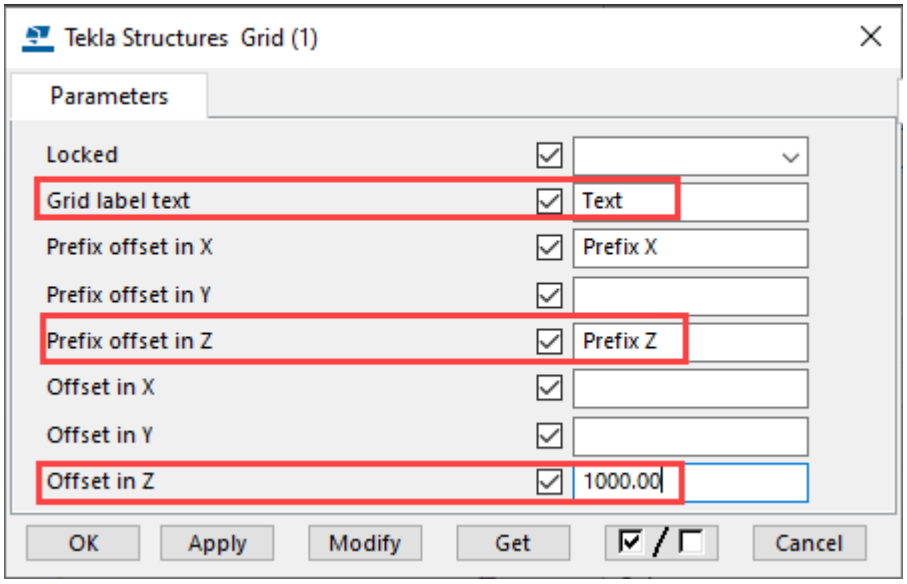


3. Click **User-defined attributes...**

4. Fill in the necessary user-defined attribute information.

   Note that you cannot leave the **Offset in Z** box empty even if the offset was 0. If the offset is 0, you still need to enter 0.

   In this example, you need to have the **Grid label text**, **Offset in Z** and **Prefix offset in Z** defined. You can also define the prefixes and the label texts in **Advanced grid label properties** in the drawing.



   Note that the **Datum level** in the drawing **View Properties** does not work in the same way as the **Offset in Z**. Setting **Offset in Z** in grid UDAs to 1000.00 gives the same result as setting the **Datum level** to -1000.00. The view **Datum level** setting moves coordinates, and level marks are relative to the coordinates. The **Datum level** setting does not move the content of the view, the content just becomes relatively moved in the opposite direction.

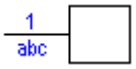
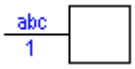
5. Click **OK** and click **Modify**.

Now you have created the necessary grid label texts and adjusted the needed values. Next, you can customize the drawing grid labels.

Customize drawing grid labels

1. Open a general arrangement drawing.
2. On the **Annotations** tab, hold down **Shift** and click **Grid labels**.
3. Define the properties for the custom grid labels:

Grid label placing	Select which grid labels are displayed by selecting the desired check boxes.
Use settings from	Define where the grid label properties are taken from:

	• **Grid** inherits the properties from the grid. Note that if you have defined settings for text color, height and font in the standard drawing **Grid properties**, the corresponding settings in **Advanced grid label properties** will changed accordingly, when you click **Modify**. • The **dialog box** option uses the settings defined in **Advanced grid label properties**.
Frame type	Select the type of the frame: Any symbol, grid label text inside, at the bottom . Any symbol, grid label text inside at the top . Any symbol with a leader line, grid label text under the leader line outside the symbol  . Any symbol with a leader line, grid label text above the leader line outside the symbol  Note that you can define a fixed size for the grid label frame by setting the advanced option XS_DRAWING_GRID_LABEL_FRAME_FIXED_WIDTH to a defined value. If you want to calculate the grid label frame size automatically, leave the value out.
File	Select a symbol file from the list.
Number	Click the **...** button and double-click the symbol. For example, the elevation symbol is by default symbol number 35 in the `xsteel.sym` file.
Color	Select the color and the line type of the grid label frame symbol.
Height	Set the height of the grid label frame symbol: **Enter height**: Enter the height in the **Height** box. **Autosize**: Adjust the symbol height automatically. **Enter Autosize minimum height**: Set the minimum height for the symbol.
Align to line	Select **Yes** to rotate the labels of vertical and inclined grid lines and align the labels with the grid lines.
Grid: **Number**	Define the **Prefix**, **Color**, **Height** and **Font** of the grid numbers. If you define the prefix in the user-

	defined attributes of the grid, you no longer need to do it here.
Grid: **Text**	Define the **Prefix**, **Color**, **Height** and **Font** of the grid label texts. If you define the prefix in the user-defined attributes of the grid, you no longer need to do it here.
Create axis text	Select **Yes** or **No**. Yes enables the options below. **No** is the default value. If you have defined prefixes and grid axis texts in the user-defined attributes of the grid, you not need to define them here.
Prefix for axis X	Define the prefix for the X axis.
Prefix for axis Y	Define the prefix for the Y axis.
Prefix for axis Z	Define the prefix for the Z axis.
Color	Define the color of the grid axis text.
Height	Define the height of the grid axis text.
Font	Define the font used in the grid axis text.

4. Save the properties for future use by entering a unique name in the **Save as** box and clicking **Save as**.
5. Click **OK** and pick a grid.

Tekla Structures customizes the grid labels and label texts according to the modifications that you made in the grid UDAs in the model, and in **Advanced grid label properties** in the drawing.

Note that if you get duplicate grid labels (traditional and custom), double-click the drawing grid and clear the check boxes of double grid labels in grid properties.

In the example below, the following properties were defined in the grid properties and user-defined grid attributes in the model:

Labels: **Z** = `Floor1 Floor2 Floor3`

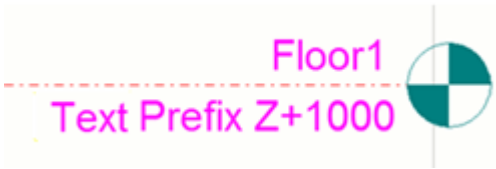
Grid label text = `Text`

Prefix offset in Z = `Prefix Z`

Offset in Z = `1000.00`

In **Advanced grid label properties**,

was selected as the **Frame type**.



Customize a grid label on a single grid line

1. Ensure that you have the **Select grid line** selection switch active .
2. In an open drawing, on the **Annotations** tab, hold down **Shift** and click **Grid labels** .
3. Define the properties for the grid labels (see instructions above).
4. Click **OK**, and pick a grid line. You can also select several grid lines using area selection.

Tekla Structures customizes the grid labels and label texts on the selected grid line according to the modifications you made.

Modify custom grid labels

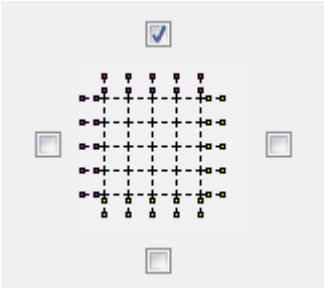
You can modify the custom grid labels by selecting the grid labels.

1. Ensure that you have the **Select grid** selection switch active .
2. In an open drawing, on the **Annotations** tab, hold down **Shift** and **Grid labels** .
3. Only select the check box of the grid label that you want to customize.
4. Define the properties for the custom grid label.
5. Click **Modify** and select the grid label.

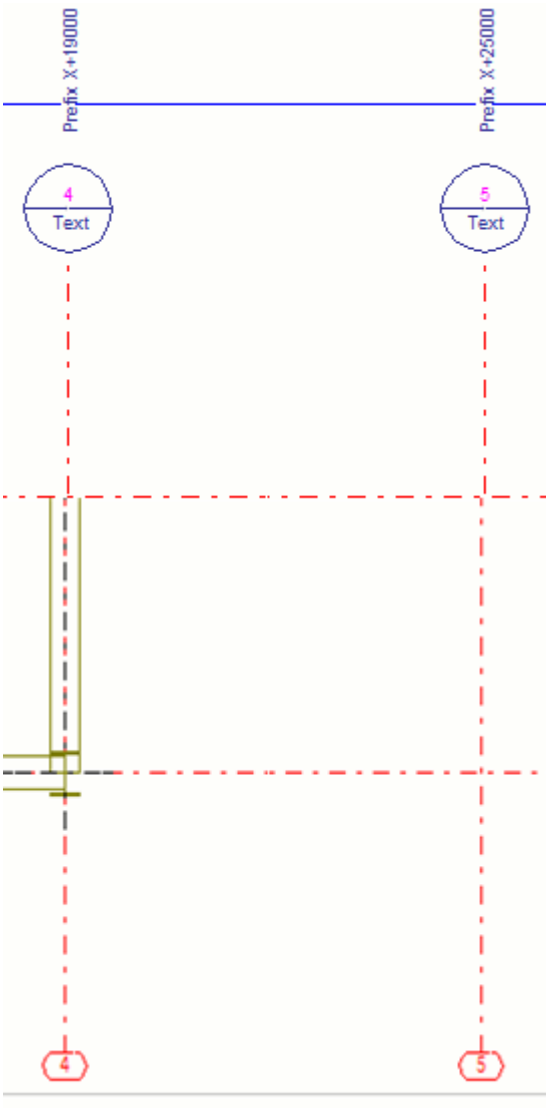
Customize grid labels only at one end of a grid line

You can have regular and customized labels at different ends of a grid line.

1. Ensure that you have the **Select grid** selection switch active .
2. In an open general arrangement drawing, on the **Annotations** tab, hold down **Shift** and click **Grid labels** .
3. Only select the check box of the grid label you want to customize.



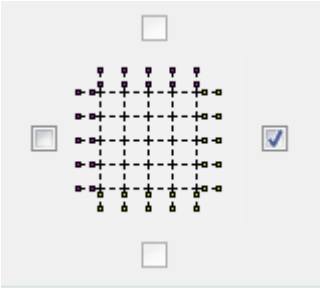
4. Define the properties for the custom grid label.
5. Click **OK** and select the grid.
6. Select the selection switch.
7. Select the desired grid lines, right-click and select **Properties...**, and only select the check box at the opposite end of the grid line. Area select is handy in grid line selection.
8. Click **Modify**. The custom grid label is displayed at the one end of the grid line, and the traditional grid label at the other end.



Add different customized grid labels on horizontal and vertical grid lines

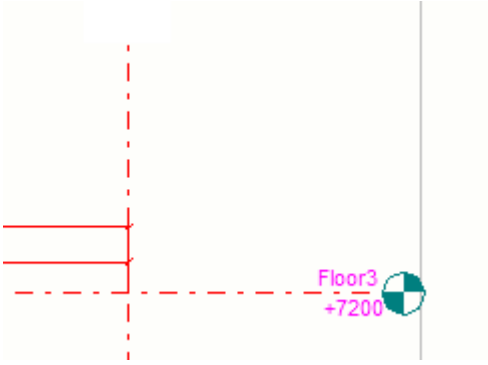
You can use different customized labels in horizontal and vertical grid line labels.

1. Ensure that you have the **Select grid** selection switch active .
2. In an open drawing, on the **Annotations** tab, hold down **Shift** and click **Grid labels**.
3. In the **Advanced grid label properties** dialog box, only select the check box for the right horizontal grid labels.



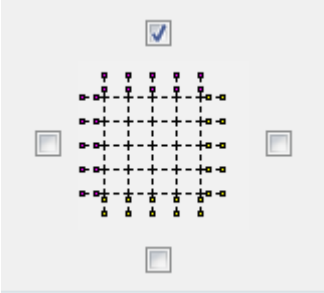
4. Define the properties for the horizontal grid labels. For example, for elevation grid label, set up an elevation symbol.
5. Click **OK** and select the grid.

   Custom grid labels are added on the horizontal grid lines. There are no grid labels on the vertical grid lines.



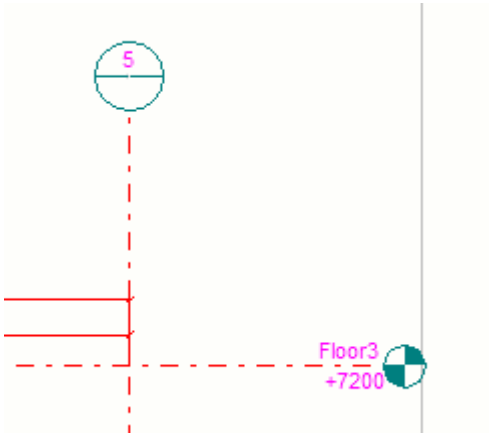
6. Hold down **Shift** and click **Grid labels**.

7. In the **Advanced grid label properties** dialog box, only select the check box for the upper vertical grid labels.



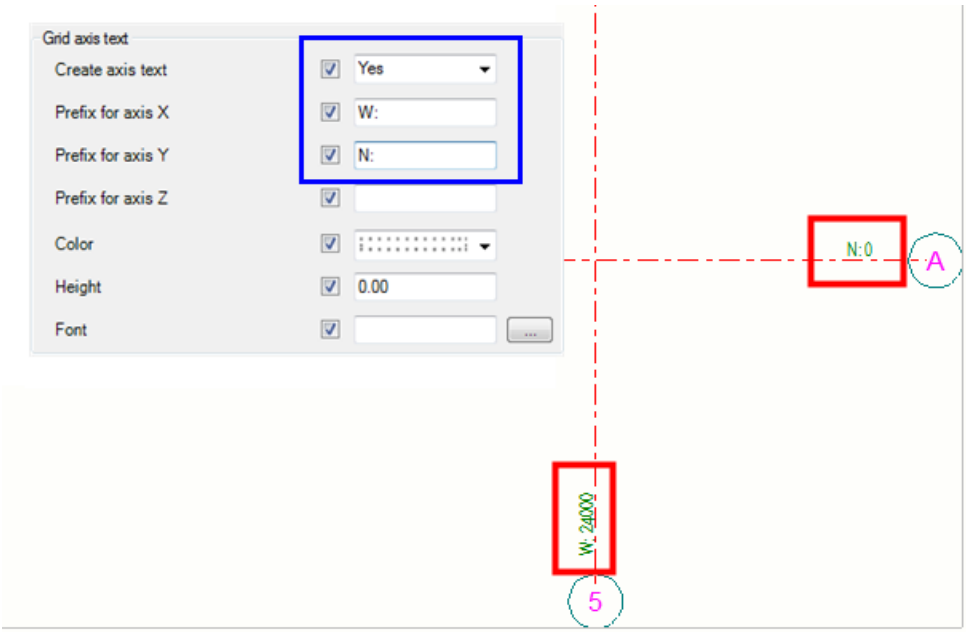
8. Define the properties for the vertical grid labels. For example, select the desired frame label type.
9. Click **OK** and select the grid line.

Custom grid labels are added on the vertical grid lines of the selected grid. The horizontal grid lines have the labels that you added earlier.

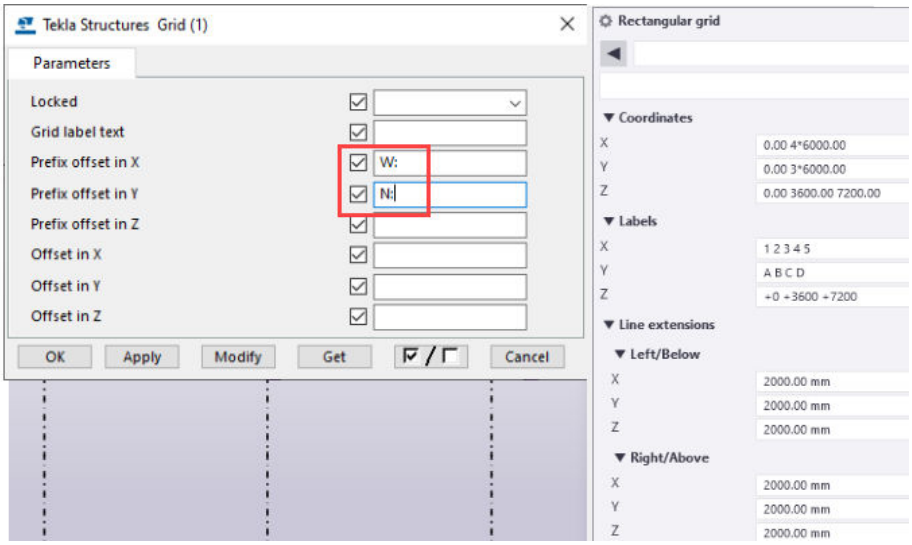


Use model grid coordinates and prefix as text on grid axis in a drawing

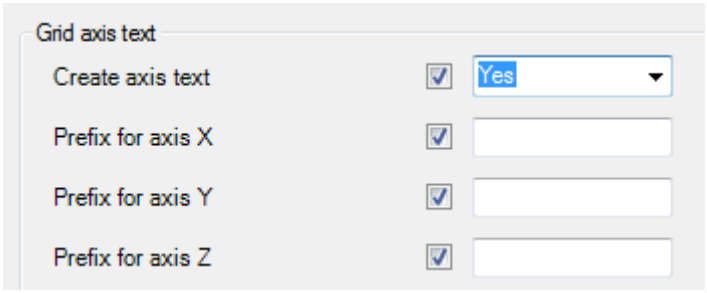
1. On the **Annotations** tab, hold down **Shift** and click **Grid labels**.
2. Do any of the following:
   - Define the following settings to show grid coordinates automatically along grid lines:

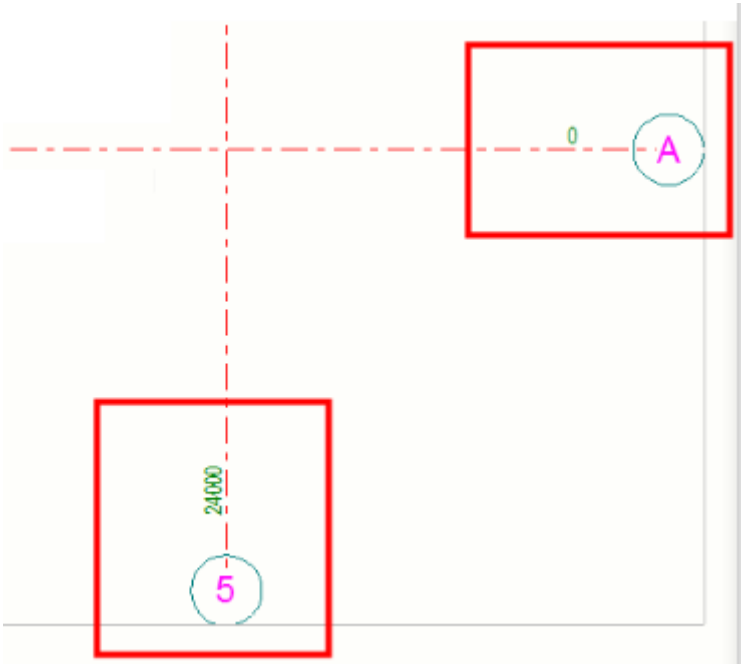


- You can also do this in another way and use different prefixes:
  a. First in the model grid user-defined attributes, add in **Prefix offset in X** and in **Prefix offset in Y** values W: and N: in the following way:



  b. Then in the drawing, in the **Advanced grid label properties** dialog box, set **Create axis text** to **Yes**:

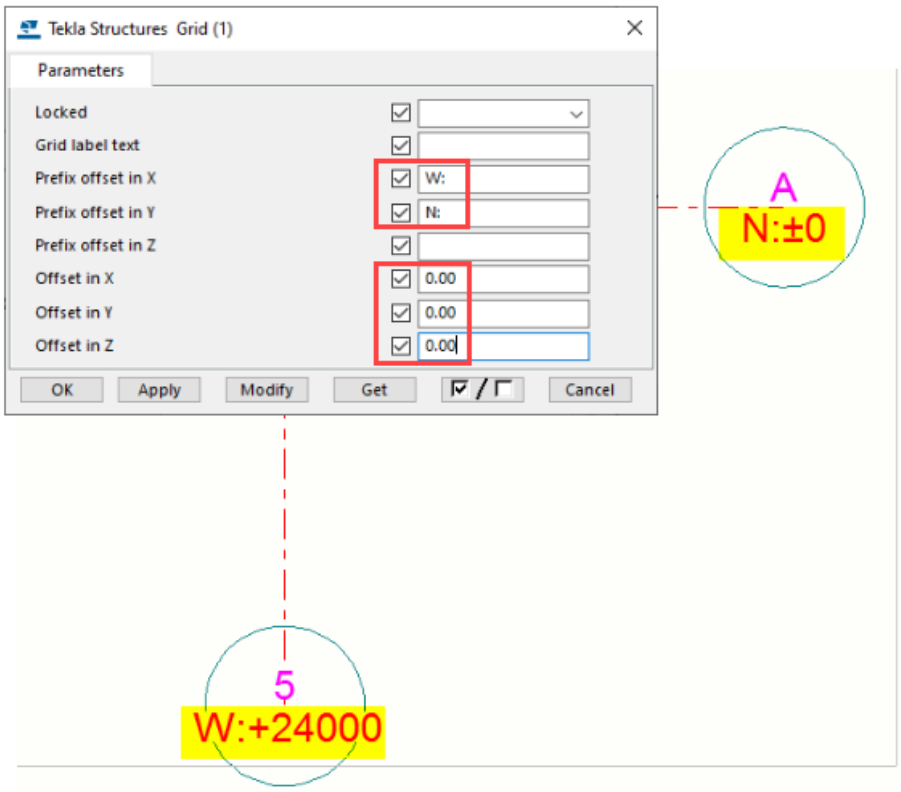




If you want to write prefixes to these values, in **Advanced grid label properties**, add the **Prefix for axis X** and **Prefix for axis Y** information.

c. To get model grid coordinates shown in the drawing grid labels automatically, in the model, go to user-defined grid attributes, and change the offset values to 0.

When you do this, go to drawing **Advanced grid label properties**, and set **Create axis text** to **No**.



Limitations

- The grid lines are not refreshed automatically.
  - If you have hidden grid lines or otherwise modified them, you need to click **Modify** in **Advanced grid label properties** to refresh the grid labels.
  - If you drag grid line handles, the custom grid labels are not moved together with the handle, until you click **Modify** in **Advanced grid label properties**.
- Advanced grid labels are not supported in multidrawings, key plans or in cloning, or when moving views to another drawing.
- Not all frame types are allowed for all axises.

4.22 Show reference objects in drawings

Reference models or reference objects can be shown in general arrangement, assembly, and cast unit drawings. You might want to use 3D plant models or architectural drawings as reference objects, for example.

You can select whether to show a reference object, and select if the hidden lines or own hidden lines are visible. You can also select whether to show the reference object as outlines or as wireframe, and define the line color and type. Reinforcement in reference objects can also be shown.

The appearance options can be modified in drawing properties before you create the drawing, but the visibility options only when a drawing is open.

You can add associative notes (page 329) to reference objects.

1. In an open drawing, double-click the drawing view frame to open the drawing view properties.
2. Click **Reference object** in the options tree on the left.

   The **Content** tab lists all the reference objects included in the drawing.

   NOTE The list only shows the reference objects that have objects located the view area.

3. To show a reference object in the drawing, click the row of the reference object, and in the **Visibility** column, select **Visible**.
4. Go to the **Appearance** tab and set the representation for the reference object:

   Wireframe: Shows the reference object as a wireframe. With this option, opening drawings is faster.

   Outline: The reference objects are drawn in the same way as Tekla Structures native parts. The reference object outlines are shown. With this option, you have the **Hidden lines** and **Own hidden lines** options available.

   Hidden lines: Shows the hidden lines in the reference object that are hidden by other reference models or parts.

   Own hidden lines: Shows the hidden lines in the reference object that are hidden by the reference object itself.
5. Modify the color and the type of **Visible lines** and **Hidden lines**.
6. Click **Modify** to modify the reference object in the selected view with the new settings.

Modify reference object representation

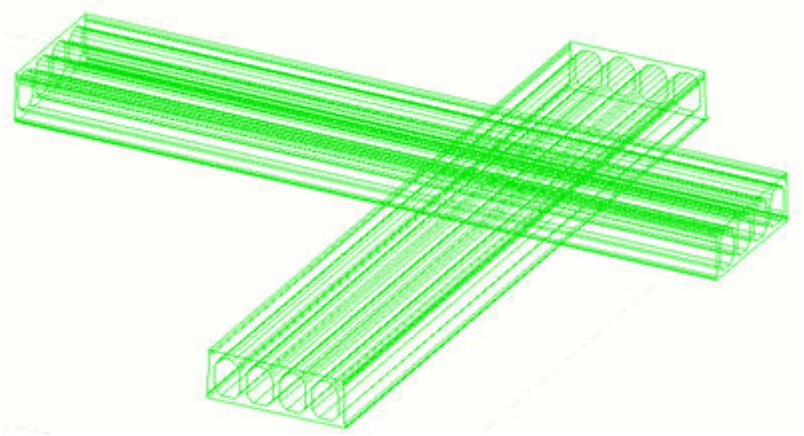
You can modify the reference object representation in an open drawing.

1. To open the reference object properties, click the reference object in the drawing.

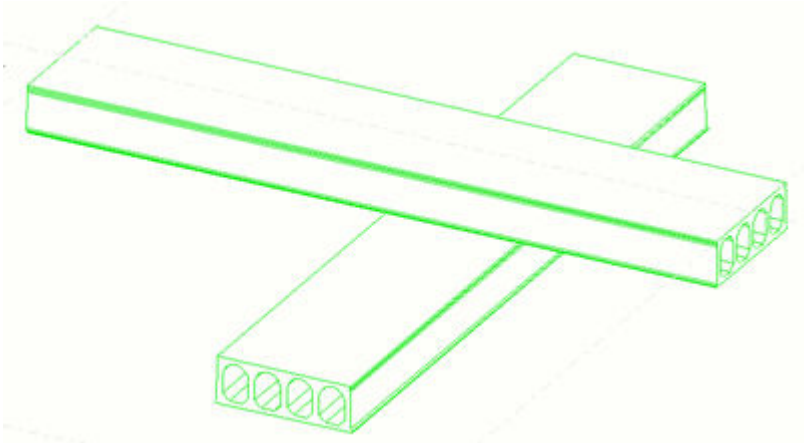
   If the property pane is not open, double-click the reference object.
2. Modify the reference object representation, line color, and line type as required.
3. Click **Modify**.

Reference object representation examples

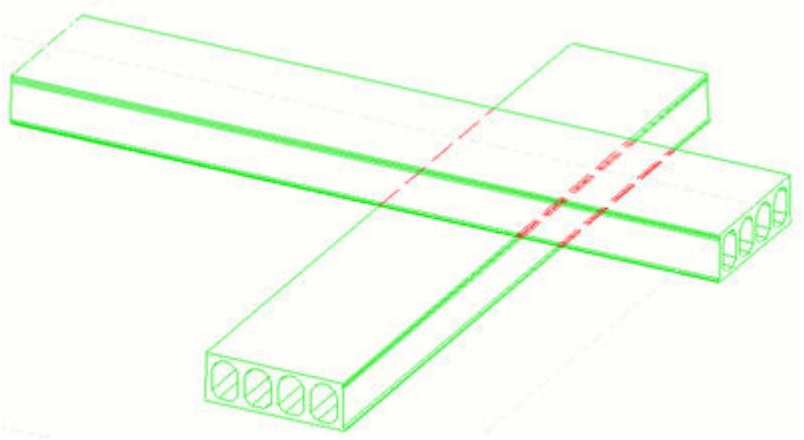
Below is an example of the **Wireframe** representation:



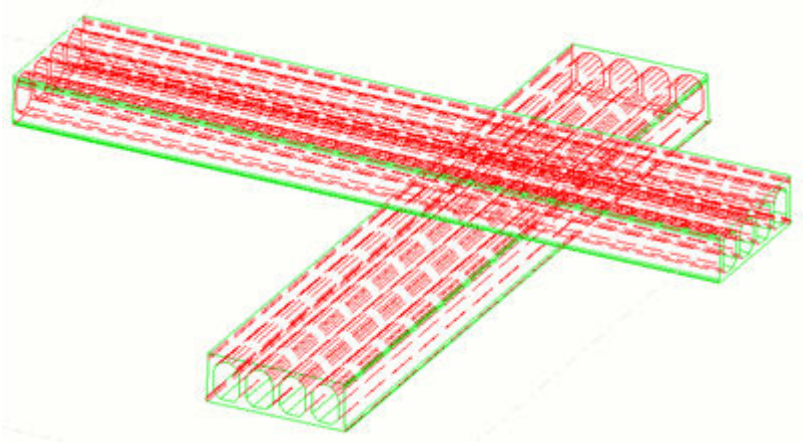
Below is an example of the **Outline** representation. **Hidden lines** and **Own hidden lines** are not activated:



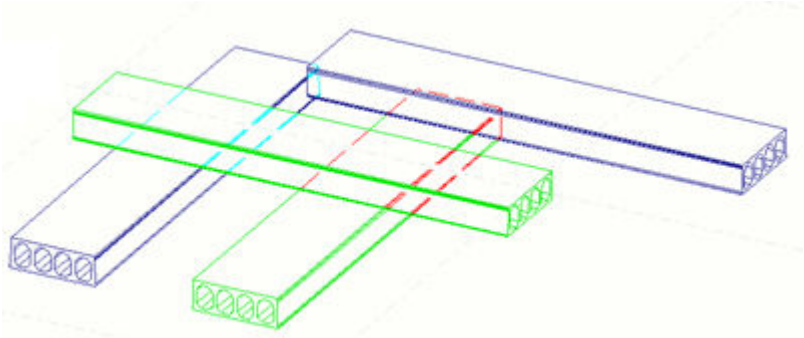
Below is an example of the **Outline** representation. **Hidden lines** is activated, and shown in red.



Below is an example of the **Outline** representation. **Hidden lines** and **Own hidden lines** are both activated, and shown in red.



Below is an example of how the parts are shown with reference objects. Reference object visible lines are shown in green, and hidden lines in red. The part visible lines are shown in blue, and the hidden lines in cyan.



4.23 2D Library in drawings

Drawing 2D Library allows you to quickly select objects in any drawings and save them as 2D drawing details. You can insert details in drawing views and drawings in any model. You can create details out of many kinds of drawing objects, such as parts, texts, notes, drawing sketch objects, or DWG files. In addition to details, you can browse and insert DWG files and images directly to your drawings from the **Drawing 2D Library**. It allows you to browse for details in other folders and use them in your drawing. Your company can create a collection of standard details that can be distributed in the system, project or firm folders and reused in different drawings and projects.

Limitations

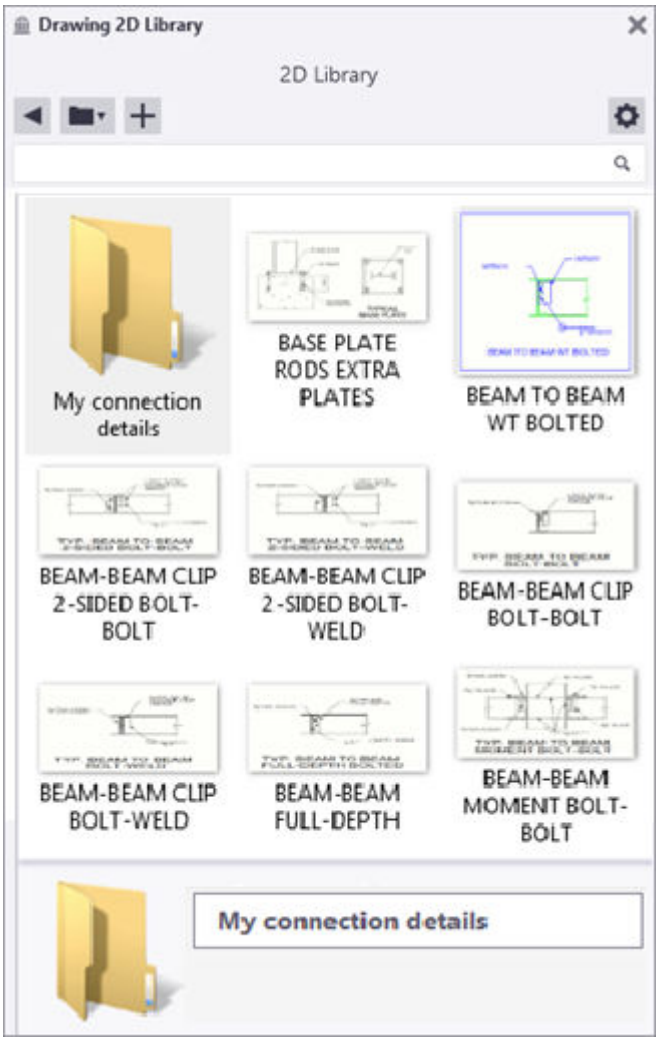
- Multi-colored part section hatches cannot be recognized in detail creation, and the part face hatch will be used instead.
- Templates or images cannot be captured in detail creation.

- Manually added weld marks cannot be captured in detail creation. This also applies to certain other manually added marks, such as revision marks and level marks.
- Details inserted in container views (views around linked/copied views) may be placed incorrectly.
- Inserted details do not rotate if the view is rotated.
- Details are not placed correctly in manually rotated views.
- Inserted details cannot be cloned with the drawing.
- You cannot use pattern lines (**Drawing** --> **Line** --> **Pattern line**) for 2D detailing.

Open and view Drawing 2D Library

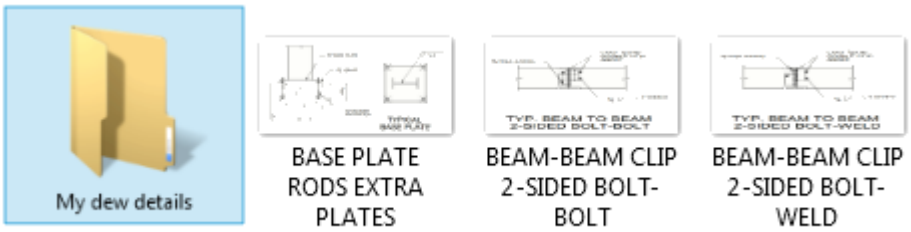
Drawing 2D Library is located in the Tekla Structures side pane. It is available when a drawing is open.

1. Open a drawing.
2. Open the **Drawing 2D Library** by clicking the **Drawing 2D Library** button in the side pane.



3. Click the **Folder** button to check the contents of the various folders:
   - **Current model** shows the details in the `\Drawing Details` folder under the current model folder. Your details are saved here, and here you can also create new subfolders. This folder may be empty if you have not created any details or used details in other folders.
   - **Project** and **Firm** show the details in the subfolder `\Drawing Details` under the project and firm folders if you have defined paths to these folders as values for the advanced options `XS_PROJECT` and `XS_FIRM`, respectively.
   - **System** shows the details in the subfolder `\Drawing Details` in any of the folders defined as a value for the advanced option `XS_SYSTEM`.
   - **Browse...** allows you to browse to any folder for details.
   - When you use details in project, firm, system or any other folder than `\Drawing Details` in the current model folder, the details are copied to the current model folder.

- When you update a detail with new or changed objects, all instances of the detail in any drawings in the current model will be updated as well. If you are fetching the detail that you update from the firm folder, the detail will not change in the firm folder. Global changes can only be done by the administrators of the firm, project and system folders.
- If the administrator updates a detail in the firm, project or system folder, any already inserted detail instances will not get automatically updated in any project. To do this, you would need to manually copy the updated detail file and replace what's in the model folder.
- **New folder** allows you to create a new folder in the current model folder in the subfolder `\Drawing Details`.
- Any subfolders in any of the folders are shown next to the details in the selected folder in **Drawing 2D Library** view. Double-click the subfolder to see the included details.



4. Search for details by entering a search term in the search box at the top of **Drawing 2D Library**. Tekla Structures shows the matching details in the view. Details are only searched in the currently displayed folder and its subfolders.

Insert a detail to a drawing from 2D Library

In **2D Library**, you can add details located in the system, project, firm or current model folder to any of your drawings.

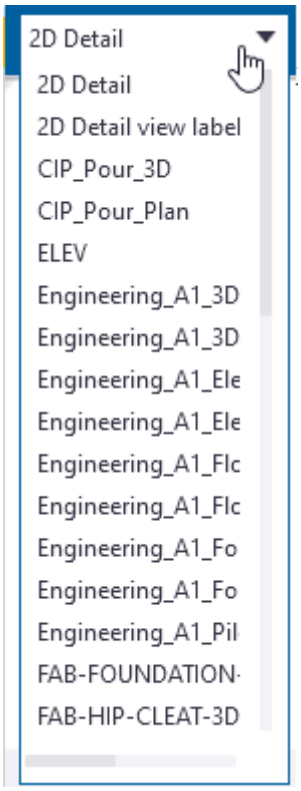
1. Open a drawing.
2. Click the **Drawing 2D Library** button in the side pane to open **Drawing 2D Library**. To browse to another folder for details, click the **Folder** button , and select **Current model**, **System**, **Project** or **Firm**. You can also browse further to other folders by selecting **Browse...**

3. If you always want to insert a detail inside a view when necessary, click the **Options** button and select **Create view if needed**.

   Details created within their own views are automatically created as fixed views.

   You may want to create a view for the detail if you place it outside drawing views. The new view will use the currently applied view properties and get the same scale as the inserted detail. If the **Create view if needed** option is not selected, the inserted detail does not get a view of its own, and dimensioning will not work properly in the detail.

   By default the `standard` view properties are used. You can also create a new view properties file to be used for detail views. This will let you have a specific view label in detail views, for example. You can select the new properties file, or any of the existing view properties files, on the **Options** menu . The selected view properties file settings will be applied to every 2D detail container view.



4. Click a detail and then click the drawing view or drawing to place the detail.

Tekla Structures inserts the detail. The detail is inserted as a plug-in, which means that the detail objects are grouped and stay together when you select or move the detail in a drawing.

Press **Esc** to interrupt picking insertion points.

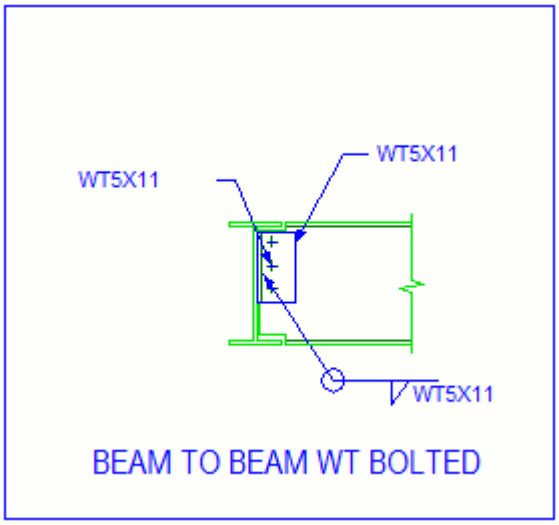
You can also pick a secondary reference point for rotation. To do this, right-click the detail in the **Drawing 2D Library** and enable **Add secondary reference point**. When you insert the detail, Tekla Structures asks you to give 2 input points. The secondary reference point will determine the direction of the detail.

Create a new detail in Drawing 2D Library

You can create new details in the current model folder or its subfolders.

1. In an open drawing, add the objects that will shape your detail.

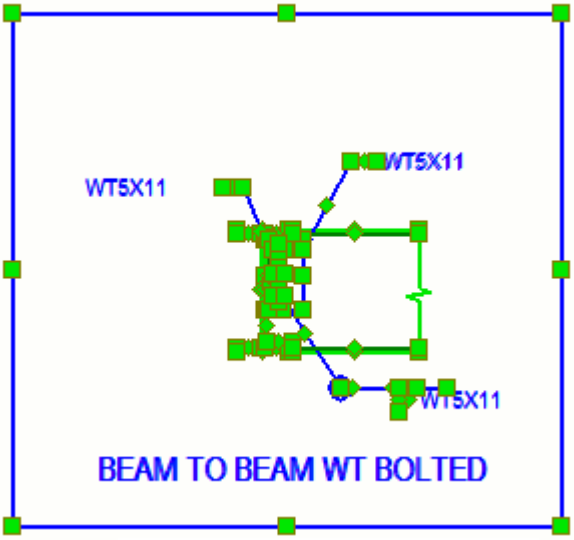
   In the example below, the detail represents a beam-to-beam connection. The detail contains texts, circles, lines, polylines, and symbols that have been added on top of a drawing and surrounded by a frame.



2. Click the **Drawing 2D Library** button in the side pane to open **Drawing 2D Library**.

3. To include grids in the detail, in the **Options** menu, select **Include grids**.

4. Select the objects that you want to include in the detail.



5. Add the detail in **Drawing 2D Library**:

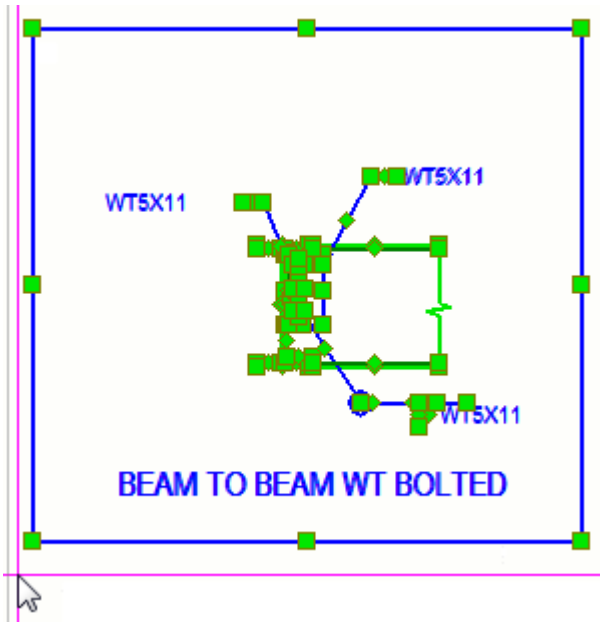
   a. If you have subfolders in the current model folder, double-click the subfolder in **Drawing 2D Library** view where you want to save the new detail.

      You can create a detail even when the current model folder is not selected. The detail will be stored in the current model folder, and a status bar message will indicate the name of the new detail.

   b. Click the **New detail from selected objects** button at the top of **Drawing 2D Library**.

      A message at the bottom of **Drawing 2D Library** asks you to pick a reference point.

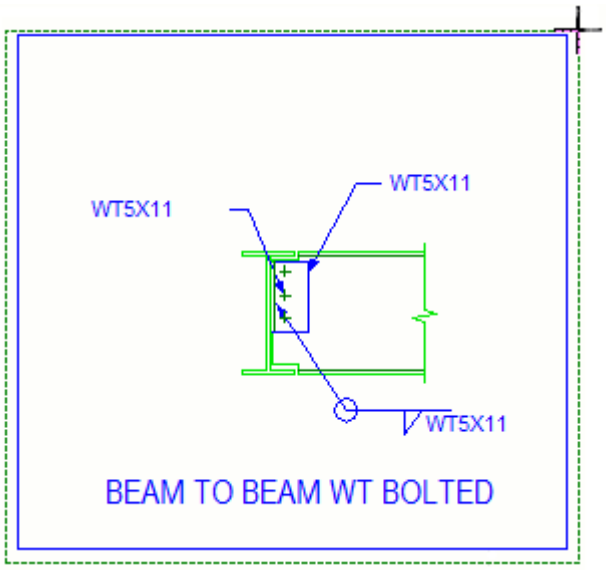
   c. Pick a reference point in the drawing.



TIP You can also pick a secondary reference point for rotation. To do this, right-click the detail in the **Drawing 2D Library** and enable **Add secondary reference point**. When you insert the detail, Tekla Structures asks you to give 2 input points. The secondary reference point will determine the direction of the detail.

A message at the bottom of **Drawing 2D Library** asks you to capture a sample image by picking two points.

d. Pick two points to capture an image of the detail.



Tekla Structures creates the detail and adds it to **Drawing 2D Library** in the `\Drawing Details` folder (or its subfolder, if defined) under the current model folder. This folder is automatically created when a detail is created in

the current model for the first time. Tekla Structures saves the detail and its metadata (name, description) as a `.ddf` file. The metadata is used when you search for details in **Drawing 2D Library**. The captured image is saved as a `.png` file.

Create a new folder in Drawing 2D Library and copy/move to the folder

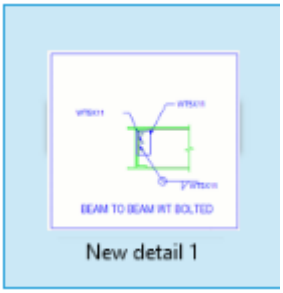
You can create details in a new folder or copy or move details to the new folder in the current model folder.

1. In **Drawing 2D Library**, click the **Folder** button , and select **New folder**. You can rename the new folder by clicking the folder and entering a name at the bottom of the **Drawing 2D Library** pane.
2. Right-click a detail you want to copy or move and click **Cut** or **Copy**.
3. Right-click the new folder and select **Paste**. Tekla Structures copies or moves the selected detail.
4. If you want to add some details in a system folder, create a subfolder in a system folder (defined by `XS_SYSTEM`) and rename it `Drawing Details`, then copy details from the model folder to the new `\Drawing Details` system folder using Windows Explorer. You can copy or move details in the same way to the project and firm folders.

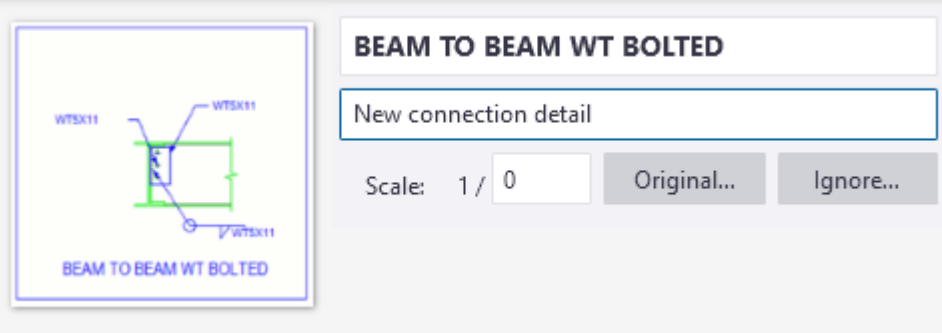
Modify detail properties in Drawing 2D Library

You can modify detail properties of the details in the current model folder. You cannot modify the detail objects, for example, the texts, marks, or lines inside a detail, because the details are grouped. You first need to explode the detail, and then update it.

1. In an open drawing, browse for a folder, and click a detail in **Drawing 2D Library**. A new detail looks like the one below in **Drawing 2D Library** when you have not touched its properties yet:



2. Modify the detail properties at the bottom of the **Drawing 2D Library** pane:



a. Edit the detail name and the description of the detail.

b. Select the original scale, enter a scale, or ignore the scale.

   The **Scale** value refers to the scale of the view where the detail was originally created. The **Scale** value should not normally need to be touched. This value is used when inserting the detail, for adapting the detail to the scale of the target view, so that dimensioning, for example, will work correctly.

   Example:

   You create a detail A from objects in a 1/10 scale view. So the scale of the new detail gets recorded as 1/10 in **Drawing 2D Library**. You then insert detail A (which has scale 1/10) into a scale 1/50 view W. The detail will look 5 times smaller than in the original view, but dimensioning will give the same results in both views.

   You create another detail B from objects in a 1/5 scale view. So the scale of the new detail gets recorded as 1/5 in **Drawing 2D Library**. You then insert this detail (which has scale 1/5) into the scale 1/50 view W. The detail will look 10 times smaller than in the original view, but again dimensioning will give the same results in both views.

   You then change the scale of the detail A in the view W from 1/10 to 1/5. This makes Tekla Structures calculate the size of the detail as if the original view had had the scale 1/5. So just as detail B, the detail A will now look 10 times smaller than in the original view. However in this case, dimensioning will now give only 1/2 of the expected results (which is the ratio between the scale you have entered and the actual original scale).

   When you click **Ignore scale**, the detail will have the same visual size regardless of the view scale, corresponding to the visual size it had when it was created. This is indicated by the scale value 1/0. Therefore dimensioning will not be correct if you use the **Ignore scale** option.

c. To change the sample image, move the mouse pointer over the image on the left, click the **Capture new** button, and then pick two points in the drawing.

3. To view and modify the detail properties in the drawing, double-click the inserted detail.

Explode a detail

You can explode an inserted detail into lines and texts, for example, to update the detail with new objects.

For example, you may have created a detail and something does not look quite right. You can explode the detail, make modifications and update the detail with the changes.

Another example could be that you have a set of company-specific details that you are inserting details from. You can explode a detail and edit it, and make a new detail out of it.

- In a drawing, right-click an inserted detail, and select **Explode**.

The detail is exploded to lines and texts. Now you can edit the detail, and then update the detail.

Explode dimension included in details

You can create native dimensions in details instead of lines and symbols.

1. In **Drawing 2D Library**, click the **Options** button and select **Explode native dimension**.
2. Insert a detail containing dimensions in a drawing.
3. Right-click the inserted detail, and select **Explode**. The detail is exploded and the included dimensions are exploded to native dimensions.

Explode symbols included in details

You can explode symbols included in details, and make them independent of the local symbol files.

Drawing 2D Library saves the drawing symbols as symbols, which means that later, when you insert the detail, you need to have the correct symbols files present. You can avoid this by exploding the symbols.

1. In **Drawing 2D Library**, click the **Options** button and select **Explode native symbols**.
2. Insert a detail containing drawing symbols in a drawing.
3. Right-click the inserted detail, and select **Explode**. The detail and the included symbols are exploded to lines and texts.

Update objects in a detail

You can modify all instances of a detail by updating the detail with new or modified objects.

1. Insert the detail in a drawing and explode it by right-clicking the detail and selecting **Explode**.
2. Modify the exploded detail objects or add new ones.
3. Select all detail objects, including any new ones.
4. Right-click the detail in **Drawing 2D Library** view and select **Update detail with selected objects**. You are asked to pick a new reference point. If you are updating the detail in the same view as originally, you do not need to pick a new reference point, otherwise you need to pick a new reference point.

   The detail gets updated. When you update a detail with new objects, the detail will get updated in all drawings where it is used.

Insert a .dwg file to a drawing from Drawing 2D Library

You can insert `.dwg` files from the **2D Library** to drawings as reference files.

1. Open a drawing.
2. Click the **Drawing 2D Library** button in the side pane to open the **Drawing 2D Library**.
3. Browse for the folder containing the `.dwg` files in the **Drawing 2D Library**. You can also move your reference files to a `\Drawing Details` folder under the current model folder.
4. If you always want to insert a detail inside a view when necessary, click the **Options** button and select **Create view if needed**.
5. Click a `.dwg` file and then click the drawing view or drawing to place the .dwg file. Tekla Structures inserts the `.dwg` file as a reference object in the drawing, not as a drawing detail. If the file is inserted from outside the model folder, the file will be copied to the model folder first and the insertion path will be relative to that.

   You cannot update the `.dwg` file with new objects, but you can select an inserted `.dwg` along with any other objects and create a new detail.

Insert an image to a drawing from Drawing 2D Library

You can insert image files from the **Drawing 2D Library** to drawings.

1. Open a drawing.

2. Click the **Drawing 2D Library** button in the side pane to open the **Drawing 2D Library**.
3. Browse for the folder containing image files in the **Drawing 2D Library**. You can also move your image files to a `\Drawing Details` folder under the current model folder.
4. If you always want to insert a detail inside a view when necessary, click the **Options** button and select **Create view if needed**.
5. Click an image file and then click the drawing view or drawing to place the image.

   The image is inserted. If the file is inserted from outside the model folder, the file will be copied to the model folder first and the insertion path will be relative to that.

   Note that the image detail thumbnail is created automatically.

4.24 Indicate cut lines in Tekla Structures drawings

Cut lines are lines that indicate with zigzag or dash-and-dot lines in different colors that a part is partially outside the view border.

Create cut lines

You can create cut lines to visualize the lines that are partially outside the view frame. You can create cut lines automatically for all parts in the view, or for selected parts.

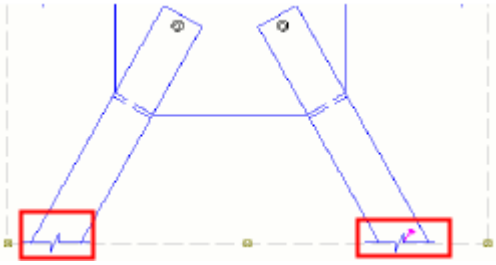
Limitations:

- Cut lines cannot be created for polybeams.
- **View extension for neighbor parts** setting is not taken into account.

1. On the **Drawing** tab, click **Create cut lines**.
2. Do any of the following:
   - Select the type (zigzag or dash-and-dot) for the line from the list of types.
   - Select the color for the line from the color list.
   - Enter a scale for the line in the box next to the list of types.
3. Do one of the following:
   - To create cut lines for all parts in the view, select the view.
   - To create cut lines for selected parts, select the parts.

4. Click **Create**.

Example



Update cut lines

1. Open a drawing that contains cut lines.
2. On the **Drawing** tab, click **Create cut lines**.
3. Do one of the following:
   - To update the cut lines of all parts in the view, select the view.
   - To update the cut lines of selected parts, select the parts.
4. Click **Create**.

Tekla Structures removes all previously created cut lines, and creates new ones that are up-to-date.

Delete cut lines

1. On the **Drawing** tab, click **Create cut lines**.
2. Do one of the following:
   - To delete the cut lines of all parts in the view, select the view.
   - To delete the cut lines of selected parts, select the parts.
3. Click **Delete**.

4.25 Explode drawing plug-ins and use ordinary drawing objects

You can explode drawing plug-ins into base objects, and then edit and use them as any other drawing objects. For example, if you want to modify COG

dimension in a way that is not available for the plug-in in the plug-in properties, you can explode the COG dimension and then modify its properties in dimension properties.

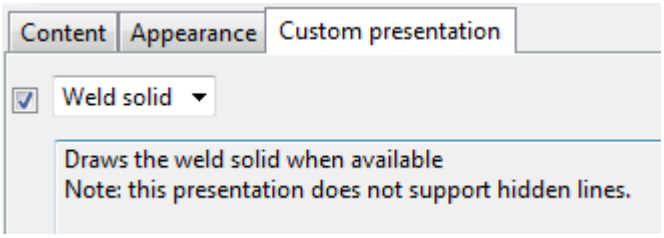
1. Click a drawing object that has been created using a plug-in.
2. Right-click and select **Explode**.

Tekla Structures explodes the plug-in into base objects that are added to the view. Now you can edit and use the exploded objects as any other drawing objects through object properties.

4.26 Custom presentations in drawings

You can modify the appearance of many of the objects in drawings using custom presentations. Custom presentations are delivered in Tekla Warehouse as extensions. The presentations can be controlled on the view level and object level.

When you have downloaded a custom presentation for an object type, the **Custom presentation** tab will become available in the drawing property dialog for that object. The list will only show custom presentations that are available for that specific object type – part, weld, mark, etc.

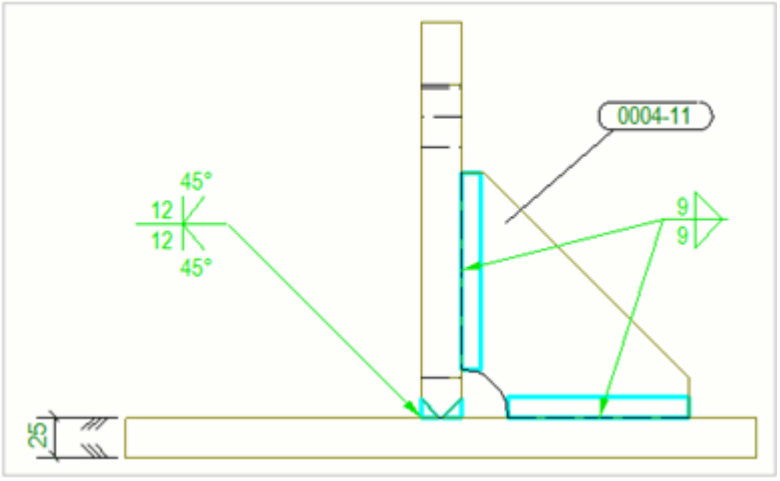


The following object types support custom presentations:

- welds and weld marks
- parts and part marks
- neighbor parts and part marks
- grid lines
- texts
- associative notes

Examples

In the example below, the **Weld solid** custom presentation is used for drawing the weld solids.



In the following example, **Corners Only** custom presentation is used for laser layout projection. Drawing all the lines of a part slows down the laser and makes it too dim to see on the layout table.

Custom presentations in Tekla Warehouse

Center of gravity

GA schedules

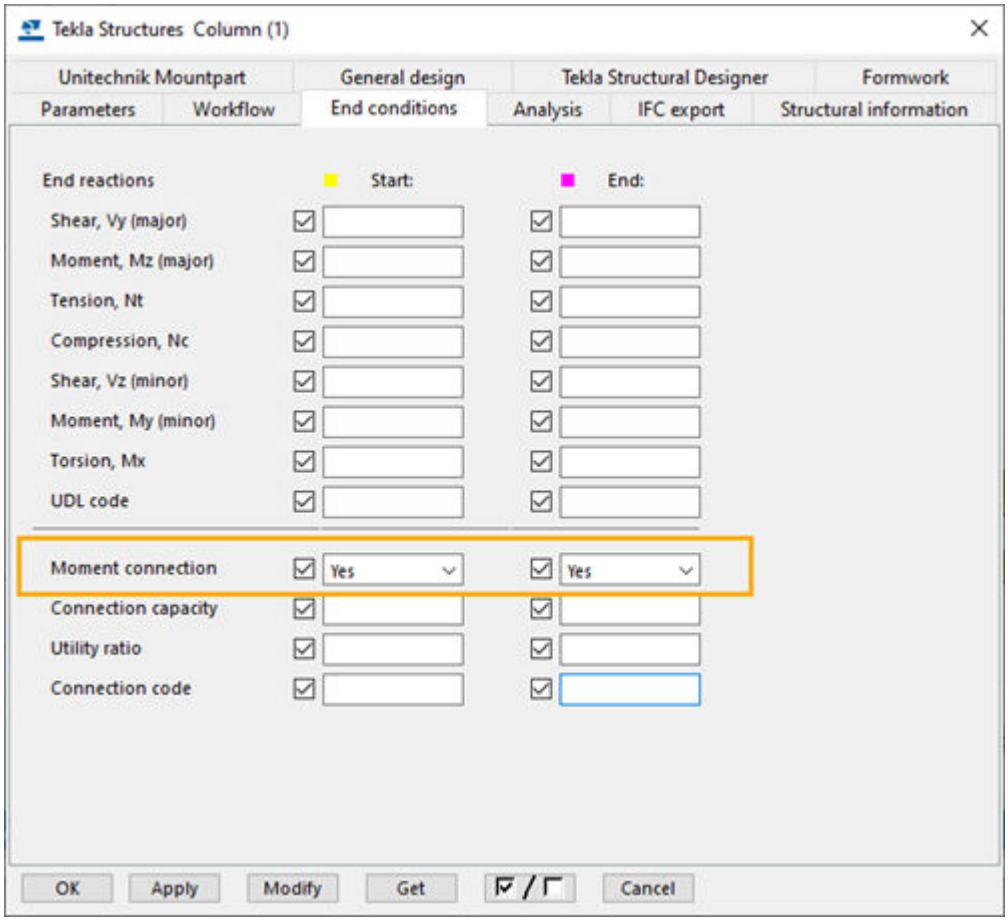
See also

Edit drawings (page 166)

4.27 Moment connection symbols in Tekla Structures drawings

In general arrangement drawings, moment connections show the beams that are connected to columns with rigid connections.

To display the moment connection symbols for a part in drawings, you need to set the **Moment connection** to **Yes** on the **End conditions** tab in the user-defined attributes of the part in the model.



You can create, update and delete moment connection symbols in general arrangement drawings using the **Create moment connection symbols** macro.

Create moment connection symbols

In general arrangement drawings, you can create moment connection symbols to show the beams that are connected to columns with rigid connections. The symbols are created according to part end releases. You can create moment connection symbols automatically for all parts in a drawing view, or for selected parts.

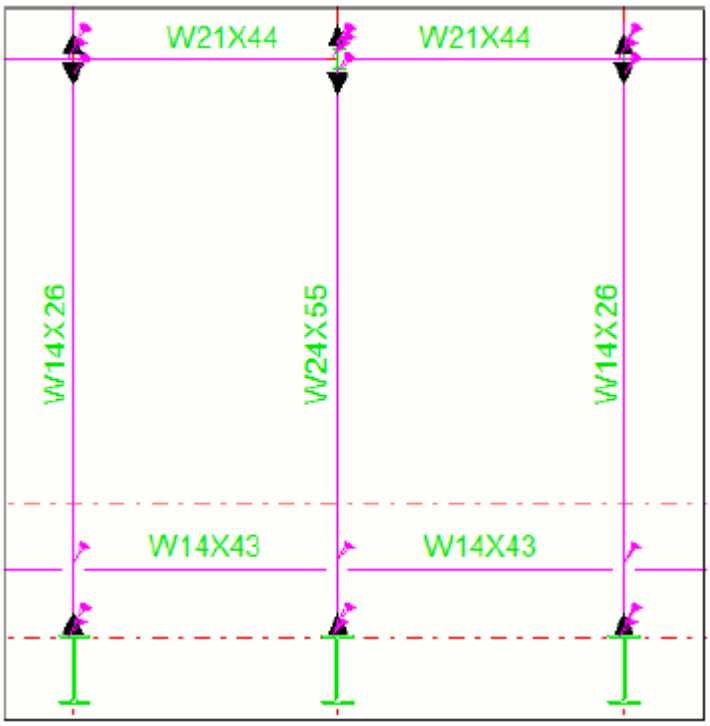
Limitations: Moment connection symbols are created to reference lines. This means that offsets are not taken into use.

1. Open the drawing.

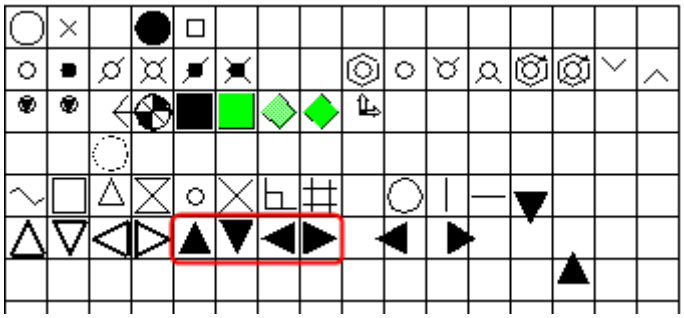
2. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
3. Search for **Create moment connection symbols** .
4. Click **Create moment connection symbols** .
5. In the **Create moment connection symbols** dialog box, select the color for the symbols from the color list.
6. Enter a scale for the symbols in the box next to the color list.
7. Do one of the following:
   - To create moment connection symbols for all parts in a drawing view, select the view.
   - To create moment connection symbols for selected parts, select the parts.
8. Click **Create**.

   The moment connection symbols are created according to analysis part and connection release properties:
   - If a connection exists, start and end release information is read from the **Start releases** and **End releases** tabs in analysis part properties.
   - If a connection does not exist, release information is read from the **End conditions** tab in the user-defined attributes of the part.

Example



TIP The moment connection symbol that is used depends on if it points to right, left, down, or up. The symbols are retrieved by default from the `xsteel.sym` symbol file. If you want to use another symbol, you can modify the symbols in Symbol Editor by opening the `xsteel.sym` symbol file and modifying the symbols 87 (right symbol), 86 (left symbol), 85 (down symbol), or 84 (symbol up) and saving the changes.



Update moment connection symbols

If you want to remove all previously created moment connection symbols and create new ones that are up to date, you can update the moment connection symbols.

1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
2. Search for the **Create moment connection symbols** macro.
3. Click **Create moment connection symbols** .
4. Do one of the following:
   - To update the moment connection symbols of all parts in the view, select the view.
   - To update the moment connection symbols of selected parts, select the parts.
5. Click **Create**.

When you do this, Tekla Structures removes all previously created symbols, and creates new ones that are up-to-date.

Delete moment connection symbols

You can delete moment connection symbols of all parts in a view or of selected parts.

1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
2. Search for the **Create moment connection symbols** macro.
3. Click **Create moment connection symbols** .
4. Do one of the following:
   - To delete the moment connection symbols of all parts in the view, select the view.
   - To delete the moment connection symbols of selected parts, select the parts.
5. Click **Delete**.

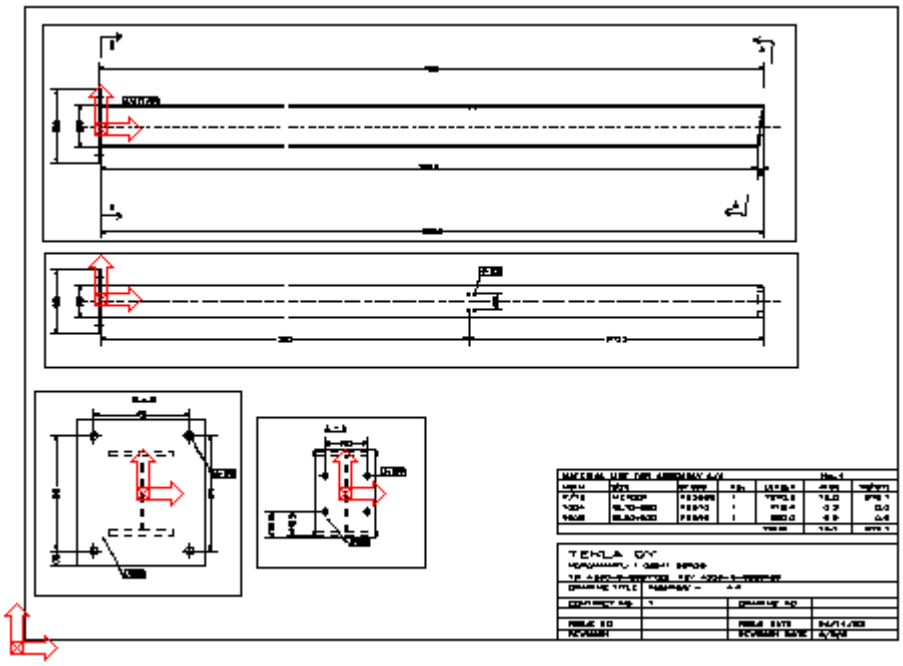
4.28 User coordinate system (UCS)

UCS is a local user coordinate system you can use in a drawing view. It is easier to position drawing objects in view, when you can place objects relative to a user-defined point of origin, or a base point, in the drawing view.

Tekla Structures shows the UCS symbol in the current drawing view, when you create, copy, move, or modify objects.

You can define a different point of origin for the UCS for each drawing view, and change the UCS point of origin as often as you like.

The following example shows several views, each with their own UCS.



NOTE To position an object using the global coordinate system, you need to calculate the coordinates of the object from the point of origin of the drawing, not a drawing view.

Set a new UCS

You can set a new UCS (user coordinate system) using one point or two points.

1. Open a drawing.
2. On the **Views** tab, click **User coordsys** and click one of the following commands:
   - Select **Set origin** to set the new UCS using one point.

- Select **Set origin by two points** to set the new UCS using two points.

  Tekla Structures displays the UCS symbol with a crosshair marking the center point.

3. Click the view where you want to place the origin.
4. If you are using two points, pick a point to define the direction of the x axis.

Toggle between two user coordinate systems

You can toggle between two user coordinate systems that have the same point of origin: the UCS following the axes of the drawing view and the oriented UCS you have created.

- To toggle between the coordinate systems, open a drawing, go to the **Views** tab and click **User coordsys** --> **Toggle orientation** (**Ctrl + T**).

Reset UCS

You can reset the UCS to its original position in the current drawing view or in all drawing views.

Do one of the following:

To	Do this
Reset the UCS in the current drawing view	In an open drawing, go to the **Views** tab, and click **User coordsys** --> **Reset current** (**Ctrl + 1**).
Reset the UCS in all drawing views	In an open drawing, go to the **Views** tab, and click **User coordsys** --> **Reset all** (**Ctrl + 0**).

5 Manage drawings

You can manage your drawings in **Document manager**. Depending on the workflow phase, you have several tools available for managing the drawings. You can update drawings when the model changes, lock drawings, freeze drawings, issue drawings, revise drawings, and delete drawings.

To	Click the link below:
Learn how to set up **Document manager** and use it for searching drawings, for example	Document manager (page 552)
Update saved drawings because of model changes	Update drawings when the model changes (page 575)
Indicate that a drawing is not available for editing by locking it	Lock drawings (page 576)
Select whether to allow Tekla Structures to update all associative objects on top of the drawing views	Freeze drawings (page 577)
Mark drawings ready for issuing	Mark drawings ready for issuing (page 578)
Mark a drawing issued when it has been released for fabrication	Issue drawings (page 579)
Delete a drawing that you do not need anymore	Delete drawings and file documents (page 590)
Renumber all GA drawings	Delete drawings and file documents (page 590)
Delete unnecessary drawing files without waiting for Tekla Structures to automatically delete them	Delete unnecessary drawing files in single-user mode (page 589)
Revise drawings, and attach information about the changes you have made	Revise drawings (page 580)
Manage drawing versions	Version control for drawings (page 584)

To	Click the link below:
Use the old drawing list	Drawing list (page 591)

5.1 Document manager

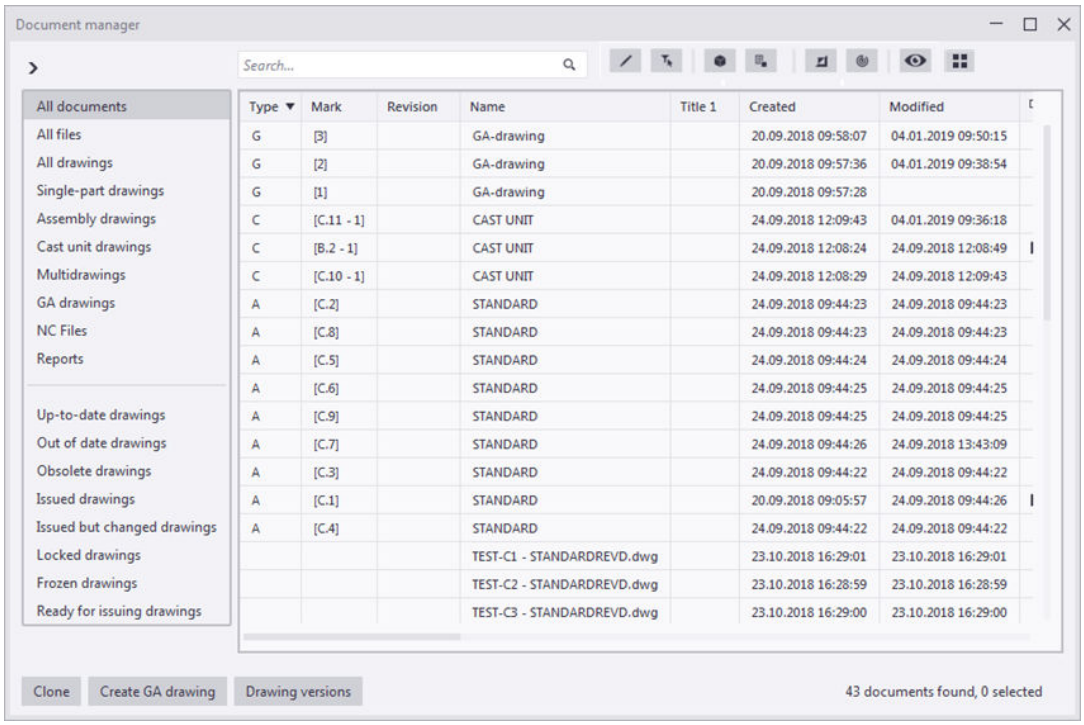
With **Document manager** you can list and manage your drawings and other types of documents efficiently.

You can reorganize, freeze, show and hide columns, edit some of the list data directly, and organize documents and drawings in categories, for example. You can open drawings, and if you have a related application installed, you can open other types of files, such as PDF files, NC files and DWG files. **Document manager** has a powerful search tool, which supports wildcards and the most common search operators. You can save your searches in search-based categories, and also create manual categories.

Open Document manager

Open **Document manager** in one of the following ways:

- On the **Drawings & reports** tab on the ribbon, click the **Document manager** button.
- Enter `document manager` in **Quick Launch**. You can also enter `drawing list`.
- Press **Ctrl+O** if a drawing is open, or press **Ctrl+L** if you are in the model. You can also customize the shortcut. You can find it in the **Keyboard shortcuts** dialog box by searching either `document manager` or `drawing list`.
- Select objects in the model, open the contextual toolbar, and select **Open or create drawings**. Then list the drawings for the selected objects in **Document manager** by selecting **Show drawings for selected objects...** in the menu.



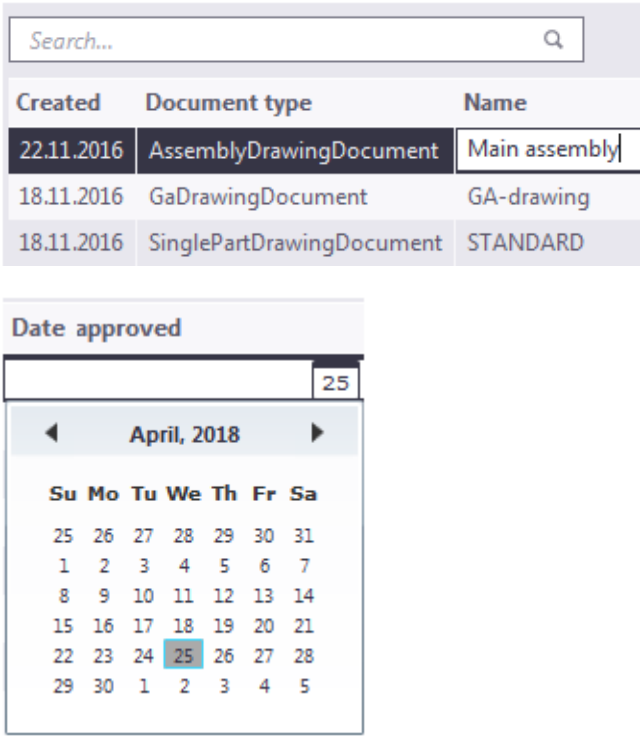
Filter, sort and adjust Document manager content

To	Do this
List documents in a certain category	• Click a category on the left. • You can also hold down **Ctrl** or **Shift** and select several categories. • You can show/hide the categories list by clicking the arrow in the top left corner of the dialog box. • To exclude documents from or include documents in an individual category in **Document manager**, right-click the category and select **Exclude/Include**. For example, if you just want to see drawings and no other documents, right-click the **All files** category and select **Exclude**.
List all documents in the model, both drawings and file documents	• Click the **All documents** category. The file `DocumentManagerFileDocumentSettings.txt` in your model, project, firm or system folder defines which other document types are shown in addition to drawings.

To	Do this
List all file documents in the model, only including other document types than drawings	• Click the **All files** category.
List all drawings in the model	• Click the **All drawings** category.
List up-to-date drawings only	• Click the **Up-to-date drawings** category.
List drawings that need to be updated	1. Click the **Up-to-date drawings** category. 2. Click the **Invert document visibility** switch.
Only list the documents that you have selected from the list	• Activate the **Show selected documents only** switch. When the switch is active, it is blue . Note that when you activate the **Show selected documents only** mode, **Document manager** also show documents that you then create or modify, as long as they match the active search and selected category criteria. These documents will then appear in the list as unselected to distinguish them from the already selected documents.
Hide the currently visible documents in the list and list all other documents	• Activate the **Invert document visibility** switch. When the switch is active, it is blue . • When the switch is active, it is blue . The name of the switch changes to **Reset document visibility** to indicate that clicking the button again will reset the visibility and not invert the visibility a second time. • If you click a category or enter a manual search, this switch will be deactivated. • Any documents that you have excluded from **Document manager** using the **Exclude** command will be ignored by **Invert document visibility** and will never be shown.

To	Do this
Show changes from now on	• Activate the **Show changes from the checkpoint set when the button is activated** switch. When the switch is active, documents that have changed since the activation of the switch will be displayed. • When you have enabled the **Show changes from the checkpoint set when the button is activated** switch in **Document manager**, and then when you disable this switch, you will get the following message: **Turning off 'Show changes' will reset the changes checkpoint and lose the history of which documents have changed since the checkpoint was set. Are you sure you wish to continue?**. Select **Create new category**. • If this switch is active, and you run numbering, any drawings who's status then changes (if parts were modified, for example) will then be automatically shown in the list.
Reset the document list	• Click the **Show all documents** button. This resets the document list so that all documents are visible, and all searches and filtering are cleared. Excluded categories remain excluded. This feature also provides a convenient way to restore the document list to a default state when recording macros.
Reorder columns	• To reorder columns, drag the columns to new locations.
Sort columns	• To sort a column, click a column title. A small arrow shows that the column is used for sorting and the sort order. • To sort by several columns, hold down **Shift** and click column titles.
Freeze columns	• To freeze one or more columns so that columns to the left of the selected column will not scroll horizontally, right-click a column and select **Freeze at this column**. • To unfreeze, right-click and select **Unfreeze columns**.
Show or hide columns	• To hide a column, right-click a column and select **Hide column**.

To	Do this
	• To show hidden columns, right-click a column and select **Show all columns** , or **Show column** and select a hidden column you want to show. • The column order, visibility and sorting order are saved in the model folder in an user-suffixed file `DocumentManagerDataGridSettings_<user>.xml`. Default values are read from `DocumentManagerDataGridSettings.xml` located in the system folder defined by the `XS_SYSTEM` advanced option if there is no user-specific settings file available. The user settings file can be located in all standard locations (model, project, firm, environment). You can copy the user-specific file file to be shared at the project, firm or environment level. Then you need to remove the username suffix from the file name. If you are copying a file to use in a model, you might need to change or add the username suffix in the file name to match the user that will be using Tekla Structures on that model.
Display time	• Right-click a column that shows date information, and select **Time display** and one of the available options: **No time** **Hours and minutes** **Hours, minutes and seconds** If the property only supports date and no time, these options will not be available.
Edit **Document manager** list data directly	1. First ensure that the edit switch is active. 2. Then click the data cell you wish to edit, such as **Name**, **Title 1** - **Title 3**, **Freeze**, **Lock**, **Ready for issuing**, or a cell that comes from drawing UDAs, such as date cells.

To	Do this
	 When the edit switch is active in **Document manager**, and you move the mouse cursor over a cell that is not editable, the cursor changes to indicate that you cannot edit the cell:
Show and modify drawing views in the model	1. Select one or more drawings in the document list. 2. Click the **Show drawing views in the model** button in the top-right corner of the **Document manager**window. When this button is activated, it changes color: The views in the selected drawings are now displayed in the model. You can then use direct modification to work with the view extents and view plane of every view that is contained within each selected drawing in the currently open model views.

To	Do this
Delete drawings or other documents	Select drawings or file documents and click **Delete**, or right-click and select **Delete**. If you have selected both drawings and file documents, you get separate confirmation messages for drawings and file documents. If you hold down **Shift** while you select **Delete**, Tekla Structures will not ask you to confirm the deletion of documents.

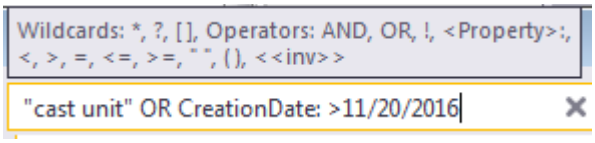
Search for documents

Use the search box at the top to look for the desired documents. Only the documents currently shown in the list are searched.

Search is case insensitive and partial. For example, searching for the string `con` will match with `concrete` and `second`.

You do not need to write the operator AND, just use a space between search terms. For example, searching for the string `steel grade` will match with `steel beam grade` but not with `steel beam`.

You can search for documents within a specific date range or exact date, and define multiple search strings.



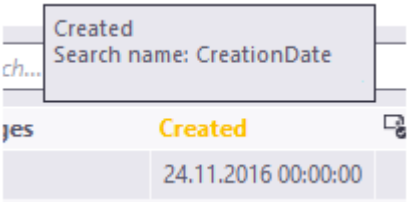
You can save searches to avoid typing in frequent searches repeatedly. This is done by saving the current search as a new category. You can also combine existing categories together to easily create more complex queries.

Wildcards and operators

See the table below for explanations and examples of the different wildcards and operators you can use.

Notation	Explanation
*	Match zero to any number of characters. Fore example, `H*T` matches **h**o**t**, **h**ea**t**, and **h**eavy lif**t**er
?	Matches any single character. For example, `B?-1` matches B1-1, B2-1, Bb-1.

Notation	Explanation
[]	Defines a matching group. The matching group matches any single character out of a set of characters that you give between the brackets. For example, the search `A[ABC]1` matches AA1, AB1, and AC1.
AND	The search expressions on both sides must match. This is the default operator, and you do not have to type it in the search string. For example, `steel AND grade` finds drawings that contain `steel grade S235JR` but does not find drawings that contain `grade S235JR` or `steel S235JR`.
OR	One of the search expressions on either side must match. For example, `steel OR grade` finds drawings that contain `grade S235JR`, `steel grade S235JR` or `steel S235JR`.
! -	These are NOT operators. Use these operators to exclude from the search, for example, `!steel` and `-steel` both mean that the search will not find documents including the string `steel`. If the name of the document that you want to find contains a dash (–), enclose the search expression in quotes, including the dash, for example, `"C100-C500"`, then your document is found, for example, `C.1000-C100-C500`.
()	Groups search expressions. Use parentheses () to create more complex expressions. For example: `(B2 AND concrete) OR B3` matches all drawings containing B3 and also the subset of B2 drawings that mention concrete.

Notation	Explanation
""	Disables special characters inside the quotes. Use for phrases that contains spaces or if you want to search for characters that are listed as wildcards or operators in this table. For example, `Mark: "2 - 1"` matches drawings containing [B.2 - 1] and [C.2 - 1].
<Property name>:	Replace <Property name> with one of the properties displayed to limit the search scope to that property. You can search for documents based on the internal name of the property (search name). The search name is shown in the property column header tooltips when you move the mouse pointer over the header. The search name is case sensitive.  For example, `CreationDate: 11/20/2020` matches drawings created on the 20th of November 2020.
<	Less than. For example, `CreationDate: < 11/20/2020` matches drawings created before the 20th of November 2020. This operator does not work for fields that contain other characters than numbers.
>	Greater than. For example, `CreationDate: > 11/20/2020` matches drawings

Notation	Explanation
	created after the 20th of November 2020. This operator does not work for fields that contain other characters than numbers.
=	Exact match. Disables wildcards and partial matching. For example, `CreationDate: = 11/20/2020` matches drawings created on the 20th of November 2020, and `Mark: = [B.2 - 1]` matches drawings containing [B.2 - 1] only. If you type a wildcard, asterisk (*) for example, it is considered as the character *.
<=	Less than or equal. For example, `CreationDate: < 11/20/2020` matches drawings created on the 20th of November 2020 or before.
>=	Greater than or equal. For example, `CreationDate: > 11/20/2020` matches drawings created on the 20th of November 2020 or after.
<<inv>>	Invariant (language-independent) search. Use if you want to match fields written in different languages. For more information, see below.

Searching with date and time

- You can search date and time in two different ways depending on the date and time format Tekla Structures is using:
  - `dd.mm.yyyy` or `mm/dd/yyyy` for dates, for example, `30.11.2018` or `11/30/2018`
  - `hh:mm:ss` or `hh:mm:sspm` and `hh:mm:ssam` for time, for example, `17:34:45` or `05:34:45pm`
- Note that advanced options `XS_IMPERIAL_DATE` and `XS_IMPERIAL_TIME` determine which date and time format is active. Only the active format is

used, which means that a non-imperial date will be ignored if imperial date is the active date format.

- You can also enter partial dates and date ranges:
  - `2018`: any date and time in the year
  - `6.2018` (or `6/2018`): any date and time in the month
  - `24.06.2018` (or `06/24/2018`: any time in the specified day
  - `"06.06.2018 14"` : any time in the range `14:00` to `14:59` on `06.06.2018`. Note that you need to use quotes because otherwise the space is seen as the end of one search expression.

Complex search example

`Mark: FP !??FP` finds all marks that begin with FP. The search matches [FP123] and [FPZ23] but does not match [AB123FP].

Invariant search values

The search syntax `<<inv>>(...)` allows you to search for documents and create categories that work in the same way regardless of the language in use.

The values to use when doing invariant `<<inv>>(...)` searches are listed below.

- Property name: `IssuedStatus`
  - `NotIssued`
  - `Issued`
  - `IssuedButModified`
- Property name: `DrawingUpToDateStatus`
  - `DrawingIsUpToDate`
  - `PartsWereModified`
  - `DrawingIsUpToDateButMayNeedChecking`
  - `IncludedDrawingModified`
  - `OriginalPartDeleted`
  - `AllPartsDeleted`
  - `NumberOfPartsInNumberingSeriesIncreased`
  - `NumberOfPartsInNumberingSeriesDecreased`
  - `DrawingWasCloned`
  - `DrawingWasUpdated`
  - `CopiedViewChanged`

    - `DrawingWasSplitted`
    - `MovedViewDeleted`
    - `MovedViewLabelChanged`
- Property name: `Changes`
    - Same values as `DrawingUpToDateStatus` and `IssuedButModified`
- In date and time search, you can only use the non-imperial formats (`dd.mm.yyyy` and `hh:mm:ss`) :
    - `<<inv>>(10.5.2017)`
    - The invariant search ignores the advanced options `XS_IMPERIAL_DATE` and `XS_IMPERIAL_TIME`.

For **Freeze**, **Lock**, **Ready for issuing** and **Master drawing** the following values can be searched for in invariant search and in normal search:

- `True`
- `False`

Create search-based document categories

There are default categories in **Document manager**, and you can create categories of your own by saving search strings.

To create a search-based category:

1. Enter a search string that finds the documents you want to categorize.

   For more information about the allowed wildcards and operators, see the searching instructions above.
2. Right-click in the category list on the left and select **New**.
3. Define the category properties:
    - **Name**: Enter the name to be shown in the category list. If you use a translation key beginning with the prefix albl_, the name is translated if the translation is found in `DocumentManager.ail` (in `\messages \DotAppsStrings\`). When a valid translation key is used, the translation will be shown in the dialog box.
    - **Search string**: Enter here the search string that gives the desired list of documents. The default for the new category is the active search string. You can test the string by clicking the **Test** button. Do not leave the Search string box empty.
    - **Tooltip (optional)**: Text that is displayed when you move the mouse pointer over a category. If you use a translation key beginning with the prefix albl_, the tooltip is translated if the translation is found in

DocumentManager.ail. When a valid translation key is used, the translation will be shown in the dialog box.

- **Group**: Enter a number that specifies the group of the category. There is a separator in the category list between the different groups.

  The default value for a new category is 0. Negative values are allowed.

- **Sort index**: Enter a number that specifies the location of the category in the category list, and the smaller the number is, the higher the category is on the category list. Categories are sorted first by **Group**, then **Sort index** and then by **Name**.

  The default value for a new category is 0. Negative values are allowed.

- **Combining operator**: Enter AND or OR. When you select several categories at a time, categories with OR rule are combined first and then the ones with AND rule.

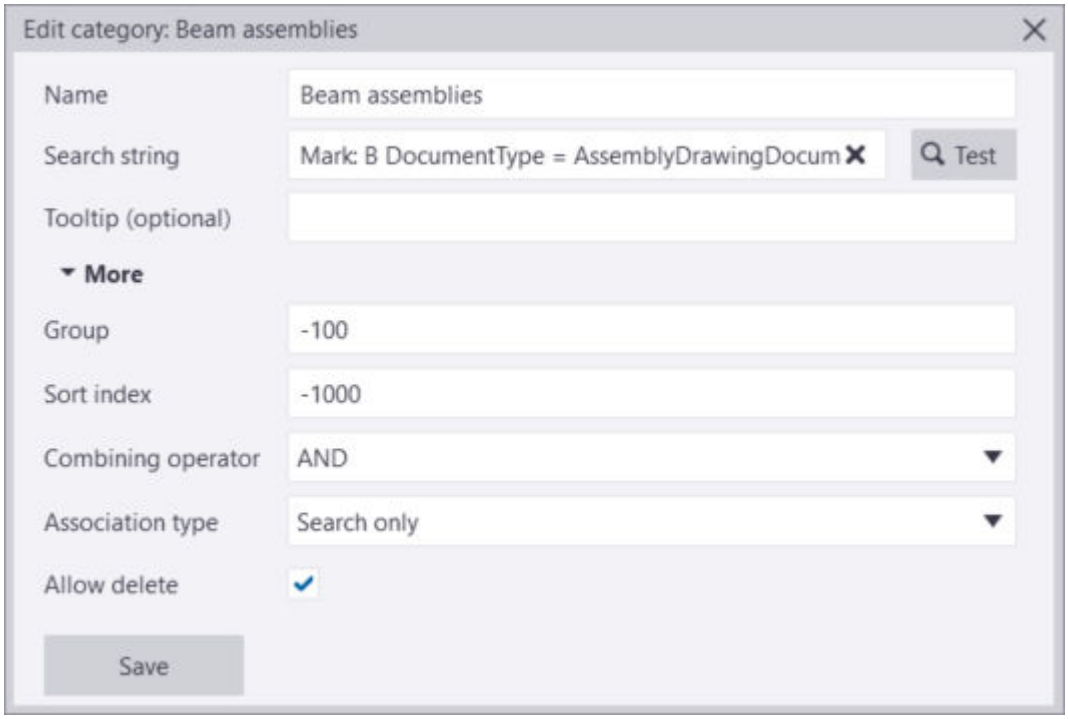
- **Association type**: Change the category association type. The options are **Search only**, **Manual only**, and **Manual and search**. For more information about changing the category type, see section "Change category association type" below.

- **Allow delete**: If you select this, the **Delete** command will be available for this category when you right-click the category in the categories list.

All changes in categories are saved in DocumentManagerCategories_<user>.xml in the current model folder. You can rename the file to DocumentManagerCategories.xml and move it to a folder defined with the advanced options XS_PROJECT, XS_FIRM or XS_SYSTEM, or keep it in the current model folder. The default categories are defined in DocumentManagerCategories.xml located in the system folder defined by the advanced option XS_SYSTEM.

4. Click **Save**.

Below is an example of a category that lists all assembly drawings that have the character `B` in the mark:



Create manual document categories

Sometimes you may need to have a document category in **Document manager** for documents that would be difficult to categorize using the search. You can simply select documents, create a manual category and add your documents there. You can also add documents to and remove documents from existing manually created categories, and combine categories, both search-based and manual ones. A manual category may contain both drawings and file documents.

Creating a manual category may be useful, for example, when you have enabled the **Show changes from the checkpoint set when the button is activated** feature in **Document manager**, and then when some documents have changed and are listed, you can add the changed documents in the manual category.

TO	Do this
Create a new manual category and add documents there	Select the documents from the list, right-click and select **Manual category** --> **Add to new** . Then type a name for the category, an optional tooltip,

TO	Do this
	and define other necessary settings. Then click **Save**. Options are the same as in the search-based categories, except that the functionality related to search strings, and the **Test** button are missing if there are documents associated. If there are no documents associated, then the search controls are visible and the category could become a search-based category or a manual category. The newly created or modified manual categories are saved to the `DocumentManagerCategories_<user>.xml` file under the current model folder.
Save recently changed documents in a new manual category	When you have enabled the **Show changes from the checkpoint set when the button is activated** switch in **Document manager**, and when you disable this switch, you will get the following message: **Turning off 'Show changes' will reset the changes checkpoint and lose the history of which documents have changed since the checkpoint was set. Are you sure you wish to continue?**. Select **Create new category**. You can also select to turn the switch off and lose the history.
Edit a manual category	Right-click the category in the category list and select **Edit**.
Add documents to an existing manual category	Select the documents from the list, right-click and select **Manual category** --> **Add to existing** and then select the desired category. All compatible manual categories are listed. You can add the selected documents to several categories.
Remove documents from a manual category	Select the documents from the list, right-click, select **Manual category** --> **Remove from** and then select the category from which to remove the documents. You can also select manual categories first so that all documents in those categories are listed, right-click the list, and select **Manual category** --> **Remove from selected** .

TO	Do this
Delete a manual category	Right-click the category in the category list, and select **Delete**.

Note that if a document ID cannot be found when loading previously saved manual categories, an entry will be written to the **Document manager** log identifying the manual category and the document ID of the missing document. This situation can arise if a drawing or file document is deleted when **Document manager** is not open and that document was referenced in a manual category.

Change category association type

You can use the setting **Association type** in the **Edit category** and **New category** dialog boxes to define if a category is search based, manual, or both.

- If the category is of type **Search only** or **Manual and search**, you need to define a **Search string** in the dialog box.
- If you create a category by first selecting documents from the documents list, the association type is **Manual only** by default.
- If you create a category from the category list, the association type is **Search only** by default.
- You can change the association type later. If you change the type from **Search only** to **Manual only**, a message is displayed telling that the **Search string** for that particular category will be cleared. If you change the type from **Manual only** to **Search only** or **Manual and search**, you need to define a **Search string**.

Find model objects that have associated drawings in Document manager

You can use **Document manager** for identifying the objects in the model that have associated drawings.

Before you start, ensure that you only have the **Select parts** selection switch active. Otherwise part selection may take a long time in large models.

1. Open a model view where all objects are clearly visible.
2. Do one of the following:
   - Highlight the objects associated to individual drawings by selecting the individual drawings in **Document manager** and clicking **Select objects in the model for selected drawings**.

- Highlight all objects that have associated drawings by selecting all drawings in **Document manager** (**Ctrl+A**) and clicking **Select objects in the model for selected drawings**

TIP
- To see the found objects more clearly in the model, right-click the model and select **Show only selected** to only show the parts that the **Select objects in the model for selected drawings** command has found and hide all others.
- After completing the step above you can highlight the objects that do not have drawings by holding down the **Ctrl** key and selecting the entire model (hold down the left mouse button and drag all the way from left to right).

Find drawings associated to objects selected in model

You can list drawings associated to the objects you have selected in the model only. This is an easy way to identify the drawings associated with a specific part, assembly or cast unit.

1. Open a model view where all objects are clearly visible.
2. Select the desired objects in the model.
3. In **Document manager** , click **Select and show only drawings containing parts currently selected in the model**.

   Tekla Structures only selects and shows the drawings associated with the objects that you selected in the model. The result also depends on the selected category, which means that if you only have the GA drawings category selected, GA drawings are only shown in the list.

TIP You can also show drawings for the selected model objects in **Document manager** by using the contextual toolbar: Click **Open or create drawings** and select **Show drawings for selected objects...** from the displayed menu.

Open documents

To open a drawing, do any of the following:

- Double-click the drawing.
- Select the drawing from the list and click the **Open** button at the bottom.

- Right-click the drawing in the list and select **Open**.
- To open the previous or next drawing in the list, click the **Open previous drawing** button or the **Open next drawing** button at the bottom.

You can also reopen an updated drawing:

- When you have a drawing open and you have edited it manually, open the **Document manager**, and open the same drawing again.

If you have a related software installed, you can also open other documents, such as NC files, DWG files, or PDF files. Do any of the following:

- Double-click the document.
- Select the document from the list and click the **Open** button at the bottom.
- Right-click the document in the list and select **Open**.

To open a folder of the selected file document in Windows Explorer, right-click the document and select **Open folder**.

Issue, lock, freeze and mark ready for issuing

- You can lock, freeze or mark ready for issuing by selecting the drawing from the list and clicking the appropriate column next to the drawing. When you do this, first ensure that the edit list switch is active. You can also right-click drawings and select the commands from the context menu, or use the buttons at the top for quick access. Note that you can hide the buttons at the top by clicking the arrow next to the buttons.

For more information about these functionalities, click the following links:

Mark drawings ready for issuing (page 578)

Issue drawings (page 579)

Freeze drawings (page 577)

Lock drawings (page 576)

Revise drawings

- To revise a drawing, select a drawing and click the **Revision** button. When you revise drawings, you can attach information about the revision changes. Tekla Structures displays this information alongside the revision number or mark. The revision date appears in the revision table. The revision number or mark is shown in the **Document manager**, and the revision information can be included in reports. You can also add revision marks containing revision information manually in an open drawing.

For more information about revising drawings, see Revise drawings (page 580). For details about adding revision marks, see Add revision marks in drawings (page 370).

Copy Document manager rows to clipboard

You can copy data from the **Document manager** list and paste the data in another application, such as a spreadsheet application:

1. Select a number of documents in the **Document manager** list.
2. Right-click and select **Copy rows to Clipboard**.
3. Open the other application and paste the row data there.

Create GA drawings

You can create GA drawings quickly and easily in **Document manager**:

1. In **Document manager**, click **Create GA Drawing**. The **Create general arrangement drawing** dialog box is displayed.
2. Select the model views of which you need GA drawings, and then create the drawings by clicking **Create**.

Configure file documents to include in Document manager

The configuration file `DocumentManagerFileDocumentSettings.txt` defines which file documents are included in **Document manager**. This file also specifies the folders to search for. This file can be located in any of the standard search locations (model, project, firm and system). By default this file is located in the system folder defined by the advanced option `XS_SYSTEM`, and you can copy it from there to model, project, firm or system folders for modifications.

The format of each line is `<folder_path>\*.<file_name_extension>|<include_subfolders>` , where

- `<folder_path>` is a relative or absolute path. A relative model subfolder can be specified using a leading period (.).
- `*.<file_name_extension>` specifies the accepted file types. You can use standard wildcards ? and *.
- `<include_subfolders>` specifies if the subfolders are included. Set to `true` to include subfolders and to `false` to not to include subfolders.

Examples:

```
.\Plotfiles\*.dxf|false
.\DSTV_Profiles\*.nc1|false
.\DSTV_Plates\*.nc1|false
```

Configure drawing UDAs to show in Document manager columns

The `DocumentManagerUDAs*.txt` files define the drawing UDAs that are shown in the **Document manager** columns. The file name can be either `DocumentManagerUDAs.txt` or `DocumentManagerUDAs_<suffix>.txt` where `<suffix>` can be any string. These files can be placed in a folder defined with the advanced options `XS_FIRM`, `XS_PROJECT`, `XS_SYSTEM` or `XS_INP`, and in the model folder. **Document manager** also searches for `DocumentManagerUDAs*.txt` files recursively in extension folders and subfolders. By default, `DocumentManagerUDAs.txt` is located in the `\inp` folder defined by the advanced option `XS_INP`, and you can copy it from there to the other supported folders for modifications. The contents of each file found are merged together.

List all the drawing UDAs that are needed in this file, one UDA on each line. The UDA names are case sensitive. The UDA must exist in the `objects.inp` file.

Below is an example of the `DocumentManagerUDAs.txt` file in the Common environment:

```
DR_APPROVAL_DATE
DR_APPROVAL_SENT
DR_APPROVED_BY
DR_ASSIGNED_BY
DR_ASSIGNED_TO
DR_CHECKED_BY
DR_CHECKED_DATE
DR_DRAWN_BY
```

`comment`

For more information about UDAs in drawings, see User-defined attributes in drawings (page 895).

Document manager log

All errors occurring in the **Document manager** functionality are written in the **Document manager** log file `\logs\DocumentManager_<user>.log` under the current model folder. This log file also lists the full paths of all the settings files that have been loaded (`DocumentManagerUDAs*.txt`, `DocumentManagerFileDocumentSettings.txt`, `DocumentManagerDataGridSettings*.xml` and `DocumentManagerCategories*.xml`), which helps you to see which settings files **Document manager** is finding. The log also tells if a settings file cannot be found.

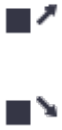
The log file is limited in size, and will be archived as `\logs \DocumentManager_<user>.bak.log` when the maximum size has been reached.

Example:

```
Loading UDA settings file C:\TeklaStructuresModels\New model
3\DocumentManagerUDAs.txt
Loaded column layout settings from file: C:\TeklaStructuresModels\New
model 3\DocumentManagerDataGridSettings_user.xml
Loaded frozen column name: Title1 from file: C:\TeklaStructuresModels\New
model 3\DocumentManagerDataGridSettings_user.xml
Loaded the column sort order settings from file: C:\TeklaStructuresModels
\New model 3\DocumentManagerDataGridSettings_user.xml
Loaded 16 document categories from file: C:\TeklaStructuresModels\New
model 3\DocumentManagerCategories_user.xml
No DocumentManagerFileDocumentSettings.txt file found. File documents
will not be shown.
```

Drawing status flags and related status messages

Flag	Column where flag displayed	What's shown in Changes column	Description
		Parts modified	The parts in the drawing have changed, for example, parts have been added or deleted, or part properties have changed.

Flag	Column where flag displayed	What's shown in Changes column	Description
		**Quantity increased** or **Quantity decreased**	The actual drawing is up to date, but the number of identical parts has changed.
		All parts deleted	All parts related to the drawing have been deleted from the model.
			The drawing is locked and you cannot open it for editing. You can check who marked the drawing from the **Locked by** column.
			The drawing is frozen. Changes made to the model objects, which have drawing objects associated with them, are no longer available in the drawing.
			The drawing has been added as a master drawing in the **Master Drawing Catalog**.
		Drawing was updated	The drawing is marked ready for issuing. You can check who marked the drawing from the **Ready for issuing by** column.
		Drawing was updated	The drawing has been updated.

Flag	Column where flag displayed	What's shown in Changes column	Description
		Linked drawing changed	You have a linked drawing in this drawing and the linked drawing has been modified.
		Copied view changed	A copied view has been modified.
		Drawing was cloned	The drawing is a cloned drawing. The flag disappears when you save and close the drawing.
			The drawing has been issued. You might want to re-issue drawings that have been sent to site.
		Issued drawing changed	The issued drawing has been edited or otherwise changed.

Other available functionalities in Document manager

In **Document manager**, the following functionalities are also available:

- Print to a PDF file, plot file (.plt) or printer (page 603)
- Revise drawings (page 580)
- Create multidrawings (page 118)
- Copy a drawing to a new sheet (page 164)
- Clone from Document manager (page 153)
- Add a cloning template master drawing in Master Drawing Catalog (page 142)
- Delete drawings and file documents (page 590)
- Version control for drawings (page 584)
- Update part and weld marks in drawings (page 336)

- Modify drawing properties of an existing drawing (page 77)
- User-defined attributes in drawings (page 895)
- Snapshots in drawings (page 170)

5.2 Update drawings when the model changes

Saved drawings need updating because of model changes. Cast-unit, assembly, single part and multidrawings use position numbers as identifiers. Many changes affect numbering, so many times renumbering is needed. Before updating drawings you need to ensure that the numbering is correct.

The drawings needing updates are indicated by flags in **Document manager**. Renumbering model objects after creating drawings may also generate flags.

NOTE
- General arrangement drawings do not require model numbering to be updated. If you use part position numbers in part marks, you must number the model to get updated marks, because old and not up-to-date part marks have question marks in them. If you do not use numbering, you can modify general arrangement drawings without numbering. Then, for example, profile changes are highlighted with change symbols.
- When you update multidrawings, also the linked drawings are updated.

Do the following after changing the model:

1. Check the numbering settings by going to the **Drawings & reports** tab and clicking **Numbering settings** --> **Numbering settings** .
2. Select **Compare to old** for both the new and modified parts.
3. Number all model objects that have same numbering series settings on the **Drawings & reports** tab by clicking **Perform numbering** --> **Number series of selected objects**, or number only new or modified model objects by clicking **Perform numbering** --> **Number modified objects** .
4. Check the **Document manager** for status flags.
5. To find the parts that have been affected, select each drawing marked with the flag in the **Document manager** and click the **Select objects** button.

   Tekla Structures highlights the affected parts in the model.

   Do the following:

   a. Check the numbering history log for renumbered parts by going to the **File** menu and clicking **Logs** --> **Numbering history log** .

`Part` or `Assembly` at the beginning of a line in the numbering history log indicates that Tekla Structures has renumbered parts or assemblies, as in the following example:

```
Part     guid: ID56CC370F-0000-027E-3134-353633303233  series:MC/1  MC/0 -> MC/1
Part     guid: ID56CC370F-0000-0282-3134-353633303233  series:MC/1  MC/0 -> MC/2
Part     guid: ID56CC370F-0000-0286-3134-353633303233  series:MC/1  MC/0 -> MC/3
Part     guid: ID56CC370F-0000-028A-3134-353633303233  series:MC/1  MC/0 -> MC/2

Assembly guid: ID56CC370F-0000-027D-3134-353633303233  series:C/1  C/0 -> C/1
Assembly guid: ID56CC370F-0000-0281-3134-353633303233  series:C/1  C/0 -> C/2
Assembly guid: ID56CC370F-0000-0289-3134-353633303233  series:C/1  C/0 -> C/2
Assembly guid: ID56CC370F-0000-0285-3134-353633303233  series:C/1  C/0 -> C/3
Assembly guid: ID56C42A49-0000-0022-3134-353536393636  series:C/1  C/0 -> C/4
```

b. To find the renumbered parts in the model, select the relevant entries from the numbering history log. Tekla Structures highlights the corresponding parts in the model.

6. Select the affected drawings from **Document manager** and click **Update**.
7. If you have new parts in the model, create drawings for them.

See also

Manage drawings (page 551)
Document manager (page 552)
Recreation of drawings (page 88)

5.3 Lock drawings

You can indicate that a drawing is not available for editing by locking it. When a drawing is locked, it cannot be accidentally modified. The geometry of the locked drawing still changes when the model changes.

Tekla Structures flags locked drawings for update if the model changes.

1. In **Document manager**, select the drawing you want to lock.
2. Ensure that the edit switch is active.
3. Click the **Lock** column next to the selected drawing.

   You can also select this command from the context menu, or click the **Lock** button at the top. When you do this, you can lock several drawings at a time.

   The **Locked by** column in the **Document manager** shows who has locked the drawing. If you have logged in to your Trimble Identity, your account name is shown. Otherwise your user name is shown.

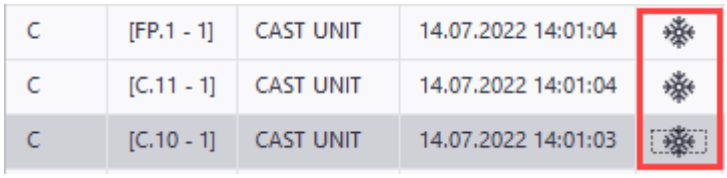
4. To unlock the drawings, click the **Lock** column again, or select the locked drawings and click the **Unlock** button at the top.

See also

Document manager (page 552)

5.4 Freeze drawings

You can select whether to allow Tekla Structures to update all associative objects on top of the drawing views. The geometry of the model is always updated, but freezing is used to stop the drawing intelligence (associativity) of drawing objects on top of the model views, and prevent them from being updated. For example, parts are updated, but dimensions, marks, views, and drawing shapes are not. Use freezing only to avoid changes in drawings while updates are made in the model, not just for noting that the drawing has been edited. You can freeze/unfreeze several drawings at a time.



How freezing affects drawings

Freezing affects drawings in the following way:

- The associativity does not disappear from a frozen drawing. When you unfreeze the drawing, the associativity works again.
- Freezing does not have any effect on the cloning result. If you edit the drawing, it does not matter whether you freeze the drawing before or after editing it.
- If a drawing is frozen, the associative drawing objects are not updated when the drawing is updated. This means that the dimensions and views are not updated, and the marks do not follow the parts if the parts have been moved.
- If a drawing is frozen, and the part changes in the model, the geometry of the part is updated in the frozen drawing when the drawing is updated.
- Unfreezing the drawing before cloning does not have any effect on the cloning result. This means, for example, that it does not matter whether you keep the drawing frozen all the time or temporarily unfreeze it before cloning.
- If you unfreeze a drawing before update, the drawing is updated normally.

Freeze general arrangement drawings

1. If you have the GA drawing open, close it without saving it.
2. In **Document manager**, select the drawing you want to freeze.
3. Ensure that the edit switch is active.
4. Click the **Freeze** column next to the drawing you selected.

   You can also select this command from the context menu, or click the **Freeze** button at the top. When you do this, you can freeze several drawings at a time.
5. When freezing is no longer necessary, select the frozen drawings and click the **Freeze** column again, or click the **Unfreeze** button at the top.

Freeze single-part, cast unit and assembly drawings

1. Save the model.
2. Open a drawing.
3. If the drawing does not look the way you want, close it without saving it.
4. Reopen the model. Do not save it.
5. In the **Document manager**, select the drawing you want to freeze.
6. Click the **Freeze** column next to the drawing you selected.

   You can also select this command from the context menu, or click the **Freeze** button at the top. When you do this, you can freeze several drawings at a time.
7. Open the drawing.
8. When freezing is no longer necessary, select the frozen drawing and click the **Freeze** column again, or click the **Unfreeze** button at the top.

5.5 Mark drawings ready for issuing

When a drawing is ready to be released for fabrication, you can mark it ready for issuing by using the option **Ready for issuing** in **Document manager**. When there are changes in the model, the geometry of the drawings that are marked ready for issuing is updated. You can also edit the marked drawings just like any other drawings.

1. In the **Document manager**, select the drawing to be marked ready for issuing.

2. Ensure that the edit switch is active.

3. Click the **Ready for issuing** column next to the drawing that you want to mark.

   You can also select this command from the context menu, or click the **Mark drawing ready for issuing** button at the top. When you do this, you can mark several drawings at a time.

The drawings are marked ready for issuing, and flagged with

in the **Ready for issuing** column in **Document manager**. You can check who marked the drawing from the **Ready for issuing by** column.

If you want to cancel the marking, use the **Unmark drawing ready for issuing** button at the top.

TIP You can also mark a drawing ready for issuing by selecting the **Mark drawing ready for issuing** check box in the save confirmation message box, which appears when you close an unsaved drawing.

You can add in your reports the information whether the drawing is marked ready for issuing, and who has marked it using the template attributes and

See also

Document manager (page 552)

5.6 Issue drawings

When a drawing has been released for fabrication, it should be marked as issued in **Document manager**. The geometry of the issued drawings is updated when the model changes. Issuing only prevents the recreation of the drawing during update.

The issuing information can be used to filter what is shown in the **Document manager** and in templates.

1. In **Document manager**, select the drawings to be issued.
2. Right-click and select **Issue** --> **Issue** , or click the **Issue** button at the top.

   Tekla Structures marks the issued drawings with a flag

When an issued drawing has been edited or otherwise changed, the color of the flag changes to and the text **Issued drawing changed** is displayed in **Document manager**. You can reissue the drawing by right-clicking the drawing and selecting **Issue** --> **Issue** .

To unissue a drawing, select the drawing, right-click and select **Issue** --> **Unissue**, or click the **Unissue** button at the top.

TIP To show the issue date in a report, add the template field in the appropriate report template.

See also

Manage drawings (page 551)

Document manager (page 552)

5.7 Revise drawings

When revising drawings, you can attach information about the revision changes. Tekla Structures displays this information alongside the revision number or mark. The revision date appears in the revision table. The revision number or mark is shown in **Document manager**, and the revision information can be included in reports. You can also add manual revision marks in an open drawing.

Type	Mark	Revision	Name	Title 1	Created ▼
A	[C.8]		STANDARD		14.07.2022 14:01:01
A	[C.9]	1	STANDARD		14.07.2022 14:01:00
A	[C.5]		STANDARD		14.07.2022 14:01:00

TIP One example of the report using revision information is `drawing_issue_rev.xsr`, which shows the most recent revision dates of drawings.

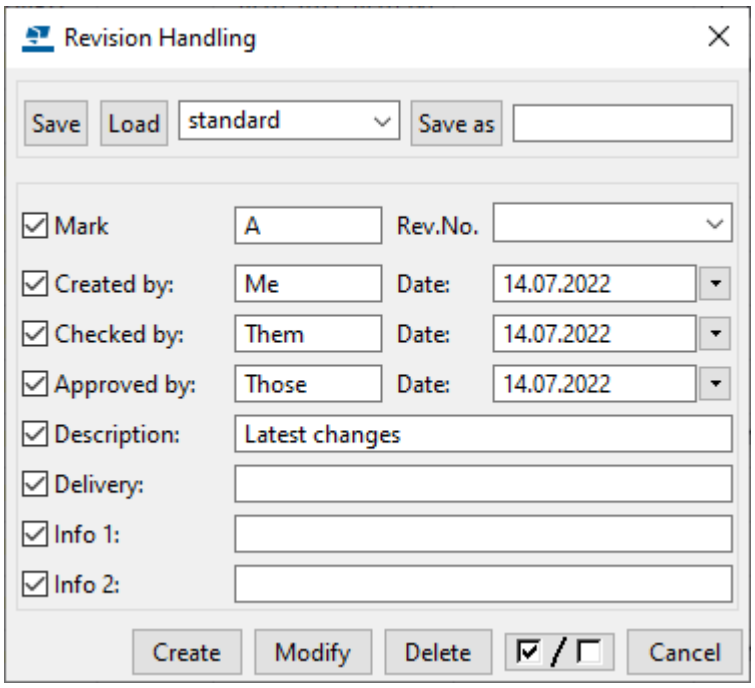
You can show the revision information in revision marks in drawings. For more information about adding revision marks, see Add revision marks in drawings (page 370).

Create drawing revisions

You can create revisions to follow the changes in the drawing using the **Revision** command in the **Document manager**.

1. In **Document manager**, select the drawings to revise.
2. Click **Revision**.

   The **Revision Handling** dialog box is displayed.



3. Enter a mark in the **Mark** box.

   Marks can be numerical or alphabetical such as 1, 2, 3... or A, B, C...
4. Fill in the **Created by** information and select the creation date from the calendar that is displayed when you click the down-arrow next to the **Date** option.
5. If you need to give information about who checked and approved the drawing, enter the information in the corresponding boxes and select the appropriate dates.
6. Enter a revision description in the **Description** box.
7. Enter delivery information in the **Delivery** box.
8. Enter any additional information in the **Info 1** and **Info 2** boxes.
9. Click **Create**.

The revision number or mark is now displayed in the **Document manager**. When you open the drawing, you can see the revision information in the revision table if there is one in the drawing.

TIP Each drawing has its own unique revision number, but several drawings can share the same revision mark, date, and other information. To attach the same

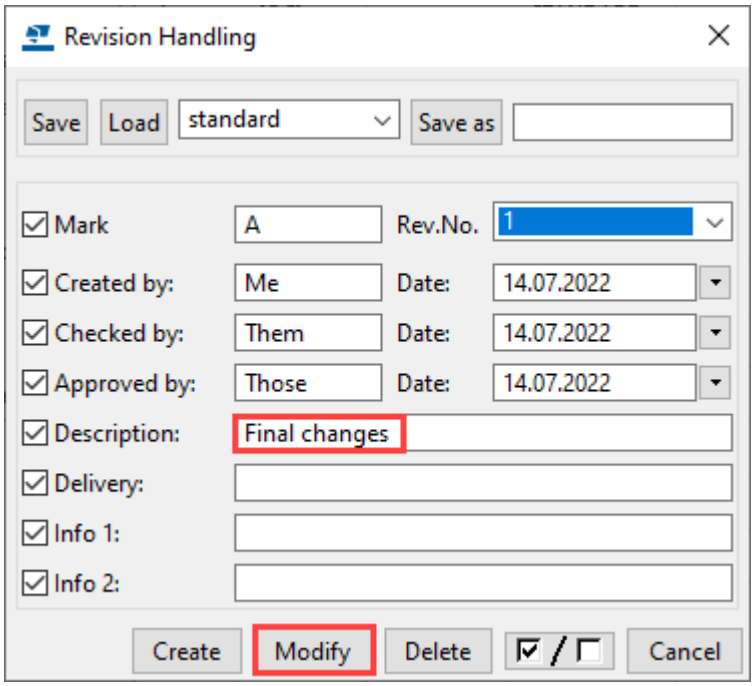
revision information to several drawings simultaneously, select multiple drawings from **Document manager** and then click the **Revision** button.

To show the revision mark instead of the revision number in the **Document manager**, set the advanced option XS_SHOW_REVISION_MARK_ON_DRAWING_LIST to `TRUE`.

Change drawing revisions

You can change revision information in an existing revision.

1. Open the **Document manager** and select a revised drawing.
2. Click **Revision**.
3. In the **Revision Handling** dialog box, select the number of the revision that you want to change from the **Rev.No.** list.
4. Change the revision information as required.
5. Click **Modify**.



6. Close the **Revision Handling** dialog box.

Delete drawing revisions

You can delete unnecessary drawing revisions.

1. Open **Document manager**, and select a revised drawing.
2. Click **Revision...**.

3. In the **Revision Handling** dialog box, select the revision number from the list next to the **Mark** box.
4. Click **Delete**.

   When you delete a revision, Tekla Structures automatically adjusts the remaining revision numbers for that drawing. The revision marks remain the change.

Attributes used in drawing revisioning

Drawing revisioning attributes can be included in drawing templates and report templates.

All revisioning attributes and the corresponding options in the **Revision Handling** dialog box are listed below.

Revision handling attribute	Option in the Revision Handling dialog box
MARK	The revision mark in the **Mark** box.
NUMBER	The revision number in **Rev.No.**
CREATED_BY	The **Created by** information of the revision.
DATE_CREATE	**Date** next to **Created by**.
CHECKED_BY	The **Checked by** information of the revision.
DATE_CHECKED	**Date** next to **Checked by**.
APPROVED_BY	The **Approved by** information of the revision.
DELIVERY	The **Delivery** information of the revision.
DESCRIPTION	The revision **Description**.
DATE_APPROVED	**Date** next to **Approved by**.
INFO1	The **Info 1** text of the revision.
INFO2	The **Info 2** text of the revision.
LAST	The revision number of the latest revision in **Rev.No.**.
LAST_CREATED_BY	The **Created by** information of the latest revision.
LAST_CHECKED_BY	The **Checked by** information of the latest revision.
LAST_DATE_CHECKED	The **Checked by Date** of the latest revision.

Revision handling attribute	Option in the Revision Handling dialog box
LAST_DATE_APPROVED	The approval **Date** of the latest revision.
LAST_DELIVERY	The **Delivery** information of the latest revision
LAST_MARK	The revision mark of the latest revision in the **Mark** box.
LAST_DESCRIPTION	The **Description** of the latest revision.
LAST_DATE_CREATE	The **Created by** date of the latest revision.
LAST_APPROVED_BY	The **Approved by** information of the latest delivery.
LAST_INFO1	The **Info 1** text of the latest revision.
LAST_INFO2	The **Info 2** text of the latest revision.

5.8 Version control for drawings

Version control is available for all drawing types allowing you to review what a drawing was at a certain stage. You can list different versions of the same drawing, open drawing versions, show their snapshots, and change the current drawing version. You can list drawing versions in other models, and deleted drawing versions that are no longer available in **Document manager**. You can also open a deleted GA drawing as a new drawing.

Listing drawing versions is useful when you want to revert to an older version of the drawing for some reason. Version control is also available in Tekla Model Sharing, and you can control unintentional changes made to drawings and revert to the version that has the correct content.

- Every time you save a drawing, a new version of the drawing is saved as a `.dg` file.

  The advanced option XS_DELETE_UNNECESSARY_DG_FILES defines if the unnecessary drawing files (`.dg` files) are automatically deleted or not. To automatically delete the drawing files that are no longer used, set `XS_DELETE_UNNECESSARY_DG_FILES` to `TRUE`.

  The XS_DELETE_UNNECESSARY_DG_FILES_SAFETY_PERIOD advanced option defines how long the drawing files are kept before they are deleted. The default safety period is 7 days. This means that after 7 days, the drawing files that are not used are deleted when the `XS_DELETE_UNNECESSARY_DG_FILES` advanced option is set to `TRUE`.

  If you want to keep all drawing versions, set `XS_DELETE_UNNECESSARY_DG_FILES` to `FALSE`.

- Note that using **Save as** does not always contain needed drawing history data and drawing update may fail.

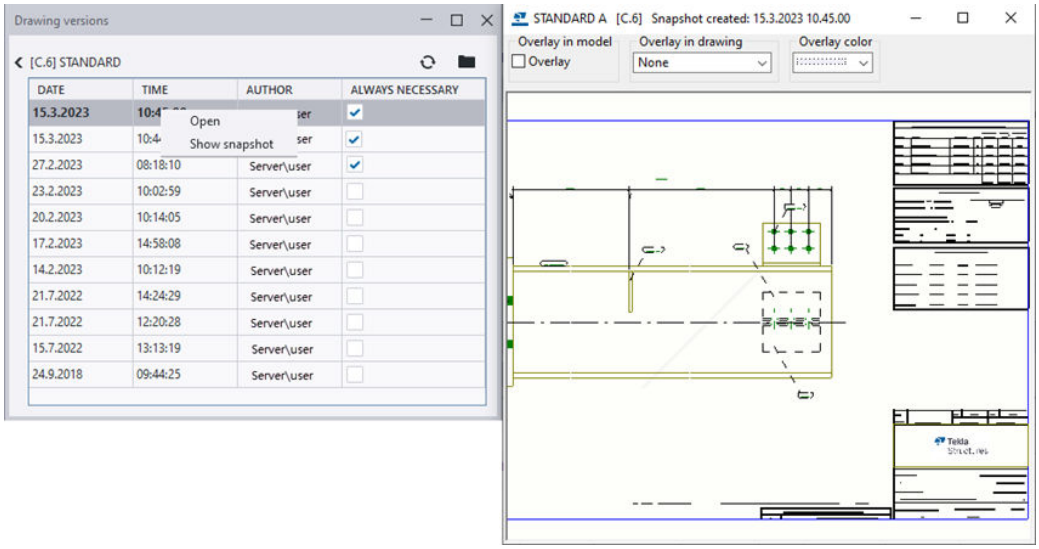
Open the drawing versions list

- To list the different versions of a drawing in the current model, click **Document manager** on the **Drawings & reports** tab, select a drawing from the list and click the **Drawing versions** button at the bottom.

  The current drawing version appears bolded.

  The drawing snapshot is displayed in a separate window. A snapshot is taken automatically by default when you save a drawing. The advanced option to control the automatic saving of snapshots is XS_DRAWING_SNAPSHOT_CREATION.

  Snapshots are not created at drawing creation if the advanced option `XS_CREATE_SNAPSHOT_ON_DRAWING_CREATION` is set to `FALSE`. If you open a drawing and save it, you get a snapshot if `XS_DRAWING_SNAPSHOT_CREATION` is set to `TRUE`.



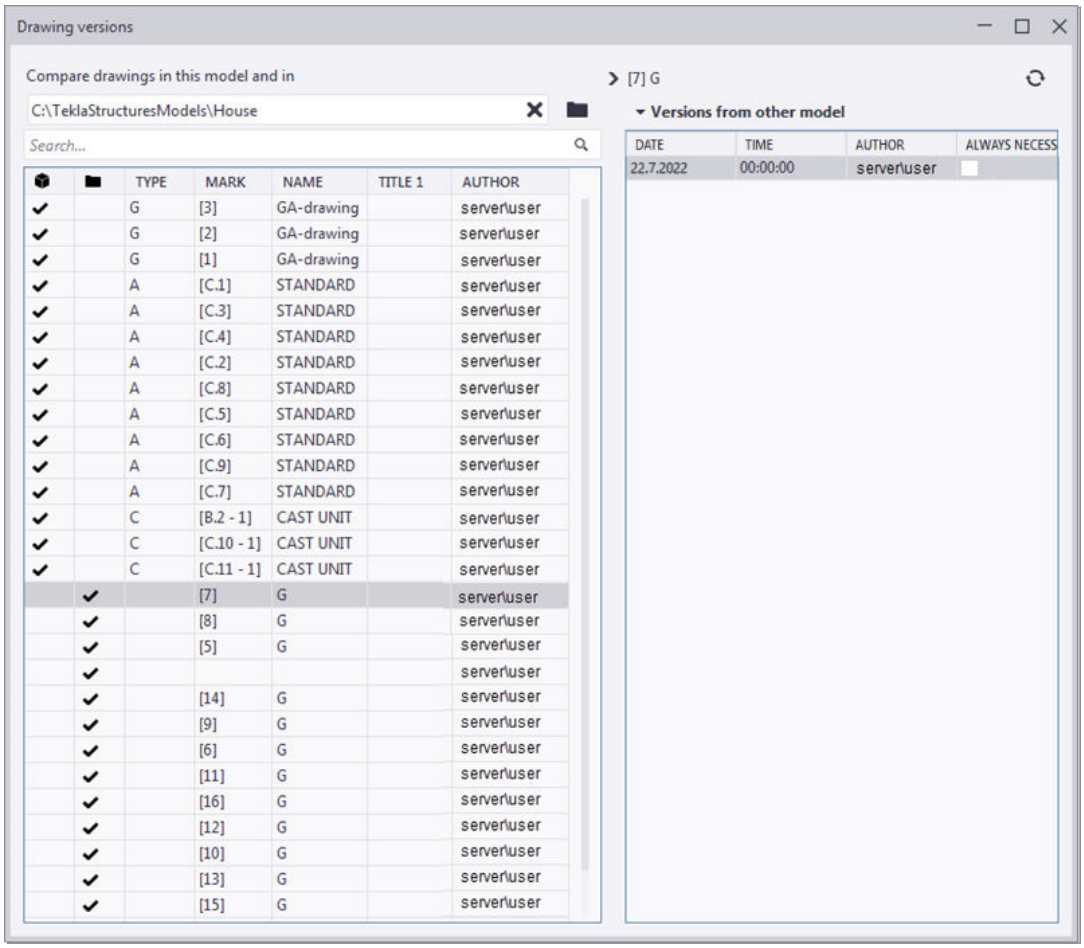
- To list all drawings and their versions related to the model, even the deleted drawings, go to **Document manager**, and click **Drawing versions** without selecting a drawing in the **Document manager** list. In this **Drawing versions** dialog box, you can see the drawing versions, open deleted GA drawing versions as new GA drawings, open the different versions, and show their snapshots.

What can you do with drawing versions

To	Do this
Open a drawing version	1. In the **Document manager** list, select a drawing and click the **Drawing versions** button at the bottom. 2. Right-click a drawing version. 3. Select **Open**.
Show snapshot for a drawing version	1. In the **Document manager** list, select a drawing and click the **Drawing versions** button at the bottom. 2. Right-click a drawing version. 3. Select **Show snapshot**.
Save another drawing version as the current version	1. In the **Document manager** list, select a drawing and click the **Drawing versions** button at the bottom. 2. Open another version of the drawing. 3. Close the drawing version. 4. When you are asked **Do you want to keep the changes to the drawing?**, answer **Yes**. This makes the drawing version the current drawing version.
Open a deleted GA drawing version as a new GA drawing	1. Click the **Drawing versions** button without selecting a drawing in the **Document manager** list. 2. Select a deleted GA drawing version from the list on the right in the **Drawing versions** dialog box, right-click and select **Open as new**.
Show drawing versions from another model	1. Click the **Drawing versions** button without selecting a drawing in the **Document manager** list. The drawings in the current model are listed.

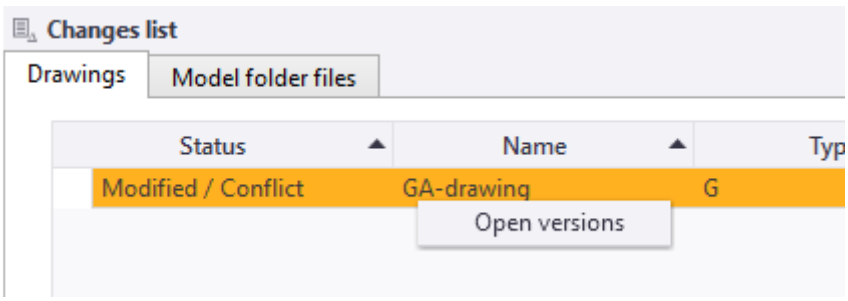
To	Do this
	2. In the upper-left corner, browse to the folder of the other model, and click **Select folder**. The drawings in the other model are listed on the left, and their versions on the right.
Keep the selected drawing versions	As drawing versions might be defined to be automatically removed, you might want to protect some versions from the removal. 1. In the **Document manager** list, select a drawing and click the **Drawing versions** button at the bottom. 2. Select the check box in the **Always necessary** column next to the drawing versions that you want to protect from the removal.

Below is an example of comparing drawings in two models: in the current model and in the model that you select in the upper left corner of the **Drawing versions** dialog box. In the dialog box, you can clearly see which drawings are in which model. The drawing versions are listed on the right.



Drawing versions in Tekla Model Sharing

- In Tekla Model Sharing, when users modify the same drawing in their local version of the model and one user writes out, the **Changes list** shows a conflict in other users' local version of the model when they read in.
- You can open the **Drawing versions** dialog box from the **Changes list**. Select the changed drawing, right-click and select **Open versions** to view the drawing versions and the changes made to the drawing.



- To keep the number of drawing files reasonable, use the XS_DELETE_UNNECESSARY_DG_FILES and XS_DELETE_UNNECESSARY_DG_FILES_SAFETY_PERIOD advanced options.
- If you want to protect some of the drawing versions from being automatically removed when XS_DELETE_UNNECESSARY_DG_FILES is set to `TRUE`, use the **Always necessary** check box in the **Drawing versions** dialog box. Set the **Always necessary** option individually for each drawing version you want to keep by selecting the corresponding check box.

5.9 Delete unnecessary drawing files in single-user mode

By default, all unnecessary drawing files are deleted automatically after seven days. You can delete all your unnecessary drawing files in the single-user mode using the **Remove unnecessary drawing files** command without waiting for Tekla Structures to automatically remove them.

To delete the files, you need to have full privileges. You can find the **Remove unnecessary drawing files** command through **Quick Launch**.

Note that the **Remove unnecessary drawing files** command does not work in models shared with Tekla Model Sharing. Instead, all unnecessary drawings are deleted automatically each time when a user writes out their changes.

NOTE If you have worked with the drawings (edited, deleted) after you saved the last time, remember to save before you use the **Remove unnecessary drawing files** command.

---

1. Search for the command **Remove unnecessary drawing files** in **Quick Launch**.
2. When Tekla Structures finds the command, select it and press **Enter**.

   Tekla Structures deletes all drawings that do not have a corresponding drawing in the database.

TIP By default, the privileges are full. If you want to restrict the command usage, add the following line in the `privileges.inp` file:

```
action:RemoveUnnecessaryDrawingFiles [who] [access]
```

`[who]` is `everyone` or `<Windows_logon_name>`, or `<domain_name>`

`[access]` can be `none`/`view`/`full`.

In the following example, only the administrator can use the **Remove unnecessary drawing files** command:

```
action:RemoveUnnecessaryDrawingFiles everyone none
action:RemoveUnnecessaryDrawingFiles ORGANIZATION\admin full
```

5.10 Delete drawings and file documents

If some drawings or other file documents become unnecessary, you can delete them from **Document manager**.

NOTE In some configurations, deleting drawings or other file documents from **Document manager** is not allowed.

---

1. In the **Document manager**, select the drawings and other file documents you want to delete.
2. Right-click and select **Delete**, or click the **Delete** button at the bottom of the **Document manager** window.
3. When Tekla Structures asks you to confirm the deletion, select **Yes**.

   If you press down **Shift** while you select **Delete**, Tekla Structures will not ask you to confirm the deletion of documents.

   If you have selected both drawings and file documents, you get separate confirmation messages for drawings and file documents.

   You can undo the deletion by clicking **Undo** on the **Quick Access Toolbar** or pressing **Ctrl+Z**.

The file document files are deleted from the respective folders immediately. The `.dg` files are not deleted from the`\drawings` folder immediately, but after seven days by default. If you want to delete them immediately in the single-user mode, enter **Remove unnecessary drawing files** in **Quick Launch**.

The advanced option that controls how often the drawing files will be deleted is XS_DELETE_UNNECESSARY_DG_FILES_SAFETY_PERIOD. The advanced option that controls if the drawing files are deleted automatically is XS_DELETE_UNNECESSARY_DG_FILES.

If you work in the multi-user mode, see Delete unnecessary drawing files in multi-user mode.

Note that in multi-user mode, when drawings are created, they become as duplicates in **Document manager**. One of them will also be marked issued. If you delete one entry, the duplicate one will also be deleted. This applies to all types of drawings. When you save and reopen the model, the duplicate drawings in the list become single entries again.

TIP If you have deleted some GA drawings, and you want to have the GA drawings renumbered, in modeling mode, enter **Renumber all GA drawings** in **Quick Launch** and press **Enter**. This command renumbers all GA drawings starting from number 1 in the order that they were originally created. Use this command in single-user mode only.

Document manager list before running the **Renumber all GA drawings** command:

G	[9]		GA-drawing		24.09.2018 08:45:49
G	[6]		GA-drawing		24.09.2018 07:54:06
G	[5]		GA-drawing		24.09.2018 07:54:00
G	[4]		GA-drawing		24.09.2018 07:53:48
G	[2]		GA-drawing		24.09.2018 07:53:27
G	[1]		GA-drawing		24.09.2018 07:53:22

Document manager list after running the **Renumber all GA drawings** command:

G	[6]		GA-drawing		24.09.2018 08:45:49
G	[5]		GA-drawing		24.09.2018 07:54:06
G	[4]		GA-drawing		24.09.2018 07:54:00
G	[3]		GA-drawing		24.09.2018 07:53:48
G	[2]		GA-drawing		24.09.2018 07:53:27
G	[1]		GA-drawing		24.09.2018 07:53:22

See also

Manage drawings (page 551)

Document manager (page 552)

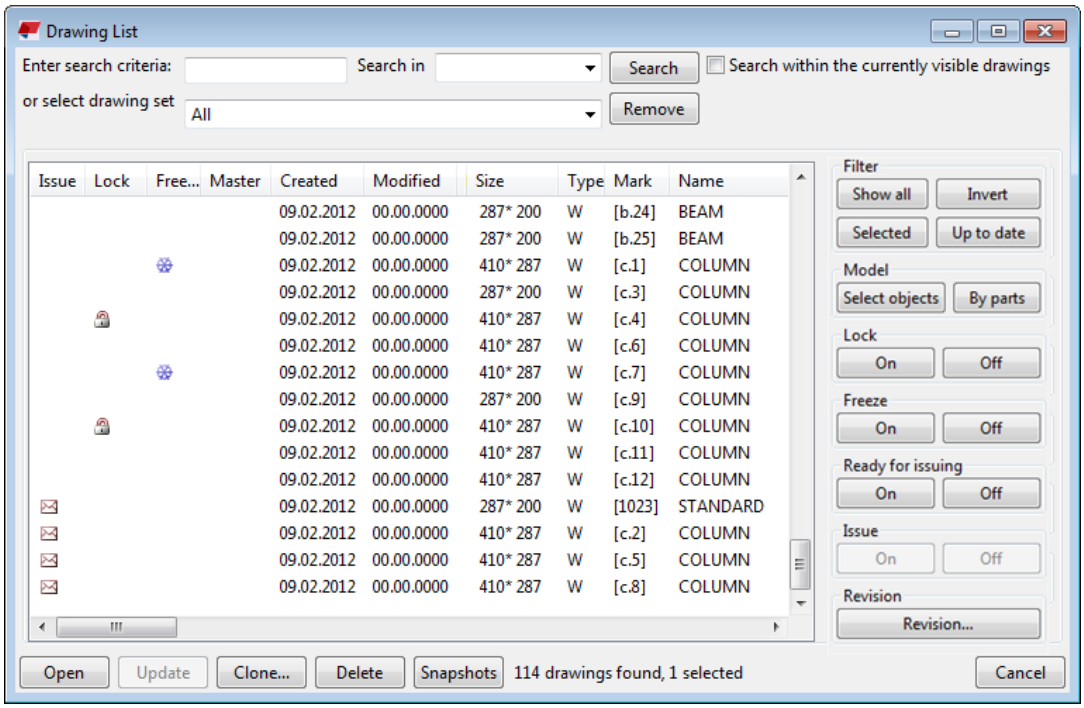
5.11 Drawing list

All drawings in a Tekla Structures model are displayed in the **Drawing list** dialog box. You can search, sort, select, and display drawings according to different criteria, and open the drawings displayed in the list. You can take snapshots of the drawings, and locate the drawing parts in the model, and to find out if a part is included in any of the drawings.

Document manager is enabled by default instead of **Drawing list**. All commands and buttons that would have launched **Drawing list** in earlier Tekla Structures versions will from version 2018i onwards launch **Document manager**. If you want to enable the old **Drawing list** instead, set the advanced option `XS_USE_OLD_DRAWING_LIST_DIALOG` to `TRUE` in the **Drawing properties** category of the **Advanced options** dialog box. If you do this, **Document manager** will be disabled.

For details about **Document manager**, see Document manager (page 552).

For details about issuing, freezing, locking and revising drawings, see Manage drawings (page 551).



Open Drawing list

To open the **Drawing list** dialog box, do one of the following:

- In the model, on the **Drawings & reports** tab, click **Document manager** (**Ctrl + L**).
- In an open drawing, on the **Drawing** tab, click **Document manager** (**Ctrl + O**).
- Enter `document manager` in **Quick Launch**. You can also enter `drawing list`.
- You can also customize the shortcut. You can find it in the **Keyboard shortcuts** dialog box by searching either `document manager` or `drawing list`.

What is displayed in the Drawing list

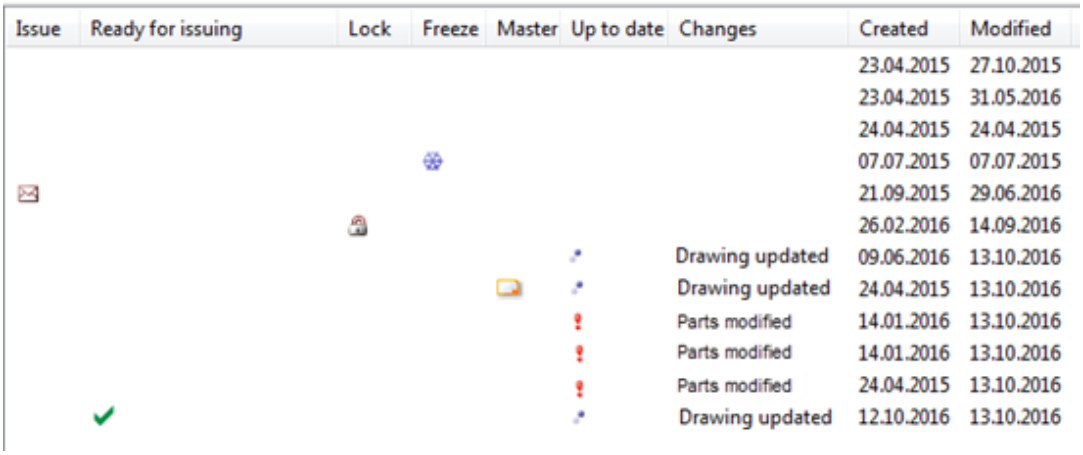
Option	Description	For more information
Issue, **Ready for issuing**, **Lock**, **Freeze**, **Master**, and **Up to date**	These columns contain flags indicating the drawing status.	Manage drawings (page 551)
Changes	Textual information about the changes in the drawing. For example, if the drawing has been cloned, it says **Cloned** in this column.	

Option	Description	For more information
Created	The creation date of the drawing.	
Modified	The latest modification date of the drawing.	
Revision	The revision number or revision mark of the drawing. By default, Tekla Structures shows revision numbers. To show revision marks instead, set the advanced option `XS_SHOW_REVISION_MARK_ON_DRAWING_LIST` to `TRUE`.	Revising drawings (page 580)
Type	The drawing types are identified by the following letters: • **W** for single-part drawings. • **A** for assembly drawings. • **C** for cast unit drawings. • **G** for general arrangement drawings. • **M** for multidrawings. **U** (unknown) means that an error has occurred, and you need to delete the drawing.	Drawing types (page 89)
Size	The paper size the drawing uses.	
Mark	In a single-part drawing mark is the part position, and in an assembly drawing the assembly position from which the drawing was created.	XS_SHOW_REVISION_MARK_ON_DRAWING_LIST

Option	Description	For more information
	You cannot change drawing marks. Mark contains the sheet number as well, if it is not zero (0). You can choose to have Tekla Structures base the mark of cast unit drawings on the position number or the ID (GUID) of the cast unit.	
Name	The name given for the drawing in the drawing properties dialog box.	Rename drawings (page 167)
Title 1, **Title 2**, and **Title 3**	Extra drawing titles added in the drawing properties dialog box.	Giving titles to drawings (page 167)
Ready for issuing by	Indicates who has marked the drawing ready for issuing.	Mark drawings ready for issuing (page 578)
Locked by	The **Drawing list** column shows who locked the drawing. If you have logged in to your Trimble Identity, your account name is shown. Otherwise your user name is shown.	Lock drawings (page 576)
User-defined attributes	You can show up to 20 user-defined attributes in the **Drawing list**. These user-defined attributes must be added in the **User-defined attributes** dialog box or panel in drawing properties. To include a user-defined attribute in drawing lists, the user-defined attribute must have the option `special_flag` set to `yes` in the `objects.inp` file.	User-defined attributes in drawings (page 895)

Drawing status flags

Tekla Structures uses certain symbols called *flags*, to indicate the status of the drawings. The columns **Issue**, **Ready for issuing**, **Lock**, **Freeze**, **Master** and **Up to date** contain the flags, and the potential additional information is displayed in the **Changes** column. If a drawing does not have any flag symbol, it is up to date.



How to read drawing status information

The table below explains the meaning of the status flags and information about the status of a drawing in the **Drawing list**.

Flag	Column where shown	Information in the Changes column	Description
	Up to date	**Parts modified**	The parts in the drawing have changed, for example, parts have been added or deleted, or part properties have changed.
		Quantity increased or **Quantity decreased**	The actual drawing is up to date, but the number of identical parts has changed.
	Up to date	**All parts deleted**	All the parts related to the drawing have been deleted.

Flag	Column where shown	Information in the Changes column	Description
	Lock		The drawing is locked and you cannot open it for editing.
	Freeze		The drawing is frozen. Changes made to the model objects, which have drawing objects associated with them, are no longer available in the drawing.
	Master		The drawing has been added as a master drawing in the **Master Drawing Catalog**.
	Ready for issuing	**Drawing updated**	The drawing is marked ready for issuing. You can check who marked the drawing from the new **Ready for issuing by** column.
	Up to date	**Linked drawing changed**	You have a linked drawing in this drawing and the linked drawing has been modified.
	Up to date	**Copied view changed**	A copied drawing has been modified.
	Up to date	**Drawing updated**	A frozen drawing has been updated.
	Up to date	**Drawing updated**	Drawing that has been marked

Flag	Column where shown	Information in the Changes column	Description
			ready for issuing has changed.
	Up to date	**Cloned**	The drawing is a cloned drawing. The flag disappears when you save and close the drawing.
	Issue		The drawing has been issued. For example, you might re-issue drawings that have been sent to site.
	Issue	**Issued drawing changed**	The issued drawing has been edited or otherwise changed.

Filter drawing list contents

To filter the **Drawing list** contents, do any of the following:

To	Do this
Display a predefined set of drawings	Select a set of drawings from the **or select drawing set** list. Saved search results are also displayed in this list.
List all drawings	Click **Show all**.
Invert the contents of the current list	Click **Invert**. This feature allows you to list the opposite of the current content. For example, if you had selected to display **Locked drawings**, clicking **Invert** shows all drawings except the locked ones.
List selected drawings only	Select the drawings and click **Selected**.
List up-to-date drawings only	Click **Up to date**.
List drawings that need to be updated	1. Click **Up to date**. 2. Click **Invert**.

To	Do this
Lists only drawings associated with the objects you have selected in the model.	1. Select objects in the model. 2. Click **By parts**. This is an easy way to identify the drawings associated with a specific part, assembly or cast unit. This works also for objects in GA drawings.
Sort the list according to column name	Click the column name.

Search for drawings and save search results

You can search for drawings in the **Drawing list**.

To search for drawings and save the results:

1. In the **Drawing list**, enter the search criteria in **Enter search criteria**.
2. If needed, you can limit the search so that it addresses only information in a certain column by selecting a column from the **Search in** list.
3. If needed, you can also limit the search to the currently visible drawings by selecting **Search within the currently visible drawings**.
4. Click **Search**.
5. Click **Store** to save the search results.
6. Enter a name for the search results in the **Store Search Result** dialog box and click **OK**.

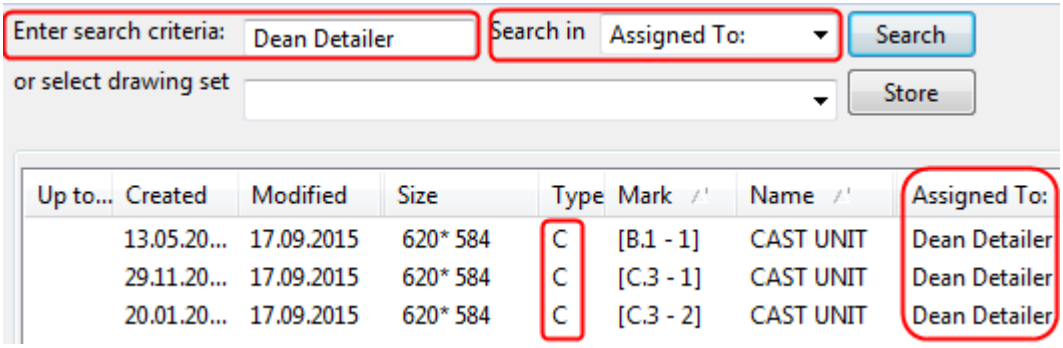
The search results are saved in the `DrawingListSearches` folder, which is created in the model folder.

You can move the saved search results to your environment, company or project folder. After you have saved the search results, they will be visible in the predefined drawing sets list in the **Drawing list** dialog box.

Example

In the following example, you want to find all cast unit drawings that are assigned to Dean Detailer:

1. From the drawing sets, select **Cast unit drawings**. Only cast unit drawings are displayed in the list.
2. In **Enter search criteria**, type `Dean Detailer`.
3. In the **Search in** list, select **Assigned To**.
4. Click **Search**.



Select drawings in Drawing list

The **Drawing list** allows you to select one or several drawings at a time. Selecting several drawings is useful, for example, if you want to lock or freeze several drawings at a time, or print several drawings.

To select drawings in the **Drawing list**:

To	Do this
Select one drawing	Click the drawing in the list.
Select several consecutive drawings	Click the first drawing, then hold down the **Shift** key and click the last drawing.
Select several non-consecutive drawings	Click the first drawing, then hold down the **Ctrl** key and click the other drawings you want to select.
Select all drawings in the list	Press **Ctrl+A**.

Check whether parts have drawings

You can use the **Drawing list** to identify parts in the model that have associated drawings.

Before you start, ensure that you have only the **Select parts** selection switch active on the **Selecting** toolbar. Otherwise selecting objects can take a long time in large models.

1. Open a model view where all parts are clearly visible.
2. In the **Drawing list**, press **Ctrl + A** to select all drawings.
3. To highlight the parts **that have drawings**, click the **Select objects** button.

   This works also in GA drawings.

   To see the found parts easier, right-click the model and select **Show only selected** to show only the parts that the **Select objects** command has found and hide all others.
4. After completing the previous step you can highlight the parts **that do not have drawings** by holding down the **Ctrl** key and selecting the entire

model (hold down the left mouse button and drag all the way from left to right).

Open a drawing when the Drawing list is open

Do any of the following:

- Double-click drawing.
- Right-click the drawing in the list and select **Open**.
- Select a drawing from the list and click the **Open** button at the bottom.

When you open a drawing, a message box is displayed showing the progress and what is happening, and a snapshot of the drawing is displayed. You can click **Cancel** to cancel the opening.

Create and view drawing snapshots

With snapshots you can take a quick look at any drawings without opening the drawing. Use this tool when you just want to check but not edit a drawing, or take a look at several drawings while trying to find the one you are looking for, for example, a certain drawing revision. You can take snapshots of all types of drawings.

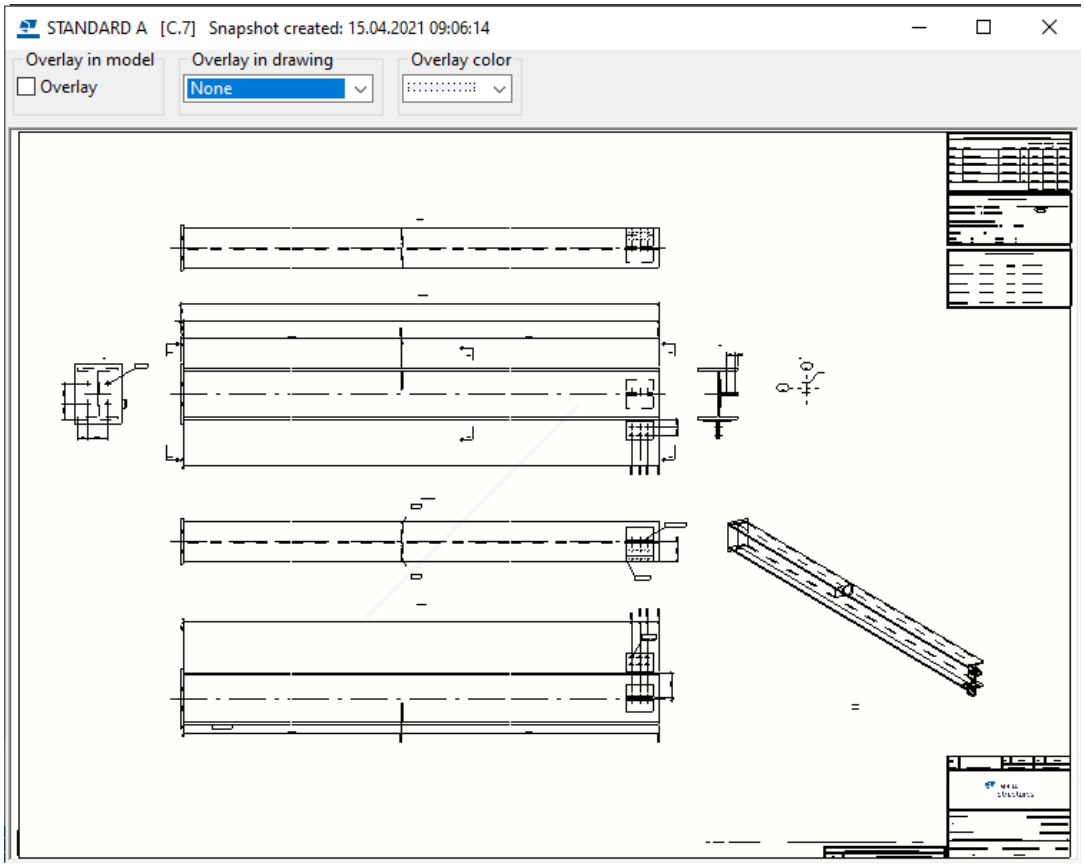
A snapshot is by default created when you open and save a drawing. The snapshot represents the situation when the drawing was last saved, so any newer changes to the model are not reflected in the snapshot.

If you have made changes in the drawing and close it by selecting the **Close** button in the upper-right corner, a message box is displayed where you can select if you want to save the drawing, and also if you want to create a snapshot of the drawing when the drawing is saved.

1. Select a drawing and open it from the **Drawing list**.
2. Save the drawing by going to the **File** menu and clicking **Save drawing**.

   The snapshot is saved in the `..\<model>\drawings\Snapshots` folder.
3. Select the same drawing from the **Drawing list**.
4. Click the **Snapshots** button at the bottom of the **Drawing list** to display the snapshot.

Below is an example of a snapshot.

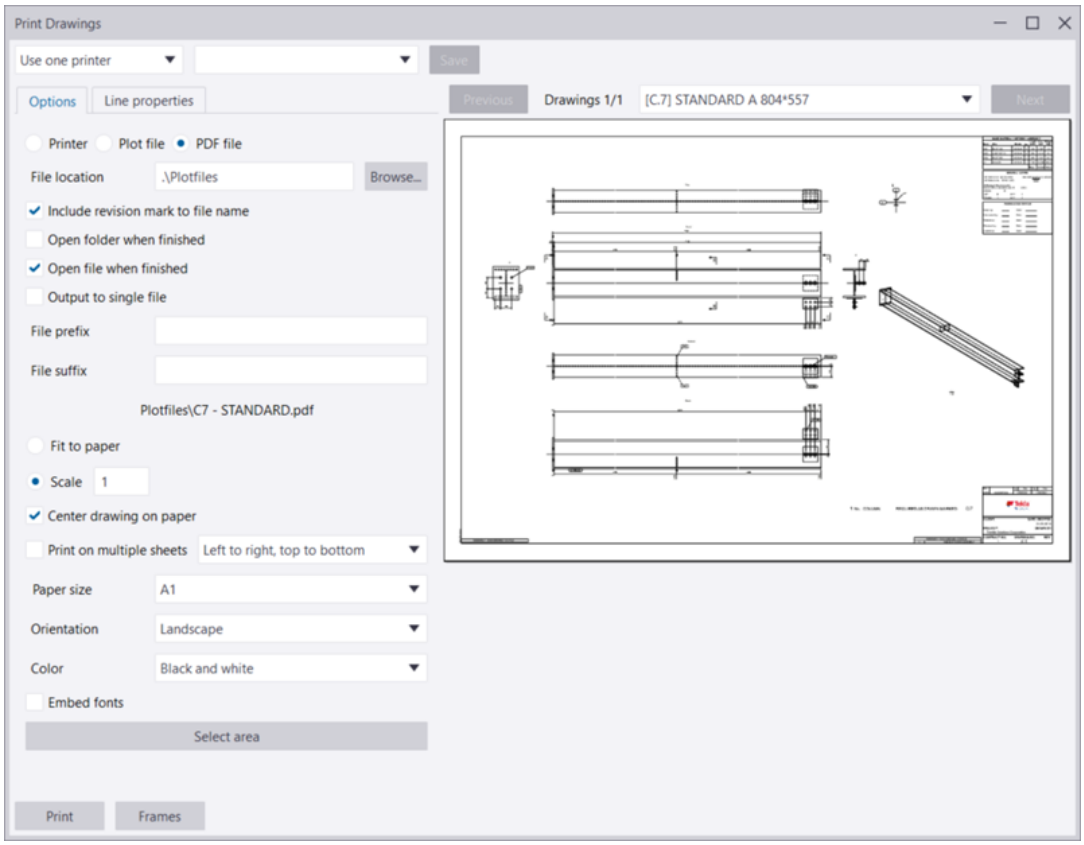


If you have selected a drawing that does not have a snapshot and click **Snapshots**, an instructional view is displayed asking you to open the drawing and save it to create the snapshot.

To create a snapshot of a drawing automatically when the drawing is created, set XS_DRAWING_CREATE_SNAPSHOT_ON_DRAWING_CREATION to `TRUE` in the **Drawing properties** category in the **Advanced options** dialog box.

6 Print drawings

You can print drawings as PDF files, save them as plot files (`.plt`) for printing with printer/plotter, or print them on a selected printer. Tekla Structures provides a real-time preview of drawings in the **Print Drawings** dialog box. You can also change the color-specific line widths.



Limitations in printing

- In an open drawing, you can only preview and print the current drawing. Having a drawing open, and you open another drawing (for example, by double-clicking another drawing in **Document manager**), the preview may

not be updated. Select the drawing again (single-click) in **Document manager** and the preview is then updated to match the selection.

NOTE You can also use the so called "old" printing functionality where you use the **Printer Catalog** dialog box and Tekla Structures printer instances. To do this, see Print drawings using Printer Catalog printer instances (old printing) (page 625).

See also

Print to a PDF file, plot file (.plt) or printer (page 603)

Create and edit drawing layouts (page 656)

Printing configuration files (page 621)

Customize print file names (page 622)

6.1 Print to a PDF file, plot file (.plt) or printer

You can print drawings and selected drawing areas to PDF files or to plot files (.plt) to be sent to a plotter or printer, or print drawings on a printer. You can also modify the output line colors and line widths (pen numbers). You can print to a single printer or to multiple printers, or print one drawing to multiple sheets.

Print a drawing

1. On the **File** menu, click **Printing** --> **Print drawings** .

   This opens the **Print Drawings** dialog box and **Document manager**.

   You can also start printing in the following ways:

   - In **Document manager**, select the drawings, right-click, and select **Print**, or click the **Print** button.

     The order of drawings matches the order they appeared in **Document manager** at the point in time when the **Print Drawings** dialog box was opened.
   - When a drawing is open, select **File** --> **Print drawings**.
   - Press **Shift+P** on the keyboard.
2. In the upper-left corner, ensure **Use one printer** is selected. This is the default option.

3. Load the desired printing settings from the settings list.

   You can also give the settings a new name. In this case, you need to give the new name before you modify any settings, otherwise your changes will be lost. For more information about the printing settings, see Printing settings files and search order (page 620).

4. In **Document manager**, select the drawings that you want to print.

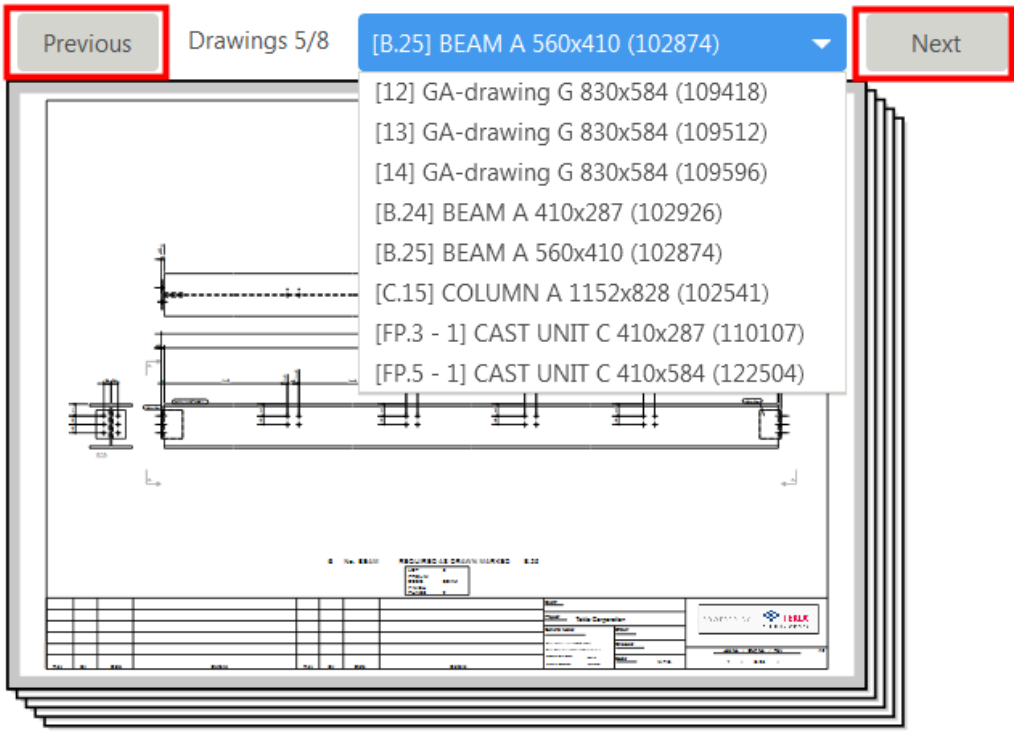
   Any out-of-date drawings are detected and you are asked whether to include them in the output or not.

   You can also print locked drawings that are up to date. If a locked drawing is not up to date, you cannot open or print it, and a failed printout will be reported. You can print any unlocked drawings unless the status of the drawing is **Original part deleted**.

5. To show a preview of the drawing, click **Click here to load a preview**.

   The preview shows the drawings always up to date. We recommend that you check the preview while you change the printing settings.

   If you have selected to print several drawings, the drawings are shown one by one in the preview. Use **Next** and **Previous** to scroll through the set of selected drawings.

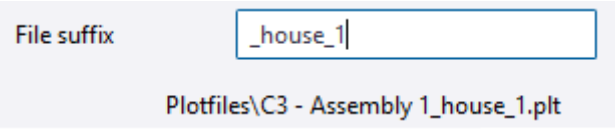


6. Select how to print:
   - **PDF file**: Converts the selected drawings to PDF format.
   - **Printer**: Sends the selected drawings to the selected printer.

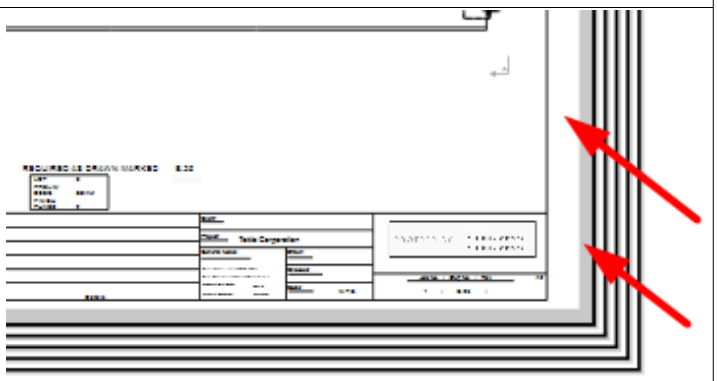
- **Plot file**: Converts the selected drawings to print files in a format suitable for the selected printer and saves them in a specified location.

7. Define the printing settings on the **Options** tab. The available settings depend on the printing option that you have selected:

Option	Description
File location	Enter the location for the PDF file or plot file, or use **Browse...** to browse to the folder. The `\Plotfiles` folder under the model folder is the default value. This option is only available if you have selected **PDF file** or **Plot file**.
Printer or plotter	Select the printer or plotter. Click **Properties...** to adjust printer properties.
Include revision mark to file name	Add the mark of the latest revision of the printed drawing to the print file name. Revision number is used by default. If you always want to use the revision mark instead, set the advanced option XS_SHOW_REVISION_MARK_ON_DRAWING_LIST to `TRUE`. If the revision mark is still not displayed, check the following advanced options and include REVISION_MARK in the value: XS_DRAWING_PLOT_FILE_NAME_A, XS_DRAWING_PLOT_FILE_NAME_W, XS_DRAWING_PLOT_FILE_NAME_G, XS_DRAWING_PLOT_FILE_NAME_M, or XS_DRAWING_PLOT_FILE_NAME_C, depending on the drawing type. This option is only available if you have selected **PDF file** or **Plot file**.
Open folder when finished	Open the PDF file or plot file folder in Windows Explorer after the printout has been created. This option is only available if you have selected **PDF file** or **Plot file**.
Open file when finished	Open the PDF file after it has been created. This option is only available if you have selected **PDF file**.
Output to single file	Print the selected drawings to a single PDF file.

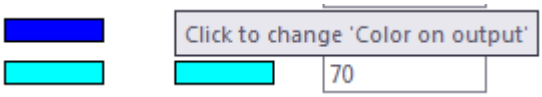
Option	Description
	If you do not select this option, each of the selected drawings will be printed in a PDF file of its own. This option is only available if you have selected **PDF file**.
File name	Give a file name to a PDF file. The file name is needed if you are printing to a single file. The PDF file name can also be controlled by a couple of advanced option switches for customizing print file names (page 622). These switches do not work for single combined multiple drawings PDF file. Sometimes the drawing file name may have an illegal character, which cannot be printed. Because of this, Tekla Structures already replaces large number of illegal characters with an underscore "_". Check the file naming conventions provided by Microsoft "Naming Files, Paths, and Namespaces". This option is only available if you have selected **PDF file** and **Output to single file**.
File extension	Specify a file name extension for the plot file. The default is `plt`. This option is only available if you have selected **Plot file**.
File prefix **File suffix**	Enter a specific prefix and/or suffix in the PDF or plot file name. When you enter a prefix or suffix, the file name preview under the **File prefix** and **File suffix** boxes will reflect the change immediately.  The file name can also be controlled by advanced option switches for customizing print file names (page 622). This option is only available if you have selected **PDF file** or **Plot file**.
Fit to paper	Fit the drawing to a specific paper size. This setting does not work if you have selected **Scale**.

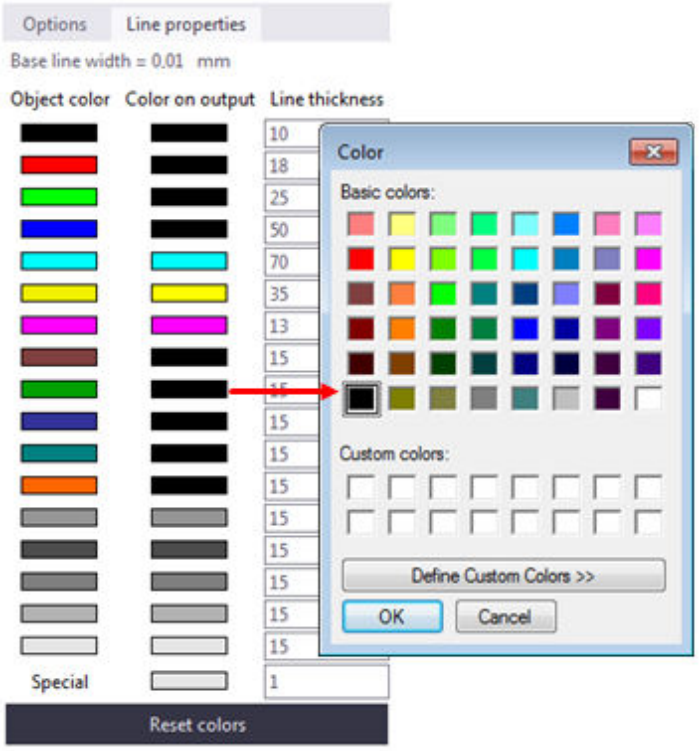
Option	Description
Scale	Force the printout to a specific scale. The **Scale** value will turn red if the drawing cannot fit on the specified sheet. This setting does not work if you have selected **Fit to paper**.
Center drawing on paper	Center the drawing on the sheet (or sheets).
Print on multiple sheets	Print on multiple sheets and specify the direction of printing the sheets. This is particularly useful if your drawing contains several drawings. Select to print from **Left to right, top to bottom** or from **Bottom to top, right to left**. When you use **Print on multiple sheets**, always set a particular **Scale**. The preview shows how the drawing is divided into sheets.
Paper size	Define the paper size or use automatic size. With the **Auto** setting Tekla Structures selects the paper size that has the least wasted area when the scaled print is fitted to the printable area on the sheet. Printers are often unable to print on the full area of a sheet, and leave borders. The *printable area* is determined for the selected printer when the option **Printer** or **Plot file** is selected. For PDF files, the printer is not known, so the output is sized to the full sheet. However, when printing a PDF, the same problem exists and the drawing content is fitted to the printable area of whatever printer is being used. The printable area is shown with white background and the non-printable border is shaded gray in the image below.

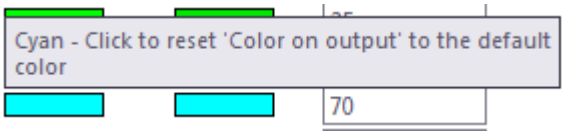
Option	Description
	 Two configuration files affect paper sizes and drawing sizes: `PaperSizesForDrawings.dat` and `DrawingSizes.dat` . For more information, see Printing configuration files (page 621)
Orientation	Define the orientation or use automatic orientation. The **Auto** setting means that the orientation that wastes least space is selected automatically.
Color	Select if the output is to be **Color**, **Black and white** or **Grayscale**.
Number of copies	Define the number of paper copies to be printed. This option is only available if you have selected **Printer**.
Collate	Collate the printout when you are printing multiple copies. This option is only available if you have selected **Printer**.
Embed fonts	Embed the fonts in a PDF file. This ensures that fonts may be reproduced in a system that does not have the same fonts installed, but this also increases the file size. In certain cases, fonts may be embedded automatically. When using non-Latin fonts, we recommended that you use embedding, otherwise the PDF may not be displayed correctly. Note that you can also define a default Unicode font for the advanced option XS_DEFAULT_UNICODE_FONT_DRAWING_PRINTING . If the selected font does not contain all characters in the text, the font is defined by this advanced option. The default value is Arial Unicode MS. This font might not be installed on your machine by

Option	Description
	default, and you may need to install it. You can also define another font that contains the characters you need and that you have installed on your machine. This option is only available if you have selected **PDF file**.
Select area	Select a rectangular area from an open drawing to only show and print that area. This option only works when you have a drawing open. All settings in the **Print Drawings** dialog box also work when **Select area** is selected, and you can change the orientation, line width, and paper size, for example.
Show entire drawing	When you have selected an area with **Select area**, the **Show entire drawing** button appears, and you can use it to show the entire drawing in the preview again.

8. Go to the **Line properties** tab to map colors to line widths (pen numbers) and set the printout colors:

Option	Description
Object color	Shows the basic set of object colors.
Color on output	• Set the printout color by clicking a color box under **Color on output** and selecting a replacing color from the displayed colors. You can also define custom colors. 

Option	Description
	 A different output color is often used when you just need one or two lines in color and the rest in black. If you do not want to print a certain color, you can replace it with white in the output. The output color settings can be used for all printing options (printer, plot file and PDF file). The output color settings are saved in and loaded from the printing settings files. If you have selected **Printer line colors** through **File** --> **Settings** , and change a line color, the change is immediately shown in the drawing. The **Color on output** only applies when the **Color** option is set to **Color** on the **Options** tab in the **Print Drawings** dialog box.
Line thickness	• Enter the line width (pen number) for each color in the boxes. Line width is expressed as a multiple of the advanced option `XS_BASE_LINE_WIDTH` value. The default value for this advanced option is 0.01 mm. For example, line width 25 will give a line weight of 0.25 mm.

Option	Description
	Note that the black and white color mode always shows the printer line width on the screen, whereas the color and grayscale color modes only show the printer line width on the screen if the **Printer line widths** switch is active in **File** --> **Settings** . Also note that the screen is refreshed when printer settings, such as line width values, are modified. You can define **Invisible** color for parts and shapes in drawings in the part or shape properties. The **Invisible** color is not shown in printouts, neither on paper nor on PDF. If you are having problems with showing correct line width, see Line width in drawings (page 617).
Reset colors	You can reset the printout colors: • To reset an individual output color, click the corresponding **Object color** color box. The **Color on output** color box will change to have the same color.  • To reset all output colors to be the same as the object colors, click the **Reset colors** button.

9. Save your printing settings by using the **Save** button in the upper-left corner. For more information about the printing settings and the search order, see Printing settings files and search order (page 620).

   You can also add frames and foldmarks (page 656) to you printouts.

10. Click **Print** to print the drawings in the PDF format or as plot files, or to send them to a printer according to the settings you defined in the dialog box. Each drawing is sent to the printer as a separate print job.

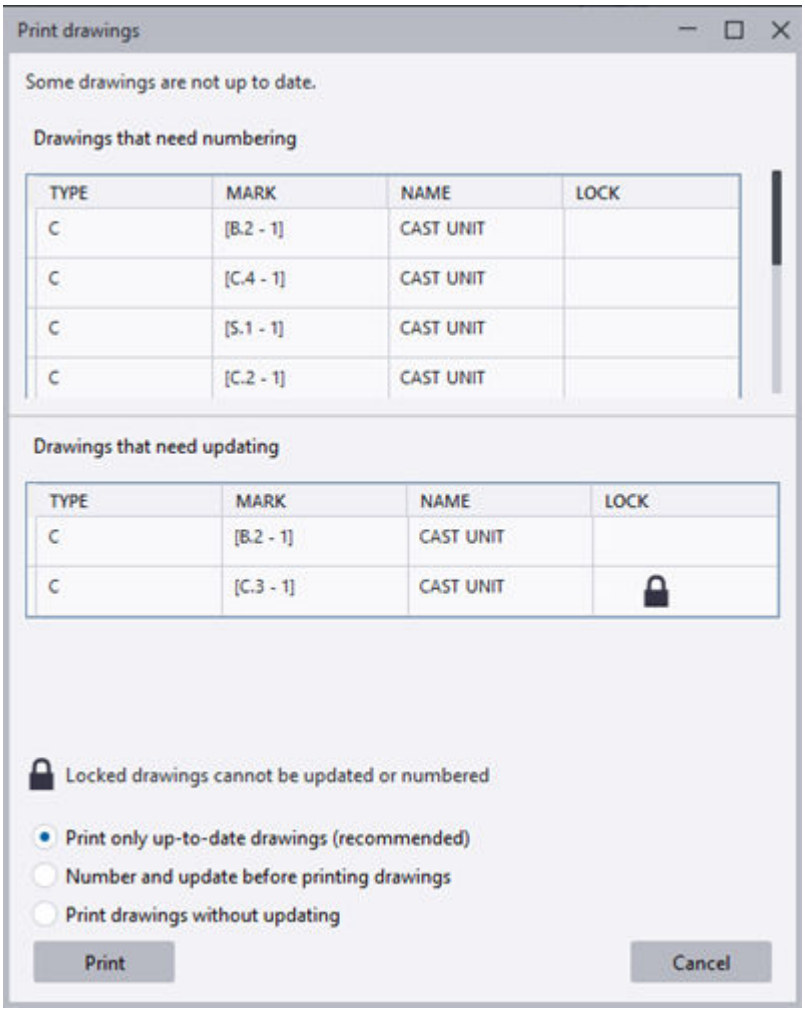
    All printing errors are written to a log file in the model folder: `logs \DPMPrinter_<username>.log .`

11. If you have selected drawings that need numbering or updating, a dialog box is displayed, and you can select how to proceed:

    NOTE You cannot update or number locked drawings.

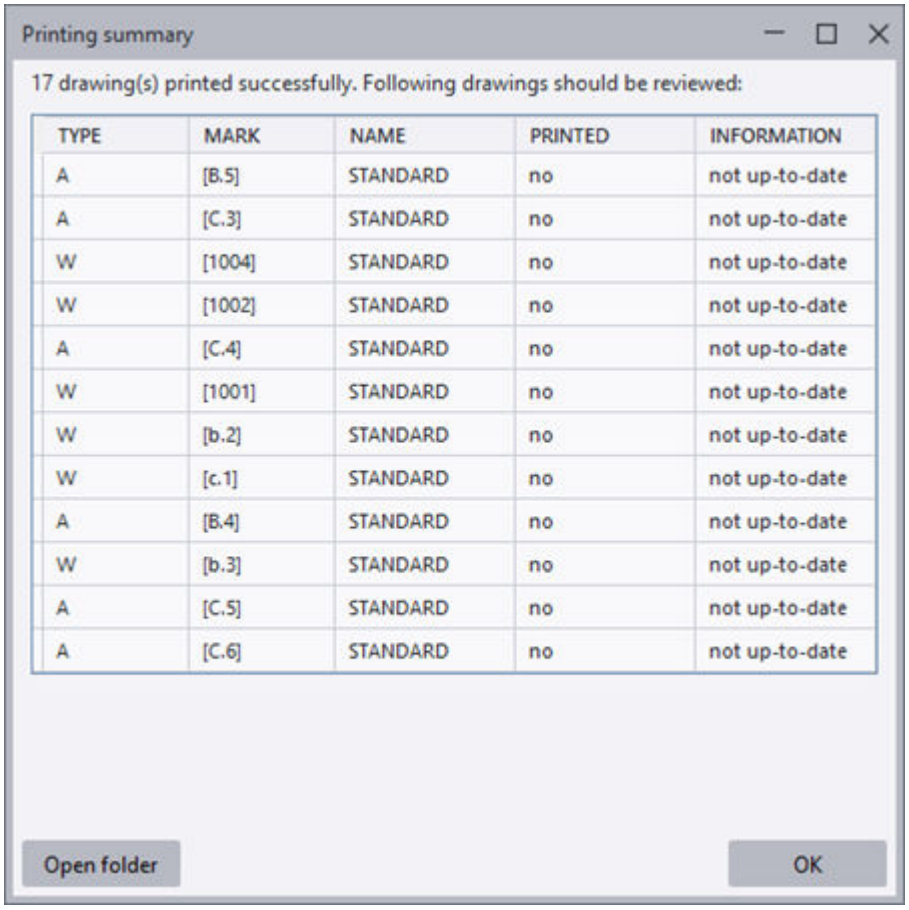
    - **Print only up-to-date drawings (recommended)**
    - **Number and update before printing drawings**

- **Print drawings without updating**
- To cancel the whole process, click **Cancel**.



12. Click **Print**

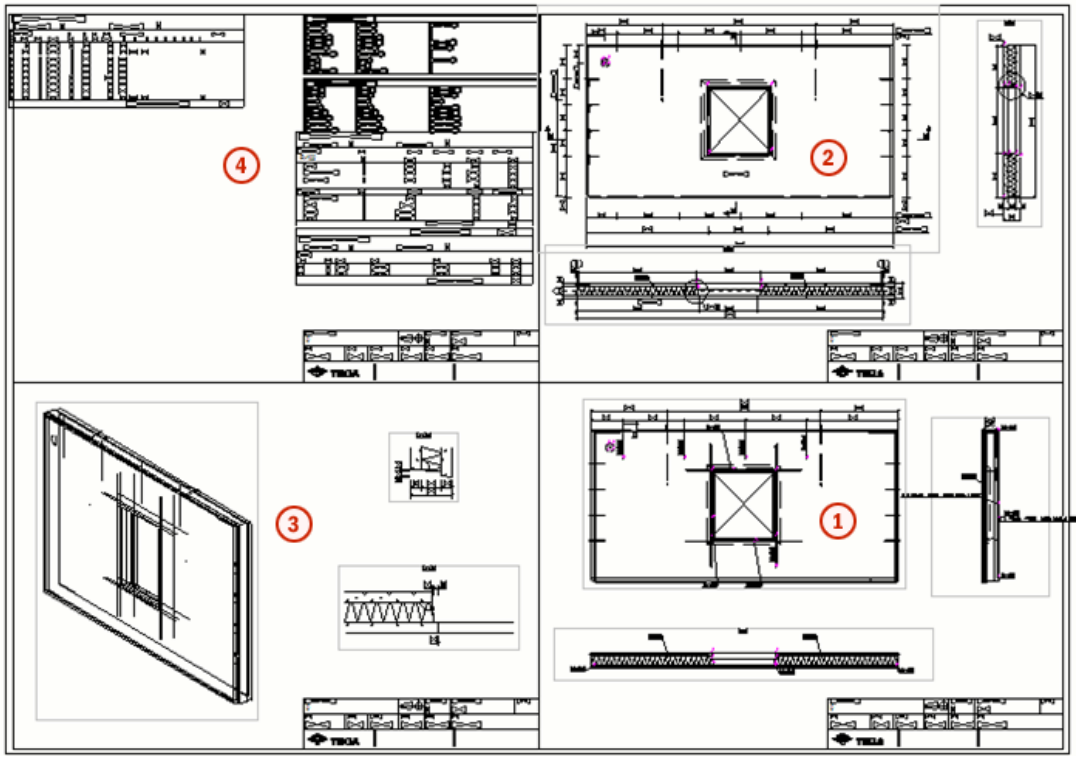
A summary is displayed showing you the printing result and a list of drawings that need attention.



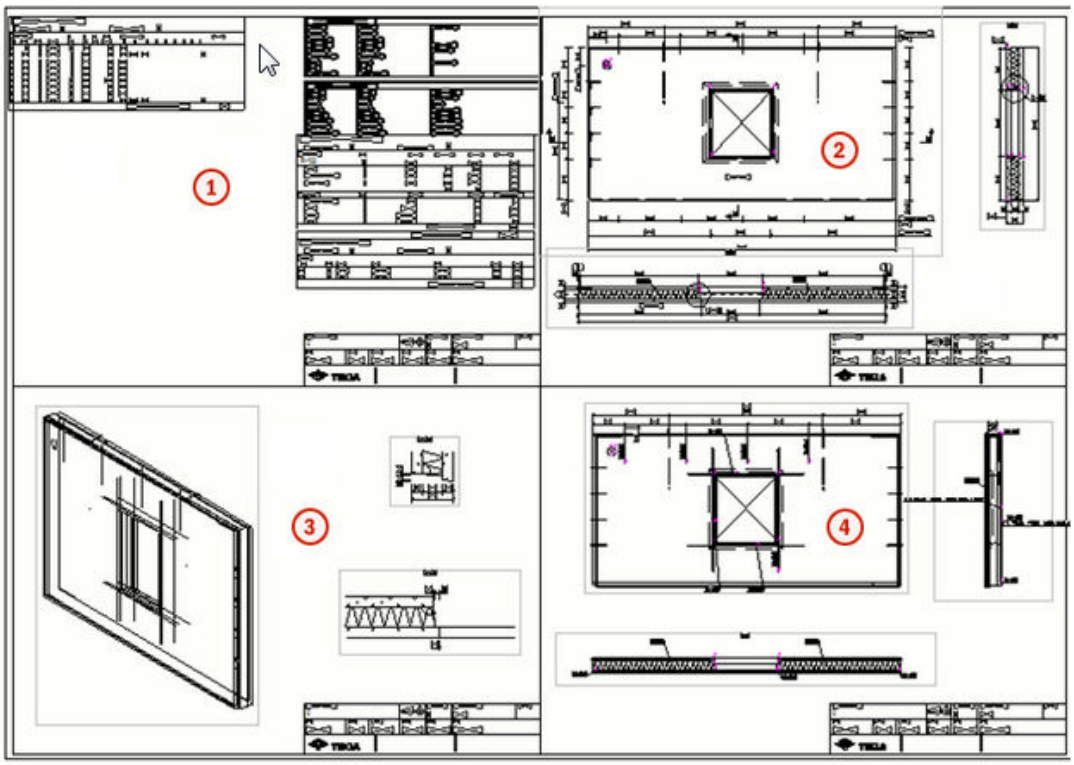
Example: Print to multiple sheets

The **Print on multiple sheets** setting on the **Options** tab prints the drawing on multiple sheets. You need to specify the direction of printing the sheets. This is particularly useful if your drawing contains several drawings.

In the example below, the option **Bottom to top, right to left** has been selected. The numbers indicate the printing order of the sheets.



In the next example, the option **Left to right, top to bottom** has been selected.



Print to multiple printers

You can print to more than one printer in one go based on the paper size of each selected drawing. When printing to multiple printers you typically have different printers for handling different paper sizes. Tekla Structures automatically selects the appropriate printer for each drawing.

To print to multiple sheets, you need to:

- Create and save the printing settings for the necessary printers
- Print by using **Use multiple printers** mode.

Create single printing settings

To be able to print to multiple printers, you first need to create single printing settings for each of the printers you want to print to:

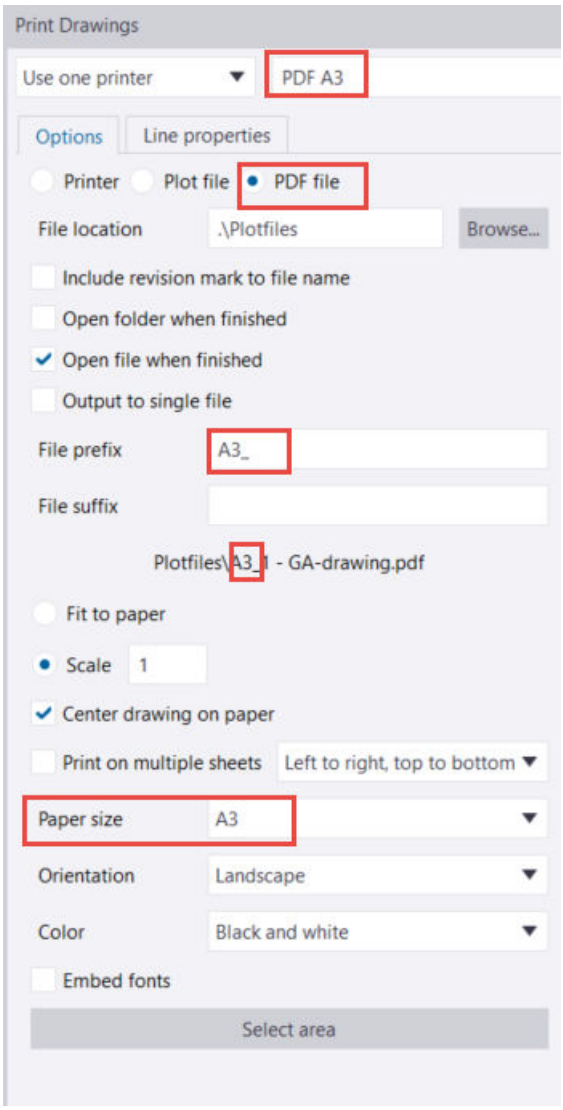
1. On the **File** menu, click **Printing** --> **Print drawings** .
2. Select **Use one printer**.
3. Define the printing settings as desired on the **Options** tab and on the **Line properties** tab. Select the output type and printer, and define the paper size that this particular printer will handle in **Use multiple printers** mode.
4. Save the settings with a desired name by clicking **Save**.
5. Repeat this for each of the desired paper sizes. Do not use the size option **Auto**.

   For example, you could create the following single printer settings files with the output type set to **PDF file**:

   - `PDF A4`: Paper size set to A4, file prefix set to `A4_`
   - `PDF A3`: Paper size set to A3, file prefix set to `A3_`
   - `PDF A2`: Paper size set to A2, file prefix set to `A2_`

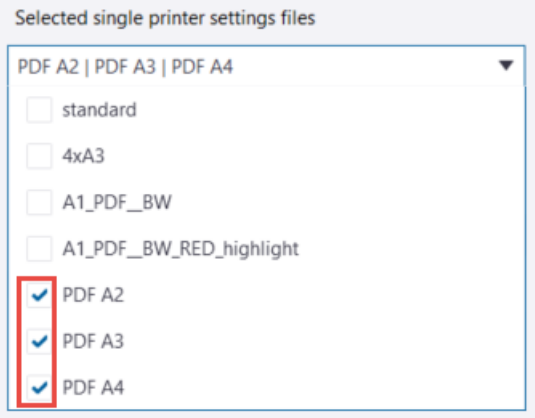
   When printing a set of drawings in the multiple printers mode using the above single printer settings files, all A4 drawings will generate `.pdf` files with prefix `A4_`, all A3 drawings will have prefix `A3_`, and all A2 drawing will generate `.pdf` files with prefix `A2_`.

   If you want to print more than one sheet size to the same printer in **Use multiple printers** mode, create a single printer settings file for each paper size, and specify the same printer in all of these files.

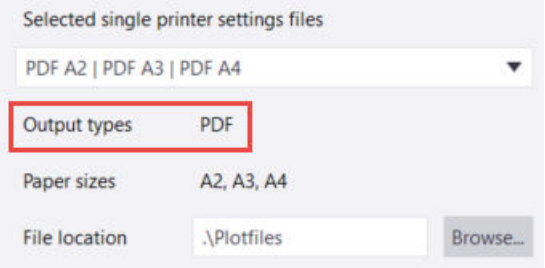


Print to multiple printers

1. On the **File** menu, click **Printing** --> **Print drawings** .
2. In the upper-left corner, select **Use multiple printers** mode.
3. In the **Selected single printer settings files** list, select the settings files to use in printing. You can select all or just some of the single printer settings files.



The output type (printer, plot file, PDF file) is defined by each selected single printer settings file. Typically you would select settings files with the same output type. **Output types** lists the output types that are specified in the selected single printer settings files.



4. Change other necessary settings on the **Options** tab. The available settings depend on the output type you have selected. The settings are described in the "Print to a single printer" section above.
5. Change necessary line width and color settings on the **Line properties** tab.
6. In the displayed **Document manager**, select the drawings that you want to print.

Line width in drawings

You can set the printer line width (pen number) in the **Print drawings** dialog box, but you may have some challenges in showing the lines correctly in the drawing on the screen. You can solve this issue by adjusting the printer line widths, or by using some predefined printer settings in the printer definition file `plotdev.bin`.

Change line width for printed drawings

You can change the line width (pen number) for printing. To do this, open the **Print Drawings** dialog box and go to the **Line properties** tab:

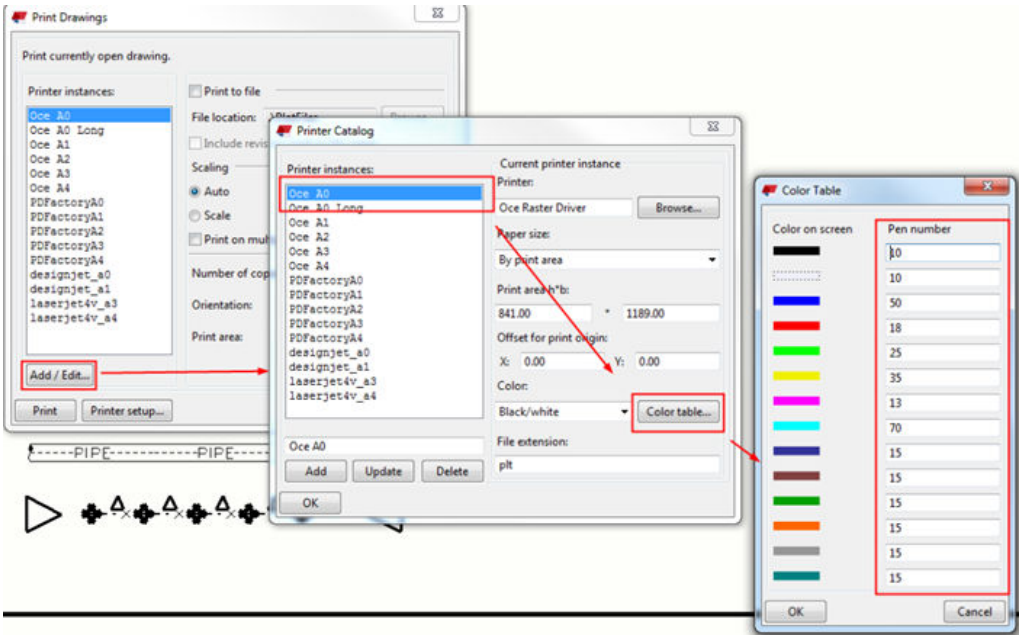
The base line default width is 0.01, and this can be changed using the advanced option XS_BASE_LINE_WIDTH. For example, to get the line width of 0.25 mm, enter the number 25.

Set the line width shown on the screen

The line width that you see in the drawing is not the real line width that you get in the printed drawing. There is a way to affect how the lines are shown in the drawing.

To be able to set the line width (pen number) for the drawings in a Tekla Structures model:

1. Take old printing into use by setting the advanced option XS_USE_OLD_PLOT_DIALOG to `TRUE`.
2. Define the line settings for the first printer in the list.



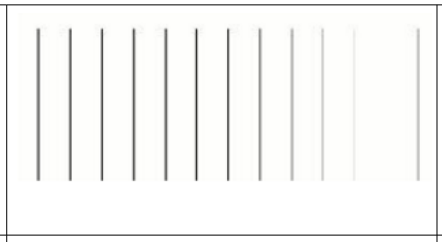
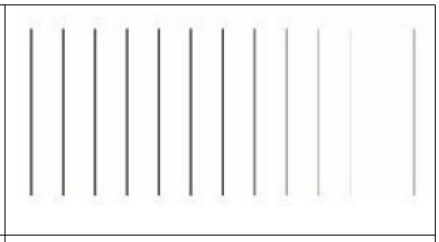
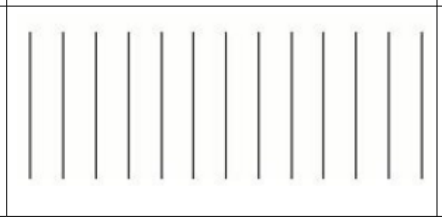
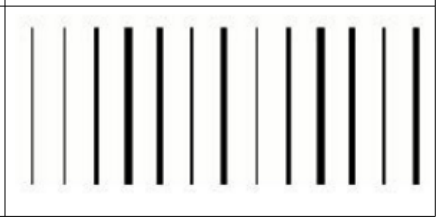
3. Take new printing into use by setting the advanced option `XS_USE_OLD_PLOT_DIALOG` to `FALSE`.
4. Go to the **File** menu and check if the **Printer line widths** switch is selected. This switch takes the settings from the old printer dialog settings. If **Printer line widths** is not selected, the line widths are only shown in black and white mode, if selected, the line widths are shown for all color modes.

   You can toggle between color modes by pressing **B**.

The file `plotdev.bin` in `..\ProgramData\Trimble\Tekla Structures \<version>\environments\default\system` contains some predefined line widths for the first printer instance. These are not the same that you use for printing, unless you use defaults in the **Print Drawings** dialog box.

You can save printer definitions file `plotdev.bin` in the current model folder or in the project and firm folders, and in a folder indicated by the advanced option `XS_DRIVE`. Tekla Structures searches first for `plotdev.bin` in the model, project and firm folders, then in the folder indicated by the advanced option.

	Without line width (pen number) set up in plotdev.bin	**With settings for line width for the first printer in plotdev.bin**
Color		

Grayscale		
Black and white		

See also

Change the pen numbers (line thickness) for colors (old printing) (page 647)

6.2 Printing settings files and search order

Tekla Structures printing settings in the **Print Drawings** dialog box are stored in two files: `<user>_ PdfPrintOptions.xml` and `PdfPrintOptions.xml`. When you open a new model, the `PdfPrintOptions.xml` file is loaded. The changes that you make in printing settings are automatically saved in `<user>_ PdfPrintOptions.xml`, and settings in this file are loaded when you re-open the model. The PDF report options are read from settings files with the name `report.PdfPrintOptions.xml` (in the printing dialog box, this settings file is shown as `report`).

You can create printing settings files for different printing purposes and load them later on. You can also create and share common settings throughout an organization.

You can load existing printing settings or save the current printing settings to either an existing printing settings file or a new file. The first settings file name in the control list will be `standard` and any other settings file names will be listed alphabetically after this. The latest settings used are automatically saved to `<model>\attributes\<user>_PdfPrintOptions.xml` (where `<user>` is the current Windows user when the dialog box is closed. The printing settings that you save using the **Save** button are saved to the `<model>\attributes\` folder with the following names:

- The `standard` file is saved as `PdfPrintOptions.xml` .
- The report file is saved as `report.PdfPrintOptions.xml`.
- All other printing settings names have the filename `<SettingsName>.PdfPrintOptions.xml`. For example, if you use the

name `MyPrintingSettings`, the settings will be saved as `MyPrintingSettings.PdfPrintOptions.xml`.

- If a file already exists it will be overwritten.
- You can move the saved printing settings files to the following locations so that other models and/or other users can access the settings:
  - XS_PROJECT
  - XS_FIRM
  - XS_DRIVER
  - XS_SYSTEM
  - XS_USER_SETTINGS_DIRECTORY
- When you open the dialog box, Tekla Structures searches the above locations in the listed order for any available settings files and adds them to the setting list.
- It will load the settings from the first of the following settings files it finds:
  - `<model>\attributes\<user>_PdfPrintOptions.xml`
  - `<model>\attributes\PdfPrintOptions.xml`
  - `PdfPrintOptions.xml` from the standard search locations listed above.

See also

Print drawings (page 602)

6.3 Printing configuration files

In printing, two configuration files are needed. They affect paper sizes and drawing sizes: `PaperSizesForDrawings.dat` and `DrawingSizes.dat`.

- `PaperSizesForDrawings.dat` defines a list of paper size names allowed to be used, and their dimensions. The `PaperSizesForDrawings.dat` file is by default located in the `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\system` folder.
- `DrawingSizes.dat` provides a list of properties that should be set up to match the drawing sizes configured in the Tekla Structures drawing layout definitions. It is used to link those drawings to the paper size name that each drawing was set up for, and provides information about the drawing sizes and the margins around them. The `DrawingSizes.dat` file is located in the `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\system` folder.

- The exact file location may vary depending on the folder structure of your environment files.

Default values are provided in the files installed with the printing functionality. These values are appropriate in most cases. The optimum values depend on the details of the existing drawing layout definitions. If the printed area of the drawing needs to be moved, or if inappropriate paper sizes are selected, see instructions for adjusting values in these two files.

Make copies of the original configuration files if you want to modify the settings, and place the copies in appropriate folders. You may have several copies of the configuration files on your computer. When needed, these files are searched for and the first one found is used in the following search order:

- model folder
- project folder as defined by the `XS_PROJECT` advanced option
- firm folder as defined by the `XS_FIRM` advanced option
- system folder as defined by the `XS_SYSTEM` advanced option

If no files are found default values will be used.

TIP When making copies of the configuration files, keep them first in a test model folder. Validate the results before using them through the project, firm or environment folders. Also, keep backup copies of the configuration files in a safe location as re-installing later versions of Tekla Structures may overwrite your own settings.

See also

Print drawings (page 602)

6.4 Customize print file names

You can affect the way Tekla Structures automatically names the `.pdf` files and plot files by using certain drawing-type-specific advanced options.

1. On the **File** menu, click **Settings** --> **Advanced options** and go to the **Printing** category.
2. Enter values for any or all of the advanced options , , , , or , depending on the print file drawing type.

   The letter at the end of the advanced option indicates the drawing type. Use single `%` around the values. See the table below for available values. You can combine several values. The values are not case sensitive.
3. Click **OK**.

Example:

The example below results in the assembly drawing `.pdf` name `E_P1_PLATE_Revision=2.pdf`:

```
XS_DRAWING_PLOT_FILE_NAME_A=E_%NAME.%_%TITLE%%REV?
_Revision=%%REV%.pdf
```

Possible values

Value	Example of the result in the print file name	Description
`%NAME%` `%DRAWING_NAME%`	P_1	Part, assembly, or cast unit position, using the file name format `prefix_number`.
`%NAME.-%` `%DRAWING_NAME.-%`	P-1	Part, assembly, or cast unit position, using the file name format `prefix-number`.
`%NAME.%` `%DRAWING_NAME.%`	P1	Part, assembly, or cast unit position, using the file name format `prefixnumber`.

`%REV%`
`%REVISION%`
`%DRAWING_REVISION%`
	2	Drawing revision number.

`%REV_MARK%`
`%REVISION_MARK%`
`%DRAWING_REVISION_MARK%`
	B	Drawing revision mark.

`%TITLE%` `%DRAWING_TITLE%`	PLATE	Drawing name from the drawing properties dialog box.
`%UDA:<drawing user-defined attribute>%`	Painted	Value of a user-defined drawing attribute. The user-defined drawing attributes are defined in `objects.inp`. The actual values for the user-defined attributes are entered in the drawing-specific user-defined attributes dialog box.
`%REV? - <text>%`	2 - Rev	Adds conditional prefixes. In this example, if `REV` exists, Tekla Structures adds the text between ? and % to the filename.

Value	Example of the result in the print file name	Description
%TPL:<template attribute>%	Base plate	You can use template attributes that can be found in Template Editor. The actual values for these attributes are entered in the drawing properties dialog box. Examples: • %TPL:TITLE1% • %TPL:TITLE2% • %TPL:TITLE3% • %TPL:DR_DEFAULT_HOLE_SIZE% • %TPL:DATE% • %TPL:TIME% • %TPL:DR_DEFAULT_WELD_SIZE%

NOTE The print file name switches %DRAWING_NAME% and %NAME% that should produce an underscore in the print file name (P_1) do not work if XS_ASSEMBLY_POSITION_NUMBER_FORMAT_STRING does not use a separator between the values (for example, %ASSEMBLY_PREFIX% %ASSEMBLY_POS%), or if XS_USE_ASSEMBLY_NUMBER_FOR is set.

To make the switches work, do the following:

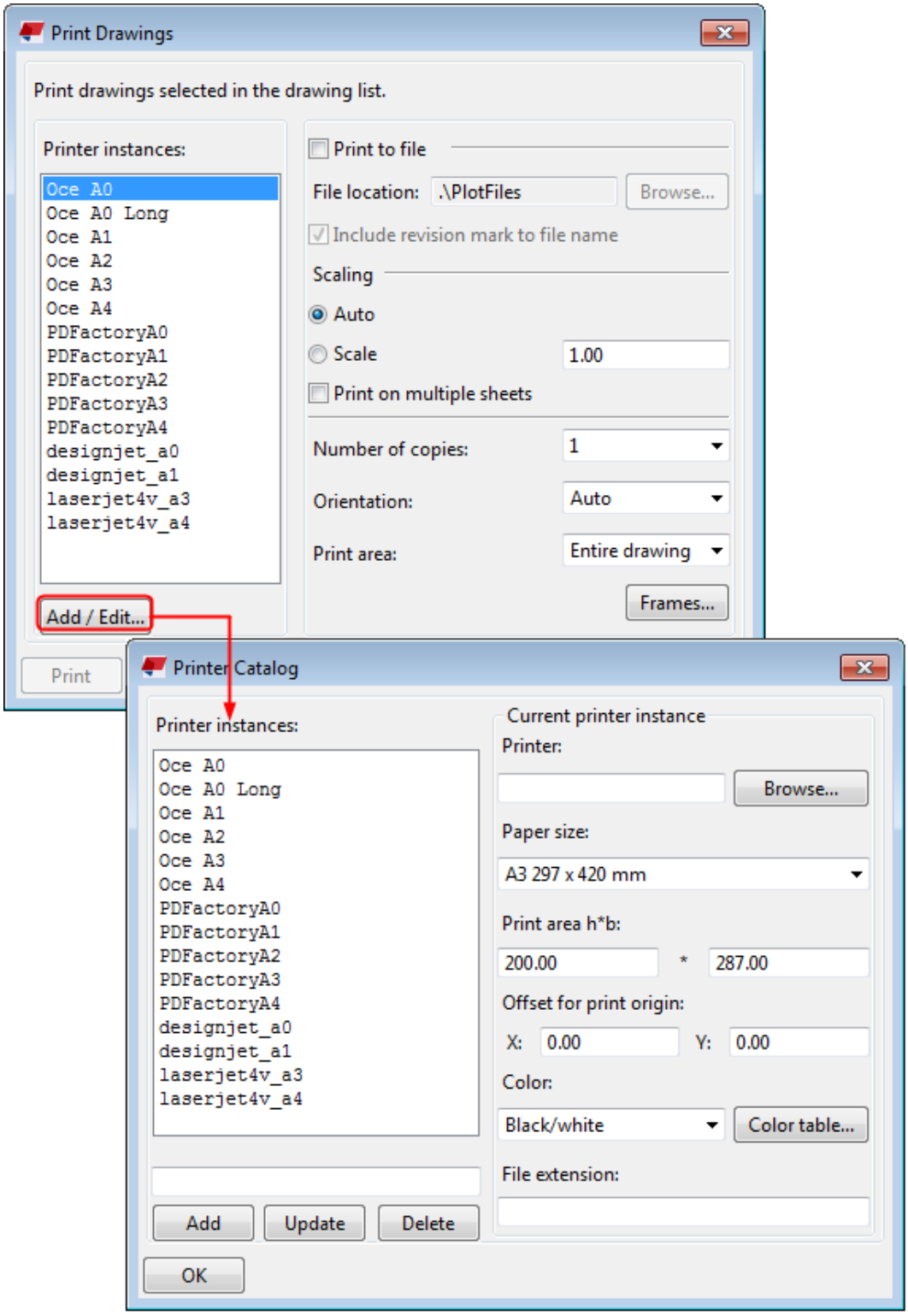
- If you want to use XS_ASSEMBLY_POSITION_NUMBER_FORMAT_STRING, use a dot (.) slash (/) or hyphen (-) between the values, for example, %ASSEMBLY_PREFIX%.%ASSEMBLY_POS%, or similar.
- Leave XS_USE_ASSEMBLY_NUMBER_FOR empty.

NOTE When you define a switch in the **Advanced options** dialog box, you only use single percent signs around the switch %xxx% . When you define a switch for an advanced option in an .ini file, you need to use double percent signs around the switch %%xxx%%. For example, type %%BOLT_NUMBER%%*D% %HOLE.DIAMETER%% for the advanced option XS_BOLT_MARK_STRING_FOR_SIZE when you define it in an .ini file.

7 Print drawings using Printer Catalog printer instances (old printing)

You can also use Tekla Structures own **Printer Catalog** printer instances for printing drawings. To do this, you need to set the advanced option `XS_USE_OLD_PLOT_DIALOG` to `TRUE`.

Setting XS_USE_OLD_PLOT_DIALOG to `TRUE` takes the **Printer Catalog** dialog box in use, and printer instances need to be set.



Setting up printers in the Microsoft Windows environment is not covered in this documentation. It is assumed that printers are set up and tested in your environment. For further information about on setting up printers, contact your system administrator.

If you prefer to print using the "new" printing functionality, see Print to a PDF file, plot file (.plt) or printer (page 603) .

Click the links below to find out more:

- Print single drawings (old printing) (page 627)
- Print multiple drawings with different sizes at one go (old printing) (page 630)
- Create .pdf files (old printing) (page 631)
- Print to file (old printing) (page 632)
- Print to multiple sheets (old printing) (page 633)
- Printing settings in Print Drawings dialog box (old printing) (page 635)
- Customize print file names (old printing) (page 637)
- Frames and fold marks in drawings (old printing) (page 639)
- Set up printer instances in Printer Catalog (old printing) (page 642)
- Printing tips (old printing) (page 648)

7.1 Print single drawings (old printing)

The instructions below apply when you are printing using **Printer Catalog** printer instances, which means that you have set the advanced option `XS_USE_OLD_PLOT_DIALOG` to `TRUE` in **File menu** --> **Settings** --> **Advanced options** --> **Printing** .

Before you print the drawing, check that the printer instance settings are correct and the layout contains the correct drawing size settings.

1. Open a drawing.
2. On the **File** menu, click **Print drawing**.
3. In the **Print Drawings** dialog box, select the printer instance you want to use.
4. If needed, modify the printing settings and add frames and fold marks in the printout.
5. Click **Print**.

TIP Shortcut for opening the **Print Drawings** dialog box: **Shift+P**.

Examples

To see some examples of printing single drawings, click the links below:

Example: Print on A4 in landscape (old printing) (page 628)

Example: Print on A3 in portrait (old printing) (page 629)

Example: Print A3 drawing on A4 paper (old printing) (page 630)

See also

Frames and fold marks in drawings (old printing) (page 639)

Print drawings using Printer Catalog printer instances (old printing) (page 625)

Printing settings in Print Drawings dialog box (old printing) (page 635)

Set up printer instances in Printer Catalog (old printing) (page 642)

Example: Print on A4 in landscape (old printing)

The instructions below apply when you are printing using **Printer Catalog** printer instances, which means that you have set the advanced option `XS_USE_OLD_PLOT_DIALOG` to `TRUE` in **File menu** --> **Settings** --> **Advanced options** --> **Printing** .

This example describes how you can print on A4 paper in landscape style in black and white.

NOTE In this example, it is assumed that the print area h*b is defined h being along the longer side of the paper and b being along the shorter side of the paper. When using some other printer driver you might have to change h*b values if you find out that the printer driver uses h along the short side of the paper.

---

1. Open a drawing and double-click the drawing background.
2. In drawing properties dialog box, click **layout**.
3. Define the following settings:
   - Set **Size definition mode** to **Specified size**.
   - Set **Drawing size** to `287` * `200`.
   - You can also use **Autosize**. Then you need to make sure that you have defined appropriate **Fixed sizes** or **Calculated sizes**.
4. Click **Modify** and **OK**.
5. On the **File** menu, click **Print drawing**.
6. In the **Print Drawings** dialog box, click the printer instance you want to use.
7. Click **Add / Edit...** and check that the printer settings are correct:

- **Paper size**: **A4 210 x 297 mm**
- **Print area h*b**: 287 x 200
- **Color**: **Black/white**

8. Click **Update**.
9. Click **OK**.
10. Set **Scaling** to **Scale** and enter 1.

    In this case, also **Scaling** setting **Auto** would produce a similar printout because drawing size and h*b are the same.
11. Set **Orientation** to **Landscape** (or **Auto**).
12. Set **Print area** to **Entire drawing**.
13. Click **Print**.

Example: Print on A3 in portrait (old printing)

The instructions below apply when you are printing using **Printer Catalog** printer instances, which means that you have set the advanced option XS_USE_OLD_PLOT_DIALOG to TRUE in **File menu** --> **Settings** --> **Advanced options** --> **Printing** .

This example describes how you can print on A3 paper in portrait style in black and white.

1. Open a drawing and double-click the background.
2. In drawing properties dialog box, click **layout**.
3. Define the following settings:
   - Set **Size definition mode** to **Specified size** .
   - Set **Drawing size** to 287 * 410.
   - You can also use **Autosize**. Then you need to make sure that you have defined appropriate **Fixed sizes** or **Calculated sizes**.
4. Click **Modify** and **OK**.
5. On the **File** menu, click **Print drawing**.
6. In the**Print Drawings** dialog box, click the printer instance you want to use.
7. Click **Add / Edit...** and check that the printer settings are correct:
   - **Paper size**: **A3 297 x 420 mm**.
   - **Print area h*b**: 410 x 287
   - **Color**: **Black/white**

8. Click **Update**.
9. Click **OK**.
10. Set **Scaling** to **Scale** and enter `1`.
11. Set **Orientation** to **Portrait** (or **Auto**).
12. Set **Print area** to **Entire drawing**.
13. Click **Print**.

Example: Print A3 drawing on A4 paper (old printing)

The instructions below apply when you are printing using **Printer Catalog** printer instances, which means that you have set the advanced option `XS_USE_OLD_PLOT_DIALOG` to `TRUE` in **File menu** --> **Settings** --> **Advanced options** --> **Printing** .

This example describes how you can print an A3 drawing on A4 paper. This is useful, for example, when you need draft drawings that do not have to show the correct scale.

1. Open the A3 drawing.
2. On the **File** menu, click **Print drawing**.
3. In the **Print Drawings** dialog box, click the printer instance that you want to use.
4. Click **Add / Edit...**, click the printer instance that you are going to use and ensure that:
   - It prints to A4 paper.
   - The **Print area h*b** value takes into account the printer hard margins. In this case, the **Print area h*b** could be `287*200`.
5. If you changed any settings, click **Update** and **OK**.

   If you did not change any settings in the **Printer Catalog**, click **OK** to return to the **Print Drawings** dialog box.
6. Set **Scaling** to **Auto**.

   When you use **Auto**, and the drawing is bigger than the paper, the drawing is scaled down to fit the paper.
7. Click **Print**.

7.2 Print multiple drawings with different sizes at one go (old printing)

You can print multiple drawings from the **Document manager**, and print drawings of different sizes at the same time.

The instructions below apply when you are printing using **Printer Catalog** printer instances, which means that you have set the advanced option `XS_USE_OLD_PLOT_DIALOG` to `TRUE` in **File menu** --> **Settings** --> **Advanced options** --> **Printing** .

1. In the **Document manager**, select the drawings you want to print.
2. Right-click the selected drawings and select **Print**.
3. In the **Print Drawings** dialog box, indicate the printer instances you want to use.

   To select several printer instances, hold down **Ctrl** and select the printers.

   When you select drawings of several sizes and several printer instances, Tekla Structures sends each drawing to the printer instance that is using the smallest paper size on which the drawing will fit. For example, if you have two printer instances selected, one A4 and one A3, Tekla Structures will send A4 drawings to the A4 printer instance and A3 to the A3 printer instance.
4. Set **Scale** to **1**.

   This makes it possible for Tekla Structures to select and use the printer instance that is using the correct paper size.
5. If necessary, modify other printing settings and add frames and fold marks in the printout.
6. Click **Print**.

See also

Set up printer instances in Printer Catalog (old printing) (page 642)

Frames and fold marks in drawings (old printing) (page 639)

Print drawings using Printer Catalog printer instances (old printing) (page 625)

Print single drawings (old printing) (page 627)

7.3 Create .pdf files (old printing)

You can use any standard pdf printer to create .pdf files, such as pdfFactory, Win2PDF, or Adobe Acrobat. You can print several drawings at a time, and use several printer instances.

The instructions below apply when you are printing using **Printer Catalog** printer instances, which means that you have set the advanced option `XS_USE_OLD_PLOT_DIALOG` to `TRUE` in **File menu** --> **Settings** --> **Advanced options** --> **Printing** .

Before you start creating .pdf files using Adobe Acrobat, ensure that you have Adobe Acrobat and Adobe Distiller installed and configured with the Adobe Postscript Printer Driver set up to print to file. Consult your Adobe documentation for more information.

Also ensure that you have an Adobe postscript printer instance in the Tekla Structures printer catalog.

1. In **Document manager**, select the drawings of which you want to create .pdf files.
2. Right-click the selected drawings and select **Print Drawings...**.
3. In **Print Drawings** dialog box, select the pdf printer instances you want to use.

   If you select several printer instances, Tekla Structures sends each drawing to the printer instance that is using the smallest paper size on which the drawing will fit. For example, if you have two printer instances selected, one A4 and one A3, Tekla Structures will send A4 drawings to the A4 printer instance and A3 to the A3 printer instance.
4. If needed, modify the printing settings and add frames and fold marks in the PDF.
5. Click **Print**.

   Tekla Structures creates the .pdf files and saves them in the folder that you specified when you defined the printer instance. The file will have the name shown in **Document manager** with the extension `ps`.

Limitations

Do not use the **Print to file** option when you create .pdf files.

See also

Printing settings in Print Drawings dialog box (old printing) (page 635)

Add an Adobe postscript printer instance (old printing) (page 644)

Frames and fold marks in drawings (old printing) (page 639)

Print drawings using Printer Catalog printer instances (old printing) (page 625)

7.4 Print to file (old printing)

You can print to a file using a print-to-file printer instance. The file is by default printed to the `\Plotfiles` folder under model folder, but you can change the folder.

The instructions below apply when you are printing using **Printer Catalog** printer instances, which means that you have set the advanced option `XS_USE_OLD_PLOT_DIALOG` to `TRUE` in **File menu** --> **Settings** --> **Advanced options** --> **Printing** .

Before you start, make sure you have a printer instance set up to print to file.

1. In **Document manager**, select all drawings that you want to print.
2. Right-click the selected drawings and select **Print Drawings...**
3. Click a printer instance that is set up to print to file.
4. Select the **Print to file** check box.

   Specify the folder. You can use the folder specified during printer instance setup or click **Browse...** to locate the destination folder in the **Browse For Folder** dialog box.

   If you do not enter a folder, Tekla Structures creates the files in the current model folder or in the folder defined by the advanced option `XS_DRAWING_PLOT_FILE_DIRECTORY`.
5. If needed, modify other printing settings and add frames and fold marks in the printout.
6. Click **Print**.

Tekla Structures prints the selected drawings to files in the specified folder using the drawing names.

See also

Printing settings in Print Drawings dialog box (old printing) (page 635)

Add a print-to-file instance (old printing) (page 643)

Frames and fold marks in drawings (old printing) (page 639)

Print drawings using Printer Catalog printer instances (old printing) (page 625)

7.5 Print to multiple sheets (old printing)

If your drawing is very large, you can print it on multiple sheets. With correct scaling settings, Tekla Structures automatically calculates the required amount of sheets.

The instructions below apply when you are printing using **Printer Catalog** printer instances, which means that you have set the advanced option `XS_USE_OLD_PLOT_DIALOG` to `TRUE` in **File menu** --> **Settings** --> **Advanced options** --> **Printing** .

Before printing to multiple sheets, make sure that the layout of the drawing supports printing on several smaller sheets. Remember that Tekla Structures automatically adds a 5 mm margin to the printouts.

Also ensure that you have set up the printer instance correctly for printing to multiple sheets.

1. Open a drawing.
2. On the **File** menu, click **Print drawing** .
3. In the **Print Drawings** dialog box, select the printer instance you want to use.
4. Select the **Print on multiple sheets** option.
5. In **Scaling**, set **Scale** = `1`. This preserves the scale. The number of sheets is rounded up.

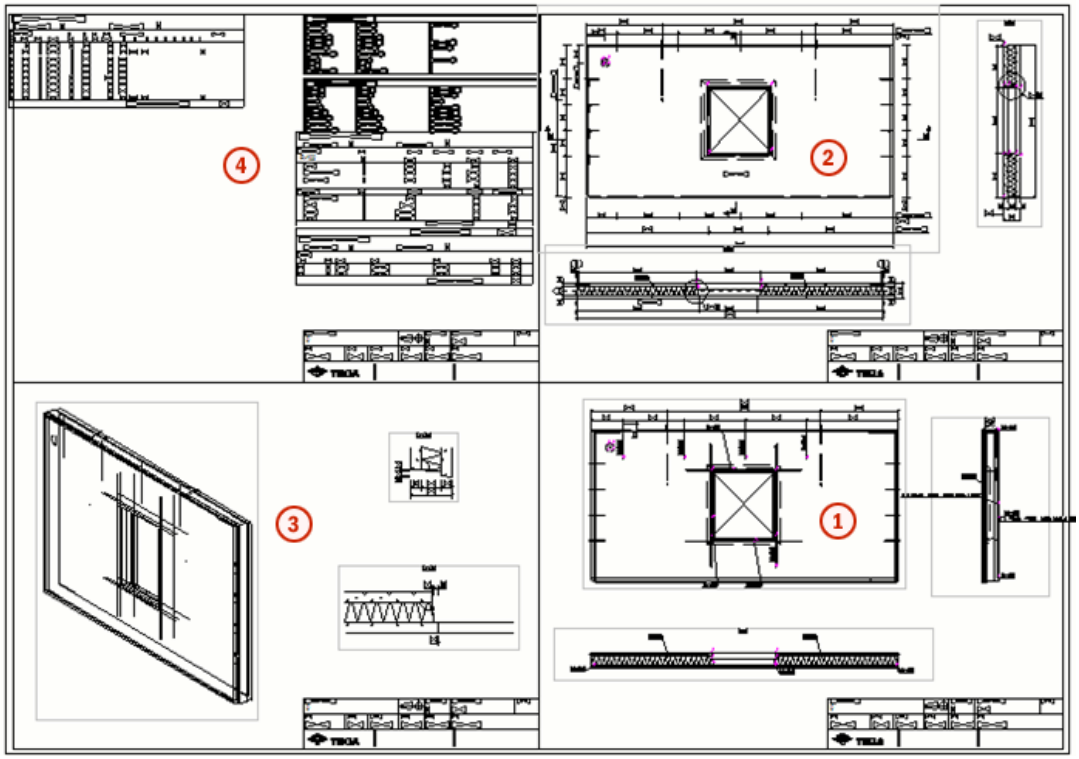
   Do not use **Auto** when you print to multiple sheets.

   Tekla Structures calculates the required amount of sheets needed to print the drawing.
6. If needed, modify other printing settings and add frames and fold marks in the printout.

   If you set **Orientation** to **Auto**, Tekla Structures selects an orientation that results in the smallest number of printed sheets.

Tekla Structures prints the drawing to multiple sheets so that it prints the lower right corner the first, and the upper left corner the last (see the numbered sheets in the example below).

If you want to have drawing frames and/or the title blocks for each smaller size sheet, you need to use proper table layout as shown in the example below.



TIP Use the advanced option `XS_PRINT_MULTISHEET_BORDER` to set borders that are left out from the smaller sheets.

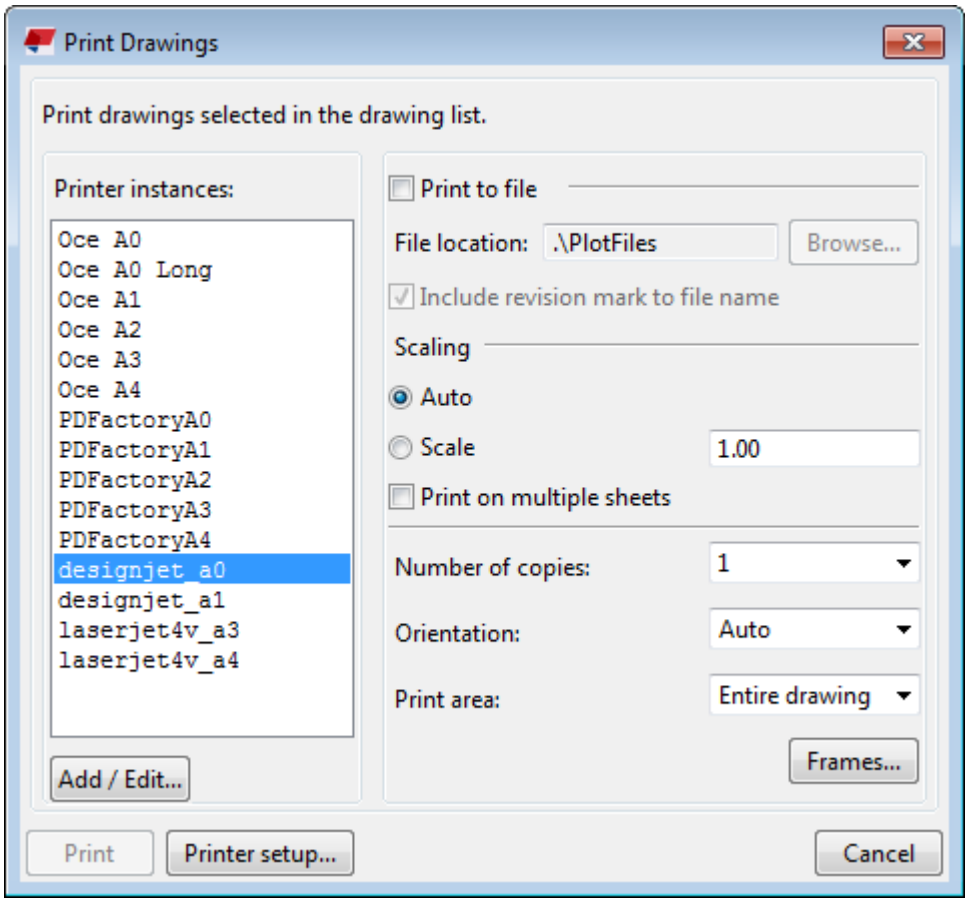
See also

Printing settings in Print Drawings dialog box (old printing) (page 635)

Print drawings using Printer Catalog printer instances (old printing) (page 625)

7.6 Printing settings in Print Drawings dialog box (old printing)

The **Print Drawings** dialog box contains options for setting up printing. This dialog box is only displayed if you have set `XS_USE_OLD_PLOT_DIALOG` to `TRUE` in **File menu** --> **Settings** --> **Advanced options** --> **Printing**.



Setting	Description
Print to file	Prints the drawing to a file.
Include revision mark to file name	Adds the latest revision of the printed drawing to the filename. Revision number is used by default. However, if you set the advanced option to `TRUE`, the revision mark is used.
Scaling	**Auto** fits the drawing to the **Print area h*b** size, i.e. fits it to the paper. This is useful, for example, when you are printing draft drawings on A4. With this option you can include everything in the printout, but the scale of the drawing may be affected as drawing size is fitted to the **Print area h*b**. With **Auto**, if the drawing size is smaller than the paper, the drawing is scaled up to fill the defined **Print area h*b**, keeping the aspect ratio.

Setting	Description
	Entering an exact scale in the **Scale** box scales the drawing manually to the defined scale. Examples on **Scale**: 1.0 = 100%, 0.9 = 90%
Print on multiple sheets	Prints the drawing on multiple small sheets.
Number of copies	Defines the number of copies.
Orientation	**Auto** orientates the drawing to suit the paper. **Landscape** prints the drawing horizontally - as it is displayed on the screen. **Portrait** prints the drawing vertically.
Print area	**Entire drawing** prints the entire drawing. **Visible area** prints the area visible in the current drawing window.
Add / Edit...	Add (page 642) or delete printer instances, or modify their settings.
Frames...	Opens a dialog box where you can select to print frames and fold marks (page 639).
Printer setup...	Opens the Windows print setup dialog box where you can change the printing settings for the current Tekla Structures session only. The changes that you make are not saved for the printer permanently.

7.7 Customize print file names (old printing)

By default, Tekla Structures uses the drawing names as print file names. You can customize these file names using an advanced option indicating the drawing type, and entering switches defining the format of the print file name as a value.

To customize the filenames:

1. On the **File menu**, click **Settings** --> **Advanced options** and go to the **Printing** category.
2. Enter switches for the advanced options , , , , and .

   You can also combine several switches. The switches are not case sensitive.
3. Click **OK**.

Example

The example below results in the filename `E_P1_PLATE_Revision=2.dxf`:

```
XS_DRAWING_PLOT_FILE_NAME_A=E_%NAME.%_%TITLE%%REV?_Revision=%
%REV%.dxf
```

Switches for customizing print file names

Use the following switches when you want to customize the print file name format. If you are defining these in an `.ini` file, use double %%. Use single % in the **Advanced options** dialog box.

Switch	Example of the result	Description
`%NAME%` `%DRAWING_NAME%`	P_1	Part, assembly, or cast unit position, using the filename format prefix_number.
`%NAME.-%` `%DRAWING_NAME.-%`	P-1	Part, assembly, or cast unit position, using the filename format prefix-number.
`%NAME.%` `%DRAWING_NAME.%`	P1	Part, assembly, or cast unit position, using the filename format prefixnumber.

`%REV%`
`%REVISION%`
`%DRAWING_REVISION%`
	2	Drawing revision number, if **Include revision mark to file name** is checked in the **Print Drawings** dialog box.

`%REV_MARK%`
`%REVISION_MARK%`
`%DRAWING_REVISION_MARK%`
	B	Drawing revision mark, if **Include revision mark to file name** is checked in the **Print Drawings** dialog box.

`%TITLE%` `%DRAWING_TITLE%`	PLATE	Drawing name from the drawing properties dialog box.
`%UDA:<drawing user-defined attribute>%`	Painted	Value of a drawing user-defined attribute. The drawing user-defined attributes are defined in `objects.inp`. The actual values for the user-defined attributes are entered in the drawing-specific user-defined attributes dialog box.
`%REV? - <text>%`	2 - Rev	Adds conditional prefixes. In this example, if `REV` exists, Tekla Structures adds the text between ? and % to the filename.

Switch	Example of the result	Description
%TPL:<template attribute>%	Base plate	You can use here template attributes that can be found in Template Editor. The actual values for these attributes are entered in the drawing properties dialog box. Examples: • %TPL:TITLE1% • %TPL:TITLE2% • %TPL:TITLE3% • %TPL:DR_DEFAULT_HOLE_SIZE% • %TPL:DATE% • %TPL:TIME% • %TPL:DR_DEFAULT_WELD_SIZE%

NOTE The print output file name switches %DRAWING_NAME% and %NAME% that should produce an underscore in the print file name (P_1) do not work if XS_ASSEMBLY_POSITION_NUMBER_FORMAT_STRING does not use a separator between the values (for example, %ASSEMBLY_PREFIX% %ASSEMBLY_POS%), or if XS_USE_ASSEMBLY_NUMBER_FOR is set.

To make the switches work, do the following:

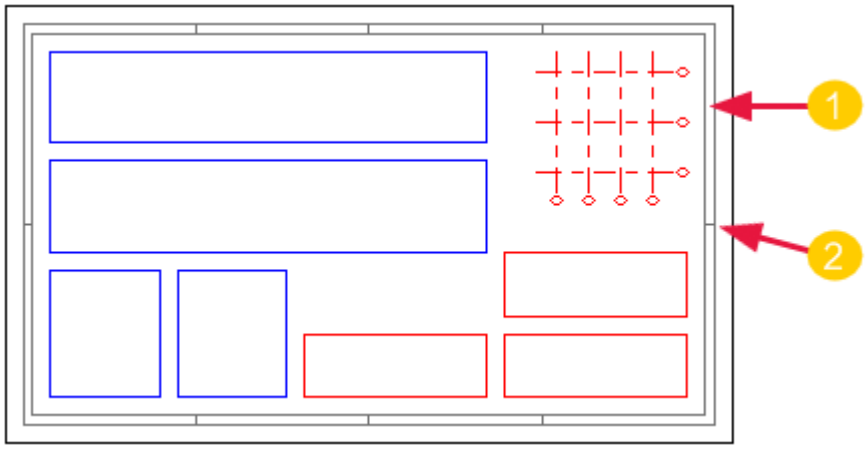
- If you want to use XS_ASSEMBLY_POSITION_NUMBER_FORMAT_STRING, use a dot (.) slash (/) or hyphen (-) between the values, for example, %ASSEMBLY_PREFIX%.%ASSEMBLY_POS%, or similar.
- Leave XS_USE_ASSEMBLY_NUMBER_FOR empty.

7.8 Frames and fold marks in drawings (old printing)

You can have a frame around drawing contents, or two drawing frames inside each other. You can add default Tekla Structures frames around the drawing contents, or use DWG/DXF files in the table layouts as drawing frames.

You can add frames and fold marks only when the advanced option XS_USE_OLD_PLOT_DIALOG is set to TRUE (old printing).

You can create fold marks as guides for folding printed drawings. They are small lines between and perpendicular to the drawing frames.



1. Frame
2. Fold mark

See also

Add frames and fold marks in printouts (old printing) (page 640)

Create and edit drawing layouts (page 656)

Add frames and fold marks in printouts (old printing)

You can add frames around printed drawings, and fold marks to indicate folds. You can select a color for the frames and fold marks

The instructions below apply when you are printing using **Printer Catalog** printer instances, which means that you have set the advanced option `XS_USE_OLD_PLOT_DIALOG` to `TRUE` in **File menu** --> **Settings** --> **Advanced options** --> **Printing** . If this advanced option is set to `FALSE`, you are using the newer printing functionality, where you cannot set the frames and fold marks through the **Print Drawings** dialog box.

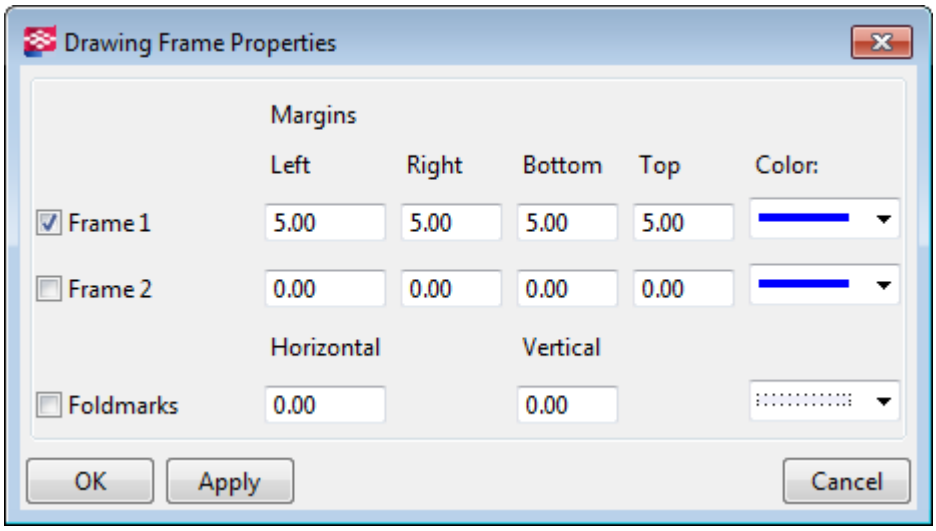
Drawing frames to be printed are controlled in the `standard.fms` file under the system folder. There is no saving option in the **Drawing Frame Properties** dialog box, which is why the default values are located in a standard file. You can save the standard file in the model folder, and then copy it to the project or firm folders, if needed.

If you want to save a set of standard files in the model folder, see Standard files.

1. On the **File** menu, click **Printing** --> **Print drawings**.
2. In the **Print Drawings** dialog box, click **Frames...**.
3. In the **Drawing Frame Properties** dialog box, select the check boxes of the frames you want to print.

4. In **Margins**, enter in millimeters the distance between each frame and the left, right, bottom, and top edge of the paper.
5. Select a color for each frame.
6. To print the foldmarks, select the **Foldmarks** check box.
7. Enter in millimeters the horizontal and vertical distances of the first foldmarks from the lower right corner of the outer frame, and between the other fold mark.
8. Select a color for the fold marks.
9. Click **OK**.

Below is an example of the properties dialog box contents and the standard file.




```
dia_drframe.drframe1_en 1
dia_drframe.drframe2_en 0
dia_drframe.fold_en 0
dia_drframe.x1 5.000000
dia_drframe.y1 5.000000
dia_drframe.x2 5.000000
dia_drframe.y2 5.000000
dia_drframe.pen 4
dia_drframe.x1_2 0.000000
dia_drframe.y1_2 0.000000
dia_drframe.x2_2 0.000000
dia_drframe.y2_2 0.000000
dia_drframe.pen_2 4
dia_drframe.fold_width 0.000000
dia_drframe.fold_height 0.000000
dia_drframe.fold_pen 0
```

See also

Table sets (page 669)

7.9 Set up printer instances in Printer Catalog (old printing)

You need to set up printer instances in **Printer Catalog** for different purposes: for printing to .pdf, to print file, or to different printers, and to print in different sizes.

You can only set up Tekla Structures printer instances in **Printer Catalog** if you have set the advanced option `XS_USE_OLD_PLOT_DIALOG` to `TRUE` in **File menu** --> **Settings** --> **Advanced options** --> **Printing**. If this advanced option is set to `FALSE`, you are using the newer printing functionality, and the **Printer Catalog** is not available.

Tekla Structures uses Microsoft Windows printer drivers to write the printed data directly to a printer, print file or .pdf.

There are two steps in setting up printer instances in Tekla Structures:

- First, you need to add a printer instance (page 642) in the **Printer Catalog**. By default, several printer instances are already defined.
- Next, you need connect the printer instances to printer drivers and adjust the printer instance settings such as paper size and print area (page 645). You can also connect one printer driver to several printer instances, for example, to print in different sizes on the same printer.

See also

Add a print-to-file instance (old printing) (page 643)

Add an Adobe postscript printer instance (old printing) (page 644)

Line thickness (pen number) in Color Table (old printing) (page 646)

Add a printer instance (old printing)

You need to add printer instances in **Printer Catalog** to be able to print. This applies to printing when the advanced option `XS_USE_OLD_PLOT_DIALOG` is set to `TRUE`.

The instructions below apply when you are printing using **Printer Catalog** printer instances, which means that you have set the advanced option `XS_USE_OLD_PLOT_DIALOG` to `TRUE` in **File menu** --> **Settings** --> **Advanced options** --> **Printing** .

1. On the **File** menu, click **Printing** --> **Choose printer**.

2. In **Printer Catalog**, click an existing printer instance that has similar settings to the one you want to add.
3. Enter a new name for the new printer instance in the box under the **Printer instances** list.
4. Click **Add**.
5. Click **Browse...** to access the **Select Printer** dialog box, which shows a list of Microsoft Windows printer drivers currently configured in your system.
6. Click a printer driver and then **OK**.
7. Select the paper size (page 645).
8. Enter the print area (page 645) in **Print area h*b** (height and width).
9. If necessary, use **Offset for print origin** to move the drawing print origin.
10. Select **Black/white**, **Gray Scale** or **Color**.

    If you select **Color**, Tekla Structures prints lines with the colors defined in the drawing properties.
11. Click **Color table...** to map pen size to the on-screen line color. The background color is not printed.
12. Click **Update**.
13. Click **OK**.
14. Confirm that you want to save changes to the model folder.

See also

Print drawings using Printer Catalog printer instances (old printing) (page 625)

Define printing paper size and print area h*b (old printing) (page 645)

Line thickness (pen number) in Color Table (old printing) (page 646)

Add a print-to-file instance (old printing)

To print to a file, you need to add a print-to-file printer instance in **Printer Catalog**. This applies to printing when the advanced option `XS_USE_OLD_PLOT_DIALOG` is set to `TRUE`.

1. On the **File** menu, click **Printing** --> **Choose printer**.
2. In **Printer Catalog**, click **Add**.
3. Enter a printer instance name for the printer driver, followed immediately (no spaces) by `@path\folder\`. The folder must already exist. For example, `11X17@d:\small\`

You can also leave the folder out. If you do this, Tekla Structures prints the file in the current model folder or in the folder specified for the advanced option XS_DRAWING_PLOT_FILE_DIRECTORY.

WARNING XS_DRAWING_PLOT_FILE_DIRECTORY overrides the folder defined in the **Printer Catalog**.

4. Click **Add**.
5. Click **Browse...** to access the **Select Printer** dialog box and click a printer driver that is configured to print to file and then click **OK**.
6. In **Paper size**, select **By print area**.
7. Enter the print area (page 645) in **Print area h*b** (height and width).
8. Enter a file name extension, for example, plt for a print file.
9. Select **Color**, **Gray Scale** or **Black/white** as the color.
10. Click **Color table...** to change the pen sizes of different colors, if needed.
11. Click **Update**.
12. Click **OK**.
13. Confirm that you want to save changes to the model folder.

TIP One way to send different drawing sizes to different folders is to set up a Microsoft Windows printer to print to file for each paper size you will use. Enter a different destination folder for each paper size in the **Printer instances** list in Tekla Structures.

See also

Print drawings using Printer Catalog printer instances (old printing) (page 625)

Define printing paper size and print area h*b (old printing) (page 645)

Line thickness (pen number) in Color Table (old printing) (page 646)

Print to file (old printing) (page 632)

Add an Adobe postscript printer instance (old printing)

To print to a .pdf file, you need to add a Adobe postscript printer instance in **Printer Catalog**. This applies to printing when the advanced option XS_USE_OLD_PLOT_DIALOG is set to TRUE.

The instructions below apply when you are printing using **Printer Catalog** printer instances, which means that you have set the advanced option

XS_USE_OLD_PLOT_DIALOG to TRUE in **File menu** --> **Settings** --> **Advanced options** --> **Printing** .

1. On the **File** menu, click **Printing** --> **Choose printer**.
2. In **Printer Catalog**, click **Add**.
3. Enter a new printer instance name followed immediately (no spaces) by @ and the folder where Adobe Distiller should look for files. For example, A4_PDF@c:\plots\pdf\in\ .
4. Click **Add**.
5. Click **Browse...** to access the **Select Printer** dialog box. Click the Adobe postscript printer driver and then **OK**.
6. Set **Paper size** to By print area.
7. Enter the print area (page 645) in **Print area h*b** (height and width).
8. Enter the file name extension ps.
9. Select **Black/white**, **Gray Scale** or **Color** as the color.
10. Click **Color table...** to change the pen sizes of different colors, if needed.
11. Click **Update**.
12. Click **OK**.
13. Confirm that you want to save the changes to the model folder.

See also

Print drawings using Printer Catalog printer instances (old printing) (page 625)

Define printing paper size and print area h*b (old printing) (page 645)

Line thickness (pen number) in Color Table (old printing) (page 646)

Create .pdf files (old printing) (page 631)

Define printing paper size and print area h*b (old printing)

You need to define paper size and print area h*b for each printer instance you add in **Printer Catalog**. This applies to printing when the advanced option XS_USE_OLD_PLOT_DIALOG is set to TRUE.

The instructions below apply when you are printing using **Printer Catalog** printer instances, which means that you have set the advanced option XS_USE_OLD_PLOT_DIALOG to TRUE in **File menu** --> **Settings** --> **Advanced options** --> **Printing** .

1. On the **File** menu, click **Printing** --> **Choose printer**.
2. In the **Printer Catalog**, select the printer instance.
3. Use the **Paper size** setting to select the paper size used in printing:

- Named paper size: Tekla Structures lists a named paper sizes for most sizes A3 and smaller. Selecting one of the listed sizes is recommended for A3 printers and smaller.
- **By print area**: The printer selects the paper size based on the print area. This option is recommended for A2 printers and larger. If you use a named paper size with a larger printer than A3, drawings larger than the print area are cropped to the size of the print area.
- **None**: Sends no size information to the printer. This option provides legacy support for Xsteel 5.0 and is not recommended for use in other circumstances.

4. Enter the **Print area h*b**:
   - Tekla Structures uses the print area values to position the printout on the paper. Ensure that the values for **h** and **b** relate to the paper size required.

     Usually the value is the paper size less the hard clip margins of the printer. For example, if the paper size is 297*420, the print area could be 407 * 284. See your printer documentation for the hard clip margins of your printer.
   - Usually for roll feed printers, **h** defines the roll width direction and **b** defines the roll feed direction. Usually for tray feed printers, **h** defines the tray feed direction, and **b** defines the tray width direction. Enter the values, and test how the printing works. If you can see, that the direction is wrong, switch the **h** and **b** values.
   - When you are printing from Tekla Structures, it uses the values set for the printer in the Tekla Structures printer instance settings in the **Printer Catalog**, and overrides for these settings the Windows printer settings.
5. Click **OK** and confirm the change.

TIP To print to different paper sizes, you can define several printer instances, each using a different paper size, but all connected to the same physical printer. For more information about setting up print devices in Microsoft Windows, see your operating system documentation.

See also

Add a printer instance (old printing) (page 642)

Printing tips (old printing) (page 648)

Line thickness (pen number) in Color Table (old printing)

You can only set Tekla Structures pen numbers in the **Color Table** of **Printer Catalog** if you have set the advanced option `XS_USE_OLD_PLOT_DIALOG` to

TRUE in **File menu** --> **Settings** --> **Advanced options** --> **Printing**. If this advanced option is set to FALSE, then you are using the newer printing functionality, and the **Printer Catalog** is not available.

Pen numbers in the **Color Table** dialog box refer to the line weights used in the printed drawing. By default, pen number 0 corresponds to a line weight of 0.01 mm. The final line weight in a printed drawing is the default pen thickness multiplied by the pen number. For example, pen number 25 will give a line weight of 0.25 mm.

- The line thicknesses displayed on the screen are taken from the first printer instance on the **Printer instances** list in the **Printer Catalog**. When you print a drawing, the line thickness is taken from the printer instance you use for printing.
- In color drawings, the lines are shown with different thicknesses if the **Printer line widths** check box is selected in **File menu** --> **Settings** .
- In black and white drawings, Tekla Structures shows the black lines on the screen using pen number thickness defined for the color in the **Color Table**.
- You can change the default line thickness using the advanced option .

See also

Change the pen numbers (line thickness) for colors (old printing) (page 647)

Change the pen numbers (line thickness) for colors (old printing)

You can change the pen numbers for colors in **Printer Catalog** to show and print lines with different thicknesses. This applies to printing when the advanced option XS_USE_OLD_PLOT_DIALOG is set to TRUE.

The instructions below apply when you are printing using **Printer Catalog** printer instances, which means that you have set the advanced option XS_USE_OLD_PLOT_DIALOG to TRUE in **File menu** --> **Settings** --> **Advanced options** --> **Printing** .

1. Open a drawing.
2. On the **File** menu, click **Printing** --> **Printer catalog**, and select a printer instance.
3. Click **Color table**
4. Enter or change a pen number.

   For example, to get line weight of 0.25 mm, enter 25.

   You can change the default line thickness 0.01 using the advanced option XS_BASE_LINE_WIDTH.
5. Click **OK**.

6. In a color drawing, on the **File** menu, click **Settings** and select **Printer line widths**, otherwise you cannot see the changes on the screen.

See also

Colors in drawings (page 61)

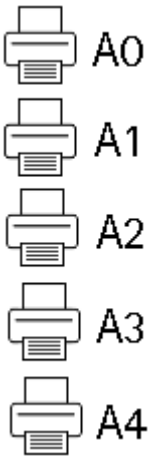
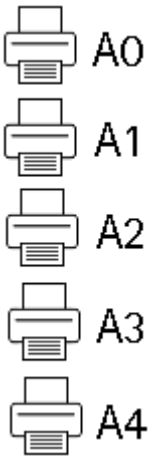
Line thickness (pen number) in Color Table (old printing) (page 646)

7.10 Printing tips (old printing)

Here are some tips that help you print drawings the way you want.

- If you are printing a drawing to a smaller paper, the line weights are scaled accordingly. This means that no lines are drawn too thickly and the drawings are more readable.
- You can force a drawing to use a different size paper: Open the drawing and go to **Drawing Properties** --> **Layout** . In **Size definition mode**, select **Specified size**. Insert the required sheet size into **Drawing size**.
- If a drawing does not fit to the paper or is printed to a wrong location, use the **Offset for print origin** option in the **Printer Catalog** to move the drawing print origin for the selected printer instance.

  The advanced options `XS_PLOT_ORIGIN_MOVE_X` and `XS_PLOT_ORIGIN_MOVE_Y` move the print origin in the x or y direction for all printer instances. If you set values for the advanced options, they are used instead of the **Offset for print origin** value.
- You can disable the print date information to prevent the loss of information when working with multi-user models. This is useful in cases where a user modifies drawings while another user prints the same drawings. Use the `XS_DISABLE_DRAWING_PLOT_DATE` to disable the print date.
- You can add several Windows printer drivers for the same physical printer, and define an exact paper size for each printer driver in Windows printer driver printing preferences or printing defaults. Then in Tekla Structures, you can set printer instances so that you select the printer driver that matches the desired paper size. See the table at the end.
- If you only have one Windows printer driver for one physical printer, define the largest paper size you are going to use for that printer driver in Windows printer driver printing preferences or printing defaults, for example, A0. Then in Tekla Structures, you can use that one printer driver, and set several printer instances for it, each using the necessary paper size. See the table below.

Physical printer	Windows printer drivers	Tekla Structures printer instances
One printer. 	Separate Windows printer drivers for each needed size. A0 A1 A2 A3 A4	Separate Tekla Structures printer instances are defined for each needed size, each printer instance is using the printer driver with a matching paper size.  A0 A1 A2 A3 A4
One printer. 	One Windows printer driver. The printer driver is set to the biggest needed paper size. A0	Separate Tekla Structures printer instances are defined for each needed size, the same printer driver is used for each printer instance.  A0 A1 A2 A3 A4

See also

Print drawings using Printer Catalog printer instances (old printing) (page 625)

Set up printer instances in Printer Catalog (old printing) (page 642)

8 Define automatic drawing settings

Drawing settings tell Tekla Structures what the drawing should look like and what it should include. Automatic drawing settings are defined before drawings are created.

Automatic drawing settings

Automatic drawing settings are settings that are defined through:

- Drawing, view and object properties in various drawing types. The properties are stored in property files. Properties can be defined separately for each drawing that you create, but it is advisable to save the most commonly used settings in property files for future use, for example, in **Master Drawing Catalog**. When creating a new drawing, always start by loading automatic drawing properties that you think contain the best settings for the drawing you are creating, and then adjust the settings as required before creating the drawing. Properties can be adjusted also after you have created the drawing.

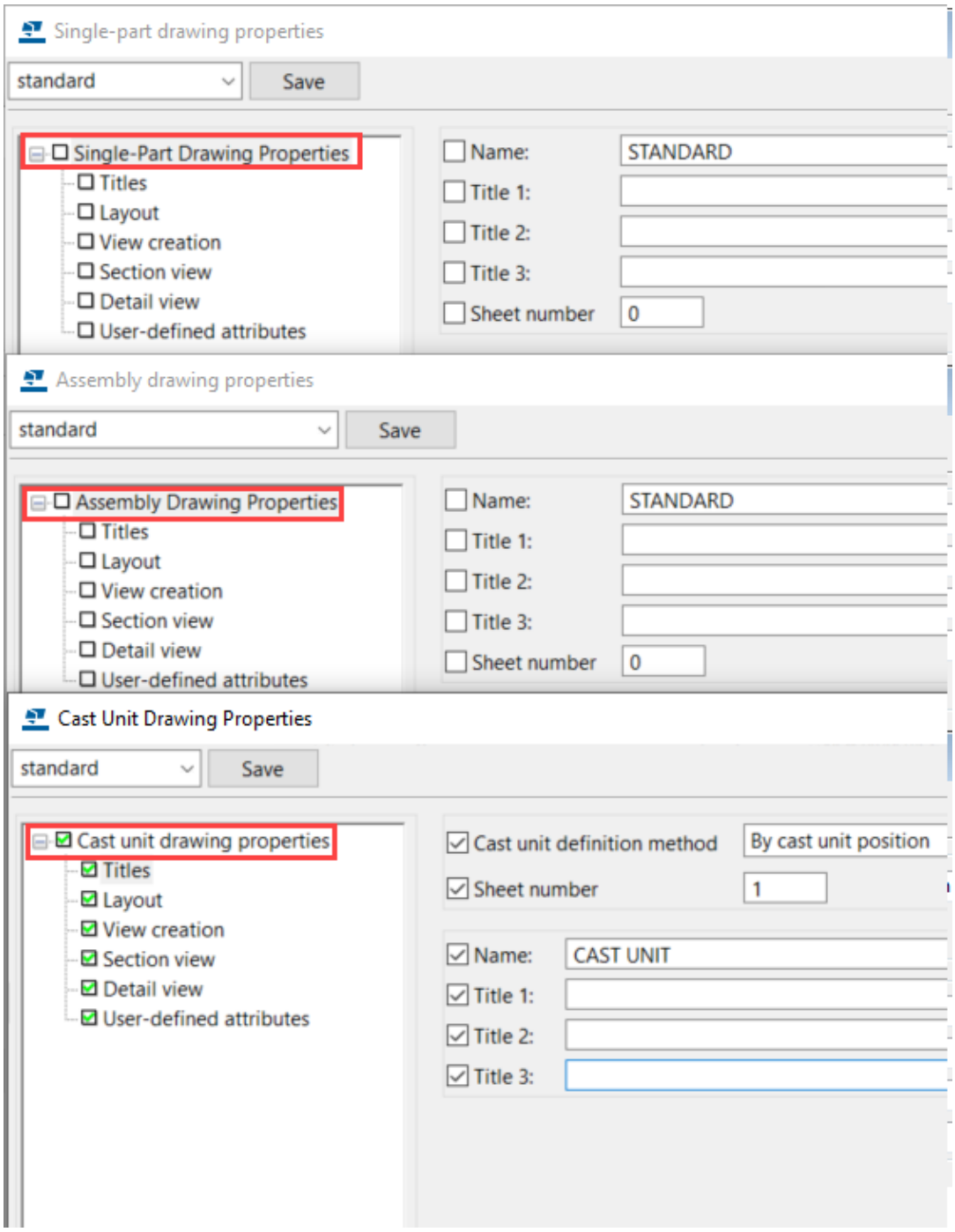
  You can open the drawing properties dialog boxes for setting the automatic drawing properties by going to the **Drawings & reports** tab, selecting **Drawing properties** and then selecting the drawing type.
- Drawing settings that are defined through various options and advanced options in **Options** and **Advanced options** dialog boxes.
- Additional settings files, such as rebar_config.inp (page 1023) for setting reinforcement, and hatch_types1.pat (page 866) for setting hatch patterns.

Single-part, assembly and cast unit drawing properties

Single-part, assembly and cast unit drawings have two types of automatic properties: drawing specific and view specific.

Drawing-specific properties apply to the whole drawing: coordinate system, coordinate system rotation, drawing titles, drawing layout, user-defined

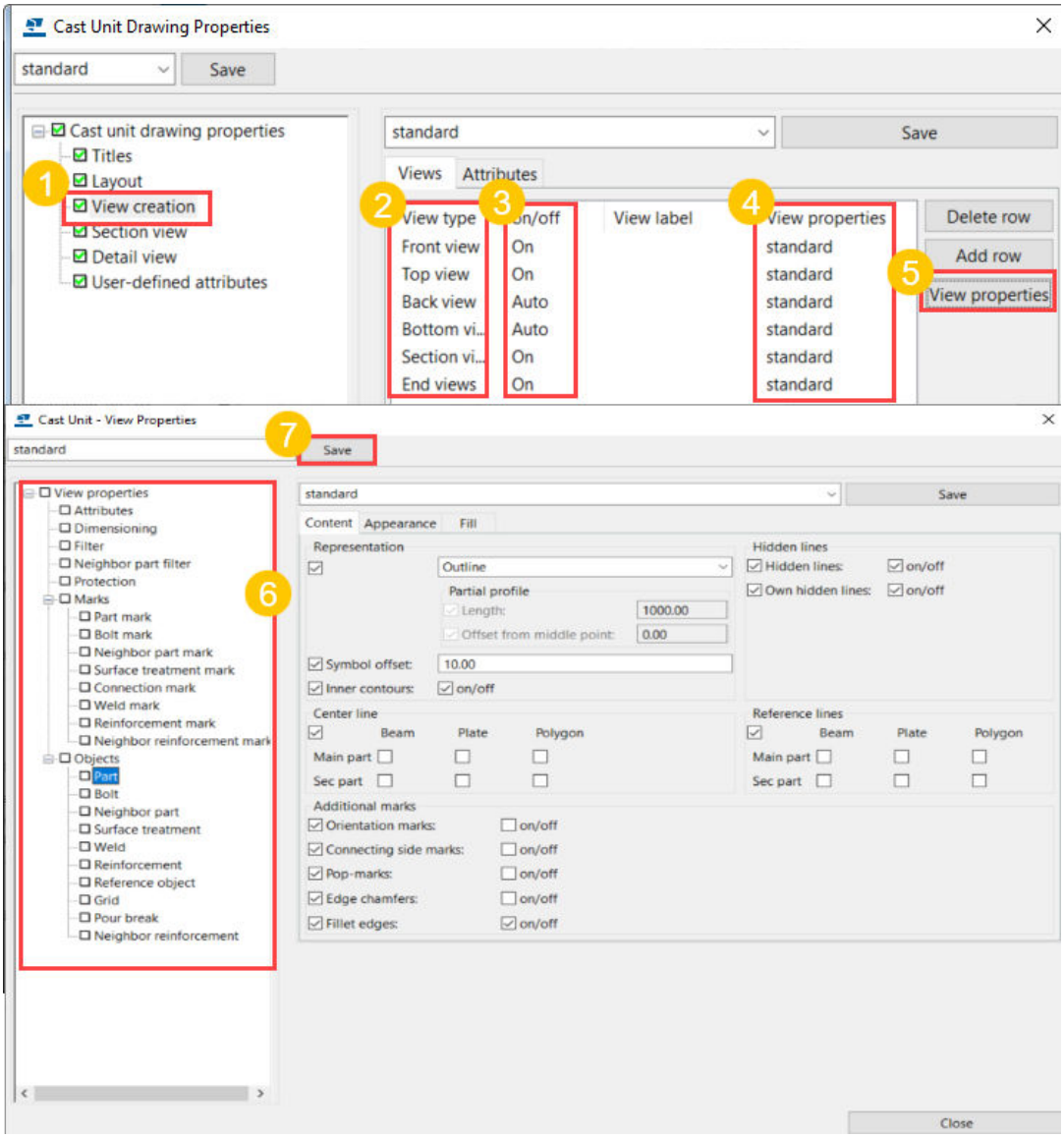
properties, and some detail and section view properties. In assembly drawings and single-part drawings, there are also some weld and bolt settings that are common for all drawing views. You can find these settings in the **Common settings for all views** area on the **View creation** page.



View-specific properties are defined separately for each view that you select to create. For example, you might want to show marks in one view, dimensions in another, and surface treatment in the third one. You may also want to define

different protection settings for each view. You can create as many views as you need.

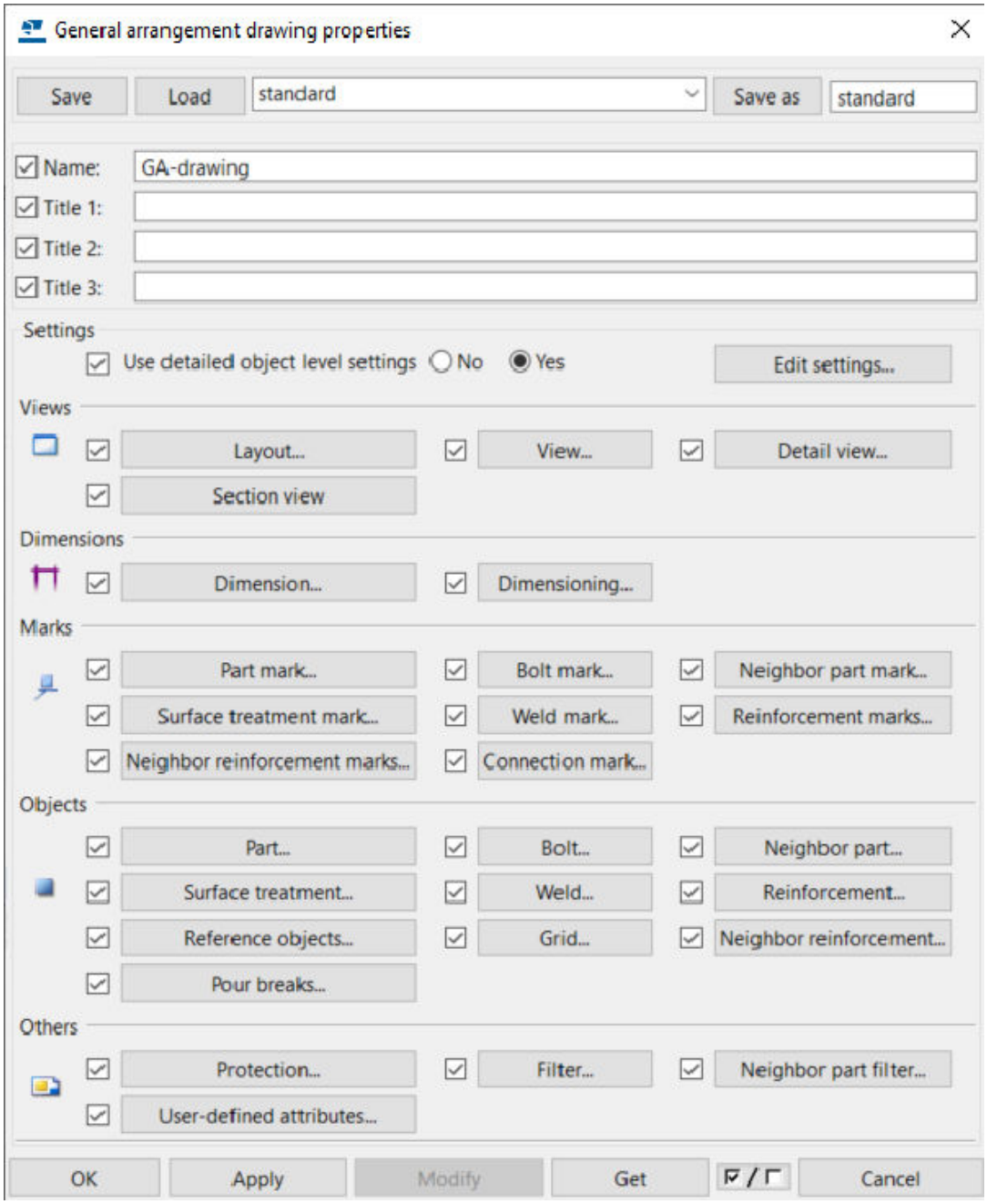
To specify the drawing views to create and the properties to use, follow the numbered path shown in the image below. First go to the **View creation** page, and select the views to create, and then select view properties to be used for the views. If you need to adjust view properties, click **View properties** and adjust the view properties, including protection settings, dimensions, filters, marks and objects. Always use **Save** to save the view properties, otherwise your changes are not saved.



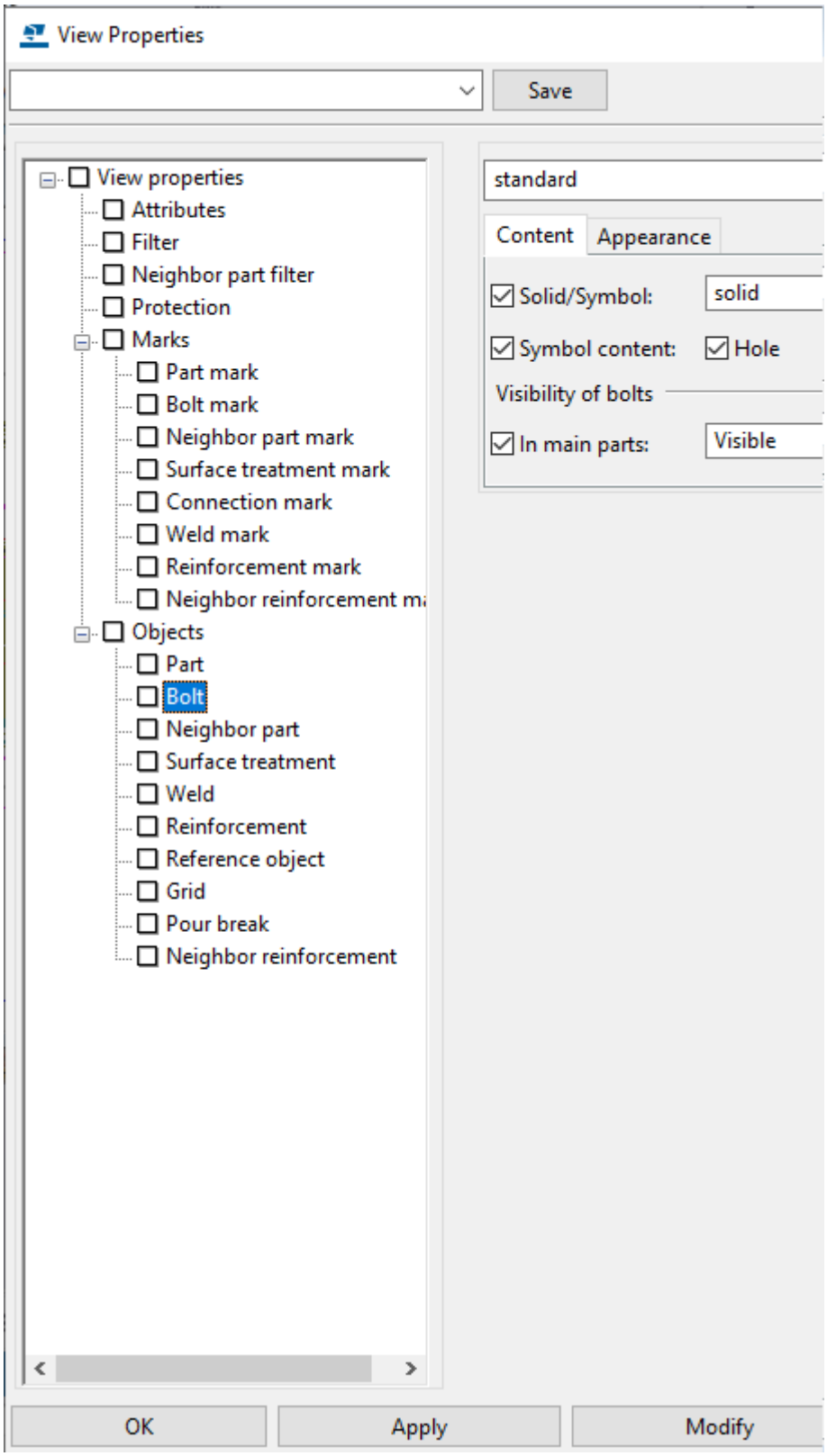
Drawing-level and view-level properties in single-part, assembly and cast unit drawings can be modified in an open drawing by double-clicking the drawing background or the view frame, respectively. The available properties are the same as in the dialog boxes shown above.

General arrangement drawing properties

Automatic drawing properties for general arrangement drawings can be defined on drawing level before creating the drawing. You can modify the drawing level properties in an open drawing by double-clicking the drawing background.

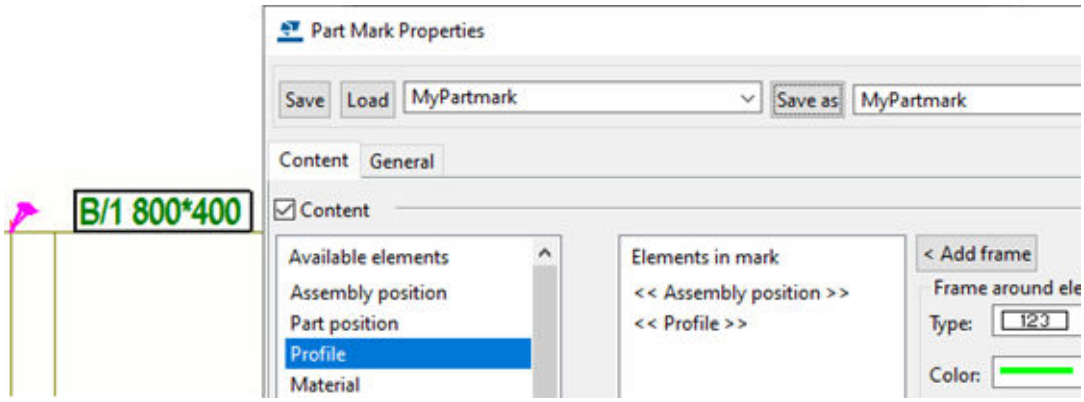


You need to select the drawing views that you want to create after you start the creation of a general arrangement drawing. You can adjust the view-level properties in an open drawing by double-clicking a view frame.


View Properties
Save
View properties
Attributes
Filter
Neighbor part filter
Protection
Marks
Part mark
Bolt mark
Neighbor part mark
Surface treatment mark
Connection mark
Weld mark
Reinforcement mark
Neighbor reinforcement ma
Objects
Part
Bolt
Neighbor part
Surface treatment
Weld
Reinforcement
Reference object
Grid
Pour break
Neighbor reinforcement
standard
Content
Appearance
Solid/Symbol:
solid
Symbol content:
Hole
Visibility of bolts
In main parts:
Visible
OK
Apply
Modify

Individual dimension, mark and object properties

You can modify the dimension, mark and object properties manually in an open drawing and save the changed properties in property files to be use later on for different purposes.



See also

Set automatic drawing properties before creating drawings (page 74)

Modify drawing properties of an existing drawing (page 77)

Modify drawing view-level properties (page 76)

Modify drawing object properties (page 78)

Detailed object level settings (page 79)

How Tekla Structures applies drawing properties in drawing creation (page 87)

8.1 Define drawing layout

A drawing layout defines which tables to include in a drawing and where the tables are located. Each drawing layout has its own table sets and applicable drawing sizes. You can create and edit drawing layouts through **Layout editor**.

The drawing layout connects drawing table sets to drawing sizes. Tekla Structures has several predefined drawing layouts. Each drawing type; assembly, single-part, cast unit, general arrangement, and multidrawing, can have its own individual drawing layout with necessary table sets. You can also create customized drawing layouts that can be used for more than one drawing type.

Using different layouts you can, for example, set assembly drawings to use A1 and A2 size drawing sheets, and single-part drawings to use A3 and A4 sheets. Another example is that you may want to include a material list in assembly drawings, but not in general arrangement drawings.

You can also customize the table sets within the same drawing layout, so that different drawing sizes have different table sets. For example, A1 sheets and

A4 sheets can have the same drawing layout, but they may require different positioning of table sets. By managing the number and location of tables, you can ensure that the tables fit to each drawing size. You can also exclude some tables from the drawing layout and edit them as part of the open drawing, so that the drawing layout does not change.

In previous versions of Tekla Structures, table sets were not associated to any drawing sizes if you used the **Specified size** option in drawing properties. In case you open an old layout file without associated table sets, Tekla Structures generates a drawing size to each table set automatically and names the drawing size in the following format: `###_[TABLE SET NAME]`. If needed, you can then customize the table set of each drawing size in **Layout editor**.

If you use the **Autosize** in drawing properties, Tekla Structures selects the smallest available drawing size that will fit the drawing views and the connected table set. You can select which drawing sizes are compatible with **Autosize** in the **Drawing size settings** dialog box.

Tekla Structures saves the new layouts you create in separate files with the file name extension `.lay`. The layout files are located in the `\attributes` sub-folder in the model folder. You can copy layout files to project or firm folders defined by the `XS_FIRM` and `XS_PROJECT` advanced options.

Note that the `.lay` files created in **Layout editor** are not compatible with **Drawing Layout** tool and **Layout editor** in previous Tekla Structures versions. This means that you cannot use new drawing layouts in Tekla Structures 2019i or earlier versions.

When an old `.lay` file is opened, all table sets in that `.lay` file that are not tied to any fixed drawing sizes in the same `.lay` file are converted to independent table sets. In the conversion, a new fixed size is automatically generated for each table set, and the table set is then tied to the newly generated size. The generated drawing sizes are always either A1 (841 mm x 594 mm) in non-imperial environments or ANSI D (864 mm x 559 mm) in imperial environments. The names of the generated drawing sizes are of the form `###_<name_of_converted_table_layout>`.

Create and edit drawing layouts

Use **Drawing layout editor** to create new drawing layouts and edit existing ones. Working on the layout, you can add, modify, move, and remove tables, add and modify drawing sizes, and select if you want to see table names or hidden tables in the drawing layout. Frames and fold marks are also added to drawings in **Drawing layout editor**.

Open Drawing layout editor

- To access **Drawing layout editor**, do any of the following:

To access Drawing layout editor	Do this
Through the **File** menu	• On the **File** menu, select **Editors** --> **Drawing layout editor** .
Through a drawing properties dialog box	1. Click **Layout**. 2. On the right side of the **Layout** list, click **Edit**.
Through the ribbon in the modeling mode	• On the **Drawings & reports** tab, click **Drawing properties** --> **Drawing layout editor** .
In an open drawing	Do any of the following: • Right-click an existing table set and select **Open Layout editor**. • Double-click an existing table set and select **Drawing layout editor**.
Through **Quick Launch**	1. Type `Drawing layout` in the **Quick Launch** box. 2. Select **Drawing layout editor** from the list.

The layout editing mode opens.

Create a new drawing layout

1. On the **Layout editor** ribbon, click **Create layout**.
2. Name the new drawing layout and select the initial drawing size.

   The drawing layout name can contain a maximum of 63 characters.

   Note that you can apply the layout to additional drawing sizes later.
3. Click **Create**.
4. In the **Layout editor** side pane, select which drawing types the new layout applies to.
5. Customize the drawing layout according to your needs.

   For example, add new tables to the layout or create and modify the available drawing sizes. See detailed instructions below.
6. To save the drawing layout, click **Save as** on the **Layout editor** ribbon.
7. In the **Save layout** dialog box, type a name for the drawing layout in the **Save layout with a name** box.
8. Select the drawing sizes to which you want to apply the current tables, drawing view margins, and spaces between drawing views.
9. To ensure that the drawing sizes that you have created for the current drawing layout are also available for printing, select the **Synchronize listed sizes with paper sizes for printing** check box.

If Tekla Structures detects new drawing sizes, a copy of the `Papersizesfordrawings.dat` file is created in the `\attributes` folder under the model folder.

Note that the drawing sizes and paper sizes are only synchronized in the current model.

10. Click **Save**.

The drawing layout is saved in the `\attributes` folder under the model folder as a `.lay` file. Note that the `.lay` files are not compatible with the **Drawing layout** tool or the **Layout editor** that were available in previous versions of Tekla Structures. This means that you cannot use the new layout with older versions of Tekla Structures.

You can now continue working with drawing layouts or close the layout editing mode by clicking **Close Layout editor**.

Add new tables to a drawing layout

Here, we refer to all the elements that you can add to the drawing layout as *tables*. The elements include:

- Template files (`.tpl`)
- DWG/DXF files

  For example, you might have some details in a DWG or DXF file that you want to show in certain types of drawings and therefore add the file in the drawing layout.
- Key plans

  Tekla Structures automatically includes the correct object in the key plan. You can use drawings that contain only one view of the correct scale as a key plan. Tekla Structures only uses the view from the original drawing. The view position, drawing size, and templates of the original drawing are not relevant to the key plan.

The combination of tables in the drawing layout is called a *table set*.

To add new tables to the drawing layout:

1. In the **Drawing size** list, select a drawing size to which you want to add tables.

   You can apply the changes to multiple drawing sizes when you save the drawing layout.

2. On the **Layout editor** ribbon, click **Add tables**.
3. In the **Available tables** dialog box, click the table that you want to add.

4. If you are adding a DWG or DXF file, or a keyplan, do one of the following:

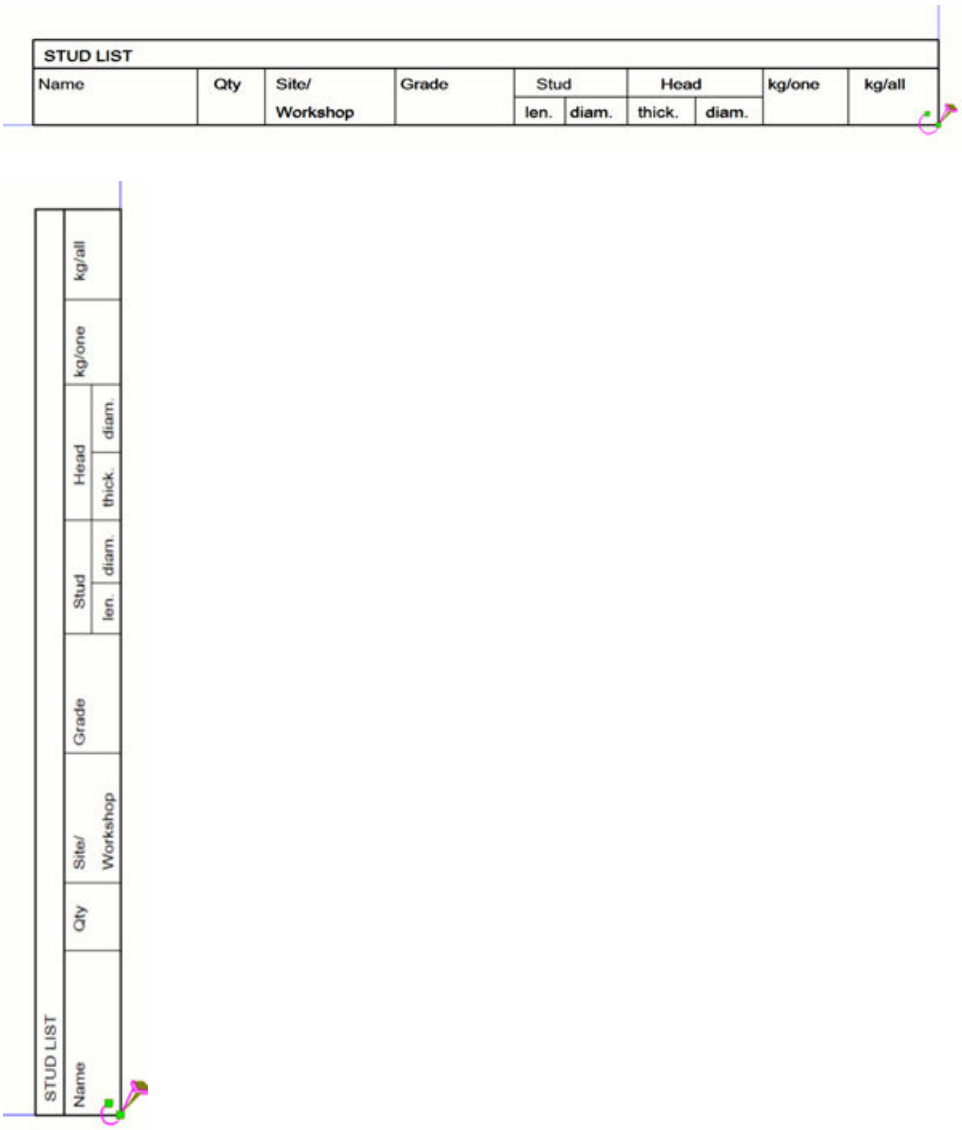
To	Do this
Add a DWG or DXF File	a. Select the file in the list of available tables. b. Browse to find and select the DWG/DXF file that you want to add to the drawing. c. Click **Open**. If the DWG/DXF file can be found under the model folder, a relative path will be used, otherwise absolute path is used. If you want to use a relative path (current model/project/firm/env), do the following: a. Copy the DWG/DXF file in your model folder. b. Insert the DWG/DXF file. c. Save the layout. The path to your DWG/DXF file will now be defined with path ".\", which will in runtime evaluate using same search order as other layout content.
Add a keyplan	a. Select **Keyplan** in the list of available tables. b. In the **Drawings** dialog box, select an existing key plan drawing and click **Select**. Note that keyplans are shown in drawings even if the layout was created in a different language than what is used when the drawing containing the keyplan is opened.

5. Close the **Available tables** dialog box.
6. Click the point in the drawing layout where you want to add the table.

   The tables are automatically anchored to the drawing view frame.
7. If you want to change the table scale or rotation, select the table and type new **Scale** or **Rotation** values.

Note that you can only edit the scale and rotation of template files (`.tpl`).

Rotation and scale changes are applied to the drawing layout immediately. When you scale or rotate a table, the table remain anchored to the same position.



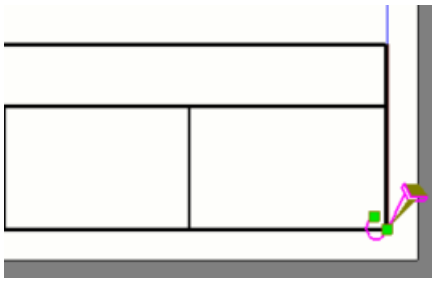
8. If you want to allow the table to overlap with a drawing view, select the **Overlap with views** check box.

   If you do not select the **Overlap with views** check box, Tekla Structures keeps the drawing views outside the tables area when it creates or recreates drawings. Note that overlap changes are applied to the drawing layout only during drawing recreation and creation.

Move tables in the drawing layout

1. On the **Drawings & reports** tab, click **Drawing properties** --> **Drawing layout editor** .
2. From the **Drawing layout** list, select the drawing layout.
3. In the drawing layout, do either of the following:
   - To move one table, click the table that you want to move.
   - To move several tables, hold down the left mouse button and drag a box around the tables that you want to move.

   Two green handles appear.

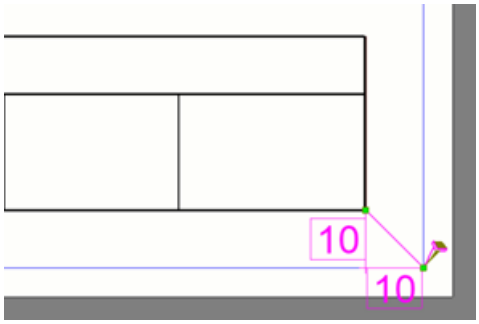
   

   The handle that is further from the table determines where the table is anchored. The closer handle determines where one corner of the table is located.
4. Do one of the following:
   - Click the table that you want to move, and hold down the left mouse button.
   - Pick one of the green handles, and hold down the left mouse button.
5. Drag the tables to a new position.

   The tables snap to offsets of 1 millimeter in metric environments, or 1/16 inch in the US imperial environment.

   Tekla Structures shows the distance between the handles in both the vertical and the horizontal direction. The distance is shown in either millimeters or fractions of inches, depending on the environment that you are using.
6. To place the tables, release the left mouse button.

The tables are moved to the new position.



Replace tables in the drawing layout

1. In an open drawing layout, right-click a table that you want to replace.
2. In the context menu, select **Replace table**.
3. Select a new table from the **Replace table** dialog box and click **Replace**.

The table is replaced and the name of the new table is shown in the **Tables in use** list. All the settings that you had defined for the old table are preserved.

Remove tables from the drawing layout

- Do any of the following:
  - In the drawing layout, right-click a table and select **Delete**.
  - In the drawing layout, click a table and press **Delete** on the keyboard.
  - Select a table in the **Tables in use** list and press **Delete** on the keyboard.

Adjust drawing sizes, frames, and fold marks

Frames and fold marks can be customized individually for each drawing layout, or `.lay` file. However, the frame properties are initially read from the `standard.fms` file with global frame and fold mark properties. If you do not want to use the global frame and fold mark properties in a drawing layout, adjust the appropriate properties in the **Drawing size settings** dialog box.

1. In the **Layout editor** side pane, click the **Edit** button on the right side of the **Drawing size** list.

2. In the **Sizes in use** section in the **Drawing size settings** dialog box, edit the drawing sizes according to your needs:

To	Do this
Add a new drawing size	a. Click **+** under the list of existing drawing sizes. A new line is added at the bottom of the list of drawing sizes. b. Either select a predefined drawing size from the list in the **Name** column, or type a name for the new drawing size. The predefined drawing sizes in the **Name** list are defined in the `PaperSizesForDrawings.dat` file. By default, the file is located in the `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\system folder` folder. c. To adjust the width and height, double-click the **Width** and **Height** boxes and type new values. d. If you want to allow Tekla Structures to use the drawing size when automatically selecting suitable drawing sizes for drawings, select the **Autosize** check box. For more information on autosizing, see Define drawing size and drawing view scale (page 673).
Adjust the width and height of a drawing size	a. Select the drawing size whose width and height you want to modify. b. Double-click the **Width** and **Height** boxes and type new values. **TIP** You can change the page orientation by right-clicking the **Width** or **Height** box and selecting **Swap orientation**. The width and height of the drawing size are swapped.
Enable using the drawing size for automatic drawing sizes	a. Select the drawing size that you want to use in autosizing. b. Select the **Autosize** check box.
Delete a drawing size from the drawing layout	• Click **-** on the right side of the drawing size that you want to delete.

3. In the **Frames** section, adjust the frames:
   a. Select which frames you want to have in the drawing layout.

      We recommend that you use at least one frame in your drawings. If you want to use only one frame, select the **Primary** frame.

      If you use fold marks in your drawings, select both the **Primary** and the **Secondary** frames, and use the secondary frame as the outer frame.

   b. Type the distances in millimeters or inches between each frame and the left, right, bottom, and top edges of the paper.

   c. Select a color for each frame.
4. If you use fold marks in your drawing layouts, define the fold mark properties:
   a. Type the horizontal and vertical distances between fold marks in millimeters or inches.

   b. Select a color for the fold marks.
5. To save the settings and close the **Drawing size settings** dialog box, click **OK**.

Adjust the position of drawing views

To adjust the position of drawing views, you need to have a drawing open.

1. Scroll to see the **Drawing views** section of the **Layout editor** side pane.
2. In the boxes under the **Margins** heading, type the horizontal and vertical distances between the outermost drawing view and the drawing edges or the table edges.

   Note that Tekla Structures automatically adds 5 millimeters to the distances that you type.
3. In the boxes under the **Spaces** heading, type the horizontal and vertical spaces between drawing view frames.

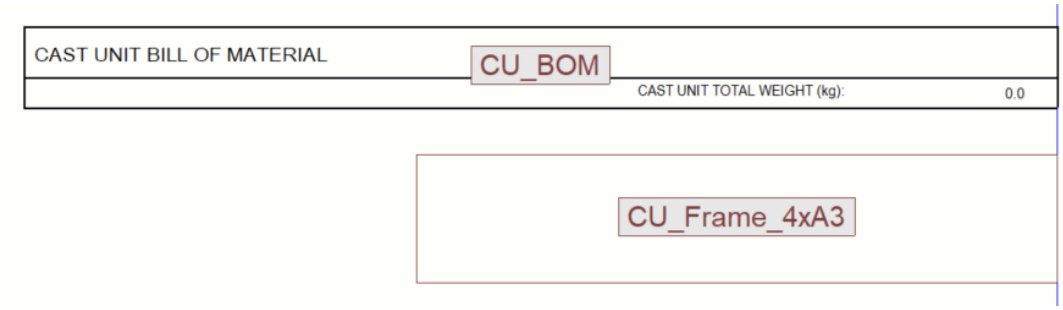
Note that the **Drawing views** settings work together with the XS_DISABLE_VIEW_CENTERING_[drawing type] advanced options. By setting the values of these advanced options to `VER`, `HOR`, or both, you can disable automatically centering drawing views in horizontal, vertical, or both directions.

You can enable or disable centering of drawing views manually for all drawing types except for cast unit drawings. In cast unit drawings, centering of drawing views is always enabled.

Adjust the visibility of tables in the drawing layout

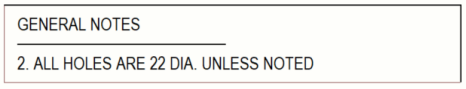
The drawing layout contains some information that you may not always need to see, because some tables are not relevant in all drawings that use the same drawing layout. These tables are called *hidden tables*. If the content of a drawing changes, these tables may appear again. For example, a table may become visible if you add a revision.

By default, hidden tables are shown as simple boxes that have a known width and a default height. In the image below, the upper table is a regular table, while the lower table is a hidden table.



If necessary, you can choose to hide hidden tables from the drawing layout completely.

- Do any of the following:

To	Do this
Hide or show hidden tables	• In the **Layout editor** side pane, click next to **Show hidden tables**. • To show hidden tables again, click next to **Show hidden tables**.
Hide or show table names	• In the **Layout editor** side pane, click next to **Show table names**. The table names are hidden.  • To show hidden tables again, click next to **Show table names**. The table names are shown again.

Edit a drawing layout

TIP You can remove or rename drawing layouts in the file folder. Go to the `\attributes` sub-folder under the model folder, find the right `.lay` file, and remove or rename the file.

1. On the **Drawings & reports** tab, click **Drawing properties** --> **Drawing layout editor** .
2. From the **Drawing layout** list, select the drawing layout that you want to edit.
3. Edit the drawing layout according to your needs.

   For example, you can add, move or remove tables, or change the drawing sizes.
4. To save the drawing layout, click **Save as** on the **Layout editor** ribbon.
5. If you want to save the edited layout with another name, in the **Save layout** dialog box, type a name in the **Save layout with a name** box.

   If you do not type a new name for the layout, the changes you made overwrite the existing layout (`.lay`) file.
6. Select the drawing sizes to which you want to apply the current tables, drawing view margins, and spaces between drawing views.
7. To ensure that the drawing sizes that you have created for the current drawing layout are also available for printing, select the **Synchronize listed sizes with paper sizes for printing** check box.

   If Tekla Structures detects new drawing sizes, a copy of the `Papersizesfordrawings.dat` file is created in the `\attributes` folder under the model folder.

   Note that the drawing sizes and paper sizes are only synchronized in the current model.
8. Click **Save**.

The changes that you made are saved to the drawing layout (`.lay`) file. All drawings that use the edited layout are automatically updated to match the changes.

You can now continue working with drawing layouts or close the layout editing mode by clicking **Close Layout editor**.

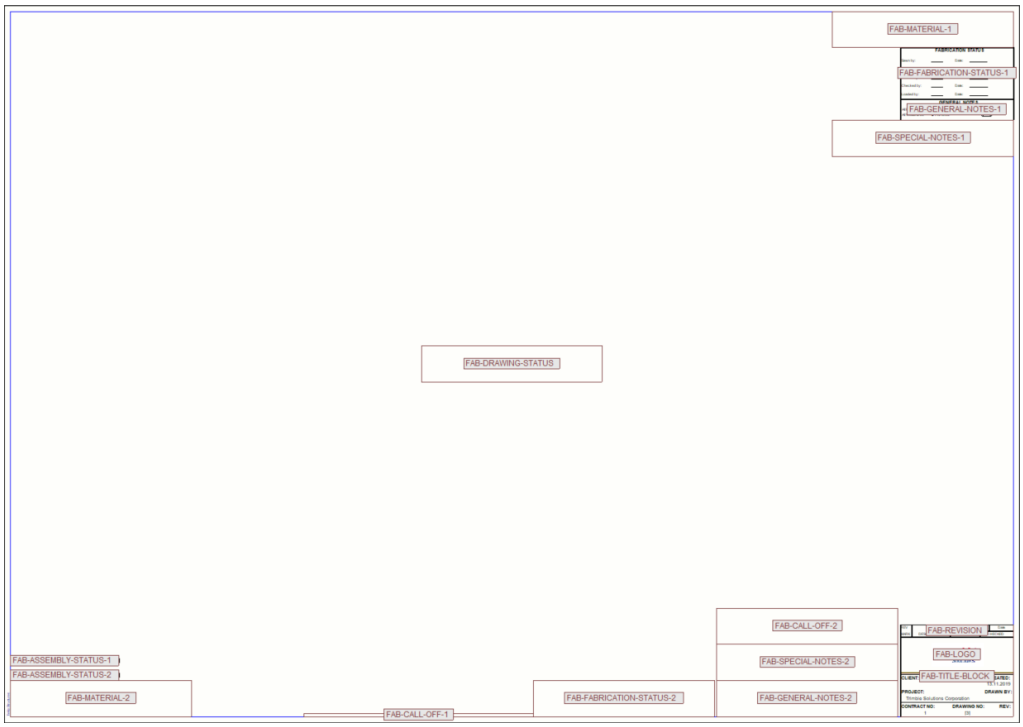
Example: Use different table sets for different drawing sizes within a drawing layout

With **Drawing layout editor**, you can create separate table sets for different drawing sizes within the same drawing layout. This way, you can ensure that the tables fit to each drawing size.

In this example, we will create one table set for the `A1`, `A2`, and `A3` drawing sizes, and another table set for the `A4` drawing size. We will do this for an existing drawing layout.

1. From the **Drawing layout** list in the **Layout editor** side pane, select the drawing layout that you want to edit.
2. Click the **Edit** button on the right side of the **Drawing size** list.
3. In the **Drawing size settings** dialog box, click **+** and create the `A1` drawing sizes.
4. To close the **Drawing size settings** dialog box, click **OK**.
5. In the **Drawing size** list, select **A1**.
6. On the **Layout editor** ribbon, click **Add tables**.
7. Add tables and move them to appropriate positions in the drawing layout.

   See an example below.



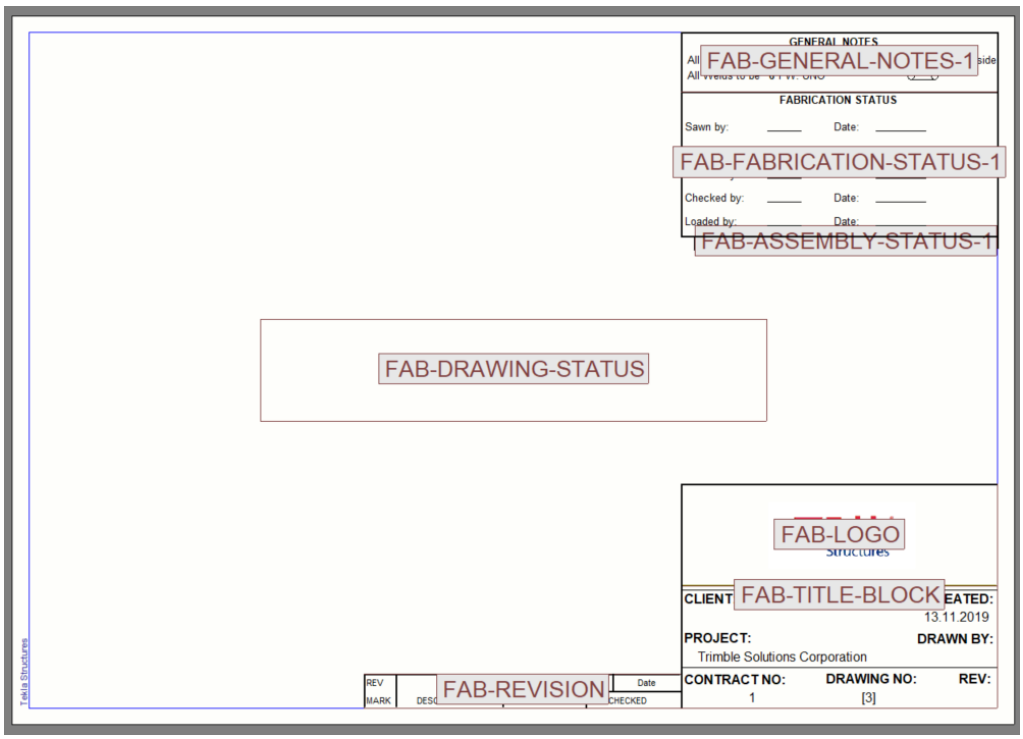
The above drawing layout shows both hidden tables and table names.

When you have placed the tables as you wish, we can apply the table set to more drawing sizes.

8. Again, click the **Edit** button on the right side of the **Drawing size** list.
9. In the **Drawing size settings** dialog box, click **+** and create the following drawing sizes:
   - A2
   - A3
   - A4

   The table set that you created for the A1 drawing size is automatically copied for the new drawing sizes.

   In this example, we do not want the A4 drawing size to use the same table set, so we will continue to adjust the table set.
10. To close the **Drawing size settings** dialog box, click **OK**.
11. In the **Drawing size** list, select **A4**.
12. Move, add or remove tables. See an example below.



The above drawing layout shows both hidden tables and table names.

13. On the **Layout editor** ribbon, click **Save as**.
14. In the **Save layout** dialog box, type a name for the drawing layout.

15. Clear the **A1**, **A2**, and **A3** check boxes.

    This way, the new table set is only applied to the `A4` drawing size.

16. Click **Save**.

Tables in drawing layout

Tables are Template Editor templates added in Tekla Structures drawings containing information on model objects. In Template Editor, the tables are referred to as graphical templates.

The term *table* refers to various elements in a drawing layout, such as:

- Tables (such as revision tables)
- Title blocks
- Lists (such as part and bolt lists)
- General notes
- Key plans
- DWG/DXF files

If you change the model, Tekla Structures updates the contents of the affected drawings and tables created in Template Editor. The contents of the tables are filled in by Tekla Structures at run time.

The available graphical templates are read from the following folders in the following order, and shown in the **Available tables** list in the **Layout editor** side pane:

- Template folder (`XS_TEMPLATE_DIRECTORY`)
- Current model folder
- Project folder (`XS_PROJECT`)
- Firm folder (`XS_FIRM`)
- Environment-specific system templates folder (`XS_TEMPLATE_DIRECTORY_SYSTEM`)
- System folder (`XS_SYSTEM`)

See also

Edit tables in Template Editor (page 671)

Create and edit drawing layouts (page 656)

Table sets

A table set is a group of tables or templates included in a drawing of a particular type and size.

The table set defines:

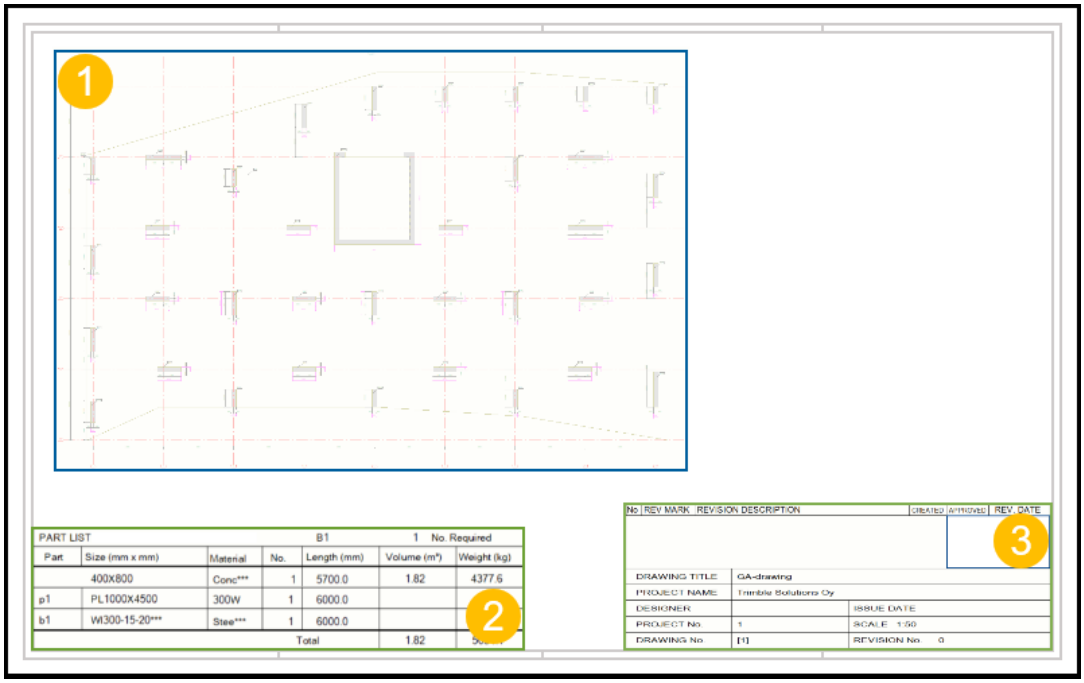
- Which tables are included in the drawing
- Where the tables are located in the drawing
- How much space Tekla Structures leaves between the drawing frame and views and between the drawing views

Table sets define the background of the drawing, not the number or location of the drawing views to include.

You can either use the same table sets with different drawing sizes, or give each drawing size its own table set. For example, if the number of views changes in a drawing and Tekla Structures chooses a new drawing size, Tekla Structures may also choose another table set automatically.

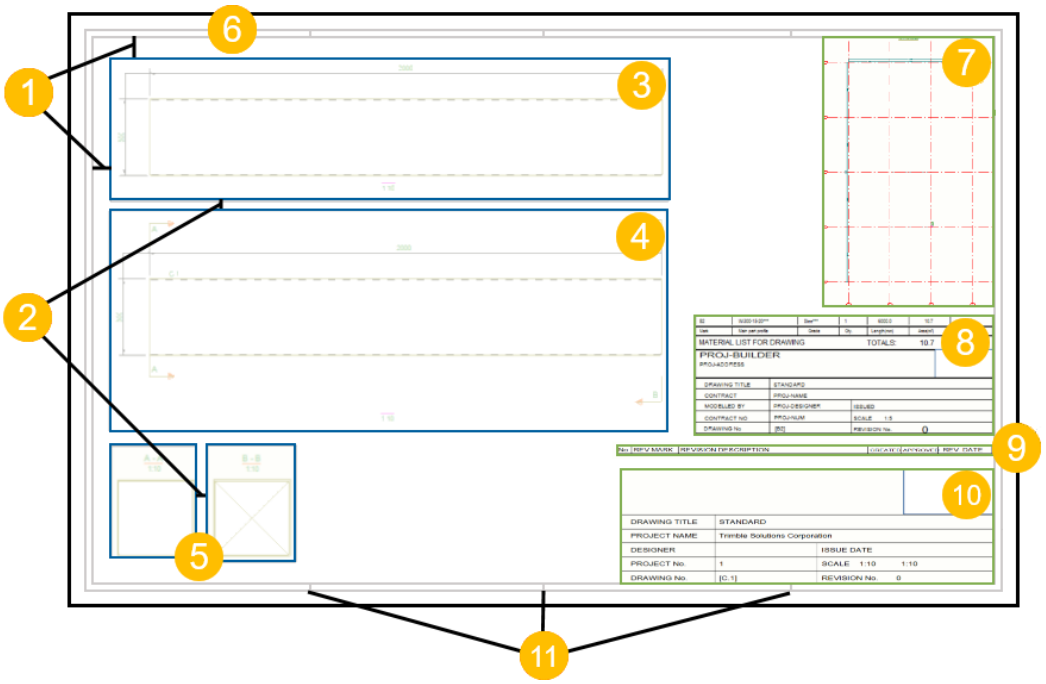
The examples below illustrate the relationship between the table set and drawing views. The drawing views are blue, and the elements of the table set are green.

Below is an example of a GA drawing layout.



1. General arrangement drawing view
2. Part list on assembly or cast unit level
3. Revision table and title block

Below is an example of an assembly drawing layout.



1. Margins between the drawing frame and the outermost views
2. Spaces between the views
3. Top view
4. Front view
5. Section views A-A and B-B
6. Drawing frame
7. Key plan
8. Material list
9. Revision table
10. Drawing title block
11. Fold marks

See also

Create and edit drawing layouts (page 656)

Tables in drawing layout (page 669)

Select a new layout for your drawing (page 174)

Edit tables in Template Editor

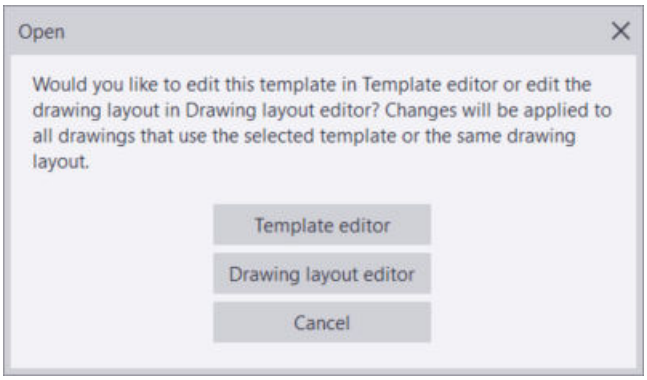
If you need to edit a table in a drawing layout, you can open it in Template Editor (TplEd). In Template Editor, drawing layout tables are called templates.

You can only open tables with templates created or saved in Template Editor version 3.2. or later.

If your templates are located in a protected folder, the templates are read-only, and you cannot save modified templates in a protected folder. In this case, you need to start Tekla Structures as an administrator.

1. In a drawing, double-click the table you want to modify.

   The following dialog box opens:

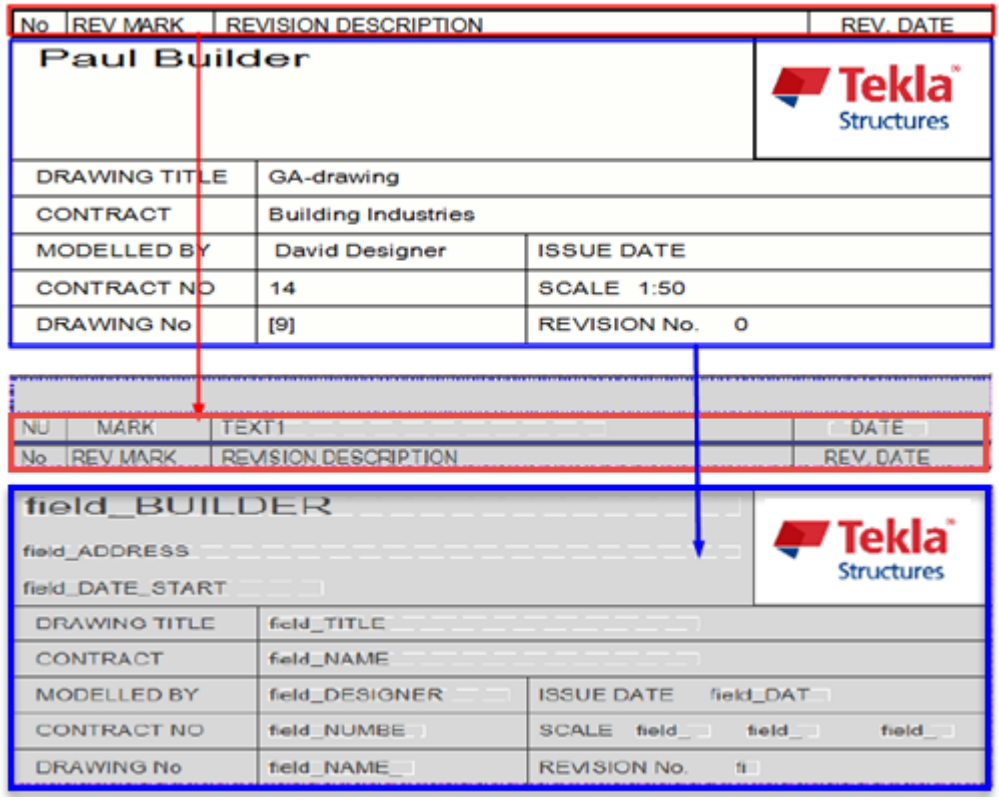
2. Click **Template Editor**.

   Tekla Structures starts Template Editor, and the selected template is displayed.

3. Modify the template and save the changes by selecting **File** --> **Save** , or **Save as** to use another folder, for example the model folder.

Template Editor user's guide in PDF format: Tekla Structures PDF documentation.

The example below illustrates how a table looks in a drawing and in Template Editor. In this example, there is a revision table and a title block. The revision table is located above the title block and bound to the title block in the layout.



8.2 Define drawing size and drawing view scale

Tekla Structures offers you three combinations of settings that you can use for defining the drawing size and the drawing view scale. You can use exact drawing view scale and automatic drawing size, automatic drawing view scale and exact drawing size, or both automatic drawing scale and drawing size.

What happens in each of the three combinations of settings:

- **Autosizing**: If you always want to use a certain drawing view scale, then you would set the exact scale to be used and let Tekla Structures automatically select the drawing size that suits the selected scale.

  You can define different scales for main views and section views. All main views in a drawing automatically use the same scale unless you fine-tune individual views manually.

- **Autoscaling**: If you always need to use a certain drawing size, for example, A3, A4 or A1, then you would set the size and let Tekla Structures automatically select the drawing view scale that suits the selected size.

  Tekla Structures first tries to use the preferred scale for the drawing views, then the alternative scales, and selects the largest possible scale.

- **Autosizing and autoscaling**: If you are not bound to any specific drawing sizes or drawing view scales, you might want to let Tekla Structures take care of both.

Set exact drawing view scale and automatic drawing size

You can let Tekla Structures find appropriate sizes and table sets for drawings. Tekla Structures automatically updates drawings to adapt to these changes by using different drawing sizes and table sets.

Using exact drawing view scale and automatic drawing size is very useful, for example, when the number, size, or location of drawing views may change.

Note that the following instructions do not apply to GA drawings.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. In drawing properties, click **Layout** in the option tree on the left.
4. Set **Size definition mode** to **Autosize**.

   The available sizes for autosizing are defined in the **Drawing size settings** dialog box, which you can access through **Layout editor**.
5. Go to the **Scale** tab and set **Autoscale** to **No**.

   This way Tekla Structures uses the exact scale that you have set for the main views and section views.
6. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
7. On the **Attributes 1** tab, set the desired **Scale**.
8. Save the view properties and click **Close**.
9. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

When you create the drawing, Tekla Structures creates the views using the selected scale and selects the smallest drawing size where the views fit in the drawing.

Note that each drawing size might have its own individual table set, so the table set might also change when Tekla Structures adapts another drawing size. Tekla Structures uses only drawing sizes that use the drawing layout that you have selected from the **Layout** list.

Set exact drawing size and automatic drawing view scale

If you need to use a specific size in your drawings, for example, A3, A2, or A1, you can specify the exact size, and let Tekla Structures automatically select the suitable drawing view scale. The drawing size should always be smaller than the actual paper size to accommodate printer margins.

Note that the following instructions do not apply to GA drawings.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. In drawing properties, click **Layout** in the options tree on the left.
4. On the **Drawing size** tab, set **Size definition mode** to **Specified size** .
5. Select the size from the **Drawing size** list.

   The available drawing sizes are defined through the (page 656).
6. Select the table set from the **Layout** list.
7. On the **Scale** tab, set **Autoscale** to **Yes**.
8. Set the **Main view scales** and **Section view scales**.

   Enter the denominators of scales and separate them with spaces. For example, enter "5 10 15 20" for the scales 1/5, 1/10, 1/15, and 1/20.
9. Select the **Scale change mode** option that defines the relationship between the scales of main and section views within a drawing.

   The options are:

   - **main = section**: The scales of the main and section view are equal.
   - **main < section**: Main view scales are smaller than section view scales.
   - **main <= section**: Main view scales are smaller than or equal to section view scales.
10. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
11. Enter the preferred scale.

    Do the same for each view that you create.
12. Save the view properties and click **Close**.

13. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

Tekla Structures creates the drawing using the specified size. Tekla Structures first tries to use the preferred scale for the drawing views, then the alternative scales, and selects the largest possible scale.

Autoscale and autosize drawings

If you are not bound to any specific drawing sizes or drawing view scales, you can let Tekla Structures take care of both.

First set autoscaling and after that autosizing.

Note that the following instructions do not apply to GA drawings.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. On the **Attributes 1** tab, set the preferred drawing view scale in the **Scale** box.

   Do the same for all of the views you plan to create.
5. Click **Save** to save the view properties. Do this for all of the views you have modified.
6. Click **Close** to return to drawing properties.
7. Click **Layout**, go to the **Scale** tab and set **Autoscale** to **Yes**.
8. Set the alternative **Main view scales** and **Section view scales**.

   Enter the denominators of scales and separate them with spaces. For example, enter "5 10 15 20" for the scales 1/5, 1/10, 1/15, and 1/20.
9. Select the **Scale change mode**, which defines the relationship between the scales of main and section views within a drawing.

   The options are:

   - **main = section**: The scales of the main and section view are equal.
   - **main < section**: Main view scales are smaller than section view scales.
   - **main <= section**: Main view scales are smaller than or equal to section view scales.
10. Enter the **Preferred size**.
11. On the **Drawing size** tab, set **Size definition mode** to **Autosize**.

12. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

When you use both autoscaling and autosizing at the same time, Tekla Structures follows the steps below:

- First Tekla Structures tries to find a drawing size where the drawing contents fit by first trying to use the exact scale defined in **View Properties** --> **Attributes 1** and the smallest drawing size defined in the current layout. The sizes are defined through **Drawings & reports** --> **Drawing properties** --> **Drawing layout editor** .

  For more information, see Create and edit drawing layouts (page 656).
- Then Tekla Structures increases the drawing size until it reaches the **Preferred size** defined in **Layout** --> **Scale** .
- If the drawing fits with the original scale, Tekla Structures tries to increase the scale using the alternative main and section view scales you defined in **Layout** --> **Scale**.
- If the drawing does not fit to any of the defined scales, Tekla Structures starts to increase the drawing size until the contents fit, using **Autosize**. When necessary, Tekla Structures changes to another appropriate table set within the current layout.
- When the views fit, Tekla Structures begin to increase the scale again so that the final drawing uses the largest possible scale.

8.3 Define object protection and placement settings in drawings

When you create a drawing, Tekla Structures uses predefined rules for positioning marks and dimensions. Marks and dimensions are automatically placed in the first suitable position.

The following settings are considered when placing the objects:

- The protected areas defined in drawing view properties in single-part, cast unit and assembly drawings. In GA drawings they can be defined on both levels. How the protection settings are applied depends on the drawing order of the objects in the drawing: the parts are drawn first, then the marks and then the dimensions.
- The placing and leader line type settings (page 826) of the annotation objects defined in the annotation object properties. Marks are usually drawn before dimensions, but if the marks have leader lines, they are drawn after the dimensions.
- The predefined mark location and part orientation settings for some marks. For more information, see Set a predefined location for beam, bracing and column marks (page 824).

- The modeling direction of parts.
- XS_MARK_INTELLIGENT_PLACING: Marks are automatically placed according to the mark placing algorithm if this advanced option is set to TRUE (default). The placement algorithm tries to avoid crossing leader lines, and also places marks otherwise more clearly.
  XS_MARK_INTELLIGENT_PLACING is available in the category **Marking - general** in the **Advanced options** dialog box. The algorithm applies to all marks and associative notes, except weld marks.
- XS_MARK_PLACING_ANGLE_CLOSE_TO_45_DEGREES: If you set this advanced option to TRUE (default), marks with leader lines are placed in a 45-degree angle if allowed by the protection settings.
- XS_MARK_INTELLIGENT_ POST_FREEPLACE_NEARBY: If you set this advanced option to TRUE (default), Tekla Structures first places the marks avoiding crossing leader lines and after that runs the place nearby command, which ensures that the mark locations follow the protection settings. If you set this advanced option to FALSE, the crossing mark check is performed, but the place nearby command is not run, so some of the protection settings may not be followed.

When you create a drawing, Tekla Structures places the annotation objects according the automatic placement settings and protection settings. You can modify the placement settings in drawing view properties and on object level for individual objects. The protection settings are also considered when you add annotation objects manually.

When you create a drawing, Tekla Structures places the views in the drawing according the settings in the drawing layout and view properties. In final drawings, you can select whether to use fixed or free view placement of views, drag drawing views to new locations, or align them.

To	Click a link below to find out more
Specify protected areas in your drawings	Protect areas in a drawing (page 679)
Arrange drawing objects	Arrange annotation objects (page 416)
Align drawing objects	Align selected drawing objects (page 417)
Specify how the marks or dimensions are placed	Define placement settings for marks (page 683) Define placement settings for dimensions (page 685)
Specify whether the view placement is fixed of free	Define automatic free or fixed placement for drawing views (page 686)

See also

Placement properties for marks, notes, dimensions, texts and symbols in drawings (page 1033)

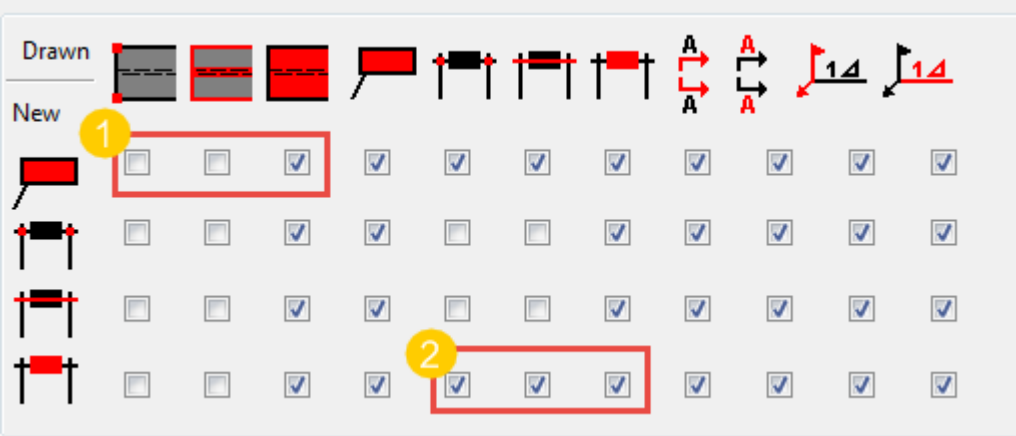
Protect areas in a drawing

You can define protected areas in drawings to prevent text, marks or dimensions from being placed in that area. When Tekla Structures places text, marks, dimensions or other annotation objects in a drawing, it first checks the protection settings.

The drawing view properties contain pre-defined protection settings, which you may change. In GA drawings, you may also define protection settings on the drawing level.

For example, In GA drawings you often want to have different protection settings in plan views and section or detail views. In plan views, you may want to show marks inside the member, for example, a slab. In section and detail views all marks need to be outside the member.

Below is an example of the protection properties dialog with explanations of the selections.



(1) With a selection like this, text and marks may overlap with part corners and edges, but not with part content.

(2) With a selection like this, dimension values may not overlap with other dimension arrowheads, lines or values.

The objects at the top of the dialog box define the areas to be protected, see descriptions below:

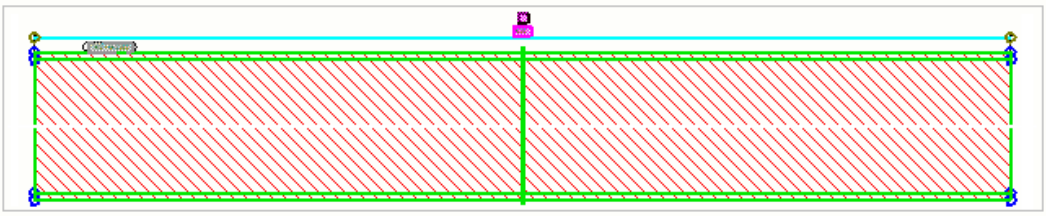
Column	Description
	Part corners

Column	Description
	Part edges
	Part content
	Text, mark or weld mark
	Dimension arrowhead
	Dimension line
	Dimension value
	Cutting line
	Section mark
	Weld arrow
	Weld mark content

The objects on the left in the dialog box define which objects, or object elements, Tekla Structures cannot place in the protected areas. See the descriptions below:

Row	Description
	Text, mark or weld mark
	Dimension arrowhead
	Dimension line
	Dimension value

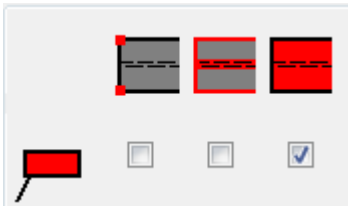
You can check which areas are protected by using the **Show protection** command. You can enter this command in the **Quick Launch** box. This command shows the protected areas with colors.



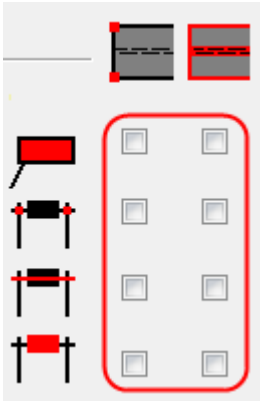
Define automatic protection settings in single-part, assembly and cast unit drawings

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Click **View creation** in the options tree, and define the drawing views you want to create.
4. In the **View creation**, select a view row, and click **View properties**
5. Click **Protection** in the options tree.

   The settings are same for all drawing types.
6. Select check boxes to indicate the areas that you want to protect from text, marks, weld marks, associative notes, dimension arrowheads, dimension lines or dimension values.
   - For example, to prevent text, marks, dimensions or other annotation objects from being placed over the inside area of parts, select the third check box on the first row. The first row defines the placement of annotation objects, and the icon above the third check box indicates the inside area of parts. With this setting, Tekla Structures may place annotation objects on the corners and edges of parts.

     
   - If you clear the first two columns of check boxes, Tekla Structures does not protect part corners and edges. This increases drawing speed and

decreases the memory requirements. You can still protect these areas by using the third column of check boxes for part protection.

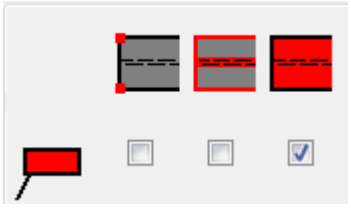


7. Click **Save** to save the view properties.
8. Click **Close** to return to drawing properties.
9. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

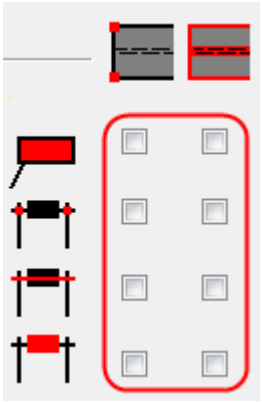
Define automatic protection settings for GA drawings

In GA drawings, automatic protection settings can only be defined on drawing level.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type. This time, select **GA drawing**.
2. Load drawing properties that are as close to the ones you need as possible.
3. Click **Protection**.
4. Select check boxes to indicate the areas that you want to protect from text, marks, weld marks, associative notes, dimension arrowheads, dimension lines or dimension values.
   - For example, to prevent text, marks, dimensions or other annotation objects from being placed over the inside area of parts, select the third check box on the first row. The first row defines the placement of annotation objects, and the icon above the third check box indicates the inside area of parts. With this setting, Tekla Structures may place annotation objects on the corners and edges of parts.



- If you clear the first two columns of check boxes, Tekla Structures does not protect part corners and edges. This increases drawing speed and decreases the memory requirements. You can still protect these areas by using the third column of check boxes for part protection.



5. Click **Save as** and save the protection settings with a unique name, and **OK** to return to drawing properties.
6. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

Modify protection settings in an existing drawing on view level

You can modify the protection settings on view level in all drawing types.

1. When the drawing is open, click the view frame of the view where you want to have different protection settings.
2. Click **Protection** in the options tree.

   The settings are same for all drawing types.
3. Adjust the settings as necessary and click **Modify**.

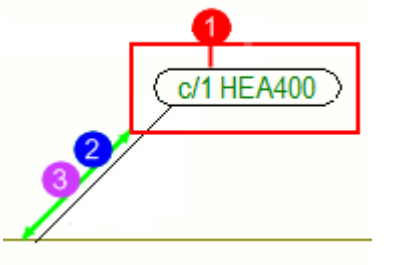
In GA drawings, you can also adjust the drawing level protection settings. To do so, double-click the drawing background, click **Protection**, adjust the settings and click **Modify**.

Define placement settings for marks

You can define automatic placement settings for the marks before creating a drawing. You can modify the settings in an open drawing.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.

3. Single-part, assembly and cast unit drawings: Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. Click an object that you want to adjust, for example **Part mark**.
5. On the appropriate tab, click the **Place...** button to open the **Placing** dialog box.
6. Adjust the placement settings. The options available in the dialog box vary depending on the annotation object type.
   - In **Minimum distance**, enter the closest distance Tekla Structures uses to place the marks. See number 2 in the image below.
   - In **Maximum distance**, enter the farthest distance Tekla Structures uses to place the marks. If no place for mark is found within the specified distance, Tekla Structures will force the mark to that distance. If you set this option to 0, the maximum distance behaves like infinity. See number 3 in the image below.
   - In **Search margin**, enter the empty margin that you want to have around the marks. See number 1 in the image below.
   - Note that if you use a high **Search margin** and **Minimum distance** values, the mark placement does not work properly.
   - To move marks away from each other, use **Search margin**, not **Minimum distance**. Keep the minimum distance setting as small as possible to reduce the possibility of marks overlapping.
   - Select the **Quarter** to define the areas Tekla Structures searches for a space to place the object.



   (1) Search margin

   (2) Minimum distance

   (3) Maximum distance
7. Click **OK**.
8. Single-part, assembly and cast unit drawings: Click **Save** to save the changes in view properties and **Close** to return to drawing properties.
9. General arrangement drawings: Click **OK**.
10. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

TIP If you are not happy with the mark placement settings in a drawing, you can change the settings in the mark properties on the object level (page 972), which has more options available. In an open drawing, you can also change the placement settings for other manually added annotation objects, such as notes, texts, and symbols.

To open the object level properties, double-click the mark, text, note, or symbol. If you use the free placing method , you let Tekla Structures to decide the location of the annotation object. If you use the fixed placing method, you can place the annotation object in any location, and the annotation object stays where you place it even though you update the drawing. With free placing, Tekla Structures tries to find the optimal place for the annotation object. You can select several or all annotation objects in a view and change the placing settings this way.

See also

Define object protection and placement settings in drawings (page 677)

Protect areas in a drawing (page 679)

Define placement settings for dimensions

You can set dimension placement to free or fixed, and control the space between parallel dimension lines and the location of the dimension in relation to the object it is dimensioning. You can also select whether to place short dimension text inside or outside the dimension.

In single-part, assembly and cast unit drawings, the dimension placement settings can be defined in an open drawing, saved into a dimension properties file and loaded in another drawing, or taken into use in the object level properties. In general arrangement drawings, most of the settings can be defined on the drawing and view level, but placement settings on the object level only.

Remember that in addition to dimension placement settings, the placement is also affected by **Protection** settings. Tekla Structures uses the protection settings (page 679) to prevent marks and dimensions from being placed in protected areas.

To adjust dimension placement settings in an open drawing and save the settings for later use:

1. Click a dimension in the drawing. If the property pane is not open, double-click the dimension.
2. Go to the **Placing** section and enter the desired space between two parallel dimension lines in **Dimension lines spacing**.
3. In the **Short dimensions** list, select whether to place short dimension text inside or outside of the dimensions.

4. Define the **Placing method**:
   - **Free** lets Tekla Structures to decide the location and direction of the dimension based on the direction settings.
   - Use **Fixed** to place the dimension in any location. When you use the setting **Fixed**, the dimension stays where you place it even though you update the drawing, whereas with **Free**, Tekla Structures tries to find the optimal place for the dimension.
   - The option direction options define where Tekla Structures places dimensions relative to the dimensioned object. You can select either **Negative direction** or **Positive direction**, or both. **Positive direction** places the dimension further away from and **Negative direction** closer to the dimensioned object. This setting affects the **Free** setting as well.
5. In **Distance s**, enter the empty margin that you want to have around the dimension. If Tekla Structures cannot place the dimension at the minimum distance, it moves the dimension by the value you enter in the **Distance s** box. Tekla Structures tries to place the dimension using the **Distance s** value until it finds a place for the dimension.
6. In **Distance d min**, enter the closest distance Tekla Structures uses to place the dimension.
7. Save the dimension properties using **Save** . You can also give the dimension properties another name and save in another file.
8. Click **Modify** to modify the selected dimension properties in the open drawing.

The dimensions are placed according the changes you made. Now you have a dimension properties file that you can load whenever you need to adjust the dimension placement settings in the same way. For example, you can load these properties while adjusting **Dimensioning rule properties**, or in an open general arrangement drawing in the **Dimension Properties** dialog box.

See also

Define object protection and placement settings in drawings (page 677)

Dimension properties (page 932)

Placement properties for marks, notes, dimensions, texts and symbols in drawings (page 1033)

Define automatic free or fixed placement for drawing views

You can keep the views in the same location (fixed) or let Tekla Structures find a suitable place for the view (free) during drawing updates.

In general arrangement drawings, this setting can only be defined on view level in an open drawing. In single-part, assembly and cast unit drawings, you can set the view placement before creating drawings, and in an open drawing.

To define automatic free or fixed placement of in single-part, assembly and cast unit drawings:

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. On the **Attributes 1** tab, select one of the following options:
   - Set **Place** to **fixed** to always keep the view in the same location when you update the drawing.
   - Set **Place** to **free** to let Tekla Structures find a suitable place for the view when you update the drawing.
5. Click **Save** to save the view properties.
6. Click **Close** to return to drawing properties.
7. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

NOTE The Arrange views (page 200) command only affects views where the **Place** is set to **free**. **fixed** views are not moved.

See also

Define object protection and placement settings in drawings (page 677)

8.4 Define drawing views

With automatic drawing views we mean views that you select to create before creating a single-part, assembly or cast unit drawing. You can define the desired drawing view properties separately for each view before you create the drawings.

When you create general arrangement drawings, you cannot select the views to be created in the general arrangement properties dialog box, but you select them at GA drawing creation. However, you can define automatic settings that apply to all the views you create in a general arrangement drawing.

To	Click the links below to find out more:
Select the views that you want to create in single-part, assembly or cast unit drawings	Define the views to create in single-part, assembly and cast unit drawings (page 689)
Define automatic view properties for general arrangement drawings	Define automatic view settings for general arrangement drawings (page 690)
Define automatic view properties for section views	Define automatic section view properties (page 717)
Define the contents of the main view and section view labels before you create the drawing	Define view labels and view label marks (page 691)
Define how Tekla Structures places the projections of a part in cast unit, single-part, and assembly drawings	Set drawing view projection type (page 693)
Include single-part drawings of the individual parts that make the assembly in assembly drawings	Include single-part drawings in assembly drawings (page 695)
Adjust the orientation of parts by changing the coordinate system, rotating parts in drawing views, setting viewing direction of columns, beams or bracings, or changing the project north	Part orientation in drawing views (page 697)
Show neighbor parts in drawing views	Show neighbor parts in drawings (page 705)
Shorten or lengthen parts in model or drawing views	Shorten or lengthen parts (page 708)
Unfold polybeams and bent plates in single-part drawings according to unfolding parameters	Unfold polybeams in drawings (page 712)
Undeform warped or cambered parts and show the developed shape of deformed parts in drawings	Undeform deformed parts in drawings (page 713)
Show symbols for part openings and recesses (blind holes) in drawing views	Show part openings and recesses in drawings (page 715)

See also

View properties in drawings (page 912)

Section view properties (page 929)

Define the views to create in single-part, assembly and cast unit drawings

Before you create single-part, assembly or cast unit drawings, you need to select the views that you want to include automatically. You can select to create some or all of the main views, section views, end views, and 3D views. At the same time, you can set the necessary view properties.

To select the drawing views to be created and set the view properties:

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.. Select single-part, cast-unit or assembly drawing.
2. Load the drawing properties file that you want to modify from the list at the top.
3. Click **View creation**.
4. Go to the **Attributes** tab and change the settings as required.

   These settings apply to all views in the drawing. Here you can select the coordinate system, set the coordinate system rotation, and undeform warped or cambered parts.
5. On the **Views** tab, select the views you want to create. You can create as many views as you like.
   - If you select **Off**, Tekla Structures does not create the view, but dimensions the parts in the available views. If you set all four main views off, Tekla Structures will still create one front view.
   - If you select **On**, Tekla Structures always creates the view, even if it was not necessary in order to show the dimensions. For section views, Tekla Structures creates one additional section view showing the middle of the main part. For end views, Tekla Structures creates an end view from one end of the main part.
   - If you select **Auto**, Tekla Structures creates the view if it is necessary in order to show the dimensions. For section views, Tekla Structures creates the necessary number of views to show all the dimensions. For end views, Tekla Structures also creates another end view from the other end of the main part, if there are dimensions at that end.
6. For each of the views that you create, select the view properties that you want to use in the **View properties** column.

   Note that if you have selected **Auto** in the **on/off** column as the creation method, Tekla Structures will use the standard view properties even if you define another view property file here.

   The lists contain predefined view properties for different types of drawings, also the view properties that you save in the **View Properties** dialog box. For more information about view properties, see View properties in drawings (page 912).

7. Check the view properties for each view by selecting the view from the list and clicking **View properties** button, and modify the properties as required.
8. Check the settings on the **Attributes 1** tab in **View Properties**.

   Here you can set the view scale and size, view extension distance, view place, and rotation of 3D views, show a reflected view and apply detailed object level settings for the selected view.
9. Check the settings on the **Attributes 2** tab in **View Properties**.

   Here you can undeform warped or cambered parts, shorten parts, show openings and recesses, select whether to se the location by the model origin or a base point, set the datum point for elevations and select the dimension creation method in the selected view.
10. Check the settings on the **Label** tab in **View Properties**.

    Here you can define the label text and position, add a symbol in the label and show the view direction marks in the selected view.
11. Go throug the options in the options tree, and modify dimensioning, protection, mark and building object settings as required.
12. Save the view properties by clicking **Save**.
13. Click **Close**.
14. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

See also

View properties in drawings (page 912)

Section view properties (page 929)

Define drawing views (page 687)

Set automatic drawing properties before creating drawings (page 74)

Define automatic view settings for general arrangement drawings

Before you create general arrangement drawings, define the automatic view properties.

1. Click **Drawings & reports** --> **Drawing properties** --> **GA drawing** .
2. Load the drawing properties file that you want to modify from the list at the top.
3. Click **View...** and load the view properties you want to modify.

4. On the **Attributes** tab, change the settings as required.

   Here you can set the view scale, view extension distance, show a reflected view, show openings and recesses, set the datum point for elevations and show pours.

5. Go to the **Shortening** tab and define the part shortening settings.

   Here you can select whether you want to cut parts, set the minimum part length, and set the space between cut parts.

6. Go to the **Label** tab and define the view label text, symbol and position.
7. If you want to create an anchor bolt plan drawing, go to the **Anchor bolt plan** tab and set **Show as anchor bolt plan** to **Yes**.

   You can also select, whether you want to create detail views in anchor bolt plans and set the enlarged part view scale.

8. Save the view properties.
9. Click **OK** to return to drawing properties.
10. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

See also

View properties in drawings (page 912)

Define drawing views (page 687)

Create anchor bolt plans using saved settings (page 127)

Set automatic drawing properties before creating drawings (page 74)

Define view labels and view label marks

All drawing views can have view labels, which may contain text and symbols. You can set the contents of the main view labels and section view labels before you create the drawing. You can also adjust the content in an open drawing.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Do one of the following depending on the drawing type:

   Single-part, assembly and cast unit drawings:

   a. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.

b. Click **Attributes** and go to the **Label** tab.

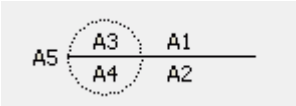
General arrangement drawings:

a. Click **View...**

b. Go to the **Label** tab.

4. Click the **...** button next to **A1** - **A5** to open the **Mark Contents** dialog box.

   The illustration in the view properties dialog box is only one way to position the label text. When you modify the positioning, the illustration in the dialog box does not change.



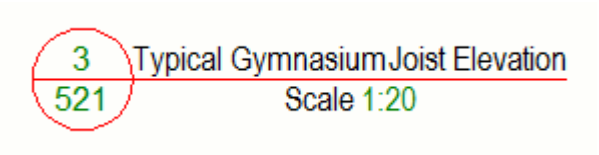
5. On the **Content** tab, select the elements that you want to include in the view label mark.
6. If needed, select an element from the list and click **< Add frame** and select the frame **Type** and **Color**.
7. If needed, select an element from the list and select the text **Color**, **Font** and **Height**.
8. Go to the **Position** tab and set the text position, horizontal and vertical offset, and the text alignment.

   Text positioning depends on whether you use a symbol or not.
9. Click **OK**.
10. Select the view label **Symbol** you want to use in the label.

    You can simply use only a label or add a symbol in it. You can also set the color, size, line length and the position of the view label.
11. Select the **Vertical** and **Horizontal** position for the view label.
12. To save the changes, click **Save**.
13. **Single-part, assembly and cast unit drawings:** Click **Close**.

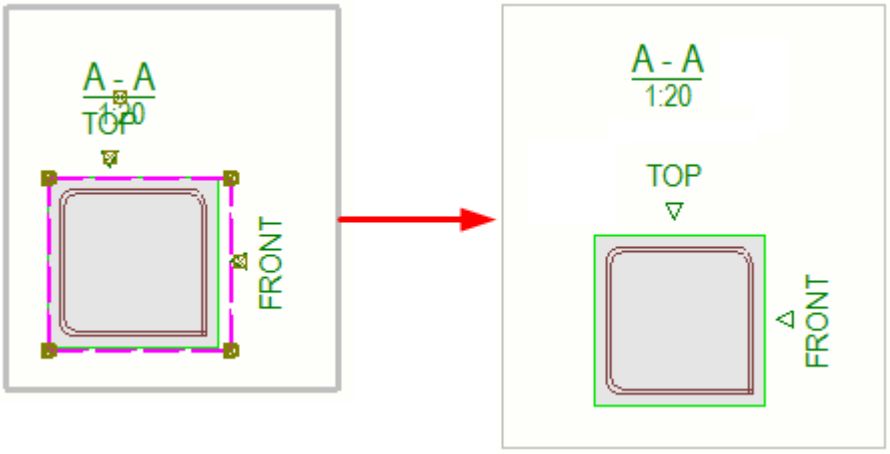
    General arrangement drawings: Click **OK**.
14. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

See below for examples of view labels:



Drag the view label to the desired page in an open drawing.

The view frame is resized automatically, if necessary.



For more information on the elements available in view label marks, see View, section view and detail view label mark elements (page 981).

For more information about the positioning of the view label marks, see View properties in drawings (page 912).

Define section view properties for all views in a drawing

If you want to use the same section view properties, such as start number or letter, cutting line, contents, and text position, in all your section views, you can define the properties on the drawing level. For more information, see Define automatic section view properties (page 717).

See also

View properties in drawings (page 912)

Section view properties (page 929)

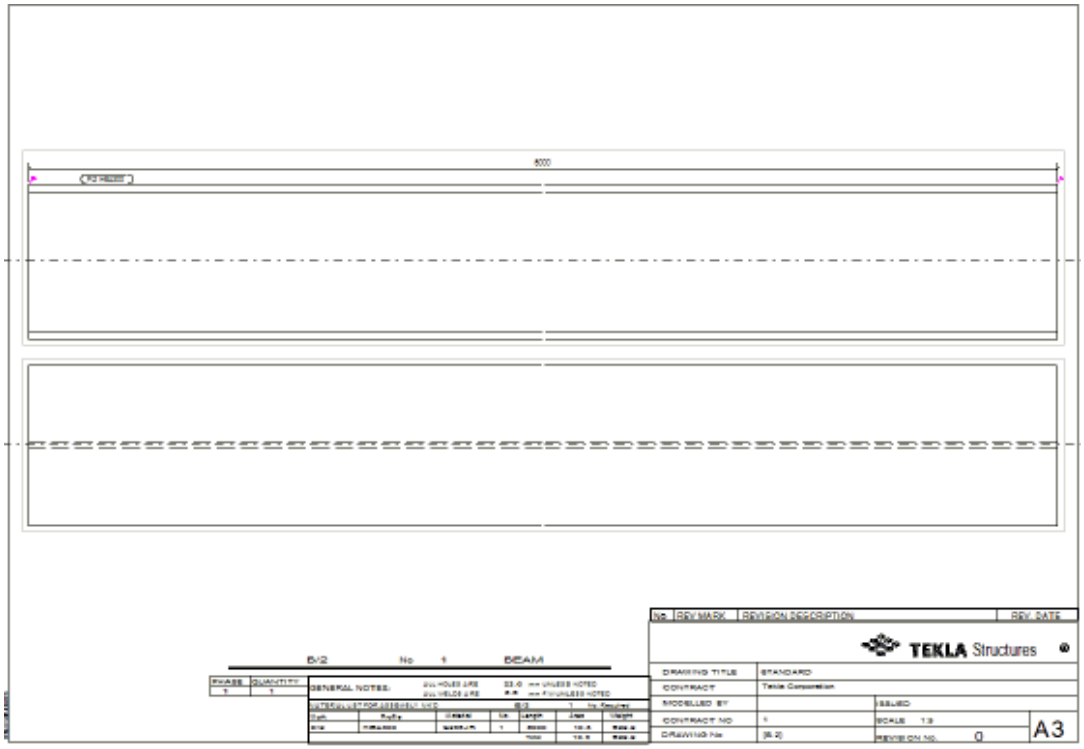
Define drawing views (page 687)

Set drawing view projection type

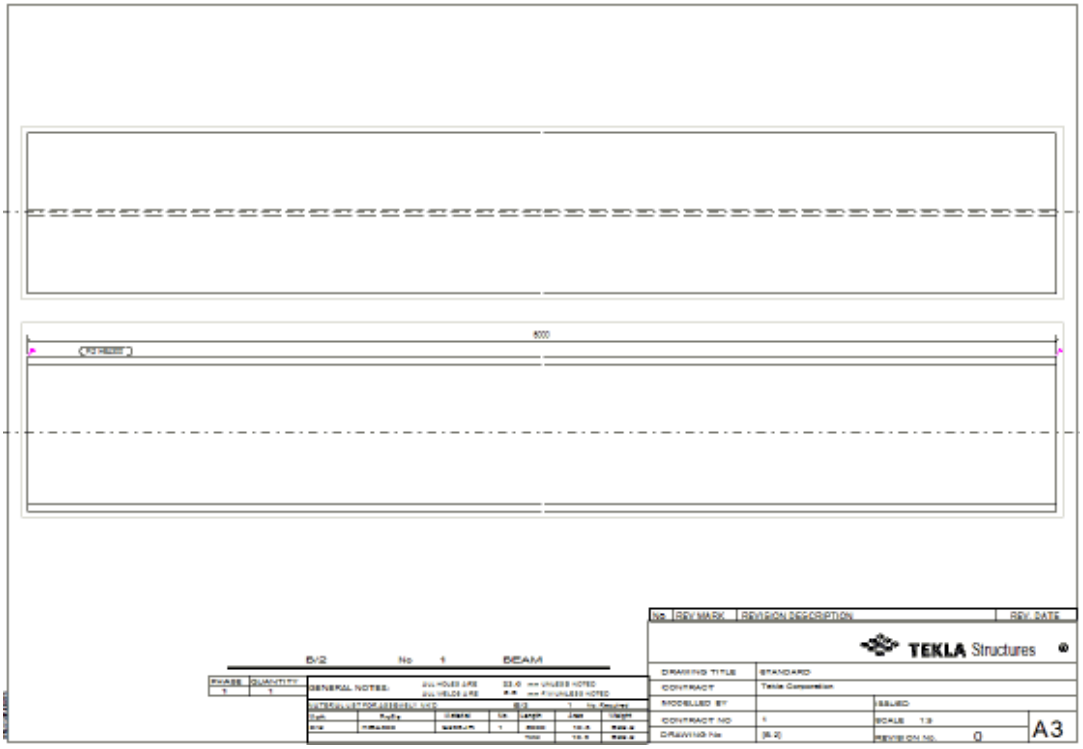
Projection type defines how Tekla Structures places the projections of a part in cast unit, single-part, and assembly drawings. Projection type affects the order of the views in the drawing.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load the properties you want to change.
3. Click **Layout** and go to the **Other** tab.
4. Select one of the following:
   - **First angle** (also referred to as European projection).
   - **Third angle** (also referred to as American projection).
5. To save the properties in the properties file, click **Save**.
6. Click **OK** and create the drawing.

First-angle projection:



Third-angle projection:



See also

Define drawing views (page 687)

View properties in drawings (page 912)

Layout properties (page 910)

Include single-part drawings in assembly drawings

You can include in assembly drawings single-part drawing views of the individual parts that make the assembly. You can use existing single-part drawings in **Document manager** or create new single-part drawing views.

1. On the **Drawings & reports** tab, click **Drawing properties** --> **Assembly drawing** .
2. Load the desired assembly drawing properties.
3. Click **Layout** and go to the **Other** tab.
4. Set **Include single-parts** to **Yes**.

   This activates the **Single-part attributes** list.
5. In the **Single-part attributes** list, select the desired drawing properties to be used in the single-part view. The properties file `standard` is the default.
6. Click **Save** to save the drawing properties in the properties file.

7. Click **OK** and create the drawing.

The value of the advanced option XS_USE_EXISTING_SINGLE_PART_DRAWINGS_IN_ASSEMBLY_DRAWING affects how Tekla Structures creates the single-part views. If the option is set to TRUE, Tekla Structures will use single-part drawings in the . If it is set to FALSE, or if there is no existing single-part drawing for a given part, a new view will be created according to the **Include single-parts** setting. The default value is FALSE.

Tekla Structures also maintains the original scale in a single part drawing in an assembly drawing when you set the layout to include single part drawings, and set the option XS_USE_EXISTING_SINGLE_PART_DRAWINGS_IN_ASSEMBLY_DRAWINGS is set to TRUE. If you do not want to maintain the scale of the existing single part drawing, set the advanced option XS_USE_EXISTING_SINGLE_PART_DRAWINGS_SCALE can be set to FALSE. If you do this, thee scale of the included single part drawing will follow the scale of the assembly drawing, or advanced option XS_SINGLE_SCALE if it is set.

The following advanced options all affect how the single-part drawing views behave:

XS_SINGLE_CENTERED_SCREW

XS_SINGLE_CLOSE_DIMENSIONS

XS_SINGLE_CLOSE_SHORT_DIMENSIONS

XS_SINGLE_COMBINE_DISTANCE

XS_SINGLE_COMBINE_MIN_DISTANCE

XS_SINGLE_COMBINE_WAY

XS_SINGLE_DIMENSION_TYPE

XS_SINGLE_DRAW_PART_AS

XS_SINGLE_EXCLUDE

XS_SINGLE_FORWARD_OFFSET

XS_SINGLE_NO_SHORTEN

XS_SINGLE_ORIENTATION_MARK

XS_SINGLE_PART_EXTREMA

XS_SINGLE_PART_SHAPE

XS_SINGLE_SCALE

XS_SINGLE_SCREW_INTERNAL

XS_SINGLE_SCREW_POSITIONS

XS_SINGLE_USE_WORKING_POINTS

XS_SINGLE_X_DIMENSION_TYPE

XS_USE_EXISTING_SINGLE_PART_DRAWINGS_SCALE

XS_NO_END_VIEWS_TO_INCLUDED_SINGLE_DRAWINGS

See also

Add single-part views in assembly drawings (page 206)

Define drawing views (page 687)

Define part orientation in drawing views

In single-part, assembly, and cast unit drawings, you can adjust the orientation of the parts in the drawing views by selecting the appropriate coordinate system and by rotating the parts. You can also separately set the viewing direction for columns, beams, and bracings in assembly drawings. The project north setting also affects part orientation.

You can:

- change the angle from which the part, assembly, or cast unit is viewed, how the part, assembly, or cast unit is rotated, and how the dimensions in the drawing view are oriented
- Rotate a part, assembly, or cast unit in a drawing view around its local axes.
- Select which side of a steel or timber part is always shown in the drawing main view.
- Define the front view direction separately for columns, beams, and bracings.
- Change the plate orientation in drawing views.

There are also many ways available for indicating the part orientation in drawings. For more information, see Indicate part orientation (page 857).

Change the coordinate system

You change the angle from which objects are viewed, how the object is rotated, and how the dimensions in the drawing view are oriented.

The coordinate system defines:

- The angle from which the part, assembly, or cast unit is viewed.
- How the part, assembly, or cast unit is rotated.
- The orientation of the dimensions in the drawing view.

NOTE Plate parts and assemblies whose main part is a plate do not follow these rules exactly but the rules defined in the Change plate

orientation in drawings section. The plate orientation affects assembly drawings too.

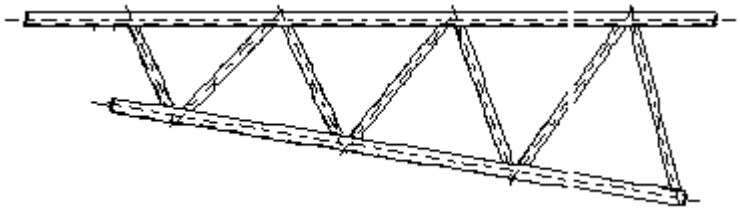
To change the coordinate system:

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load the drawing properties that you want to change.
3. Click **View creation** and go to the **Attributes** tab.

   The settings affect all views in a drawing.
4. In **Coordinate system**, select one of the available coordinate systems:

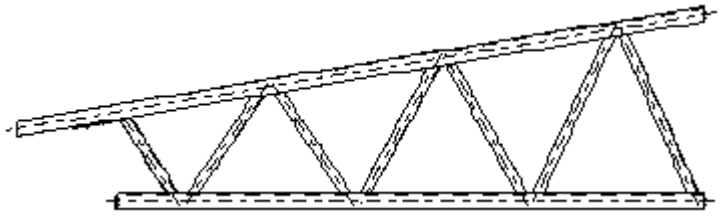
   - **local**

     Tekla Structures uses the local coordinate system of the main part. The x axis of the part is parallel to the x axis of the drawing, and the start point (the end point created first) of the part is on the left. The start point is marked with yellow, and the end point created second is marked with pink.

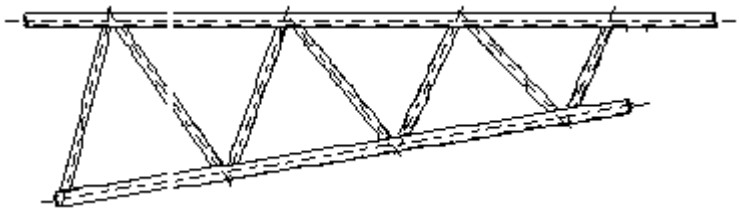
   - **model**

     Tekla Structures uses the global coordinate system. The part has the same position in the drawing as it has in the model. This is one option when you want to show columns vertically. You can also use this option to display sloping parts in position. Tekla Structures cannot display horizontally skewed parts.

   - **oriented**

     Tekla Structures uses the local coordinate system of the main part, but the coordinate system is oriented so that the x axis of the part points to the right even if the part was created from right to left.

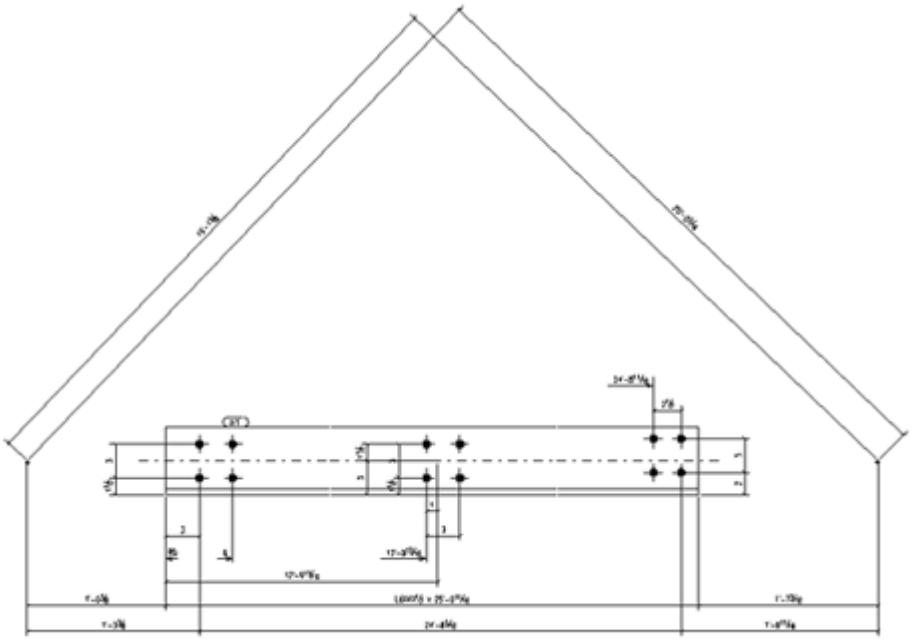


- **horizontal brace**

  Tekla Structures automatically rotates the drawing views so that the front view is from the top of the model. This is used for skewed braces. It automatically rotates the front view around the x axis.

- **vertical brace**

  Tekla Structures automatically rotates the drawing views so that the front view is in the same plane as the brace in the model. This is used for skewed braces. It automatically rotates the front view around the x axis.



- For concrete parts, the option **Fixed** rotates the front view so that it shows the casting direction (the face that is top in form) of the concrete part, if it is defined in the model.

5. To save the changes, click **Save**.
6. Click **OK** and create the drawing.

Rotate parts in drawing views

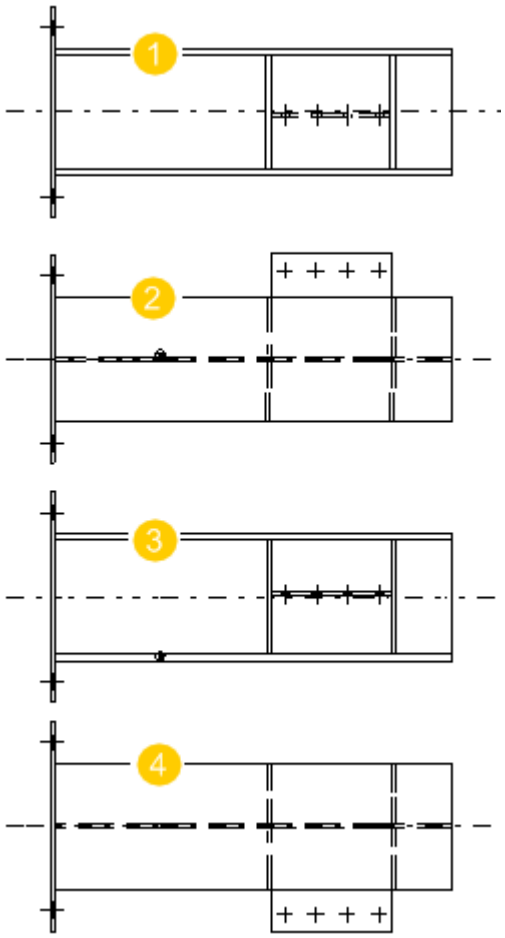
You can rotate a part, assembly, or cast unit in a drawing view around its local axes.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load the drawing properties that you want to change.
3. Click **View creation** and go to the **Attributes** tab.

   The settings affect all views in a drawing.
4. In **Rotate coordinate system**, specify the angle:
   - With the **Around X**, you can rotate in steps of 90 degrees (**0**, **90**, **180**, **270**).
   - With the **Around Y**, you can rotate in steps of 180 degrees (**0**, **180**).
   - With the **Around Z**, you can specify any angle.
5. To save the changes, click **Save**.
6. Click **OK** and create the drawing.

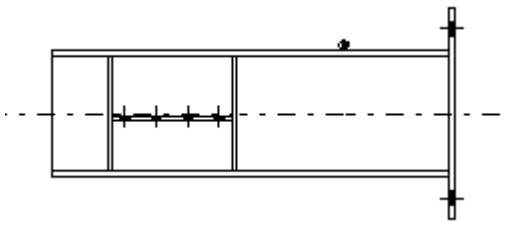
Examples

Below there are some examples of rotating an object around the x axis:

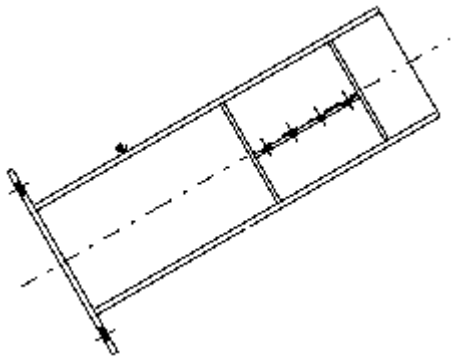


1. 0 degrees
2. 90 degrees
3. 180 degrees
4. 270 degrees

Below is an example of rotating the same part 180 degrees around the y axis:



Below is an example of rotating the same part 30 degrees around the z axis:



Select the steel or timber part face that is shown in front drawing view

You can select which face of a steel or timber part is shown in the drawing main (front) view by using the user-defined attribute **Fixed drawing main view**.

The user-defined attribute **Fixed drawing main view** controls the drawing coordinate system for steel and timber parts. This UDA is only taken into account when you use the coordinate system **Fixed** in drawing properties. When the fixed coordinate system is used, the part is rotated so that the front view shows the part face that has been selected with the **Fixed drawing main view** UDA.

1. In the model, double-click a steel or timber part to open part properties, and click the **User-defined attributes** button.
2. On the **Parameters** tab, click **Fixed drawing main view**, and select one of the options:
   - **Top**
   - **Back**
   - **Bottom**
   - **Start**
   - **End**
   - **Front**
3. Click **Drawings & reports** --> **Drawing properties**, and select assembly or single part drawing properties.
4. Click **View creation** in the options tree, go to the **Attributes** tab, and set **Coordinate system** to **Fixed**.
5. Click **OK** to activate the settings, and create the drawing using the current settings.

NOTE If you set the advanced option `XS_SET_FIXEDMAINVIEW_UDA_TO_AFFECT_NUMBERING` to `STEEL`, `TIMBER` or `MISC`, and if identical steel, timber or miscellaneous material parts have

different options selected for **Fixed drawing main view**, they get different assembly position numbers.

If you set this advanced option, the **Top in form face** command is available also for non-concrete material in the model.

Set viewing direction for columns in assembly drawings

In assembly drawings, you can define the front view viewing direction separately for columns.

NOTE Do not change the viewing direction settings in the middle of the project. If you change the settings, some drawings may disappear.

1. On the **File menu**, click **Settings** --> **Options** and go to the **Orientation marks** settings.
2. Under **Viewing direction**, use the **Columns in assembly drawing** option to set the front view direction of columns:
   - The values are **As beam and bracing**, **North**, **East**, **South** and **West**. Select **As beam and bracing** to use the same viewing direction as you use for beams and bracings. This is the default value.
   - If you have set the coordinate system to **local** in **View creation** properties, Tekla Structures uses the coordinate system of the column when setting the viewing direction of the front view.
   - If you have set the coordinate system to **oriented**, the column is in a horizontal position, and the viewing direction of the front view is the option you select (**North**, **East**, **South** or **West**).
   - If you have set the coordinate system to **model**, the column is in a vertical position, and the viewing direction of the front view is the option you select (**North**,**East**, **South** or **West**).
3. Click **OK**.

Set viewing direction for beams and bracings in assembly drawings

In assembly drawings, you can define the front view viewing direction separately for beams and bracings.

NOTE Do not change the viewing direction settings in the middle of the project. If you change the settings, some drawings may disappear.

1. On the **File menu**, click **Settings** --> **Options** and go to the **Orientation marks** settings.
2. Under **Viewing direction**, use the **Beams and bracings in assembly drawing** option to set the front view direction:

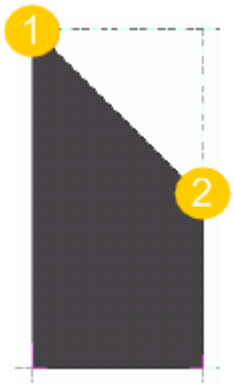
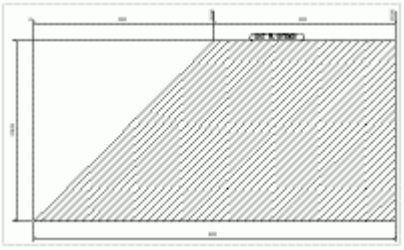
- The values are **North or east**, **North or west**, **South or east**, and **South or west**. The default value is **North or east**.
- If the beam or bracing is parallel to the X axis of the model, it is also parallel to the X axis in the drawing.
- If you have set the coordinate system to **model**, and the beam or bracing is sloped, it is also sloped in the drawing.

3. Click **OK**.

Change plate orientation in drawings

Plates created with the **Plate** command are automatically oriented in drawings. The longest side of the plate always faces downwards in the drawing. You can affect this orientation in single-part drawings and assembly drawings.

Automatic plate orientation:

Example	Description
	Contour plate in the model view: 1. First creation point 2. Second creation point
	The same contour plate in a single-part drawing

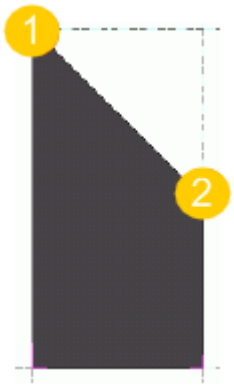
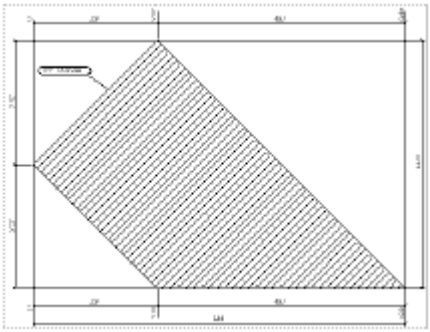
Instead of using automatic plate orientation, you can set the plate main axis to follow the line created by the first and second points you pick, regardless of the plate dimensions. This enables you to define the plate orientation in drawings or reports.

To define the contour plate orientation with first and second picked points:

1. Create the contour plate.

   The first and second points you pick also define the plate's main axis.

2. Double-click the plate to open the contour plate properties.
3. Click **User-defined attributes**, and click the **Orientation** tab.
4. Select **From 1st to 2nd creation point** in the **Main axis direction** list.
5. Click **Modify**, and close the dialog box.
6. Click **Drawings & reports** --> **Perform numbering** --> **Number modified objects** to update numbering.
7. To view the orientation of the plate, create a single-part drawing of the plate.

Example	Description
	Contour plate in the model view: 1. First creation point 2. Second creation point
	Single-part drawing of the plate. The user-defined attribute **Main axis direction** is set to **From 1st to 2nd creation point**.

NOTE You can also affect the orientation of the plates using the advanced options XS_POLYGON_SQUARE_CORNER_PREFERENCE_FACTOR and XS_POLYGON_PERPENDICULAR_EDGE_PREFERENCE_FACTOR.

Show neighbor parts in drawings

You can select which neighbor parts to show in drawings and also automatically extend the view boundary if necessary.

Neighbor parts refer to (optionally shown) parts that are close to the part that is depicted in a drawing. Depending on settings, the neighbor parts may be

parts somehow connected to the part in question, or just parts that happen to be close by.

For more information about neighbor part properties, see Part and neighbor part properties in drawings (page 1004).

TIP If you do not want to see neighbor part extensions in drawing views, set the advanced option `XS_VISUALIZE_VIEW_NEIGHBOR_PART_EXTENSION` to `FALSE`.

Show neighbor parts in drawing views

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. Click **Neighbor part...**
5. On the **Visibility** tab, select the parts that you want to show using the following options:

   Neighbor parts:

   - **None** does not show neighbor parts.
   - **Connected parts** shows all parts connected to the model object.
   - **Connecting parts** shows only the parts the model object is connected to.
   - **All components** combines the **Connected parts** and the **Connecting parts** options.
   - **By extreme** show all parts within the boundaries of the main and secondary part. This setting is affected by the value entered for **View extension for neighbor parts** on the **Attributes 1** tab.

   Main/Secondary parts:

   - **Main parts** shows only neighbor parts that make the main part of an assembly or a cast unit.
   - **Secondary parts** shows only neighbor parts that are secondary parts of an assembly or a cast unit.
   - **Both** shows both main and secondary parts of an assembly or a cast unit.

   Skew parts:

   - **Yes** shows skew parts as neighbor parts in the drawing.
   - **No** does not show skew parts as neighbor parts.

Bolts:

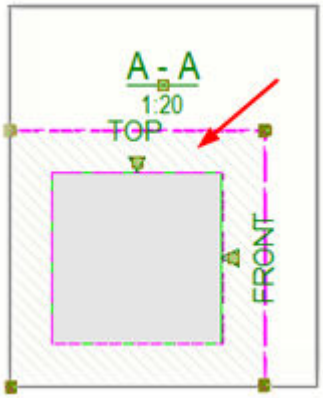
- **Yes** shows the bolts in the neighbor parts.
- **No** does not show the neighbor part bolts.

6. On the **Content** tab, set the representation of the neighbor part and neighbor part bolts, and which lines and marks are shown
7. On the **Appearance** tab, set the colors and line types you want to use in neighbor parts.
8. Click **Attributes** in the options tree and enter a value by which to extend the view in the **View extension for neighbor parts** box.

   Try different values and check which one suits your needs. Often too big values do not work very well. If you set the value to `0`, neighbor part extension is not shown.

   For more information about view properties, see View properties in drawings (page 912).
9. To save the changes, click **Save**.
10. Click **Close** to return to drawing properties.
11. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

In the following example, **View extension for neighbor parts** is set to `100`. No neighbor parts are located in this area.



Show neighbor parts in general arrangement drawings

In GA drawings, you need to define neighbor parts using neighbor part filters because neighbor parts are not automatically detected. The parts that fulfill the filtering criteria will be treated as neighbor parts. You must also define a filter for normal parts to get the neighbor parts working.

1. On the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing** .

2. Load drawing properties that are as close to the ones you need as possible.
3. Click **Filter...** to go to the **General - Filter Properties** dialog box, and create a filter by **Part: Name** and **Object: Object type** for all parts that should be treated as normal parts. Then click **OK**.
4. Click **Part...** to go to the **General - Part Properties** dialog box, and on the **Content**, **Appearance**, and **Fill** tabs, define the needed part properties. For example, select a fill that shows the normal parts in a different way than the neighbor parts. Then **OK**.
5. Click **Neighbor part filter...** to go to the **General - Neighbor Part Filter Properties** dialog box, and create a filter by **Part: Name** and **Object: Object type** for all parts that should be treated as neighbor parts. Then click **OK**.
6. Click **Neighbor part...** to go to the **General - Neighbor Part Properties** dialog box, and on the **Visibility** tab, select **By extreme** to show all parts within the boundaries of the main and secondary part.

   This setting is affected by the value entered for **View extension for neighbor parts** on the **Attributes** tab in view properties. **None** does not show neighbor parts.
7. Still in the **General - Neighbor Part Properties**, on the **Content**, **Appearance**, and **Fill** tabs, define the needed part properties. For example, select a fill that shows the neighbor parts in a different way than the normal parts. Then click **OK**.
8. Click **View...**, and on the **Attributes** tab, enter a value by which to extend the view in the **View extension for neighbor parts** box.

   Try different values and check which one suits your needs. Often too big values do not work very well. If you set the value to `0`, neighbor part extension is not shown.

   For more information about view properties, see View properties in drawings (page 912).
9. Click **OK** to return to drawing properties.
10. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

The neighbor parts are shown with a different color in the GA drawings. Note that you can also do the same in an open drawing by modifying the drawing properties (page 438).

Shorten or lengthen parts

You can use the shortening functionality in the model to make the part longer or shorter in the drawing than in the model. You can also shorten or lengthen parts in drawing views.

Lengthening can be useful for adding length to concrete precast parts in the cast condition while the model stays in the erected state. The most common use for this is to account for pre-stressing elastic shortening, where the part actually shrinks a fraction of an inch after casting and after the strands are cut.

Shorten a part in the model

You can shorten parts in the model. When you do that the true length of the part is decreased in the drawing.

1. Double-click a part to open the part properties in the property pane.
2. Go to the **Deforming** tab.
3. In the **Shortening** box, define the degree of shortening.
4. Click **Modify**.

When drawings are created, Tekla Structures decreases the true length of the part by the value defined in the **Shortening** box. Shortening is applied linearly along the length in drawings.

TIP To show the dimensions of the shortened part correctly in the drawings, set **Undeformed** to **Yes** on the **Attributes** tab in the **View creation** panel in drawing properties. For more information about undeformed parts in drawings, see Undeform deformed parts in drawings (page 713).

Lengthen a part in the model

You can lengthen parts in the model. When you do this, the true length of the part is increased in the drawing.

To have a concrete part lengthened in cast unit drawings, you must enter a negative value for shortening in part properties.

1. Double-click a part to open the part properties in the property pane.
2. Go to the **Deforming** section.
3. In the **Shortening** box, enter a negative value.

   For example, `-20` would result in a part that is cast 20 units longer in the drawing than the part in the model.
4. Click **Modify**.

Shorten parts in drawing views

If parts are large and do not include any important details, you can shorten them in drawing views by cutting them.

Parts are only cut in empty areas. If there is something important, for example, a stiffener in the part, the part is not cut in that area, because that area is not considered empty.

You can also shorten parts view by view, see Shorten parts in selected views (page 427).

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. Go to the **Attributes 2** tab.
5. In **Cut parts**, select one of the following:
   - **Yes** to cut in both X and Y direction.
   - **Only in x direction**
   - **Only in y direction**
6. In **Minimum cut part length** and **Space between cut parts**, select how to cut the middle regions of the parts in drawing views.

   The **Minimum cut part length** defines how long the part must at least be to get shortened. The length of the part must be at least twice the entered value.

   The **Space between cut parts** defines the distance between cut parts on paper. For example, try 3.0 mm.
7. Set **Cut skew parts** to **Yes** to also cut skewed parts in views.

   Setting **Cut skew parts** to **No** cuts secondary parts that are skewed less than 5.7 degrees in relation to the main part.
8. To save the changes, click **Save**.
9. Click **Close**.
10. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

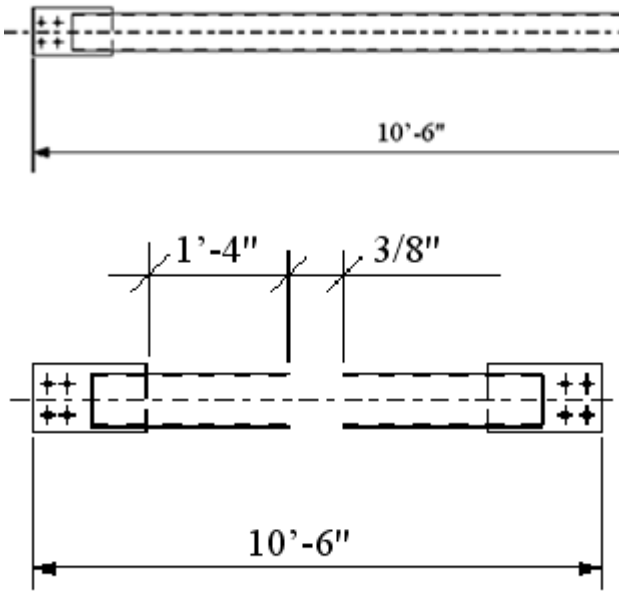
Related advanced options

- You can show view shortening symbols in drawings by setting the advanced options `XS_DRAW_VERTICAL_VIEW_SHORTENING_SYMBOLS_TO_PARTS` and `XS_DRAW_HORIZONTAL_VIEW_SHORTENING_SYMBOLS_TO_PARTS` to `TRUE` through **File menu** --> **Settings** --> **Advanced options** --> **Drawing properties** .

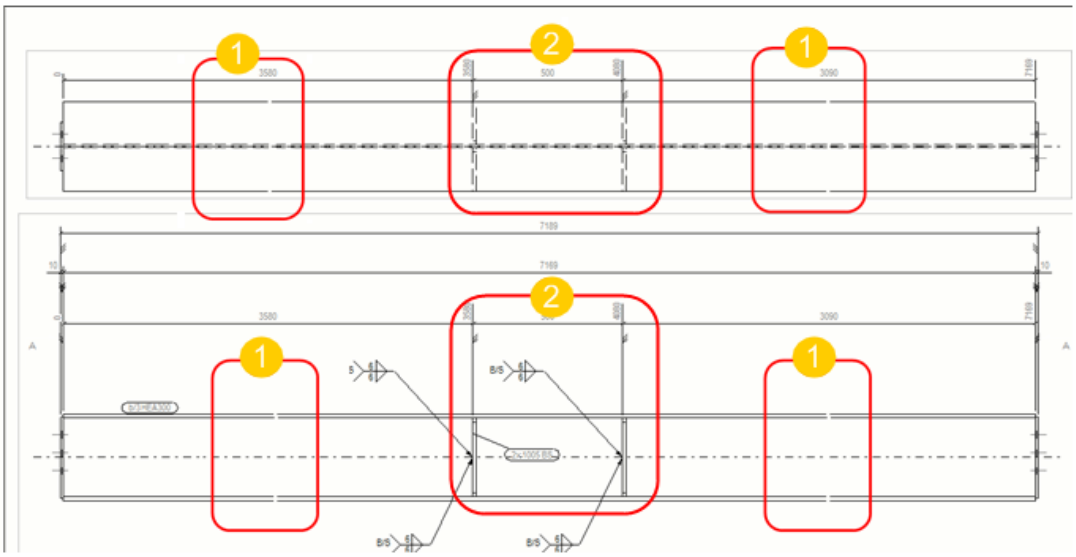
- You can also control the appearance of the view shortening symbol with the advanced options `XS_SHORTENING_SYMBOL_COLOR`, `XS_SHORTENING_SYMBOL_LINE_TYPE`, and `XS_SHORTENING_SYMBOL_WITH_ZIGZAG`.

Examples

Below is an example of a part before and after cutting. Note that the width is the same in the non-cut and cut part. **Minimum cut part length** is 1' 4" and cut length is 3/8".

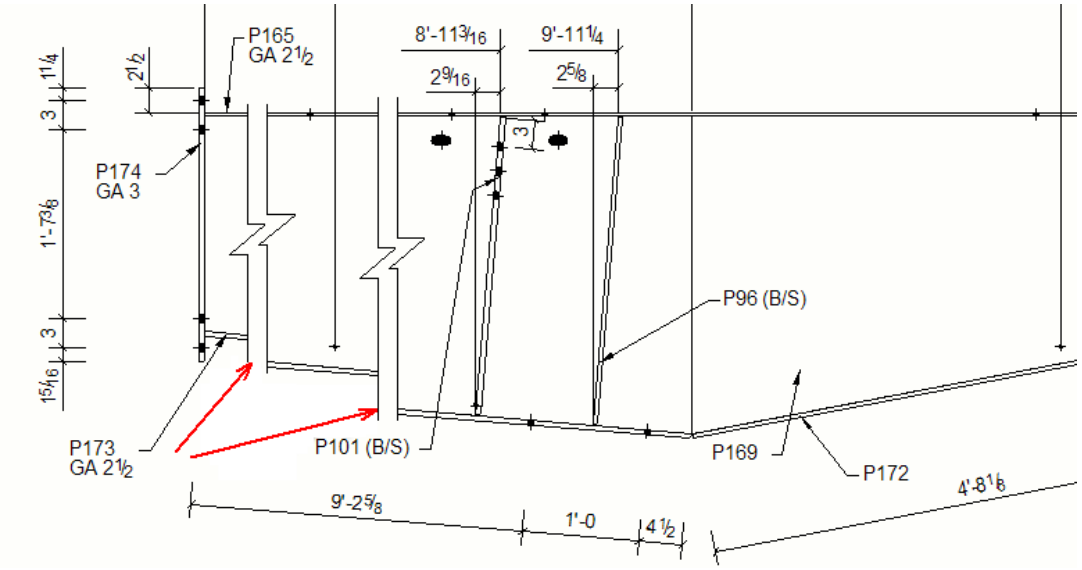


The following example describes the meaning of **Minimum cut part length**, **Space between cut parts**, and an area that is not considered to be empty in the part, and therefore the part is not cut. **Minimum cut part length** is set to 650, which means that the part is shortened in the view at 650.



1. **Space between cut parts** is set to 1. This means the distance between cut parts on paper (not in model).
2. There is not enough empty area between the stiffeners, and that is why the part is not shortened at **Minimum cut part length**.

Below is an example of using the advanced options `XS_DRAW_VERTICAL_VIEW_SHORTENING_SYMBOLS_TO_PARTS` and `XS_SHORTENING_SYMBOL_WITH_ZIGZAG`.



Lengthen shortened parts in drawing views

You can stretch shortened drawing views to fill up empty areas of the drawing .

After Tekla Structures has scaled drawing views and selected the drawing size, it may stretch shortened views to fill up empty areas of the drawing.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Click **Layout** and go to the **Other** tab.
4. Set **Expand shortened parts to fit** to **Yes**.
5. To save the changes, click **Save**.
6. Click **OK** and create the drawing.

Unfold polybeams in drawings

When you create a drawing, you can automatically unfold polybeams and bent plates in single-part drawings. Tekla Structures unfolds the polybeams

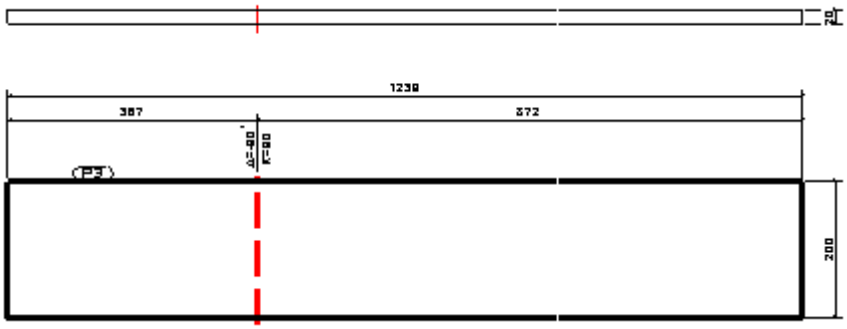
according to unfolding parameters, which define the location of the neutral axis when a profile is unfolded.

Limitations:

- You can unfold only beams that have been created with the **Polybeam** command. You cannot unfold beams created with the **Curved beam** command.
- You can unfold a polybeam only on one plane.

1. On the **Drawings & reports** tab, click **Drawing properties** --> **Single-part drawing** .
2. Load drawing properties that are as close to the ones you need as possible.
3. Click **View creation** in the options tree and go to the **Attributes** tab.
4. Set **Unfolded** to **Yes**.
5. To save the changes, click **Save**.
6. Click **OK** and create the drawing.

   Tekla Structures unfolds the polybeam in the single-part drawing.



NOTE The **Unfolded** setting in the **View Properties** properties dialog box on the **Attributes 2** tab is ignored in drawing creation when you set this option on the **Attributes** tab of the **Single-part drawing properties** dialog box.

See also

View properties in drawings (page 912)

Undeform deformed parts in drawings

You can undeform warped or cambered parts and show the developed (undeformed) shape of deformed parts in drawings.

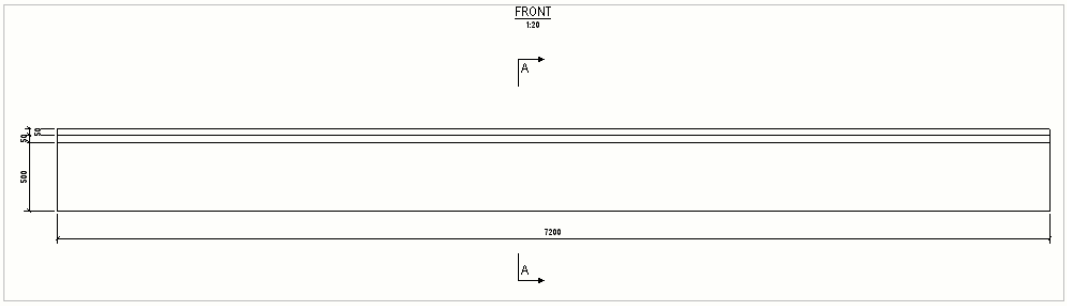
Deformed parts are parts that have been warped or cambered in the model. You may want to undeform these parts if you want a concrete part to have two states: as erected (in the model view) and as cast (in the drawing view), for example.

NOTE Part shortenings are hidden if you set **Undeformed** to **No**.

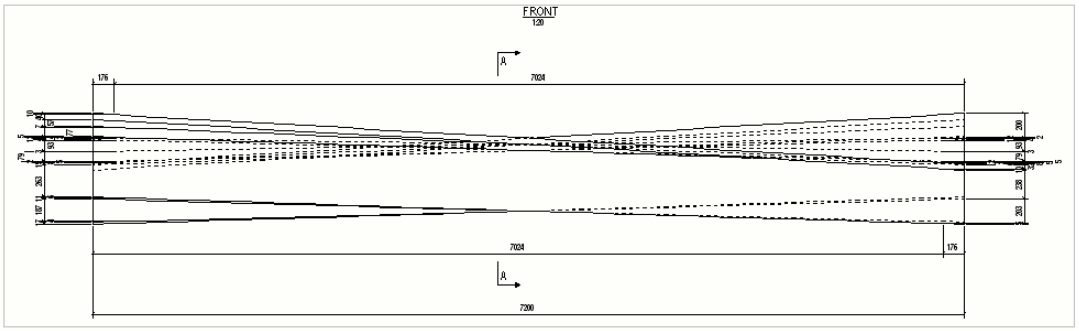
1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Click **View creation** and go to the **Attributes** tab.
4. To hide deforming angles and cambering, set **Undeformed** to **Yes**.
5. To save the changes, click **Save**.
6. Click **Close**.
7. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

The created drawing shows the developed shape and dimensions of the part.

See below for an example of a undeformed part in a drawing.



See below for an example of a warped part in a drawing.



NOTE The **Undeformed** option value in the **View Properties** properties dialog box on the **Attributes 2** tab is ignored in drawing creation when the **Undeformed** option is set to a value on the **View creation** --> **Attributes** tab.

See also

View properties in drawings (page 912)

Show part openings and recesses in drawings

You can select whether you want to show symbols for part openings and recesses (blind holes) in drawing views.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Depending on the drawing type, do one of the following:

   Single-part, assembly and cast unit drawings:

   a. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.

   b. Go to the **Attributes 2** tab.

   c. Set **Show openings/recess symbol** to **Yes**.

   d. Save the view properties and click **Close**.

   General arrangement drawings:

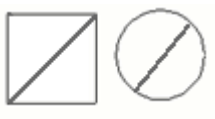
   a. Click **View...**

   b. On the **Attributes** tab, set **Show openings/recess symbol** to **Yes**.

   c. Click **OK**.
4. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

By default, Tekla Structures displays openings and recesses in the following way:

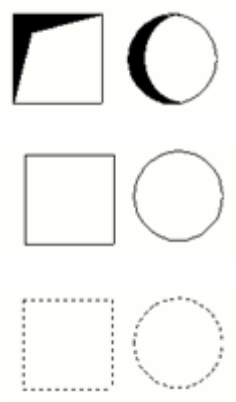
Type of opening	Shown as	Examples
Hole through a part	Hole symbol	

Type of opening	Shown as	Examples
Recess in the front face of a part	Recess symbol and bounding lines shown as unbroken lines	
Recess in the back face of a part	Recess symbol and bounding lines shown as dashed lines Remember to switch hidden lines on for parts.	

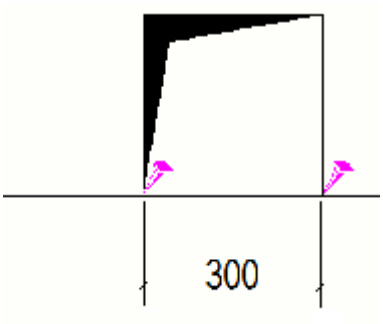
Add symbols in openings and recesses

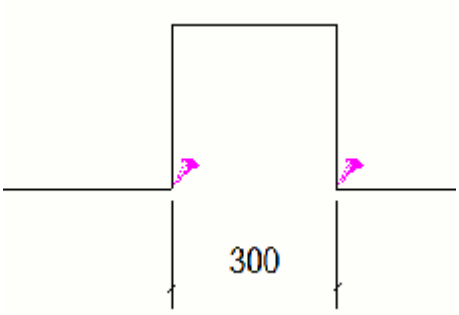
Tekla Structures has some advanced options that you can use for adding symbols in openings and recesses in your drawings.

1. On the **File menu**, click **Settings** --> **Advanced options** and go to **Drawing properties**.
2. Set the advanced option `XS_USE_CROSS_FOR_OPENING_SYMBOL` to `FALSE` to show the openings and recesses as follows:

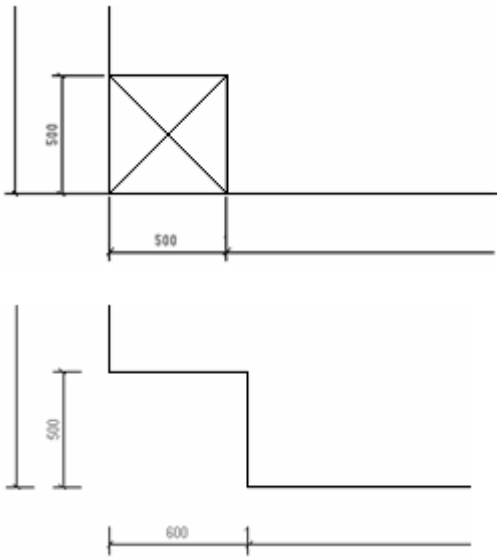


3. Set `XS_USE_OPENING_SYMBOL_IN_BORDER_HOLES` to `TRUE` to show opening/recess symbols in openings located at part borders. This advanced option is by default set to `FALSE`. The symbol used depends on the setting of the advanced option `XS_USE_CROSS_FOR_OPENING_SYMBOL`.





4. Set XS_USE_OPENING_SYMBOL_IN_CORNER_HOLES to TRUE to show opening/recess symbols in openings located in part corners. This advanced option is by default set to FALSE. The symbol used depends on the setting of the advanced option XS_USE_CROSS_FOR_OPENING_SYMBOL.



See also

View properties in drawings (page 912)

Define drawing views (page 687)

Define automatic section view properties

You can set some automatic properties for section views before you create a drawing. Properties for automatic section views need to be set in two places in drawing properties: in the **Section view** properties, and in **View creation** -->

View properties . The settings in the **Section view** properties apply to all section views in the drawing.

For a list and descriptions of section view properties, see Section view properties (page 929).

Define automatic section view properties

Note that for general arrangement drawings, you can only modify the **Start number or letter of section view and symbol label** setting.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Click **Section view**.
4. On the **Attributes** tab, set values for either **Fit by parts** or **Section depth** and **Distance for combining cuts**. Remember to select the **Size** check box first:
   - **Fit by parts** setting works as an alternative to **Section depth** and **Distance for combining cuts**, and it shows the whole part in the sections.
   - **Section depth** defines the positive and negative depth of the section view when sections are not combined. In an open drawing, you can adjust the section view depth also by dragging the view boundary.
   - **Distance for combining cuts** defines the distance range for combining cut views.
   - You can additionally control which section views get combined using the advanced option `XS_DRAWING_CUT_VIEW_COMPARISON_CRITERIA`.
5. Still on the **Attributes** tab, set the direction of the **Left section**, **Middle section** and **Right section** to **left** or **right**.
6. Go to the **Cutting line** tab and set the section mark line length and offset (distance between the section mark and the section).
7. Go to the **Section mark** tab and modify the section mark settings:
   a. Click the **...** button next to **A1** - **A5** to open the **Mark Contents** dialog box.
   b. Select the elements that you want to include in the mark.
   c. If needed, select an element from the list and click **< Add frame** and select the frame **Type** and **Color**.
   d. If needed, select an element from the list and select the text **Color**, **Font** and **Height**.

e. Go to the **Position** tab and select the side to show the text, the text position, the horizontal and vertical offset, and the text rotation options.

f. In **Start number or letter of section view and symbol label**, select whether you want to start the section view and section symbol labels with a number or a letter:

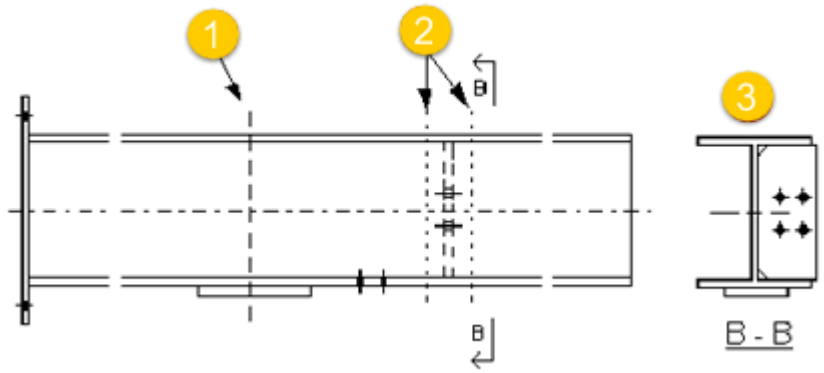
- You can enter any number starting from 1, or any letter A - Z or a - z (also shown in uppercase in the label).
- If you use a letter, and the entered string is longer than one letter, only the first letter is shown. If you use numbers, all entered numbers are shown.
- The start number in the label changes only when you change it in the drawing properties before creating a drawing, and when you change it in an existing drawing and recreate the drawing, in which case the labels for all automatically included section views and all new section views will change.

g. Click **OK** to return to drawing properties.

8. Click **View creation** and add the section and end views you want to create.
9. Still in the **View creation** panel, select the view and the properties that you want to change and click **View properties**.
10. Adjust the view properties as required.
11. Click **Save** to save the view properties.
12. Click **Close**.
13. Repeat steps 9 - 12 for all the section and end views you create.
14. To save the changes, click **Save**.

Examples of section view and mark settings

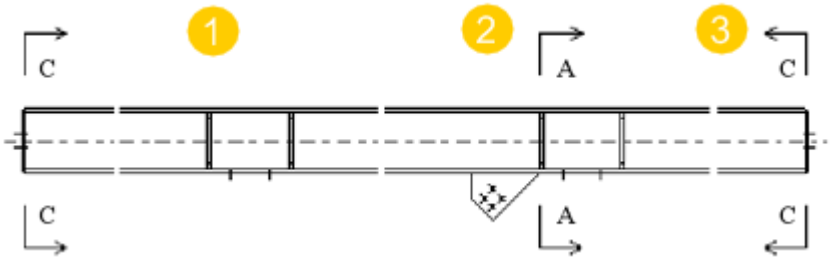
Combining section views



1. Distance for combining cuts = 1’- 4"
2. Section depth = 4"
3. Combined sections

Section view direction

The arrow in the section view symbol indicates the direction of the section view, as shown below:



1. Left section, right direction
2. Middle section, right direction
3. Right section, left direction

Section marks

See below for examples of section marks:

Show section and end view direction marks in drawings

You can display view direction marks in section views and end views in drawings.

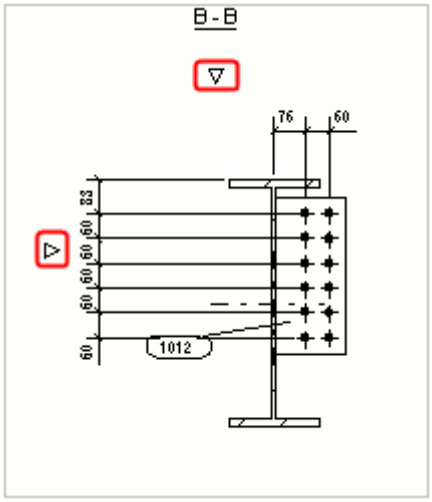
For details about view properties, see View properties in drawings (page 912).

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.

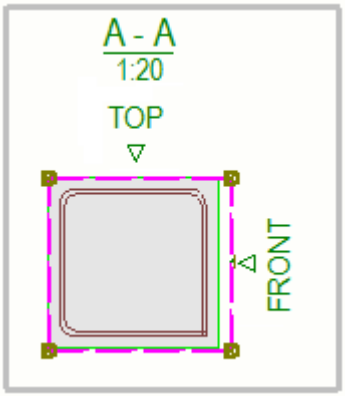
   Select a section view or an end view.
4. Go to the **Label** tab in **View Properties**.
5. Select one of the options in **View direction marks**: **Show marks**:
   - **Symbol only**
   - **Label only**
   - **Symbol and label**
   - **None** does not show any marks.
6. Define the height of the symbol and text label in **Height**.

   If you try to use 0, you will get an error message.
7. To save the changes, click **Save**.
8. Click **Close**.
9. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

The view direction mark is shown with a small symbol (optionally with a text label) around the end or section view.



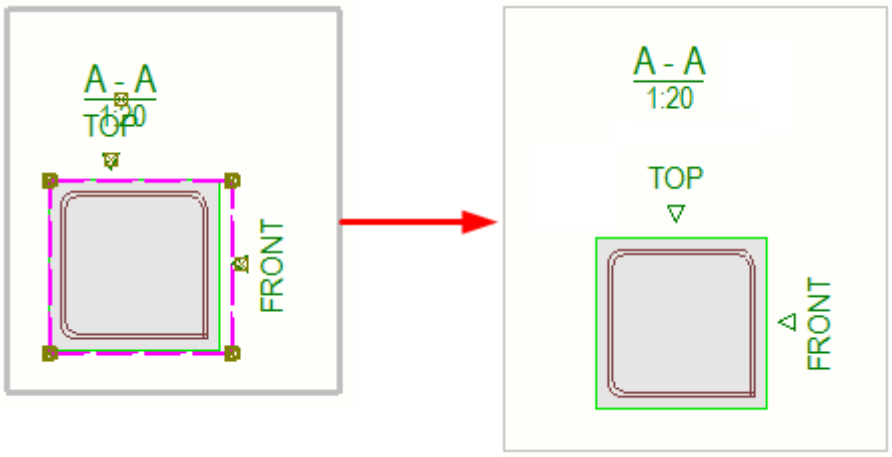
The view direction mark position follows the label position setting. In the image below, **Center by view restriction box** has been selected for the label.



Tips

- You can drag view direction marks to a better place in a drawing view: click the view frame to activate the handles, point the handle, press and hold

down the left mouse button and drag. The view frame is resized automatically, if necessary.



- You can define the view direction mark symbol in **File menu** --> **Settings** --> **Advanced options** --> **Drawing properties** by using the following advanced options:
  - XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_BACK
  - XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_BOTTOM
  - XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_FRONT
  - XS_DRAWING_VIEW_DIRECTION_MARK_SYMBOL_TOP

    The default symbol is `xsteel@66`.

Define the location for end views and section views

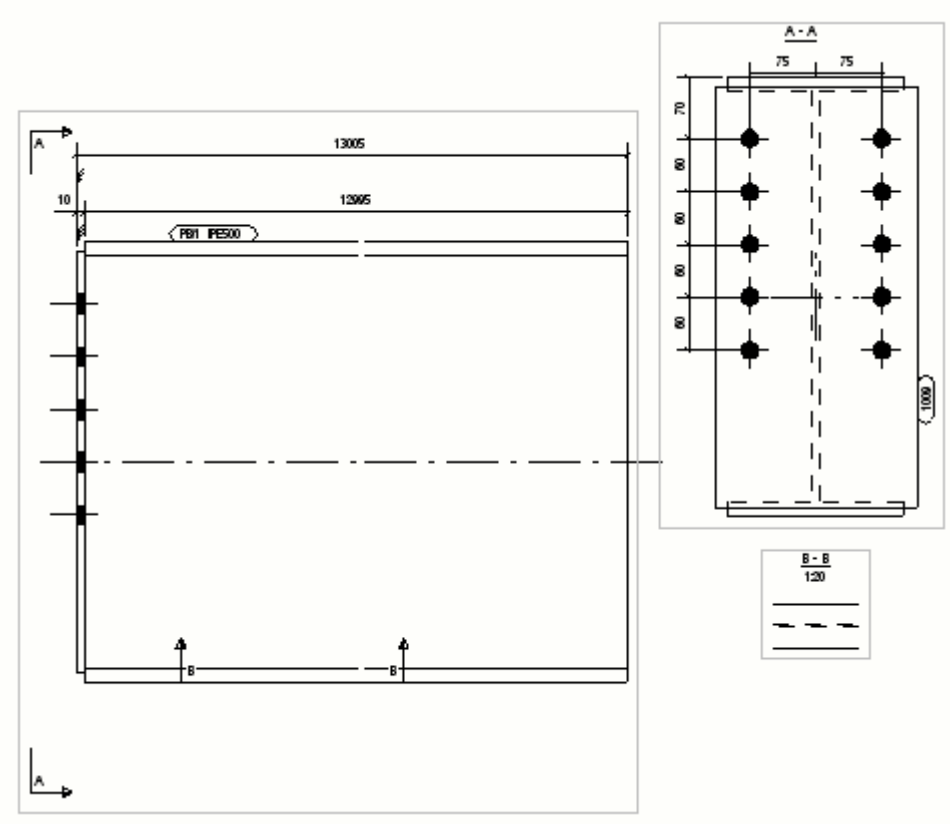
You can always place section views and end views next to the main view or in any empty location in a single-part, assembly, and cast unit drawing.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Click **Layout** and go to the **Other** tab.
4. Set **Align end views with main view** to **Yes** to place the views next to the main view.
5. Set **Align section views with main view** to **Yes** to place the views next to the main view.
6. To save the changes in a drawing properties file, click **Save** at the top.
7. Click **OK** and create the drawing.

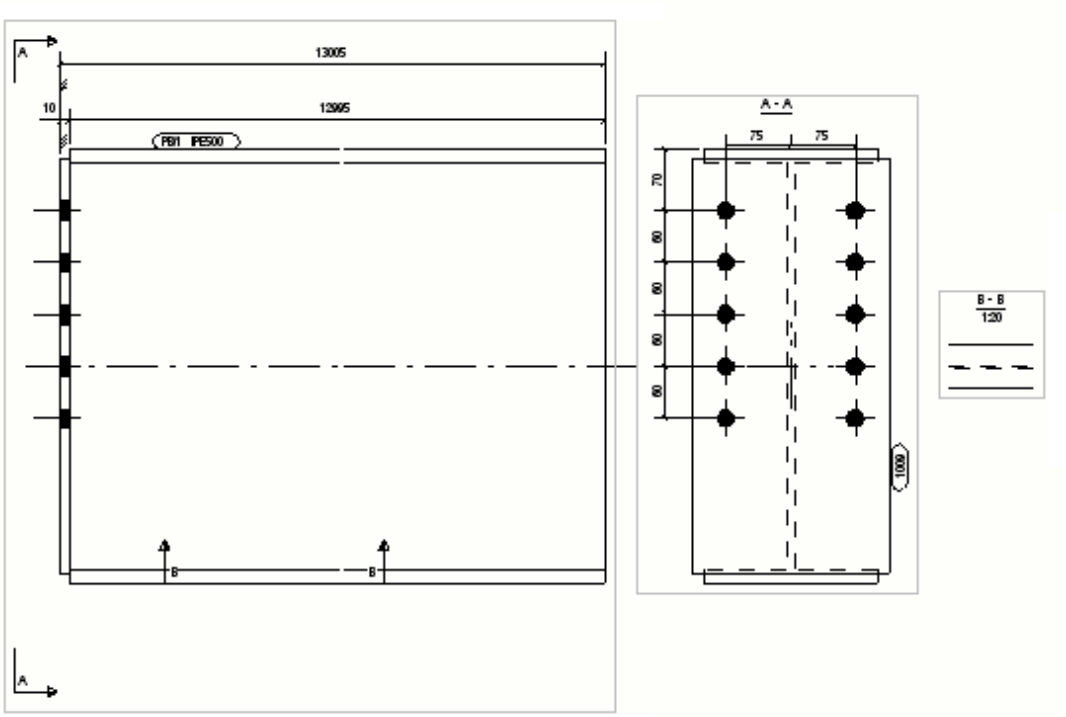
   If you select **No**, Tekla Structures places the section and end views in any available location.

Example

End and section views in any location (**No** selected).



End and section views beside the main view (**Yes** selected).



For more information about layout properties, see Layout properties (page 910).

8.5 Define dimensioning

Dimensions are associative annotation objects that represent building object measurements. Dimensions are more than lines or vectors; they are interactive callouts of geometry. In automatic dimensioning Tekla Structures creates dimensions in the whole drawing or in drawing views based on dimensioning settings you define before creating the drawing.

In single-part, assembly and cast unit drawings automatic dimensions are set view by view.

In general arrangement drawings, automatic dimensions are set for the whole drawing.

You can define automatic dimensions settings before you create a drawing and you can also modify the settings after you have created the drawing.

To	Click the links below to find out more
Create automatic dimensions in single-part, assembly or cast unit views	What are automatic view-level dimensions (page 726) Add automatic view-level dimensions (page 730)

To	Click the links below to find out more
Check the settings affecting dimension creation, and have a look at some examples	Dimensioning rule properties (page 942)
Create a filter that is needed in view-level dimensioning for selecting the objects that you want to dimension	Create a drawing view filter for view-level dimensioning (page 744)
See examples of different combinations of dimensioning types and settings	Various scenarios of using different dimensioning types (page 754)
Use the traditional way of dimensioning in the Dimensioning dialog box by using the dimensioning type Integrated	Add automatic view-level dimensions using Integrated dimensioning type (page 758)
Create dual dimension tags automatically in all types of drawings	Add automatic dual dimensions (page 779)
Control the dimensions that Tekla Structures adds for unfolded parts	Add dimensions to unfolded parts (page 780)
Create minimum and maximum position dimensions for bolts	Add minimum and maximum position dimensions to bolts (page 781)
Add extensions to dimension lines	Create dimension line extensions (page 782)
Adjust absolute dimensions	Change the appearance of absolute dimensions (page 783)
Exaggerate narrow dimensions to make them easier to read	Create exaggerated dimensions (page 783)
Use another prefix in radial dimensions	Change the prefix in radial dimensions (page 785)
Dimension plates using advanced options	Add dimensions to plates (page 786)
Adjust profile dimensioning using dimension planes table	Add dimensions to profiles (page 790)
See examples of sloped dimension texts	Sloped dimension texts (page 793)
Add automatic dimensions in general arrangement drawings	Add automatic dimensions to general arrangement drawings (page 794)

What are automatic view-level dimensions

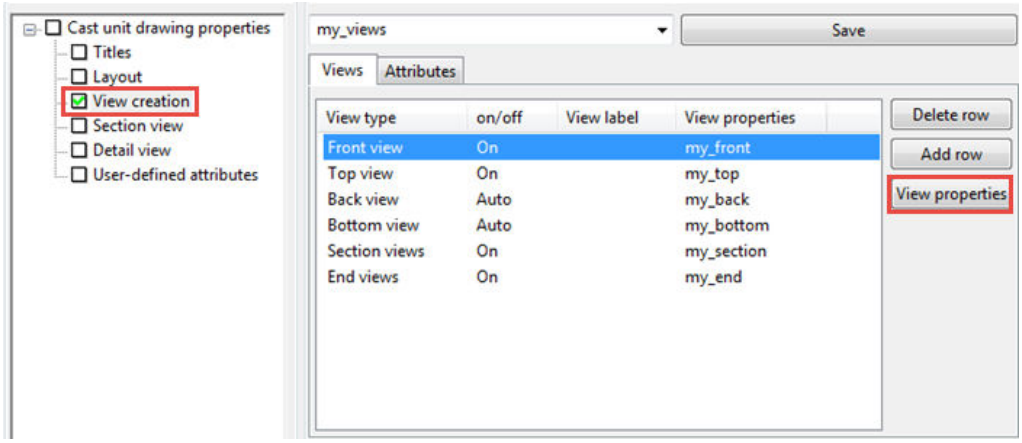
Automatic view-level dimensioning gives you full control on the dimensions in each drawing view you create, with a lot of dimensioning options. Automatic

view-level dimensioning can be used in single-part, assembly and cast unit drawings.

In view-level dimensioning, the dimensions are created based on the rules you define. You can define what you want to dimension, where the dimensions are placed, in which order they are created, and which settings you want to use for each dimension. You can dimension shapes and holes, for example.

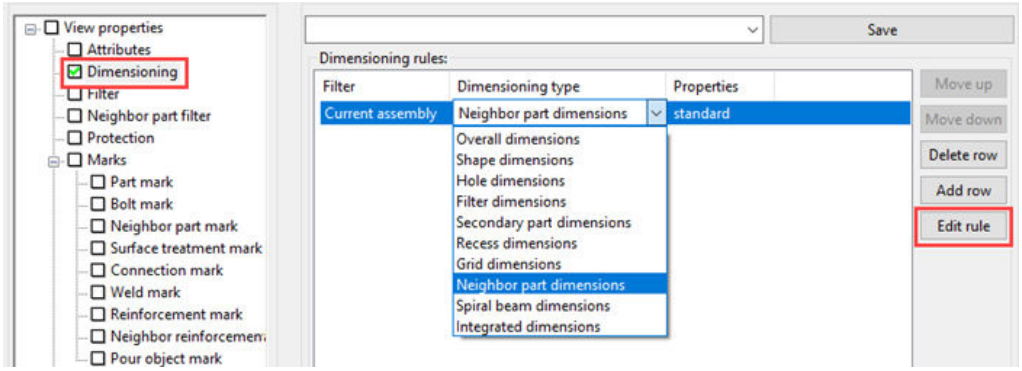
Below you can find short description on how to proceed in dimensioning.

1. When you click the **View creation** option in the options tree in drawing properties, you are able to select the views to be created and the view properties to be used.



2. Click **View properties**.
3. Click **Dimensioning** in the options tree.

   In the **Dimensioning** panel, you can add rules by clicking **Add row**. Then select which dimensioning rules you want to use in the **Dimensioning type** column and the desired dimensioning rule properties file.



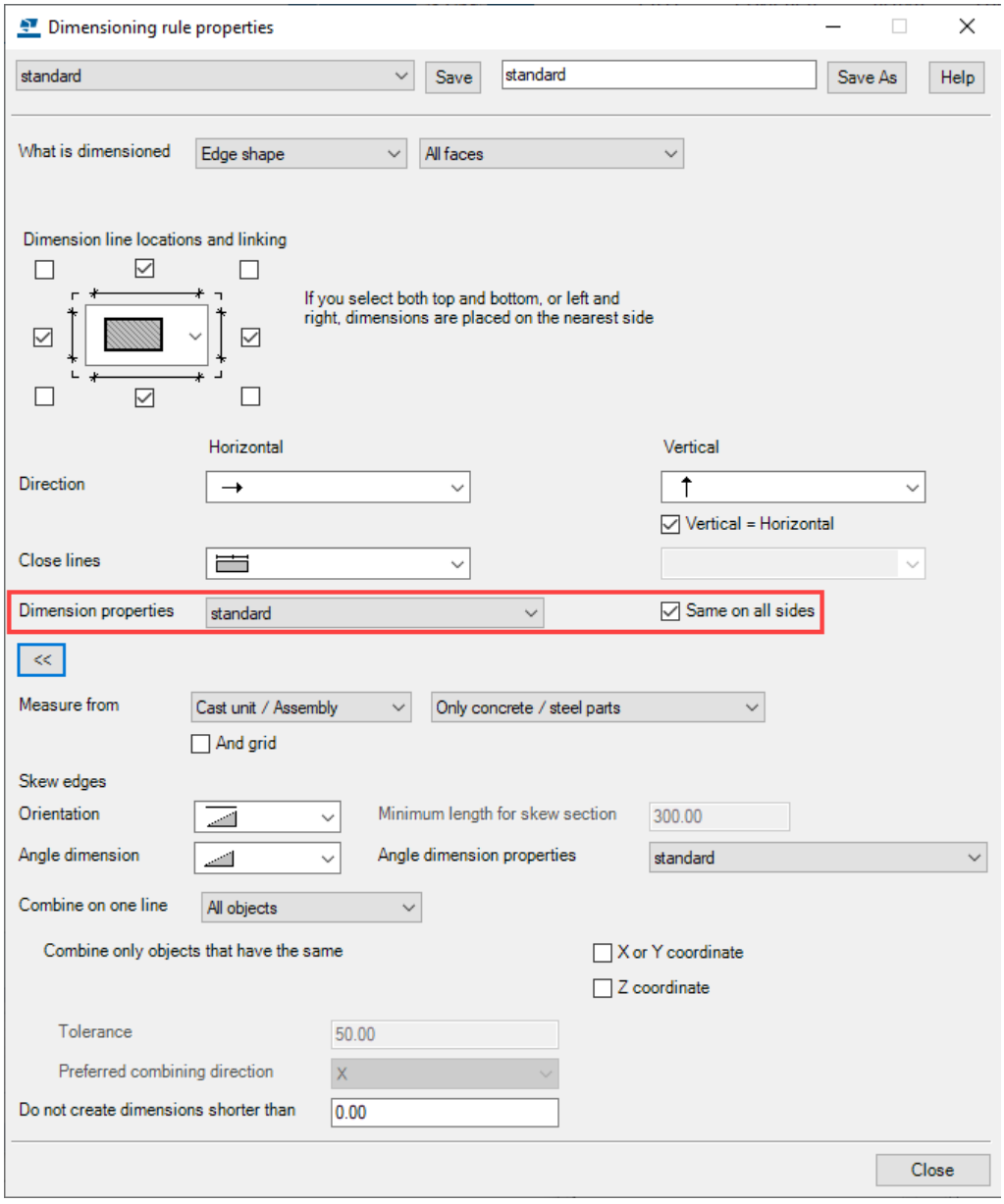
4. You can modify the selected rule by clicking **Edit rule**.

   In the **Dimensioning rule properties** dialog box, you can select what to dimension, how to dimension, which objects to measure from, where to place the dimensions, and select the dimension properties. The **Properties** list contains properties files that you have saved in the object

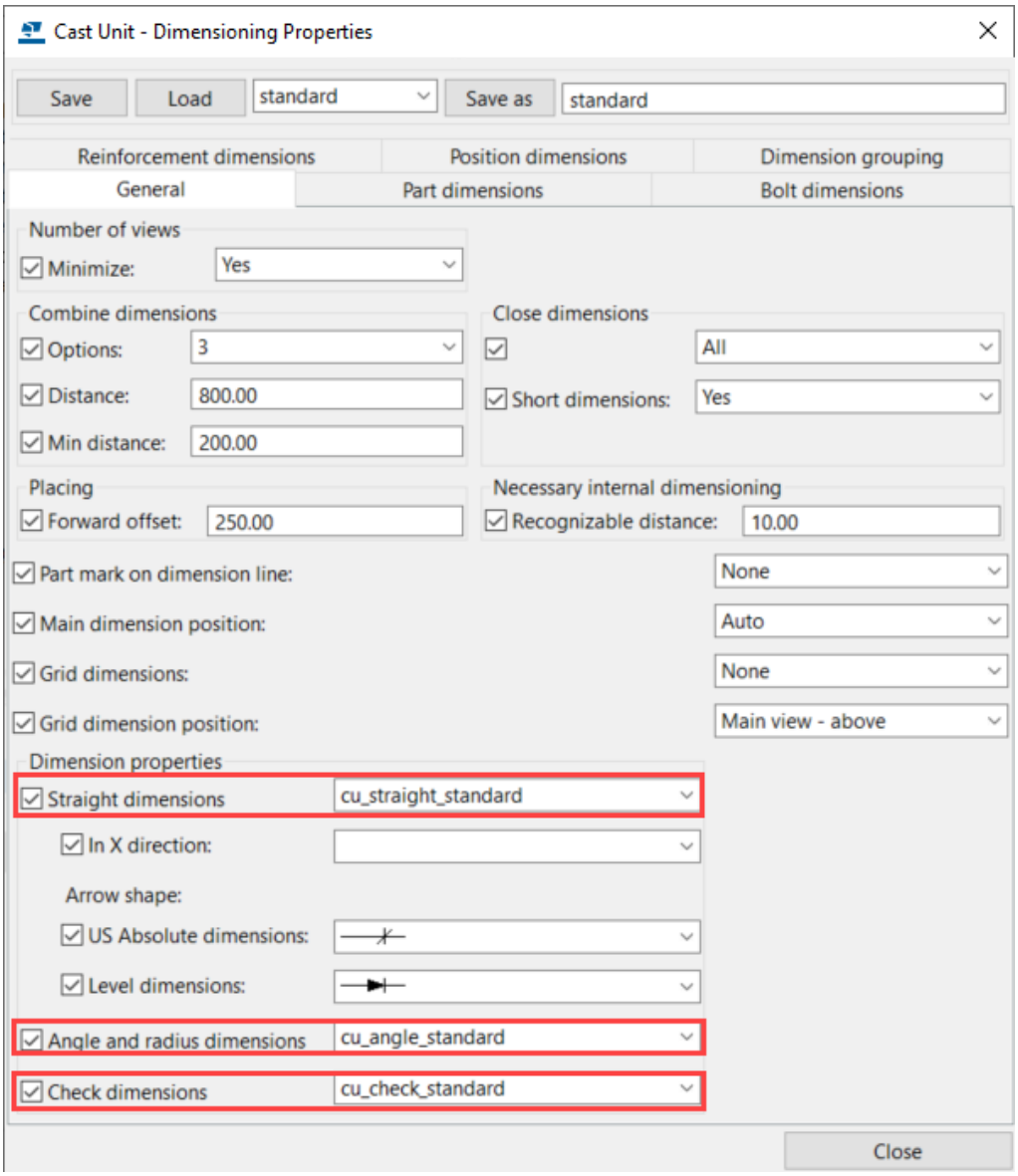
level **Dimension Properties** dialog box in an open drawing. For example, you may want to use some special font or color in the dimensions. To do this, double-click a dimension in a drawing, make the necessary changes and save the properties file. Then you can load the properties here.

The model folder is the dimensioning rule working folder. Dimensioning rule files can also be read from firm and project folders defined by the advanced options XS_FIRM and XS_PROJECT. You can also define relative paths, for example, `XS_FIRM=".\ts"` can be used for reading files from the 'ts' folder inside the model folder.

To select different dimension line properties for each side, deselect the **Same on all sides** and select the dimension properties from the lists.



If you select **Integrated dimensions**, the **Dimensioning Properties** dialog box is displayed. Make your changes to the settings on the tabs and save the properties file with a unique name using **Save as**. You can also load dimension properties here as well.



5. When you have set the rule properties, give a name to the rules file and click **Save As**.
6. Click **Close** to return to the **Dimensioning** panel.
7. Ensure that you have selected correct dimensioning properties files for the dimensioning rules.

8. Enter a unique name for the view properties in the box at the top of the **View Properties** dialog box and click **Save**.

Now you can select the saved view properties for a view in the **View creation** panel. These view properties contain the saved dimensioning properties.

See also

Dimensioning rule properties (page 942)

Add automatic view-level dimensions (page 730)

Add automatic view-level dimensions using Integrated dimensioning type (page 758)

Add automatic view-level dimensions

The next examples will go through the basic workflow of creating automatic dimensions on view level. The goal is to create drawing properties that you can use later on to create similar drawings, including all necessary views, having the dimensions that you want, just by loading the appropriate drawing properties file and then creating the drawing.

If you want to use Integrated dimensions, see Add automatic view-level dimensions using Integrated dimensioning type (page 758), or dimension spiral beams, see Dimension spiral beams (page 444).

The workflow consists of four tasks:

1. Create drawing properties
2. Define drawing views and drawing view settings
3. Define dimensioning rules
4. Create and apply dimensioning rule properties

Define the drawing property file

Create a drawing property file that will pull together all the settings that you define in drawing properties, including view-level dimension settings.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.

2. In drawing properties, load the properties that you want to use as a basis for the new properties by selecting from the list at the top.

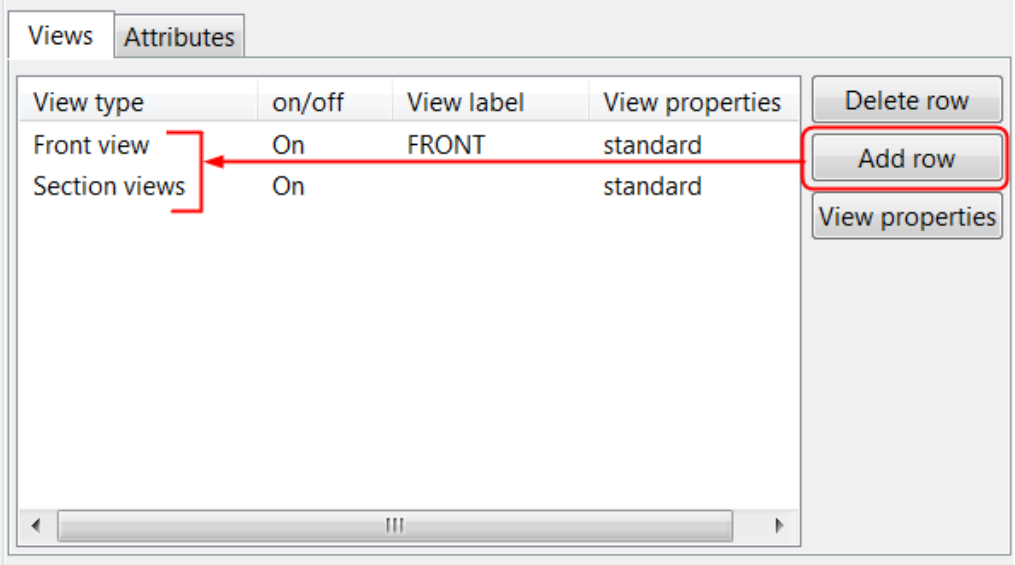
   If you do not have any suitable drawing properties available, give the drawing property file a unique name and save the properties by clicking **Save**.

Now you have created a drawing property file where you can save the new dimensioning settings.

Define the drawing views to be created

Create the desired views and define the view properties to use:

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load the drawing properties that you created and saved in the phase 1 of this workflow.
3. Click **View creation** in the options tree.
4. In the next panel, click **Add row** to add new views in your drawing.



5. For those views that you want to create, set the **on/off** control to **On**.

   If you select **Auto**, the view is created if relevant dimensions are created with the used dimensioning settings. If relevant dimensions are not created, neither is the view. Tekla Structures can automatically decide whether dimensions are relevant or not.

Now you have defined the views that you want to have in the drawing you will create. You can save the list of views using **Save**, and then load it if you need the same set of views in another drawing.

Define view dimensions

Define the dimensioning rule properties to be used in the drawing views that you just created.

The model folder is the working folder. Dimensioning rule files can also be read from firm and project folders defined by the advanced options XS_FIRM and XS_PROJECT. You can also define relative paths, for example, `XS_FIRM=".\ts"` can be used for reading files from the 'ts' folder inside the model folder.

If you are planning to use filters to select the parts that you want to dimension, you need to create the drawing view filters first, for example, for selecting embeds, inner panels or outer panels.

You need to create separate dimensioning rules for each dimensioning type. For example, the rules made with **Overall dimensions** are valid for **Overall dimensions** only, not for **Shape dimensions**, for example.

1. Select a view in the **View creation** panel and click **View properties**.
2. In the **View Properties** dialog box, click **Dimensioning** in the options tree to define the dimensions to be created for the selected view.
3. Click **Add row** to add a rule.

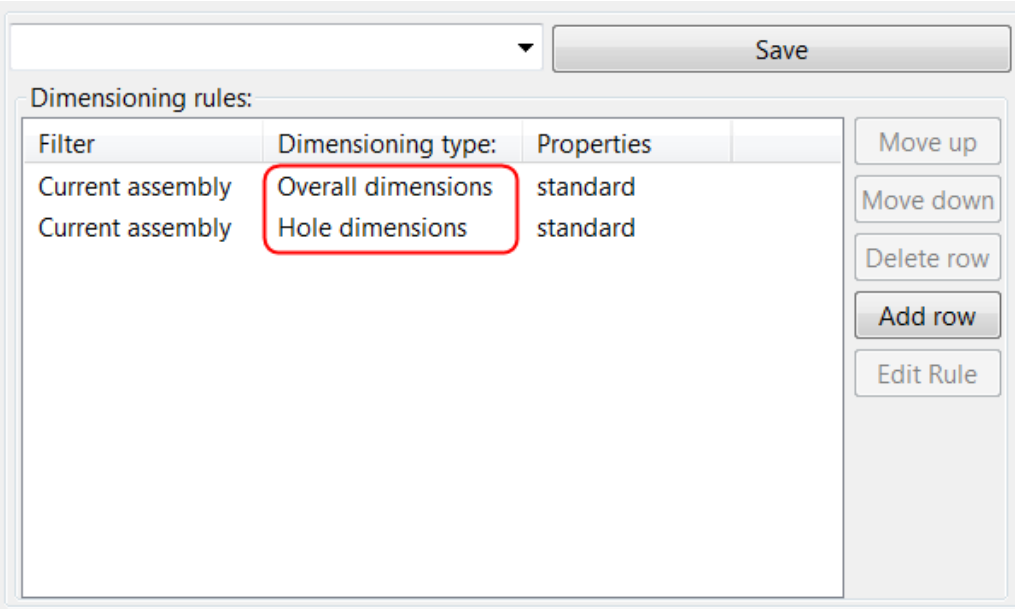
   Here we add two rows.

   The order of the rules in the list defines the order of the dimension lines in the drawing: the dimension created by the first rule is placed closest to the dimensioned object

   At the moment, the filter can only be set for the **Integrated dimensions** in this panel. You are able to select the filter in the **Dimensioning rule properties** dialog box, and you can leave the selection **Current assembly** in the **Filter** column for all rules.

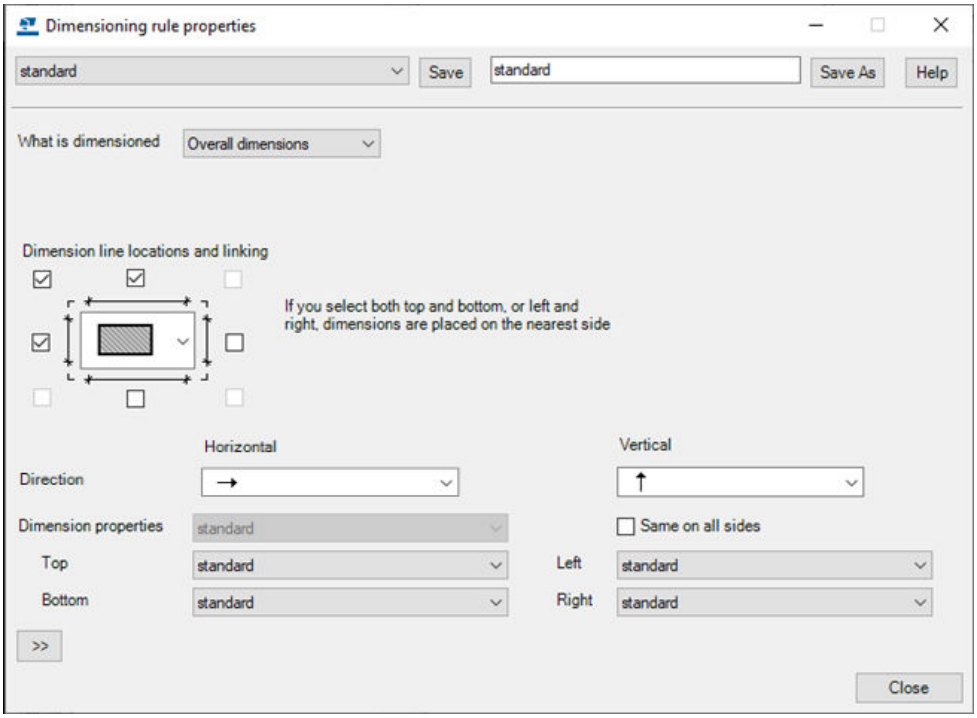
   Leave **Current assembly** in the **Filter** column.
4. Select the **Dimensioning type** for the selected rules.

Here we select overall dimensions and hole dimensions:

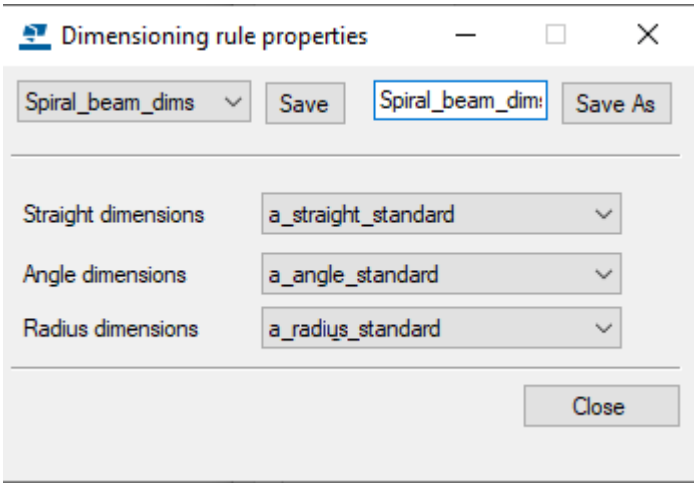


5. Click one of the rules and click **Edit rule**.
6. Depending on the selected dimensioning type, a specific **Dimensioning rule properties** dialog box is displayed. Do one of the following:
   - For most of the dimensioning types, you need to define what to dimension, and where and how to place the dimensions. From the **Dimension properties** list, select a suitable set of saved dimension properties to change the dimension appearance, dimension text font size or color, for example. If you want, you can define different

dimension line properties for each side by deselecting **Same on all sides** and selecting different dimension properties.

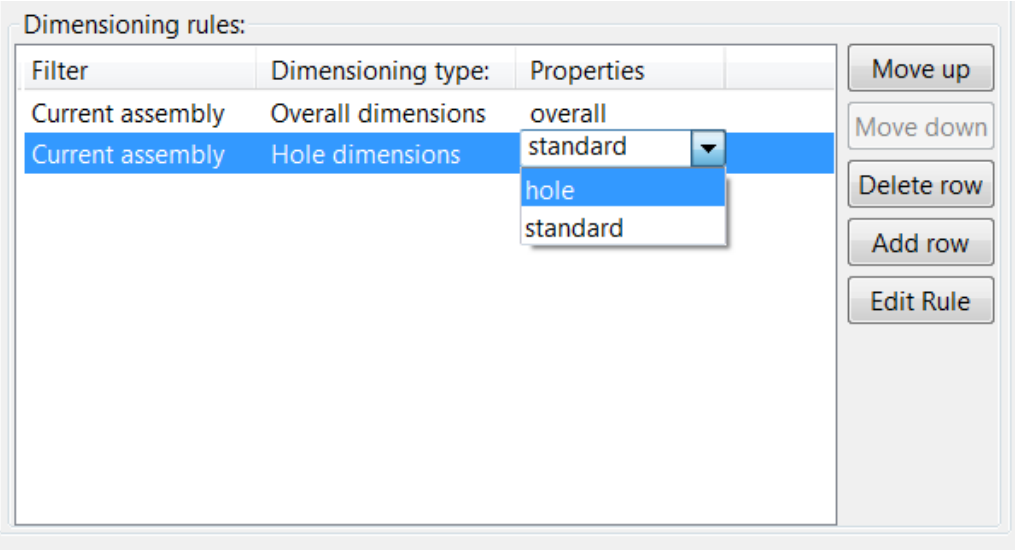


- If you have selected dimensioning type **Spiral beam dimensions**, select predefined dimension properties. If none of the available properties suit your needs, open a drawing, click **Drawing** --> **Properties** --> **Dimension** when a drawing is open, and edit and save the needed dimension properties so that they are available for selecting in the spiral beam **Dimensioning rule properties** dialog box for the three dimension types.



7. Give the dimensioning rule a unique name and click **Save As**.
8. Click **Close**.

9. Define other dimensioning rules needed for the view following the steps 5 - 8.
10. Select correct properties for the rules.



Even though the dimension lines are created and placed by default in the order that you define them in the **View creation** panel, Tekla Structures searches for the first suitable location for the dimension lines according to the placement and protection settings. So the dimension placement may not always follow the creation order. Check the result and adjust the location of the dimension lines if necessary.

11. In the top-left corner, give a unique name to the view properties and click **Save** to save your changes to the view properties file.

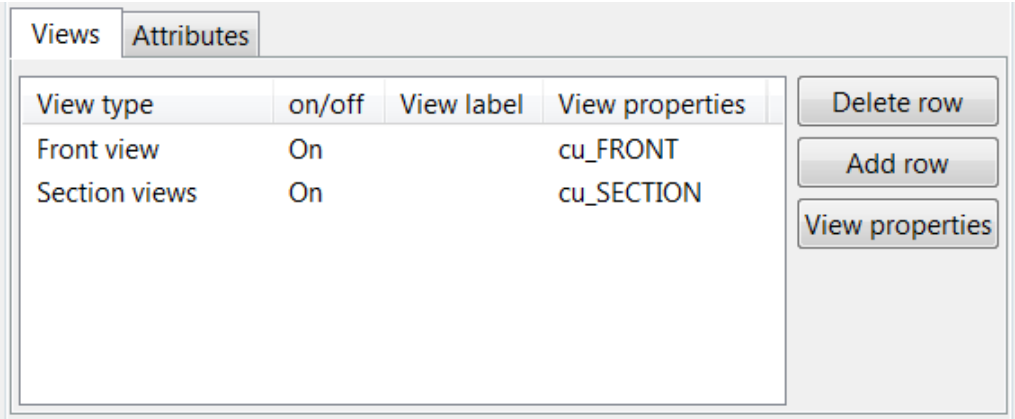
Now you have created new view properties containing two types of dimensions. You can connect this properties file with a drawing view and use the defined dimensions in that view.

Connect view properties to views and save drawing properties

Connect then new view properties to drawing views and save drawing properties.

1. In the **View creation** panel, select correct view properties for the views that you are creating.

In the example below, one front view and one section view are created, and the views have been connected to view properties `cu_FRONT` and `cu_SECTION`.



2. Remember that you created or loaded the drawing properties file in phase 1 of this workflow. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

Tekla Structures creates the drawing according to the definitions in various properties files.

Example workflow: Create automatic overall and hole dimensions on view level

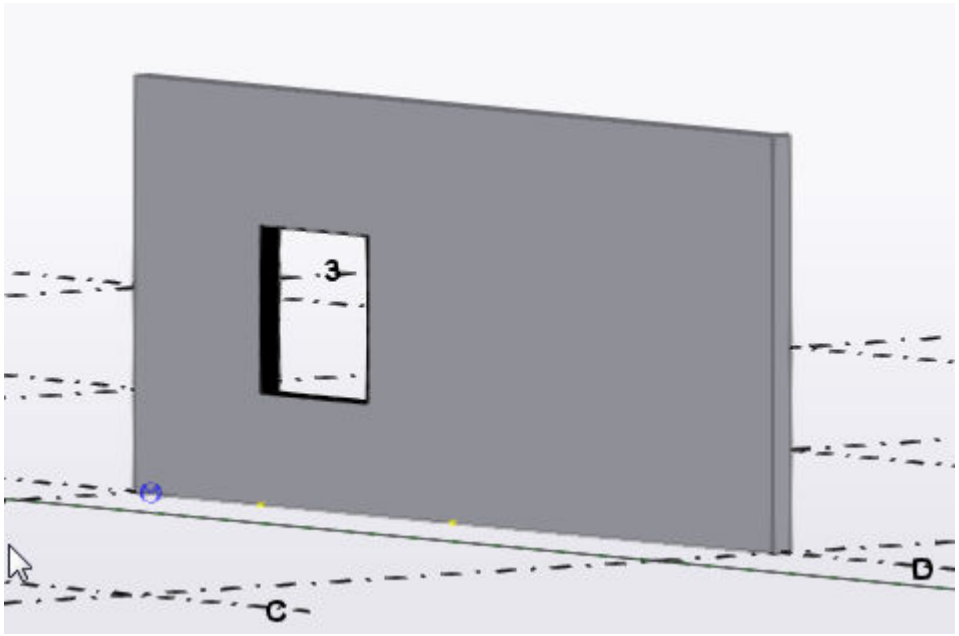
In this example workflow, you will create a cast unit wall panel drawing that contains

- one front view with automatic overall and hole dimensions
- one section view with overall dimensions

In overall and hole dimension rules you will apply the dimension properties that you have earlier created and saved manually in a cast unit drawing. You will save the created dimensioning rule properties in view properties. Finally, you will save the created view properties in drawing properties and create a cast unit drawing.

Before starting, in an open cast unit drawing, open the property pane, and select **Straight dimension** in the object list. Set the dimension text font size to `5.00` and save the properties with the name `dim_font_5`. Then set the dimension color to `red` and save the properties with the name `dim_red`.

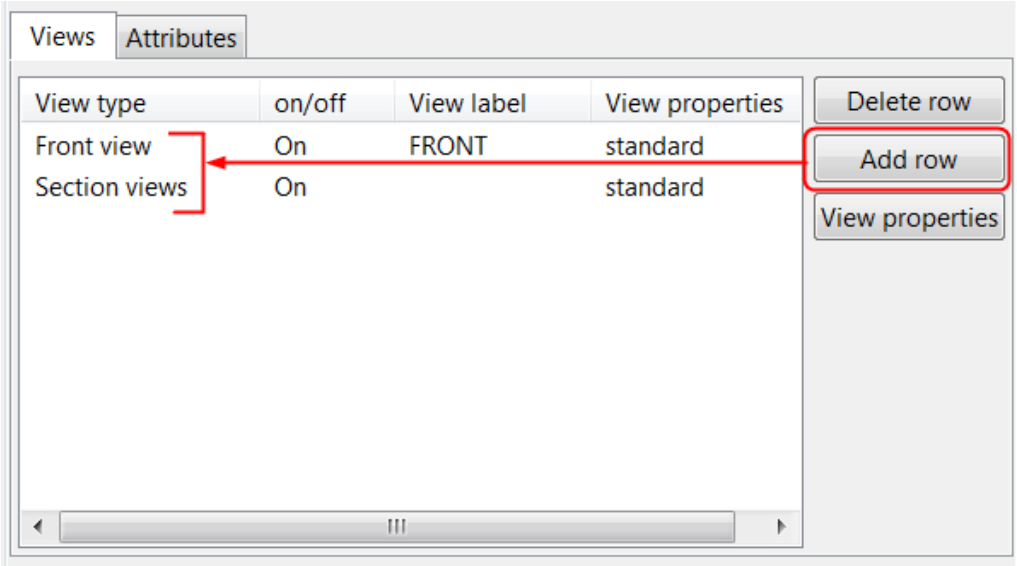
In this example, you are going to dimension the following cast unit wall panel in the model:



Define the views to be created

1. On the **Drawings & reports** tab, click **Drawing properties** --> **Cast unit drawing** .
2. Load drawing properties that are as close to the ones you need as possible.
3. Click **View creation** in the options tree.
4. In the **View creation** panel, click **Add row** to add new views in your drawing.

   In this example, you want to add two views, one front view and one section view.



5. Set the **on/off** setting to **On** for the views that you want to create.

If the list contains extra views, set them to **Off** or use the **Delete row** button to delete them.

Now you have defined the views that you want to create. Next, you need to define the dimensions that you want to have in the front view and section view.

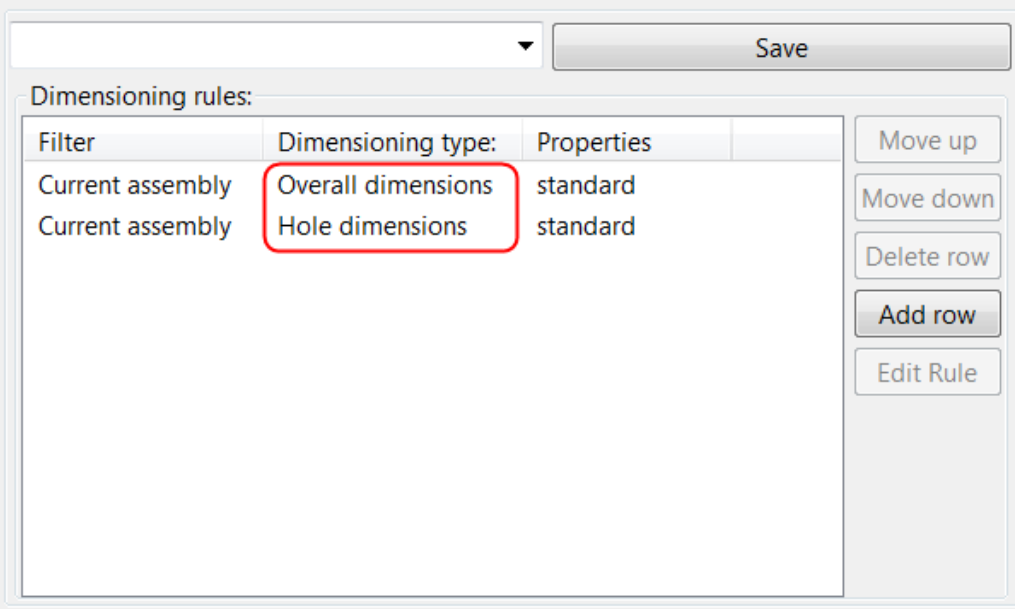
Define front view dimensions

1. Select a view in the **View creation** list.

   In this example, select one **Front view**.
2. Click **View properties** and then click **Dimensioning** in the option tree to define the dimensions to be created in the front view.
3. In the **Dimensioning** panel, use **Add row** to add two new dimension rules in the dimensioning rules list.
4. Select **Overall dimensions** as the first rule and **Hole dimensions** as the second.

   The order of the rules in the list defines the order of the dimension lines in the drawing: the dimensions created by the first rule is placed closest to the dimensioned part.

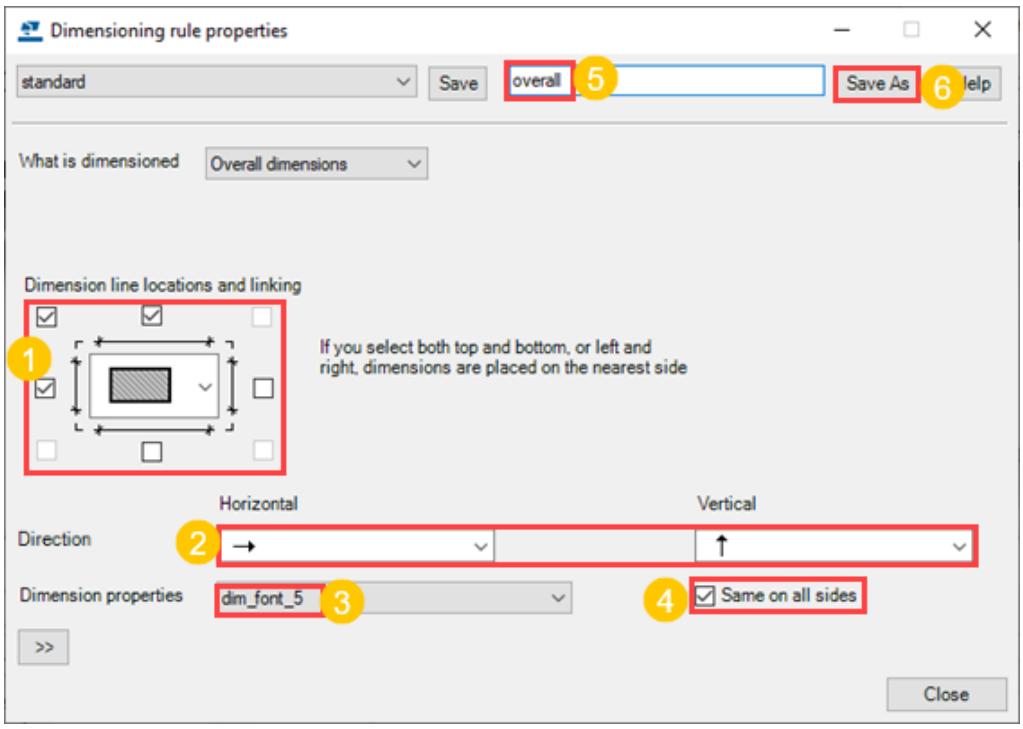
   Leave **Current assembly** in the **Filter** column for both rules.



5. To the define overall dimension rules, click the **Overall dimensions** row, and click **Edit rule**.

6. In the **Dimensioning rule properties** dialog box, define what to dimension, where and how to place the dimensions, and which dimension properties to use.
   - Select the check boxes above and on left side of the object, and also the check box in the upper-left corner to link the dimensions together.
   - Use default values in the **Start point** lists. The default values are left for the **Horizontal** and bottom for the **Vertical** dimension.
   - From the **Dimension properties** list, select a suitable set of saved dimension properties. In this example, select the dimension properties file `dim_font_5`, which contains a definition for a larger font.
   - Give the dimensioning rule a unique name and click **Save As**.

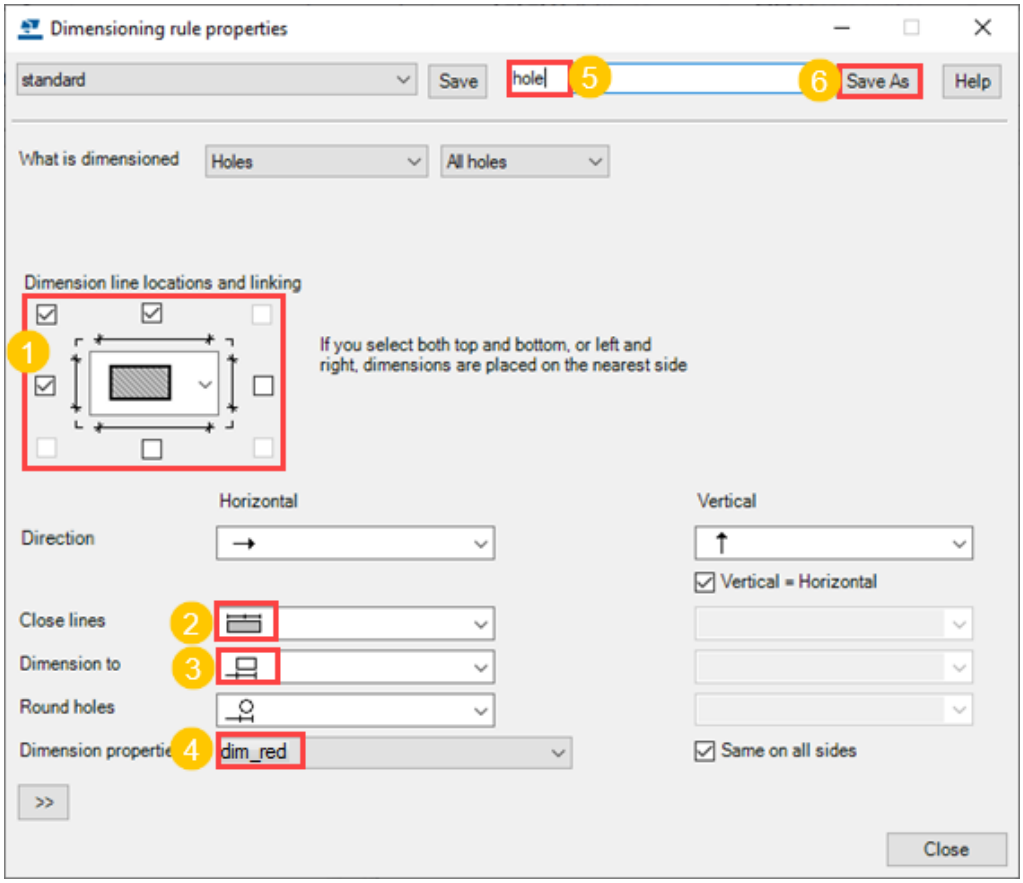
     In this example, the name `overall` is used.



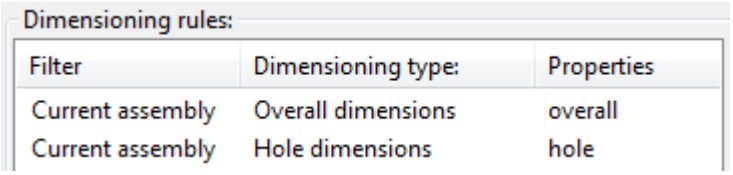
7. Click **Close**.
8. Next, define the hole dimensions. In the **Dimensioning** panel, select **Hole dimensions** from the dimensioning rules list and click **Edit rule**.
9. Create dimensioning rules for hole dimensions:
   - Select the check boxes above and on left side of the object, and also the check box in the upper-left corner to link the dimensions together.
   - Use the default values in the **Start point** lists.
   - In **Close lines**, select the setting that extends the dimension lines to the other end of the cast unit.
   - In **Dimension to**, select the setting that dimensions to both ends.

- From the **Dimension properties** list, select a suitable set of saved dimension properties. In this example, select the dimension property file `dim_red` containing a definition for red dimensions.
- Give the hole dimensioning rule a unique name and click **Save As**.

  In this example, the name `hole` is used.

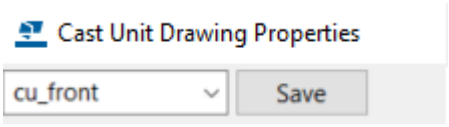


10. Click **Close**.
11. For the **Overall dimensions** rule, select `overall` properties, and for the **Hole dimensions** rule, select `hole` properties from the **Properties** column.



12. In the **View Properties** dialog box, give the front view properties a unique name and click **Save**.

    In this example, the front view properties are saved with the name `CU_Front`.



Now you have saved the view properties for the front view containing overall and hole dimensions. Leave the **View Properties** dialog box open for further modifications.

Define section view dimensions

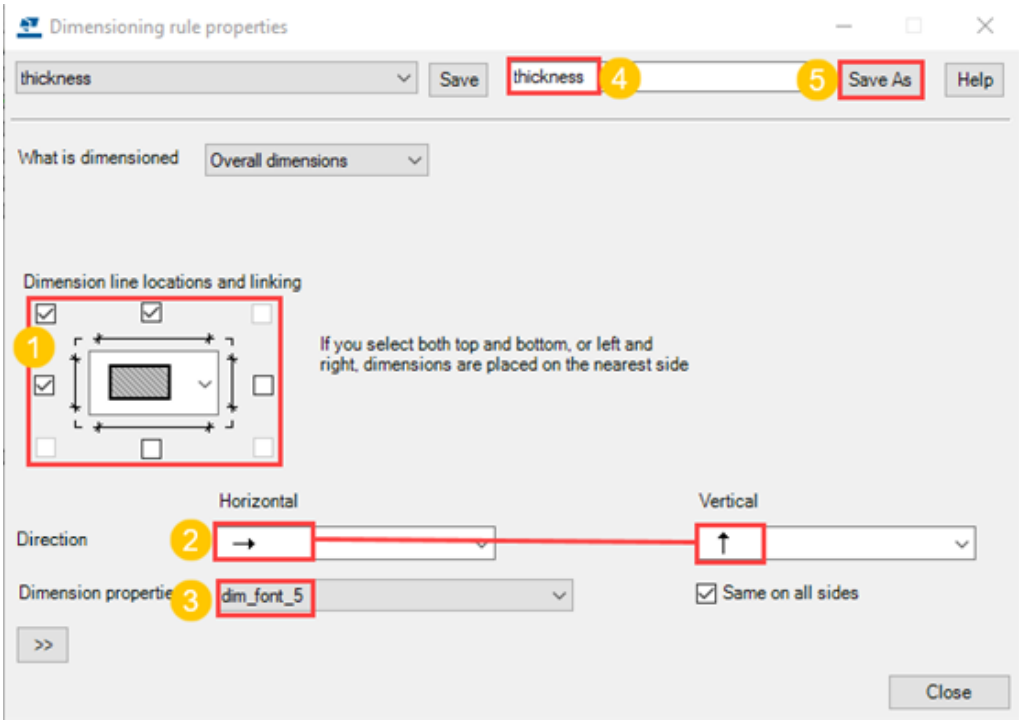
A section view is also needed in the cast unit drawing, because you want to show the wall thickness. Next, you will create overall dimensions for the section view.

1. In the **View creation** panel, select the **Section views** row and click **View properties**.
2. Load the view property file `CU_Front`.

   You can start creating new view properties on the basis of already existing view properties.
3. Click **Dimensioning** in the options tree.
4. In the **Dimensioning** panel, delete the unnecessary hole dimension rule by clicking the **Hole dimensions** row and **Delete row**.

   You will only need the overall dimensions in the section view.
5. Click the **Overall dimensions** row and click **Edit rule**.
6. Create a dimensioning rule for the overall dimensions in the section view:
   - Select the check box below the object only, because you only want to show the thickness.
   - Select the same dimension properties as for the overall dimensions in the front view, because you want to show the dimension text with a little bit larger font: `dim_font_5`.
   - Give the rule a unique name and click **Save As**.

     In this example, the name `thickness` is used.



7. Click **Close**.
8. In the **Dimensioning** panel, select `thickness` in the **Properties** column as the property file for the overall dimensioning rule.
9. Give a unique name for the section view properties and click **Save As**.

   In this example, the name `CU_Section` is used.
10. Click **OK**.

    Now you have saved the view properties for the section view containing overall dimensions.

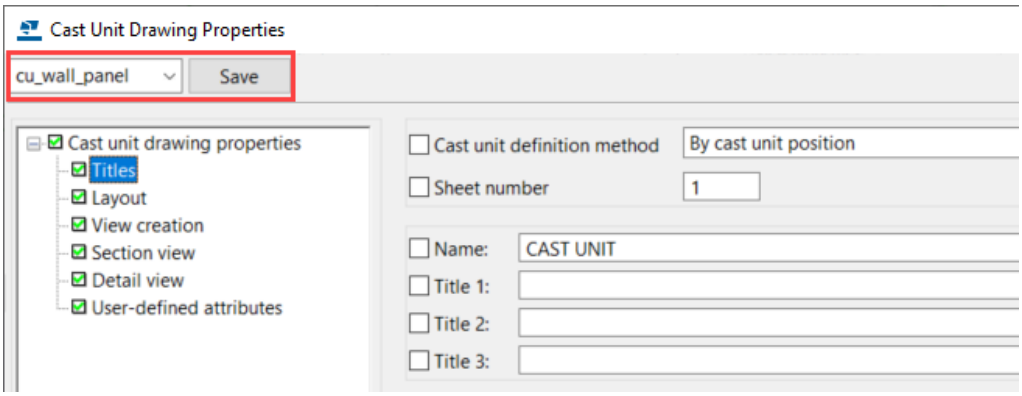
Connect view properties to views and save drawing properties

1. In the **View creation** panel, select `CU_Front` for the front view and `CU_Section` for the section view.

View type	on/off	View label	View properties
Front view	On		CU_Front
Section views	On		CU_Section

2. In the **Drawing Properties** dialog box, give the drawing properties a unique name and click **Save**.

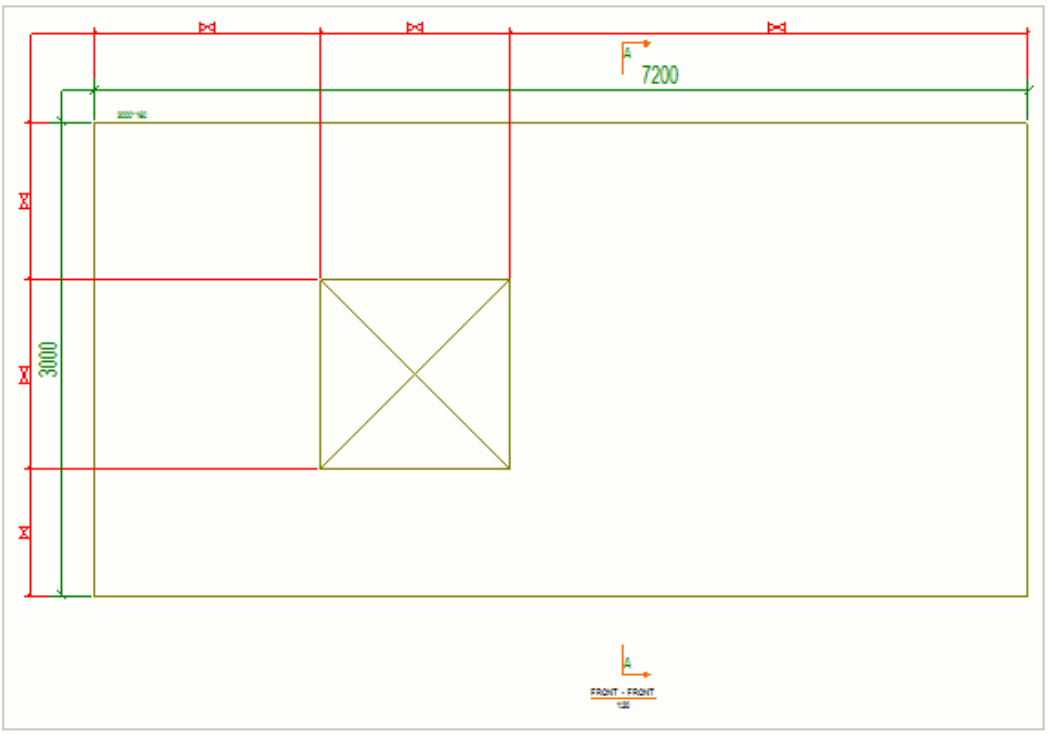
   In this example, the name `cu_wall_panel` is used.



3. Click **OK** and create the cast unit drawing.

Tekla Structures creates the cast unit drawing according your definitions in different property files. The cast unit drawing contains a front view and a section view. The overall dimensions in both views have a little bit larger font, and the front view has red hole dimensions. Only the wall thickness is dimensioned in the section view.

You can use the drawing property file `cu_wall_panel` later on when you need drawings with similar settings.



TIP You can still change the dimensioning settings in the views after creating the cast unit drawing:

1. Double-click the drawing view frame to open the view properties dialog box.
2. Click **Dimensioning** in the options tree to open the **Dimensioning** panel where you can select and then edit the dimensioning rules.

Create a drawing view filter for view-level dimensioning

You need to create a drawing view filter to use the **Filter dimensions** dimensioning type in view-level dimension creation in single-part, assembly and cast unit drawings. The filter needs to be created on view level, because you create the dimensions view by view.

Create a drawing view filter

You need to create a drawing view filter to use the **Filter** option in view-level dimension creation in single-part, assembly and cast unit drawings. The filter needs to be created on view level, because you create the dimensions view by view.

NOTE If you want to improve the performance of dimensioning, check if you have some rules that are not creating any dimensions and remove them. Every rule consumes time even if it does not find any objects to dimension.

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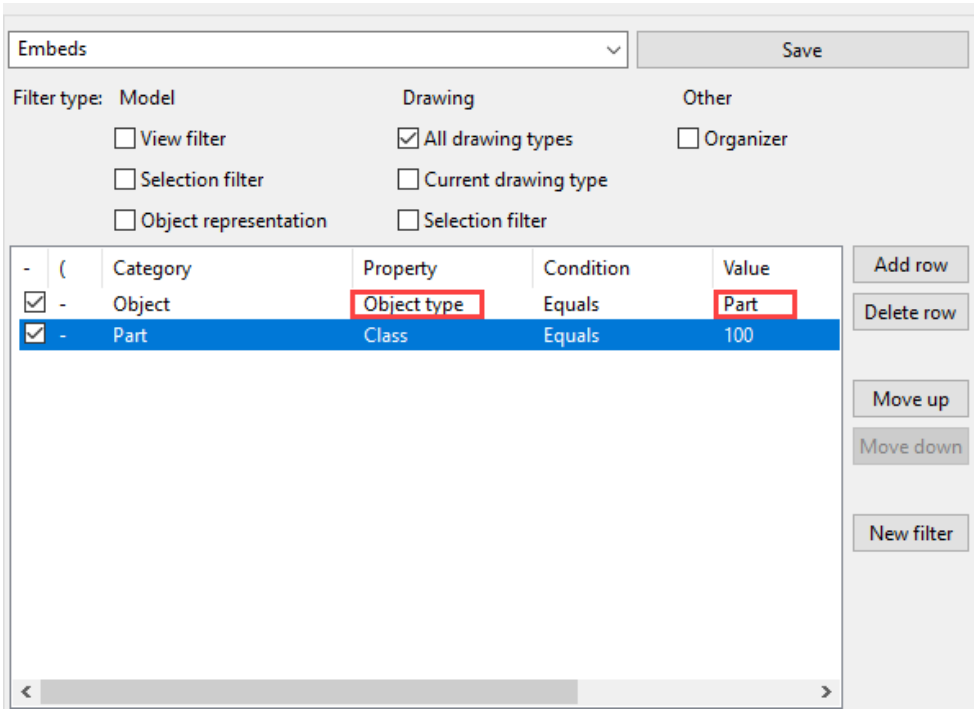
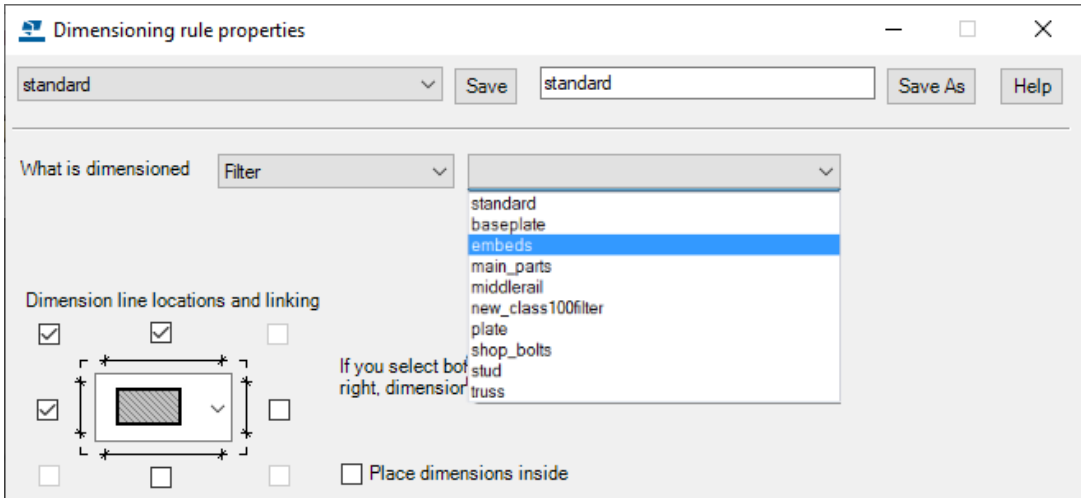
1. In an open drawing, double-click the drawing view frame to open the **View Properties** dialog box.
2. Click **Filter**.
3. Click **Add row** and define the filter properties.
   - Add a row first to define the **Object type** object category. This needs to be defined for all drawing view filters that are going to be used in dimensioning definitions. Set the **Value** to **Part** or **Reinforcing barReinforcing bar**
   - Then add a filter row that selects all parts in a specific class, for example.



4. Use **Save** to save the filter with a unique name.
5. Click **Cancel** to close the dialog box.

Now you can select the drawing view filter from the filter list in the **Dimensioning rule properties** dialog box and use it for dimensioning. If the filter is not displayed immediately after creation, refresh the list by closing and opening the dialog box.



For more information about rule properties, see Dimensioning rule properties (page 942).

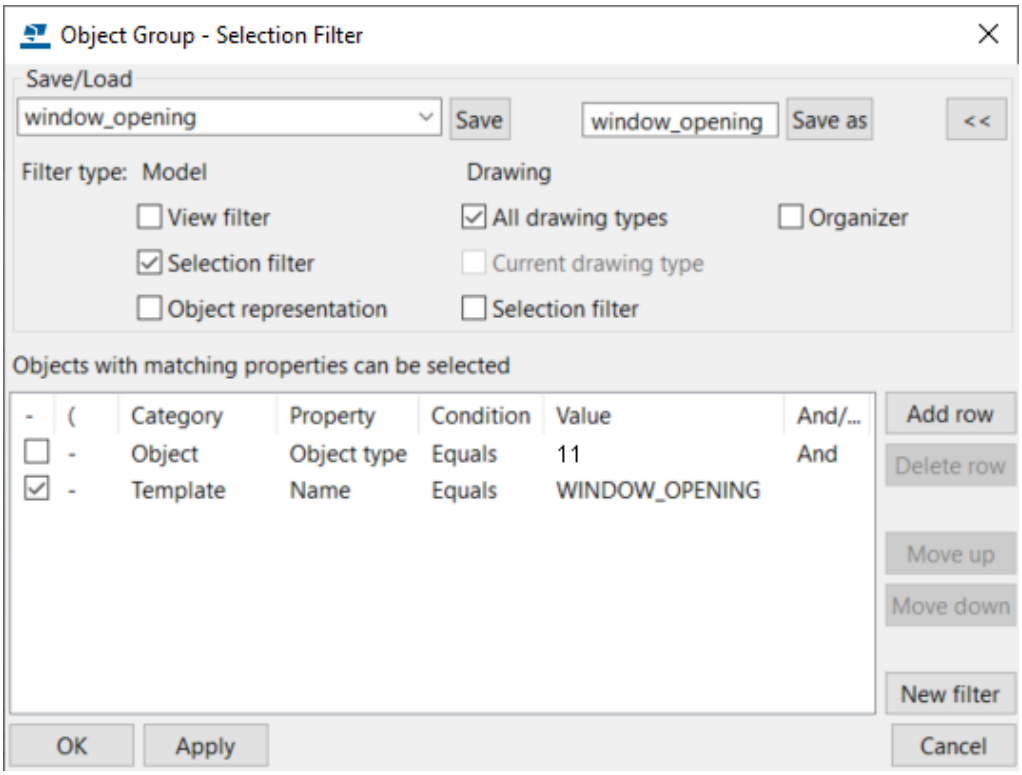
Create a filter for holes and recesses

You can create a filter for cut parts. The filter can be a drawing view filter or a model selection filter.

To create a model selection filter for holes and recesses:

1. Click the **Selection filter** button on the **Selecting** toolbar.
2. In the **Object Group - Selection Filter** dialog box, add the first row:
   - Select **Object** as category, **Object type** as property and **Equals** as condition.
   - To fill 11 in the value, use **Select from model**, and select a cut part.
3. Add the second row:
   - Select **Template** as category, enter `NAME` in uppercase as property and select **Equals** as condition.
   - Enter the name of the template as the value.
4. Select check boxes **Selection filter** and **All drawing types**.

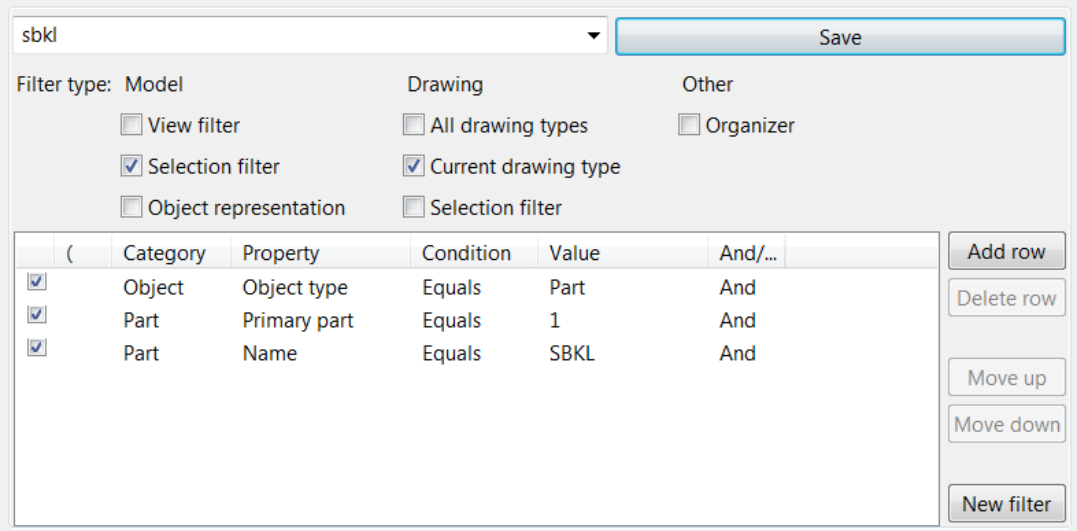
   Too see all available filter types, click the button
5. Save the filter with a unique name using **Save as**.



For more information about creating filters, see Create filters.

Create a filter for the main part of an assembly

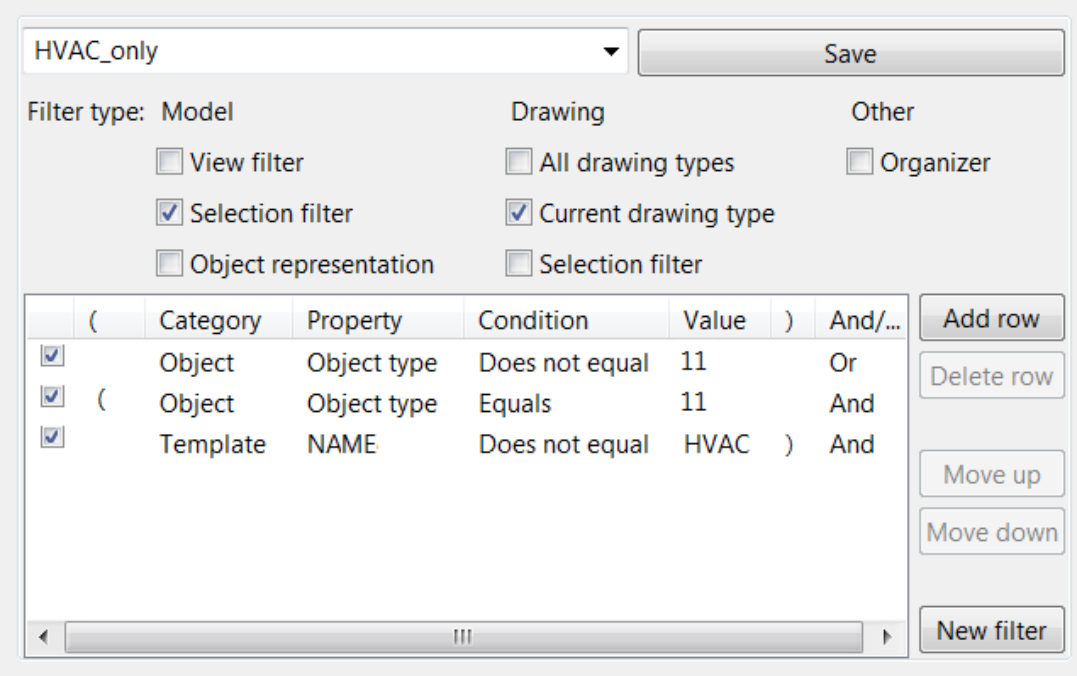
When your subassembly consists of many objects, but you only want to select the main part for dimensioning, you can create a filter for that.



Create an exclude filter for dimension tag

Often several objects need to be excluded from dimension tag content. You can exclude everything else from the tag content, except the parts that you want to include.

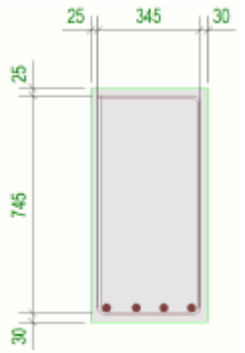
With the example filter shown below it is possible to exclude everything else from the tag content, except cut parts that have the name 'HVAC'.



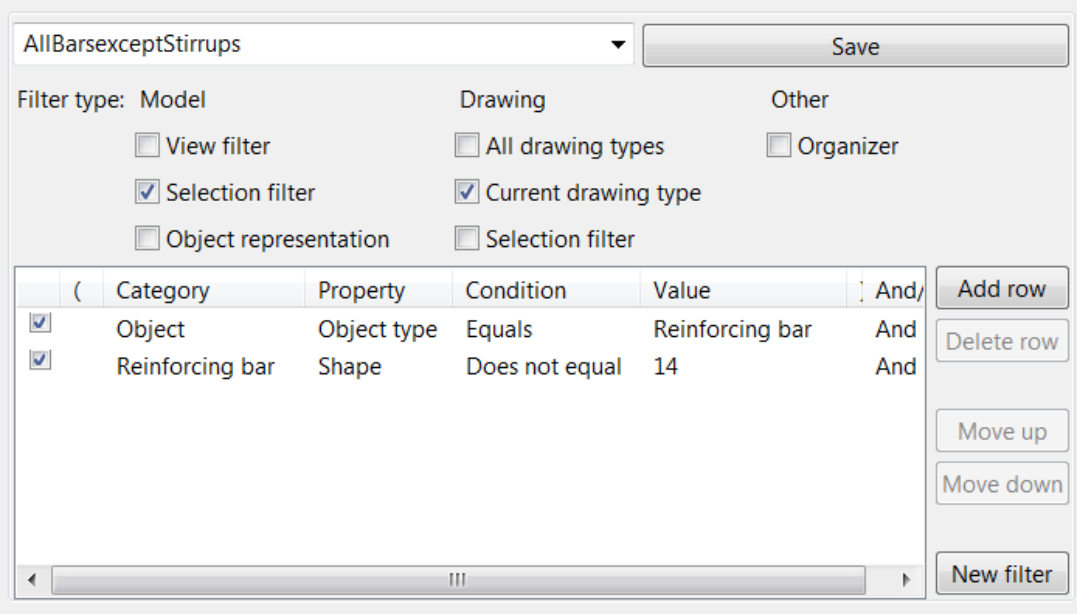
Create an exclude filter for stirrups in section views

The view-level dimensioning currently dimensions reinforcing bar polygon points along the center line of the reinforcing bar. You can filter, where stirrups (shape 14) are excluded but all other reinforcing bars are dimensioned.

Below is an example of a stirrup in section view. In some cases, the resulting dimensions are created similarly regardless of the hook type.



To avoid this kind of results, you can use suitable filtering criteria. See below for an example of a filter, where stirrups (shape 14) are excluded but all other reinforcing bars are dimensioned.



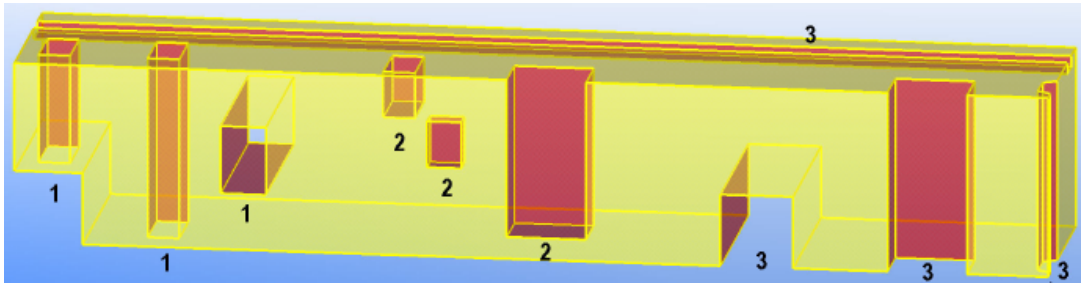
Dimensioning method of shapes, holes and recesses

The logic and functionality of shape dimensions, hole dimensions and recess dimensions in view-level dimensioning is clear and predictable. The definition, which geometry is shape, hole or recess is done only once for the object to be dimensioned, and that definition is used in all drawing views.

The definition of shape and hole is done by looking at the shadow of an object in three directions X, Y and Z in the following way:

- Points along the outer edge of the shadow are dimensioned as a shape.
- Inner loops in the shadow will be dimensioned as holes.
- All other geometry points that are not visible in the shadow, will be dimensioned as recesses.

In the image below there are some examples of each geometry type:

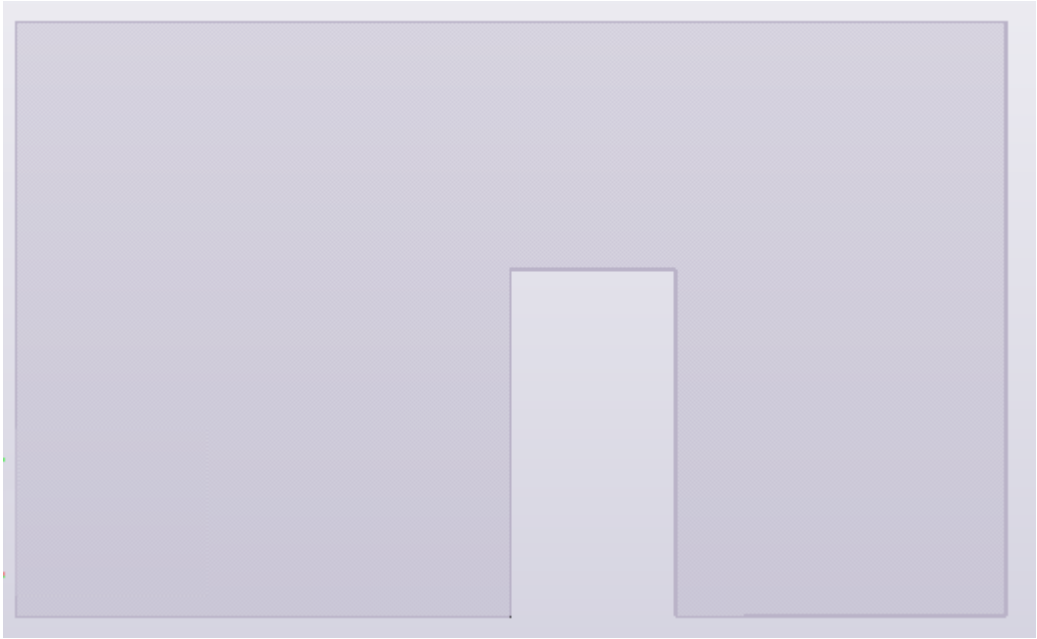


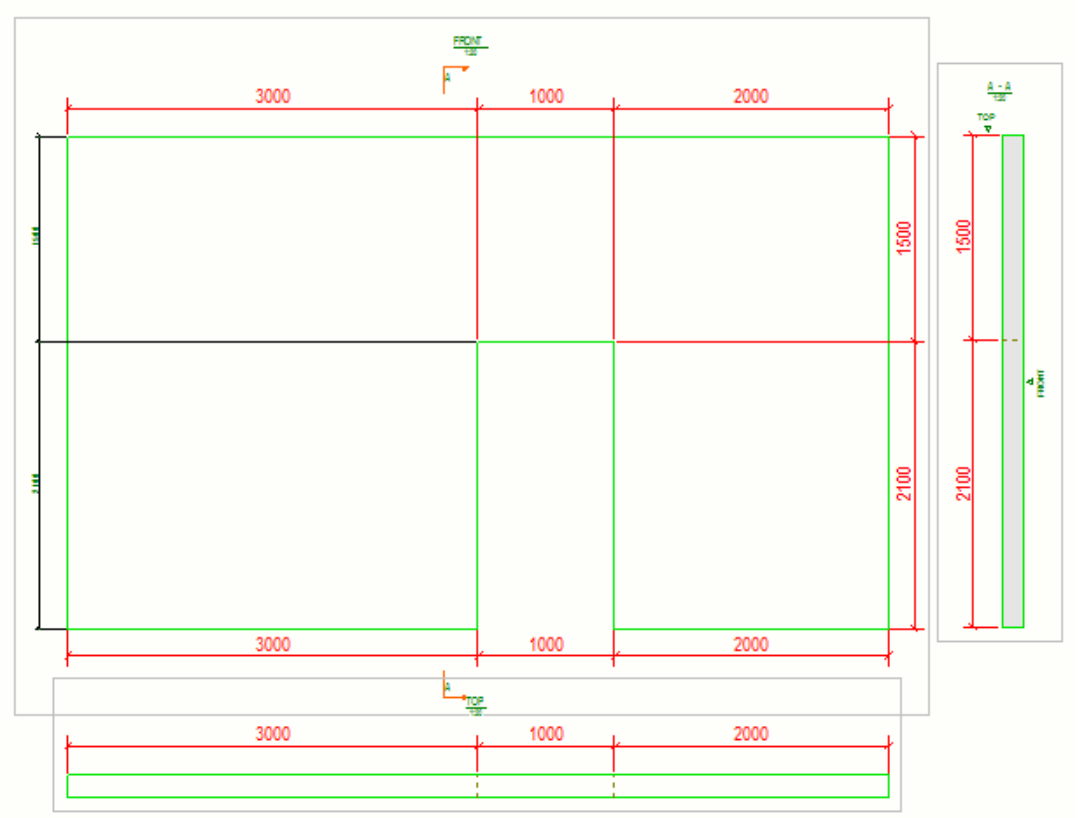
1. Holes

2. Recesses
3. Shapes

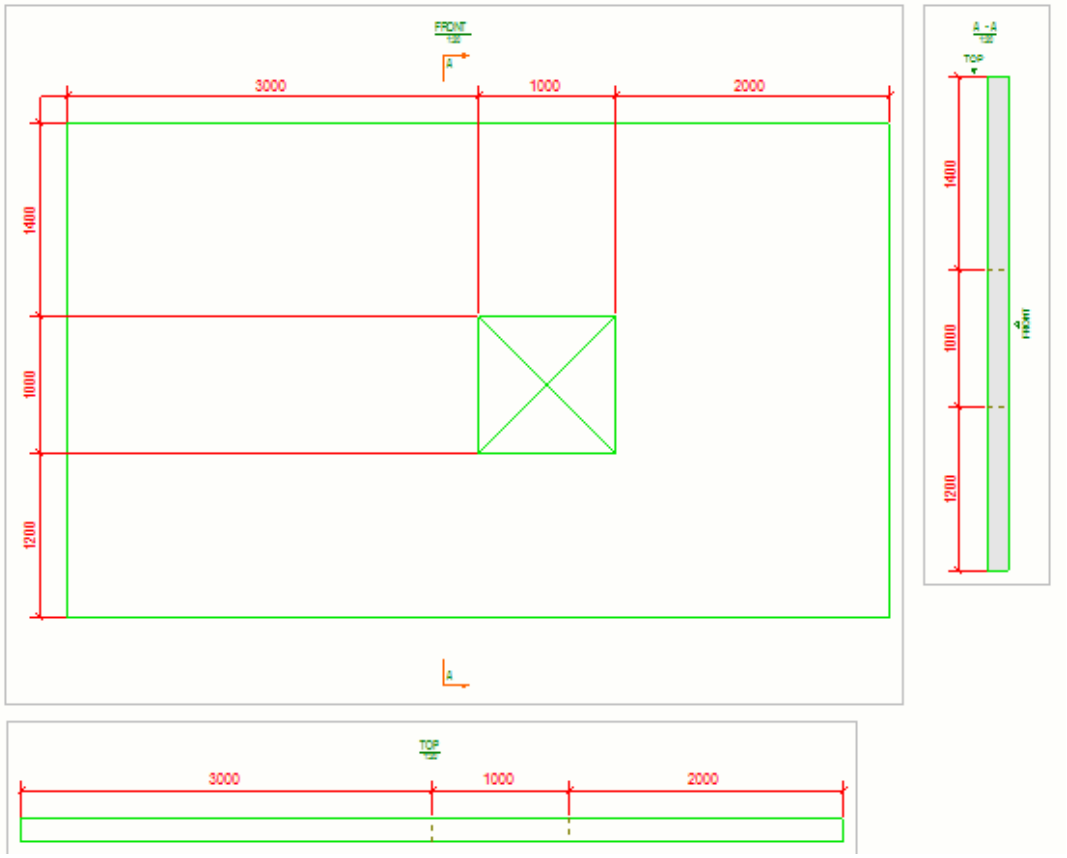
Examples

Below is an example of a shape in a model object and the dimensions in a drawing:

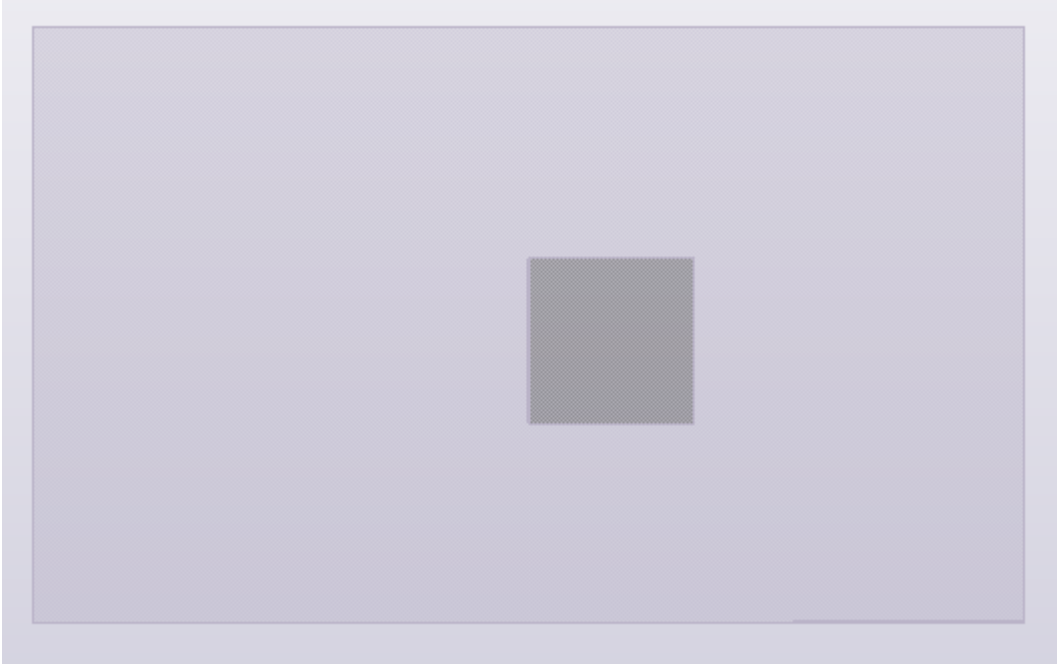


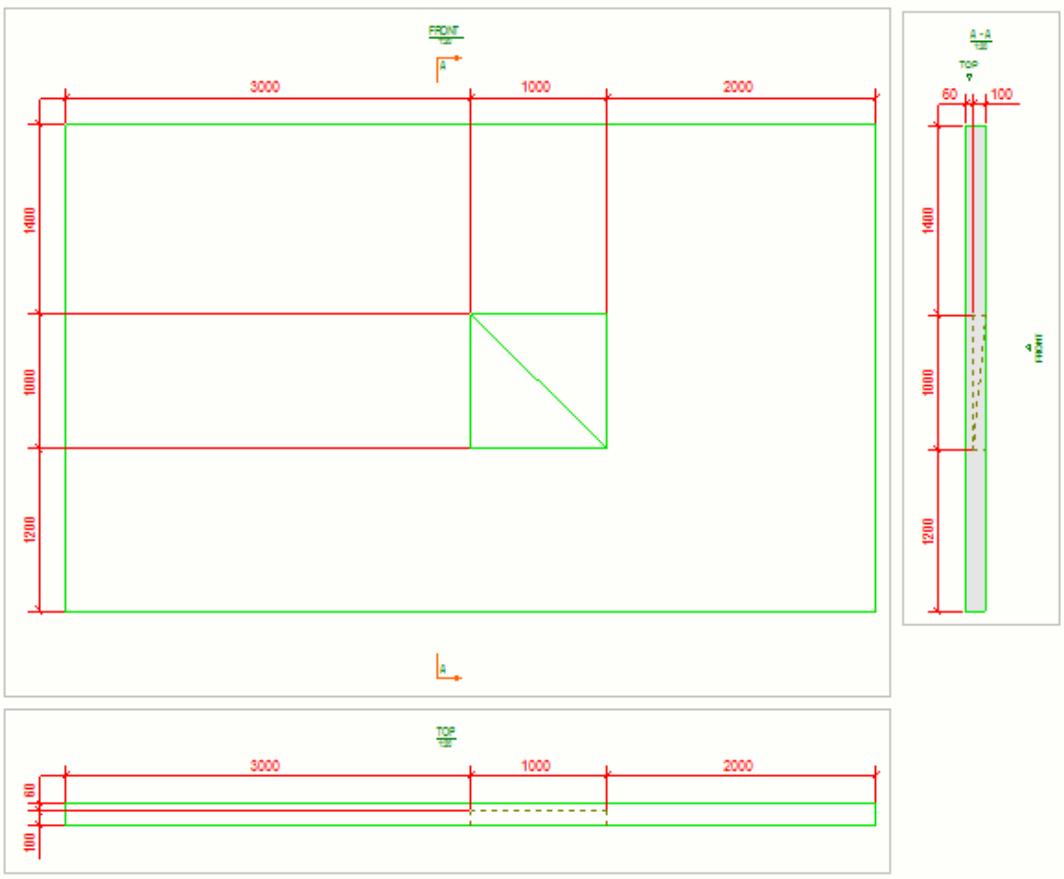


Below is an example of a hole in a model object and the dimensions in a drawing:



Below is an example of a recess in a model object and the dimensions in a drawing:





See also

What are automatic view-level dimensions (page 726)

Dimensioning rule properties (page 942)

Various scenarios of using different dimensioning types

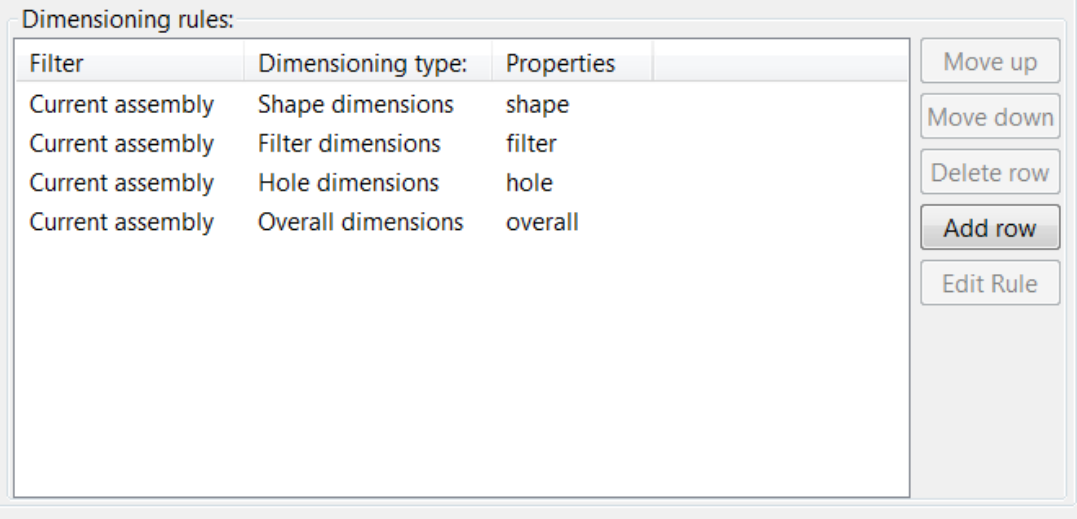
Depending on what you want to dimension and how, you can create different sets of dimensioning rules to get what you need. Have a look at the examples scenarios below.

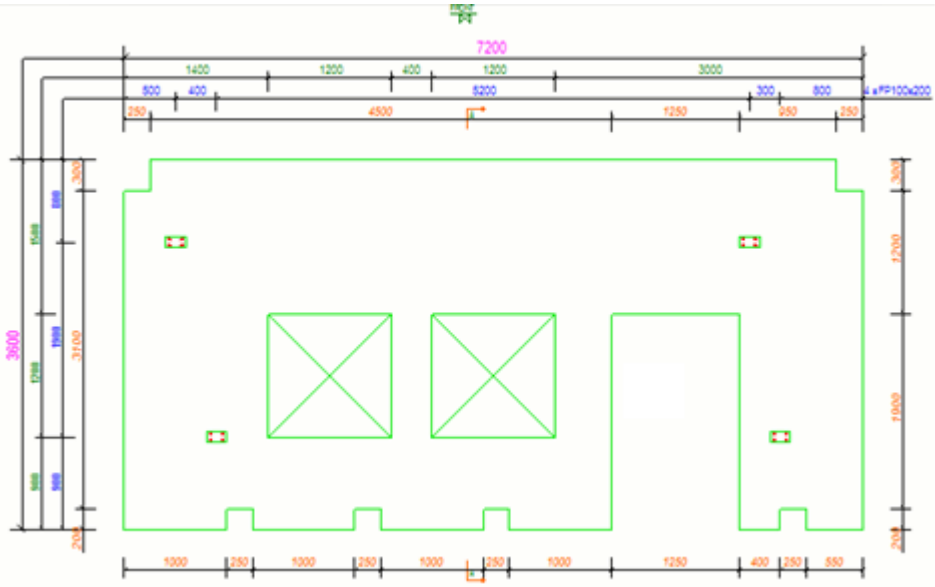
Use view-level dimensioning only

In this scenario, dimension types suitable for precast objects are used only.

Each rule creates one dimension line on selected sides of the cast unit or assembly. In the image below, four rules have been defined and four dimension lines are created. Only the first rule (shape dimensions) is configured to create dimensions on all sides. Other rules are configured to create dimensions only on two sides. Rules are executed in the order they

appear in the list; the topmost rule first, then the second, and so on. The first rule is closest to the part being dimensioned.

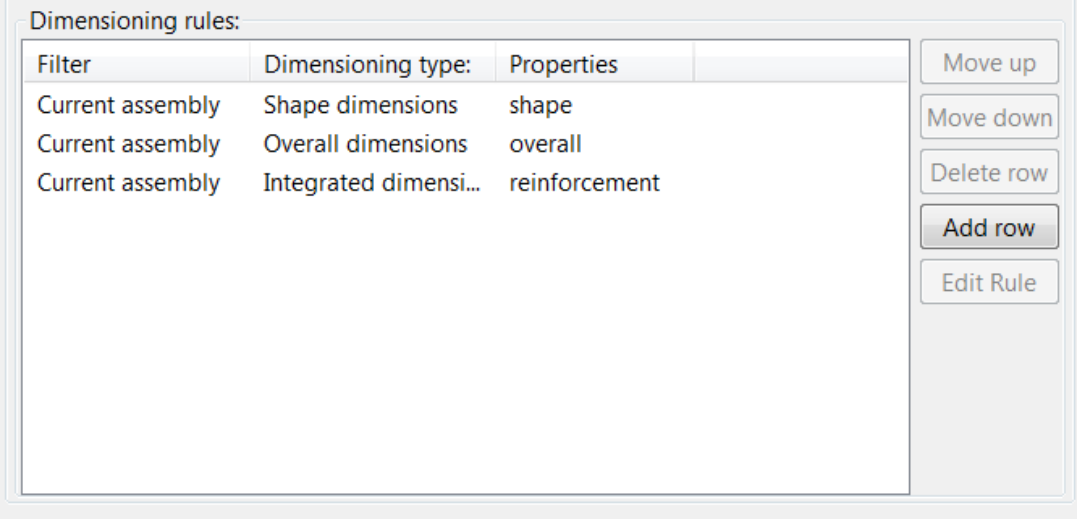


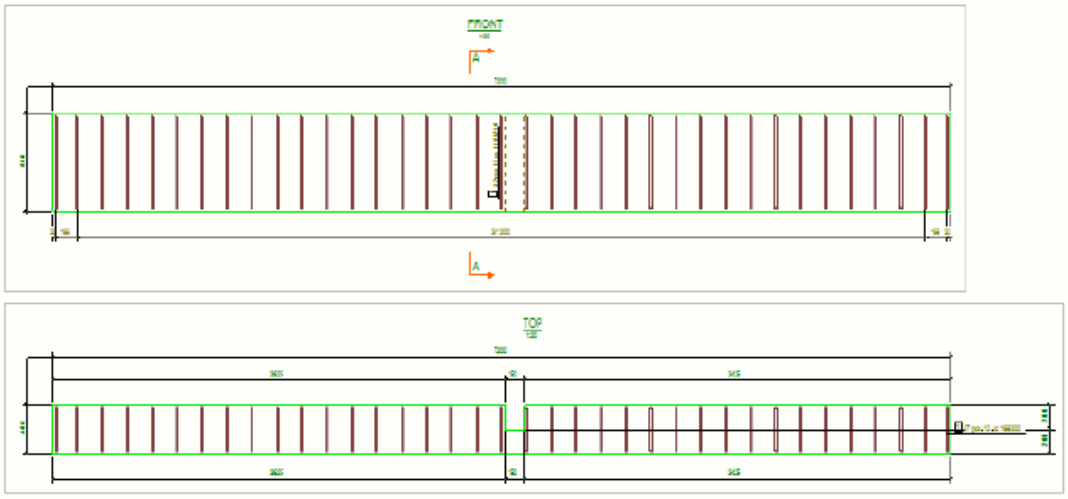


Use both view-level dimensioning and integrated dimensioning

Both view-level and integrated dimensioning methods are used.

In this example, integrated, overall and shape dimensions are created for both top and front view.

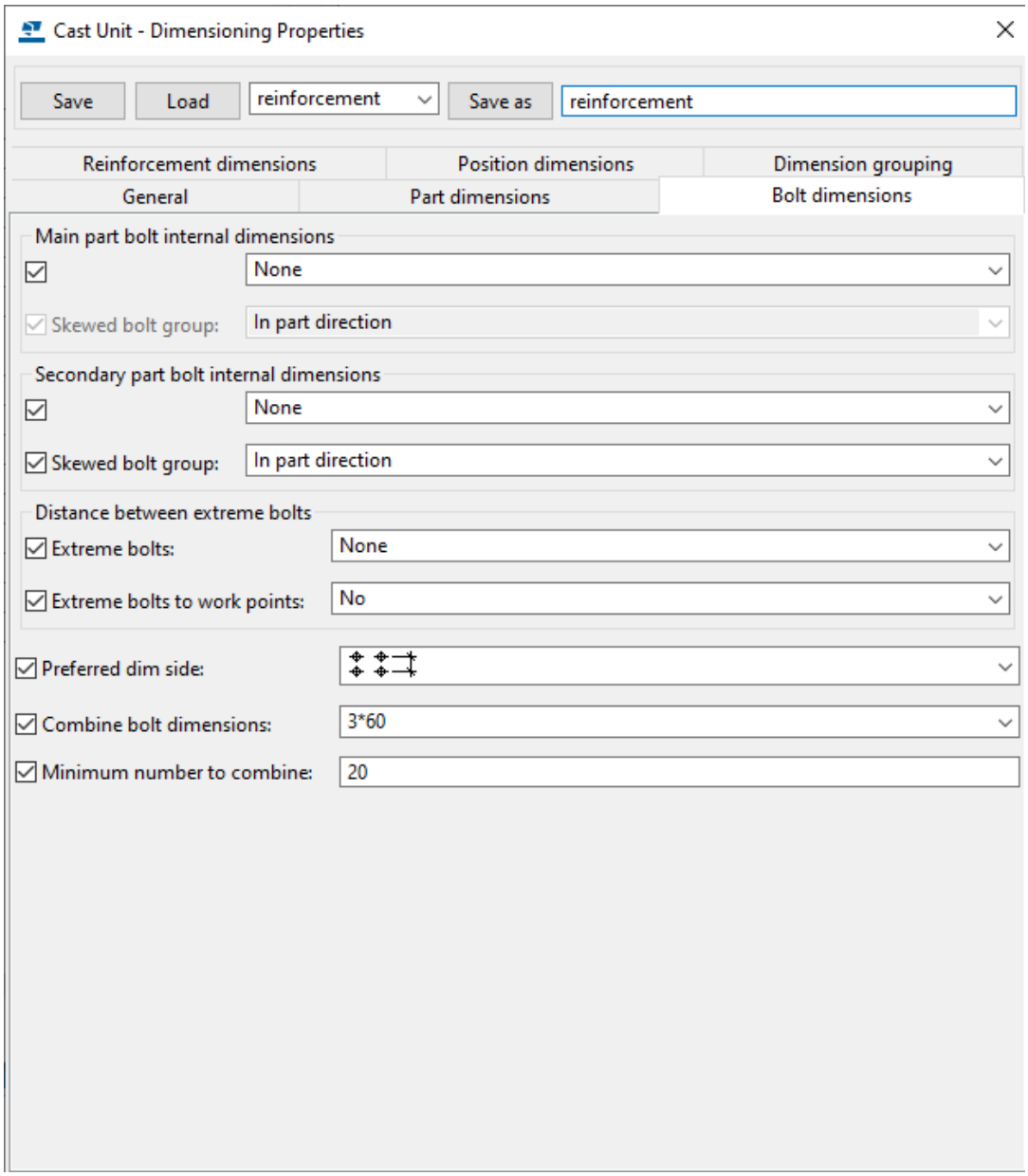




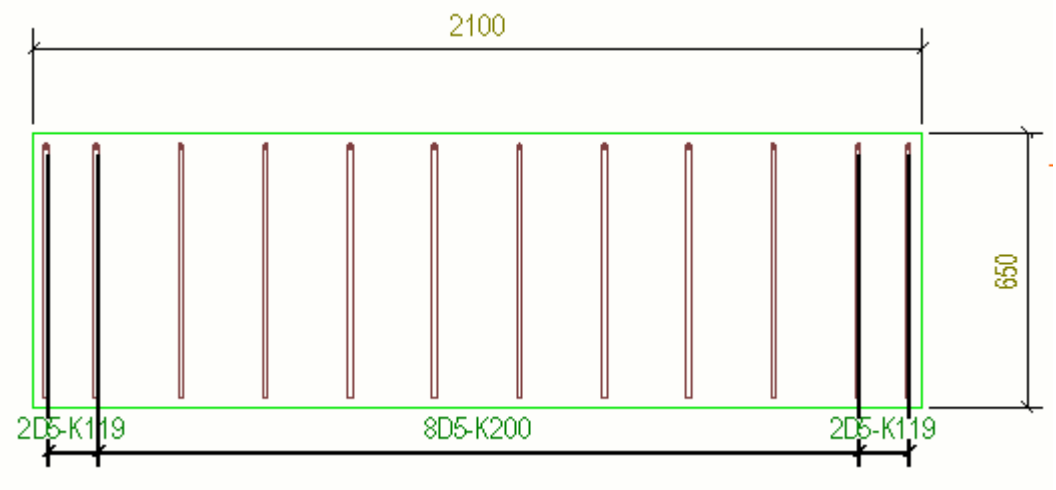
Use integrated dimensioning only

In this scenario, integrated dimensioning is used for creating the reinforcing bar dimensions.

When you select **Integrated dimensions** as the **Dimensioning type** and click **Edit rule** in the **View Dimensioning Rules** dialog box, the **Dimensioning Properties** dialog box is displayed. Go to the **Reinforcement dimensions** tab and define the desired properties. On the **General** tab there are some options for controlling dimension properties, and you can load object level dimension properties for various dimension types. Save the changes with **Save** or **Save as** to save the properties file for later use.



When you return to the **View Dimensioning Rules** dialog box by clicking **Close**, you can attached the new integrated dimensioning properties to the **Integrated dimensioning** rule.



Add automatic view-level dimensions using Integrated dimensioning type

In the **Dimensioning** dialog box, you can control what gets dimensioned and how in a drawing. You can experiment with different combinations of options to achieve different kinds of dimensioning effects.

Add dimensions using Integrated dimensioning type

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. Click **Dimensioning**.
5. Select **Integrated dimensions** as the dimensioning type, select the desired dimensioning rule properties, and click **Edit rule**.
6. Select the dimensions to create and adjust the related settings.

   The available tabs and settings depend on the drawing type:

   - On the **Part dimensions** tab, select the part dimensions to create and adjust the related settings (Part dimensions tab (page 959)).
   - On the **General** tab, adjust settings related to minimizing the number of views, dimension type, combining dimensions, closing dimensions, limit for dimensioning asymmetry in secondary parts, forward offset, grid dimensions, dimension position, and part marks on dimension line (General tab (page 959)).

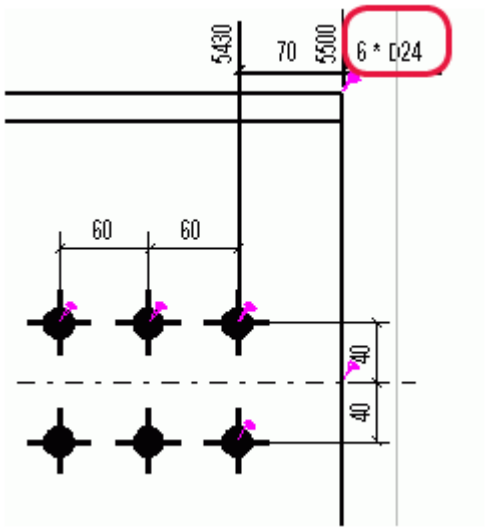
- On the **Position dimensions** tab, select the position dimensions to create. They indicate the position of parts in relation to the main part or to work points (Position dimensions tab (page 959)).
- On the **Bolt dimensions** tab, select the bolt dimensions to create, combine bolt dimensions and select the side for the dimension (Bolt dimensions tab (page 959)).
- On the **Dimension grouping** tab, group dimensions and adjust the related settings (Dimension grouping tab (page 959)).
- On the **Sub-assemblies** tab, create dimensions for parts in sub-assemblies and adjust the related settings (Sub-assemblies tab (page 959)).
- On the **Reinforcement dimensions** tab, create dimensions for reinforcing bar groups in cast unit drawings, add dimension marks and adjust the related settings (Reinforcement dimensions tab (page 959)).

7. Save the dimensioning settings by clicking **Save** and close the dialog box by clicking **Close**.
8. Save the view properties by clicking **Save** and return to the drawing properties dialog box by clicking **Close**.
9. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

Group identical objects to the same dimension line

You can group identical parts, bolts, components, and cuts or shapes to the same dimension line in integrated dimensioning. You also have the option to include automatic dimension tags to grouped dimensions.

1. In the **Dimensioning Properties** dialog box, go to the **Dimension grouping** tab.
2. In **Activate dimension grouping**, select the objects that you want to group.
3. Highlight a row (**Parts**, **Bolts**, **Components** or **Cuts/Shapes** in the **Activate dimension grouping** list and select the elements by which you define the identical conditions in **grouping properties**.
4. In **Automatic tagging**, select the appropriate options to include automatic dimension tags.
5. If you want Tekla Structures to update dimension grouping automatically, set the option **Update grouping when model changes** to **Yes**.
6. Click **OK**.



TIP You can change the dimension tag contents (page 260) in the final drawing and include some other elements in the tag.

Add elevation dimensions

You can add elevation dimensions (level marks) in your drawings for the start and end points of parts in integrated dimensioning. Tekla Structures dimensions elevations relative to a reference point, which you can change.

For example, if the elevation is 5000 mm, and you set the reference point to 200, the elevation changes to 4800 mm. You can also change the elevation dimension prefix, which in the English version is **EL** by default.

You can also add level marks manually (page 342).

To change the reference point and create elevation dimensions using another prefix:

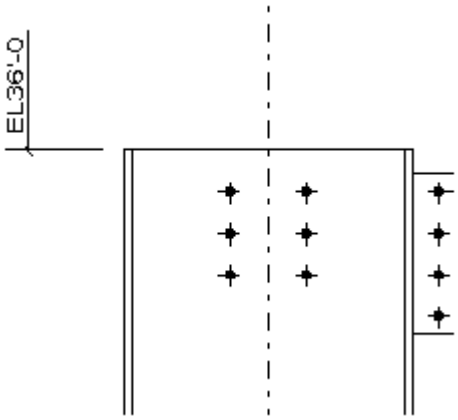
1. Go to the **Position dimensions** tab, and set **Elevation dimensions** to **On**.
2. Click **OK**.
3. In the drawing properties dialog box, click **View** --> **Attributes 2**.
4. Do one of the following:
   - To use a specific value, select **Specified** from the **Datum point for elevations** list and enter a value in the **Datum level** box.
   - To measure elevations relative to the view plane, select view plane from the **Datum point for elevations** list.
5. Save the drawing properties and create the drawing.
6. To change the prefix, open the `dim_operation.ail` file in a text editor that supports UTF-8 coding. Recommended editors are Visual Studio and Notepad++.

This file is located in the `Tekla Structures/<version>/messages/` folder. Replace **EL** with a new prefix on the following line in the file:

```
string dim_operation_dim_elevation_prefix{entry = ("enu",
"EL");};
```

NOTE Shortening value added in the user-defined properties of a part affects also elevation dimensions.

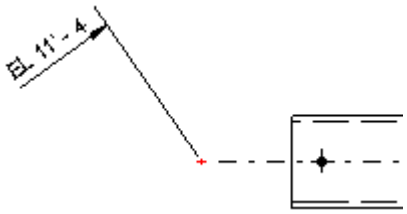
Example



Limitations

Tekla Structures creates elevation dimensions for skew parts only if the parts are in the same position in the drawing as they are in the model. This means that the coordinate system must be set to **model**.

If you are using **local**, **oriented**, or **brace** coordinate systems, Tekla Structures does not draw the elevation dimensions for skewed parts by default. If you want to create elevation dimensions, set the advanced option XS_DRAW_SKEWED_ELEVATIONS to `TRUE` in **File** --> **Settings** --> **Advanced options** --> **Dimensioning - parts** . See below for an example of an elevation dimension for a skew part.



For more information about part orientation, see Define part orientation in drawing views (page 697).

Create check dimensions

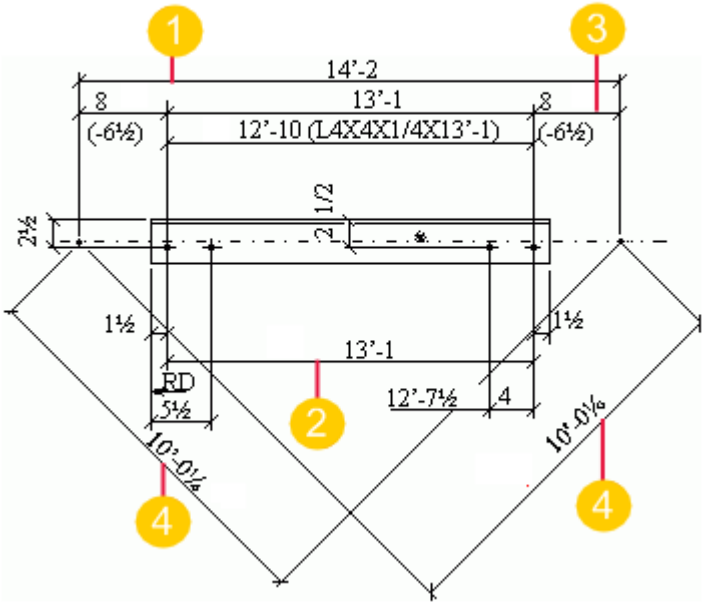
In integrated dimensioning, you can create additional dimensions, check dimensions, to check the accuracy of dimensions. Check dimensions are

usually in thinner text than other dimensions. They are not required for fabrication or erection, and they are mainly used for checking detailing, not for assembling parts.

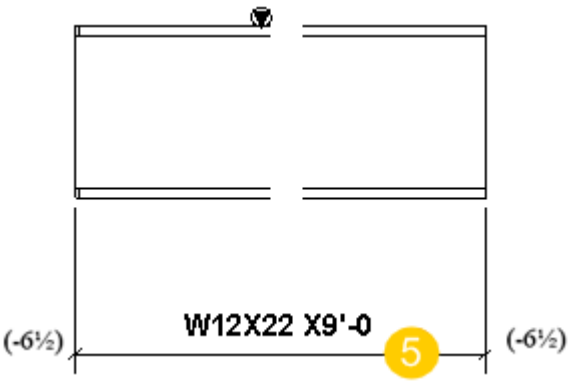
Tekla Structures uses work points to create check dimensions. Work points can be either the points between which the part was originally created, or the intersection point of the reference lines of the parts. The reference line location depends on the part position **At depth** set in part properties. If it is **Middle**, the reference line is the center line, if it is **Front**, the line is located in the part front edge, etc.

Knock-off dimension is a special type of check dimension that dimensions the distance from work points to the end of the part.

Check dimensions:



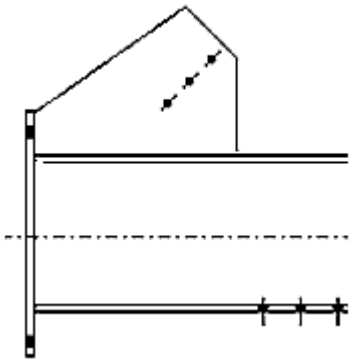
Knock-off dimensions:

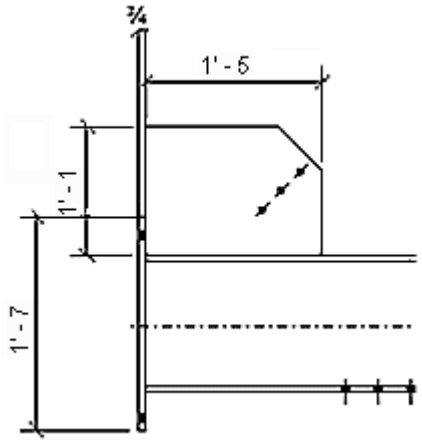
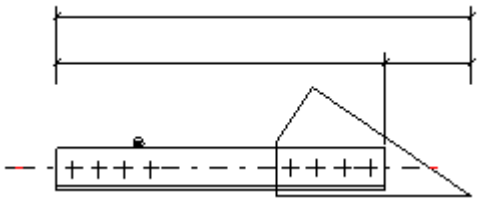
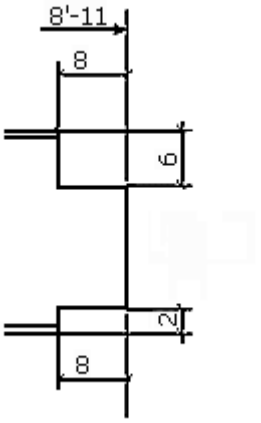


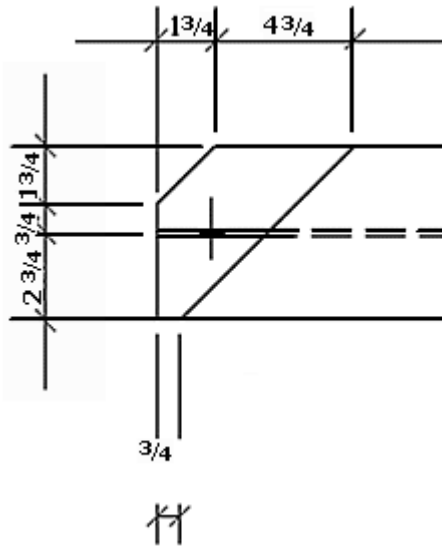
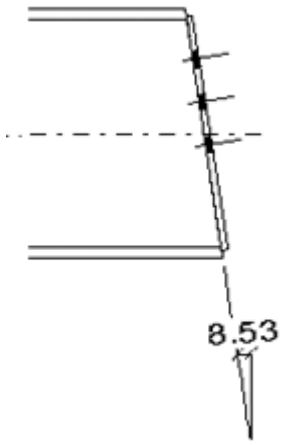
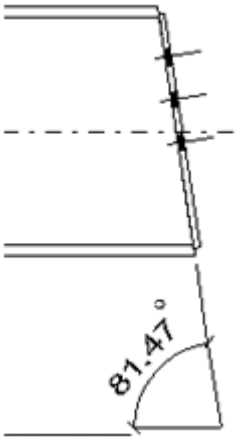
To	Do this	Number
Create check dimensions between outermost work points.	In the **Dimensioning** dialog box, click **Part dimensions** and set **Main part work points** to **Yes**.	**(1)**
Create check dimensions between outermost bolts.	In the **Dimensioning** dialog box, click **Bolt dimensions** and set **Extreme bolts** to **Assembly** or **Main part**.	**(2)**
Create check dimensions from outermost work point to first bolt.	In the **Dimensioning** dialog box, click **Bolt dimensions** and set **Extreme bolts** to **Assembly** or **Main part** and **Extreme bolts to work points** to **Yes**.	**(3)**
Create horizontal and vertical check dimensions between the work points in a skewed brace.	In the **Dimensioning** dialog box, click **Position dimensions** and set **Main part skew position** to **Yes**.	**(4)**
Create check dimensions between the work points, such as the intersections of main and neighbor part reference lines.	In the **Dimensioning** dialog box, click **Position dimensions** and set **Position bolts to** or **Position parts to Working points** or **Both**.	
Create check dimensions to the bolt hole locations in the main part.	In the **Dimensioning** dialog box, click **Position dimensions** and set **Main part bolt position** to **On**.	
Create knock-off dimensions.	In the **Dimensioning** dialog box, click **Part dimensions** and set **Knock-off dimensions** to **On**.	**(5)**

Example: Part dimensioning

Here are some examples of what the part dimensions look like in integrated dimensioning with different settings selected on the **Part dimensions** tab.

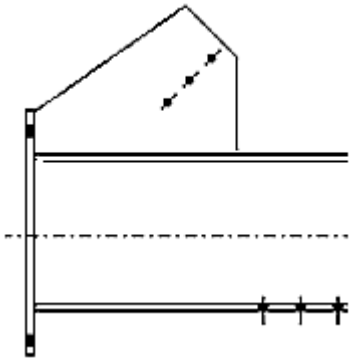
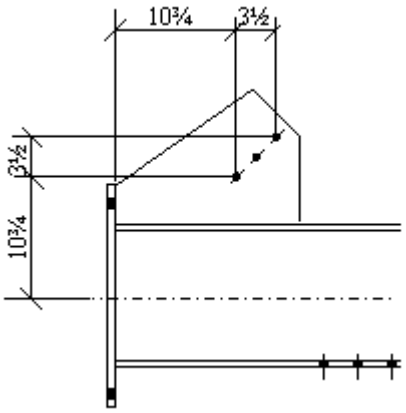
Dimensioning setting	Example
Internal dimensions set to **None**	

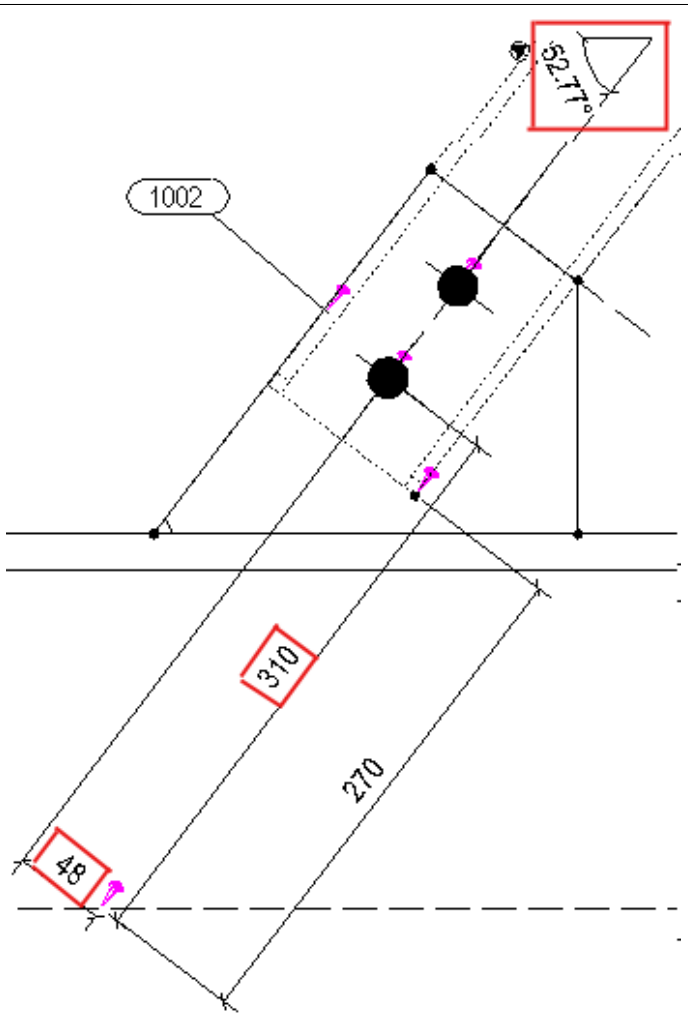
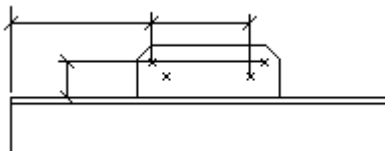
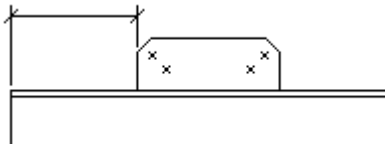
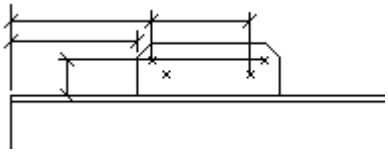
Dimensioning setting	Example
Internal dimensions set to **All**.	
Overall dimensions	
Main part shape (**Shape dimensions**) set to **On**.	

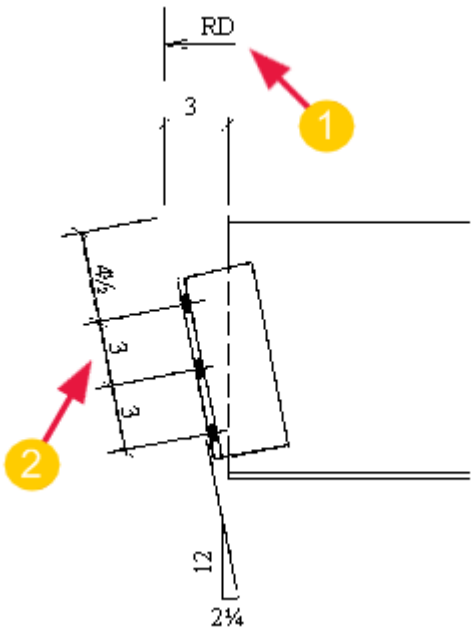
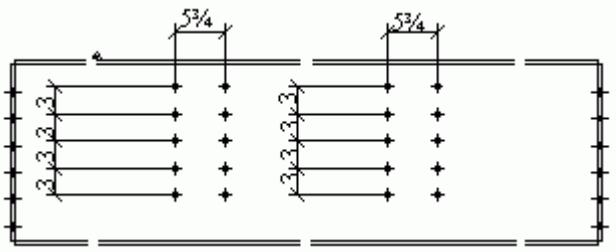
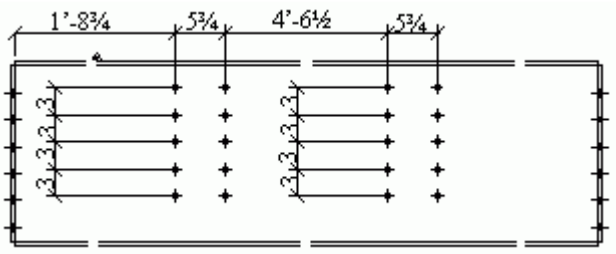
Dimensioning setting	Example
Bevel dimensions set to **On**.	
Bevel angle set to **Angle of cut**.	
Bevel angle set to **Angle of beam**.	

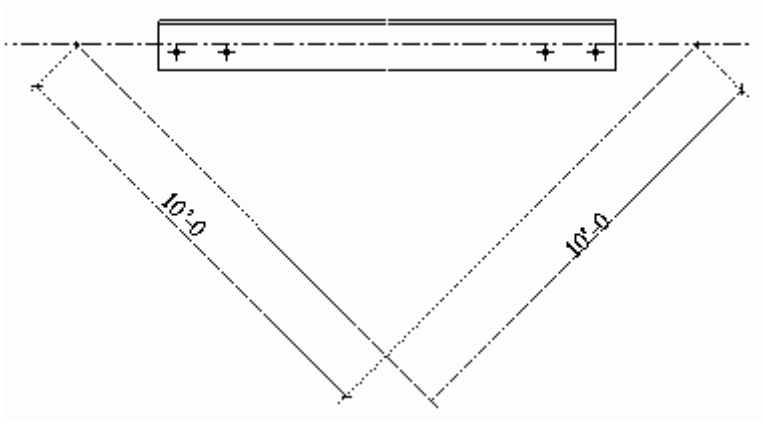
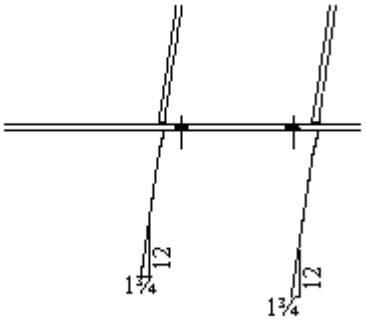
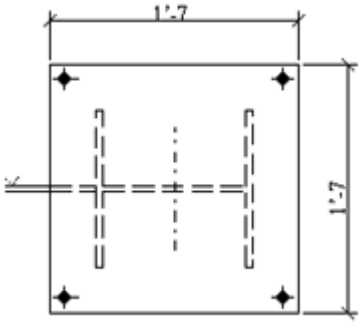
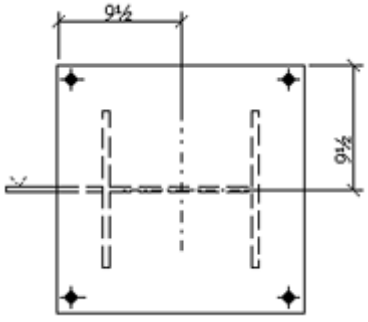
Example: Position dimensioning

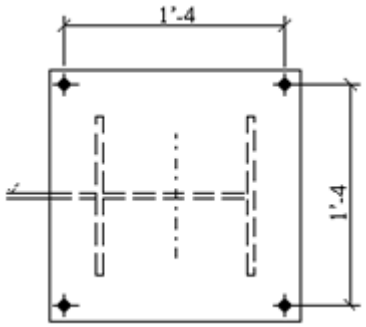
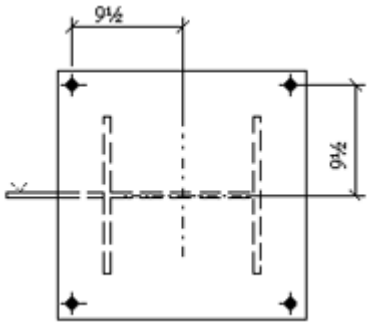
Here are some examples of what the position dimensions look like in integrated dimensoning with different settings selected on the **Position dimensions** tab.

Dimensioning setting	Example
Position parts to is set to **None**.	
Position parts to is set to **Main part**.	

Dimensioning setting	Example
Position bolts to is set to **Working points**.	
Secondary part is dimensioned **By bolt**.	
Secondary part is dimensioned **By part**.	
Secondary part is dimensioned **By both**.	

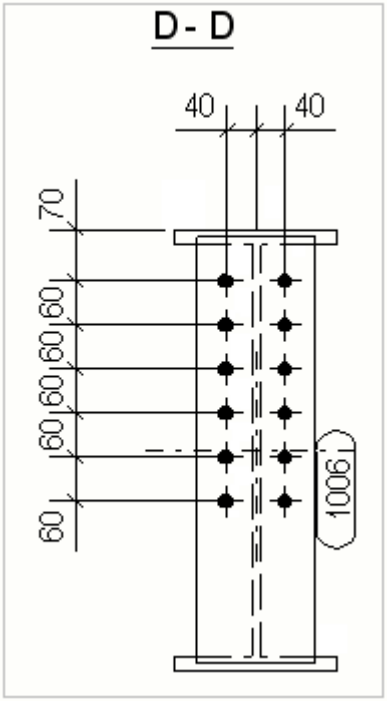
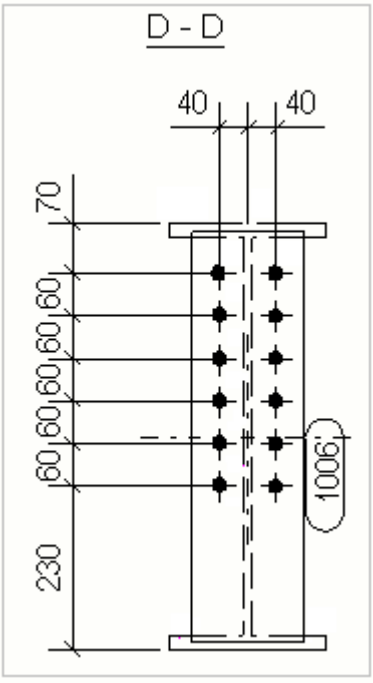
Dimensioning setting	Example
Secondary part is dimensioned **By bolt**. **Secondary part dimension direction** is **Neighbor part**. **Position from** is set to **Work point**.	 **(1)** Running dimensions start from the intersection of the main and secondary part (=work point) **(2)** Dimensions are aligned with the neighboring part
Main part bolt position is set to **Off**. (**Main part bolt internal dimensions** is set to **Internal** on the **Bolt dimensions** tab.)	
Main part bolt position is set to **On**. (**Main part bolt internal dimensions** is set to **Internal** on the **Bolt dimensions** tab.)	 By default, create minimum and maximum position dimensions are not created for bolts. For information on how to create these dimensions, see Add minimum and maximum position dimensions to bolts (page 781).

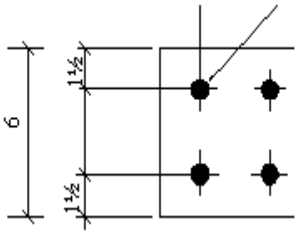
Dimensioning setting	Example
Main part skew position is set to **Yes**.	
Skew position is set to **angle**.	
Centered part is set to **Internal**.	
Centered part is set to **Position**.	

Dimensioning setting	Example
Centered bolt is set to **Internal**.	
Centered bolt is set to **Position**.	

Example: Closing dimension

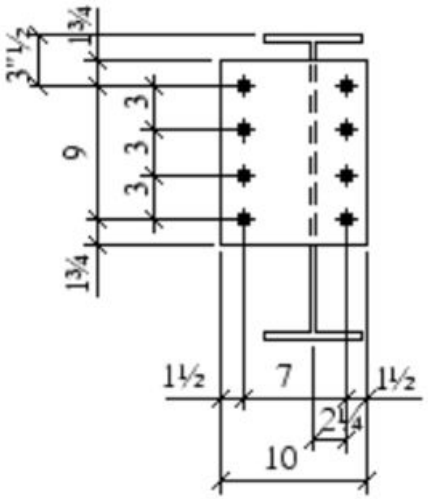
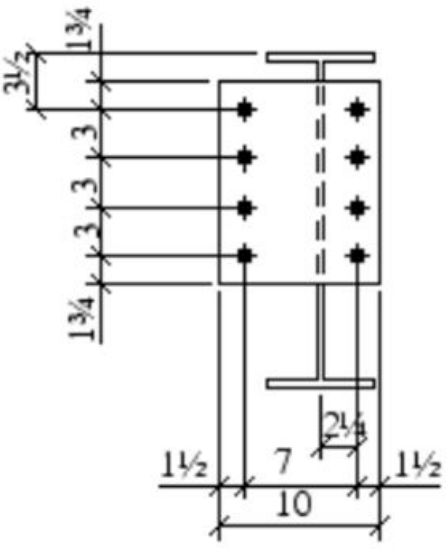
Here are some examples of how Tekla Structures creates dimensions in integrated dimensioning with different options selected in the **Close dimensions** area on the **General** tab.

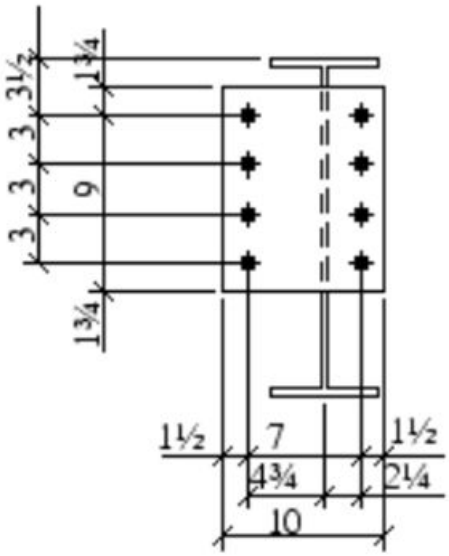
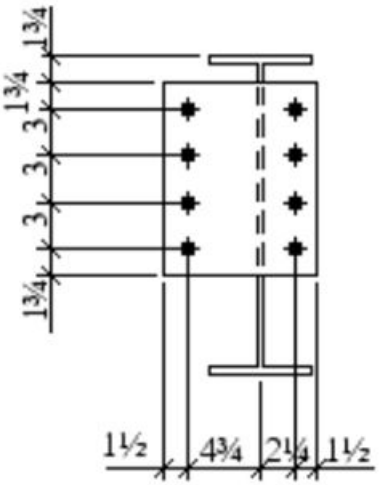
Closing option	Example
Close dimensions is set to **No**.	
Close dimensions is set to **All**.	

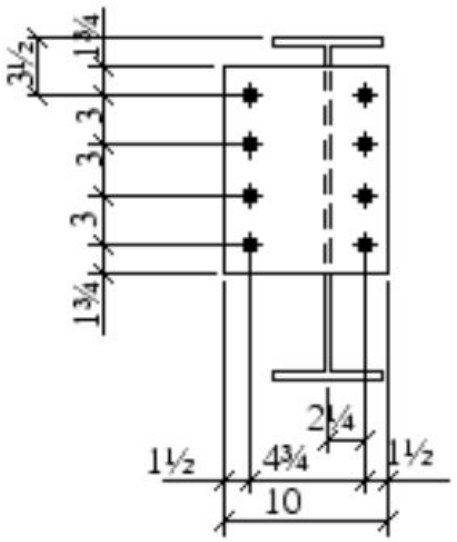
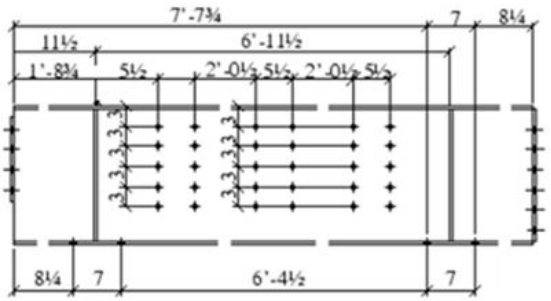
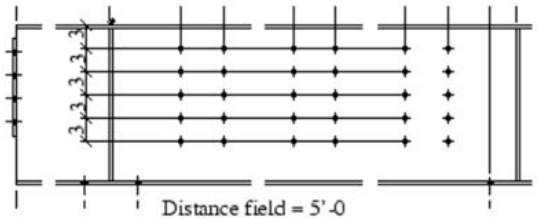
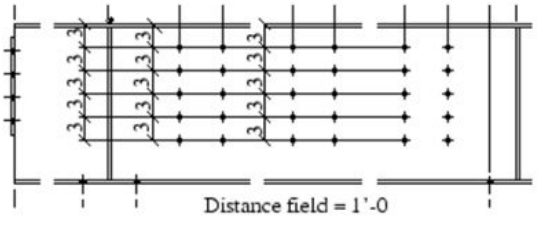
Closing option	Example
Short dimensions is set to **No**.	

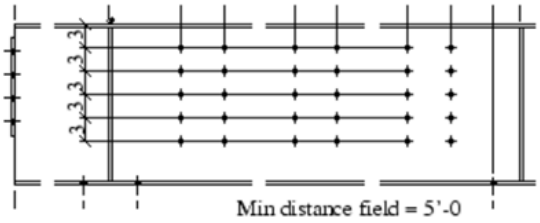
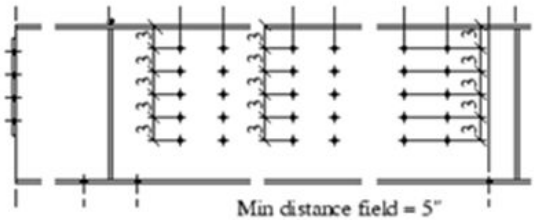
Example: Combine dimensions

Here are some examples of how Tekla Structures combines dimensions in integrated dimensioning with different options selected on the **General** tab.

Combining option	Example
Option **No** prevents dimensions from being combined.	
Option **1** combines part position dimensions with part internal dimensions, and bolt group internal dimensions with bolt edge distances. Bolt position dimensions are not combined with bolt internal dimensions.	

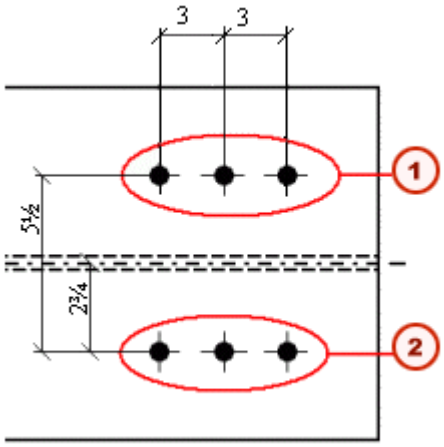
Combining option	Example
Option **2** combines the part position dimension with part internal dimensions and bolt group internal dimensions. Bolt internal dimensions are combined with bolt position dimensions. Edge distances are shown separately.	
Option **3** combines bolt internal dimensions and position dimensions in the same dimension line.	

Combining option	Example
Option **4** combines bolt group position dimensions with part position dimensions. Part and bolt internal dimensions are not combined with this option, but bolt internal dimensions are combined with bolt edge distances.	
Option **5** combines internal dimensions and the position dimension of bolt groups where there are several bolt groups.	
Option **4.5** uses a combination of option **5** for the main part and option **4** for the secondary parts.	
Distance 5’-0	
Distance 1’-0	

Combining option	Example
Min distance 5’-0	
Min distance 5"	

Example: Combine bolt group dimensions

For dimensioning and marking purposes, Tekla Structures treats bolt groups located close together in integrated dimensioning as one group on the basis of the minimum number of dimensions to combine and format selected on the **Bolt dimensions** tab.

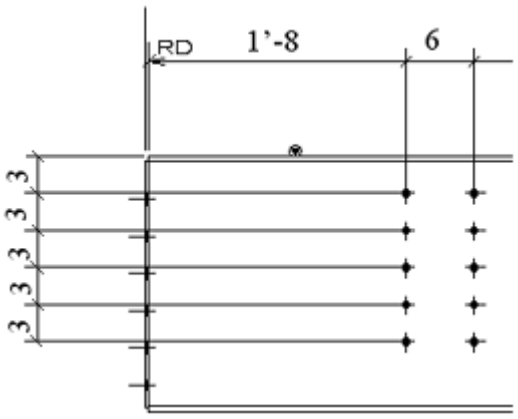
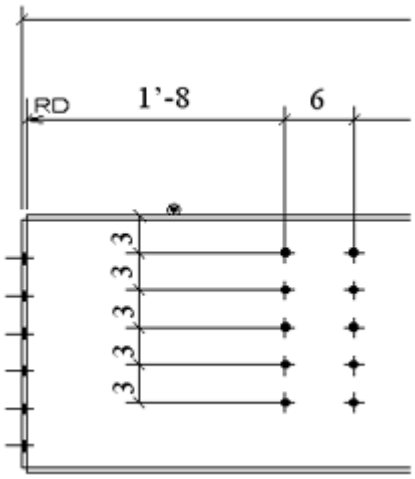


(1) Bolt group 1

(2) Bolt group 2

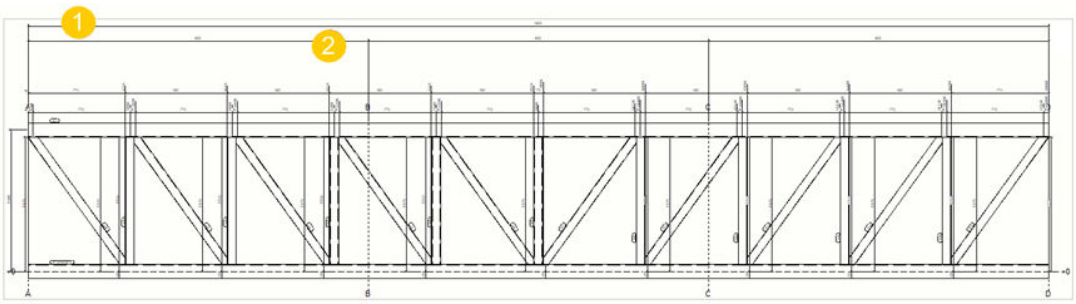
Example: Forward offset

Here are some examples of how Tekla Structures places dimensions in integrated dimensioning with different forward offset values set on the **General** tab.

Forward offset setting	Example
Forward offset greater than the 1’-8 dimension to the hole group.	
Forward offset set to a smaller value.	

Example: Grid dimensions

Here is an example of how Tekla Structures creates dimensions in integrated dimensioning with different options selected in the **Grid dimensions** area on the **General** tab.



(1) Option **Overall** selected

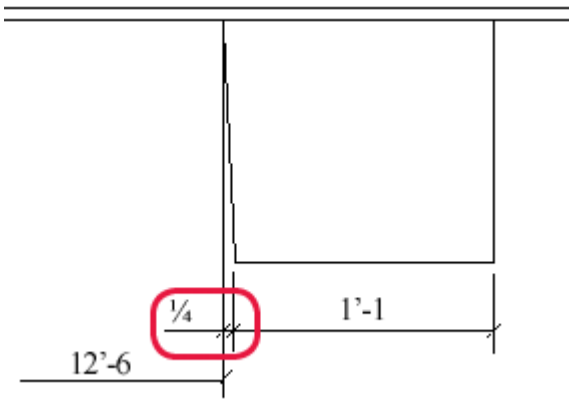
(2) Option **Individual spans** selected

Example: Recognizable distance

Here is an example of how Tekla Structures uses the **Recognizable distance** setting in integrated dimensioning. If you set value for **Recognizable distance** on the **General** tab, and the asymmetry of the parts is smaller than the distance you entered, a dimension is added.

This setting is used, when the **Internal** dimension is set to **Necessary**. Recognizable distance dimension is not necessarily needed if the part can be assembled correctly without it.

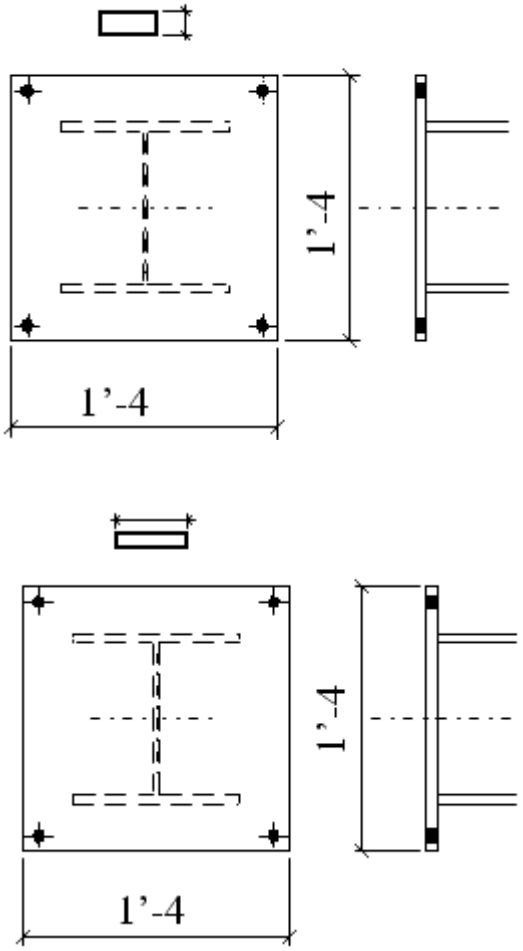
A typical example is a rectangle that is almost as long as it is wide.



Example: Preferred dimension side

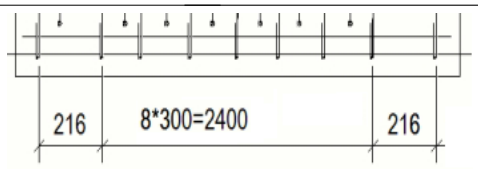
You can set the preferred dimension side for parts and bolts on the **Part dimensions** tab and **Bolt dimensions** tab in integrated dimensioning. The

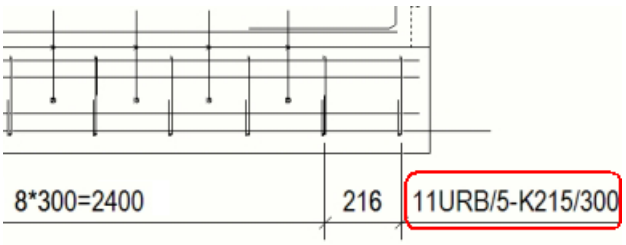
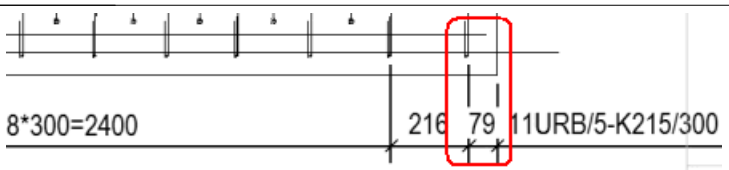
examples below show how the different settings for **Preferred dim side** look like for part dimensions.



Example: Reinforcement dimensions

Here are some examples of how Tekla Structures creates dimensions for reinforcing bar groups in integrated dimensioning with different options selected on the **Reinforcement dimensions** tab.

Settings	Example
Dimensions for reinforcing bar groups is set to **On**, no dimension tags speci ied in dimension properties.	

Settings	Example
Dimensions for reinforcing bar groups is set to **On**, dimension tags speci ied in dimension properties.	
Dimensions for reinforcing bar groups is set to **On**, dimension tags speci ied in dimension properties, closing dimensions added to the edge of the part in dimension properties (**Part edge** set to **Yes**).	

Add automatic dual dimensions

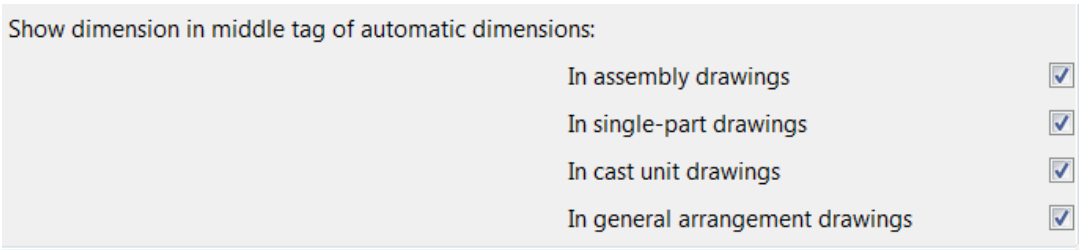
You can create dual dimension tags automatically in all types of drawings.

Limitations:

Dual dimensions can only be shown in relative and US absolute dimensions, but not in absolute dimensions.

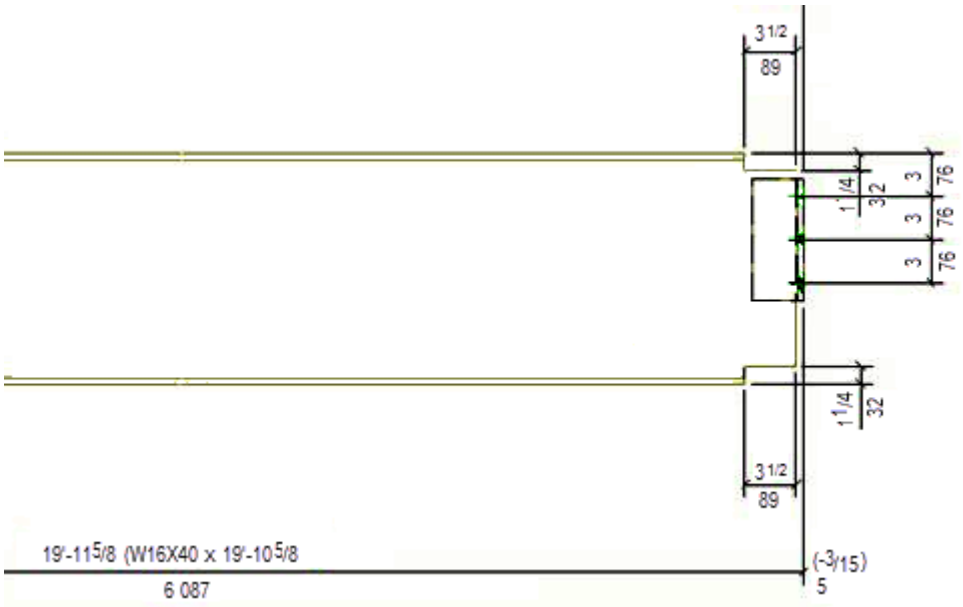
1. Click On the **File menu**, click **Settings** --> **Options** and go to **Drawing dimensions** settings.
2. Set the units, format and precision.
3. Select the drawing types where you want to have the dual dimensions.
4. Click **OK**.

When Tekla Structures creates the drawing, it adds in the lower dimension tag in the selected unit and format, and adds the text DIMENSION in the middle dimension tag in the **Dimension Properties** dialog box.



Example

Below is an example of dual dimensions using the units mm and format ###.



See also

Define dimensioning (page 725)

Add dual dimensions manually (page 260)

Add dimensions to unfolded parts

In single-part and assembly drawings, you can control the dimensions Tekla Structures adds for unfolded parts that have been created using **View Properties** --> **Attributes 2** --> **Unfolded** : **Yes**.

Use the advanced options in **File menu** --> **Settings** --> **Advanced options** --> **Dimensioning - unfolding** .

To	Do this
Create bending line dimensions for unfolded parts.	Set the advanced option `XS_DRAW_BENDING_LINE_DIMENSIONS_IN_UNFOLDING=TRUE`.
Create angle and radius dimensions for unfolded parts.	Set the advanced option `XS_DRAW_ANGLE_AND_RADIUS_INFO_IN_UNFOLDING=TRUE`.
Set a prefix text for an angle dimension.	Set the advanced option `XS_ANGLE_TEXT_IN_UNFOLDING_BENDING_LINE_DIMENSIONING=A=`.
Set a prefix text for a radius dimension.	Set the advanced option `XS_RADIUS_TEXT_IN_UNFOLDING_BENDING_LINE_DIMENSIONING=R=`.

To	Do this
For angle text dimensions, show the interior angle instead of the exterior angle.	Set the advanced option `XS_DRAW_INSIDE_ANGLE_IN_UNFOLDING=TRUE`.
Set the format for angle text.	Set the advanced option `XS_UNFOLDING_ANGLE_DIM_FORMAT=1`. **###**= 0 **###[.#]**= 1 **###.#**=2 **###[.##]**= 3 **###.##**= 4 **###[.###]**=5 **###.###**= 6 **### #/#**= 7 **###/##.###**= 8
Set the accuracy of the angle text.	Set the advanced option `XS_UNFOLDING_ANGLE_DIM_PRECISION=10`. **0.00**= 1 **0.50**= 2 **0.33**= 3 **0.25**= 4 **1/8**= 5 **1/16**= 6 **1/32**= 7 **1/10**= 8 **1/100**= 9 **1/1000**= 10

See also

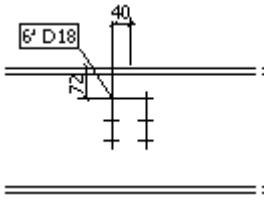
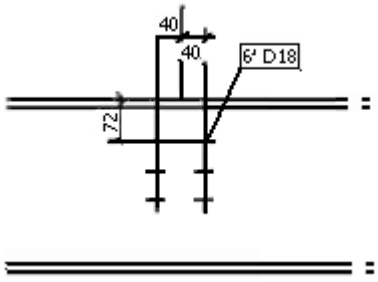
Dimension properties (page 932)

Add minimum and maximum position dimensions to bolts

By default, Tekla Structures does not create minimum and maximum position dimensions for bolts. You can use an advanced option to create the dimensions.

To create minimum and maximum position dimensions for bolts:

1. On the **File** menu, click **Settings** --> **Advanced options** and go to the **Dimensioning - bolts** category.
2. Set `XS_BOLT_POSITION_TO_MIN_AND_MAX_POINT` to `TRUE` .

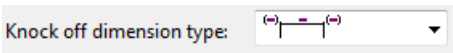
Setting	Example
Before setting the advanced option.	
After setting the advanced option to `TRUE`.	

Create dimension line extensions

You can create line extensions for dimensions that have line arrows.

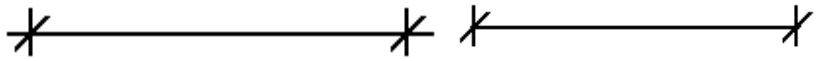
Limitations

Line extensions cannot be applied to dimensions that have different arrows from line arrows, or to knock-off dimensions of the following type:



1. On the **File** menu, click **Settings** --> **Options** and go to the **Drawing dimensions** settings.
2. Enter the length of the dimension line extension in the **Dimension line extension length for line arrow** box.

Example



Line extensions added No line extensions

Change the appearance of absolute dimensions

You can select whether to show zero at the zero point of the absolute dimensions, and also change the orientation of the absolute dimensions.

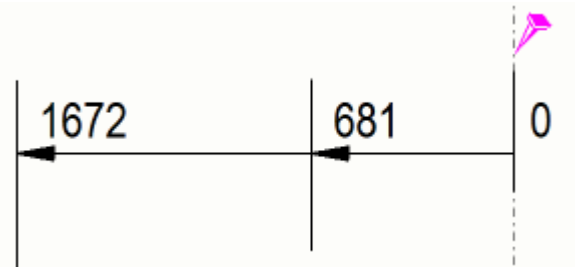
1. On the **File** menu, click **Settings** --> **Options**, and go to the **Drawing dimensions** settings.
2. Set **Show zero in absolute dimensions** to **No** if you do not want to show zero at the zero points in absolute dimensions.

   Yes is the default value.
3. Set **Draw absolute dimension values parallel to dimension line** to **Yes** to show dimensions parallel to dimension lines in absolute dimensions.

   No is the default value.
4. Click **OK**.

Example

In the following example, dimensions are parallel to the dimension line and zero is shown at the zero point.



See also

Dimension properties (page 932)

What are automatic view-level dimensions (page 726)

Create exaggerated dimensions

You can exaggerate narrow dimensions to make them easier to read.

When you enable the exaggeration of the dimensions, a dimension that is narrower than the defined limit is enlarged. If there are many exaggerated dimensions, Tekla Structures arranges them automatically. Setting exaggeration consists of selecting the exaggeration limit and the exaggeration scale, enabling the exaggeration and setting the direction, origin, width, position and height for the exaggerated dimensions.

For single-part , assembly and cast unit drawings, save the dimension exaggeration properties on object level in an open drawing into a dimension properties file, which you can take into use when you modify dimensioning rules.

Limitation: Exaggeration works only if the dimension extension lines are long. Set **Short extension line** to **No** on the **General** tab of the **Dimension Properties** dialog box.

1. On the **File** menu, click **Settings** --> **Options** and go to the **Drawing dimensions** settings.
2. Enter the exaggeration limit in the **Exaggeration limit** box.
3. Select **Paper** or **Model** as the exaggeration scaling method.

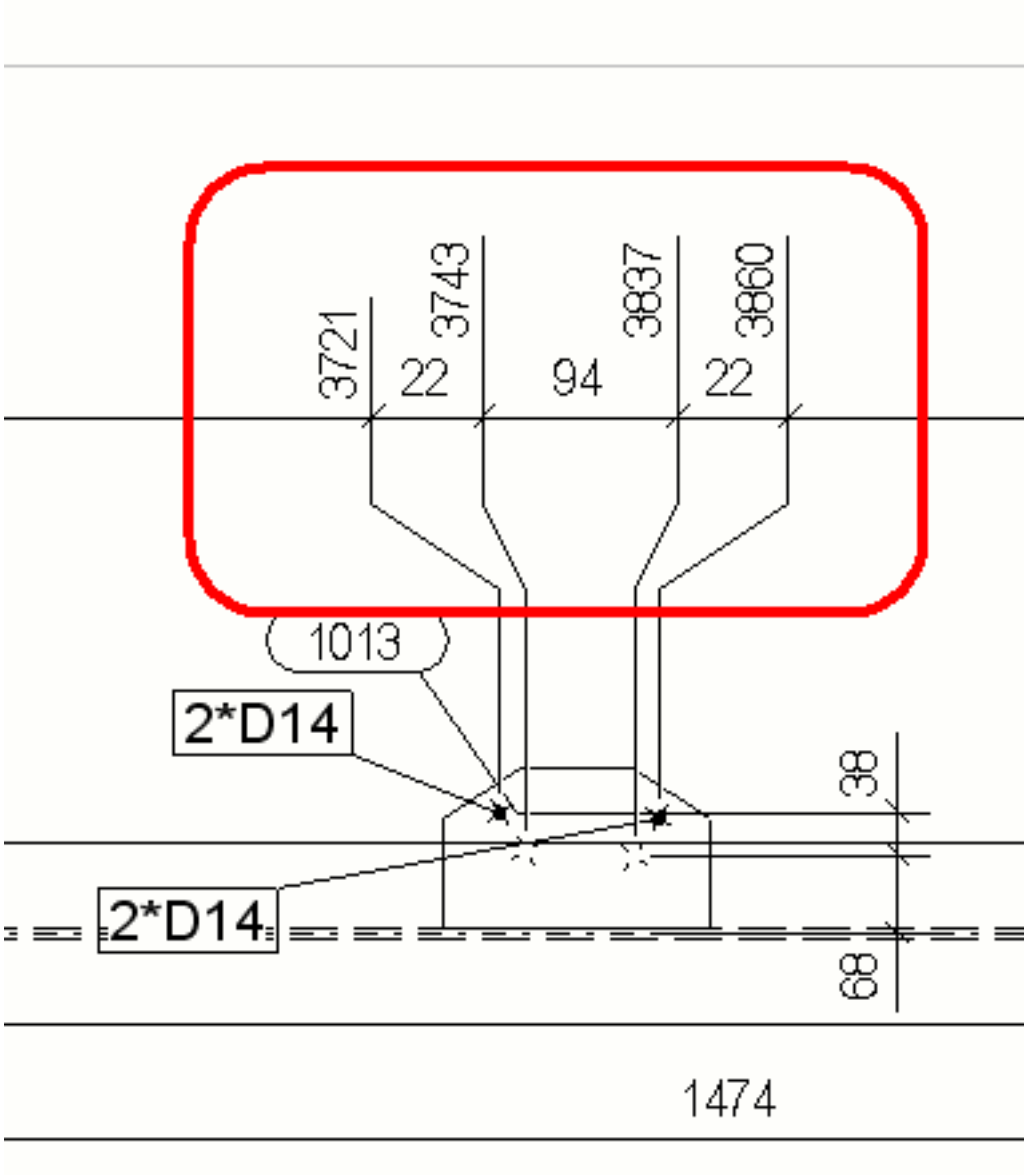
   If you select **Paper**, the exaggeration limit is multiplied by the view scale. For example, if the scale is 1:10 and the limit is 10 mm, then all the dimensions smaller than 100 mm are exaggerated.

   If you select **Model**, all dimensions smaller than 10 mm can be exaggerated regardless of the drawing scale.
4. Click **OK**.
5. Open a drawing and double-click a dimension.
6. Go to the **Marks** tab of the **Dimension Properties** dialog box.
7. Enable the exaggeration by setting **Exaggeration** to **Specified**.
8. Set the values for **Direction**, **Origin**, **Width**, **Position** and **Height**.
9. Enter a name for the dimension properties file at the top and click **Save**.
10. If you want to modify the current dimension, click **Modify**. Otherwise, close the dialog box.

    Now you have a dimension properties file that contains the exaggeration settings, which you can load later on or use in dimensioning rules.

Example

See below for an example of exaggerated dimensions:



See also

Dimension properties (page 932)

Change the prefix in radial dimensions

You can change the dimension prefix in radial dimensions.

By default, the prefix of radial dimensions is R, for example, R 200.

1. Close Tekla Structures
2. Open the file `dim_operation.ail` located in `..\Tekla Structures\<version>\messages\`.

3. Change the prefix R to Radius:

```
string dim_operation_dim_radius_prefix{ ... entry =
("enu", "R ");};

string dim_operation_dim_radius_prefix{ ... entry =
("enu", "Radius ");};
```

4. Save the changes and reopen Tekla Structures.

See also

Add dimensions manually (page 207)

Add dimensions to plates

You can dimension plates using some advanced options in **File menu** --> **Settings** --> **Advanced options** --> **Dimensioning - parts** .

NOTE The settings in the `dim_planes_table.txt` tell Tekla Structures which dimensioning planes are possible for certain profile types, and with the advanced options, you can fine-tune the dimension position. The settings in the `dim_planes_table.txt` file are available for use only when you have set the path to `dim_planes_table.txt` as a value for the XS_PART_DIMENSION_PLANES_TABLE advanced option.

When you modify `dim_planes_table.txt` in environment folders, after changing the file, you need to:

1. Restart Tekla Structures.
2. Re-create the drawings.

Otherwise, the changes will have no impact on the drawings.

To dimension plates using the advanced options:

To	Do this
Dimension the plates to the edge that is nearest to the neighbor part	Set the advanced option XS_PART_POSITION_TO_EDGE_NEAREST_TO_NEIGHBOUR to `TRUE`.
Dimension the plates to the leading edge of the **beams**	1. Set the advanced option XS_USE_PLATE_SIDE_POSITIONING to `FALSE`. 2. Set the advanced option XS_PART_POSITION_TO_EDGE_NEAREST_TO_NEIGHBOUR to `FALSE`. 3. Set the advanced option XS_PART_POSITION_TO_LEADING_EDGE to `TRUE`.

To	Do this
Dimension the plates to the leading edge of the **columns**	1. Set the advanced option XS_USE_PLATE_SIDE_POSITIONING to `FALSE`. 2. Set the advanced option XS_PART_POSITION_TO_EDGE_NEAREST_TO_NEIGHBOUR to `FALSE`. 3. Set the advanced option XS_PART_POSITION_TO_LEADING_EDGE_IN_COLUMNS_ALSO to `TRUE`.
Dimension the plates to the trailing edge	1. Set the advanced option XS_USE_PLATE_SIDE_POSITIONING to `FALSE`. 2. Set the advanced option XS_PART_POSITION_TO_EDGE_NEAREST_TO_NEIGHBOUR to `FALSE`. 3. Set the advanced option XS_PART_POSITION_TO_LEADING_EDGE to `FALSE`. 4. Set the advanced option XS_PART_POSITION_TO_LEADING_EDGE_IN_COLUMNS_ALSO to `FALSE`.
Dimension the plates using their original reference points in the model	1. Set the advanced option XS_PART_POSITION_TO_EDGE_NEAREST_TO_NEIGHBOUR to `FALSE`. 2. Set the advance option XS_USE_PLATE_SIDE_POSITIONING to `TRUE`. **Note:** If you have created one plate from left to right and another from right to left, Tekla Structures dimensions the plates differently.

How dim_planes_table.txt and XS_PART_POSITION_TO_LEADING_EDGE work together

- If you set all plate options (beam plates and contour plates) to `FALSE` in `dim_planes_table.txt` as follows:

  ```
  5, -1.0, FALSE, FALSE, FALSE, FALSE, FALSE, FALSE
  51, -1.0, FALSE, FALSE, FALSE, FALSE, FALSE, FALSE
  ```

  In this case, the `dim_planes_table.txt` file does not affect the plate dimensioning at all, but the `XS_PART_POSITION_TO_LEADING_EDGE` advanced option defines the location to be either right or left (but not middle).

- If you enable both left and right & bottom and top options in `dim_planes_table.txt` as follows:

  `5, -1.0, FALSE, TRUE, TRUE, FALSE, TRUE, TRUE`

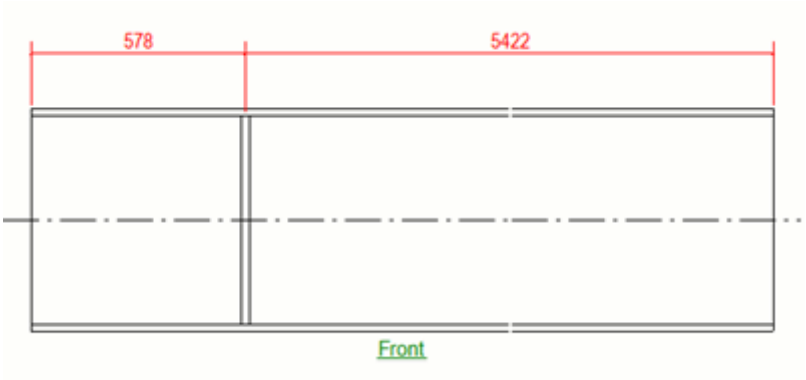
  `51, -1.0, FALSE, TRUE, TRUE, FALSE, TRUE, TRUE`

  In this case, you can still adjust which one is selected by using the `XS_PART_POSITION_TO_LEADING_EDGE` advanced option.
- If you set the dimensioning to the middle of the plates as follows:

  `5, -1.0, TRUE, FALSE, FALSE, TRUE, FALSE, FALSE`

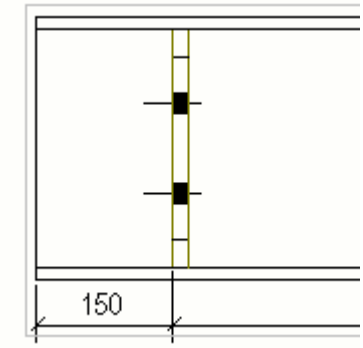
  `51, -1.0, TRUE, FALSE, FALSE, TRUE, FALSE, FALSE`

  In this case, the `XS_PART_POSITION_TO_LEADING_EDGE` advanced option does not have an impact.

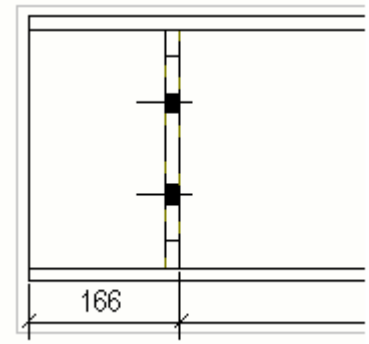


Examples

Below is an example of dimensioning plates to the leading edge.



Below is an example of dimensioning plates to the trailing edge.



In the following two examples, neighbor parts are blue, and the plate creation points are shown.

In the first example, the following values are used:

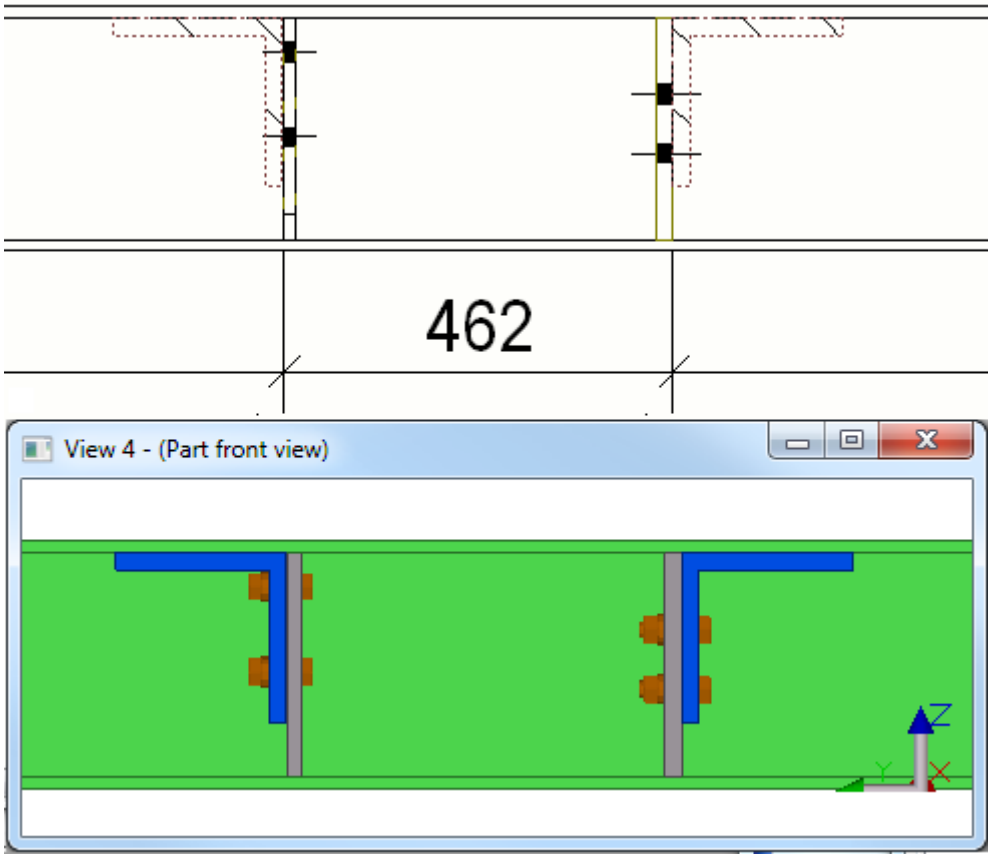
`XS_PART_DIMENSION_PLANES_TABLE`= (no value given, `dim_planes_table.txt` not used)

`XS_PART_POSITION_TO_EDGE_NEAREST_TO_NEIGHBOR=TRUE`

`XS_PART_POSITION_TO_LEADING_EDGE=FALSE`

`XS_PART_POSITION_TO_LEADING_EDGE_IN_COLUMNS_ALSO=FALSE`

`XS_USE_PLATE_SIDE_POSITIONING=FALSE`



In the second example, the following values are used:

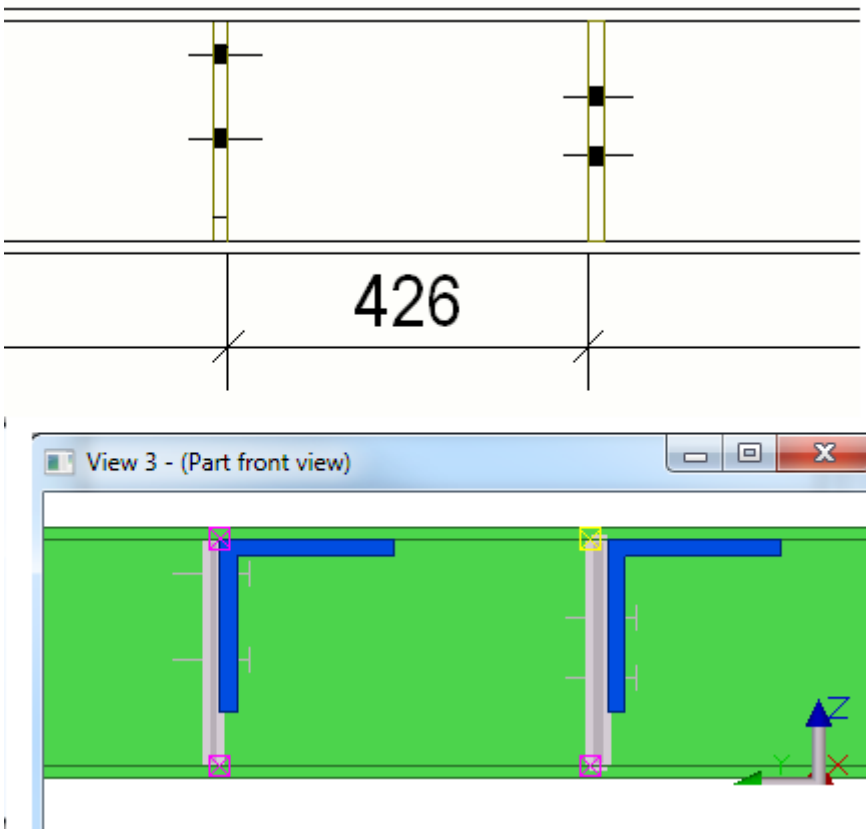
`XS_PART_DIMENSION_PLANES_TABLE`= (no value given, `dim_planes_table.txt` not used)

`XS_PART_POSITION_TO_EDGE_NEAREST_TO_NEIGHBOR=FALSE`

`XS_PART_POSITION_TO_LEADING_EDGE=FALSE`

`XS_PART_POSITION_TO_LEADING_EDGE_IN_COLUMNS_ALSO=FALSE`

`XS_USE_PLATE_SIDE_POSITIONING=TRUE`



See also

Add automatic view-level dimensions using Integrated dimensioning type (page 758)

Add dimensions to profiles

You can affect the way Tekla Structures dimensions different profiles in drawings. For example, you can have Tekla Structures always dimension round bars to the middle of the profile and large I profiles to the top.

To define dimension settings for profiles, you need to activate and then edit the dimension planes table `dim_planes_table.txt`.

1. On the **File** menu, click **Settings** --> **Advanced options** and go to the **Dimensioning - parts** category.
2. Set the advanced option XS_PART_DIMENSION_PLANES_TABLE as follows:

   ```
   XS_PART_DIMENSION_PLANES_TABLE=%XS_PROFDB%
   \dim_planes_table.txt
   ```

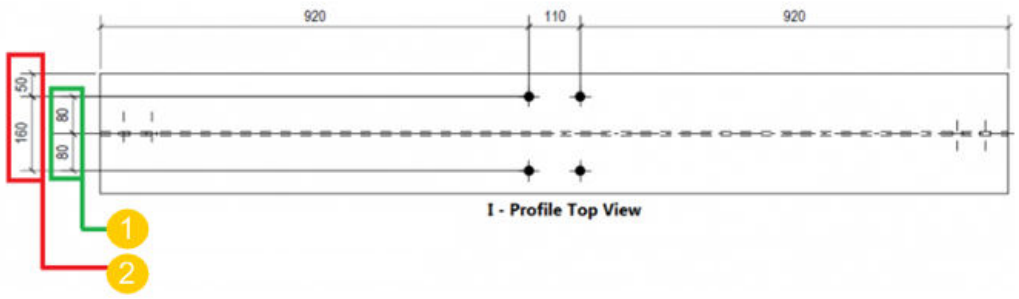
   This advanced option defines the path to the part dimension planes table.
3. Open the `dim_planes_table.txt` file in any text editor, for example, Microsoft Notepad. The file is located under the environment folder `..\Tekla Structures\<version>\environments` in the `\profiles` folder. The exact folder location may vary depending on your environment.
4. Edit the file contents and save the file.
5. To use the new settings in drawings, restart Tekla Structures and recreate the drawings. Changing the file does not automatically update existing drawings.

Example 1

In this example, the hole dimension is set from the middle of the flange instead of the flange edge for "I" Profile.

Open the file and edit the middle of the column to **TRUE*** in the **ProfType 1** row line, save and restart Tekla Structures. However, when you generate the new drawing, it will create the hole dimension from the middle of the flange.

dim_planes_table.txt
/*** DIMENSION PLANES TABLE for different profile types and sizes ***/
//Values: -1.0 in max size means no size limit!
/*** FLANGE WEB
ProfType, MaxSize, **middle**, left, right, middle, bottom, top
========== ***/
//**I-profile** - horizontal by reference line,vertical from top flange
1, **TRUE***, FALSE, FALSE, FALSE, TRUE, TRUE*



(1) Required dimension (middle of flange)

(2) Default dimension (edge of flange)

Example 2

Here is another example of a dimension planes table:

dim_planes_table.txt
FLANGE WEB
ProfType,MaxSize, middle,right, left, middle, right, left
==========
1, 300.0, TRUE*, FALSE, FALSE, FALSE, TRUE*, TRUE
7, -1.0, TRUE*, FALSE, FALSE, TRUE*, FALSE, FALSE

The line beginning with 1 means that Tekla Structures always dimensions I profiles (**ProfType** = 1) smaller than 300 mm (**MaxSize** = 300) to the middle of the flange and to the right edge of the web, no matter how the part was created.

The line beginning with 7 means that Tekla Structures always dimensions round tubes (**ProfType** = 7) to the middle of the profile

The **ProfType** numbers run in the same order as the profiles in the **Profile catalog**:

- 1 = I profile
- 2 = L profile
- 3 = Z profile
- 4 = U profile
- 5 = plate
- 6 = round bar
- 7 = round tube
- 8 = square pipe
- 9 = C profile
- 10 = T profile

- 15 = ZZ profile
- 16 = CC profile
- 17 = CW profile
- 51 = polygon plate
- 999 = sketched profile
- etc.

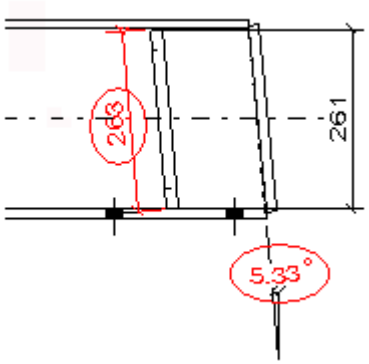
The value **-1.0** in the **MaxSize** indicates that there is no size limit for the profile.

The asterisk after **TRUE** indicates that it is the default value.

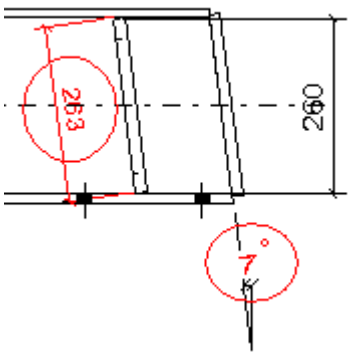
Sloped dimension texts

Tekla Structures aligns slightly sloped dimension text. If the dimension text is sloped more than a certain degree, Tekla Structures flips the text.

Below is an example of a dimension text that is slightly sloped.



Below is an example of a flipped dimension text.



The default limit for aligning dimension text is `0.1` (5.74 degrees). When this limit is exceeded, the dimension text gets flipped. To adjust this limit, use the advanced option `XS_TEXT_ORIENTATION_EPSILON`.

Add automatic dimensions to general arrangement drawings

In the general arrangement drawing **General - Dimensioning Properties** dialog box, you can create part, grid and overall dimensions, and control the way they are created. You can experiment with different combinations of options to achieve different kinds of dimensioning effects.

Dimension GA drawings

1. On the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing** .
2. Load drawing properties that are as close to the ones you need as possible.
3. Click **Dimensioning...**
4. Select the dimensions to create and modify the related settings.
5. On the **Grid** tab, adjust the grid dimension and overall dimension creation settings and the positioning of the dimensions.
6. On the **Parts** tab, adjust part dimension creation settings and the positioning of the dimensions.
7. Click **OK** and create the drawing.

Object groups in dimensioning general arrangement drawings

You can use the object groups (selection filters) that you have created in the model, or create the necessary groups through the **Parts** tab in the **General - Dimensioning Properties** dialog box using the **Object groups** button. For example, you might want to create an object group for beams of a certain size.

Dimension object groups on different dimension lines

You can use object groups to specify different objects to be dimensioned on different dimension lines.

1. On the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing**.
2. Click **Dimensioning**, and go to the **Parts** tab
3. Add the desired object groups to dimensioning rules by clicking **Add rule** and selecting the rule from the list in the **Object group** column.
4. If needed, you can create new object groups by clicking **Object group**. For example, add object group rules for beams of different size.

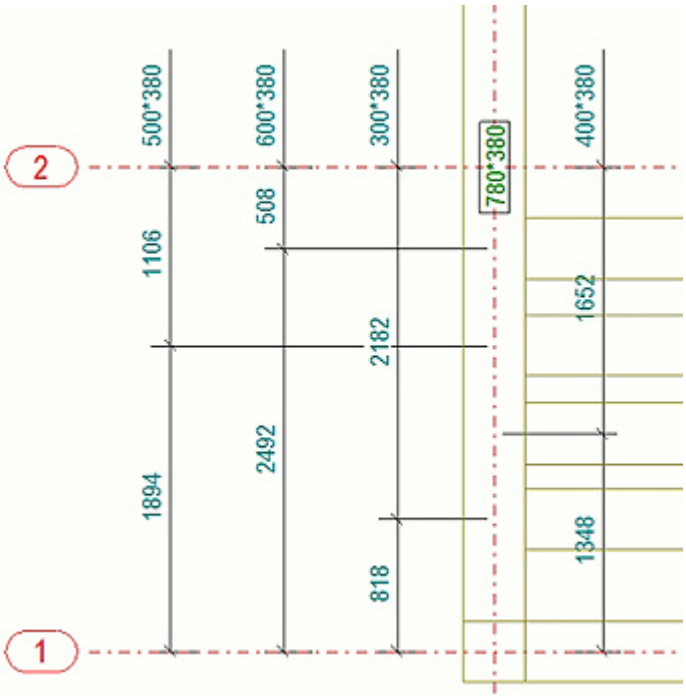
5. For each object group, select the **Positioning** option, the **Horizontal position** option, and the **Vertical position** option, depending on the type of the objects in the group.

   For example, for beam groups, set **Horizontal position** to **Left side** to position the beam dimensions to the left side of the grid.

6. If needed, in the **Tag** column, enter the text tags that you want to display for the different object groups in the drawing. For example, enter the size of the beam.

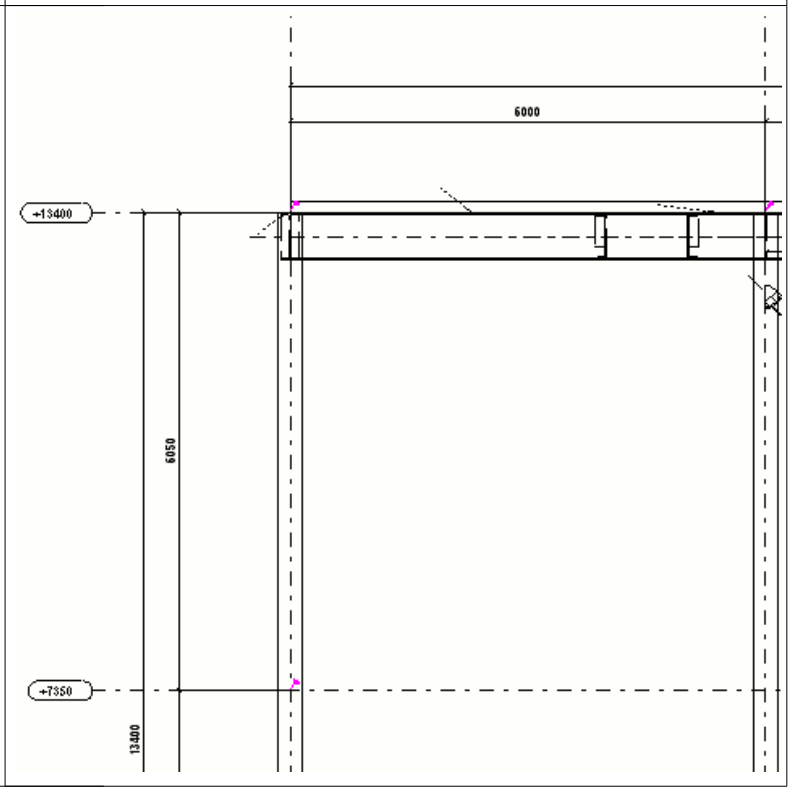
Example

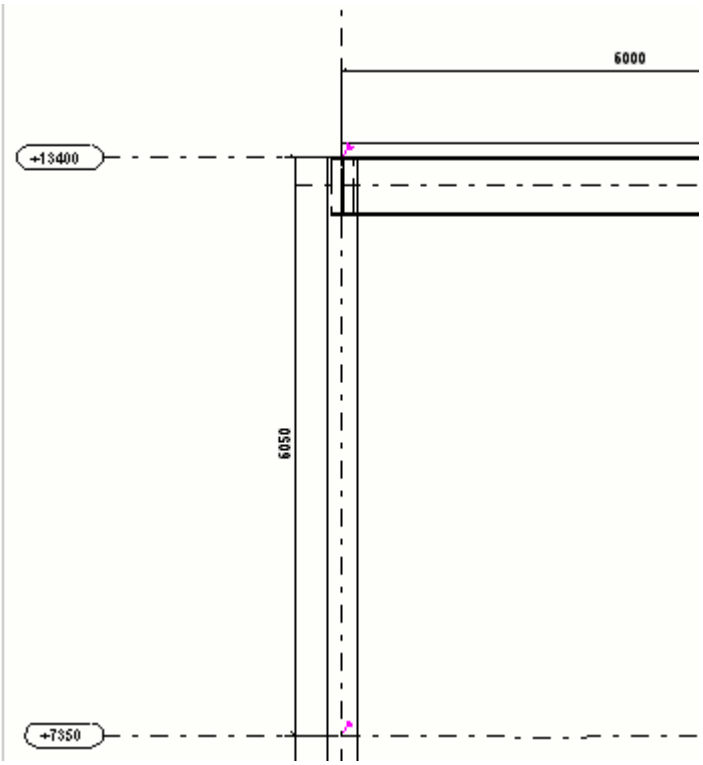
In this example, several beam groups were created, one for each beam size to be dimensioned, then the position for the dimensions in different groups was selected, and tags were added displayed for each group:



Example: Grid and overall dimensions

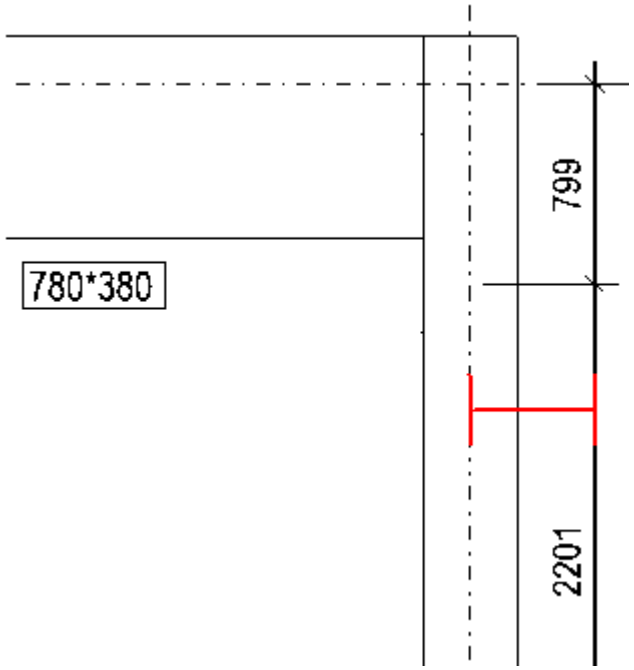
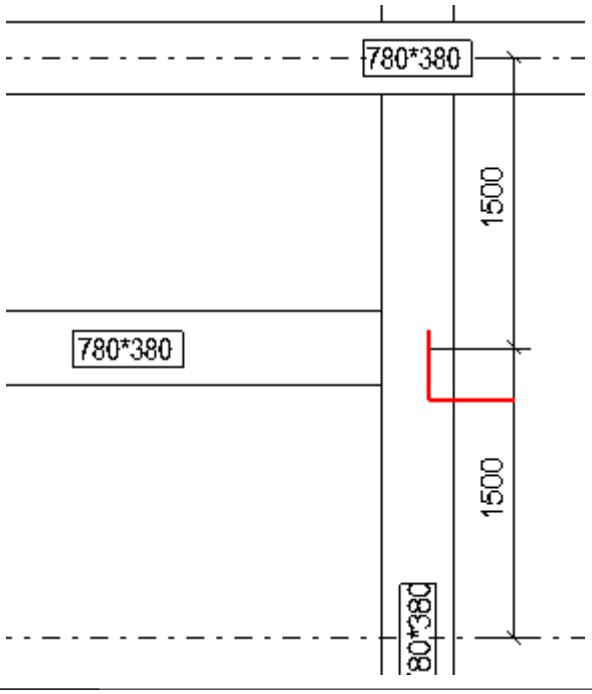
Here are some examples of what the grid and overall dimensions look like in general arrangement drawings with different settings selected on the **Grid** tab.

Dimensioning setting	Example
Grid line dimensions = **On** **Overall dimension** = **On** **Horizontal** = **Left** **Vertical** = **above**	

Dimensioning setting	Example
Grid line dimensions = **On** **Overall dimension** = **Off** **Horizontal** = **Left** **Vertica**l = **above**	

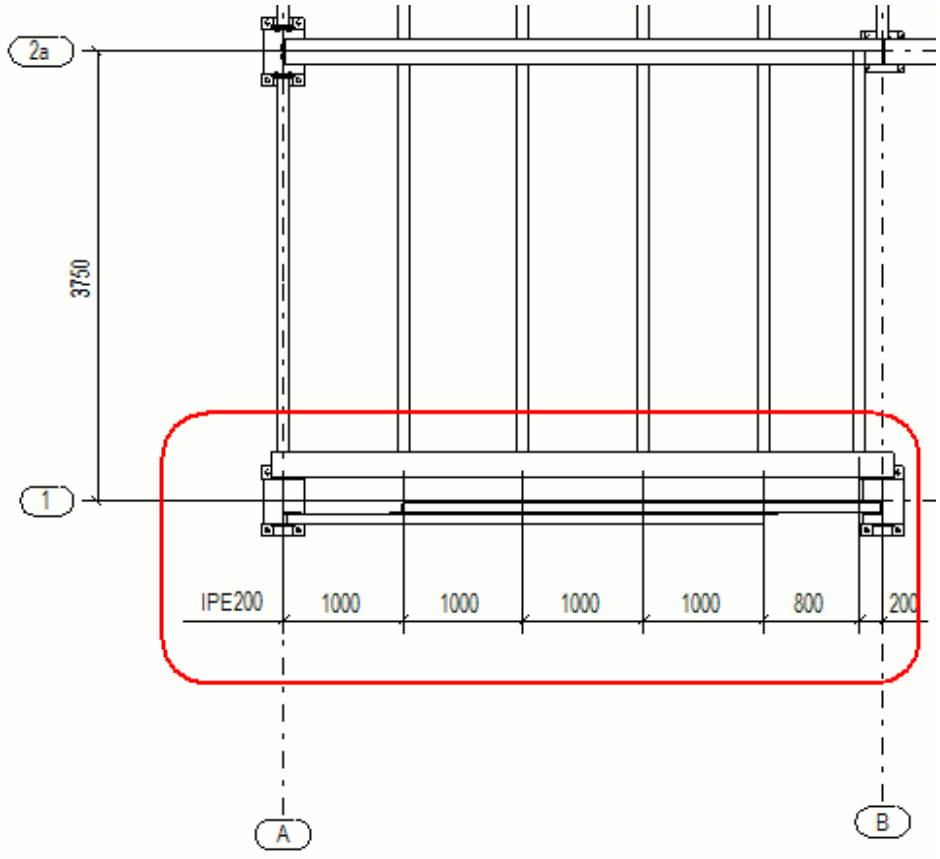
Example: Maximum leader line length options

Here are some examples of how the dimensions are positioned when you have set values for the **Maximum leader line length** options **Outside dimensions** and **Inside dimensions** on the **Parts** tab.

Dimensioning setting	Example
A value has been set for **Outside dimensions**.	
A value has been set for **Inside dimensions**.	

Example: Dimension parts partly outside the view

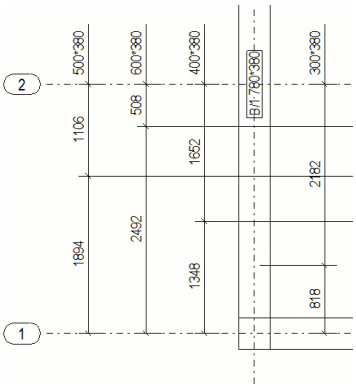
Here is an example of how the parts are dimensioned if you set the option **Include parts not entirely in the view** to **On** on the **Parts** tab of the **General - Dimensioning properties** dialog box.



Example: Limit the number of outside dimensions

Here is an example of what the part dimensions look like when you set 3 as the value for the option **Maximum number of outside dimensions** on the **Parts** tab in general arrangement drawing dimensioning properties. Tekla

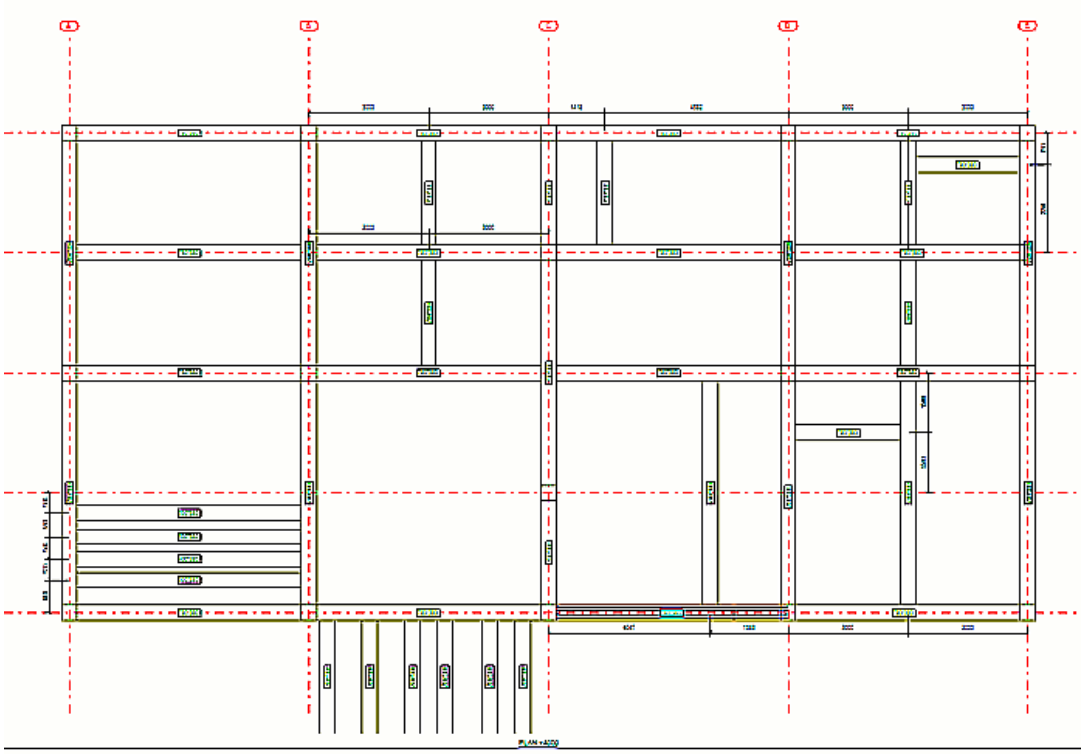
Structures creates three dimensions outside the grid and the fourth inside the grid.



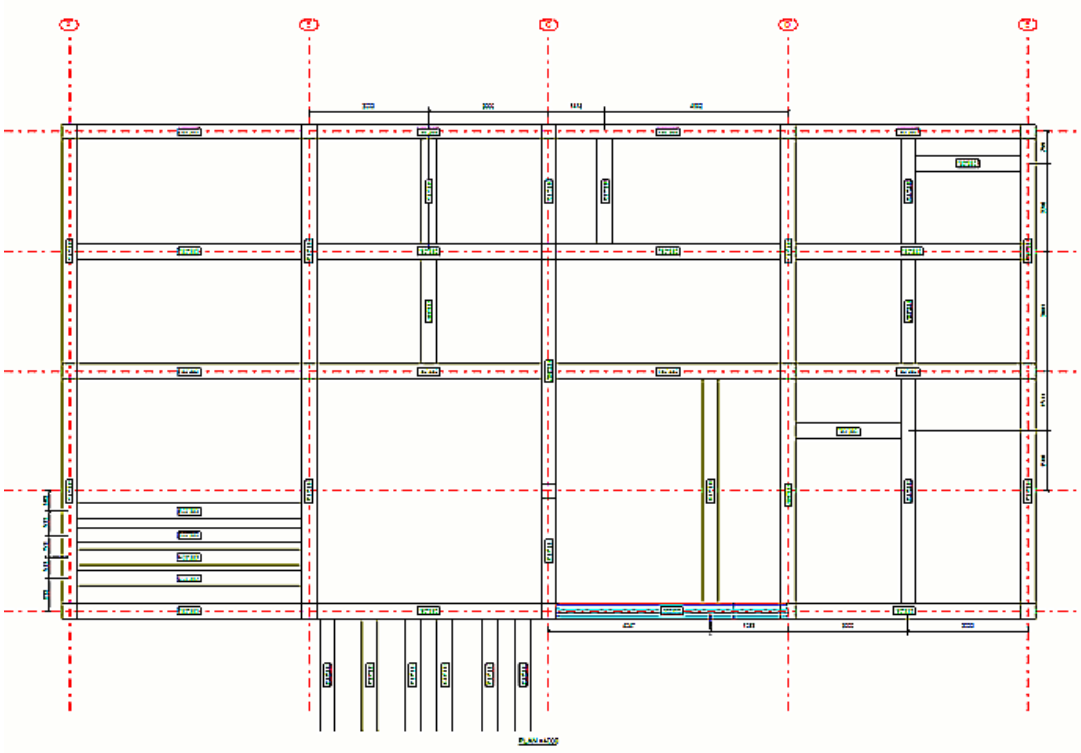
Example: Position part dimensions

Here are some examples of how the part dimensions are positioned in general arrangement drawings with different positioning settings selected on the **Parts** tab.

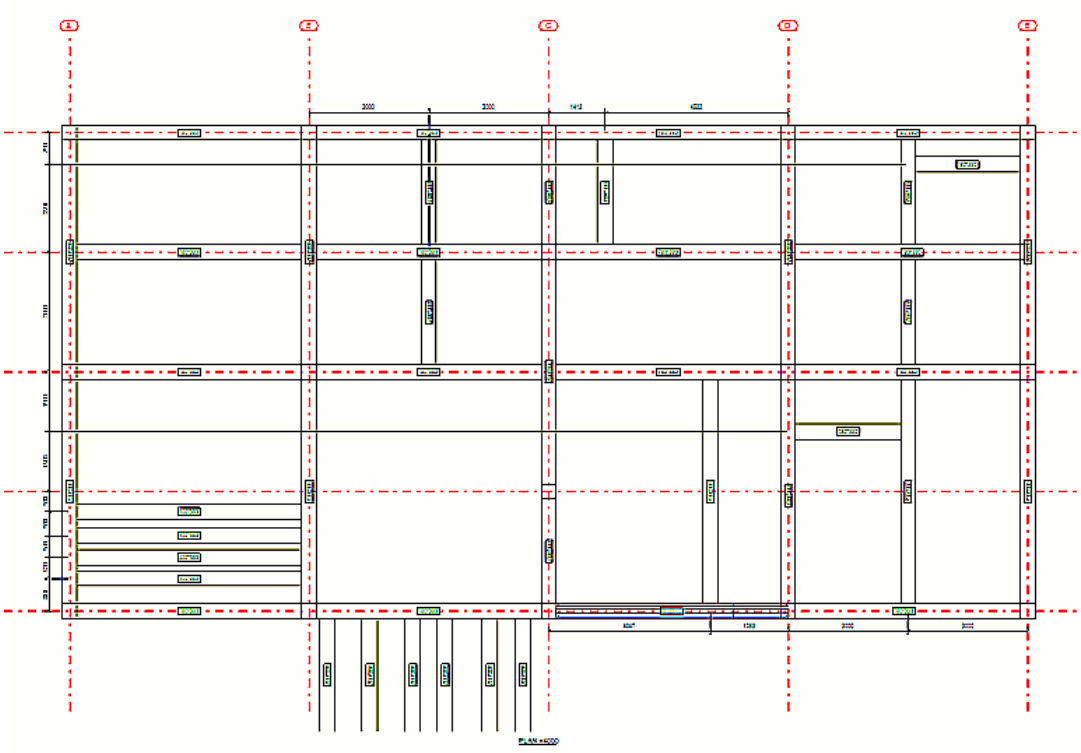
In the example below, **Positioning** is set to **Inside grid**, which places all dimensions next to or near the grid.



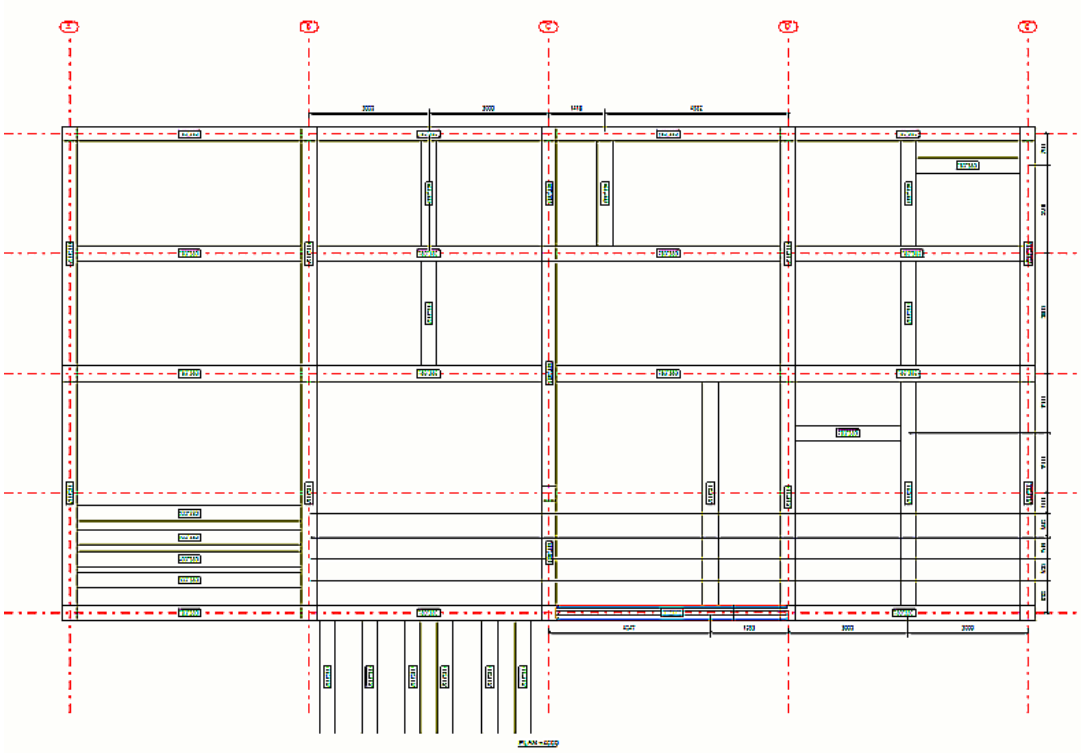
In the example below, **Positioning** is set to **Outside grid**, which places all dimensions outside the grid.



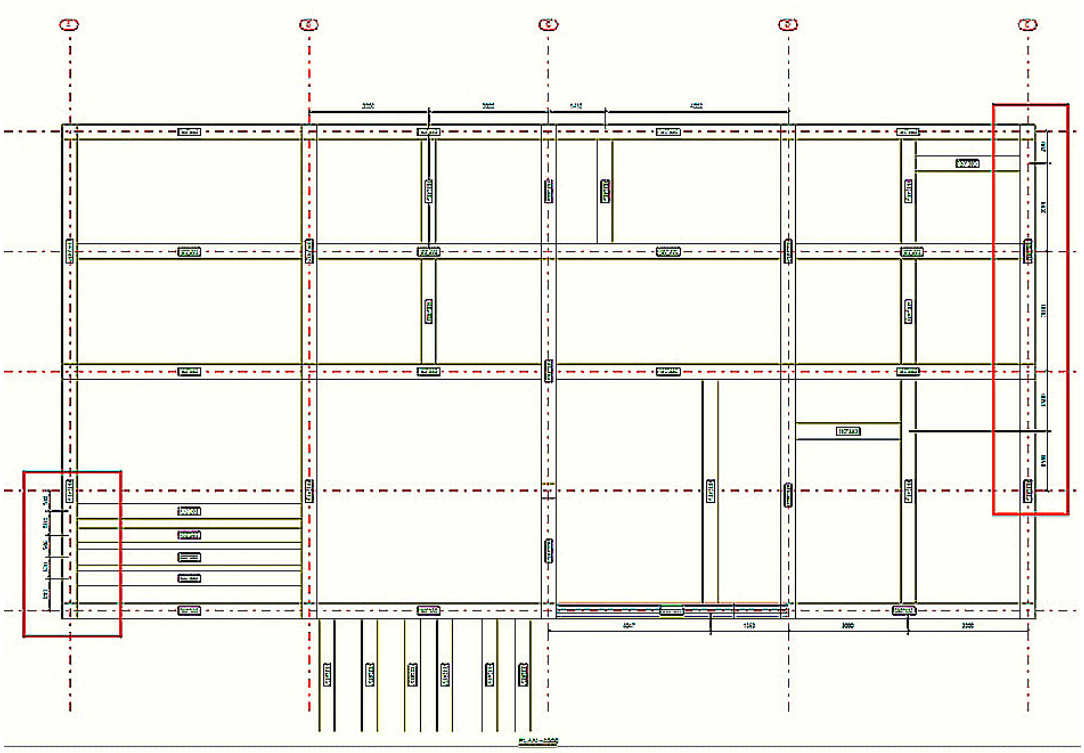
In the example below, **Horizontal position** is set to **Left side**, which places all dimensions to horizontal parts to the left of the grid.



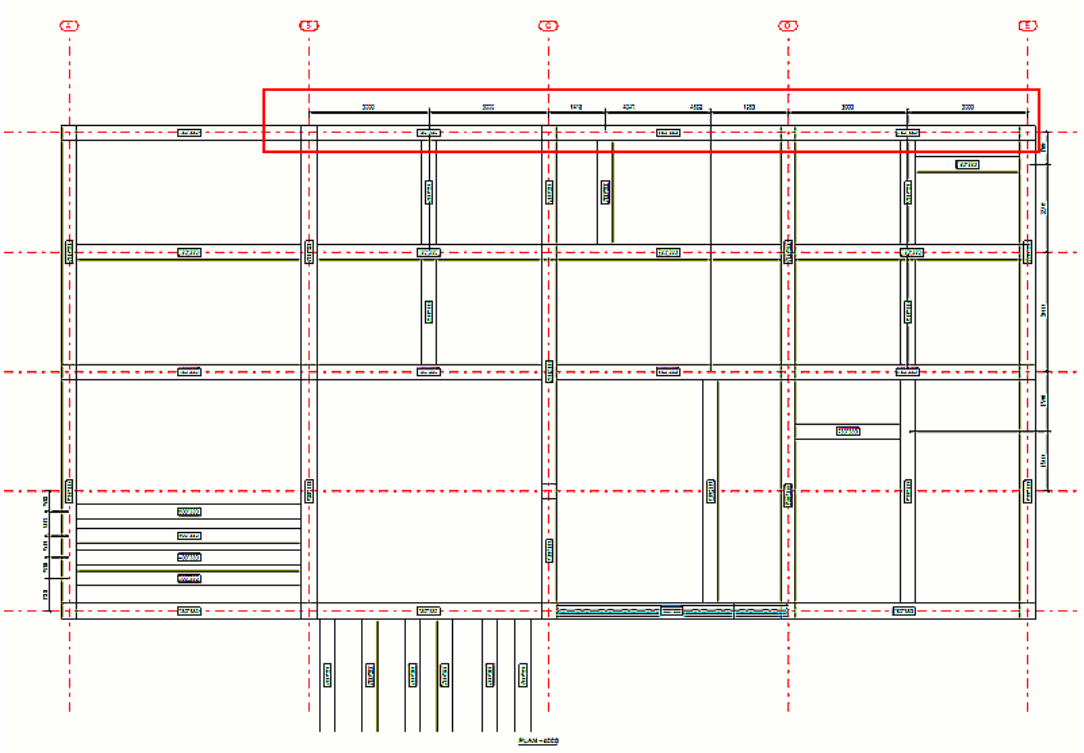
In the example below, **Horizontal position** is set to **Right side**, which places all dimensions to horizontal parts to the right of the grid.



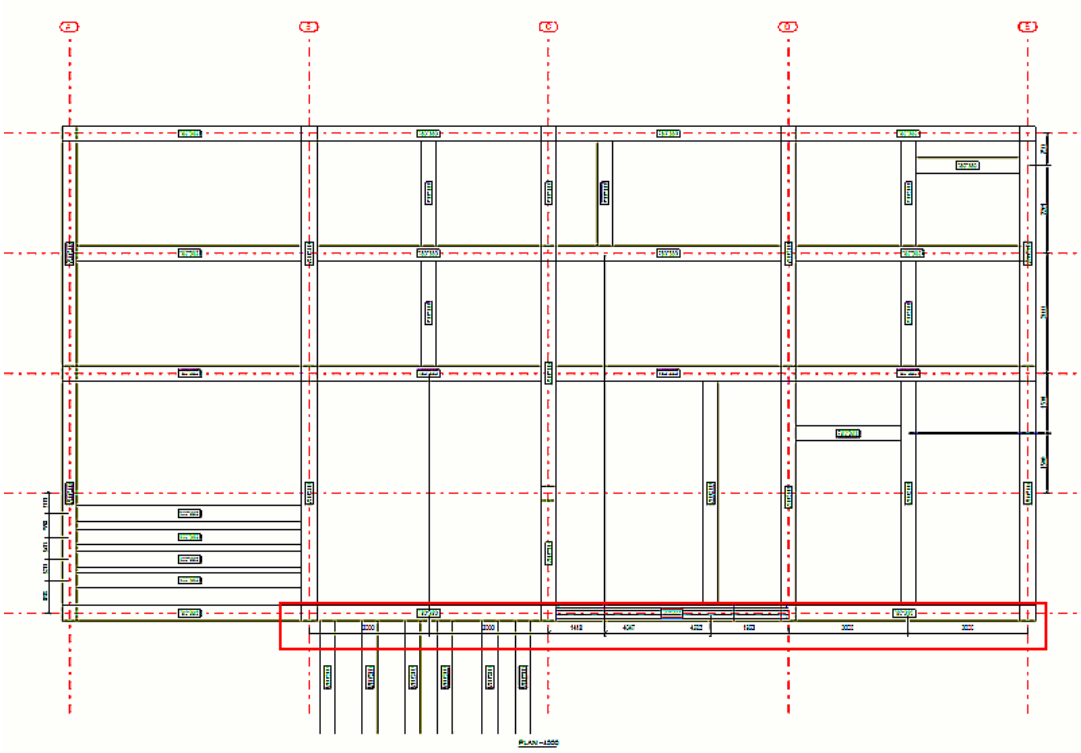
In the example below, **Horizontal position** is set to **Distributed to both sides**, which places all dimensions to horizontal parts to the side of the grid nearest the part they are dimensioning.



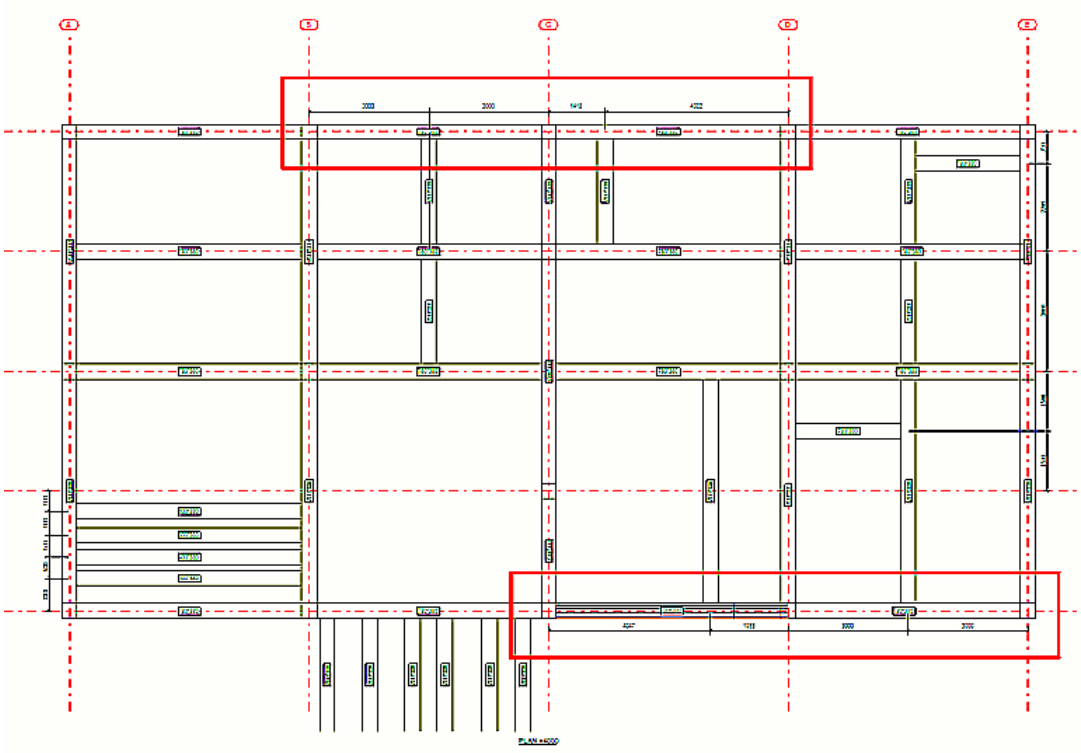
In the example below, **Vertical position** is set to **Above**, which places all dimensions to vertical parts above the grid.



In the example below, **Vertical position** is set to **Below**, which places all dimensions to vertical parts under the grid.



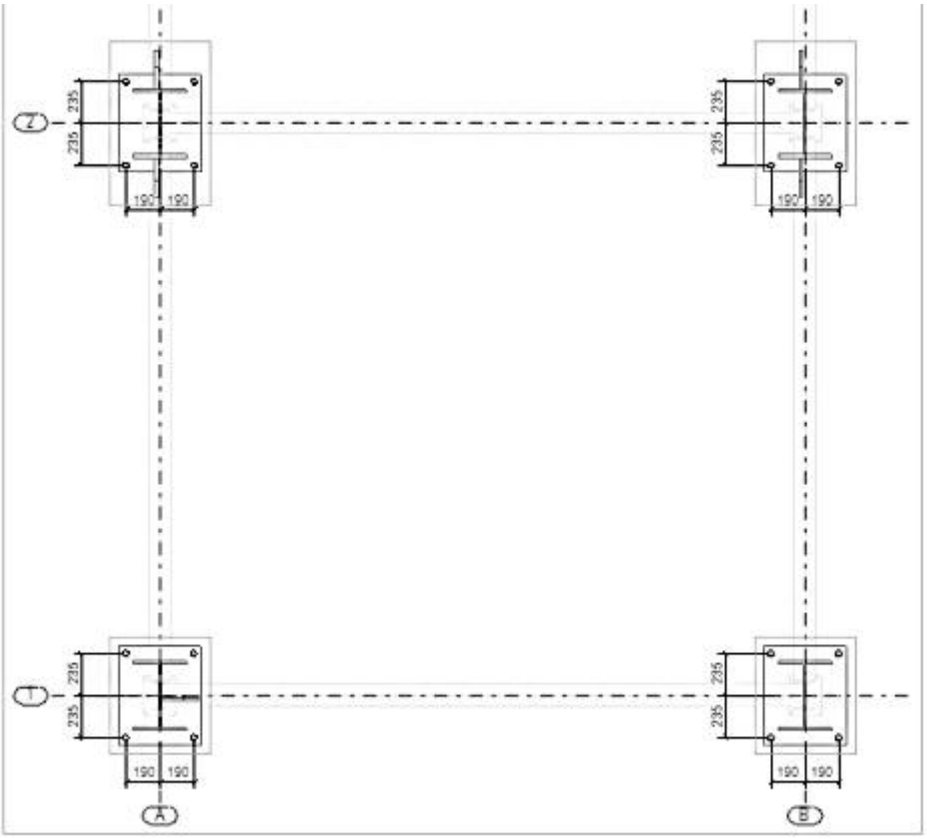
In the example below, **Vertical position** is set to **Distributed to both sides**, which places all dimensions to vertical parts to the side of the grid nearest the part they are dimensioning.



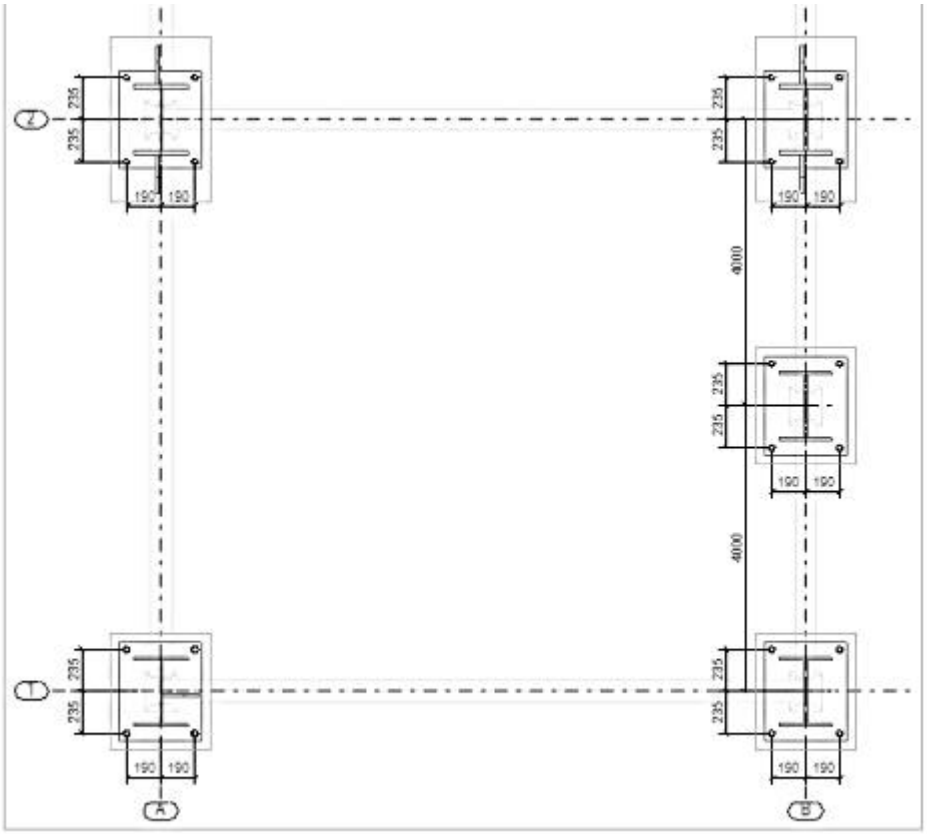
Example: Dimension anchor bolt plans

Here are some examples of what the dimensions in anchor bolt plans look like in different situations.

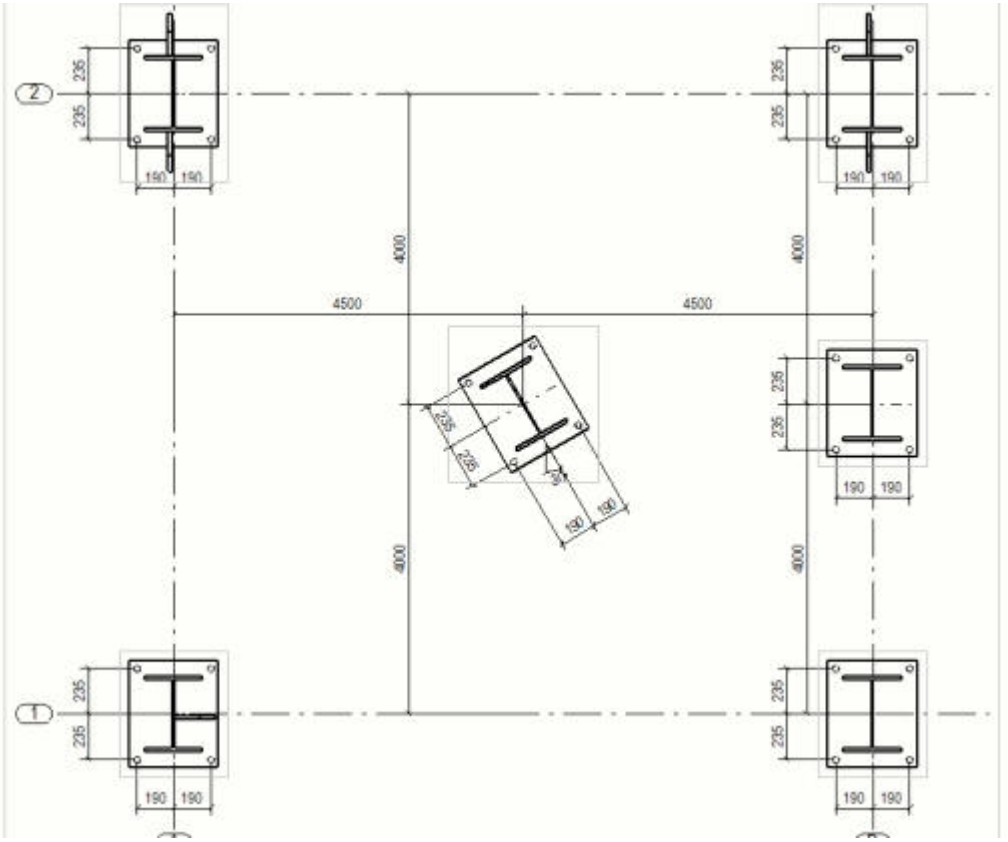
First an example of a typical situation, where all columns are located in grid line intersections:



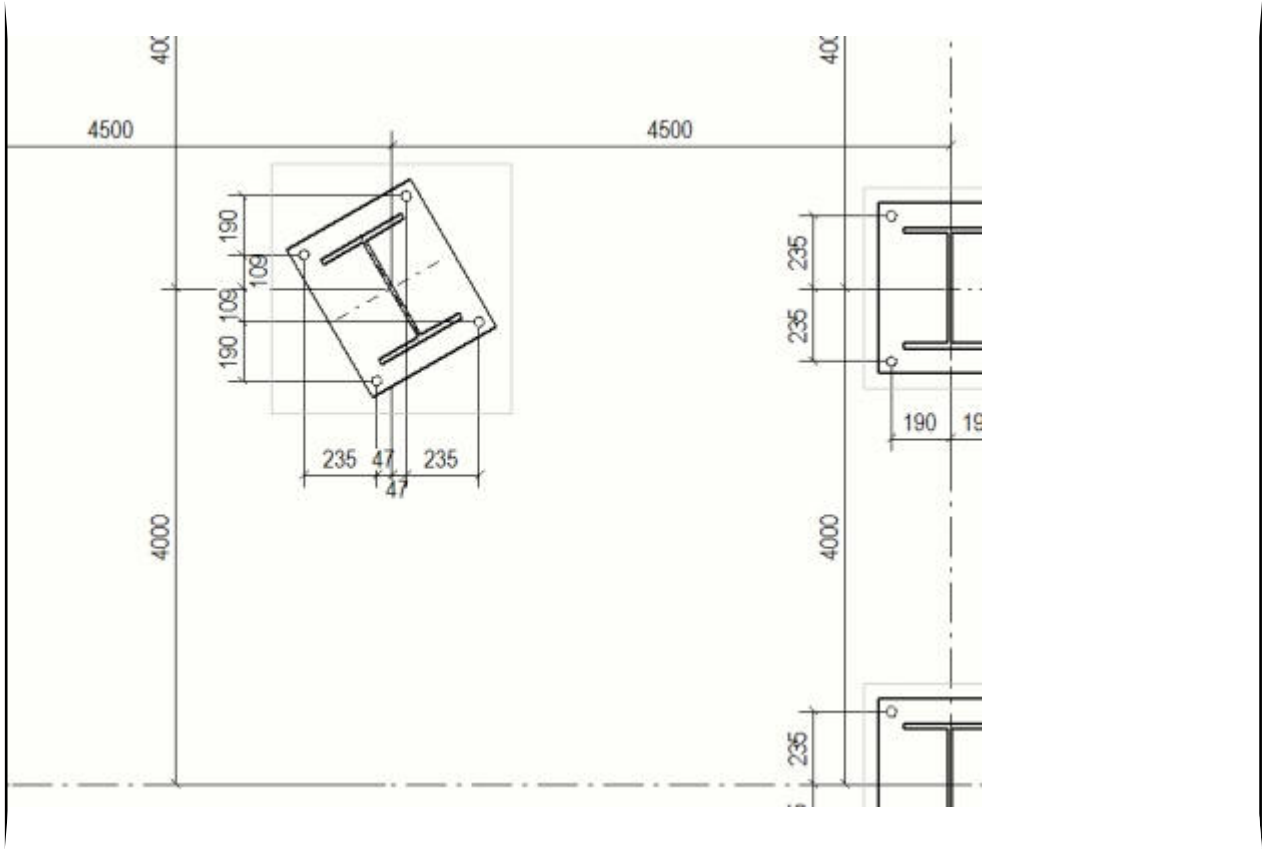
If the column reference point is not located on the grid line, Tekla Structures dimensions the reference point automatically relative to the grid lines. See the example below.



If the column is rotated relative to the drawing, the rotation is also automatically dimensioned. See the example below.



If you set the advanced option XS_ANCHOR_BOLT_PLAN_USE_VIEW_COORDSYS_FOR_BOLT_DIMENSIONS to `FALSE` and create the drawing, all dimensions are in the drawing coordinate system.



8.6 Define marks

Marks are associative annotation objects that are used for identifying individual building objects in a drawing. A mark displays a set of user-selectable properties called mark elements. By automatic marks we mean marks that Tekla Structures creates in a drawing based on the mark properties defined in the drawing properties.

The mark properties define what Tekla Structures shows in marks and how the marks are shown. In addition, Tekla Structures uses the `contentattributes_global.lst` attributes file for setting the default unit settings for some mark elements. You can use `contentattributes_userdefined.lst` when you want to configure settings of your own.

Tekla Structures is able to create the following automatic marks:

- Part marks
- Bolt marks
- Neighbor part marks
- Surface treatment marks

- Connection marks
- Model weld marks
- Reinforcement marks
- Pour object marks
- Dimension marks
- Section marks
- View label marks

You can modify the mark properties after creating the drawing, and add marks manually in an open drawing.

To	Click the links below to find out more
Set up and add automatic marks for building objects	Add automatic marks (page 811)
Indicate whether to show and merge marks	Adjust the visibility of automatic marks (page 814)
Add frames around single mark elements or around the mark itself, adjust the appearance of the mark text and leader line, and change the unit and format of an element	Adjust text, frames and leader lines of automatic marks (page 819)
Check how the location of the mark is affected by mark placement settings, the type of the leader line, predefined mark location and part orientation settings, modeling direction of parts, and drawing protection settings	Mark location (page 824)
Automatically merge part marks, surface treatment marks or reinforcement marks	Merge marks automatically (page 828)
Show the mark frame and leader line of the hidden part with a dashed or a solid line	Show mark frames and leader lines for hidden parts (page 823)
Set or change the unit and the number of decimals in measurement values for various mark elements	Change unit settings for marks (page 833)
Use advanced options to define the contents of the bolt mark **Size** element	Define size in bolt marks using advanced options (page 844)
Add level attributes in part marks and associative notes as user-defined attributes	Add level attributes in automatic part marks (page 836)

To	Click the links below to find out more
Add user-defined attributes and template attributes in marks	Add attributes in automatic marks (page 835)
Add custom graphical templates as elements in marks, for example, add a template that changes the unit and the number of decimals in measurement values in a mark	Add templates in automatic marks (page 839)
Add symbols in mark from a specific symbol file	Add symbols in automatic marks (page 844)
Add a pull-out picture of a reinforcing bar in a reinforcement mark	Add pull-out pictures in reinforcement marks (page 848)

See also

Mark properties in drawings (page 972)

Mark elements (page 981)

Define object protection and placement settings in drawings (page 677)

Add marks, notes, and texts in drawings (page 289)

Show pours in drawings (page 485)

Show pour objects, pour marks and pour breaks in drawings (page 883)

Units and decimals in drawings, reports and templates (page 893)

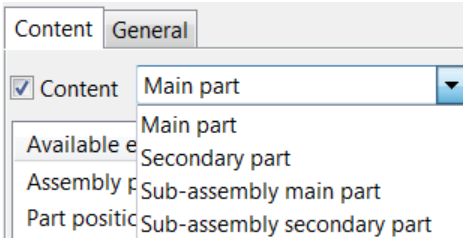
Add automatic marks

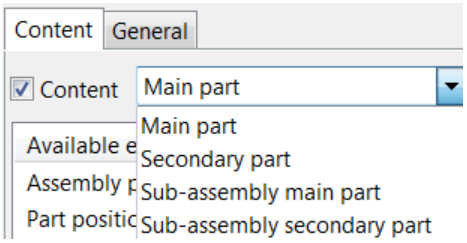
You can set up automatic marks for building objects (parts, neighbor parts, bolts, surface treatment, connections, reinforcement, and neighbor reinforcement) and save the mark properties in a property file for later use.

You can do this the view properties dialog box of single-part, assembly, and cast unit drawings. For general arrangement drawings, automatic marks can be defined on drawing level.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Do one of the following depending on the drawing type.

Drawing type	Add automatic marks
Single-part, assembly and cast unit drawings:	a. Click **View creation** in the options tree on the left, select the

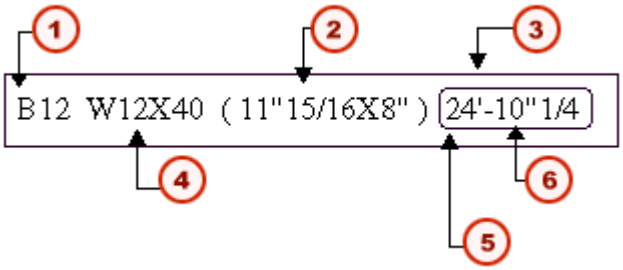
Drawing type	Add automatic marks
	view and the properties that you want to change, and click **View properties**. b. Click the mark type you want to modify, for example, **Part mark**. c. For some marks, you need to select from a list the object that you are defining the marks for. For example, for part marks, you can define part mark settings independently for main and secondary parts, and for sub-assembly main and secondary parts.  d. Add elements in the mark by double-clicking the elements in the **Available elements** list. e. Modify the element appearance (frame and font). For length, height, spacing and diameter elements, you can adjust also the unit and format. f. Use the **Move up** and **Move down** buttons to place the elements in the order you want. g. Modify the appearance, placement and visibility settings on the **Content** and **General** tabs. For example, adjust the leader line. h. Save the view properties by entering a properties file name in the box at the top and click **Save**. i. Click **Close** to return to the drawing properties.

Drawing type	Add automatic marks
General arrangement drawings:	a. Click the mark type you want to modify, for example, **Part mark...**. b. For some marks, you need to select from a list the object that you are defining the marks for. For example, for part marks, you can define part mark settings independently for main and secondary parts, and for sub-assembly main and secondary parts.  c. Add elements in the mark by double-clicking the elements in the **Available elements** list. d. Modify the element appearance (frame and font). For length, height, spacing and diameter elements, you can adjust also the unit and format. e. Use the **Move up** and **Move down** buttons to place the elements in the order you want. f. Modify the appearance, placement and visibility settings on the **Content** and **General** tabs. For example, adjust the leader line. g. Save the mark properties by entering a properties file name in the box at the top and click **Save as**. h. Click **OK** in the subdialog to save the changes, close the subdialog and return to the drawing properties.

4. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

Example

This is an example of a part mark.



1. Assembly position
2. Size
3. Mark frame
4. Profile
5. Mark element frame
6. Length

NOTE If you have created rebar marks automatically but you cannot see the marks, recreate the marks or modify the view or drawing level settings for rebar or neighbor rebar marks.

See also

Mark location (page 824)

Define placement settings for marks (page 683)

Mark properties in drawings (page 972)

Mark elements (page 981)

Add templates in automatic marks (page 839)

Add symbols in automatic marks (page 844)

Add pull-out pictures in reinforcement marks (page 848)

Add attributes in automatic marks (page 835)

Define size in bolt marks using advanced options (page 844)

Adjust the visibility of automatic marks

Using the visibility options in the mark properties you can indicate whether the marks are displayed in a drawing. You can modify these settings before creating a drawing, and also in an open drawing after creating a drawing.

To adjust the visibility of marks before creating a drawing:

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. For bolt marks and weld marks in assembly and single-part drawings, adjust the following settings located at the bottom of the **View creation** page. These settings affect all views in a drawing:
   - The **Weld mark visible** setting controls the visibility of model weld marks in assembly drawing views. The options are:
     - **In one view**: Tekla Structures automatically finds the most relevant view for showing the model weld marks. Every weld mark is shown in a drawing in one view only.
     - **In all views**: Tekla Structures adds the model weld marks in all drawing views that contain the part with the weld.
   - The **Weld size limit** setting filters welds of the defined size and bigger out of all views in a drawing. This setting is available in single-part drawings and assembly drawings.
   - The **Ignore bolt size** setting filters standard-sized bolt marks out of all views in a drawing, which means that Tekla Structures does not display bolt marks of the defined bolt size in the drawings. This setting is available in single-part drawings and assembly drawings.
4. Do one of the following depending on the drawing type. Note that all settings are not available for all mark types.

Drawing type	Adjust mark visibility settings
Single-part, assembly and cast unit drawings:	a. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**. You need to adjust the setting for all the views in the drawing separately. b. Click a mark type in the options tree. For example, click **Part mark**.

Drawing type	Adjust mark visibility settings
	• Go to the **General** tab and select whether you want to display marks by selecting one of the **Visibility in view** options. The available options depend on the mark type: • **distributed**: Distributes the marks evenly in the drawing view. Tekla Structures only creates marks that are not visible in the other views. • **always**: Always creates marks in the view, regardless of the settings in other views. Selecting the option **always** might slow down the drawing update during opening, even if you had deleted the marks manually. • Select **preferred** for one view only in a drawing. If you set other views to **distributed**, the marks are located only in the view that has the setting **Visibility in view** set to **preferred**. The option **preferred** acts like the option **distributed**, but the preferred view has a higher priority. • **none**: Does not create marks. Note that if you want to create your own marks, always use the **distributed** option. • In **Parts out of view plane**, select whether you want to

Drawing type	Adjust mark visibility settings
	display marks for parts that are out of view plane: • **Visible**: Displays marks for parts outside the view in the drawing. • **Not visible**: Displays no marks for parts outside the view in the drawing. • **For bolt marks**, select if you want to show bolt marks in main parts, secondary parts, sub-assembly main parts, or in sub-assembly secondary parts. **For bolt marks**, you can also define the **Ignore size**, which filters marks for standard-sized bolts out of drawings. Tekla Structures will not display bolt marks of the bolt size you enter here. Note that this setting is also available in drawing level properties at the bottom of the **View creation** page. • **For weld marks**, select if and how you want to show weld marks in sub-assemblies and hidden parts. **For weld marks**, you can also define the **Weld size limit** to filter welds of the defined size and bigger out of the drawing. Note that this setting is also available in drawing level properties at the bottom of the **View creation** page. c. Click **Save** to save the changes in view properties, and then click **Close** to return to drawing properties. d. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

Drawing type	Adjust mark visibility settings
General arrangement drawings:	a. Click a mark type button in the drawing properties dialog box. For example, click **Part mark...** b. Go to the **General** tab and select whether you want to display marks by selecting one of the **Visibility in view** options. The available options depend on the mark type: • **distributed**: Distributes the marks evenly in the drawing view. Tekla Structures only creates marks that are not visible in the other views. • **always**: Always creates marks in the view, irrespective the settings in other views. Selecting the option **always** might slow down the drawing update during opening, even if you had deleted the marks manually. • Select **preferred** for one view only in a drawing. If you set other views to **distributed**, the marks are located only in the view that has the setting **Visibility in view** set to **preferred**. The option**preferred** acts like the option **distributed**, but the preferred view has a higher priority. • **none**: Does not create marks. Note that if you want to create your own marks, always use the **none** option. c. In **Parts out of view plane**, select whether you want to display marks for parts that are out of view plane:

Drawing type	Adjust mark visibility settings
	• **Visible**: Displays marks for parts outside the view in the drawing. • **Not visible**: Displays no marks for parts outside the view in the drawing. d. For bolt marks, select if you want to show bolt marks in main parts, secondary parts, sub-assembly main parts, or in sub-assembly secondary parts. For bolt marks, you can also define the **Ignore size**, which filters standard-sized bolts out of drawings. Tekla Structures will not display bolt marks of the bolt size you enter here. e. Click **OK**. f. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

See also

Adjust mark visibility in an existing drawing (page 326)

Merge marks automatically (page 828)

Mark properties in drawings (page 972)

Adjust text, frames and leader lines of automatic marks

You can add frames around single mark elements and around the mark itself. You can also adjust the appearance of the mark text and leader line. For some elements, you can change the unit and format.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Do one of the following depending on the drawing type:

   Single-part, assembly and cast unit drawings:

   a. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.

b. Click a mark type in the drawing properties dialog box. For example, click **Part mark**.

General arrangement drawings:

- Click a mark type in the drawing properties dialog box. For example, click **Part mark...**

4. On the **Content** tab of the in mark properties, select one or several elements from the **Elements in mark** list and adjust the element settings:

   - To apply your changes in all of them, hold down **Shift** and click the last element in the list to select all elements.
   - To add a frame around the selected elements, click **< Add frame**.
   - Select a **Type** and **Color** for the frame.

     You can select a different frame type and color for each element you add.
   - Select the element text **Color**, **Font** and **Height**.

     You can select a different color, font and font height for each element you add.
   - If necessary, change the unit and the format of a length, height, spacing or diameter element.

     Before you can do this, you must first select the element from the **Elements in mark** list.

5. Go to the **General** (or the **Appearance**) tab adjust the mark frame and leader line settings:

   - Select the mark frame **Type** and **Color**.
   - Select the **Type** for the leader line and the **Arrow** to use.

     All marks do not have leader lines, in which case the leader line type selection is not available.
   - If you want to hide leader lines of hidden parts, set **Use hidden lines for hidden parts** to **Yes**.

     This option is not available for all marks.

6. **Single-part, assembly and cast unit drawings:** Click **Save** to save the view properties, and then click **Close** to return to drawing properties.

   General arrangement drawings: Click **OK**.

7. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

See also

Adjust part mark leader lines with advanced options (page 821)

Place reinforcement mark leader line associativity point automatically (page 821)

Mark properties in drawings (page 972)

Adjust part mark leader lines with advanced options

You can modify part mark leader line settings using advanced options. On the **File menu**, click **Settings** --> **Advanced options** and go to **Marking - parts**.

To modify part mark leader line settings using advanced options:

To	Do this
Define whether a leader line is drawn when the leader line is shorter than defined with the advanced option XS_DRAW_SHORT_LEADER_LINES_OF_PART_MARKS_MINIMUM_LENGTH	Set XS_DRAW_SHORT_LEADER_LINES_OF_PART_MARKS to `TRUE` (default) to always draw leader lines in part marks. When you set this option to `FALSE`, the leader line is not drawn if it would be shorter than the minimum you set for the advanced optionXS_DRAW_SHORT_LEADER_LINES_OF_PART_MARKS_MINIMUM_LENGTH
Give a minimum length for the leader line. If the length is less than this value, the leader line is not drawn.	Set a value in millimeters for the advanced option XS_DRAW_SHORT_LEADER_LINES_OF_PART_MARKS_MINIMUM_LENGTH.
Define the leader line start position for a leader line with a rectangular frame.	Set a value for the advanced option XS_MARK_LEADER_LINE_POSITION_TYPE_FOR_RECTANGULAR_FRAME.
Define the leader line start position for a leader line of a mark without a frame and for a leader line of a mark with a mark element frame.	Set a value for the advanced option XS_MARK_LEADER_LINE_POSITION_TYPE_FOR_NO_FRAME.
Define the length of the leader line extension.	Set a value for the advanced option XS_MARK_LEADER_LINE_EXTENSION_LENGTH.

See also

Adjust text, frames and leader lines of automatic marks (page 819)

Place reinforcement mark leader line associativity point automatically

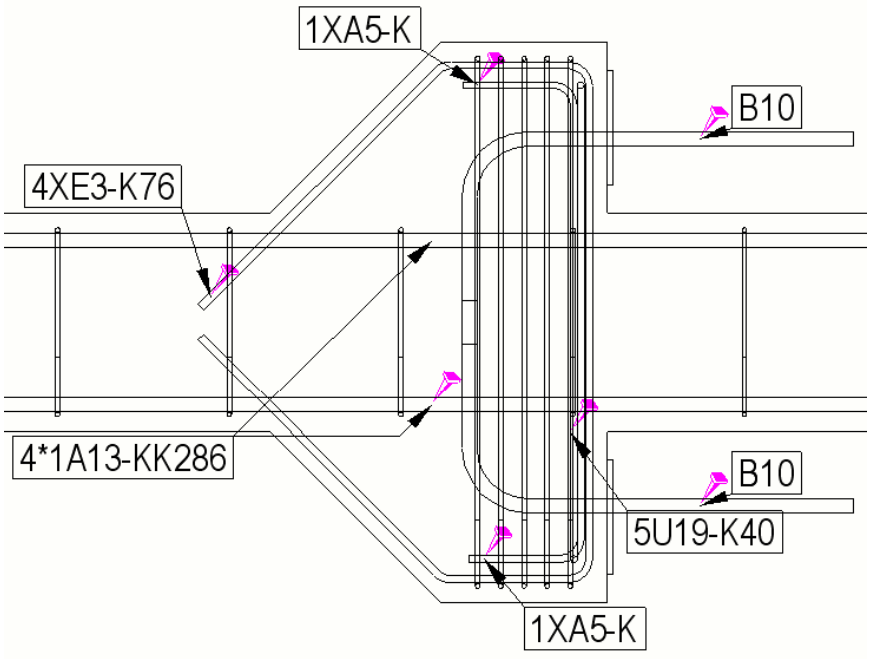
Tekla Structures places the associativity point of reinforcement mark leader lines so that it points to only one reinforcing bar. You can adjust how Tekla Structures searches for the place for the associativity point.

- On the **File menu**, click **Settings** --> **Advanced options** and go to the **Concrete detailing** category.

To	Do this
Select an optimal place for the associativity point.	Set XS_ENABLE_REBAR_MARK_LEADER_LINE_BASE_POINT_OPTIMIZATION to `TRUE`.
Define how far the other reinforcing bars must be from the associativity point in order for Tekla Structures to place the associativity point.	Set a millimeter value for XS_REBAR_MARK_LEADER_LINE_BASE_POINT_SEARCH_TOLERANCE.
Define the search step length while searching for an optimal place for the associativity point along the reinforcing bar.	Set a millimeter value for XS_REBAR_MARK_LEADER_LINE_BASE_POINT_SEARCH_STEP_LENGTH.

Example

An example showing optimized associativity points.



See also

Adjust text, frames and leader lines of automatic marks (page 819)

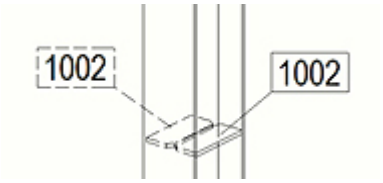
Show mark frames and leader lines for hidden parts

If a part is behind another part in the drawing so that it is hidden, you can select whether to show the mark frame and leader line of the hidden part with a dashed or a solid line.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. **Single-part, assembly and cast unit drawings:** Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. Click **Part mark**.
5. In part mark properties, go to the **General** tab.
6. Use one of the following options:

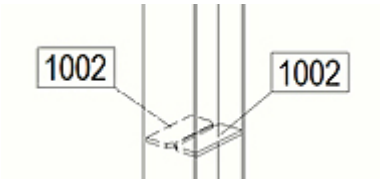
   Use hidden lines for hidden parts: **Yes**

   Part mark frame and leader line are shown with a dashed line.

   Use hidden lines for hidden parts: **No**

   Part mark frame and leader line are shown with a solid line.

7. **Single-part, assembly and cast unit drawings:** Click **Save** to save the changes in view properties and **Close** to return to drawing properties.

   General arrangement drawings: Click **OK**.
8. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

TIP With the advanced option XS_OMIT_MARKS_OF_HIDDEN_PARTS_IN_GA_DRAWINGS you can omit the marks of hidden objects in general arrangement drawings.

See also

Mark properties in drawings (page 972)

Mark location

The location of the marks in drawings is affected by several settings, not only the properties of the mark itself.

Setting	Click the links below to find out more
The placement settings in the mark properties	Define placement settings for marks (page 683)
The type of the selected leader line	How leader line type affects part mark and surface treatment mark location (page 825) How leader line type affects reinforcement mark location (page 827) How merging affects reinforcing bar group mark location (page 826)
The predefined mark location and part orientation settings	Set a predefined location for beam, bracing and column marks (page 824) Settings in the Options dialog box: Orientation settings Indicate part orientation (page 857)
Protection settings	Protect areas in a drawing (page 679)
The modeling direction of parts	Create horizontal parts
Alignment of the marks	Align selected drawing objects (page 417)

See also

Mark properties in drawings (page 972)

Set a predefined location for beam, bracing and column marks

By default, part marks are placed at the end point of the part. You can change this by adjusting the predefined location settings for beam, bracing and column marks.

1. On the **File** menu, click **Settings** --> **Options** and go to the **Orientation marks** settings.

2. In **Mark location**: **Preferred location for beams and bracings**, select **Left** or **Right** to place the mark to the left or right end of the part.
3. In **Mark location**: **Mark always to center of column in GA drawings**, select **Yes** to place part marks in the center of columns in plan views, or **No** to place part marks on the same flange in GA drawings and assembly drawings.
4. Click **OK**.

See also

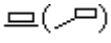
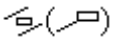
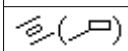
Mark location (page 824)

Mark properties in drawings (page 972)

Indicate part orientation (page 857)

How leader line type affects part mark and surface treatment mark location

Part marks and surface treatment marks have several types of leader lines you can select. The type of the leader line affects the location of the mark.

Leader line type	Description
	Always uses a leader line.
	Tries to find a space along the part for the mark. If impossible, Tekla Structures uses a leader line.
	The mark is always along the part. Lack of space might cause the mark to overlap other elements.
	The mark is always inside the part.
	The mark is always inside the part and parallel to the part axis.
	Tries to find a space for the mark inside the part. If impossible, Tekla Structures places the mark along the part with a leader line.
	Tries to find a space inside the part for the mark and align it parallel to the part axis. If impossible, Tekla Structures places the mark along the part with a leader line.
	Places the part mark along and in the middle of a part face. Note that the direction of the X axis of the object defines if there is a leader line or not. For example, column marks will get a leader line in an XY plan view whereas horizontal beam marks will get the mark along and in the middle of the part face.

See also

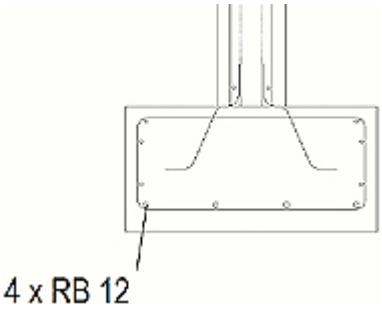
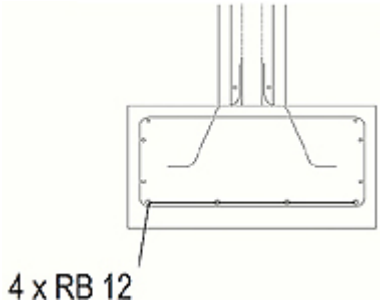
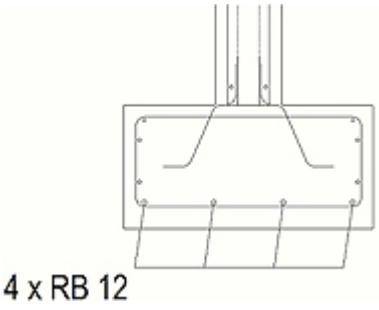
Mark location (page 824)

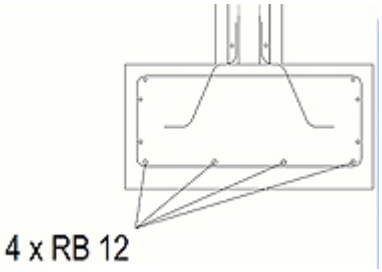
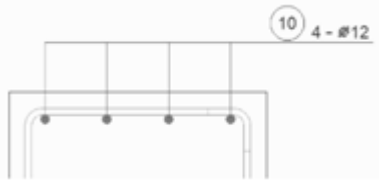
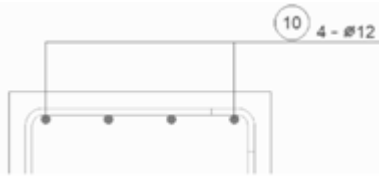
Mark properties in drawings (page 972)

How merging affects reinforcing bar group mark location

In reinforcing bar groups, Tekla Structures first tries to place the mark on the middle bar if it is visible. If that is not possible, Tekla Structures tries the next visible bar.

Below is a list of leader line options available for identical reinforcement marks and reinforcing bar group marks:

Option	Image	Example
One leader line to group		
One leader line per row		
Parallel leader lines		

Option	Image	Example
Leader lines to one point		
Perpendicular leader lines		
Leader line to first and last		

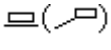
See also

Mark location (page 824)

Mark properties in drawings (page 972)

How leader line type affects reinforcement mark location

Reinforcement marks have several types of leader lines you can select. The type of the leader line affects the location of the mark. Tekla Structures tries to place the mark close to the midpoint on straight bars, or to the midpoint of the longest bar segment.

Reinforcement leader line type	Description
	Always creates a leader line.
	Tries to find a space along the reinforcing bar for the mark. If this is not possible, creates a leader line.
	The mark is always along the reinforcing bar. The mark may overlap other elements if there is not enough space.
	The mark is parallel to the reinforcing bar.

Reinforcement leader line type	Description
	The mark is parallel to the reinforcing bar on the bar outline. If there is not enough space for the mark, a leader line is created.

See also

Mark location (page 824)

Merge marks (page 356)

Mark properties in drawings (page 972)

Merge marks automatically

You can let Tekla Structures automatically merge marks.

You can automatically merge:

- Part marks and surface treatment marks
- Reinforcement marks

For more information about mark properties, including merging settings, see Mark properties in drawings (page 972).

You can also merge marks manually. For more information, see Merge marks (page 356).

Merged part marks

A merged part mark means that you have only one part mark for similar parts in a drawing, instead of a separate mark for each of the parts. Merged part marks indicate the number of included parts, and contain the defined part mark contents, and the near side and far side information. The marks are merged only in X direction of the main part.

Tekla Structures merges marks for visible parts in drawings if:

- The secondary parts are welded or bolted to the same main part.
- The parts are on the same line.
- Distances between the parts are equal.
- The parts have the same part position.
- Distance between parts is not more than what is set for the advanced option XS_PART_MERGE_MAX_DISTANCE.
- There are at least as many parts in the array as it has been set for the advanced option XS_MIN_MERGE_PART_COUNT.

Limitations

- You cannot merge part marks (assembly marks) that are not part of the same assembly.
- Tekla Structures does not merge neighbor part marks.

Advanced options in merging marks

In merging part marks, you may find the following advanced options useful:

XS_MULTIPLIER_SEPARATOR_FOR_MERGED_PART_MARK

XS_NSFS_POSTIX_FOR_MERGED_PART_MARK

XS_NS_POSTIX_FOR_MERGED_PART_MARK

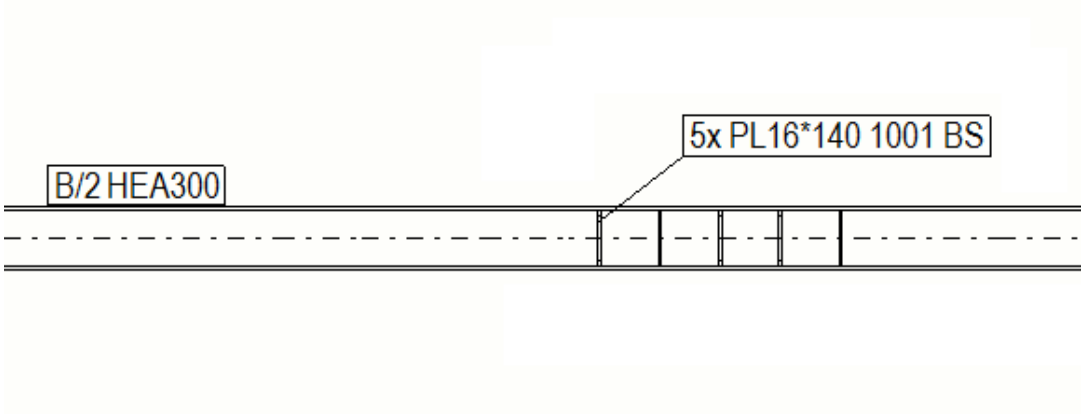
XS_FS_POSTIX_FOR_MERGED_PART_MARK

XS_PART_MERGE_MAX_DISTANCE

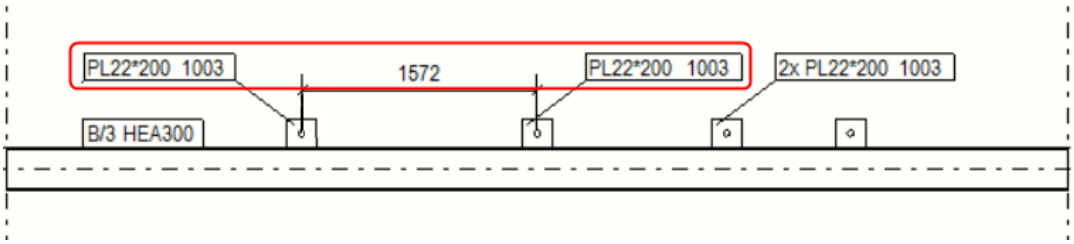
XS_MIN_MERGE_PART_COUNT

Example

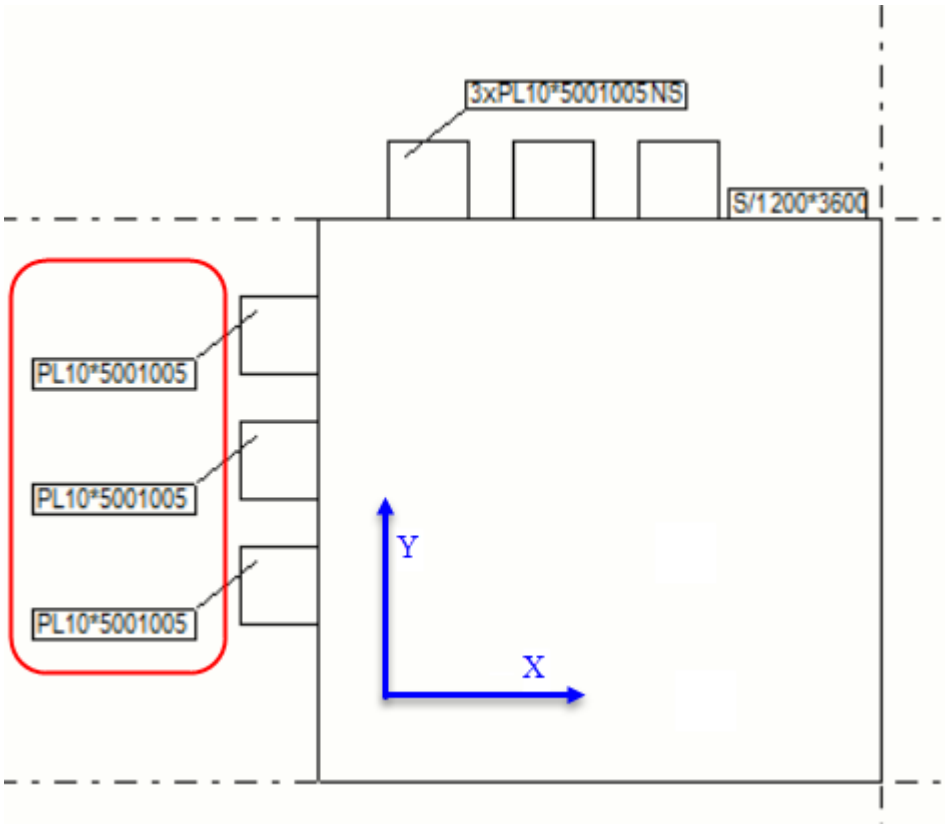
In the example below, the part marks are merged in X direction of the HEA300 beam (main part).



In the example below, the leftmost part marks are not merged, because they are too far from each other.



In the example below, the marks in the Y direction are not merged, because the marks are merged only in X direction (which in this example is horizontal).



Merge part marks automatically

You can merge part marks and surface treatment marks automatically to reduce the number of marks in the drawing.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. **Single-part, assembly and cast unit drawings:** Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. Click**Part mark** .

   If you want to merge surface treatment, click **Surface treatment mark** instead.
5. On the **General** tab in the part mark properties, set **Merge marks** to **On**.
6. **Single-part, assembly and cast unit drawings:** Click **Save** to save the changes in view properties and **Close** to return to drawing properties.

   General arrangement drawings: Click **OK**.

7. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

Tekla Structures merges marks for identical secondary parts on both faces of main parts.

Merged reinforcement marks

Tekla Structures can automatically merge similar reinforcement marks of bars, and you can also merge reinforcement marks manually. Merged reinforcement marks may include several blocks, and additional information. Blocks combine similar single marks.

NOTE To have Tekla Structures automatically merge reinforcement marks in drawings, the reinforcement must be attached to a concrete part or cast unit in the model.

Tekla Structures automatically merges marks for visible reinforcing bars in drawings if:

- The bars belong to the same concrete part or cast unit.
- The direction of the bars is the same.
- The bar marks are identical.
- The bars are close to each other.
- A straight line can be drawn through all the bars.

You can also define the distance within which to automatically merge marks for visible reinforcing bars using the following advanced options:

XS_MAX_MERGE_DISTANCE_IN_HORIZONTAL

XS_MAX_MERGE_DISTANCE_IN_VERTICAL

Merge reinforcement marks automatically

You can merge reinforcement marks automatically to reduce the number of marks in the drawing.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type. You can merge reinforcement marks in cast unit drawings and general arrangement drawings.
2. Load drawing properties that are as close to the ones you need as possible.
3. **Cast unit drawings:** Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. Click .
5. Go to the **Merging** tab of the reinforcement marks dialog box.

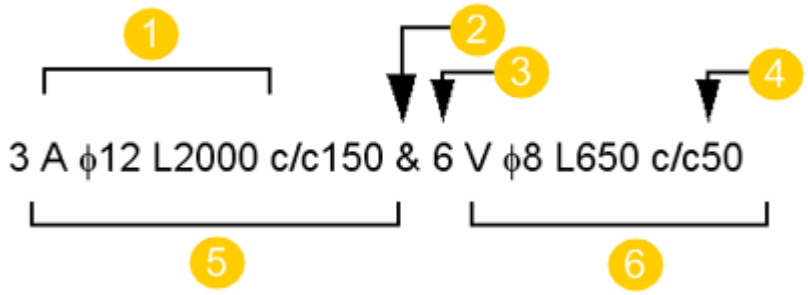
6. Select an option from the **Identical marks in same cast unit** list to merge marks and create leader lines:
   - **One leader line to group** creates one leader line for a group of reinforcing bars.
   - **One leader line per row**: Merges the marks and creates one leader line for a row of reinforcing bars.
   - **Parallel leader lines**: Merges the marks and creates parallel leader lines.
   - **Leader lines to one point**: Merges the marks and draws all leader lines to one point.
   - **No merge**: Marks are not merged, an individual leader line is created for each mark.
   - If you select **No merge**, you still need to define the mark content for the marks that Tekla Structures automatically merges on the **Merging** tab.
7. If there are several possible merge directions, select the horizontal or vertical from **Preferred direction of merge**.
8. Select the contents to be included in the merged reinforcement marks from the **Available elements** list.

   To ensure that merged reinforcement marks appear in the drawing, always include **Symbol separating blocks in mark** as the last element in the reinforcement mark. To omit the separating symbol, leave this box blank, but still include the element in the mark.
9. If needed, add a frame around the single elements in the mark. You can define the frame individually for each element.
10. Adjust the font, font height and the color of the mark text. You can adjust these settings individually for each element.
11. **Cast unit drawings:** Click **Save** to save the changes in view properties and **Close** to return to drawing properties.

    General arrangement drawings: Click **OK**.
12. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

Example

In this example, three A φ12 L2000 marks are merged to a block, and six V φ8 L650 marks to another block, and then these blocks are merged in the following way:



1. Single mark content
2. Symbol separating the blocks
3. Block prefix
4. Distance between groups
5. Block 1
6. Block 2

Change unit settings for marks

Tekla Structures uses the `contentattributes_global.lst` attributes file for setting the default unit settings for various mark elements. This file defines, for example, the unit used, and the number of decimals. You can change the unit and format for the length, height, diameter, and spacing elements in the mark, associative note and dimension mark properties dialog box. One additional way to change the unit is to add individual settings at the end of the `contentattributes.lst` file.

The `contentattributes_global.lst` file also defines default values for attributes in templates created in Template Editor.

If you change the unit and format, save the changes for future usage in a property file if necessary.

WARNING Do not edit `contentattributes_global.lst`.

For user-defined attributes in mark elements, the default unit settings are taken from the `contentattributes_userdefined.lst` file. You can use `contentattributes_userdefined.lst` also when you want to configure settings of your own. By default, this file is located in `..\Program Files\Tekla Structures\<version>\bin\applications\Tekla\Tools\TplEd\settings`, but often the location depends on your environment. The files are read from different locations in a certain search order.

The container file `contentattributes.lst` lists all the files that contain the actual attribute definitions. The order of the files included in `contentattributes.lst` defines the reading order of the files.

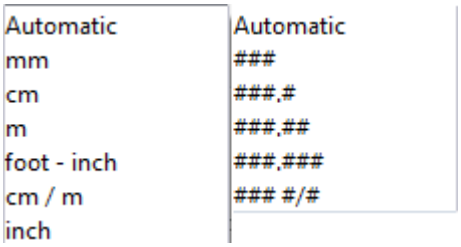
TIP You can add in the mark a template that changes the unit and number of decimals. This affects all drawings that have marks containing the changed template.

Example 1

This first example shows how you can change the unit and format of a length element in the part mark, save your changes in a property file and apply the changes in a part mark.

1. Double-click the background of an open drawing to open the **Drawing Properties** dialog box.
2. Click **Part mark**.
3. Add a **Length** element in the part mark.
4. Select **Length** from the **Elements in mark** list and adjust the unit and format as required. For example, select **mm** and **###.##**.

   The unit and format settings only become available when you select the **Length** element in the **Elements in mark** list.



5. Give a name to the property file next to the **Save as** button and click **Save as**.

   Now you can load this file later on when you need to use the same unit and format again.
6. Click **Modify**.

   All the part marks in your drawing now have the new unit and format setting for the length element.

Example 2

This second example shows a situation when you want a certain project to have certain individual settings. In this case, you can copy `contentattributes.lst` and `contentattributes_global.lst` to the model folder, and add this individual setting at the end of the `contentattributes.lst`. See the example below showing the included global attributes and user attributes files, and the added `DIAMETER` setting.

```
// ------------------------------------------------------------------------
// Template Editor 3.20 - Attributes
// ============================
//
// This is a container file where all needed attribute files are included.
//
// ------------------------------------------------------------------------

// Global attributes defined in source code
[INCLUDE .\settings\contentattributes_global.lst]

// User attributes defined in 'objects.inp'
[INCLUDE contentattributes_userdefined.lst]

DIAMETER                        FLOAT       RIGHT    TRUE       5       2         Length     mm

[BINDINGS] // Do NOT remove this line
```

See also

Mark elements (page 981)

Add templates in automatic marks (page 839)

Add attributes in automatic marks

All types of building object marks allow you to add user-defined attributes and template attributes. For example, you might want to add control numbers or specify the number of characters in part numbers in assembly or part marks.

You can add user-defined attributes and template attributes in automatic and manual marks.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. **Single-part, assembly and cast unit drawings:** Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. Click **Part mark**.
5. In the mark properties dialog box, double-click the **User-defined attribute** element in the **Available elements** list to add it in the mark.
6. Enter the user-defined attribute name in the **Mark content - user defined attribute** dialog box exactly as it appears in the `objects.inp` file.

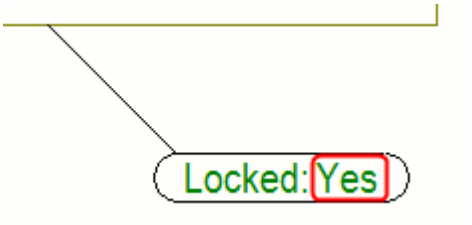
   If you need a template attribute in your mark, enter that instead.
7. **Single-part, assembly and cast unit drawings:** Click **Save** to save the changes in view properties and **Close** to return to drawing properties.

   General arrangement drawings: Click **OK**.

8. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

Example

In the following example, the user-defined attribute `OBJECT_LOCKED` has been added in the part mark after a text element **Locked:**.



See also

Add level attributes in automatic part marks (page 836)

Mark elements (page 981)

Add level attributes in automatic part marks

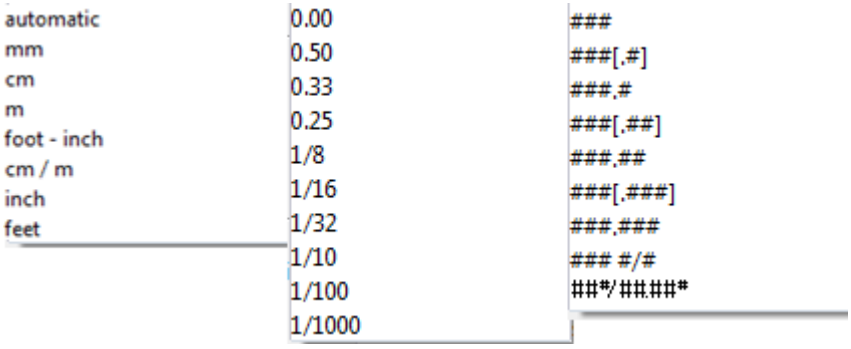
You can add level attributes, such as `TOP_LEVEL`, `BOTTOM_LEVEL`, `ASSEMBLY_TOP_LEVEL`, `ASSEMBLY_BOTTOM_LEVEL`, and `ASSEMBLY.MAIN_PART.TOP_LEVEL`, in part marks as user-defined attributes.

The level attributes take the dimension format from the `MarkDimensionFormat.dim` file. To change the dimension format for a dimension rule, change the dimension format in the dimension properties property pane in an open drawing, save the dimension properties, and load the changed dimension properties in the dimensioning rule that you use for creating the dimensions in a view.

You can add level attributes in automatic and manual marks.

To change the dimension format and add level attributes:

1. In an open drawing, click to open the property pane, and select **Straight dimension** from the property pane object list.
2. Select `MarkDimensionFormat` from the properties file list at the top.
3. Change the unit, precision and format as desired.



4. Click **Save** to save the changes in the `MarkDimensionFormat` file.
5. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
6. Load drawing properties that are as close to the ones you need as possible.
7. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
8. Click **Part markPart mark**.
9. In part mark properties, double-click the **User-defined attribute** element to add it to the mark.
10. Enter the attribute name in the **Mark content - user defined attribute** dialog box.

    You can enter the following values:

    `TOP_LEVEL`

    `TOP_LEVEL_UNFORMATTED`

    `TOP_LEVEL_GLOBAL`

    `TOP_LEVEL_GLOBAL_UNFORMATTED`

    `TOP_LEVEL_PROJECT`

    `TOP_LEVEL_BASEPOINT`

    `BOTTOM_LEVEL`

    `BOTTOM_LEVEL_UNFORMATTED`

    `BOTTOM_LEVEL_GLOBAL`

    `BOTTOM_LEVEL_GLOBAL_UNFORMATTED`

    `BOTTOM_LEVEL_PROJECT`

    `BOTTOM_LEVEL_BASEPOINT`

    `ASSEMBLY_TOP_LEVEL`

    `ASSEMBLY_TOP_LEVEL_UNFORMATTED`

ASSEMBLY_TOP_LEVEL_GLOBAL

ASSEMBLY_TOP_LEVEL_GLOBAL_UNFORMATTED

ASSEMBLY_BOTTOM_LEVEL

ASSEMBLY_BOTTOM_LEVEL_UNFORMATTED

ASSEMBLY_BOTTOM_LEVEL_GLOBAL

ASSEMBLY_BOTTOM_LEVEL_GLOBAL_UNFORMATTED

CAST_UNIT_TOP_LEVEL

CAST_UNIT_BOTTOM_LEVEL

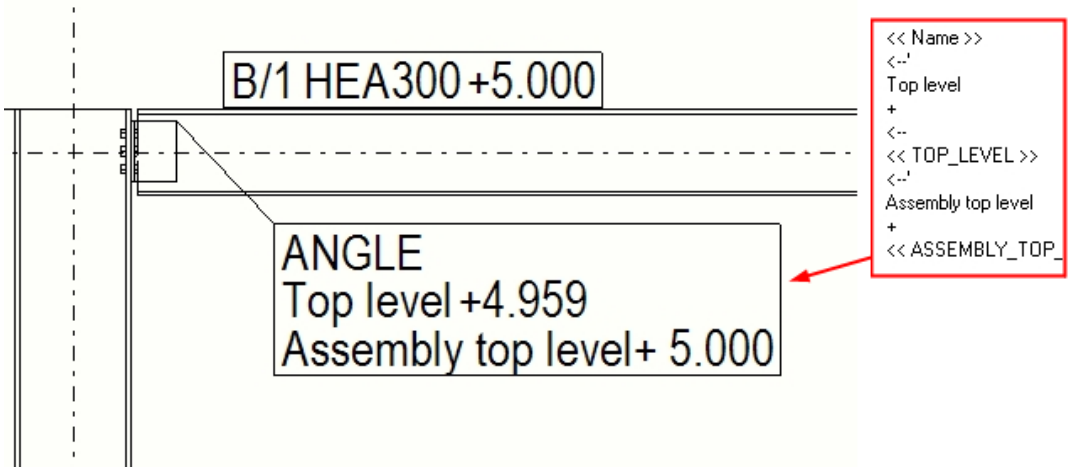
and/or the following:

ASSEMBLY.MAIN_PART.TOP_LEVEL.

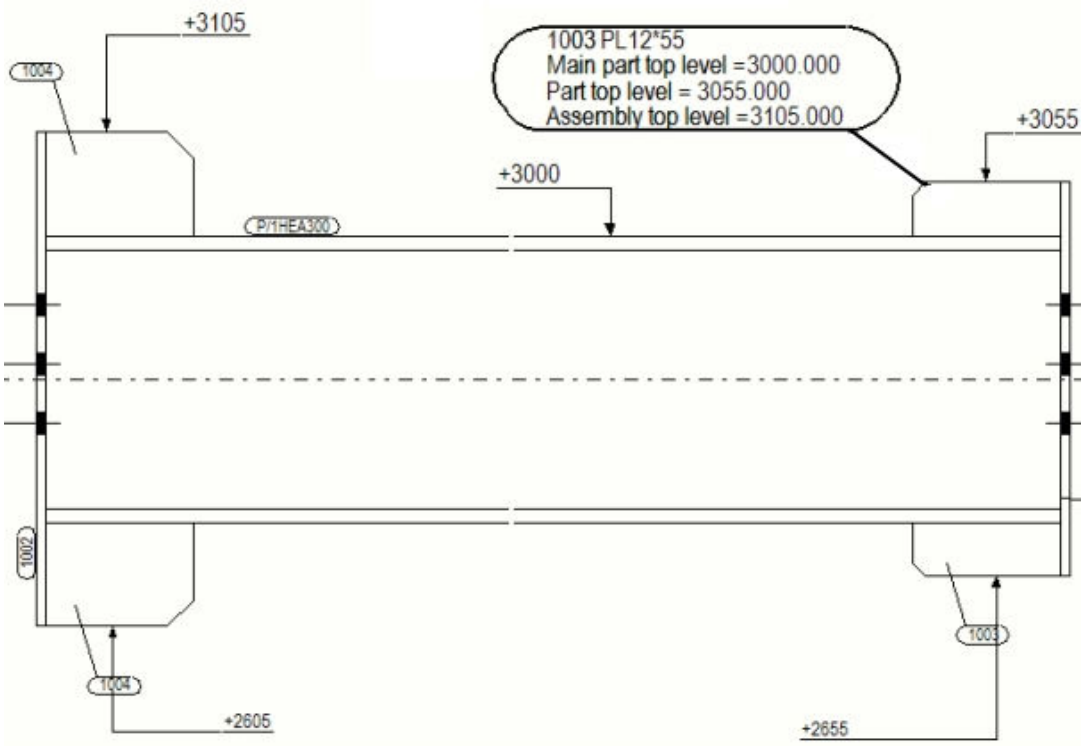
11. Click **Dimensioning** in the options tree.
12. Select a dimension rule from the list and click **Edit rule**.
13. Select MarkDimensionFormat from the **Dimension properties** list.
14. Save the dimensioning rule by clicking **Save** and click **Close**.
15. Save the view properties clicking **Save**.
16. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

Example

In the following example, TOP_LEVEL and ASSEMBLY_TOP_LEVEL have been added in the mark.



In the following example, the top level of the part itself (TOP_LEVEL), the top level of the assembly (ASSEMBLY_TOP_LEVEL), and the top level of the assembly main part (ASSEMBLY.MAIN_PART.TOP_LEVEL) have been added in the mark.



See also

Add attributes in automatic marks (page 835)

Units and decimals in drawings, reports and templates (page 893)

Add templates in automatic marks

You can create custom graphical templates (`.tpl`) with Template Editor and add them as elements in all types of marks, dimension marks and associative notes in all drawing types.

In mark templates, you can include detailed information of an embed or assembly, such as the sub-material used. Or you can use a template that changes the unit and the number of decimals in measurement values in a mark. You can also add graphical objects using the Template Editor tools.

To add templates in dimension marks or tags, you need to modify the dimension properties in the property pane in an open drawing. You can save the dimension properties, and then take the saved properties into use when you create automatic dimensions (page 730).

The size of the templates in part marks is calculated according to the actual size of template contents. Only lines and texts in the template are considered when calculating the exact size. This means, for example, that circles or bitmaps in the template do not have any effect.

Limitations: Mark templates do not support image files like the other graphical drawing templates.

Before you add a template in a mark, ensure that the template you use does not contain any margins.

Add templates in marks

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. **Single-part, assembly and cast unit drawings:** Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. Click the mark type that you want to change.

   For example, click **Part mark**.
5. In the mark properties dialog box, double-click the **Template** element in the **Available elements** list to add it in the mark.

   This will remove all other elements from the mark.
6. Select a template from the list in the **Mark content - template** dialog box. If you have not created a template yet, or want to edit the template, you can do that from here by selecting **Create new...** or **Edit...**

   Remember that if you edit the template here, the change affects all drawings that have marks containing the changed template.
7. Click **OK** to return to mark properties.
8. Save the mark properties for later using a unique name.
9. Do one of the following depending on the drawing type:
   - **Single-part, assembly and cast unit drawings:** Click **Save** to save the changes in view properties and **Close** to return to drawing properties.
   - **General arrangement drawings:** Click **OK**.
10. Click **Save** to save the drawing properties, then click **OK** and create the drawing.
11. When necessary, copy the saved mark properties files from the `<model>\attributes` folder into your firm or project folder.

TIP The mark templates are by default searched from the following folders in the following order:

`%XS_TEMPLATE_DIRECTORY%\mark`

`ModelDir\mark`

`%XS_PROJECT%\mark`

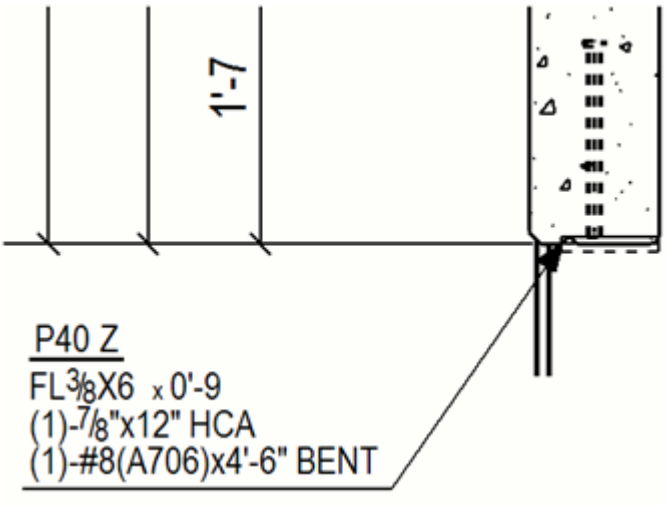
`%XS_FIRM%\mark`

`%XS_TEMPLATE_DIRECTORY_SYSTEM%\mark`

`%XS_SYSTEM%\mark`

The mark folder name can be changed using the advanced option XS_TEMPLATE_MARK_SUB_DIRECTORY.

Example of a template in a mark



Example 1: Create a mark template containing separate value fields and text elements

You can use decimals instead of fractions in your part marks in the US environment Imperial role by using a mark template. The template contains separate value fields and text elements that change the fractions to decimals, and change the number of decimals.

1. Follow the steps 1 - 5 in "Add templates to marks" above so that you have double-clicked the **Template** element in the **Available elements** list .
2. In the **Mark content - template** dialog box, click **Create new**.

   This starts the Template Editor.
3. Click **File** --> **New** and create a new graphical template.
4. Click **Insert** --> **Component** --> **Row** .
5. Select **PART** as the content type.
6. Click **Insert** --> **Text** , enter `L` and place it inside the row you just added.
7. Click **Insert** --> **Value field** and place the value field on the right side of the `L` text.

8. In the displayed **Select Attribute [Part]** dialog box, scroll down to **PROFILE - Profile**, open the profile tree, select the **HEIGHT - height** property and click **OK**.
9. Double-click the value field. In the **Value Field Properties** dialog box, change the settings as follows:
   - **Unit**: `inch`
   - **Decimals**: `1`
   - **Length**: `8`
10. Click **OK**.
11. Click **Insert** --> **Text** , enter `X` as text and place it on the right side of the value field.
12. In the similar way, add another value field for the width information (**WIDTH - Width** profile property).
13. Click **Insert** --> **Text** and add the second `X` between the value fields.
14. Add the third value field for the profile flange thickness by selecting **FLANGE_THICKNESS_1 - Flange thickness 1** profile property and modify the settings as follows:
    - **Unit**: `inch`
    - **Decimals**: `2`
    - **Length**: `4`
15. Click **Edit** --> **Properties** and minimize the height and the width of the row.
16. Click **File** --> **Save** to save the template.

    The template is by default saved as a `.tpl` file in the `\mark` folder under the model folder. You can copy this template to other models as required.
17. In Tekla Structures, click **Refresh list** in the **Mark content - template** dialog box to see the template you created.
18. Select the template and click **OK**.
19. Save the mark properties for late use with a unique name.
20. Do one of the following depending on the drawing type:
    - **Single-part, assembly and cast unit drawings:** Click **Save** to save the changes in view properties and **Close** to return to drawing properties.
    - **General arrangement drawings:** Click **OK**.
21. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

Example 2: Create a mark template containing a formula in the value field

You use decimals instead of fractions in your part marks in the US environment Imperial role by adding a formula in the mark template value field.

1. Follow the steps 1 - 5 in "Add templates to marks" above so that you have double-clicked the **Template** element in the **Available elements** list .
2. In the **Mark content - template** dialog box, click **Create new**.

   This starts the Template Editor.
3. Click **File** --> **New** and create a new graphical template.
4. Click **Insert** --> **Component** --> **Row** .
5. Select **PART** as the content type.
6. Click **Insert** --> **Value field** , and place the field.
7. In the displayed **Select Attribute [Part]** dialog box, click the **Formula** button.
8. Add the following formula in the **Formula** box and click **OK**:

   ```
   "L " + format(GetValue("HEIGHT"),"Length","inch",1) + " x "+ format(GetValue("WIDTH"),"Length","inch",1) + " x " + format(GetValue("PROFILE.FLANGE_THICKNESS_1"),"Length","inch",2)
   ```
9. Double-click the value field to open the **Value Field Properties** dialog box.
10. Set the **Data type** to text, enter a name for the field, for example, `PART_MARK`, and ensure that the length value you give in the **Length** field is high enough to fit all the characters and numbers included, for example, `20`.
11. Click **OK**.
12. Click **Edit** --> **Properties** and minimize the height and the width of the row.
13. Click **File** --> **Save** to save the template.

    The template is by default saved as a `.tpl` file in the `\mark` folder under the model folder. You can copy this template to other models as required.
14. In Tekla Structures, click **Refresh list** in the **Mark content - template** dialog box to see the template you created.
15. Select the template and click **OK**.
16. Save the mark properties for later use with a unique name.
17. Do one of the following depending on the drawing type:
    - **Single-part, assembly and cast unit drawings:** Click **Save** to save the changes in view properties and **Close** to return to drawing properties.

- **General arrangement drawings:** Click **OK**.

18. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

Add symbols in automatic marks

All types of marks allow you to add symbols in them. You can select the symbol file to be used and the symbol to be added in the mark.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. **Single-part, assembly and cast unit drawings:** Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. Click **Part mark**.
5. In mark properties, double-click **Symbol** in the **Available elements** list.
6. In the **Mark content - symbol** dialog box, click **Select...** next to the **File** box to select the symbol file you want to use.
7. When you have selected the file, click **Select...** next to the **Number** box to select the number of the symbol you want to use.
8. Click **OK**.

   Tekla Structures adds the name of the symbol file and the number of the symbol in the elements list.
9. Save the mark properties for late use with a unique name.
10. **Single-part, assembly and cast unit drawings:** Click **Save** to save the changes in view properties and **Close** to return to drawing properties.

    General arrangement drawings: Click **OK**.
11. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

See also

Add and modify symbols in drawings (page 371)

Define size in bolt marks using advanced options

You can use certain advanced options to define the contents of the bolt mark **Size** element in different types of drawings.

To change the bolt mark **Size** element content, go to **File menu** --> **Settings** --> **Advanced options** --> **Marking - bolts** .

NOTE
- General arrangement drawings have separate advanced options.
- Any changes made to the advanced options listed in the table below will take place only in the new drawings you create, and if you modify the affected bolt marks.

To	Do this
Define the contents of the size element in bolt marks.	Set a value for the advanced option XS_BOLT_MARK_STRING_FOR_SIZE. For GA-drawings, set a value for the advanced option XS_BOLT_MARK_STRING_FOR_SIZE_IN_GA.
Define the contents of the size element in bolt marks for workshop bolts.	Set a value for the advanced option XS_SHOP_BOLT_MARK_STRING_FOR_SIZE. For GA-drawings, set a value for the advanced option XS_SHOP_BOLT_MARK_STRING_FOR_SIZE_IN_GA.
Define the contents of the size element in bolt marks for site bolts.	Set a value for the advanced option XS_SITE_BOLT_MARK_STRING_FOR_SIZE. For GA-drawings, set a value for the advanced option XS_SITE_BOLT_MARK_STRING_FOR_SIZE_IN_GA.
Define the contents of the size element in slotted hole marks.	Set a value for the advanced option XS_LONGHOLE_MARK_STRING_FOR_SIZE. For GA-drawings, set a value for the advanced option XS_LONGHOLE_MARK_STRING_FOR_SIZE_IN_GA.
Define the contents of the size element in slotted hole marks for workshop bolts.	Set a value for the advanced option XS_SHOP_LONGHOLE_MARK_STRING_FOR_SIZE. For GA-drawings, set a value for the advanced option XS_SHOP_LONGHOLE_MARK_STRING_FOR_SIZE_IN_GA.

To	Do this
Define the contents of the size element in slotted hole marks for site bolts.	Set a value for the advanced option XS_SITE_LONGHOLE_MARK_STRING_FOR_SIZE. For GA-drawings, set a value for the advanced option XS_SITE_LONGHOLE_MARK_STRING_FOR_SIZE_IN_GA.
Define the contents of the size element in marks for oversized bolt holes (workshop).	Set a value for the advanced option XS_SHOP_OVERSIZEDHOLE_MARK_STRING_FOR_SIZE. For GA-drawings, set a value for the advanced option XS_SHOP_OVERSIZEDHOLE_MARK_STRING_FOR_SIZE_IN_GA.
Define the contents of the size element in marks for oversized bolt holes (site).	Set a value for the advanced option XS_SITE_OVERSIZEDHOLE_MARK_STRING_FOR_SIZE. For GA-drawings, set a value for the advanced option XS_SITE_OVERSIZEDHOLE_MARK_STRING_FOR_SIZE_IN_GA.
Define the contents of the size element in marks for tapped bolt holes (workshop).	Set a value for the advanced option XS_SHOP_TAPPEDHOLE_MARK_STRING_FOR_SIZE. For GA-drawings, set a value for the advanced option XS_SHOP_TAPPEDHOLE_MARK_STRING_FOR_SIZE_IN_GA.
Define the contents of the size element in marks for tapped bolt holes (site).	Set a value for the advanced option XS_SITE_TAPPEDHOLE_MARK_STRING_FOR_SIZE. For GA-drawings, set a value for the advanced option XS_SITE_TAPPEDHOLE_MARK_STRING_FOR_SIZE_IN_GA.
Define the contents of the size element in hole marks.	Set a value for the advanced option XS_HOLE_MARK_STRING_FOR_SIZE. For GA-drawings, set a value for the advanced option XS_HOLE_MARK_STRING_FOR_SIZE_IN_GA.

To	Do this
Define the contents of the size element in hole marks for workshop bolts.	Set a value for the advanced option XS_SHOP_HOLE_MARK_STRING_FOR_SIZE. For GA-drawings, set a value for the advanced option XS_SHOP_HOLE_MARK_STRING_FOR_SIZE_IN_GA.
Define the contents of the size element in hole marks for site bolts.	Set a value for the advanced option XS_SITE_HOLE_MARK_STRING_FOR_SIZE. For GA-drawings, set a value for the advanced option XS_SITE_HOLE_MARK_STRING_FOR_SIZE_IN_GA.

You can use any combination of text and the following options as the value for the above advanced options. You can use the options in any order, and make calculations.

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

Enclose each option in % characters.

To use special characters, enter a backslash (\) followed by an ASCII number.

With length-type options, but not in calculations, you can specify the number of decimals. For example, HOLE.DIAMETER.2 shows (tapped) hole diameters with two decimals.

Example

Example of using the advanced options:

```
XS_LONGHOLE_MARK_STRING_FOR_SIZE=D%HOLE.DIAMETER%
(%HOLE.DIAMETER+LONG_HOLE_X%x%HOLE.DIAMETER+LONG_HOLE_Y%)
```

```
XS_SITE_HOLE_MARK_STRING_FOR_SIZE=D%HOLE.DIAMETER%
```

Please note that for calculations the operator (+, -, *, /) must not be outside the "%" marks:

Correct: `%OPTION1*OPTION2%`

Incorrect: `%OPTION1%*%OPTION2%`

For example, if `OPTION1` = 5.0 and `OPTION2` = 3.0, the results would be "15" and "5*3".

Add pull-out pictures in reinforcement marks

You can add a pull-out picture for a reinforcing bar in the reinforcement mark to illustrate the shape and dimensions of the bar in the drawing.

Pull-out pictures can be added in automatic and manual created reinforcement marks.

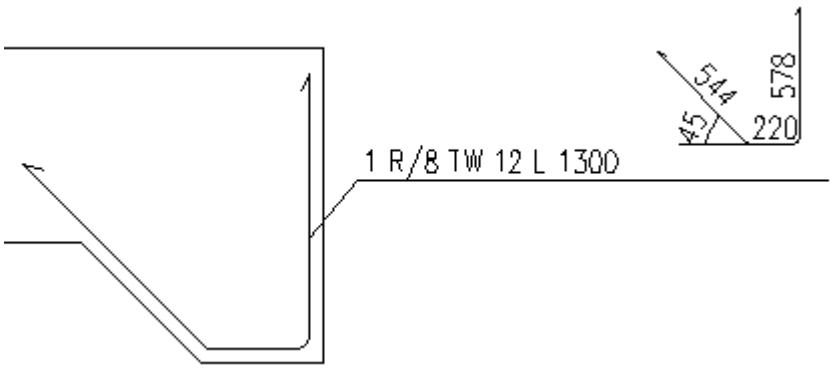
Add a pull-out picture in automatic reinforcement marks

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. **Cast unit drawings:** Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. Click **Reinforcement mark** in the options tree.
5. To add a pull-out picture in the reinforcement mark, double-click the **Pullout picture** element in the **Available elements** list.
6. In the **Pullout picture** dialog box, select the scaling option:
   - **Auto** auto-scales the pull-out picture.
   - **One factor** and **Two factors** scale the pull-out picture according to the values that you enter.

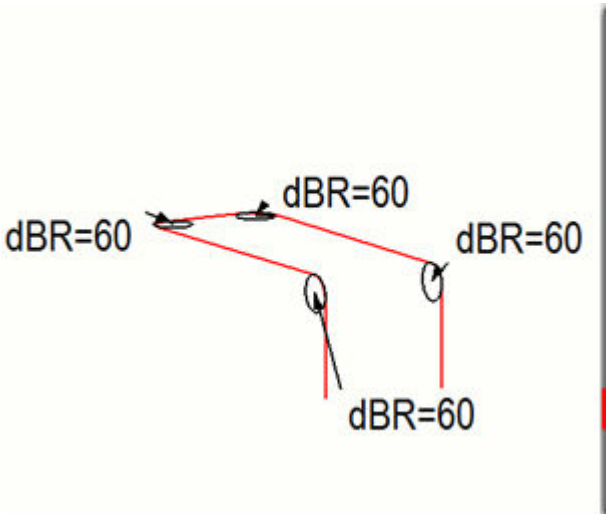
     The scale of the pull-out is relative to the scale of the drawing view. For example, if the drawing view scale is 1/10 and the pull-out scaling is 2, the actual scale of the pull-out in the drawing view is 1/5.

7. In **Rotation**, select the rotation of the pull-out: **Automatic**, **Plane** or **3D**. If the pull-out is 3D, and you select **Automatic**, Tekla Structures automatically shows the pull-out in 3D.
8. In **End marks**, define the shape of bar ends in the pull-out.
9. Select **Dimensions** to show bar dimensions in the pull-out.
10. Select **Exaggeration** to show reinforcing bar hooks more clearly in the pull-out.
11. Select **Bending radius** to show the bending radius in form of diameter of the bending roll.
12. Select **Bending angle** to show bar bending angles in the pull-out.
13. Select **Coupler/end anchor symbols** to show graphical symbols representing the rebar couplers and end anchors in pull-out pictures.
14. Select **Placeholders** to show placeholders instead of dimensions in pull-out pictures. In the **Pullout picture** dialog box in object level reinforcement mark properties, the **Placeholders** option is only available if the XS_ENABLE_PULLOUT_PLACEHOLDERS advanced option is set to `TRUE`. In the **Pullout picture** dialog box in view level reinforcement mark properties, this option is always available.
15. Click **OK**.
16. Save the mark properties for future use with a unique name.
17. **Cast unit drawings:** Click **Save** to save the changes in view properties and **Close** to return to drawing properties.

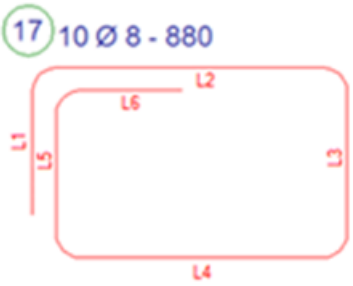
    General arrangement drawings: Click **OK**.
18. Click **Save** to save the drawing properties, then click **OK** and create the drawing.



A 3D pull-out showing the bending radius:



Placeholders shown instead of dimensions:



	L1	L2	L3	L4	L5	L6	Length
17.1	10	24	14	24	14	10	89
17.2	10	24	19	24	19	10	98
17.3	10	24	23	24	23	10	107
17.4	10	24	27	24	27	10	115
17.5	10	24	32	24	32	10	124
17.6	10	24	36	24	36	10	133
17.7	10	24	41	24	41	10	142
17.8	10	24	45	24	45	10	150
17.9	10	24	49	24	49	10	159
17.10	10	24	54	24	54	10	168

TIP To change the leader line length, color, line type, or representation of a pull-out, open the (page 1023) file located in the system folder defined by the advanced option `XS_SYSTEM` and edit the following lines: `PullOutLeaderLineMinLength`, `PulloutColor`, `PulloutVisibleLinetype` and `PulloutRepresentation`.

Add a pull-out picture in a reinforcement mark manually

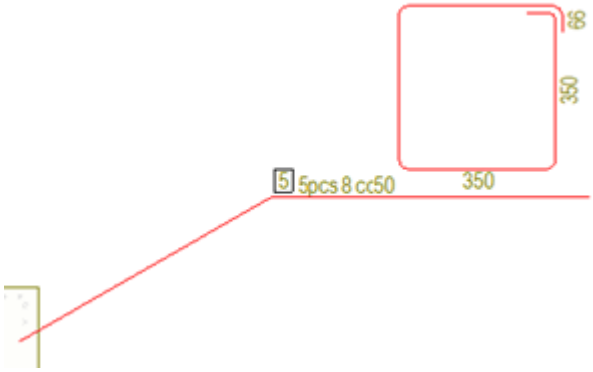
1. In an open drawing containing reinforcement and reinforcement marks, double-click a reinforcement mark.

   The reinforcement mark properties are displayed in the property pane.
2. In the mark editor preview, click a location the mark where you want to add the pull-out picture.
3. To add a pull-out picture element in the mark, click the **New element** button in the editor and select **Pullout picture**.
4. In the **Pullout picture** dialog box, define the pull-out picture properties.

   For property details, see the instructions for automatic marks above.
5. Click **OK**.

6. To apply the changes to the reinforcement mark, click **Modify** in the property pane.

Tekla Structures adds the pull-out picture in the mark.



See also

Mark elements (page 981)

8.7 Define drawing parts and neighbor parts

Parts and neighbor parts in drawings are building objects that will exist in the real structure or which will be closely related to the structure.

Parts and neighbor parts have drawing properties that affect the way that the part is shown in the drawing.

To	Click the links below to find out more
Define what is shown in a part and how the part is shown	Define automatic drawing part properties (page 852)
Define what is shown of neighbor parts and how the neighbor parts are shown. You can also control the visibility of the neighbor part bolts. Also check some examples of the part settings.	Define automatic neighbor part properties (page 856)
Indicate the part orientation by using part marks, by including compass direction in marks, and showing orientation marks and connecting side marks	Indicate part orientation (page 857)
Check and change part and neighbor part properties	Part and neighbor part properties in drawings (page 1004)

Define automatic drawing part properties

You can define what is shown in a part and how the part is shown.

Limitations: The color of the center lines can be changed only on the drawing and view level, not on the object level. For center lines, you can only adjust the color in object properties dialog box. You can adjust the line type of part center lines with the advanced option XS_CENTER_LINE_TYPE.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. **Single-part, assembly and cast unit drawings:** Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**..
4. Click **Part**.
5. On the **Content** tab, select the part representation, whether to show hidden lines, center lines and reference lines, and which additional markings to show.

   A reference line is a line between the points from which a part is created.

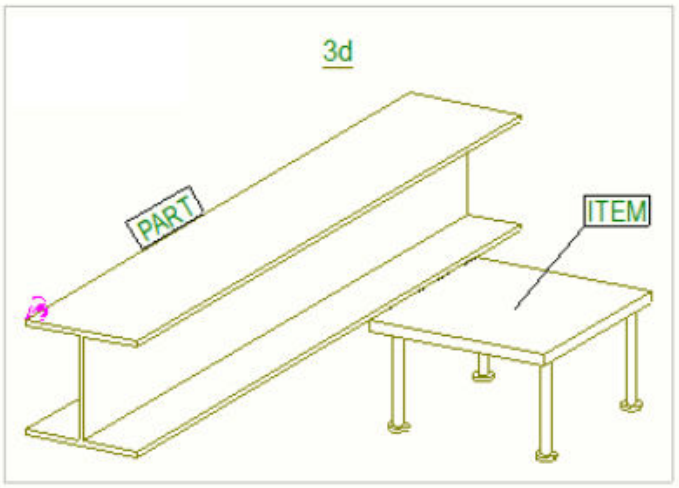
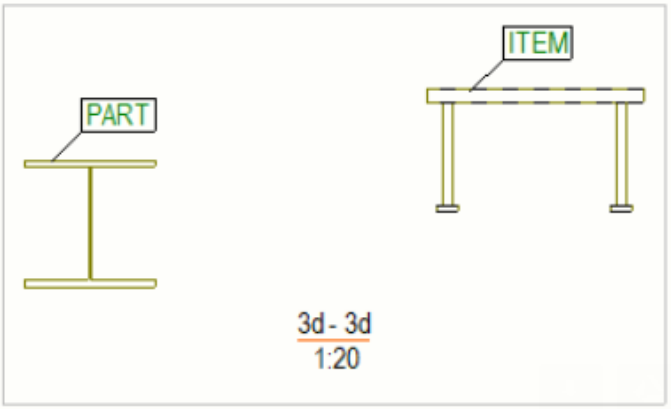
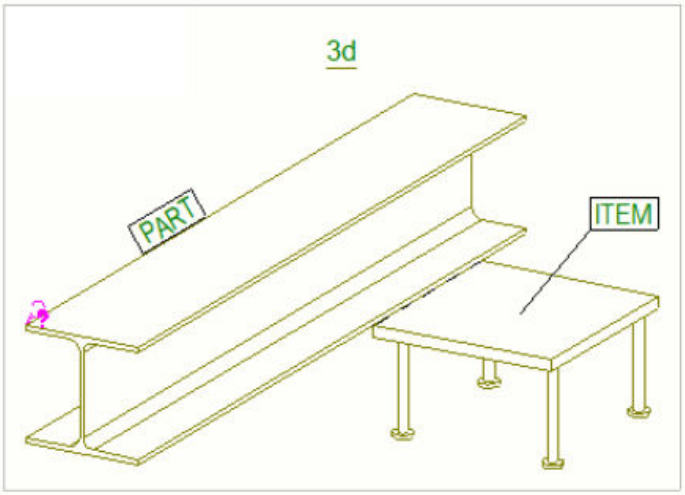
   Note that **Bounding box** is a good option to use with complex items with an extrema box containing many polygons slowing down drawings, because **Bounding box** makes drawings faster.
6. On the **Appearance** tab, select the color and type of the lines.

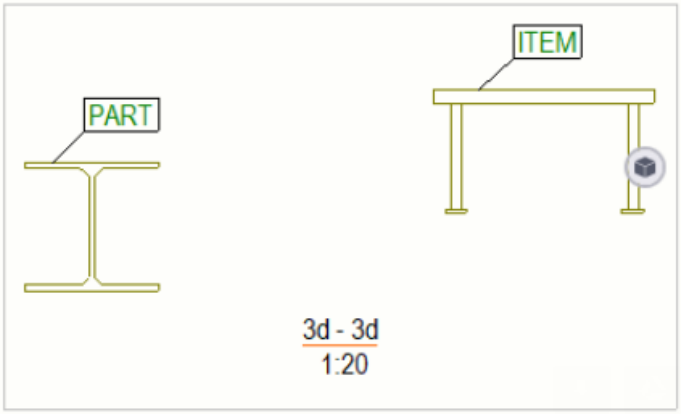
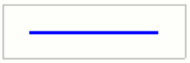
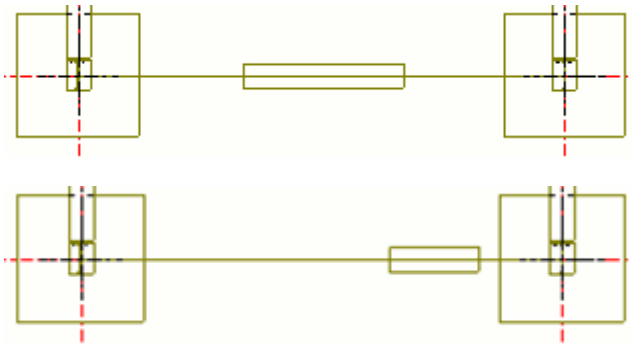
   The color of the center lines can be changed only on the drawing and view level, not on the object level. For center lines, you can only adjust the color in the properties dialog box, not the type. You can adjust the line type of part center lines with the advanced option `XS_CENTER_LINE_TYPE`.
7. On the **Fill** tab, set the part and section fill options.
8. Do one of the following depending on the drawing type:
   - **Single-part, assembly and cast unit drawings:** Click **Save** to save the view properties. Then return to drawing properties by clicking **Close**.
   - **General arrangement drawings:** Click **OK** to return to drawing properties.
9. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

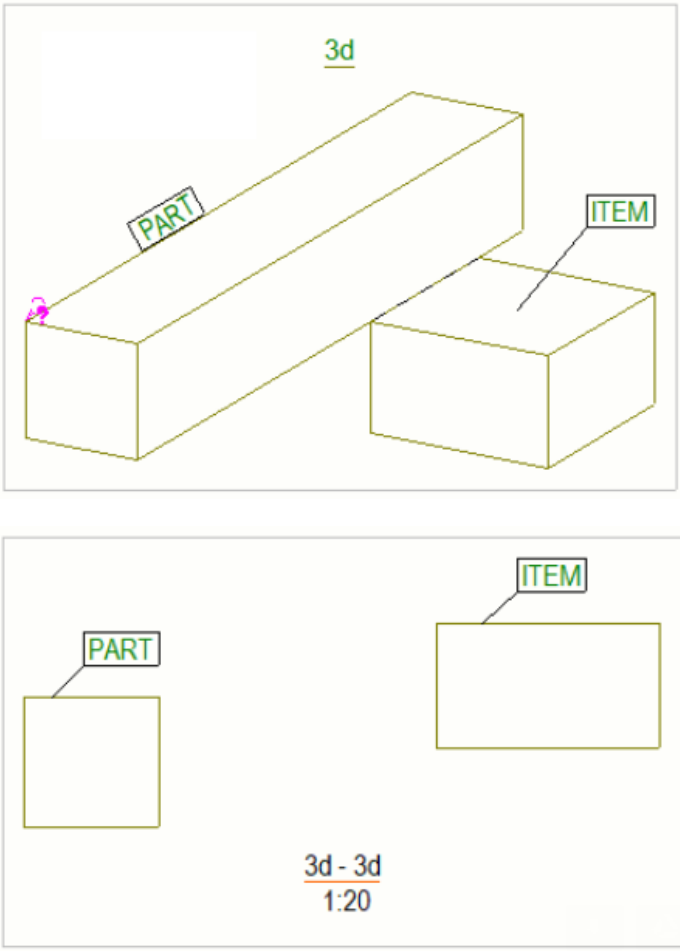
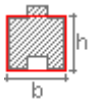
You can also change the part reference line type using the advanced option XS_DRAWING_PART_REFERENCE_LINE_TYPE, and use the advanced XS_DRAWING_POINT_SCALE to scale the points Tekla Structures uses to create reference lines.

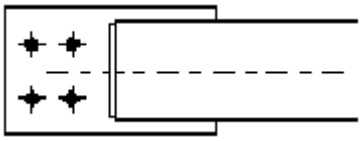
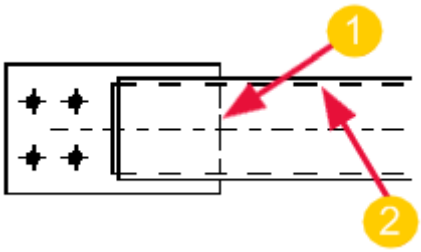
Example: Part representations

Here are some examples of what parts and items look like with different settings defined in the **View Properties** dialog box.

Setting	Example
Representation is set to **Outline**.	 
Representation is set to **Exact**.	

Setting	Example
	
Edge chamfers is selected in **Additional marks**.	
Representation is set to **Symbol**.	
Representation is set to **Symbol with partial profile**. You can change the partial profile settings **Length** and **Offset from middle point**. In the first example on the right, the default values are used. In the second example, both the length and the offset have been adjusted.	

Setting	Example
Representation is set to **Bounding box**.	
Representation is set to **Bounding box**.	
Representation is set to **Base box**.	
Representation is set to **Exact** and **Symbol offset** is set to `0.00`. **Center line** is selected.	
Representation is set to **Exact** and **Symbol offset** is set to `10.00`. **Center line** is selected.	

Setting	Example
Hidden lines is **not** selected.	
(1) Hidden lines is selected. Hidden lines for other parts are shown. **(2) Own hidden lines** is selected. Hidden lines for the main part are shown.	

See also

Part and neighbor part properties in drawings (page 1004)

Define automatic neighbor part properties

You can define what is shown of neighbor parts and how the neighbor parts are shown. You can also control the visibility of the neighbor part bolts.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Single-part, assembly and cast unit drawings: Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. Click **Neighbor part**.
5. Select the desired visibility options on the **Visibility** tab.
   - For more information, see Show neighbor parts in drawings (page 705).
6. On the **Content** tab, select the neighbor bolt representation, whether to show hidden lines, center lines and reference lines, and which additional markings to show.

   A *reference line* is a line between the points from which a part is created.
7. On the **Appearance** tab, select the color and type of the lines.

The color of the center lines can be changed only on the drawing and view level, not on the object level. For center lines, you can only adjust the color in the properties dialog box, not the type.

You can adjust the line type of part center lines with the advanced option XS_CENTER_LINE_TYPE.

8. Single-part, assembly and cast unit drawings: Click **Save** to save the view properties. Then return to drawing properties by clicking **Close**.
9. General arrangement drawings: Click **OK** to return to drawing properties.
10. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

You can also change the neighbor part reference line type using the advanced option XS_DRAWING_PART_REFERENCE_LINE_TYPE, and use the advanced XS_DRAWING_POINT_SCALE to scale the points Tekla Structures uses to create reference lines.

See also

Define drawing parts and neighbor parts (page 851)

Part and neighbor part properties in drawings (page 1004)

Indicate part orientation

Part orientation marks indicate the erection direction of assemblies and cast units. There are many ways available for indicating part orientation: by using part marks, by including compass direction in marks, and showing orientation marks and connecting side marks.

To check how the location of the mark is affected by mark placement settings, the type of the leader line, predefined mark location and part orientation settings, modeling direction of parts, and drawing protection settings, see Mark location (page 824).

Use part mark as an orientation mark in general arrangement drawings

In general arrangement drawings, part marks appear at the same end as in assembly drawings. Parts with the same assembly position are always marked at the same end.

1. Select **Drawings & reports** --> **Drawings properties** --> **Assembly drawing** .
2. Load drawing properties that are as close to the ones you need as possible.

3. In assembly drawing properties, click **View creation** --> **Attributes** and set the **Coordinate system** to **oriented** or **model**.

   You must use **oriented** or **model** if you want the **Viewing direction** settings in **File menu** --> **Settings** --> **Options** --> **Orientation marks** to have effect.

4. Click **Save** to save the properties and **Close** to close the dialog box.
5. In the model, click **Drawings & reports** --> **Numbering settings** --> **Numbering settings** and clear the **Beam orientation** and **Column orientation** check boxes.

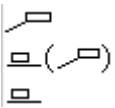
   When you do this, similar assemblies get the same number regardless of the part orientation.

6. Go to **File menu** --> **Settings** --> **Options** --> **Orientation marks** and set the following options:
   - Set the desired **Viewing direction** for beams, bracings and columns.
   - Set **Mark always to center of column in GA drawings** to **No**.
   - Set **Preferred location for beams and bracings** to **Left** or **Right**.

     The preferred location determines the end where the mark is placed.

7. Click **OK**.
8. Check the leader line type through **Drawings & reports** --> **Drawings properties** --> **GA drawing** --> **Part marks** --> **General** .

   Place the mark near the part end, not in the middle of the part using one of the following settings:

9. Click **Save** to save the drawing properties and **OK** to close the dialog box.
10. Create assembly (and single-part) drawings using the settings you modified.

    If you have not created a drawing earlier, the modeling direction of the part that has the smallest id is used, except if the top-in-form face has been defined.

11. Create general arrangement drawings using the settings you modified.

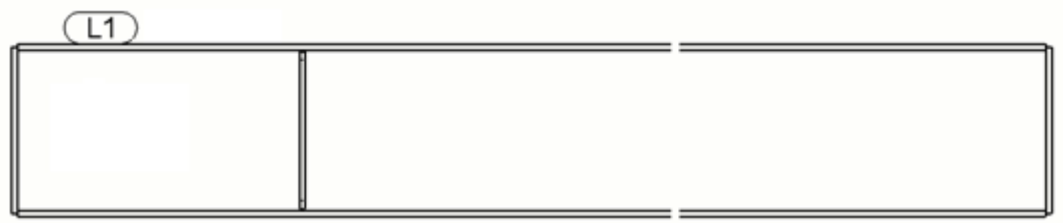
NOTE
- In general arrangement drawings, the cast unit part marks behave in the same way as the assembly part marks, except when **Top in form face** is set to **Front** or **Back** on the **Parameters** tab of the part's **User-defined attributes** dialog box. These settings tell the end where the part mark is placed.
- When you update an assembly drawing so that the mark is placed at the other end of the part, remember to update the marks in the

corresponding general arrangement drawing. Tekla Structures does not do this automatically.

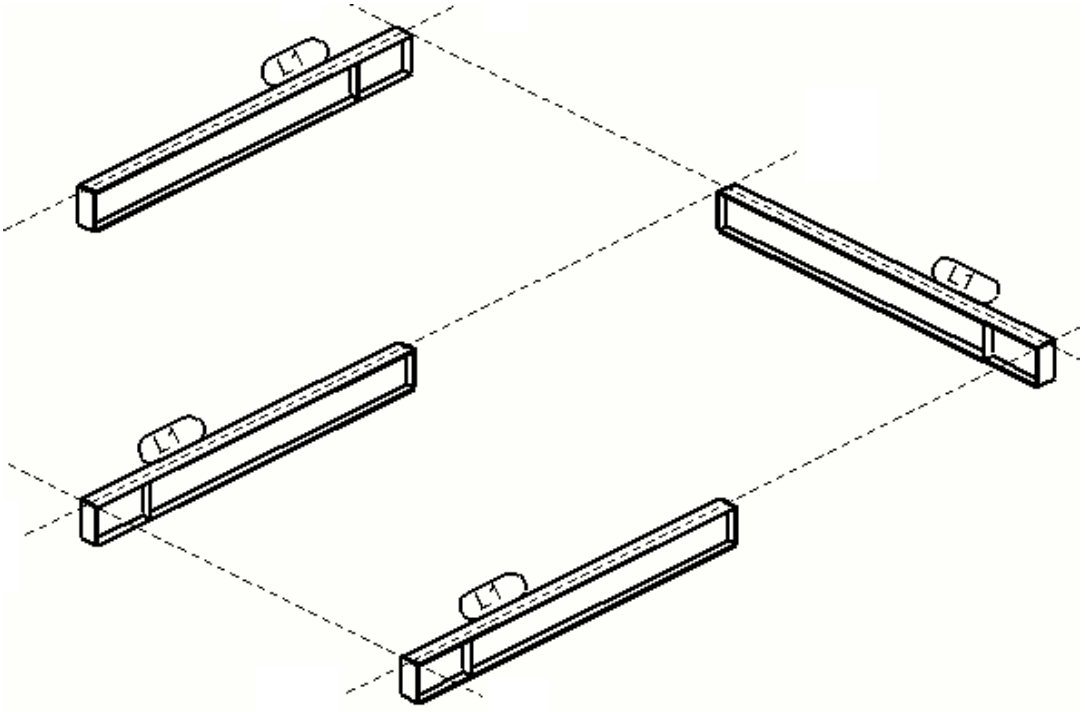
- Also note that if you have set the advanced option XS_UPSIDE_DOWN_TEXT_ALLOWED to TRUE, the text reading direction indicates the part installation direction.

Example

The part mark is positioned at the stiffener side:



The marks are positioned at the same side irrespective of the orientation of the beams:



Show compass direction in part marks

You can include face direction information in part marks. No matter where you add or move the mark, the face direction stays the same. The face direction

indicates the compass direction (North, East, South, West) of the face where the mark appears.

Limitation: Tekla Structures indicates the face direction only if it is the same for all assemblies or cast units with the same assembly or cast unit position number.

1. Check in which direction north is in the model by clicking **File** --> **Settings** --> **Options** --> **Orientation marks** and checking **Project north (degrees counterclockwise from global x)**.

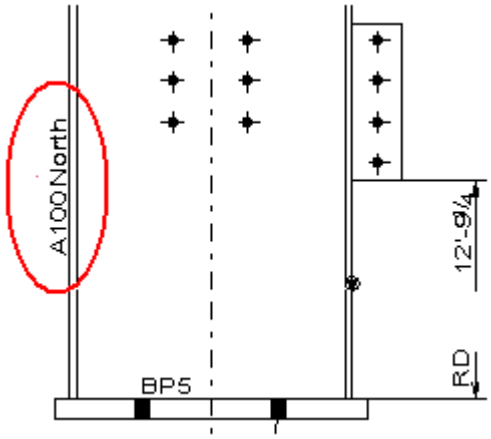
   For more information about the orientation mark settings in the **Options** dialog box, see Settings in Options dialog box.

2. In the model, click **Drawings & reports** --> **Numbering settings** and select the **Column orientation** check box.

   This forces Tekla Structures to show the face direction in the part mark for two similar columns with different orientation.

3. In the desired assembly or cast unit drawing view properties, click **Part mark** and insert the **Face direction** element in the part mark.

Now the assembly or cast unit drawings show the face direction in the part marks.



TIP To show compass direction in GA drawings, go to **File** --> **Settings** --> **Options** --> **Orientation marks** and set **Mark always to center of column in GA drawings** to **No**.

Show orientation marks (north marks)

You can use orientation marks or north marks to indicate the erection direction of assemblies. The default orientation mark is a triangle inside a circle.

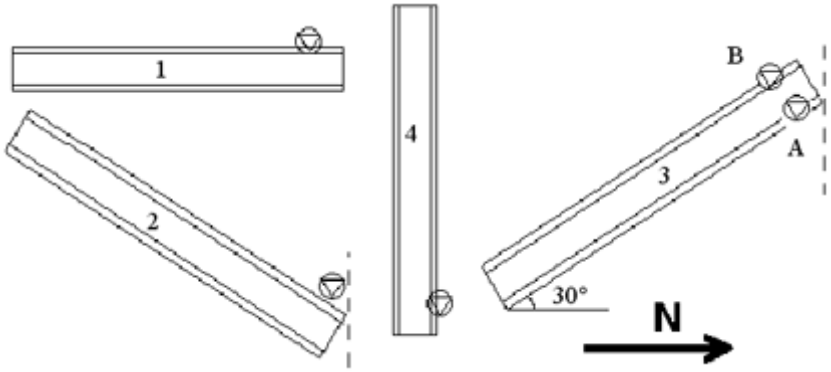
NOTE The orientation mark is not drawn if the part is perpendicular to the view, for example, for columns in a plan view.

1. On the **Drawings & reports** tab, click **Drawing properties** --> **Assembly drawing**.
2. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
3. Click **Part** --> **Content** and select the **Orientation marks** check box.
4. Save the view properties and click **Close**.
5. Save the assembly drawing properties and click **OK**.
6. If needed, define which parts Tekla Structures considers to be columns, braces or beams by setting the skew limits through **File menu** --> **Settings** --> **Options** --> **Orientation marks** --> **Skew limit** .
7. Use the following advanced options to adjust orientation marks:
   - XS_ORIENTATION_MARK_DIRECTION
   - XS_ORIENTATION_MARK_MOVE_DIST_FOR_BEAMS
   - XS_ORIENTATION_MARK_MOVE_DIST_FOR_COLUMNS
   - XS_NORTH_MARK_SYMBOL
   - XS_HIDDEN_NORTH_MARK_SYMBOL
   - XS_NORTH_MARK_SCALE
   - XS_GA_NORTH_MARK_SYMBOL (general arrangement drawings)
   - XS_GA_HIDDEN_NORTH_MARK_SYMBOL (general arrangement drawings)
   - XS_GA_NORTH_MARK_SCALE (general arrangement drawings)
8. Create the assembly drawing.

For different parts, Tekla Structures draws orientation marks in the following way:

- To the top flange of beams, at the end which points closest to the north or to the direction defined in `XS_ORIENTATION_MARK_DIRECTION` (see part 1 and 2 in the illustration below)
- To the lower end of columns, on the flange which points closest to the north or to the direction defined in `XS_ORIENTATION_MARK_DIRECTION` (see part 4 in the illustration below)

- To the flange of bracing, at the end which points closest to the north or to the direction defined in `XS_ORIENTATION_MARK_DIRECTION` (see 3 A and B in the illustration below)



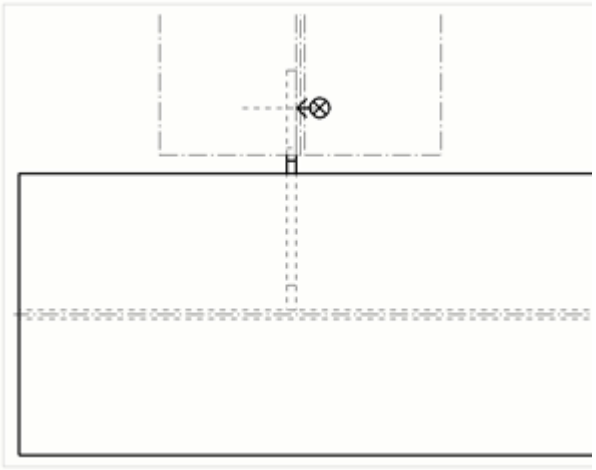
TIP You can display orientation marks for single-part views included in assembly drawings by setting the advanced option XS_SINGLE_ORIENTATION_MARK to `TRUE`.

Show connecting side marks

You can use a symbol in assembly drawings to indicate the side of a part to which a connecting part is attached.

Note that connecting side marks can be set from drawing view level properties to secondary parts only (typically shear plates like shown in the example image below). The connecting side mark to the main part of the assembly can be set on the object level in the drawing part properties.

1. Select **Drawings & reports** --> **Drawings properties** --> **Assembly drawing** .
2. Load drawing properties that are as close to the ones you need as possible.
3. Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. Click **Part**.
5. On the **Content** tab, select the **Connecting side marks** check box.
6. Click **Save** to save the view properties.
7. Click **Close**.
8. Click **Save** to save the drawing properties, then click **OK** and create the drawing.



TIP If needed, you can change the connecting side mark symbol with the advanced optionXS_CONNECTING_SIDE_MARK_SYMBOL, and useXS_MIN_DISTANCE_FOR_CONNECTING_SIDE_MARK to control whether the connecting side mark is drawn or not.

8.8 Define drawing bolts

Bolts are connecting objects that fasten parts or assemblies or attach to them. You can change how the bolts are shown in drawings.

To	Click the links below to find out more
Change how the bolts are shown in drawings, and check representation examples	Define automatic bolt properties in drawings (page 863)
Create user-defined bolt symbols	Create customized bolt symbols (page 866)
Check bolt properties	Bolt properties in drawings (page 1010)

Define automatic bolt properties in drawings

You can define what is shown for bolts and how the bolts are shown.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.

3. Click **View creation** and check the **Ignore bolt size** setting. This setting filters standard-sized bolts out of drawings, which means that Tekla Structures does not display bolt marks of the defined bolt size in drawings. This setting is available in single-part and assembly drawings.
4. Single-part, assembly and cast unit drawings: Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
5. Click **Bolt**.
6. On the **Content** tab, select the bolt representation, bolt symbol content, and the visibility of bolts in main parts.

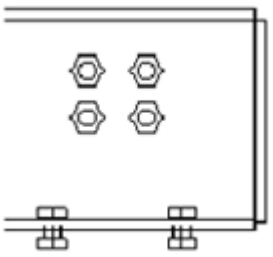
   For assembly and cast unit drawings, you can also set the visibility of bolts in secondary parts and sub-assemblies.

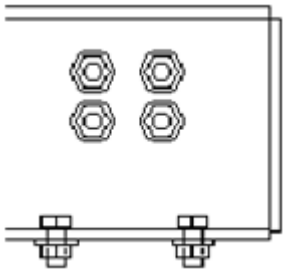
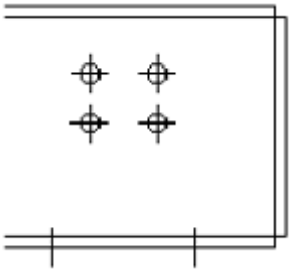
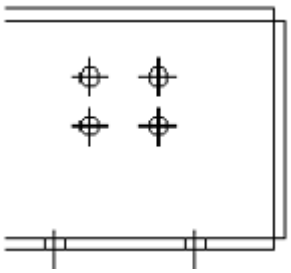
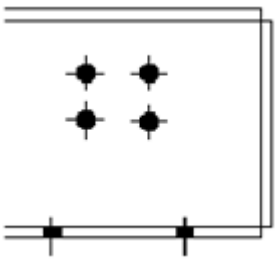
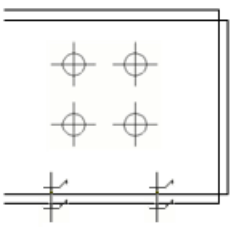
   For more information about user-defined bolt symbols, see Create customized bolt symbols (page 866).
7. On the **Appearance** tab, select the color of the bolts.
8. Single-part, assembly and cast unit drawings: Click **Save** to save the view properties. Then click **Close** to return to drawing properties.
9. General arrangement drawings: Click **OK**.
10. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

Examples of bolt representations

There are several representation options available for displaying bolts in drawings. You can select the options from the **Solid/Symbol** list.

Note that there can be a difference in bolt representation depending on whether the bolt is a workshop or site bolt, and also what kind of a symbol has been defined in the `xsteel.sym` file.

Setting	Example
solid	

Setting	Example
exact solid	
Symbol	
Symbol2	
Symbol3	Example of a workshop bolt: 
DIN symbol	

See also

Define drawing bolts (page 863)

Bolt properties in drawings (page 1010)

Create customized bolt symbols

You can create your own bolt symbols in Symbol Editor and use them in drawings. You only need to create bolt symbols if you need different bolt symbols than the ones that you can find in Tekla Structures.

1. Save the symbol file `ud_bolts.sym` in the symbol folder (usually the folder `..\Tekla Structures\<version>\environments\common\symbols\`).
2. Open Microsoft Notepad, or any text editor.
3. Create a text file consisting of lines in three columns:
   - The first column contains the bolt assembly standard.
   - The second contains the bolt diameter.
   - The third column contains the name of the symbol file and the symbol number, separated with the @ character.

     Example of file contents:

     ```
     7990 24 ud_bolts@1
     7990 25 ud_bolts@2
     ```

     Tekla Structures uses the user-defined symbol for bolts in drawings that have the standard and diameter you define in this text file.
4. Save the file with the name `bolt_symbol_table.txt`.
5. Set the name of the file as a value for the advanced option `XS_USER_DEFINED_BOLT_SYMBOL_TABLE` in **File menu** --> **Settings** --> **Advanced options** --> **Marking - bolts** as follows:

   ```
   XS_USER_DEFINED_BOLT_SYMBOL_TABLE=bolt_symbol_table.txt
   ```

   You can also enter a full path to the bolt definition file. Without the path Tekla Structures searches for the file in the model, firm, project, and system directories.
6. To use your own bolt symbol, click **Bolt** --> **Content** --> **Solid/Symbol** --> **User-defined symbol** in the drawing view properties dialog box.

See also

Define drawing bolts (page 863)

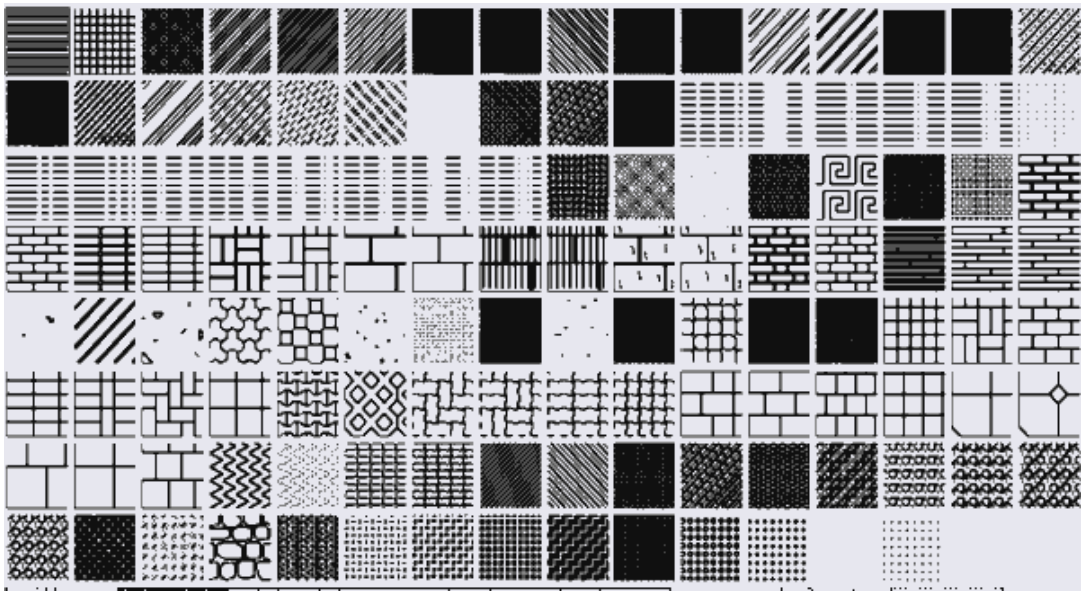
8.9 Define hatches in drawings

You can use hatching on part faces, part sections or drawing shapes, such as circles and polygons, to show different materials. You can also use hatch patterns on surface treatment.

Hatch patterns are by default located in the `hatch_types1.pat` file in the ...`\Tekla Structures\<version>\environments\<common>\inp` folder. For some environments, it is also located in `...\Tekla Structures\<version>\environments\<environment>\inp`. In these environments, the XS_INP advanced option is used to define the folder from which this file is read. For example, in the US Imperial environment the following line exists in `env_US_imperial.ini`:

```
set XS_INP=%XSDATADIR%\environments\usimp\inp\
```

The file can also be read from a firm folder defined by XS_FIRM but the location defined by `XS_INP` takes priority, so a firm folder file will not be read if a file of the same name also exists in the `XS_INP` folder.



You can also add hatch patterns to Tekla Structures:

- To add custom hatch patterns in the `hatch_types1.pat` file, see Create custom hatch patterns (page 872).
- If you have a hatch pattern file (created in a CAD software, for example), you can add it to Tekla Structures `.pat` file with a little modification, see Adding custom fill patterns.

See also

Add hatches (fills) to parts in drawings (page 868)

Create custom hatch patterns (page 872)

Define automatic surface treatment in drawings (page 881)

Hatch pattern settings (.htc) for automatic drawing hatches (page 876)

Surface treatment hatch pattern properties (surfacing.htc) (page 1014)

Colors in drawings (page 61)

Add hatches (fills) to parts in drawings

You can use hatches as fills in parts, neighbor parts, and cross sections in a drawing. You can use automatic hatches or manual hatches.

Limitations

There are some limitations in the hatch background color usage:

- Background color does not work in conjunction with hardware hatches.
- The background color does not have any effect if there is an automatic hatch available. The background color can only be changed if the automatic hatch is not defined for the material type.

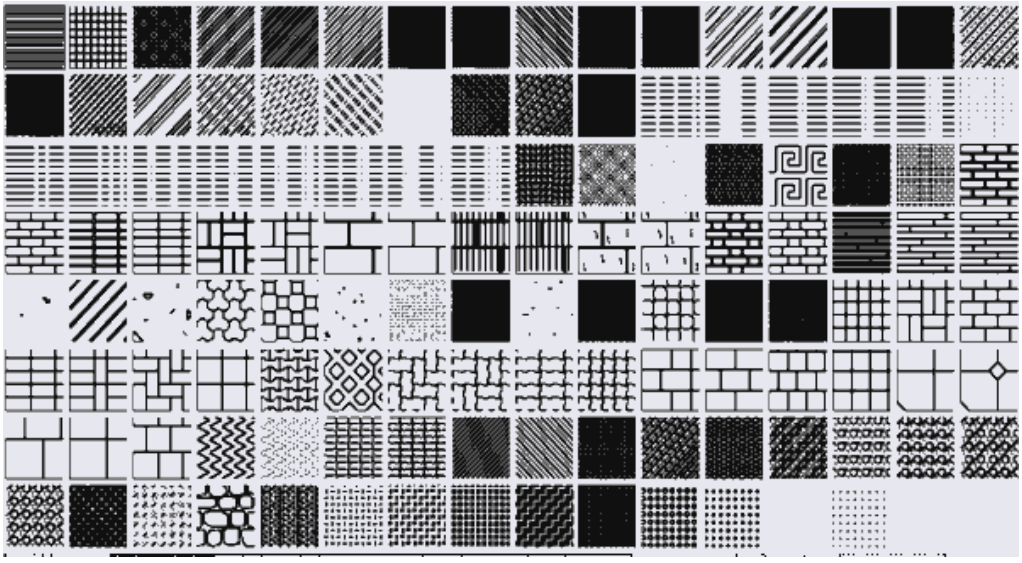
You can also use fills in sketch objects (rectangles, polygons, circles). For more information, see Draw sketch objects in drawings (page 388).

To add a fill to a part:

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Single-part, assembly and cast unit drawings: Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. Click **Part** and go to the **Fill** tab.
5. Select the hatch from the **Type** list. Select **Automatic** or a manual hatch pattern.

   To preview the hatches, click the **...** button next to the **Type** list.

   You can select the desired hatch by double-clicking it in the **Hatches** window.



If you select **Automatic**, Tekla Structures uses the hatches defined in the hatch schema file (page 876) (`.htc`). Each drawing type has its own schema file.

The names of the schema files (page 876) Tekla Structures are defined in the **Hatching** category of the **Advanced options** dialog box:

XS_DRAWING_GA_HATCH_SCHEMA

XS_DRAWING_CAST_UNIT_HATCH_SCHEMA

XS_DRAWING_SINGLE_PART_HATCH_SCHEMA

XS_DRAWING_ASSEMBLY_HATCH_SCHEMA

6. Define a color for the hatch (page 61) in the **Color** box.
7. Define a background color for the hatch in the **Background** box.

   You can define the background color only after selecting a hatch first.
8. In **Scale**, select whether to use automatic or custom scaling and rotation for hatches.

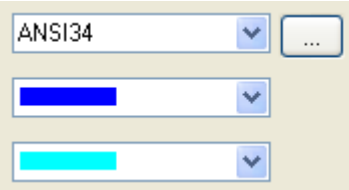
   If you select automatic scaling and rotation, Tekla Structures automatically scales the hatch to suit the profile size, and you do not need to edit each drawing manually. If you select **Custom** scaling and rotation:

   - Enter the scales in **Scaling in direction x** and **Scaling in direction y**, and select whether to **Keep ratio of x and y**.
   - Enter the angle of rotation in the **Angle** box. Angle `0.0` is for horizontal and `90.0` for vertical.
9. **Single-part, assembly and cast unit drawings:** Click **Save** to save the view properties. Then return to drawing properties by clicking **Close**.
10. **General arrangement drawings:** Click **OK** to return to drawing properties.

11. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

Example 1

In the example below, the following hatch options have been selected for cross sections:



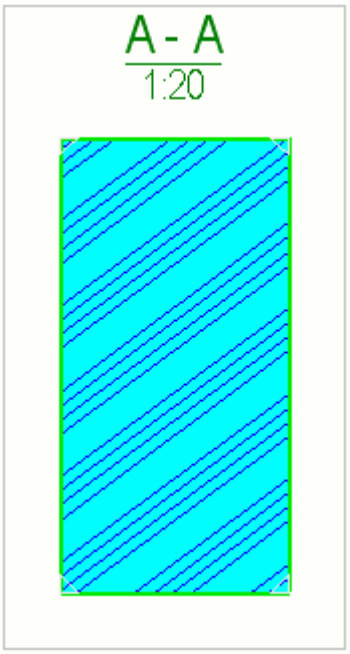
Scale: **Custom**

Scaling in direction x: 0.25

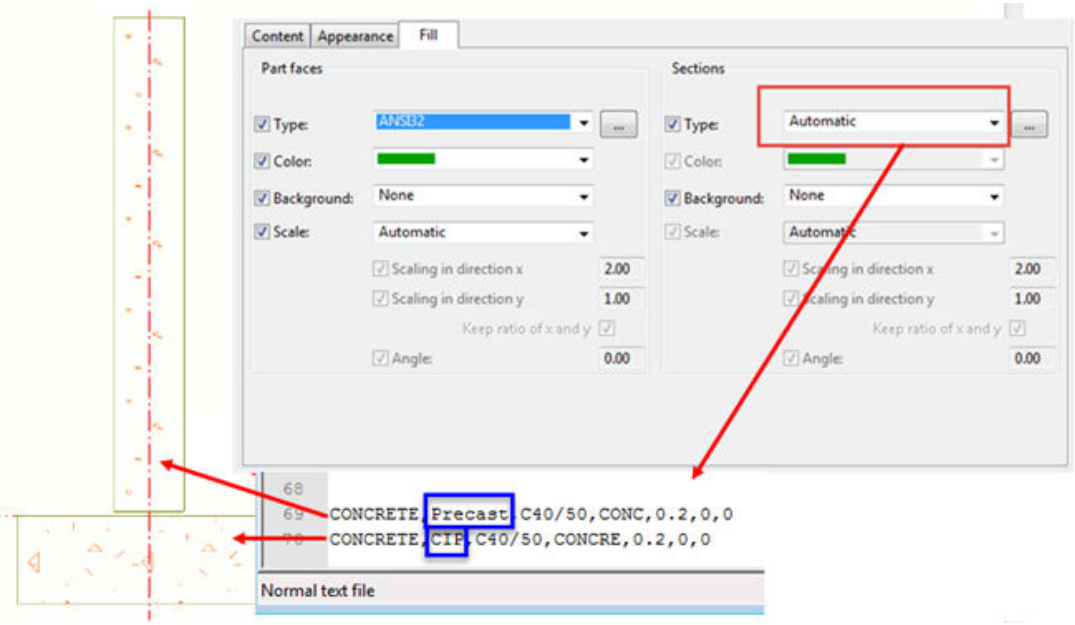
Scaling in direction y: 0.50

Keep ratio of x and y is selected.

Angle: 10.00



Cast-in-place and precast hatches use different automatic hatch depending on the cast unit type. You need to select the appropriate cast unit type in the concrete part properties.



NOTE If the hatches are not displayed in the drawing, check the session history log for the following messages:

"Solid warning: Clipped part Id: XXXX was created with second fall back and may contain overlapping volume and area."

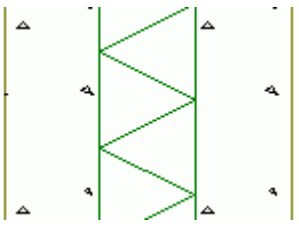
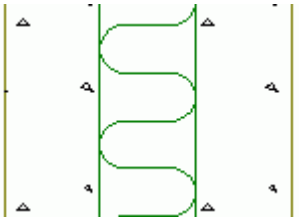
"Solid warning: Pour Id: XXX was created with second fall back and may contain overlapping volume and area."

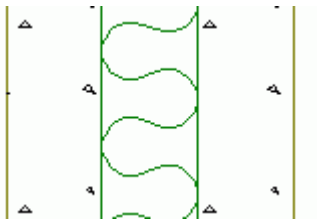
Usually it is enough that you move a part or a cut a little bit, maybe 1 mm, in any direction.

Example 2

You can use the following fill types to represent insulation:

These hatch types can be scaled and rotated.

Hatch name	Pattern
HARD_INS1	
SOFT_INS	

Hatch name	Pattern
SOFT_INS2	

Some advanced options related to hatches

XS_HATCH_SCALE_LIMIT

XS_HATCH_SPECIAL_COLOR_R

XS_HATCH_SEGMENT_BUFFER_SIZE

XS_HATCH_SPECIAL_COLOR_ACI

XS_SET_HATCH_ORIGIN_INTO_VIEW_ORIGIN

See also

Define hatches in drawings (page 866)

Part and neighbor part properties in drawings (page 1004)

Hatch pattern settings (.htc) for automatic drawing hatches (page 876)

Create custom hatch patterns

You can add hatch patterns of your own manually in the `hatch_types1.pat` file.

Add hatch patterns

1. Open the `hatch_types1.pat` file in a text editor.

   The file is located by default in the `...\Tekla Structures\<version>\environments\<common>\inp` folder. For some environments, it may also be located in the `\inp` folder under another environment folder. The file can also be read from the firm folder. For details on file location, see Define hatches in drawings (page 866).

   Use the following syntax for your hatch pattern:

   ```
   *pattern-name[, description]

   angle, X origin, Y origin, X offset, Y offset, Dash1, Dash2,,,DashN
   ```

2. Type the hatch pattern header line:

   A pattern definition starts with a header row. This starts with an asteriks (*) immediately followed by a name of no more than 31 characters (no

spaces allowed). The name can be followed by an optional description for the pattern.

Syntax: `*pattern-name[, description]`

For example: `*CHECKERED,Checkered plate`

3. Type the rows for the pattern:

   The rows following the header each define evenly spaced parallel lines used to create the pattern. Only one row is required but more complex patterns may require several rows.

   ```
   angle, X origin, Y origin, X offset, Y offset, Dash1,
   Dash2,,,DashN
   ```

   Syntax:

   ```
   0, 0.0, 0.0, 25.4, 25.4, 25.4, -25.4
   ```

   ```
   90, 12.7, 12.7, 25.4, 25.4, 25.4, -25.4
   ```

   See the example below for what this pattern looks like.

4. Save the `.pat` file.

   A pattern with the defined header will appear in Tekla Structures in the fill type lists in drawing object properties.

   When you add the hatch in a drawing object, it is inserted at a particular point (the pattern origin), which will be a point on the boundary of the fill area. The area inside the boundary is then filled by infinitely repeating the lines defined in each row of the pattern definition along the local X and Y axes of the line. Each row in the definition can define a line with different local X and Y axes and different starting points relative to the pattern origin. In this way, a pattern can be developed consisting of any number of lines drawn in any direction.

Example of a custom hatch in a drawing polygon

This example describes how to apply the CHECKERED pattern in a drawing polygon.

The pattern is defined as follows:

```
*CHECKERED,Checkered plate
```

```
0, 0.0, 0.0, 25.4, 25.4, 25.4, -25.4
```

```
90, 12.7, 12.7, 25.4, 25.4, 25.4, -25.4
```

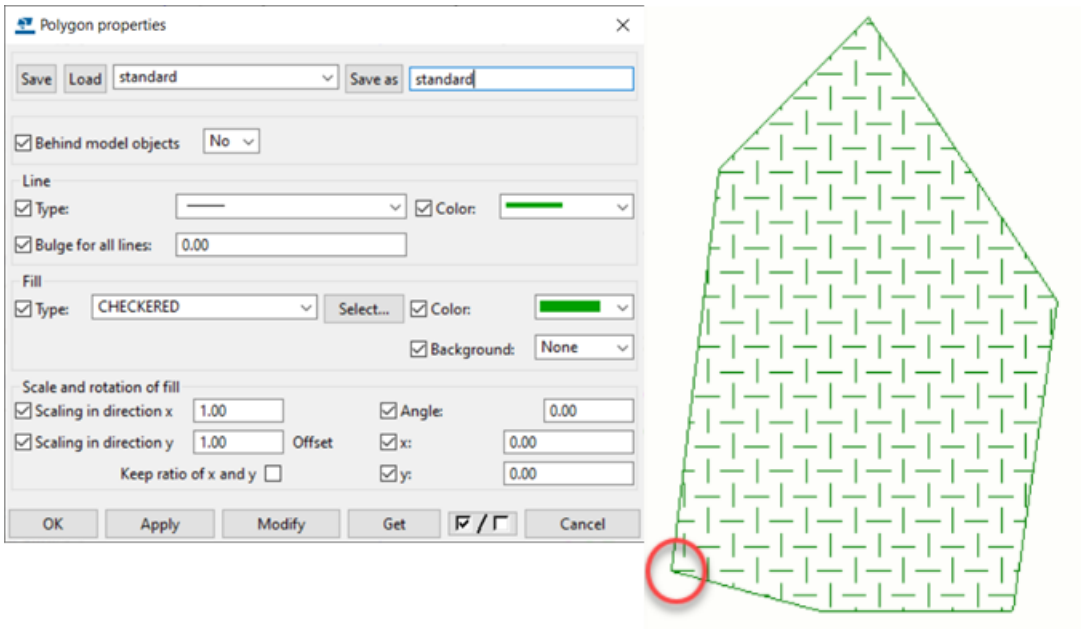
1. In Tekla Structures, create a polygon in a drawing.

   The first point of the polygon will be the origin of the hatch pattern.

2. Double-click the polygon to open its properties.
3. Select CHECKERED as the fill type.
4. Set the scale in X and Y to 1 and the angle to 0.

5. Click **Modify** to apply the pattern.

The vertex surrounded by a red circle is the first point of the polygon and hence the pattern origin.

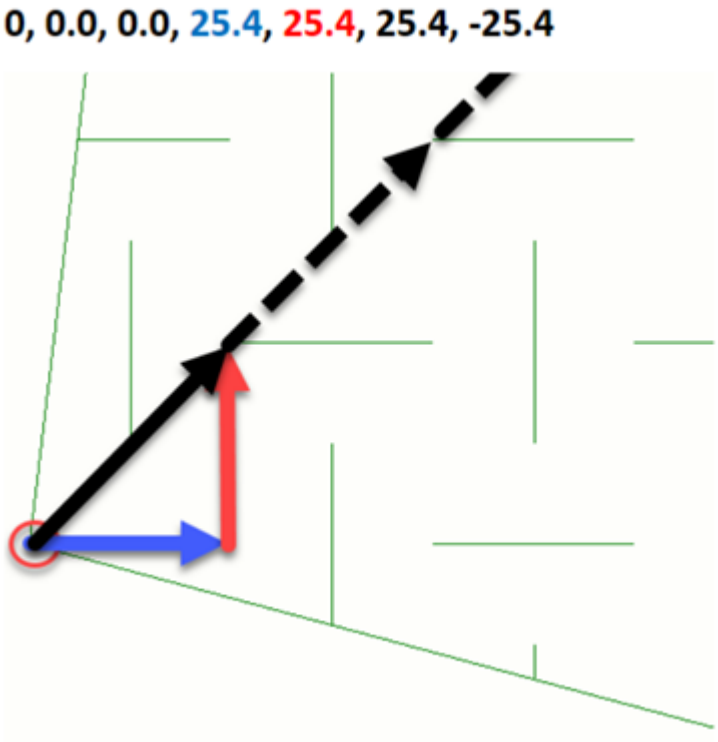


- Consider the first line of the pattern definition:

  `0, 0.0, 0.0, 25.4, 25.4, 25.4, -25.4`

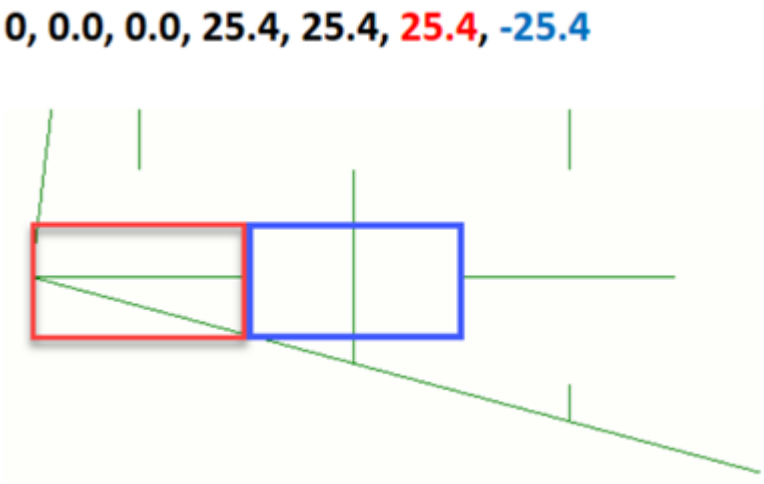
  - The 2nd and 3rd values define the X origin and Y origin. In this case, both are 0, so this line starts at the pattern origin. The angle is 0, so the line is oriented along its X axis. Since we have set the angle of the fill to 0 in polygon properties, this also corresponds to the global X axis of the drawing.
  - The 4th and 5th values (X offset and Y offset respectively) determine where copies of the line are placed. The first copy of the line will start 25.4 units in X and 25.4 units in Y from the origin. This is illustrated by the image below, where the black arrows represent the displacement

from one line to the next, and the blue and red arrows represent the X and Y components of this displacement.



- The 6th and 7th values are optional. If a line definition contains only 5 values, the line is a continuous line with no gaps. Dashed lines are created by adding pairs of positive and negative values. The positive values represent the length of the displayed sections of the line, and the negative values represent the length of the hidden sections. In this case, the line consists of a line segment of 25.4 units followed by a space of 25.4 units.

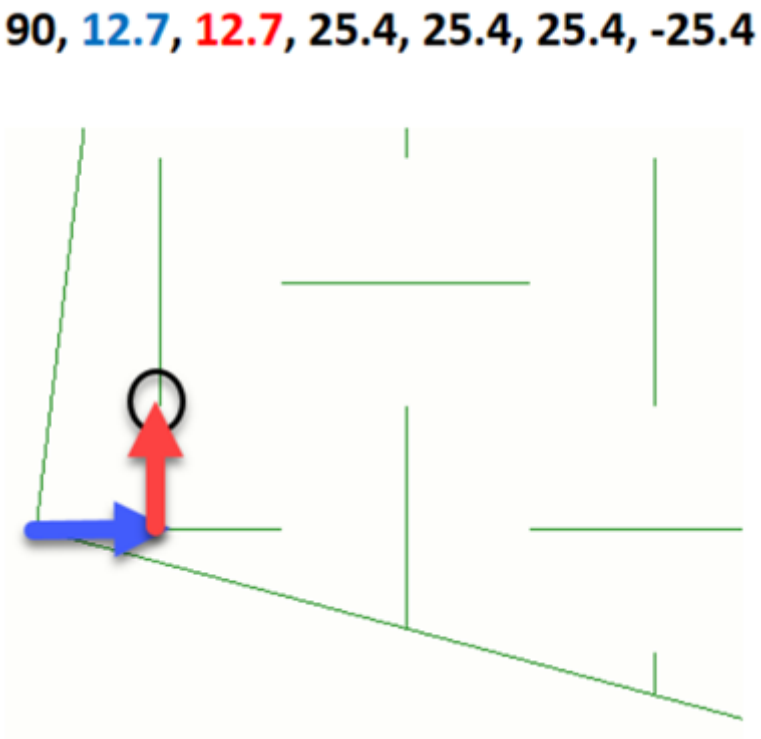
  This is illustrated in the following image:



- The second line is rotated by 90 degrees, so it is running away from the pattern origin along the global Y axis of the drawing.

  ```
  90, 12.7, 12.7, 25.4, 25.4, 25.4, -25.4
  ```

  Its X origin is 12.7 and its Y origin is 12.7. This is illustrated in the following image:



Hatch pattern settings (.htc) for automatic drawing hatches

The hatch patterns for material types and names in drawings are defined in a hatch schema file `.htc`. Different drawing types (general arrangement, assembly, single-part, and cast unit drawings) have their own schema files.

The hatch patterns defined in schema files are used when you set **Type** to **Automatic** on the **Fill** page in the part or shape properties.

Hatch schema file name and location

The name of the schema file Tekla Structures uses for each drawing type is defined in the **Hatching** category of the **Advanced options** dialog box:

- `XS_DRAWING_GA_HATCH_SCHEMA=general.htc`
- `XS_DRAWING_ASSEMBLY_HATCH_SCHEMA=assembly.htc`
- `XS_DRAWING_SINGLE_PART_HATCH_SCHEMA=single.htc`
- `XS_DRAWING_CAST_UNIT_HATCH_SCHEMA=cast_unit.htc`

The default schema files are located in the `..\Tekla Structures\<version>\environments\common\system` folder. The environment-specific schema files are located under the environments. The exact file location may vary depending on the folder structure of you environment files.

If you modify the hatch files and you want to used the modified settings file in the current model only, save them to the model folder. If you want to use the settings in all models, save the modified settings file in the firm folder.

Schema file syntax

```
Material type, Cast unit type, Material name, Hatch name,
Scale, Color, Automatic scaling and rotation
```

Examples:

`CONCRETE,,hardware_SOLID,,120` (gray shade)

`CONCRETE,Precast,C25/30,CONC,0.3,1` ("traditional" concrete Precast)

`CONCRETE,CIP,C25/30,CONCRE,0.3,1` ("traditional" concrete CIP)

`CONCRETE,,CROSS,2,1` (small crosses)

`MISCELLANEOUS,Insulation,HARD_INS1,1,,1` (straight lines)

`MISCELLANEOUS,Insulation,SOFT_INS,1,,1` (rounded lines)

`MISCELLANEOUS,Insulation,SOFT_INS2,1,,1` (completely round)

Option	Description
`Material type`	`STEEL`, `CONCRETE`, `TIMBER`, `MISCELLANEOUS` (case sensitive)
`Cast unit type`	`Precast` or `CIP`. You also need to define the cast unit type for concrete parts in part properties.
`Material name`	The material name defined for the part in part properties in the model. The character limit for the material name is 32 characters.
`Hatch name`	To check the hatch pattern names and related hatch patterns go to the **Fill** tab in **Part** properties, select a hatch pattern from the **Type** list and click the **...** button next to the list. Then click a pattern to see its name in the **Type** list. The selected hatch pattern is marked with a red frame. Hatch names are case sensitive. If you do not want to use any hatch pattern for a material, leave the hatch name field blank in the schema file.

Option	Description
`Color`	BLACK: 0 (default) WHITE: 1 RED: 2 GREEN: 3 BLUE: 4 CYAN: 5 YELLOW: 6 MAGENTA: 7 Special: 120 (use this color for gray shade) GRAYSCALE 1: 130 GRAYSCALE 2: 131 GRAYSCALE 3: 132 GRAYSCALE 4: 133 The hatch color defines the line width for the printer. If you do not define a color for a hatch in the schema file, Tekla Structures uses the default color black (0). To use the special color that is printed as color or grayscale, depending on the selected printer settings, set the color number in the `.htc` file to 120. To adjust the gray shade for Special color 120, use the following advanced options: XS_HATCH_SPECIAL_COLOR_R XS_HATCH_SPECIAL_COLOR_G XS_HATCH_SPECIAL_COLOR_B
`Scale`	Scale is a numeric value that Tekla Structures uses to scale hatches.
`Automatic scaling and rotation`	Automatic scaling and rotation defines whether you want to use automatic scaling and rotation. TRUE: 1

Option	Description
	FALSE: 0 (default) Automatic scaling and rotation requires additional processing, so use it only when necessary.

Example of the `cast_unit.htc` file:

```
/************************************/
/* Tekla Structures hatch schema file */
/************************************/

/
*****************************************************************************
**************************

    Purpose
    -------
    This file is used to define the hatch patterns for different
materials for
    Tekla Structures automatic hatching functionality. These settings
will be used
    when the hatching option is set to "Automatic" in the drawing.

    Syntax
    -------
    Material Type, Cast unit type, Material Name, Hatch name, Scale,
Color, Automatic scaling and rotation

    Available options
    -----------------
    Material Type:
    STEEL
    CONCRETE
    TIMBER
    MISCELLANEOUS

    Cast unit type:
    CIP
    Precast

    Colors:
    BLACK          0 (default)
    WHITE          1
    RED            2
    GREEN          3
    BLUE           4
    CYAN           5
    YELLOW         6
    MAGENTA        7
    Special           120 (use this color for gray shade)
    GRAYSCALE 1    130
    GRAYSCALE 2    131
    GRAYSCALE 3    132
    GRAYSCALE 4    133

    Automatic scaling and rotation:
    TRUE       1
    FALSE      0 (default)

    Instructions for use
    --------------------
    - material and hatch names are case sensitive
```

```
    - automatic scaling and rotation option requires additional
processing, so use it only when necessary
    - to adjust the gray shade for Special color 120, use the variables
      XS_HATCH_SPECIAL_COLOR_R
      XS_HATCH_SPECIAL_COLOR_G
      XS_HATCH_SPECIAL_COLOR_B

    Examples
    --------
    CONCRETE,,hardware_SOLID,,120            (gray shade)
    CONCRETE,Precast,C25/30,CONC,0.3,1          ("traditional" concrete
Precast)
    CONCRETE,CIP,C25/30,CONCRE,0.3,1            ("traditional" concrete
CIP)
    CONCRETE,,CROSS,2,1                    (small crosses)
    MISCELLANEOUS,Insulation,HARD_INS1,1,,1        (straight lines)
    MISCELLANEOUS,Insulation,SOFT_INS,1,,1         (rounded lines)
    MISCELLANEOUS,Insulation,SOFT_INS2,1,,1        (completely round)

**************************************************************************
**************************/
```

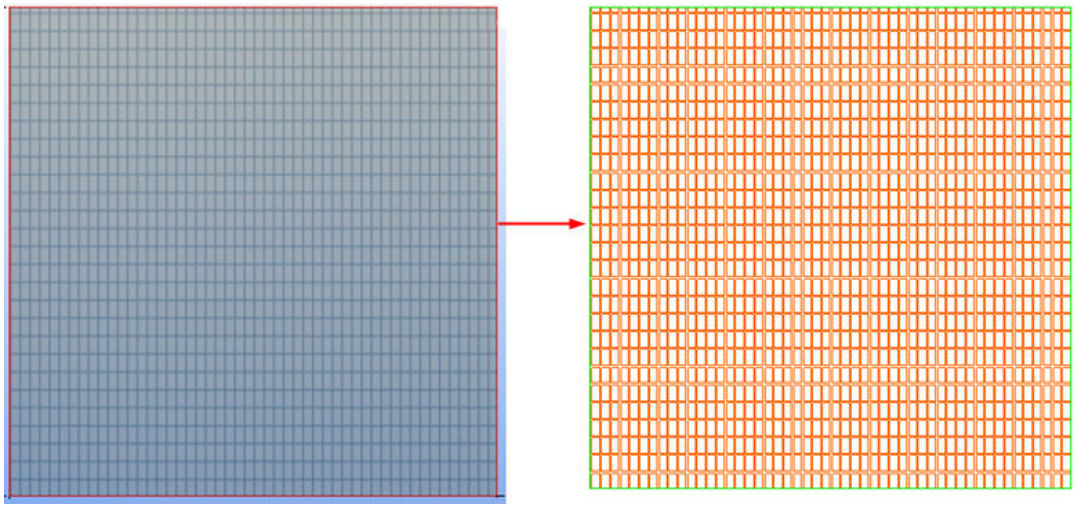
See also

Add hatches (fills) to parts in drawings (page 868)

Part and neighbor part properties in drawings (page 1004)

8.10 Define drawing surface treatment

You can add various types of surface treatment to steel and concrete parts in the Tekla Structures model and show them in drawings.



The properties of the hatch patterns to use for each surface treatment type are defined in the `surfacing.htc` file. Also the `product_finishes.dat` code file is needed listing all surface treatment codes that are used in drawings and reports, for example, TS1 for Tile Surface 1.

See also

Define automatic surface treatment in drawings (page 881)

Surface treatment properties in drawings (page 1012)

Surface treatment hatch pattern properties (surfacing.htc) (page 1014)

Define automatic surface treatment in drawings

You can define what is shown in a surface treatment and how the surface treatment is shown in drawings.

The hatch pattern Tekla Structures adds on the surface treatment depends on the type of the surface treatment that you selected in surface treatment properties in the model, and on the hatch properties defined in the surface treatment hatch pattern properties file `surfacing.htc`.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Single-part, assembly and cast unit drawings: Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. Click **Surface treatment**.
5. On the **Content** tab, select the surface treatment representation, and whether to show the surface treatment pattern, hidden lines, and own hidden lines.
6. On the **Appearance** tab, select the color and type of visible and hidden lines.
7. General arrangement drawings: Click **OK**.
8. Click **Surface treatment mark**, add the elements that you want to include in the mark and adjust the appearance of the mark as required.
9. Single-part, assembly and cast unit drawings: Click **Save** to save the view properties. Then click **Close** to return to drawing properties.
10. General arrangement drawings: Click **OK**.
11. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

See also

Define automatic surface treatment in drawings (page 881)

Surface treatment properties in drawings (page 1012)

Surface treatment hatch pattern properties (surfacing.htc) (page 1014)

Define hatches in drawings (page 866)

8.11 Define drawing welds

You can define automatic weld settings to be used in a drawing before you create a drawing. You can modify the settings in an open drawing after you have created the drawing.

Before creating a single-part drawing or an assembly drawing, you can define the representation, visibility and appearance properties (page 1036) for the model welds. For general arrangement drawings, representation settings can only be defined on view and object level, not on drawing level.

See also

Define automatic model weld properties in drawings (page 882)

Show welds in drawings (page 489)

Define automatic model weld properties in drawings

You can define what the model welds look like in your drawings before you create the drawing.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Single-part and assembly drawings: Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. Click **Weld**.
5. On the **Content** tab, define the visibility settings:
   - In **Welds** and **Welds in sub-assemblies**, select **Not visible**, **Site weld visible**, **Workshop weld visible**, or **Both visible**.
   - In **Weld size limit**, enter a weld size to filter out welds of that size and smaller.

     This is useful when you only want to show non-typical welds in a drawing.

     To set whether the weld size is an exact or minimum value, use the advanced option `XS_WELD_FILTER_TYPE`.

To filter out a standard weld type, use the advanced option `XS_OMITTED_WELD_TYPE`.

Note that you can also set the **Weld size limit** for all views in the drawing at one go at the bottom of the **View creation** page in assembly and single-part drawing properties (page 906).

- In **Representation**, select **Outline** or **Path**, and enable **Hidden lines** and **Own hidden lines**, if required.

6. On the **Appearance** tab, modify the color and line as required.
7. Single-part, assembly and cast unit drawings: Click **Save** to save the view properties. Then click **Close** to return to drawing properties.
8. General arrangement drawings: Click **OK**.
9. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

See also

Model weld properties in drawings (page 1036)

Model weld mark properties in drawings (page 996)

8.12 Define drawing pours

You can define automatic pour object settings to be used in a drawing before you create a drawing. You can modify the pour settings in an open drawing after you have created the drawing.

You can select whether to show pours and pour breaks in drawings, define the desired pour object and pour break content and appearance. For pour objects, you can also define the desired fill. Additionally, you can add automatic pour object marks in your drawings.

You can change these settings also in an open drawing.

For more information, see Show pour objects, pour marks and pour breaks in drawings (page 883).

See also

Show pours in drawings (page 485)

Pour object and pour break properties in drawings (page 1029)

Add hatches (fills) to parts in drawings (page 868)

Show pour objects, pour marks and pour breaks in drawings

You can show pour objects and pour breaks in general arrangement drawings. You can also add automatic pour object marks.

Ensure that you have enabled pour management (set the advanced option `XS_ENABLE_POUR_MANAGEMENT` to `TRUE`.)

To show pour objects and pour breaks automatically, modify their appearance and add pour object marks:

1. On the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing** .
2. Load drawing properties that are as close to the ones you need as possible.
3. In the properties dialog box, click the **View...** button and set **Show pours in drawing** to **Yes**, and click **OK**.
4. Click the **Pour object...** button to modify the properties:
   - **Content** tab: Select whether you want to see the hidden lines and own hidden lines, and the edge chamfers by clicking **on** or **off**.
   - **Appearance** tab: Set the color and type for visible lines and hidden lines.
   - **Fill** tab: Select the fill for the pour object face and/or pour object section face.
5. Click **OK**.
6. Click the **Pour object mark...** button, select the contents and appearance of the mark, and click **OK**.
7. Click the **Pour breaks...** button and set the **Visibility** to **Visible** and click **OK**.

   You can also select whether you want to show pour break hidden lines on the **Visibility** tab. On the **Appearance** tab you can change the color and type of the visible and hidden lines in pour breaks.
8. Modify the other properties as required. For example, click **Reinforcement...** and set the **Visibility of all reinforcing bars** to **Visible** to show the reinforcement in the pour drawing.
9. Save the changed properties and click **OK**.

   Now you can create the general arrangement drawing using the modified properties. The pour objects, pour object marks and pour breaks are shown accordingly.

   You can also open the pour drawing and modify the properties further on drawing, view and object level.

See also

Show pours in drawings (page 485)

Pour object and pour break properties in drawings (page 1029)

Add hatches (fills) to parts in drawings (page 868)

8.13 Define drawing reinforcement and meshes

There are numerous ways of showing reinforcing bars and meshes in drawings. You can define automatic reinforcement and mesh settings before you create the drawing, and adjust the settings in an open drawing. In addition to modifying the properties available in the properties dialog box, you can also modify the bending schedules, rounding of bar dimensions, symbols in use, and reinforcement appearance in the `rebar_config.inp` settings file.

Click the links below to find out more:

Define automatic reinforcement and reinforcement mesh properties (page 885)

Automatic grouping of rebar sets (page 889)

See also

Reinforcement object properties in drawings (page 1016)

Reinforcement settings for drawings (rebar_config.inp) (page 1023)

Part and neighbor part properties in drawings (page 1004)

Add dimensions to reinforcement manually (page 219)

Add reinforcement marks manually in drawings (page 303)

Add pull-out pictures in reinforcement marks (page 848)

Define automatic reinforcement and reinforcement mesh properties

You can define what is shown in reinforcing bars and meshes and how they are shown in cast unit and general arrangement drawings. You can also do this for neighbor reinforcement.

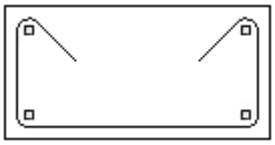
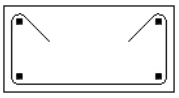
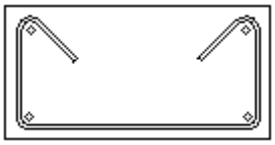
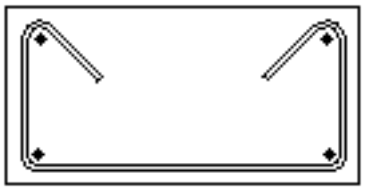
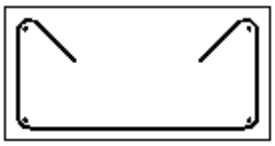
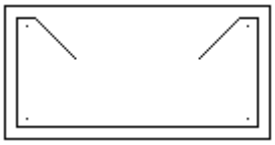
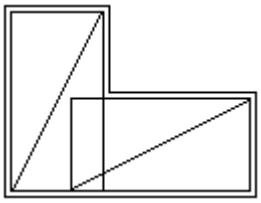
1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Single-part, assembly and cast unit drawings: Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.

4. Click **Reinforcement** (or **Neighbor reinforcement**).
5. On the **Bar content** tab, set the bar representation, bar end symbols, and the visibility of bars and lines.
6. On the **Bar appearance** tab, select the color and type of the visible lines and hidden lines.
7. On the **Mesh content** tab, set the mesh representation, mesh symbols, and the visibility of meshes, wires, bars and lines.
8. On the **Mesh appearance** tab, select the color and type of the visible lines and hidden lines.
9. On the **Rebar assembly** tab, set the visibility of the rebar assemblies, rebar assembly representation, the color and type of the visible lines, and select the mesh symbol settings.
10. General arrangement drawings: Click **OK**.
11. Click **Reinforcement mark** (or **Neighbor reinforcement mark**), add the elements that you want to have in the mark and adjust the mark appearance.
12. Single-part, assembly and cast unit drawings: Click **Save** to save the view properties. Then click **Close** to return to drawing properties.
13. General arrangement drawings: Click **OK**.
14. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

TIP
- The representation of reinforcing bars that point away from the drawing can be changed. You can customize reinforcing bar bent symbols (cross, circle, filled circle) by editing the symbol file `bent.sym`, which in the default environment is located in the folder `..\ProgramData\Trimble\Tekla Structures\<version>\\environments\common\symbols`. Or, you can create a new symbol file in the same folder, for example, `my_new_symbols.sym`, and take it into use in the rebar_config.inp (page 1023) by entering the following string in the file: `BentSymbolFile=my_new_symbols.sym`
- If you wish to show embeds while hiding reinforcement, use the XS_HIDDEN_LINES_UNHIDE_EMBEDDED advanced option.

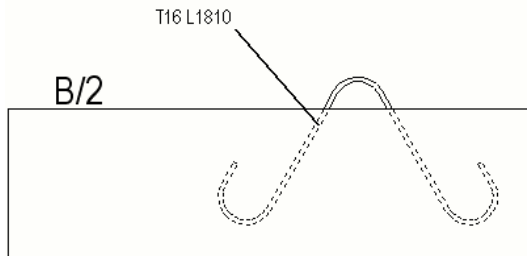
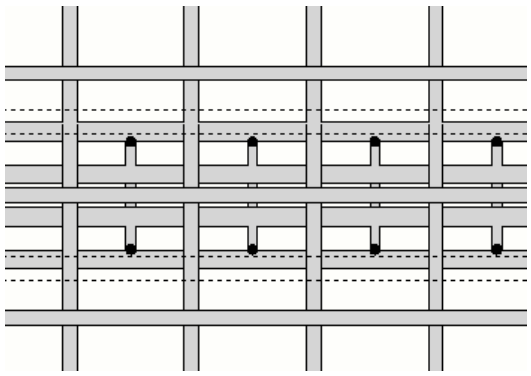
Example: Reinforcement representations

Here are examples of how the reinforcement will look with the different representation options selected on the **Bar content** tab in **Reinforcement** or **Neighbor reinforcement** properties.

Setting	Example
single line	
single line with filled ends	
double lines	
double lines with filled ends	
filled line	
stick	
outline	

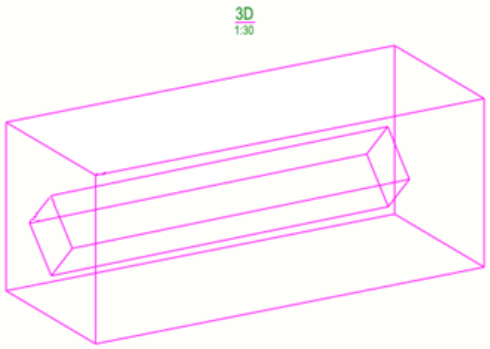
Example: Hide reinforcing bar lines in drawings

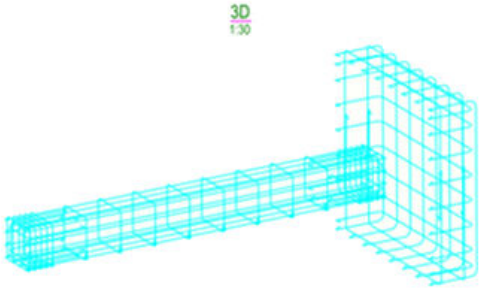
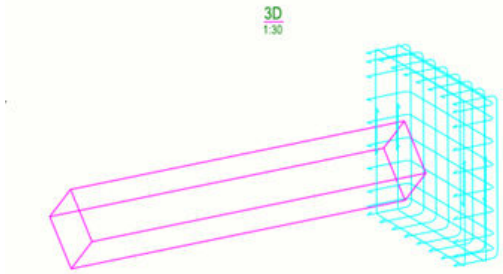
You can hide reinforcement bar lines behind other reinforcing bars and behind parts in cast unit drawings. Here are some examples of what reinforcing bars look like with different settings selected on the **Bar content** tab in **Reinforcement** or **Neighbor reinforcement** properties.

Setting	Example
Hide lines behind parts is selected. The reinforcing bar lines are hidden behind other parts.	
Hide lines behind other rebars is selected. The reinforcing bar lines are hidden behind other reinforcing bar lines.	

Example: Rebar assembly representation

Here are some examples of what rebar assemblies look like with different settings selected on the **Rebar assembly** tab in **Reinforcement** or **Neighbor reinforcement** properties.

Outline is selected. All sub-assemblies of the rebar assembly are shown as outline boxes.	: 

As individual rebar objects is selected. All rebar objects in the rebar assembly and sub-assemblies are visible.	
Sub-assemblies as outline, highest level as individual rebar objects is selected. All sub-assemblies are shown as outline boxes and the highest level of the rebar assembly is shown as individual rebar objects.	

See also

Reinforcement object properties in drawings (page 1016)

Reinforcement settings for drawings (rebar_config.inp) (page 1023)

Part and neighbor part properties in drawings (page 1004)

Define drawing reinforcement and meshes (page 885)

Automatic grouping of rebar sets for drawings

The reinforcing bars created by using the rebar set commands are automatically grouped for drawing marking and dimensioning purposes. The automatic grouping works both for simple, non-tapered groups as well as for tapered groups.

Note that the automatic grouping of rebar set bars takes cast units into consideration. If a cast unit consists of multiple parts, the bars are grouped within the cast unit instead of the parts.

Preconditions for rebar group creation

Simple, non-tapered groups are created when:

- Bars are created by the same rebar set

- Bars are side by side, placed radially, or parallel and distributed along a curve
- Bars have identical properties, including length
- Bars have the same position number

Linearly tapered bars are grouped when:

- Bars are created by the same rebar set
- Bars have identical properties, except length
- Length of bars increases linearly
- Distance between each bar end and the tapering line is less than or equal to the value set by XS_REBARSET_TAPERED_LINEAR_GROUPING_TOLERANCE for the model, or by the **Tapered linear tolerance** user-defined attribute for a rebar set or property modifier.
- There are minimum of 3 bars in a group

Bars that are tapered along a curve are grouped when:

- Bars are created by the same rebar set
- Bars have identical properties, except length
- Length of bars increases by following the curve
- Distance between each bar end and the curve is less than or equal to the value set by XS_REBARSET_TAPERED_CURVED_GROUPING_TOLERANCE for the model, or by the **Tapered curved tolerance** user-defined attribute for a rebar set or property modifier.

Spacing of bars is taken into account in grouping if the advanced option XS_REBARSET_ENABLE_BAR_GROUPING_WHEN_SPACING_DIFFERS is set to `FALSE`. Then only bars with same spacing are grouped, and each spacing zone of a rebar set automatically creates a separate group. If the advanced option is set to `TRUE`, spacing is ignored, and groups can contain bars from more than one adjacent spacing zones.

Work with rebar set groups in drawings

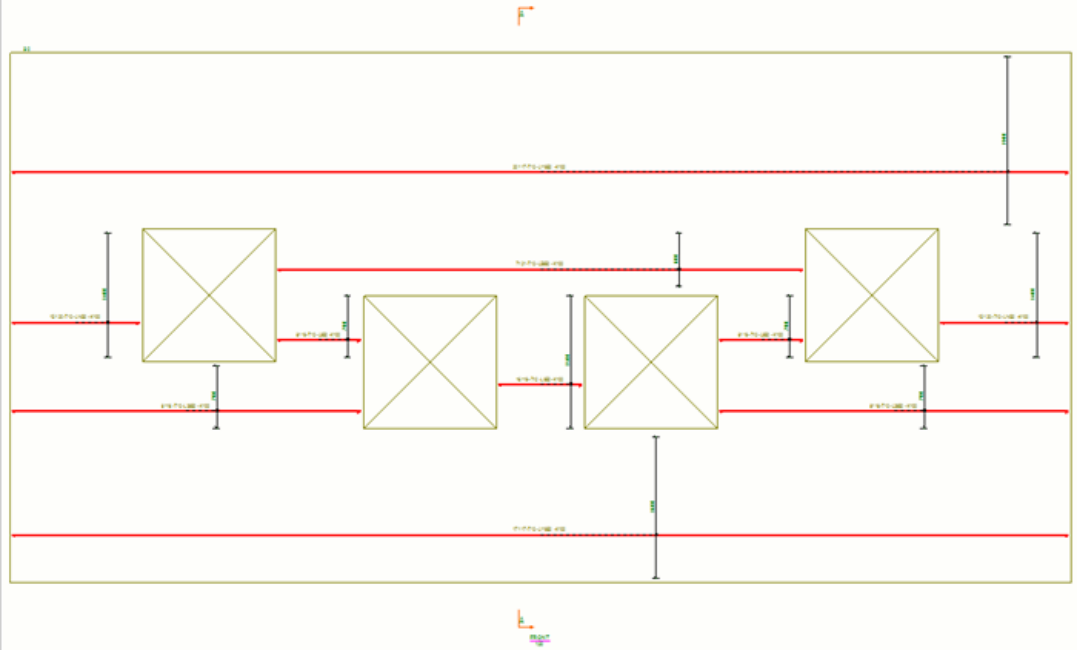
Automatically created groups behave in the drawing similarly as the rebar groups:

- There is only one mark for the rebar group, and the reinforcement mark content comes from the content setting **Group** in **Reinforcement mark** properties.

  Note that if XS_REBARSET_USE_GROUP_NUMBER_FOR_BARS_IN_TAPERED_GROUPS is set to `FALSE`, bars in tapered bar groups within rebar sets do not have a common position number (GROUP_POS), so position information is not included in reinforcement marks.

- You can adjust the rebar visibility (page 1016) in **Reinforcement** properties. For example, you can select to show **bar in the middle of group**.
- You can use the following rebar-group-specific interactive dimensioning commands (page 219) for rebar groups:
  - **Add mark** --> **Dimension mark** in contex menu.
  - **Add rebar dimension mark** in **Quick Launch**.
- Associative notes (page 329) can be created.

Below is an example of an automatically created rebar group in a drawing. The drawing uses rebar group visibility option **bar in the middle of group**, and reinforcement mark option **One leader line to group**, and the dimensions have been created with the command **Add mark** --> **Dimension mark** .



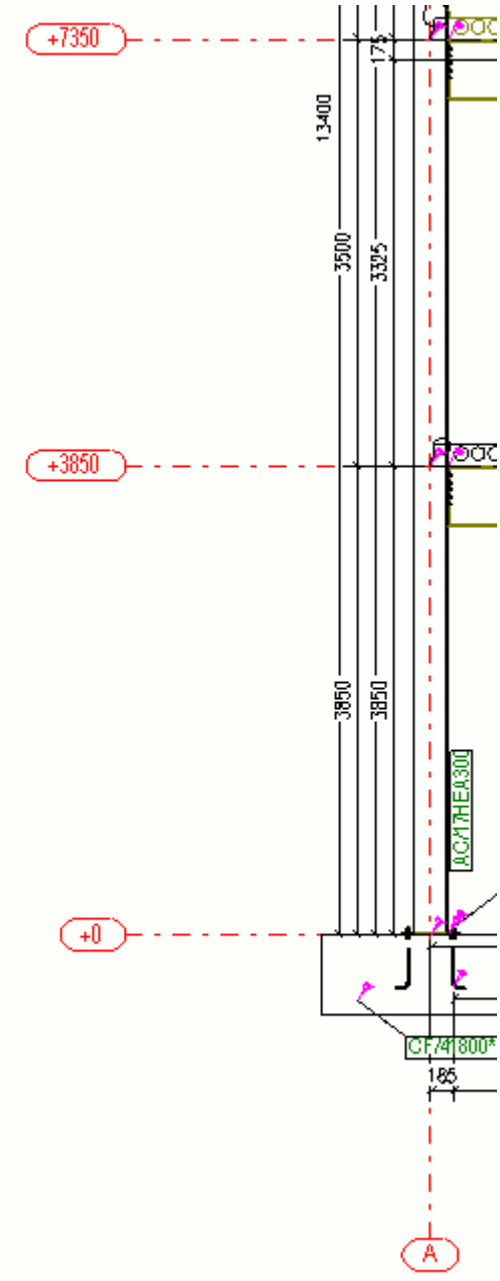
TIP If you want to fine-tune the automatic grouping, you can manually group and ungroup rebar set bars by using property modifiers.

Create a property modifier for the bars that you want to group, and in the **Rebar property modifier** properties, set **Grouping** to **Manual**.

If you need to ungroup automatically or manually grouped bars, create a property modifier with **Grouping** set to **No grouping**.

8.14 Define drawing grids

You can show grids in all types of drawings. You can change the appearance and visibility of the labels, label frames, and grid lines.



See also

Show grids in drawings (page 514)

Define automatic grid properties (page 892)

Drawing grid and grid line properties (page 1041)

Define automatic grid properties

You can modify drawing grid properties of single-part, assembly and cast unit drawings individually for each view. In general arrangement drawings, you can modify the grid properties on both view and drawing level. You can also modify individual grid properties of in an open drawing.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Single-part, assembly and cast unit drawings: Click **View creation** in the options tree on the left, select the view and the properties that you want to change, and click **View properties**.
4. Click **Grid**.
5. Modify the grid properties as required.
6. Single-part, assembly and cast unit drawings: Click **Save** to save the view properties and click **Close**.

   General arrangement drawings: Click **OK**.
7. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

TIP The advanced options XS_DRAWING_GRID_LABEL_FRAME_FIXED_WIDTH and XS_DRAWING_GRID_LABEL_FRAME_LINE_WIDTH_FACTOR allow you to adjust the grid labels further.

See also

Drawing grid and grid line properties (page 1041)

8.15 Units and decimals in drawings, reports and templates

Unit settings used in drawing objects, and reports and templates created in Template Editor are taken from different places. These settings include, for example, the used unit and the number of decimals.

In drawings, reports and templates, the units and decimals are defined in the following way:

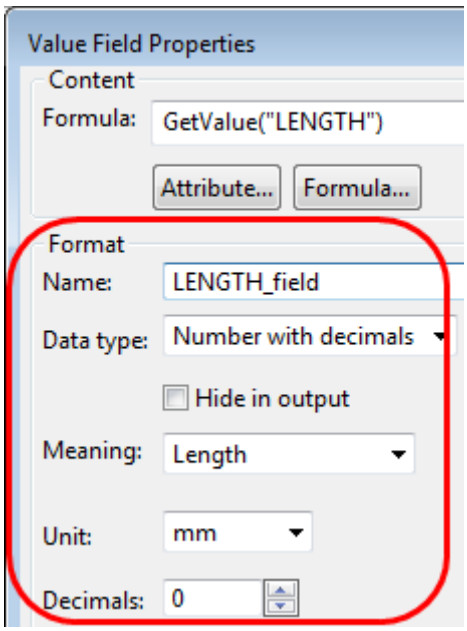
- **Unit settings for marks:** Tekla Structures uses the `contentattributes_global.lst` attributes file for setting the default unit and decimal settings for various mark elements. You can use

contentattributes_userdefined.lst when you want to configure settings of your own.

```
// Name     Datatype     Justify  Cacheable  Length  Decimals  Unit type  Unit     Precision
// XXXXX    FLOAT        RIGHT    TRUE       8       2         Length     ft-frac  1/8
// ------------------------------------------------------------------------------------------
DIAMETER    FLOAT        RIGHT    TRUE       5       0         Length     mm
```

- **Unit settings for drawing objects:** Default unit settings for drawing objects (other than marks) either come from the standard drawing properties files (standard.*) or are hardcoded in Tekla Structures.
- **Unit settings for dimensions and dimension tags:** Default unit settings for dimensions and dimension tags are defined in the **Options** dialog box on the **Drawing dimensions** page.
- **Level attributes in marks**, such as TOP_LEVEL, and BOTTOM_LEVEL take the dimension format from the MarkDimensionFormat.dim file. For more information about level attributes in part marks, see Add level attributes in automatic part marks (page 836).
- **Unit settings for reports/templates created with Template Editor:** Unit settings for reports and templates created in Template Editor can be defined in the **Value Field Properties** dialog box **Format** area or in format functions. The default unit and decimal settings are taken from the contentattributes_global.lst attributes file.

  Format area:



  Formula containing a format function:

```
if GetValue("ADVANCED_OPTION.XS_IMPERIAL")=="TRUE" then
":  "+format(double(GetValue("TOP_LEVEL")*1000),"Length","ft-inch", 1/16)
else
":  "+GetValue("TOP_LEVEL")
endif
```

8.16 User-defined attributes in drawings

Many Tekla Structures dialog boxes contain user-defined attributes for various objects. When you define a new user-defined attribute, make the definition of the user-defined attribute unique. This is because a user-defined attribute cannot have different definitions for different object types. In drawings, the user-defined attributes can be used in templates, **Document manager**, and marks, for example.

The user-defined attributes in the drawing are displayed when you click **User-defined attributes** in drawing properties.

When you define new user-defined attributes

When you define new user-defined (page 896) attributes, you need to create your own file in the company, project or firm folder. After adding your own user-defined attributes, you need to use the **Diagnose and change attribute definitions** tool to update the definitions in the model. The `object.inp` files are merged so that if there are user-defined attributes in any of the files, they are displayed in the user interface. Tekla Structures merges the files in a way that eliminates duplicate attributes. If Tekla Structures encounters the same attribute name in different `objects.inp` files, the attribute from the first read `objects.inp` file will be used.

Tekla Structures reads the `objects.inp` files from the following folders in the following order:

1. model folder
2. company folder
3. project folder
4. firm folder
5. system folder
6. inp folder

Show user-defined attributes in Template Editor

In order to show the new user-defined attribute in Template Editor, you need to add the user-defined attribute in a customized `contentattributes_userdefined.lst` file and include the name of the customized file in the `contentattributes.lst` file.

Make a copy of these modified files as the Tekla Structures installation always overwrites these files.

See also

Add attributes in automatic marks (page 835)

Document manager (page 552)

Customize print file names (page 622)

Modify automatic user-defined drawing attributes (page 896)

Modify automatic user-defined drawing attributes

You can modify the user-defined attributes before you create a drawing. The drawing UDAs may vary according to the environment.

1. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.
2. Load drawing properties that are as close to the ones you need as possible.
3. Click **User-defined attributes**.
4. On the **Workflow** tab, enter project-specific information to be displayed in drawings and in the **Document manager**.
5. On the **Parameters** tab, enter a user-defined **Comment** for drawings, projects, assemblies, parts, and so on.
6. Use the **User field 1** to **User field 8** on the **Parameters** tab to enter drawing-specific information.
7. On the **Title** tab, select if you want to use project-specific or drawing-specific information in the title block of the drawing.

   When you click **Use project settings**, you cannot enter any information in the title block fields.
8. If you selected **Use drawing settings**, the title block fields become available, and you can enter the necessary data in the fields.
9. General arrangement drawings: Click **OK**.
10. Click **Save** to save the drawing properties, then click **OK** and create the drawing.

TIP • You can control whether the modifications in user-defined attributes affect all selected drawings in the **Document manager** at the same time, even if the drawings are of different types by using the advanced option XS_DRAWING_UDAS_MODIFY_ALL_DRAWING_TYPES.

See also

Create a new user-defined drawing attribute (page 896)

Create a new user-defined drawing attribute

If you want to add a new line in the user-defined drawing attributes dialog box in drawings and a new column in **Document manager**, you need to create your own `objects.inp` and add a new user-defined attribute there. You also need to do this if you want the user-defined attribute to be available in Template Editor.

Before you start, close Tekla Structures.

1. Open the `objects.inp` file located by default in `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\inp`.
2. Save the `objects.inp` file in your model, project or firm folder.
3. Delete all user-defined attribute definitions from the `objects.inp` file, except the property definitions in the beginning of the file that describe the properties, and the sections that define the new tab to different drawing types. Also leave one property definition in section User-defined attributes for objects that you can use as a template for the new property.
4. Enter the name of the tab page where you want the new user-defined attributes to be located and the attribute properties.

   Do not enter the names of the tab pages that come with the installation, use a new one instead.
5. To show the user-defined attribute in the **Document manager** and in the user-defined attributes dialog box, set `status_flag` to `yes`.

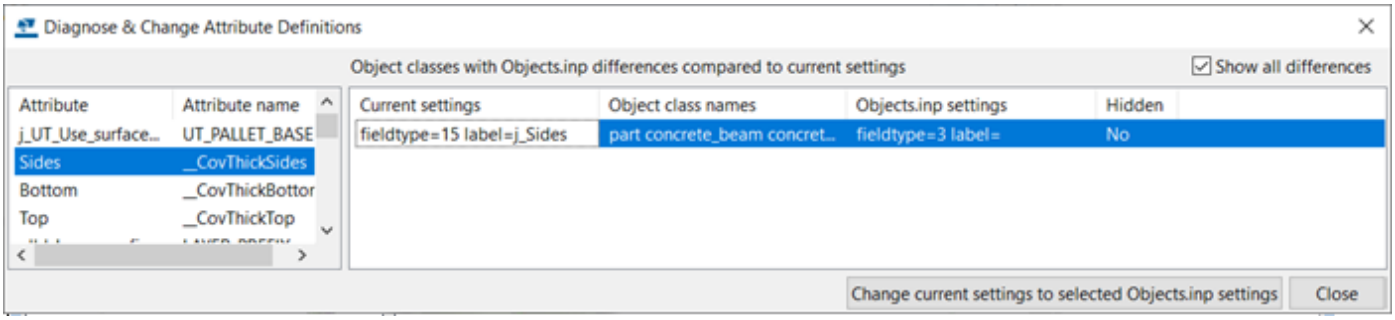
```
/*****************************************************************************/
/* User defined attributes for objects */
/*****************************************************************************/
/*****************************************************************************/
/* Common drawing attributes*/
/*****************************************************************************/
 beam(2,"Beam")
  {
  tab_page("My tab")
   {
      attribute("DESIGNED_BY","Designed By", string, "%s", yes, none, "0.0", "0.0")
    {
      value("", 0)
    }
   }
  modify(1)
}
```

6. Define the drawing types where you want to use the new tab containing the new user-defined attribute.

```
/*****************************************************************************/
/* Drawing attributes - single part */
/*****************************************************************************/
singledrawing(0,"j_Single_part_drawing")
{
 tab_page("My tab","My tab",200)
 modify(1)
}
/*****************************************************************************/
/* Drawing attributes - GA */
/*****************************************************************************/
gadrawing(0,"j_GA_drawing")
{
 tab_page("My tab","My tab",200)
 modify(1)
}
```

7. Save and close the file.
8. Update the definitions of the user-defined attributes:
   a. On the **File** menu, click **Diagnose & repair** --> **Diagnose and change attribute definitions** .

      The **Diagnose and change attribute definitions** dialog box is displayed.

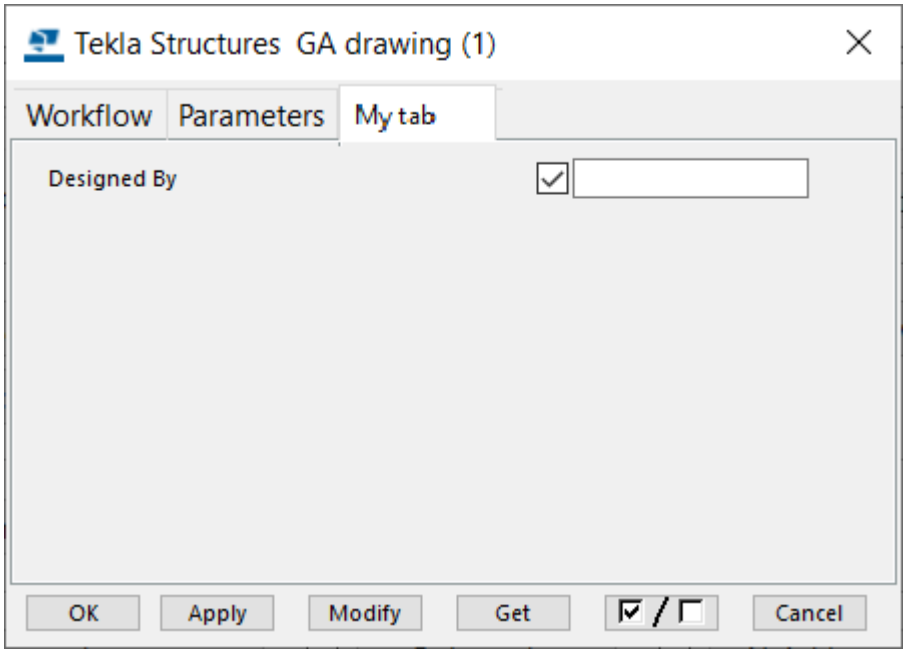


   b. If there are conflicts between your `objects.inp` and the default `objects.inp`, select the definition in the area on the right and click **Change current settings to selected Objects.inp settings**.

      The definition of the user-defined attribute is updated in the model.
9. On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type.

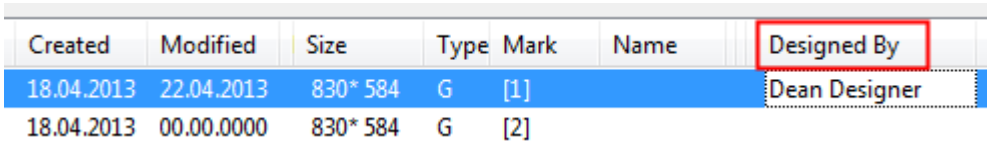
   In this example, select **GA drawing**.
10. Click **User-defined attributes...**

The user-defined drawing attributes dialog box of the general arrangement drawing shows the tab you created.



11. Use **Save as** to save the properties for future use.
12. Click **OK** and create a general arrangement drawing.
13. Right-click the created drawing in **Document manager** and select **User-defined attribute** .
14. Go to the new tab (**My tab**) and enter the needed information (the designer name in the **Designed By** box).
15. Click **Modify** and close the dialog box.

    Document manager contains a column for the new user-defined attribute, and the name of the designer is displayed:



See also

User-defined attributes in drawings (page 895)

8.17 Define customized line types in TeklaStructures.lin

You can define your own line types, and use them wherever line type settings are available. The customized line types are handled the same way as other

line types. The customized line types are defined in the `TeklaStuctures.lin` file in `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\inp`. By default, the most widely used line types are available in the file.

1. Open the `TeklaStuctures.lin` file.
2. Start every line with the letter **A** to indicate the beginning of the line type pattern.

   You can use three different objects: dashes, dots and spaces to form a line.
3. Define the length of a dash (**-**) using positive numbers.
4. Define the length of a space ( ) using negative numbers.
5. Define dots (**.**) using the number zero (**0**).

   The patterns must begin with a dash. They typically end with a space, although the space in the end is not required.
6. After you have defined the line type pattern, press **Enter**.

The `TeklaStructures.lin.id` file contains the names of the line types visible in the user interface and the unique IDs assigned for each line type. The value of the ID must be greater than 10, for example:

```
CENTER, 1000
BORDER, 1002
DASHDOT, 1003
```

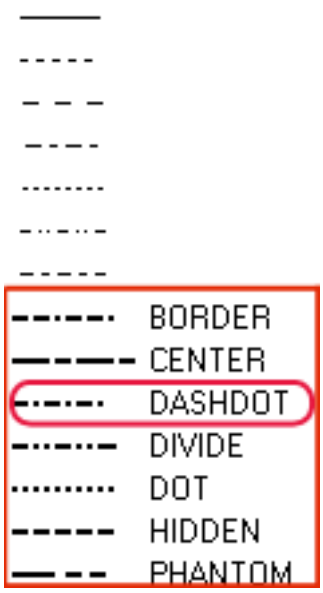
You can use `TeklaStructures.lin` also in mapping the exported line types.

NOTE If you add new customized line types, you need to add corresponding bitmaps to the `..\ProgramData\Trimble\Tekla Structures\<version>\bitmaps` folder, and name them `dr_line_type_*.bmp`, for example `dr_line_type_CENTER.bmp`.

Example 1

Line type definition for `DASHDOT` is `A, 12.7, -6.35, 0, -6.35`

This string means that the pattern starts with a dash with the length of `12.7` units, followed by a space with the length of `6.35` units, then a dot, and then again a space with the length of `6.35` units. Then the first dash is drawn again.



Example 2

Below is an example containing definitions for dotted lines:

```
*DOT,Dot . . . . . .
A, 0, -1.5875
*DOT2,Dot (.5x) .....
A, 0, -0.79375
*DOTX2,Dot (2x) . . . . . .
A, 0, -3.175
```

Printed line types

Printing will look for the line type files in the model folder, and in the locations specified by the advanced options `XS_PROJECT`, `XS_FIRM`, `XS_SYSTEM` and `XS_INP`, in that order.

See also

Drawing sketch object properties (page 1038)

9 Drawing settings

There are a lot of settings in Tekla Structures that you mainly control in the properties dialog boxes. There are also some additional settings files that you need to modify in a text editor.

Click the links below to find out more:

- General arrangement drawing properties (page 903)
- Single-part, assembly and cast unit drawing properties (page 906)
- Layout properties (page 910)
- View properties in drawings (page 912)
- Section view properties (page 929)
- Dimension and dimensioning properties in drawings (page 931)
- Mark properties in drawings (page 972)
- Mark elements in drawing marks (page 981)
- Part and neighbor part properties in drawings (page 1004)
- Bolt properties in drawings (page 1010)
- Surface treatment properties in drawings (page 1012)
- Reinforcement object properties in drawings (page 1016)
- Pour object and pour break properties in drawings (page 1029)
- Reinforcement settings for drawings (rebar_config.inp) (page 1023)
- Placement properties for marks, notes, dimensions, texts and symbols in drawings (page 1033)
- Model weld properties in drawings (page 1036)
- Drawing sketch object properties (page 1038)
- Drawing grid and grid line properties (page 1041)
- Settings in the Options dialog box: The **Options** dialog box contains some settings for controlling drawings.

9.1 General arrangement drawing properties

On the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing** . You can modify the properties after creating the drawing by double-clicking the drawing background.

For more information about creating and modifying general arrangement drawings, click the links below:

Quick start to Tekla Structures drawings (page 15)

Create general arrangement drawings (page 105)

Create general arrangement drawings using saved settings in Master drawing catalog (page 126)

The settings and options in the general arrangement drawing properties are described below.

Setting	Description	For more information
Name	The name of the drawing. This name is shown in **Document manager**, and can be included in drawing and report templates. The maximum length of the drawing name is 32 characters.	Modify drawing names and titles (page 167)
Title 1, **Title 2**, **Title 3**	Titles are shown in the **Document manager**, and can be included in drawing and report templates.	Modify drawing names and titles (page 167)
Use detailed object level settings	Set to **Yes** to use the detailed object level settings created in the **Object level settings for drawing** dialog box. To manage detailed object level settings, click **Edit**.	Detailed object level settings (page 79)
Layout...	Select the drawing layout and define drawing sizes. You can also select whether you want to list hidden objects in templates. To open **Drawing layout editor**, click **Edit**.	Define drawing layout Define drawing size and drawing view scale (page 673)
View...	Define view properties, such as scale, neighbor part view extension, reflected view,	View properties in drawings (page 912)

Setting	Description	For more information
	opening and recess symbol, datum point elevation, part shortening, view label, and anchor bolt plan settings.	
Detail view...	Define detail view properties, such as view label, detail boundary and detail mark settings.	Create drawing views (page 180)
Section view	Define the start number or letter for the section view and symbol label for the section views that you will manually add to the GA drawing.	Create drawing views (page 180)
Dimension...	Define dimension properties: dimension type, units, precision, format, placing, and appearance.	Dimension properties
Dimensioning...	Define grid dimensioning and part dimensioning settings.	General arrangement drawing dimensioning properties (page 970)
Part mark... **Bolt mark...** **Neighbor part mark...** **Surface treatment mark...** **Weld mark...** **Reinforcement marks...** **Neighbor reinforcement marks...** **Connection mark...** **Pour object mark...**	Define mark properties, such as included elements and element settings, mark visibility, mark frame, mark leader line, and placing settings.	Mark properties in drawings (page 972) Model weld mark properties in drawings (page 996) Mark elements (page 981) Show pours in drawings (page 485)
Part...	Define part properties: part representation, hidden line, center line and reference line visibility, additional mark	Part and neighbor part properties in drawings (page 1004)

Setting	Description	For more information
	visibility, part appearance, and fill settings.	
Bolt...	Define bolt properties: bolt representation, bolt symbol content, bolt visibility, and bolt appearance settings.	Bolt properties in drawings (page 1010)
Neighbor part...	Define neighbor part properties: visibility, part representation, hidden line, center line and reference line visibility, additional mark visibility, and part appearance settings. You can also define bolt representation and bolt symbol content settings for neighbor parts.	Part and neighbor part properties in drawings (page 1004)
Surface treatment...	Define surface treatment properties: surface treatment visibility, pattern visibility, hidden lines visibility and surface treatment representation settings.	Surface treatment properties in drawings (page 1012)
Weld...	Define welding properties: weld visibility, weld size limit and weld appearance settings.	Model weld properties in drawings (page 1036)
Reinforcement...	Define reinforcement and mesh properties: bar and mesh representation and visibility, bar and mesh symbol, and bar and mesh appearance settings.	Reinforcement object properties in drawings (page 1016)
Neighbor reinforcement...	Define neighbor reinforcement and mesh properties: bar and mesh representation and visibility, bar and mesh symbol, and bar and mesh appearance settings.	Reinforcement object properties in drawings (page 1016)
Reference objects...	Define reference object visibility and appearance settings.	Show reference objects in drawings (page 526)

Setting	Description	For more information
Grid...	Define grid visibility and appearance settings.	Drawing grid and grid line properties (page 1041) Customize drawing grid labels (page 516)
Pour breaks...	Define pour break visibility and appearance settings	Pour break properties (page 1029)
Pour object...	Define pour object visibility and appearance settings.	Pour object properties (page 1029)
Protection...	Define protected areas to prevent text, marks, or dimensions from being placed in these areas.	Protect areas in drawings (page 679)
Filter... and **Neighbor part filter...**	Create and modify part filters or neighbor part filters on the drawing level.	Create new filters
User-defined attributes...	Add customized information to a drawing, such as information related to workflow, and comments. This information can be displayed in **Document manager**, and you can use it in report and drawing templates and marks, and as switches when customizing print file names. The user-defined attributes and tabs shown in this dialog box are defined in the `objects.inp` file. The availability of the various user-defined attributes depends on your environment, configuration, and role.	User-defined attributes in drawings (page 895)

9.2 Single-part, assembly and cast unit drawing properties

On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type. You can modify the properties after creating the drawing by double-clicking the drawing background.

For more information about creating single-part, assembly, and cast unit drawings, click the links below:

Quick start to Tekla Structures drawings (page 15)

Create fabrication drawings (page 107)

The settings and options in single-part, assembly and cast unit drawing properties are described below.

Setting	Description	For more information
Name	Define the name of the drawing that is shown in the **Document manager**, and that can be included in drawing and report templates. The maximum length of the drawing name is 32 characters.	Modify drawing names and titles (page 167)
Cast unit definition method	This setting is available for cast unit drawings only. **By cast unit position**: A drawing will be created from each cast unit. If there are identical cast units, one of them will serve as the base cast unit for the drawing. This is the most common method for creating cast unit drawings. **By cast unit ID**: Each part in the model has a unique GUID. You can create drawings by using the cast unit GUIDs. The GUID determines the marking of the drawing. You can create several drawings from identical cast units.	
Title 1, **Title 2**, **Title 3**	Define titles that are shown in **Document manager**. The	Modify drawing names and titles (page 167)

Setting	Description	For more information
	titles can be included in drawing and report templates.	
Sheet number	Used for creating several drawings of the same part as multiple drawing sheets. The number of sheets is not limited.	Create multiple drawing sheets of the same part (page 162)
Layout	Select the drawing layout, and define drawing sizes, autoscale, projection type, view alignment and part expansion settings. You can also select whether you want to list hidden objects in templates. To open **Drawing layout editor**, click **Edit**.	Define drawing layout Define drawing size and drawing view scale (page 673)

View creation	Define the drawing views to be created. From here you can move on to modifying the view properties for each view.
	Here you can also define **Common settings for all views**:
	• **Weld mark visible** controls the visibility of model weld marks in assembly drawings. The options are:
	  • **In one view**: Tekla Structures automatically finds the most relevant view for showing the model weld marks. Every weld mark is shown in a drawing in one view only.
	  • **In all views**: Tekla Structures adds the model weld marks in all drawing views that
		View properties in drawings (page 912)
		Define the views to create (page 689)

Setting	Description	For more information
	contain the part with the weld. • **Ignore bolt size** filters standard-sized bolts out of drawings, which means that Tekla Structures does not display bolt marks of the defined bolt size in drawings. This setting is available in single-part and assembly drawings. • **Weld size limit** filters welds and weld marks of the defined weld size and smaller out of the drawing. This setting is available in single-part and assembly drawings.	
Section view	Define section view properties.	Create views in drawings (page 180) Section view properties (page 929) Add and modify section marks in drawings (page 343)
Detail view	Define the start number or letter of the detail view and detail symbol label.	Create views in drawings (page 180) Add and modify detail marks in drawings (page 347)
User-defined attributes	Add customized information to a drawing, such as information related to workflow, and comments. This information can be displayed in the **Document manager**, and you can use it in report and drawing templates and marks, and as switches when customizing print file names. The user-defined attributes and tabs shown in this dialog	User-defined attributes in drawings (page 895)

Setting	Description	For more information
	box are defined in the `objects.inp` file. The availability of the various user-defined attributes depends on your environment, role and configuration.	

9.3 Layout properties

On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type. Then click **Layout**. You can modify the properties after creating the drawing by double-clicking the drawing background.

All options in all layout property panels and dialog boxes are described below. All the described options are not available for all drawing types.

Option	Description
Drawing size tab	
Layout	Define the layout that you want to use.
Edit	Open the **Drawing layout editor** to customize drawing layouts.
List hidden objects in templates	Select **Yes** to list hidden objects in templates. **No** removes all information about the hidden parts, also from the total weight.
Size definition mode	Select **Autosize** if you want to let Tekla Structures find appropriate sizes and table sets for drawings. Select **Specified size** to specify the exact size for the drawing. The drawing size should always be smaller than the actual paper size to accommodate printer margins.
Drawing size	If you selected **Specified size**, define the drawing size here.
Scale tab	
Autoscale	Set **Autoscale** to **Yes** to let Tekla Structures automatically select the suitable scale for the drawing view.

Option	Description
Main view scales **Section view scales**	When you use autoscaling, enter the denominators of main view and section view scales and separate them with spaces. For example, enter "5 10 15 20" for the scales 1/5, 1/10, 1/15, and 1/20
Scale change mode	When you use autoscaling, set the scale change mode which defines the relationship between the scales of main and section views within a drawing: **main = section**: The scales of the main and section view are equal. **main < section**: Main view scales are smaller than section view scales. **main <= section**: Main view scales are smaller than or equal to section view scales
Preferred size	Enter the preferred size of the drawing, if you use both autosizing and autoscaling. Tekla Structures tries to find a drawing size where the drawing contents fit by first trying to use the exact scale and the smallest drawing size. If the contents do not fit, Tekla Structures increases the drawing size until it reaches the preferred size.
Other tab	
Projection type	Define how Tekla Structures places the projections of a part in cast unit, single-part, and assembly drawings. Projection type affects the order of the views in the drawing. The options are: **First angle**, also referred to as the European projection. **Third angle**, also referred to as the American projection.
Align section views with main view **Align end views with main view**	Set to **Yes** to place the views next to the main view. If you select **No**, Tekla Structures places the section and end views in any available location.
Expand shortened parts to fit	Set to **Yes** to stretch shortened views to fill up empty areas of the drawing.
Include single-parts	Set to **Yes** to include in assembly drawings the single-part drawings of the individual parts that make the assembly. Setting this to **Yes**, activates the **Single-part attributes** option.

Option	Description
Single-part attributes	Define the desired single-part drawing properties to be used in the single-part view. To do this, set **Include single-parts** to **Yes**.

See also

Define drawing size and drawing view scale (page 673)

Define drawing layout (page 655)

Set drawing view projection type (page 693)

Define automatic section view properties (page 717)

Shorten or lengthen parts (page 708)

Include single-part drawings in assembly drawings (page 695)

9.4 View properties in drawings

In drawing view properties, you can check and modify the view properties.

All drawing view properties are described here. The view properties shown in the drawing property pane differ to some extent from the view properties that you open through the drawing properties dialog boxes or drawing ribbon, and the properties may be named differently.

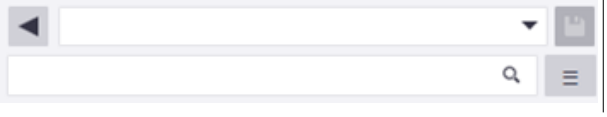
View properties in property pane

You can use the view properties in the property pane to quickly modify the properties for the views you have selected in an open drawing or for the views that you are going to create, or to load predefined view properties, for example.

To open drawing view properties in the drawing property pane:

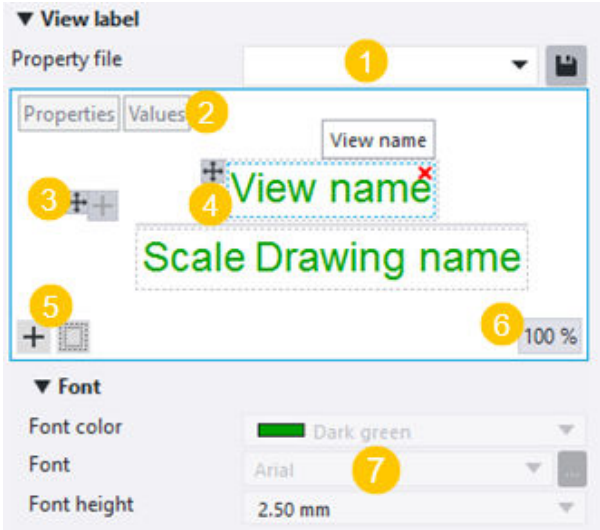
- In an open drawing, click a drawing view frame. If the drawing property pane is not open, double-click the view frame. Double-clicking the frame also opens the view properties dialog box, which has more properties available.
- In the drawing property pane object list, select **View**.

All the view properties that are available in the drawing property pane are described below.

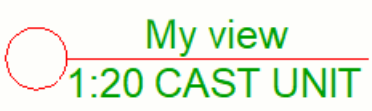
Setting	Description
Save, load and search properties in property pane	

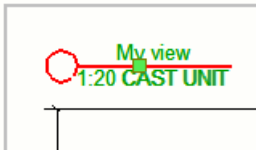
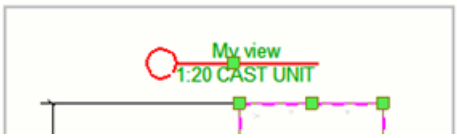
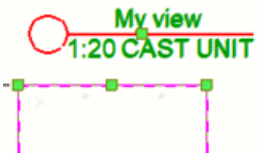
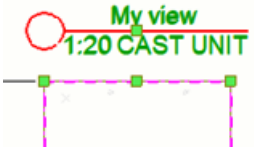
Setting	Description
	• To load existing properties, select the properties from the list. • To save the modified properties, click . You can also save the properties with another name. • To search for a property, start typing the property name.
Size and placing	
Scale	Set the view scale.
Size	**Fit by parts**: Tekla Structures fits the view contents in the drawing view frame without leaving any unnecessary space. **Define as distances**: **Depth down** and **Depth up** define the view size along the x and y axes of the view. **X min / X max** and **Y min / Y max** define the depth of the view relative to and perpendicular to the view plane.
View extension for neighbor parts	Enter a value by which to extend the view so that neighbor parts are shown. For more information, see "Show neighbor parts in drawings".
Reflected view	Display load bearing structures, such as columns and beams on a lower floor. If you select this option, structures are displayed with a continuous line, and if you do not select this option, structures are displayed with a dashed line.
Rotation around (in 3D views)	Modify the view angle of 3D views. Enter the values for the angles in Y and X directions. Rotation in a drawing view is around the local axis. Value 0.0 for both equals the front view.
Place	Set the placing for the drawing view to be fixed or free: • **Fixed**: Maintain the views in the same location during updates.

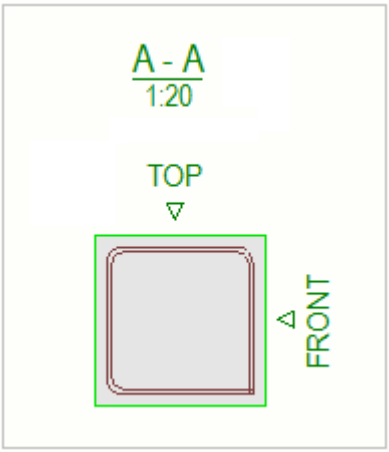
Setting	Description
	• **Free**: Let Tekla Structures find a suitable place for the view during updates. For more information, see "Define automatic free or fixed placement for drawing views" (page 686).
Part representation	
Cut parts	If parts are very long and do not include any details, you can shorten them. **Yes** activates shortening. You can also select to cut **Only in x direction** or **Only in y direction**. For more information, see "Shorten parts in drawing views" (page 708).
Cut skew parts	When selected, Tekla Structures cuts skewed parts.
Minimum cut part length	Define how long the part must at least be to get shortened. The length of the part must be at least twice the entered value.
Space between cut parts	Define the distance between cut parts.
Undeform parts	Select this option to undeform deformed parts and show the developed (undeformed) shape of the deformed parts in drawings. For more information, see "Undeform deformed parts in drawings" (page 713).
Show openings/recess symbol	When selected, Tekla Structures shows symbols in openings and recesses. For more information, see "Show part openings or recesses in drawings" (page 715).
Base points and dimensions	
Location by	Select model origin, project base point, or any base-point-defined coordinate system. **Location by** uses the project base point as the default value. • Base point data can be used to set the coordinate system for the view. The base point can be used instead of datum offset. • When the base point is set, level attributes and template attributes in

Setting	Description
	marks provide values in the specific base-point-defined coordinate system. • If you change the project base point Z or the elevation value, the level value will change accordingly when a drawing is opened. • This setting affects level marks and attributes that end with `_BASEPOINT`.
Datum points for elevations	Define the reference point for elevations. **Specified** uses the value that you enter. **View plane** measures reference points relative to the view plane.
Dimension creation	Creates the dimensions separately for the selected view only. This option affects the creation of the dimensions during cloning and redimensioning of existing drawings. The options are: **Automatically** **Do not create** **Clone** For more information, see "Clone dimensions in selected views only".
View label	
View label editor	 **(1) Property file** allows you to load and save view label contents. To activate the setting, click a label container in the editor.

Setting	Description
	(2) Click the **Properties** or **Values** button to show either the property name or the property value in the editor. These buttons are only available when you have selected an existing view in a drawing. **(3)** A new, empty label container added by using the **New container** button. Use the new element button to add elements in the container. Use the handle to drag the container to another location. **(4)** Label containers around the view label line. Click a label container to select it. Use the **New element** button to add new elements. Use the red delete button in the upper-right corner to remove the contents. To check the label container position number, point the label container handle. **(5)** Use the **New element** button to open the element list and select the element to add. The element list opens automatically when you click an empty label container. You can also search for elements. Use the **New container** button to add new containers. You may add up to five containers. You can drag elements and containers to new locations. **(6)** The % button shows the current zoom level. You can zoom in or out with the middle mouse button. Click the % button to zoom to the best fit. **(7)** Label properties
Font	
Font color, **Font**, **Font height**	Define the font color, type, and height used in the label. Clicking **...** gives more font choices.
Frame	

Setting	Description
Frame, **Frame color**	Define the frame type and color for one or several elements, or for the label container. These properties become available when you select an element or a label container. You can also add the frame from the element list.
Numeric value	
Units, **Format**	Select the desired unit and format when appropriate.
Text placement	
Alignment	Define the label mark alignment: **Left**, **Center**, or **Right**. This setting is enabled when you have selected a label mark in the label editor.
Text position	Define the text position in the label. This setting is enabled when you have selected a label mark in the label editor.
X offset for text, **Y offset for text**	Define the text offset in the X or Y direction. This setting is enabled when you have selected a label mark in the label editor.
Symbol and line	
Symbol, **Symbol color**, **Symbol size** , **Line length**	Define the symbol to be used in the view label, the color and size of the symbol, and the label line length. The symbol options are: **None** **Empty circle** **Crossed circle** **Custom**  For more information about using custom symbols, see the support article "How to add two custom symbols in a view label".
Label placement	
Vertical position	Define the vertical position of the view label:

Setting	Description
	Above places the view label at the top of the view. **Below** places the view label at the bottom of the view.
Horizontal position	Define the horizontal position of the view label: **Left aligned to view frame**  **Center aligned to view frame**  **Right aligned to view frame** **Left aligned to view restriction box**  **Center aligned to view restriction box**  **Right aligned to view restriction box**

Setting	Description
Direction marks	
Mark type, **Height**	Show view direction marks. The options are **None**, **Symbol**, **Label**, and **Symbol and label**. The label means the view label. You can also define a height for the mark. If you try to use 0, you will get an error message. 
More	
View	Open the view properties dialog box with more settings.
Object groups	Open the **Object level settings for view** dialog box to define detailed object level settings (page 79) for object groups.
Object group settings	Enable the defined object group settings.

View properties in view properties dialog boxes

There are some view properties that are displayed in the view properties dialog boxes only, not in the property pane. These properties can be used for setting up the views and view properties before you create drawings, or when you want to change some of the view properties for all views in a drawing, for example.

To open the view properties dialog box, depending on where you are:

- In the drawing property pane **Drawing view** properties, go to the **More** section and click **View**.
- In an open drawing, double-click a view frame. This displays the view properties for the current view.
- In the drawing ribbon, on the **Drawing** tab, click **Properties** --> **View**.

- On the **Drawings & reports** tab, click **Drawing properties** and select the drawing type. Then select **View creation** and a drawing view from the list, and click **View properties** on the right.
- For GA drawings, on the **Drawings & reports** tab, click **Drawing properties** and select **GA drawing**. Then click the **View...** button.

All of the settings in all view properties dialog boxes are described below.

Setting	Description
View creation in the options tree	When **View creation** is selected in the options tree in single-part, assembly, and cast unit drawings, you can define the drawing views that you want to create. From here you can move on to modifying the view properties for each view separately by clicking the **View properties** button.
Views tab	
View type and **on/off** columns	Define the main views, section views, end views, and 3D views that you want to create. Use the **Delete row** and **Add row** buttons to delete views from or add views to the list. • In **View type**, select the type of the view that you want to create. • If you select **Off**, Tekla Structures does not create the view, but dimensions the parts in the available views. If you set all four main views off, Tekla Structures will still create a front view. • If you select **On**, Tekla Structures always creates the view, even if it is not necessary in order to show the dimensions. For section views, Tekla Structures creates one additional section view that shows the middle of the main part. For end views, Tekla Structures creates an end view from one end of the main part. • If you select **Auto**, Tekla Structures creates the view if it is necessary in order to show the dimensions. For section views, Tekla Structures creates the necessary number of

Setting	Description
	views to show all the dimensions. For end views, Tekla Structures also creates another end view from the other end of the main part, if there are dimensions at that end. For more information, see "Define the views to create" (page 689).
View label column	Set the view name. To show the view name in the view label automatically, you need to define it here.
View properties column	Shows the current view properties for the selected view. You can select another view properties file from the list, or check and modify the view properties by clicking **View properties**. Note that if you have selected **Auto** in the **on/off** column as the creation method, Tekla Structures will use the standard properties even if you define another property file here.

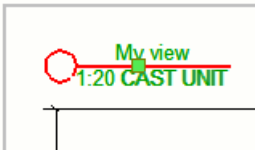
Common settings for all views	Define some common settings for all views:
	• **Weld mark visible** controls the visibility of model weld marks in assembly drawings. The options are:
	  • **In one view**: Tekla Structures automatically finds the most relevant view for showing the model weld marks. Every weld mark is shown in a drawing in one view only.
	  • **In all views**: Tekla Structures adds the model weld marks in all drawing views that contain the part with the weld.
	• **Ignore bolt size** filters standard-sized bolts out of drawings, which means that Tekla Structures does not display the bolt marks of the defined bolt size in drawings. This

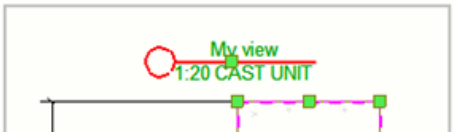
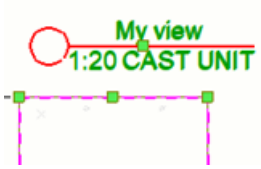
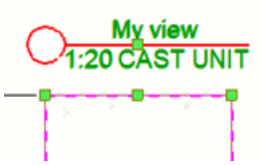
Setting	Description
	setting is available in single-part and assembly drawings. • **Weld size limit** filters welds and weld marks of the defined weld size and smaller out of the drawing. This setting is available in single-part and assembly drawings.
Recreate the drawing	**Yes** recreates the drawing.
Attributes tab	
Coordinate system	Set the coordinate system of the drawing views. The options are: local, model, oriented, horizontal brace, vertical brace, and fixed. For more information, see "Change the coordinate system" (page 697).
Rotate coordinate system **Around X**, **Around Y**, **Around Z**	Rotates the view around the x, y, or z axis of the parts using the entered values. For more information, see "Rotate parts in drawing views" (page 697).
Unfolded	This setting is only shown for single-part drawings. **Yes** shows and dimensions the bend lines in the drawing. For more information, see "Unfold polybeams in drawings" (page 712).
Undeformed	**Yes** undeforms deformed parts and shows the developed (undeformed) shape of the deformed parts in drawings. For more information, see "Undeform deformed parts in drawings" (page 713).
Attributes 1 and **Attributes 2** tabs in view properties dialog box (**Attributes** and **Shortening** tabs in **General - View Properties**) :	
Scale	Set the view scale.

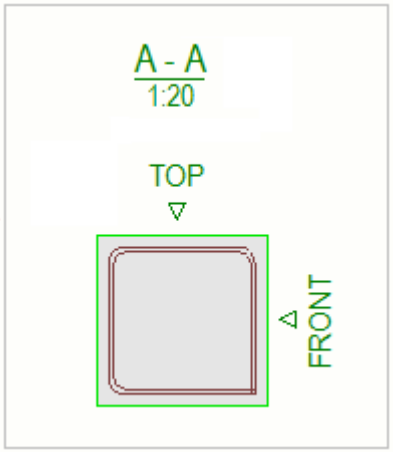
Setting	Description
Reflected view	Displays the load bearing structures, such as columns and beams on a lower floor. **Yes** displays the structures with a continuous line, and **No** displays them with a dashed line.
Rotation around (in 3D views)	Modify the view angle of 3D views. Enter the values for the angles in the Y and X directions. Rotation in a drawing view is around the local axis. Value 0.0 for both equals the front view.
View extension for neighbor parts	Enter a value by which to extend the view so that neighbor parts are shown. For more information, see "Show neighbor parts in drawings".
Size	**Fit by parts**: Tekla Structures fits the view contents in the drawing view frame without leaving any unnecessary space. **Define as distances**: **Depth down** and **Depth up** define the view size along the x and y axes of the view. **X min / X max** and **Y min / Y max** define the depth of the view relative to and perpendicular to the view plane.
Place	Set the placing for the drawing view to be fixed or free: • **Fixed**: Maintain the views in the same location during updates. • **Free**: Let Tekla Structures find a suitable place for the view during updates. For more information, see "Define automatic free or fixed placement for drawing views" (page 686).
Use detailed object level settings	Set to **Yes** to apply the detailed object level settings that you have defined for the selected object groups.

Setting	Description
Unfolded	**Yes** shows and dimensions the bend lines in the drawing. This setting is only shown for single-part drawings. For more information, see "Unfold polybeams in drawings" (page 712).
Undeformed	**Yes** undeforms deformed parts and shows the developed (undeformed) shape of the deformed parts in drawing views. For more information, see "Undeform deformed parts in drawings" (page 713).
Shortening	If parts are very long and do not include any details, you can shorten them: • **Cut parts**: **Yes** activates shortening. You can also select to cut **Only in x direction** or **Only in y direction**. • **Cut skew parts**: **Yes** cuts skewed parts. Note that setting **Cut skew parts** to **No** cuts secondary parts that are skewed less than 5.7 degrees in relation to the main part. • **Minimum cut part length** defines how long the part must at least be to get shortened. The length of the part must be at least twice the entered value. • **Space between cut parts** defines the distance between cut parts on paper. For more information, see "Shorten or lengthen parts" (page 708).
Show openings/recess symbol	**Yes** shows symbols in openings and recesses. For more information, see "Show part openings and recesses in drawings" (page 715).
Location by	Select the model origin, project base point or any base-point-defined

Setting	Description
	coordinate system. **Location by** uses the project base point as the default value. • Base point data can be used to set the coordinate system for the view. The base point can be used instead of datum offset. • When the base point is set, level attributes and template attributes in marks provide values in the specific base-point-defined coordinate system. • If you change the project base point Z or the elevation value, the level value will change accordingly when a drawing is opened. • This setting affects level marks and attributes that end with `BASEPOINT`.
Datum point for elevations	Define the reference point for elevations. **Specified** uses the value that you enter in **Datum level**. For example, if the elevation is 5000 mm, and you set the reference point to 200, the elevation changes to 4800 mm. **View plane** measures reference points relative to the view plane.
Show pours in drawing	This setting is only available in the view properties of general arrangement drawings. **Yes** shows pours in drawings. For more information, see "Show pour objects, pour marks, and pour breaks in drawings" (page 883).
Dimension creation method in this view	Clones the dimensions separately for the selected view only. This option affects the creation of the dimensions during cloning and re-dimensioning of existing drawings. The options are: **Automatically**

Setting	Description
	Do not create **Clone** For more information, see "Clone dimensions in selected views only" (page 153).
Labels tab:	
Text	Defines the view label text. Enter text in fields **A1**-**A5**, or click the ... buttons and select the label mark elements and appearance. For more information about view labels, see "Define view labels and view label marks" (page 691).
Symbol	Define the symbol to be used in the view label, the **Color** and **Size** of the symbol, and label **Line length**. The symbol options are: **None** **Empty circle** **Crossed circle** **Custom** For more information about using custom symbols, see the support article "How to add two custom symbols in a view label".
Label position	Defines the vertical and horizontal position of the view label. • For **Vertical**: **Above** places the view label at the top of the view. **Below** places the view label at the bottom of the view. • For **Horizontal**: **Left aligned to view frame** 

Setting	Description
	Center aligned to view frame  **Right aligned to view frame** **Left aligned to view restriction box**  **Center aligned to view restriction box**  **Right aligned to view restriction box**
View direction marks	Show view direction marks. You can define view direction marks for section views and end views only. The options are **None**, **Symbol only**, **Label only**, and **Symbol and label**. The label means the view label. You can also define a height for the mark. If you try to use 0, you will get an error message.

Setting	Description
	The following example shows the view direction marks together with the view label. 
Other options in the view properties options tree:	
Dimensioning	Define view-level dimension settings for each view separately. For more information on dimensioning settings, see "Dimensioning rule properties" (page 942) and "Add automatic view-level dimensions" (page 730).
Filter	Define drawing view filters. For more information, see "Create filters".
Neighbor part filter	Define drawing view filters for neighbor parts. For more information, see "Create filters".
Protection	Define protected areas to prevent text, marks or dimensions from being placed there. For more information, see "Protect areas in drawings" (page 679).
Marks	Define view-level mark properties. For more information, see "Mark properties" (page 972).
Objects	Define view-level properties for the following objects: Parts and neighbor parts (page 1004)

Setting	Description
	Bolts (page 1010) Surface treatment (page 1012) Welds (page 1036) Reinforcement and neighbor reinforcement (page 1016) Reference objects (page 526) Grid (page 1041) Pour object and pour breaks (page 1029)
Anchor bolt plan tab (**General - View Properties**)	
Show as anchor bolt plan	**Yes** shows the general arrangement drawing as an anchor bolt plan. For more information, see "Create anchor bolt plans using saved settings" (page 127).
Enlarged part view scale	Define the scale used in the enlarged part views.
Create detail views	**Yes** creates separate detail views. If you select **No**, Tekla Structures dimensions the anchor bolts in the enlarged view. Tekla Structures groups similar detail views so that similar details are drawn only once.
Detail view scale	Defines the scale used in the anchor bolt plan detail views.

9.5 Section view properties

Section view properties in single-part, assembly, and cast unit drawing properties define the section depth, section view direction, cutting line length and color, and section mark contents and appearance for the automatically created section views.

To open automatic section view properties:

- On the **Drawings & reports** tab, select **Drawing properties**, select the drawing type, and select **Section view** in the options tree.
- In an open drawing, double-click the drawing background, and select **Section view** in the options tree.

- In GA drawings, clicking the **Section view** button opens a dialog box where you can enter the section view and symbol start number or letter.

Setting	Description
Attributes tab	
Fit by parts	The **Fit by parts** setting shows the whole part in the sections. This setting works as an alternative to the **Section depth** and **Distance for combining cuts** settings.
Section depth	Define the positive and negative distances of the section view when views are not combined.
Distance for combining cuts	Define the distance range for combining cut views.
Direction	Define the view direction of the section. The options are: • **Left section** • **Middle section** • **Right section** The available values are **left** or **right**.
Cutting line tab	
Line	Define the length and offset of the cutting line.
Properties	Define the color of the cutting line.
Section mark tab	
Text	Define the text on the section mark. Click the **...** buttons next to the text boxes to open the **Mark Contents** dialog box and to modify the mark contents.
Symbol: **Color**	Define the color of the section mark symbol.
Left symbol, **Right symbol**	Define the left and right section mark symbol.
Size	Define the size of the section mark left and right symbol.
x/y	Define the offset of the section mark left and right symbol.

Setting	Description
Start number or letter of section view and symbol label	Define the start letter or number used in the section view label and in the section symbol label. You can enter any number starting from 1 or any letter A - Z or a - z (also shown in uppercase in the label). If you use letter, and the entered string is longer than one letter, only the first letter is shown in the view label and section symbol label. If you use numbers, all entered numbers are shown. The start number changes only if you change it in the drawing properties before creating a drawing, and if you change the option in an existing drawing and recreate the drawing, in which case section view and symbol labels for all automatically included section views and all new section views will change.

See also

View properties in drawings (page 912)

Define object protection and placement settings in drawings (page 677)

Define drawing views (page 687)

Define the views to create in single-part, assembly and cast unit drawings (page 689)

Create views in drawings (page 180)

9.6 Dimension and dimensioning properties in drawings

Dimension and dimensioning properties define what the dimensions look like, and which formats, units etc. are used, and what is being dimensioned and how.

Click the following link to find out more about dimension properties in property pane and in GA drawings:

- Dimension properties (page 932)

Click the following links to find out more about dimensioning properties in view-level dimensioning:

- Dimensioning rule properties (page 942)

- Dimensioning properties in drawings (Integrated dimensioning) (page 959)

Click the following link to find out more about dimensioning properties in GA drawings:

- General arrangement drawing dimensioning properties (page 970)

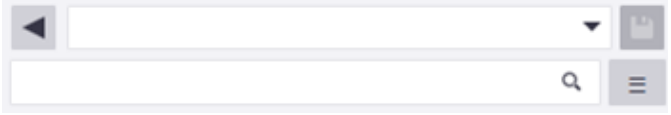
Dimension properties

Use dimension properties to view and modify the appearance, placement, and contents of the dimensions.

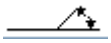
To open dimension properties:

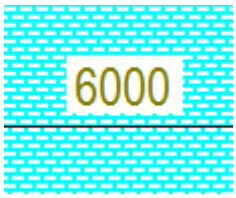
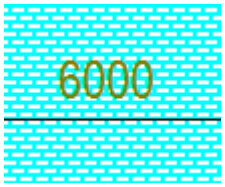
- Click a dimension in an open drawing. If the property pane is not open, double-click the dimension. The properties specific to the selected dimension are displayed.
- In an open drawing, go to the **Dimensioning** tab, hold down **Shift** and click one of the dimensioning commands. The center of gravity dimension properties (page 271) are displayed in a dialog box of their own.
- Select one of the dimension types from the drawing property pane object type list. There are five sets of dimension properties available: **Straight dimension**, **Curved dimension**, **Angle dimension**, **Radial dimension**, and **Rebar dimension mark**.
- In an open drawing, go to **Quick Launch**, start typing `dimension properties`, and select **Dimension properties** from the list. The dimension properties dialog box is displayed.
- To open the **Dimension Properties** dialog box for general arrangement drawings before creating a drawing, go to the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing** , and then click **Dimension...**.
- To open the **Dimension Properties** dialog box for general arrangement drawings when the drawing is open, double-click the drawing background and select **Dimension...**.

All of the dimension settings are described below. Note that all of the settings are not available for all dimension types.

Setting	Description
Save, load and search properties in property pane	 • To load predefined properties, select the properties from the list.

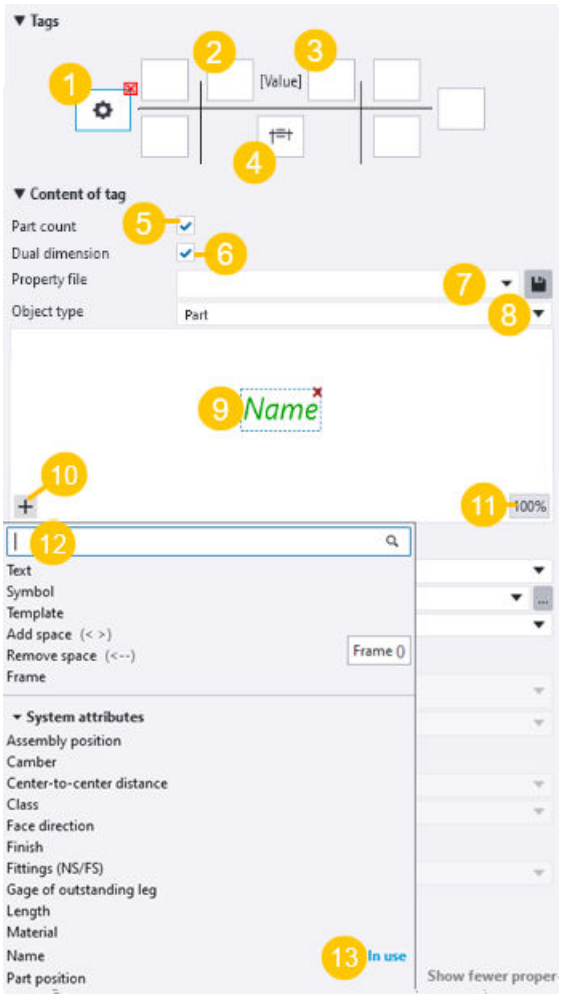
Setting	Description
	• To save the modified properties, click . You can also save the properties with another name. • To search for a property, start typing the property name.
Rebar (available for rebar dimensions only)	
Dimension to	Specifies whether dimensions are created between **All rebars** or between the **Start and end rebars** only.
Part edge	Add closing dimensions to the edge of the part in rebar group dimensions. This setting is only available if **Dimension to** is set to **All rebars**.
Subgrouping	Specifies whether subgroups are created for different rebar spacings within a rebar object so that dimension marks can display the subgroup properties. **Subgrouping** is only available when **Dimension to** is set to **All rebars**.
Extension line to visible rebar	Display extension lines to the visible rebars.
Tapered dimension type	Specifies whether skewed dimensions have **Straight** or **Skewed** representation.
Curved dimension type	Specifies whether curved dimensions have **Straight** or **Curved** representation.
Dimension line	
Straight dimension	Set the dimension type for straight dimensions. **Relative**: Add point to point dimensions. **Absolute**: Dimension from a common start point. **Relative and absolute**: Combination of point to point and common start point. **US absolute**: Dimension from a common start point, which includes a running dimension mark (RD). **US absolute 2**: Similar to **US absolute**, but it changes short dimensions to relative.

Setting	Description
	Absolute with short relatives: Similar to **Absolute**, but it changes short dimensions to relative. Also called internal absolute. This option may show both dimensions, but it does not show relative dimensions when the dimensions are long. This option shows the absolute dimensions inside the dimension lines. **Absolute with all relatives above**: Similar to **Relative and absolute**, but it places the relative dimensions above the absolute. **Elevation:** Create an elevation dimension at a picked point. This type is only available in the dimension properties of manual dimensions in drawing mode.
In x direction	This setting is available in the GA drawing dimension properties only. Override the straight setting for horizontal dimensions. If you use the blank option, Tekla Structures uses the **Straight** option settings. The x direction usually means dimensions parallel to the drawing x axis.
Curved dimension	Available for curved dimensions. Define whether to use angle units or distance units for curved dimensions.
Angle dimension	Available for angle dimensions. Define the appearance of angle dimensions.  **Degrees on side** **Degrees at angle vertex** **Triangle** **Triangle with degrees**
Triangle base length	Available for angle dimensions. Define the base length of a triangle.
Line color	Define the color of the dimension line.

Setting	Description
Arrow	Select the dimension line arrow type. You can also create arrow types of your own, see Customize dimension line arrows (page 275). In the GA drawing dimension properties, you can also separately select the dimension arrow type for **US Absolute dimensions** and **Level dimensions**.
Arrow height	Set the height for the arrowhead.
Arrow length	Set the length for the arrowhead.
Short extension line	Define whether Tekla Structures should create extension lines all of the same length, or use the short extension line automatically if the dimension line falls on a grid line. If you want to exaggerate (page 783) dimensions, you need to set this option to **No**.
Dimension value	
Font color	Select the color for the dimension mark text.
Font, **Font height**	Select the font to be used in the dimension mark. You can also change the font height.
Place on line (**Place** in GA drawing dimension properties)	Define how the dimension mark is placed relative to the dimension line: **Above dimension line** or **On dimension line**.
Background (**Background mask** in GA drawing dimension properties)	Select **Opaque** to hide the area of the drawing that is covered by the dimension mark.  Select **Transparent** to show the drawing content in the dimension mark background so that drawing linework is visible. 

Setting	Description
Frame	Define a frame for the dimension mark or select **None**. **Underline** **Rectangle** **Rounded rectangle** **Sharpened rectangle**
Units	Define the units used in dimensioning. **automatic** uses the units defined in the model. **mm** (millimeters) **cm** (centimeters) **m** (meters) **ft - in** (feet and inches) Inches are converted into integer feet, and the remaining inches are shown in inches. **cm / m** (centimeters and meters) Dimensions under 100 cm are shown in centimeters, and dimensions above 100 cm are shown in meters. Millimeters are shown as superscript. **in (decimal)** (inches, decimals used) **ft (decimal)** (feet, decimals used)
Precision	Define dimension precision: rounding, imperial units. The following are for defining precision with rounding. For example, with precision 0.33 the actual dimension 50.40 is shown as 50.33. **0.00** **0.50** **0.33** **0.25** The following are for imperial units: **1/8**

Setting	Description
	1/16 **1/32** The following are for defining precision without rounding: **1/10** **1/100** **1/1000**
Format	Define the dimension format: the number of decimals and their appearance. **###** **###[.#]** **###.#** **###[.##]** **###.##** **###[.###]** **###.###** **### #/#** The following is only available for straight dimensions: **## #/##.## #**
Digit grouping (**Use grouping** in GA drawing dimension properties)	Select **Yes** to group long dimension values.
Combine equal dimensions	Combine equal dimensions. The options are **Off**, **3*60**, and **3*60=180**. The accuracy of combining equal dimensions is 0.1. This setting is not available in GA drawing dimension properties.
Minimum number to combine	Define the minimum number of dimensions to combine. This setting is not available in GA drawing dimension properties.
Show numeric value	Select the check box to show the numeric value in dimensions. This setting is not available in GA drawing dimension properties.

Setting	Description
Tags and **Content of tag** (not available in GA drawing dimension properties, or for some dimension types)	
Dimension tag editor	The **Content of tag** section is displayed when you click a tag container.  **(1)** Available tag containers around the dimension line. Click a tag container to select it. Use the red cross button in the upper-right corner of the container to remove the contents. You can enter text or select the desired elements from the element list. In tags, you cannot use template attributes such as `MODEL_TOTAL` that refer to the whole model. Tags only check the information from the object in the drawing, and not from the whole model. **(2)** Prefix tag **(3)** Postfix tag

Setting	Description
	(4) Dual dimension tag **(5) Part count** check box: When you click a tag container, the **Part count** check box becomes available. When you select the check box, the selected tag shows the part count. **(6) Dual dimension** check box: When you click the dual dimension tag container **(4)**, the **Dual dimension** check box becomes available. When you select the check box, dual dimensions (page 260) will be applied. **(7) Property file**: Load a set of predefined dimension tag contents. You can also save the tag contents. **(8) Object type**: Define the type of the building object that you are dimensioning. The options are **Part**, **Bolt**, **Rebar**, **Surface treatment**, and **Pour object**. This setting is not available when you have selected a dimension in an open drawing. **(9)** Tag editor for adding content in dimension tags. You can drag elements to another location inside one tag container. Click the red ✖ delete button in the upper-right corner to delete the element. **(10)** Element list. Use the plus button + to open the element list, and then select the elements to add in the current tag container. You can also search for elements. **(11)** The % button shows the current zoom level in the preview. You can zoom in or out by scrolling the middle mouse button. Click the button to zoom to best fit. **(12)** Search box for searching elements. Enter the element that you want to locate. **(13)** If you have already added an element in a tag, the element is marked with text **In use** in the element list.
Font color, **Font**, **Font height**	Define the font, font color and font height for the whole tag or for the selected tag elements.
Frame, **Frame color**	Define a frame and frame color for the whole tag or for a single element.

Setting	Description
Units, **Format**	Set the unit and format for length elements. Unit settings can be modified only when a length element is selected.
Rotation	Set the dimension tag rotation. The options are: **Parallel to dimension line** does not rotate the tag. This is the default value. **Perpendicular to dimension line** rotates the tag. This setting is not available for the prefix, postfix, or dual dimension tag.
Excluded tags	Select a drawing view **Filter** that removes the defined content from tags.
Curved dimension tag type	Available for curved dimensions only. Select a tag type for a curved rebar dimension line. The tag type controls how the tags are aligned to the dimension. The options are: **Horizontal or vertical** **Horizontal** **Vertical** **Horizontal or vertical with extension line** **Follow curve** **Follow along curve**
Exaggeration (not available in GA drawing dimension properties, or for some dimension types)	
Exaggerate short dimensions	**Specified** exaggerates short dimensions (page 783) to make them easier to read.
Direction	Select the direction of the exaggeration: **Left / Down**, **Right / Up** or **Both**.
Origin	Set the **Origin**: : The origin is near the dimension line.

Setting	Description
	: The origin near the measured object.
Width	Set the desired exaggeration width.
Position	Set the desired position for the exaggerated dimension.
Height	Set the desired height for the exaggerated dimension.
Plate side marks (not available in GA drawing dimension properties, or for some dimension types)	
Side mark type	**Specified** creates plate side marks to plate dimensions (page 260). Select the left and right plate side mark type, and specify the color, size and offset for the mark. The offset means the distance of the mark from the dimension line. **Automatic** is only available in intelligent drawings, that is when the advanced option `XS_INTELLIGENT_DRAWING_ALLOWED` is set to `TRUE`.
Placing	
Dimension line spacing	Define the space between parallel dimension lines. In manually created dimensions, this setting works only if dimension **Placing** is set to **Free**.
Short dimensions	Define the text location of short dimensions: : text is inside the dimension lines : text is outside the dimension line
Placing method	• **Free** lets Tekla Structures decide the location and direction of the dimension based on the direction setting. • **Fixed** allows you to place the dimension at any point. This setting is not available in GA drawing dimension properties.

Setting	Description
Negative direction, **Positive direction**	Define the side of the dimensioned object on which Tekla Structures places the dimensions. These settings are not available in GA drawing dimension properties.
Position	The position can only be defined if you have selected **Free** as the placing method. **Distance s** is the empty margin that you want to leave around the dimension. This is called **Search margin** in GA drawing dimension properties. **Distance d min** is the minimum distance of the dimension from the part. This is called **Minimum distance** in GA drawing dimension properties.

Dimensioning rule properties

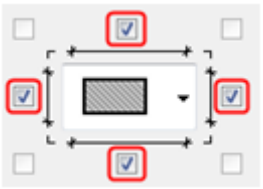
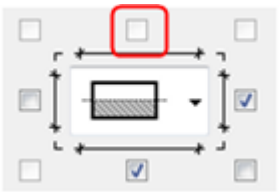
The following table describes the settings and options available in the **Dimensioning rule properties** dialog box. To open this dialog box, select **View creation** in a drawing properties dialog box, then select a view row, and click **View properties** --> **Dimensioning** --> **Edit rule** .

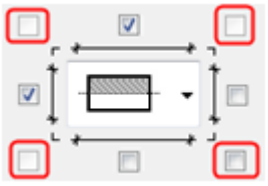
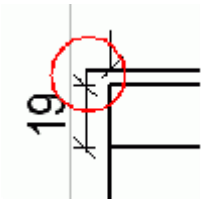
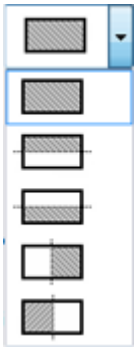
If you have selected **Integrated dimensions** as the dimensioning type, the Dimensioning properties (page 959) dialog box will be displayed instead.

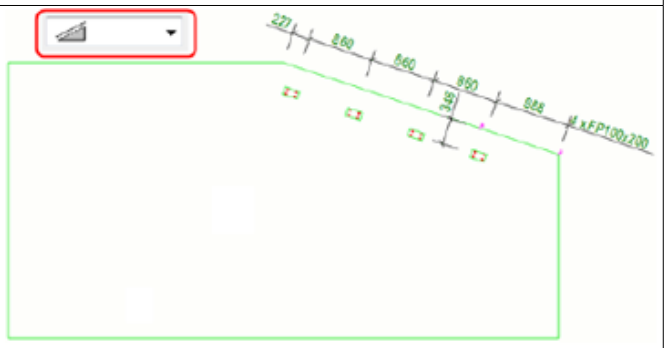
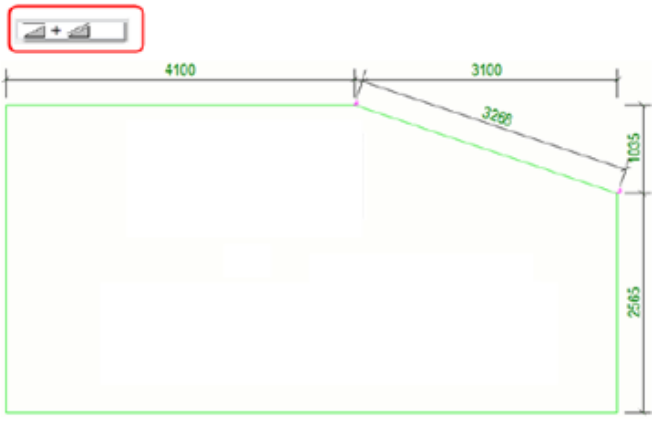
Dimensioning rule properties

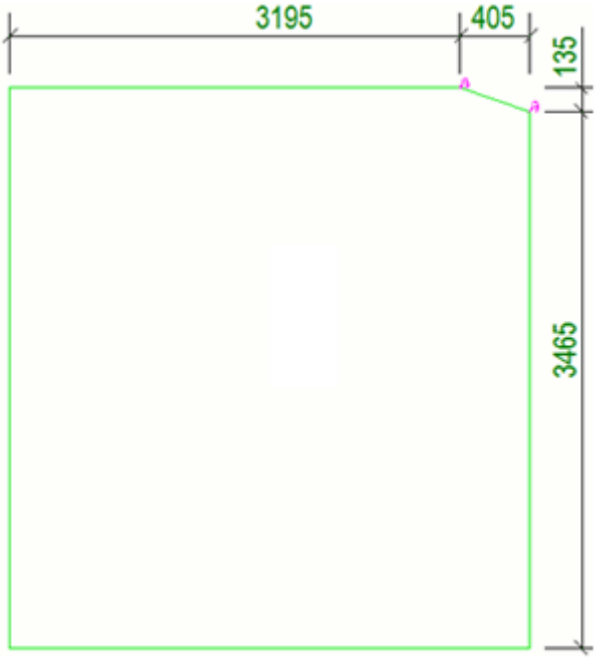
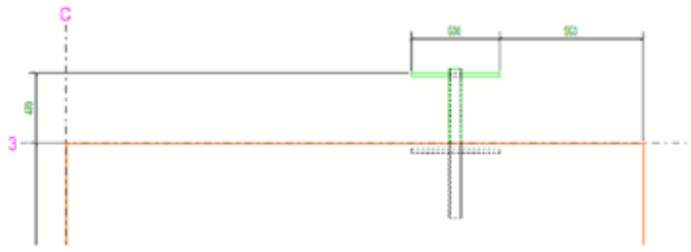
Setting	Description
What is dimensioned	Select the dimensioning type: • **Overall dimensions** creates dimensions for the bounding box of the objects you select in the **Measure from** list. • **Edge shape** creates dimensions for the edge of the object selected in the **Measure from** list. By selecting the **Visible faces** option, dimensions will be created only to faces that

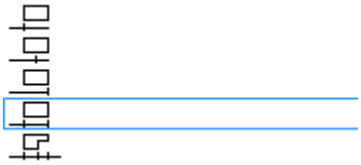
Setting	Description
	are visible in the drawing view. The other option **All faces** dimensions all faces. **All faces** is the default value, and it will be used if the dimension settings file does not contain any value for the new setting. For sandwich walls, the default setting **Cast unit / Assembly** may not give the desired result with **Edge shape**. Then you can dimension the internal and external layers separately according to the part name. • **Secondary parts** creates dimensions for the secondary parts of a cast unit or an assembly. • **Holes** creates dimensions for the holes of the objects selected in the **Measure from** list. The hole dimensions are combined according to the **Combine on one line** setting. **NOTE:** The **Holes** dimensioning type does not dimension bolts, you need to use the **Integrated dimensions** to get the bolt dimensions. • **Recesses** creates dimensions for the recesses of the objects selected in the **Measure from** list. The hole dimensions are combined according to the **Combine on one line** setting. • **Distance to grid** creates dimensions from the grid line to the bounding box of the object selected in the **Measure from** list. This setting works only when the grid lines are visible. • **Filter** can be used to dimension anything that can be filtered. It is often used when dimensioning embeds. For example, you can dimension bolt locations after you have created a filter for bolts. When you select **Filter** from the **What is dimensioned** list: • A list is displayed where you can select the filter. The filter is a drawing view filter and you need to create it beforehand to be able to use it here. • The **Only objects on the selected sides** check box is displayed allowing you to only dimension objects that are on a certain side of the main part, for example, **Front** side. By

Setting	Description
	creating separate rules for each side you can have front and back side objects dimensioned on separate dimension lines. The options are **Front**, **Top**, **Start**, **Back**, **Below**, and **End**. • **Neighbor parts** dimensions the neighbor parts. Neighbor parts are typically connecting parts, for example, columns in a beam drawing. Note that the neighbor part dimensioning works best if the view size is extended so that the neighbor parts are fully visible in the view. See an example of the neighbor part dimensioning in the "Examples of dimensions" section below.
Dimension line locations and linking	Control on which sides of the assembly/cast unit the dimensions will be created.  • Dimension line locations are rotated together with the drawing view if the view is manually rotated. • When selecting both sides, top and bottom or left and right, the objects will be dimensioned on the side that is closest to the object. • When only a part of the assembly/cast unit is selected for the rule, one of the check boxes will be dimmed and you cannot select it.  • When you have selected one vertical and one horizontal dimension line, the linking check boxes in the corners are activated, and you can

Setting	Description
	link the perpendicular dimension lines. The check boxes are dimmed by default.   • You can select objects for dimensioning from the whole assembly/cast unit or only from one half of the assembly/cast unit. If you do this, objects in the other half will be ignored in dimension creation. The default value is the whole assembly/cast unit. 
Place dimensions inside	When you select this setting, the dimensions can be placed inside a cast unit, assembly or part. This is only visible for the dimensioning type **Filter dimensions**.
Orientation	Orients the dimensions along the sloped edge of a part. Another possibility is to create horizontal or vertical dimensions. • This option is visible only when the dimensioning type **Filter** or **Edge shape** is selected. • When **Filter** is selected, there are two settings available for orientation. The first setting places the dimensions along the sloped edge:

Setting	Description
	 The second setting places the dimensions horizontally and vertically: • When **Edge shape** is selected, there are three settings available for orientation. First two settings work in the same way as the first two **Filter** settings. The first setting is the default value. The third one creates both dimensions: 
Minimum length for skew section	Defines the minimum length of skewed sections when dimensions are still created for skew sections. The default value is 300 mm. For

Setting	Description
	example, when you use the value 500 mm and the skew section is shorter than 500 mm, the dimensions are not created along the skew section, but horizontally and vertically instead. 
Direction	Defines where the origin points for the dimensions are: • The default values are left for the **Horizontal** and bottom for the **Vertical** dimension. • The setting marked with symbol chooses the nearest edge as the dimension origin, for every dimension separately. In the example below, the object is closer to the right edge, and so the horizontal dimension starts from there. 
Vertical = Horizontal	Often the settings are similar for both directions, and when this option is used, only vertical settings need to be set.

Setting	Description
Close lines	Defines whether the dimension lines extend to the other end of the cast unit or assembly or not . The dimension lines extend to the other end by default.
Dimension to	Defines which points of the selected objects are dimensioned: • **Both ends** is the default.  • If you select the center point setting and the object is a custom part (=embed), Tekla Structures will use the component insertion point, which is not always in the center. For other objects, the actual center point will be used.
Round holes	Defines whether the round holes will be dimensioned by center point or diameter : • This option is visible only when **Holes** or **Recesses** is selected. • The diameter is the default value. • Currently, round holes are identified only by cut part profile (prefix D). If you create the round hole using round chamfers, for example, the hole will not be dimensioned.
Dimension properties	Select and apply dimensions settings defined in a dimension properties (page 932) file that has been saved earlier. The default value is **standard**. This file defines the used dimension properties, such as dimension type, units, precision, format, appearance, marks, and included dimension tags. If you select **Same on all sides**, the settings in the same dimension properties file are used on all sides. If you deselect **Same on all sides**, you can select and apply different dimension line properties for **Top**, **Bottom**, **Left** and **Right** side.

Setting	Description

Measure from	Defines the objects that will be used as the origin point for the dimensions. The available settings are:
	• **Cast unit/Assembly:** This is the default value. If you select this setting, you have three more settings available:
	  • **Only concrete/steel parts**: For a cast unit, only concrete parts are used, and for steel only steel parts.
	  • **All parts**
	  • **All parts and rebars**
	• **Main part**: This setting uses cast unit or assembly main part.
	• **Part name:** If you select **Part name**, you can define the part name.
	• **Filter**: If you select **Filter**, you can use a predefined filter for selecting the objects that you want to use as dimension origin points.
	• **Current part**: Select **Current part** when you are dimensioning a single part.
	• **Bounding box**: Uses the bounding box of an object as the origin point for the dimensions. This option is only available in filter, hole, recess and secondary part dimensioning.
	• **Nearest edge**: Uses the nearest edge an object as the origin point for the dimensions. This option is only available in filter, hole, recess and secondary part dimensioning.
	• **And grid**: Adds grid lines to the dimensions. This option is only available for the **Overall dimensions**, **Shape dimensions**, **Filter dimensions**, **Secondary parts dimensions**, **Hole dimensions**, and **Recess dimensions** dimensioning types.

Combine on one line	Creates a rule based on a filter, for example, for embeds (EB_*), and then groups the embeds according to the main part name, so that embeds with different names will get their own dimension lines. The filtered object can be a part, a reinforcing bar or an assembly. The available settings are: • **All objects** (default)

Setting	Description
	• **By name** • **By position number** • **No** When **Holes** or **Recesses** is selected, the **Combine on one line** option changes to display suitable settings for holes or recesses. The available settings are: • **All holes** (default) • **Any holes of the same size** • **By cut part name** • **No**
Combine only objects that have the same **X or Y coordinate** **Z coordinate**	Combines only the dimensions of objects that are on the same horizontal or vertical line, or dimensions of objects that have the same Z coordinate. By default, these options are not selected.
Tolerance	The tolerance is the maximum distance between objects when Tekla Structures still considers the objects to be on the same line. 50 mm is the default value.
Preferred combining direction	Sets the preferred direction for combining dimensions if an object can be combined both in horizontal and in vertical direction. X is the default value.
Do not create dimensions shorter than	Defines the minimum length of the dimensions for Tekla Structures to create them. 0 is the default value, which means that all dimensions are created.
Do not dimension holes smaller than	Defines the minimum diameter for the holes for Tekla Structures to create the dimensions. With this option you can prevent the creation of dimensions for small holes. The distance is the shortest dimension of a hole. If any dimension of the hole is larger than the given value, the hole will be dimensioned in all directions. For example, with value 40, a rectangular hole of 80*30 will get both dimensions 80 and 30. Default value is 0, which means that all dimensions are created.
Component objects	Defines how component objects are dimensioned: • **By reference points** (default) places the dimensioning point to the first insertion point of the component. Only one dimension will be

Setting	Description
	created for each component regardless of the number of parts inside the components. • **As secondary objects** creates separate dimensions for each part inside the component.
Items	Defines how items are dimensioned: **By reference point**: Places the dimensioning point to the first insertion point of the item. **As secondary objects**: Creates item dimensions similarly as parts are dimensioned. This is the default value.

Dimension rule properties for spiral beam dimensioning

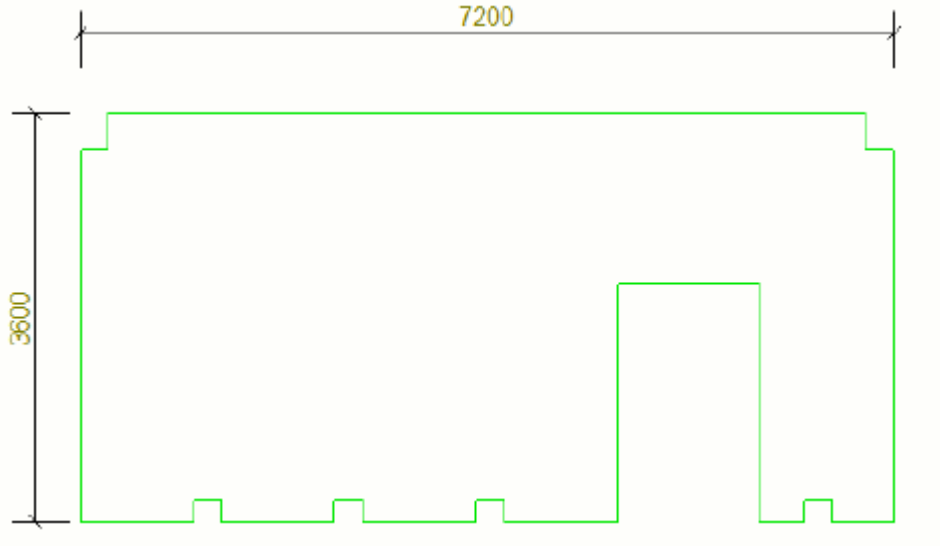
When you select dimensioning type **Spiral beam dimensions**, and click **Edit rule**, a different **Dimensioning rule properties** dialog box is displayed.

Setting	Description
Straight dimensions **Angle dimensions** **Angle and radius dimensions**	Select predefined dimension properties. If none of the available properties suit your needs, open a drawing, click **Drawing** --> **Properties** --> **Dimension**, and edit and save the needed dimension properties so that they are available for selecting in the spiral beam **Dimensioning rule properties** dialog box for the three dimension types.

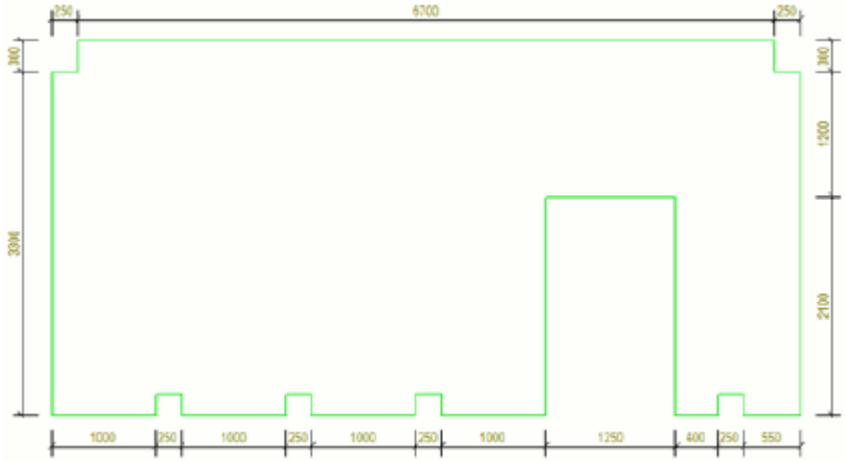
Examples of dimensions

See below for examples of dimensions created with different settings in the **Dimensioning rule properties** dialog box.

Overall dimensions

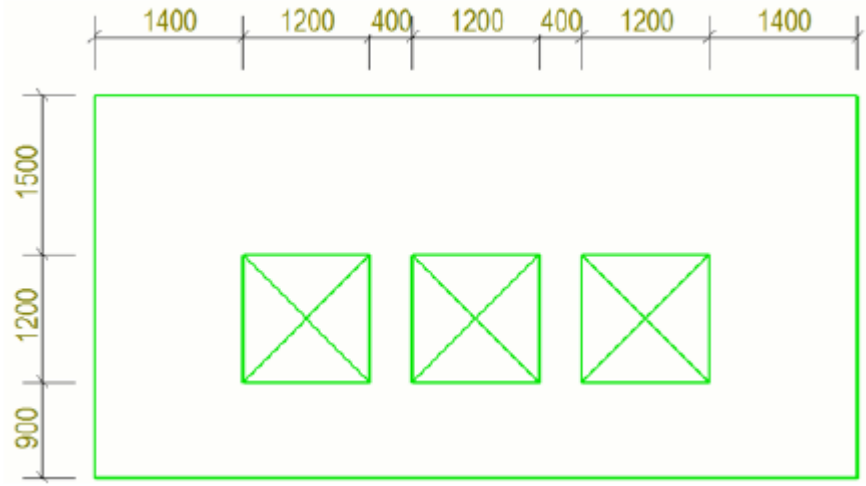


Edge shape

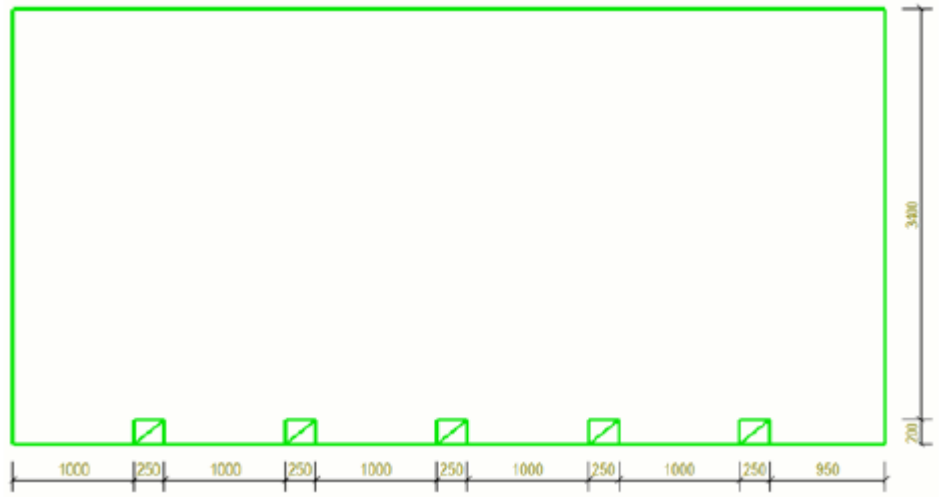


Secondary parts

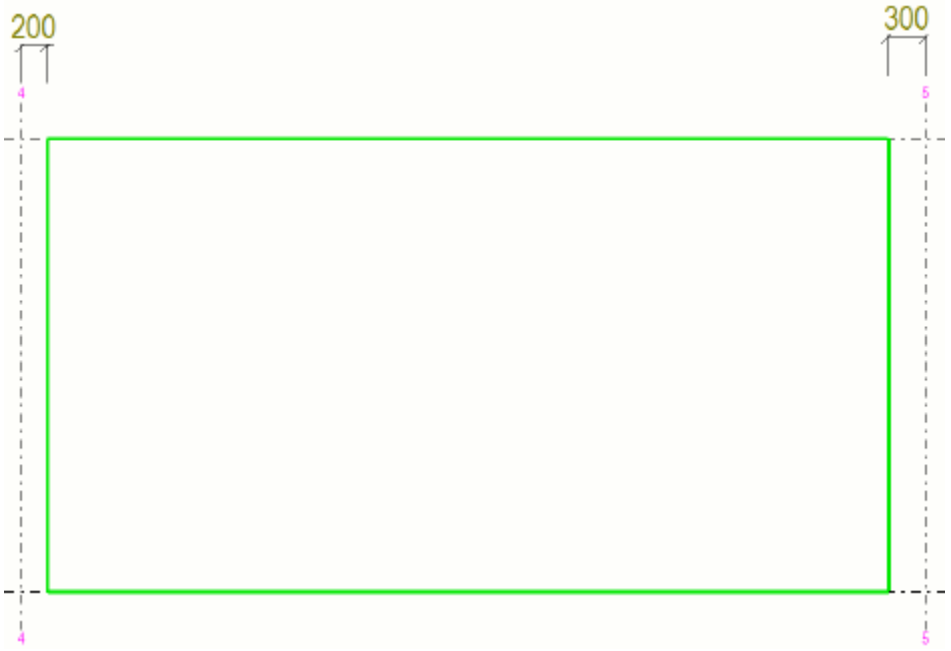
Holes



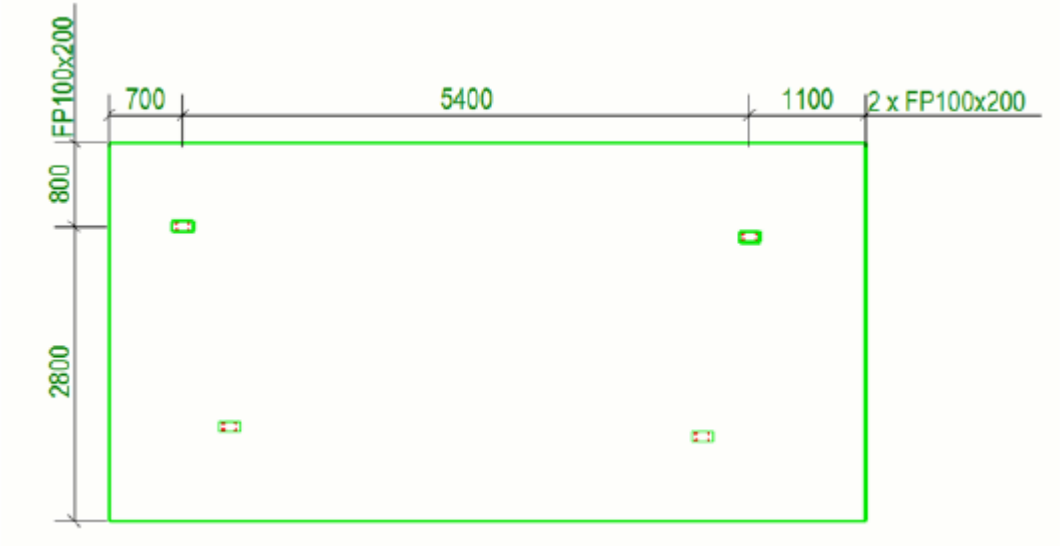
Recesses



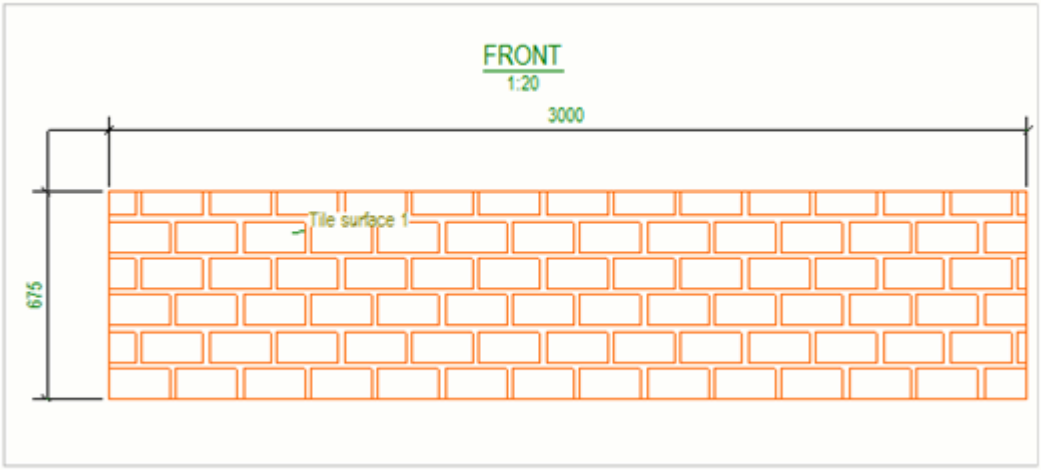
Distance to grid



Filter: Embeds

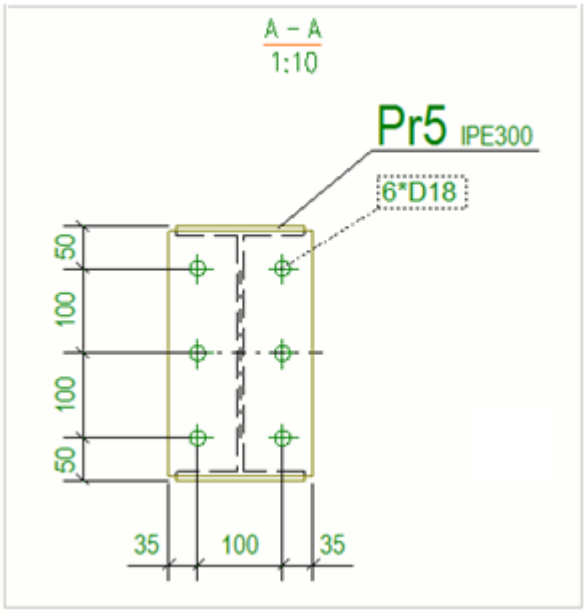


Filter: Surface treatment



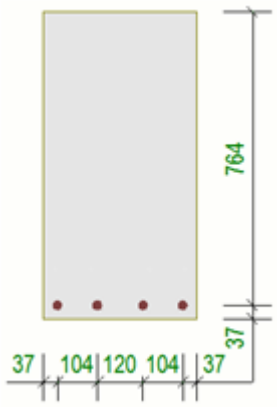
Filter: Bolts

To get the location of every bolt in the bolt group dimensioned, set **Dimension to** to midpoint :

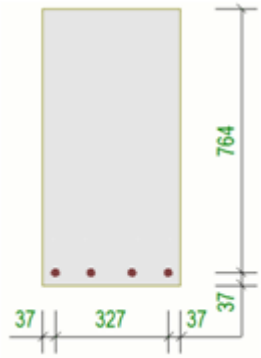


Filter: Reinforcing bars and strands

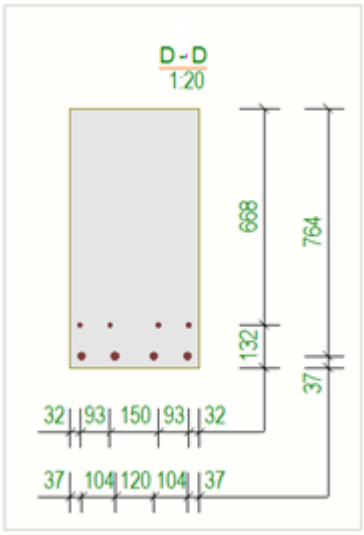
Use the midpoint option to dimension each bar in a group:



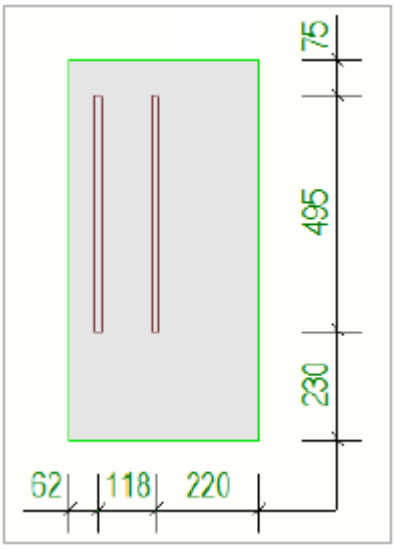
Start point and end point option will dimension the first and the last reinforcing bar in a group:



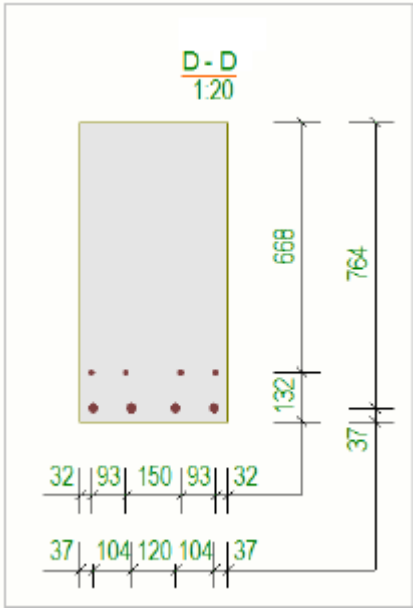
If you want to have separate dimension lines for reinforcing bars that are not of the same size, set the **Combine on one line** to **By position number**:



To make the rule creation easier, the midpoint setting of the **Dimension to** option works so that it will create dimensions to start point and end point of the reinforcing bars that are parallel to the view plane:



If you want to get different size of reinforcing bars on separate dimension lines, set **Combine on one line** to **By position number**:

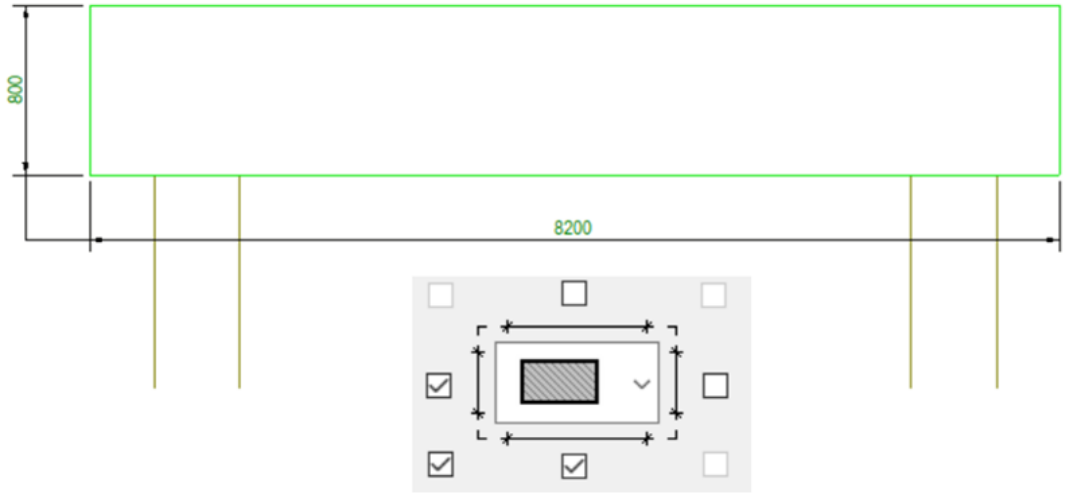


Neighbor parts

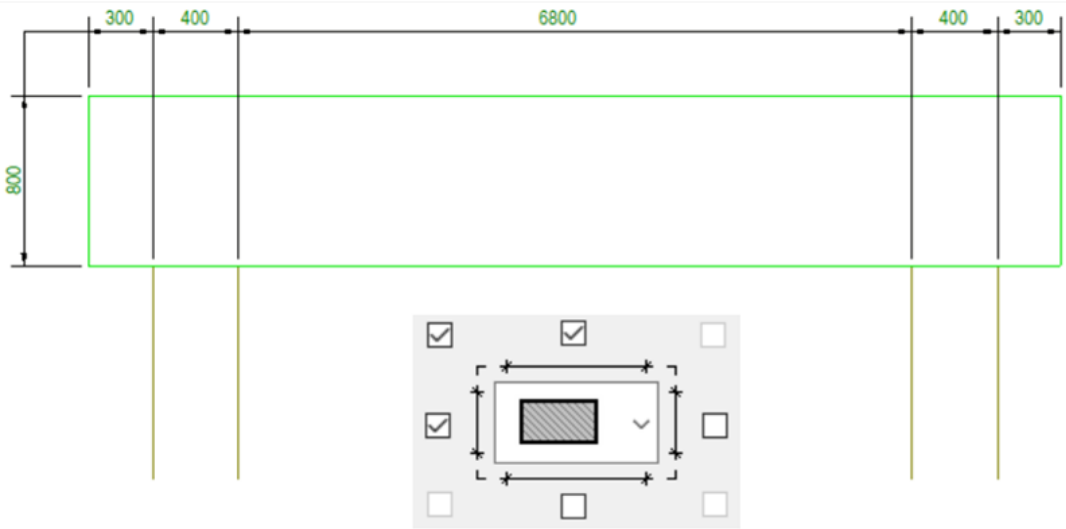
The dimensioning process uses the whole model object to locate the edges and corners where the dimensioning points should be placed. The dimensioning process is not aware of the view boundaries, and when the object extends outside the view boundaries, some dimensioning points might be placed outside the view and automatically removed, because objects outside the view boundaries are not allowed.

In the example below, the columns extend beneath the view boundaries and, with the settings shown in the image, the horizontal dimensioning points of the columns are placed on the bottom level of the column and thus removed.

For the vertical dimensioning, the top level of column is dimensioned, but the bottom level dimension point is again outside the view boundaries and thus removed.



After changing the dimensioning settings so that you place the horizontal dimension line above, the dimensioning points are inside of view boundaries and not removed. With vertical dimension line, the situation is still the same: the bottom level point is outside the view boundaries and thus removed.



Add automatic view-level dimensions (page 730)

Create a drawing view filter for view-level dimensioning (page 744)

Dimension and dimensioning properties in drawings (page 931)

Show spiral beams in drawings (page 443)

Dimensioning properties in drawings (Integrated dimensioning)

Use the **General** tab in the **Dimension Properties** dialog box to view and modify the general settings of the dimensions. This dialog box is displayed if you use **Integrated dimensions** dimensioning type in dimensioning.

General tab

Option	Description
Dimensioning type	**Standard** is used for almost all dimensioning. **Truss** meets the specific requirements needed to dimension truss drawings. It dimensions the position and length of the diagonals. The dimensioning is done only if the diagonals are secondary parts that are welded to upper and lower chords, which are main parts and not welded to any parts. If the truss welding is done some other way, standard dimensioning is used.
Minimize	**Yes** minimizes the number of views that Tekla Structures creates. Also check the settings in the drawing **View Properties** dialog box.
Combine dimensions	Combines several single dimensions into one dimension line. In **Options**, select the combination level. The larger the number, the more Tekla Structures combines dimensions. Option **4.5** uses a combination of option **5** for main part and option **4** for secondary parts. The **Distance** means the distance within which Tekla Structures combines internal dimensions. If the distance between two details is less than the defined **Min distance**, Tekla Structures combines the dimensions.

Option	Description
Close dimensions	**Close dimensions** complete dimension lines to include the entire part. **No** does not close dimensions **In X** only closes dimensions in the x direction and leaves others open **All** closes all dimensions This setting is not relevant to profile shape dimensions.
Close dimensions: **Short dimensions**	**Yes** closes short dimensions. With **No**, the open dimension is the middle one rather than the short end dimension. When you leave short dimensions open, Tekla Structures leaves out the longer dimension line in dimension lines that contain two dimensions. If dimension lines contain three dimensions, Tekla Structures leaves out the middle one. This option does not affect dimension lines with more than three dimensions.
Placing: **Forward offset**	**Forward offset** defines the distance Tekla Structures uses to search for the base point of a dimension. If Tekla Structures does not find a base point (corner) within the **Forward offset** search distance, it uses an edge point. The **Centered bolt** setting affects on how the dimension is displayed.
Necessary internal dimensioning: **Recognizable distance**	**Recognizable distance** defines the limit for dimensioning asymmetry in secondary parts. In some cases, it is important to look at the asymmetrical relationship of parts, so that an asymmetric secondary part is correctly connected to a main part. You can use **Recognizable distance** to have asymmetry reflected in dimensioning. If the asymmetry is smaller than the distance you enter here, Tekla Structures represents it using a dimension.

Option	Description
Part mark on dimension line	**None** creates no part mark on the dimension line. **On assembly overall dimension** creates a part mark on the assembly overall dimension line. **Between extreme bolts** creates a part mark on the dimension line between the outermost bolts.
Main dimension position	Defines the side on which Tekla Structures places the **Assembly overall dimensions**, **Main part work points**, and **Knock off dimensions**. **Auto** treats the main dimensions in the same way as other dimensions. **Above** places the main dimensions above the part (or to the left for vertical parts). **Below** places the main dimensions below the part. **Above** places the skew position dimensions of the main part below the part and **Below** places them above.
Grid dimensions	Creates grid dimensions. The values are: **None**, **Individual spans**, **Overall** or **Individual spans and overall**.
Grid dimension position	Sets the grid dimension position. The values are: **Main view - above** **Main view - below** **Top view - above** **Top view - below** **Bottom view - above** **Bottom view - below** **Back view - above** **Back view - below** **All views - above** **All views - below**.
Dimension properties	

Option	Description
Straight dimensions	Sets the dimension type for straight dimensions using the settings in the properties file you select.
In X direction	Otherwise same settings as in straight dimensions but overrides the straight setting for horizontal dimensions. If you use the blank option, Tekla Structures uses the **Straight dimensions** option settings. The x direction usually means dimensions parallel to the drawing x axis.
Arrow shape: **US Absolute dimensions**	Controls the type of mark used with the dimension line in US Absolute dimensions. US Absolute dimensions are dimension types **US Absolute** and **US Absolute 2** that can be defined in dimension properties (page 932). If the advanced option XS_USE_USABSOLUTE_ARROW_TYPE_FOR_ABSOLUTE_DIMENSIONS is set to `TRUE`, the arrow shape is overridden for other types of absolute dimensions as well.
Arrow shape: **Level dimensions**	Controls the type of mark used with the dimension line in level dimensions.
Angle and radius dimensions	Sets the dimension type for angle dimensions using the settings in the properties file you select.
Check dimensions	Sets the dimension type for check dimensions using the settings in the properties file you select.

Position dimensions tab

Option	Description
Position bolts to **Position parts to**	Controls from where Tekla Structures creates the part/bolt position dimensions. **None** creates no position dimensions.

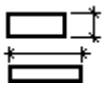
Option	Description
	Main part creates the dimensions from the main part reference line. **Working points** creates the dimensions between the work points, such as the intersections of main and neighbor part reference lines.
Embedded objects	Creates position dimensions for locating embedded objects in cast unit drawings. The embedded objects are custom components attached to the cast unit. **As secondary objects** dimensions embedded objects in cast unit drawings the same way as secondary parts. **By reference point** dimensions embedded objects to their reference point, which is the origin of the custom component.
Secondary part	Creates dimensions to bolt holes or edges of the secondary part. **None** creates no position dimensions for secondary parts. **By bolt** dimensions bolt hole locations in the secondary parts. **By part** dimensions edges of secondary parts. **By both** dimensions bolt hole locations and edges of the secondary part.
Secondary part dimension direction	Aligns dimensions with the main or neighboring part. Only for skewed clip angles or shear plates.
Position from	Sets the start point for running dimensions. Only for skewed clip angles or shear plates bolted to a neighboring part.
Reversed direction for running dimensions	**Yes** changes the direction of running dimensions. With this option, you can set the zero point to the end of a member rather than to the start.

Option	Description
Main part bolt position	**On** creates dimensions to the bolt hole locations in the main part.
Main part skew position	**Yes** creates horizontal and vertical check dimensions representing the skew position of a brace. Created between the work points of the main part. Skew check dimensions are located in the front view. Their location depends on the **Main dimension position** setting selected on the **General** tab. When the main dimensions are above the part, the skew dimensions are below, and vice versa.
Skew position	Defines how Tekla Structures dimensions skewed secondary part positions. **None** creates no dimensions for skewed secondary part positions. **angle** creates an angle dimension for the secondary part. **Dimensions** creates dimensions for the skewed position of the secondary part. **Both** creates both the angle and the dimensions.
Centered part	Controls the dimensions of centrally-placed parts. These settings have an effect only if position dimensions are created. **Internal** dimensions the centered parts overall size. **Position** dimensions the part to the main part center lines. **None** creates no centered part dimensions.
Centered bolt	Controls the dimensions of centrally-placed bolt groups. **Internal** dimensions the centered bolts spread.

Option	Description
	Position dimensions the bolts to the main part center-lines. **Centered bolt** overrides the **Secondary part bolt internal** option for centered bolts. This only applies to bolts located centrally on the part.
Elevation dimensions	**On** creates elevation dimensions.
Combine equal dimensions	Combines equal dimensions. The choices are **Off**, **3*60** or **3*60=180**. The accuracy of combining equal dimensions is 0.1.
Minimum number to combine	Defines the minimum number of dimensions to combine.

Part dimensions tab

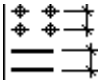
Option	Description
Internal	Creates internal dimensions for secondary parts connected to the main part. **None** creates dimensions for secondary parts. **Necessary** creates only the dimensions that are necessary for assembling the parts. **All** creates all dimensions for the secondary parts.
Main part overall dimensions	**Once** creates one overall dimension for the main part. **All** creates overall dimensions for main parts in all views. **None** creates no overall dimensions for the main part. The **Assembly part overall dimensions** settings have some effect on these options.

Option	Description
Assembly part overall dimensions	**Length only** creates overall dimensions for an entire assembly or cast unit in the x direction only. **All dimensions** creates overall dimensions for an assembly or cast unit in all directions. **Off** creates no overall dimensions for an assembly or a cast unit.
Main part work points	**On** creates a check dimension between the outermost work points.
Main part shape (**Shape dimensions**)	**On** creates dimensions to show the shape of a main part. By default, Tekla Structures automatically draws shape dimensions on both ends of a beam, even if the ends are symmetrical.
Main part radius dimensions (**Radius dimensions**)	**On** creates radius dimensions for curved chamfers and round holes in the main part. This option is available only if you have set **Main part shape** dimensions to **On**. Note that this option does not create radius dimensions for curved beams or polybeams with chamfers.
Bevel dimensions	**On** creates linear dimensions of a bevel.
Bevel angle	Creates an angle dimension and defines which side of the bevel to dimension. The choices are **None**, **Angle of cut** and **Angle of beam**.
Knock off dimensions	**On** creates check dimensions from the edge of the main part to the work point.
Preferred dim side	Sets the preferred view (front or side) for part dimensions. 
From the nearest floor level to part	**On** creates dimensions indicating the distance from the closest floor level to the bottom and/or top of the parts.

Option	Description
From grid to part centerline	**On** creates dimensions showing the offset of a part from the grid to the part center line.
From grid to part ends	**On** creates dimensions showing the offset of a part from the grid to the near or far end of the part.

Bolt dimensions tab

Option	Description
Main part bolt internal dimensions	Creates internal dimensions for bolt groups in the main part. **None** creates none of the internal bolt dimensions. **Internal** creates bolt group internal dimensions (distances between bolts). **All** creates edge distance and bolt group internal dimensions. Edge distance is the dimension from the outermost bolt to the part edge.
Main part bolt internal dimensions: **Skewed bolt group**	Indicates whether the dimensions are parallel to the part or the bolt group. The choices are **No dimensions**, **In part direction**, and **In bolt group direction**.
Secondary part bolt internal dimensions	Creates internal dimensions for bolt groups in the secondary part. The choices are **None**, **Necessary**, **Internal**, and **All**.
Secondary part bolt internal dimensions: **Skewed bolt group**	Aligns the bolt dimensions with the secondary part or bolt group. The choices are **In part direction**, **No dimensions**, and **In bolt group direction**
Distance between extreme bolts: **Extreme bolts**	Creates check dimension between the outermost bolts. The choices are **None**, **Main part**, and **Assembly**.
Distance between extreme bolts: **Extreme bolts to work points**	Creates check dimensions from the outermost bolts to the work points. **Yes** creates the check dimensions.

Option	Description
Preferred dim side	Sets the preferred view (front or side) for bolt dimensions. 
Combine bolt dimensions	Sets the format of the combined bolt group internal dimensions. You can combine bolt group internal dimensions and display them in the format **3*60** or **3*60=180**, or have single dimensions.
Minimum number to combine	Defines the minimum number of dimensions to combine.

Dimension grouping tab

Option	Description
Activate dimension grouping	Selects the objects for grouping.
Parts	Groups according to parts.
Bolts	Groups according to bolts.
Components	Groups according to components.
Cuts/Shapes	Groups according to cuts or shapes.
Automatic tagging	Defines how to display information in a dimension line.
Display tags	Displays tags.
Include part count in the tag	Includes part count in the tag.
Do not display marks for the grouped items	Does not display part marks for grouped items.
Available elements	Elements available to define identical conditions.
Add >	Adds items to **Selected elements** list.
Remove	Removes items from **Selected elements** list.
Move up	Moves the element higher in the list.
Move down	Moves the element lower in the list.
Update grouping when model changes	**Yes** updates dimension grouping automatically when the model changes.

Sub-assemblies tab

Option	Description
Dimension parts inside sub-assemblies	Defines whether to dimension parts inside sub-assemblies. **Yes** creates internal dimensions for parts inside sub-assemblies. **No** does not create internal dimensions for parts inside sub-assemblies.
Measure sub-assembly position from	Defines the position from which the sub-assembly is measured. **None** does not measure the sub-assembly position. **Bolt** measures the position of the sub-assembly from the bolts. If bolts are not included in the sub-assembly, or if it is not possible to measure the position from bolts, Tekla Structures measures the sub-assembly position from the reference point. **Extrema points** measures the position of the sub-assembly from the bounding box of the sub-assembly. **Reference point** measures the position of the sub-assembly from the reference point.

Reinforcement dimensions tab

Option	Description
Dimensions for reinforcing bar groups	**On** creates dimensions for reinforcing bar groups. This also activates the other selections on this tab.
Dimension mark settings	Sets the mark type, which can be dimension line, dimension mark, tagged dimension mark, or variants of these typical styles. The Default environment contains three predefined settings files: `rebar_dimension_line`,

Option	Description
	`rebar_dimension_mark`, and `rebar_tagged_dimension_mark` Clicking the **...** button opens the dimension properties, and you can view the settings file and also change the settings, if needed.

General arrangement drawing dimensioning properties

Use the **Grid** tab to view and modify settings for grid and overall dimensions, and the **Parts** tab to view and modify settings for part dimensions in general arrangement drawings.

To display GA drawing dimensioning properties:

- On the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing** and click **Dimensioning...**.
- In an open GA drawing, double-click the drawing background, and click **Dimensioning...**

Grid tab

Setting	Description
Grid line dimensions	**On** creates grid dimension lines.
Overall dimension	**On** creates the overall dimensions.
Dimension positioning: **Horizontal**	Position the vertical grid and overall dimension lines to the **Left** or **Right** side of the drawing or on **Both** sides.
Dimension positioning: **Vertical**	Position the horizontal grid and overall dimension lines **Above** or **Below** the drawing or **Both**.

Parts tab

Option	Description
Maximum leader line length: **Outside dimensions**	Control how close the dimension lines are positioned to the parts they are dimensioning. Defines that the outside dimension lines take the maximum leader line length from the grid line.
Maximum leader line length: **Inside dimensions**	Control how close the dimension lines are positioned to the parts they are dimensioning. Defines that the inside

Option	Description
	dimension lines take the maximum leader line length from the part reference point.
Include parts not entirely in view	**On** dimensions the parts partly outside the view. **Off** will not dimension these parts.
Maximum number of outside dimensions	Specify the maximum number of dimension lines allowed outside the grid. When you dimension different objects on different dimension lines, using this setting helps you to create cleaner drawings. Once the desired maximum is reached, Tekla Structures creates the dimensions inside the grid.
Object group dimensioning rules Specify different object groups to be dimensioned on different dimension lines. Use **Add rule** to add object group rules.	
Object group	The object group to be dimensioned.
Positioning	**No dimensions** does not create dimensions for the parts. **Inside grid** creates dimensions next to or near the parts being dimensioned. All part dimensions are placed inside the grid when the parts are inside the grid. The dimension will still be outside, even if you have selected **Inside grid** because of where the parts are in the end bay and the end being dimensioned to is next to the outside. **Outside grid** creates dimensions to parts and positions them outside the grid. **Either** creates dimensions to parts and positions them either inside or outside the grid depending on the part position and the setting of the **Maximum number of outside dimensions** option. You must use the **Either** option if you define the **Maximum number of outside dimensions**, so that Tekla

Option	Description
	Structures can position dimensions inside the grid when the maximum number of dimensions is reached outside the grid.
Horizontal position	**Left side** positions all dimensions to horizontal parts to the left of the grid. **Right side** positions all dimensions to horizontal parts to the right of the grid. **Distributed to both sides** positions all dimensions to horizontal parts to the grid nearest the part they are dimensioning.
Vertical position	**Above** positions all dimensions to vertical parts above the grid. **Below** positions all dimensions to vertical parts below the grid. **Distributed to both sides** positions all dimensions to vertical parts to the grid nearest the part they are dimensioning.

9.7 Mark properties in drawings

In the mark properties of different types of marks you can view and modify the settings affecting the mark content, placing, and appearance. You can adjust the mark properties before creating a drawing and also in an open drawing.

Note that all of the listed settings are not available for all types of marks. Furthermore, the mark properties in the drawing and view level properties are to some extent different from the properties in the property pane, or may be named differently.

For details about adding marks manually, see Add part marks manually in drawings (page 301) and Add reinforcement marks manually in drawings (page 303).

To open mark properties in the property pane or in a dialog box:

- In an open drawing, click a mark. If the property pane is not open, double-click the mark.
- Open the mark properties by selecting a mark type in the drawing property pane object list. To be able to do so, do not select any objects in the drawing.

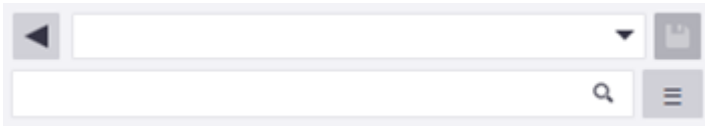
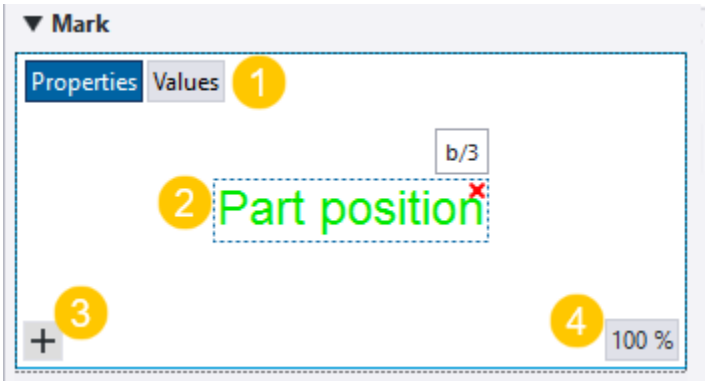
- To open the mark properties in a dialog box, in **Quick Launch**, start typing `mark properties`, and then select the properties for the desired marks from the displayed list.

On the drawing and view level, you can open themark properties in any of the following ways:

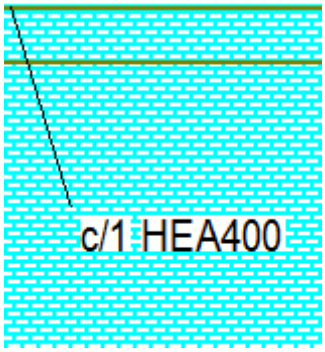
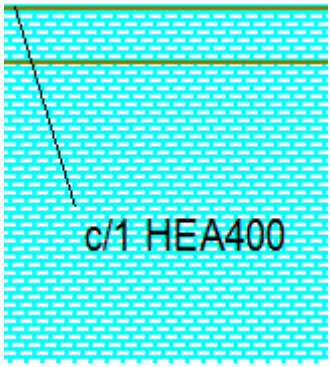
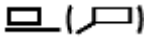
- On the **Drawings & reports** tab, click **Drawing properties**, select the drawing type, and go to the mark properties directly by clicking the button of the respective mark (GA drawings), or go to the drawing view properties and click the respective mark in the options tree (other drawing types).
- In an open drawing, double-click the drawing background, and go to the mark properties directly by clicking the button of the respective mark (GA drawings), or go to the view properties and click the respective mark in the options tree.
- In an open drawing, go to **Drawing** --> **Properties** , select **View** and click the respective mark in the options tree.

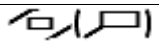
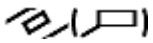
Mark properties in property pane

Most of the properties listed below are also available on the drawing and view level.

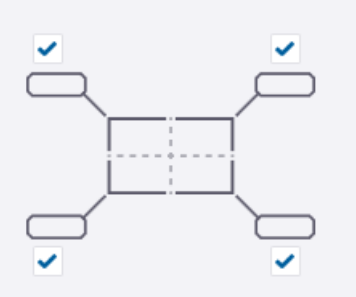
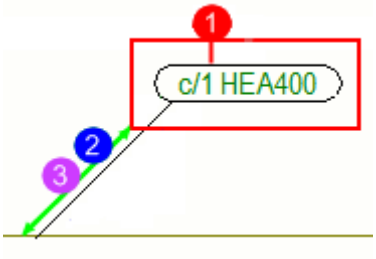
Setting	Description
Save, load and search properties in property pane	 • To load existing properties, select the properties from the list. • To save the modified properties, click . You can also save the properties with another name. • To search for a property, start typing the property name.
Mark	
Mark editor view	

Setting	Description
	(1) Click the **Properties** or **Values** button to show either the property name or the property value in the mark editor view. These buttons are available when you edit an existing mark. **(2)** The preview of the mark shows the elements that you have added, and the representation that you have selected for the mark. You can drag the elements to another location. To delete an element, click the red × delete button in the upper-right corner of the element. **(3)** New element. Use the + **New element** button to open the element list and select the elements to add in your mark. You can also search for elements. The available elements depend on the type of the selected object. If your mark already contains the element, the element has a text **In use**. For more information about mark elements, see "Mark elements in drawing marks" (page 981). **(4)** The **%** button shows the current zoom level in the preview. You can zoom in or out by scrolling the middle mouse button. Click the button to zoom to best fit.
Font	
Font color, **Font**, **Font height** (**Font** settings in view and drawing level properties)	Define the font type, color, and height used in element texts. Clicking **...** gives more font choices.
Frame properties	
Frame, **Frame color** (**Frame around mark**, and **Frame around element** on the view level and drawing level)	Define the frame type and color for one or several elements, or for the whole mark.
Numeric value	
Units and **Format**	Change the units and format of the length and diameter elements that you have selected in the editor view.

Setting	Description
Background (**Background mask** in view and drawing level properties)	Select **Opaque** to hide the part of the drawing that is covered by the mark.  Select **Transparent** to show the part of the drawing that is covered by the mark, so that line work is visible, for example. 
Leader line	Define the leader line type and line arrow type, and how the leader lines are shown for merged marks. Select one of the following **Leader line** types: **Leader line**  **Along line or leader line** **Along line** **Inside part horizontal** **Inside part along part**

Setting	Description
	 **Inside part horizontal or with leader line**  **Inside part along part or with leader line** **Along part centered** **Parallel**: The mark is parallel to the reinforcing bar. **Parallel or leader line**: The mark is parallel to the reinforcing bar line. If there is not enough space for the mark, a leader line is created.
Merged marks **Rebar group mark**	If you are merging marks, or adding rebar group marks, select one of the following **Merged marks** or **Rebar group mark** options: **One leader line to group** creates one leader line for a group of objects. **One leader line per row** merges the marks and creates one leader line for a row of objects. **Parallel leader lines** merges the marks and creates parallel leader lines. **Leader lines to one point** merges the marks and draws all leader lines to one point. **Perpendicular leader lines** allows you to create marks with perpendicular leader lines to rebar groups. Creates marks with leader lines to each rebar in a group plane. **Leader line to first and last** allows you to create marks with perpendicular leader lines to rebar groups. Creates marks to first and last rebar in a group. You can control the length of the perpendicular leader lines with the advanced option `XS_MARK_LEADER_LINE_LENGTH_FOR_PERPENDICULAR` (**File** --> **Settings** --> **Advanced options** --> **Marking - general**). The default value is 0 mm.
Arrow	Select one of the listed **Arrow** types.

Setting	Description
Identical marks in same cast unit	This setting is for merged reinforcement marks. **One leader line to group** creates one leader line for a group of reinforcing bars. **One leader line per row** merges the marks and creates one leader line for a row of reinforcing bars. **Parallel leader lines** merges the marks and creates parallel leader lines. **Leader lines to one point** merges the marks and draws all leader lines to one point. **No merge** does not merge marks. Tekla Structures creates an individual leader line for each mark. This option is only available in view and drawing level properties If you select **No merge**, you still need to define the mark content for the marks that Tekla Structures automatically merges.
Placing	
Alignment	Select where you want to align the mark: **Left** **Center** **Right** **Leader line** The **Leader line** option is available in the following mark types: bolt marks, connection marks, part marks, neighbor part marks, pour object marks, reinforcement marks, neighbor reinforcement marks, merged reinforcement marks, and surface treatment marks.
Rotation	Define the mark rotation.
Placing method	**Free** allows Tekla Structures to search for the first suitable location for the mark. **Fixed** allows you to place the mark in any location. When you use the option **Fixed**, the mark stays where it is even though you update the drawing,

Setting	Description
	whereas with **Free**, Tekla Structures tries to find the optimal place for the annotation object. For more information about placing various annotation objects, see Placement properties for marks, dimensions, notes, texts and symbols (page 1033).
Position (**Quarter** on view and drawing level)	Define the areas where Tekla Structures searches for a position to place the mark. 
Distances	 **(1) Distance s** (**Search margin** on the view and drawing level) is the empty margin that you want to leave around the mark. **(2) Distance d min** (**Minimum distance** on the view and drawing level) is the minimum distance of the mark from the part. **(3) Distance d max** (**Maximum distance** on the view and drawing level) is the maximum distance of the mark from the part. Note that if you use high **Distance s** and **Distance d min** values, the mark placement does not work properly.

Additional mark properties on view and drawing level

Mark properties on the drawing and view level contain some additional visibility settings that are not available in the property pane.

Setting	Description
Content tab	
Content	On the drawing and view level, you need to select additional information for certain types of marks. • For example, for part marks, you need to define part mark settings independently for main and secondary parts, and for sub-assembly main and secondary parts. • For bolt marks, you need to select whether the bolt marks are for site bolts or shop bolts. • For reinforcement marks, you need to select whether the reinforcement marks are for single rebars, rebar groups, or for meshes.
General (or **Appearance** in some marks) tab	
Visibility of mark: **Visibility in view**	Define the visibility of the marks in the view. **distributed** distributes the marks in the view. Tekla Structures only creates marks that are not visible in the other views. **always** always creates marks in the view, irrespective of the settings in other views. **preferred** acts as **distributed**, but the preferred view has a higher priority. Select **distributed** to one view only in a drawing. If you set other views to **distributed**, the marks are located only in the view that has the setting **Visibility in view** set to **preferred**. The option **none** creates no marks.
Visibility of mark: **Parts out of view plane**	Define whether you want to show marks for parts that are out of the view plane. **Visible** displays marks for parts outside of the view plane in the drawing. **Not visible** displays no marks for parts outside the view plane.

Setting	Description
Visibility of mark: **Merge marks**	This setting is for part marks and surface treatment marks. **On** merges marks. For more information about the merging distance and other merging principles of the part marks, see Merge marks (page 356). By default, the maximum merging distance is 1200 mm from the part.
Visibility of bolt marks: **In main parts** **In secondary parts** **In sub-assembly main parts** **In sub-assembly secondary parts**	Define the visibility of the bolt marks. **Visible** displays bolt marks. **Not visible** does not display bolt marks.
Ignore size	This setting is for bolt marks. Filters standard-sized bolt marks out of drawings, which means that Tekla Structures does not display bolt marks of the defined size in drawings. There are some advanced options that can be used in addition to the **Ignore size** setting so that you first give the ignored size and then set the advanced option: XS_OMITTED_DIAMETER_TYPE defines if the ignored size is the size of the bolt (value `BOLT`) or the hole (value `HOLE`). XS_OMITTED_BOLT_TYPE can be used for filtering out ignored bolt marks on the basis of the bolt standard. Enter the name of the bolt standard as the value, for example, 7990. You can also use wildcards, such as * or ?. XS_OMITTED_BOLT_ASSEMBLY_TYPE can be used for filtering out ignored bolt marks on the basis of the bolt type. The values are `SITE`, `SHOP`, and `SITE_AND_SHOP`.
Merging tab in the rebar mark properties	
Preferred direction of merge (on the **Merging** tab in the rebar mark properties)	**Merge vertical** merges the marks in the vertical direction of the drawing.

Setting	Description
	Merge horizontal merges the marks in the horizontal direction of the drawing.

See also

Level mark properties in drawings (page 1001)

Drawing weld mark properties (page 991)

Model weld mark properties in drawings (page 996)

Mark elements (page 981)

Define marks (page 809)

Add automatic marks (page 811)

Add part marks manually in drawings (page 301)

Add reinforcement marks manually in drawings (page 303)

Delete marks for selected parts (page 337)

Add and modify detail marks in drawings (page 347)

Add and modify section marks in drawings (page 343)

Mark elements

Some elements can be added in all marks whereas some of the elements are specific to a mark type. The common elements and the mark-specific elements are listed below.

For information about mark properties, see Mark properties (page 972).

Common elements in marks

There are some mark element types that can be used in most of the mark types.

Element	Description
Text	Add a text element in the mark and add your text there.
Symbol	Open a dialog box where you can change the symbol file in use, and select a symbol to add to the mark.
Template	Add in the mark a custom graphical template created with **Template editor**. Opens a dialog box where you

Element	Description
	can select the template that defines the contents of the mark. For more information about adding templates in marks, see "Add templates in automatic marks (page 839)". In mark templates, you can include detailed information of an embed or assembly, such as the sub-material used. Or you can use a template that changes the unit and the number of decimals in measurement values in a mark. You can also add graphical objects in templates using the **Template editor** tools.
Add space < >	Add spaces between mark elements.
Remove space<--	Add a backspace between the desired elements to remove the default space between them. The default space between the elements depends on the text height and can be changed with the advanced option XS_MARK_ELEMENT_SPACE_FACTOR .
Frame	Add a frame around one or several elements, or around the whole mark depending on your selection.
User-defined attributes **Template attributes**	Add a user-defined attribute or template attribute in the mark from the list of available attributes. In the drawing and view level properties, you need to enter the attribute name exactly as it appears in the `objects.inp` file. In marks, you cannot use template attributes such as `MODEL_TOTAL` that refer to the whole model. Marks only check the information from the object in the drawing and not from the whole model. For more information about adding user-defined attributes or template attributes in marks, see "Add attributes in automatic marks" (page 835).

Element	Description
Custom properties	Add a custom property in the mark from the list of available custom properties.

Part mark elements

You can define part mark contents independently for main and secondary parts and for sub-assembly main and secondary parts.

The following table lists all elements specific to part marks and neighbor part marks.

For more information about part marks properties, see Mark properties (page 972).

Element	Description
Assembly position	Adds the prefix and position number of the assembly.
Part position	Adds the prefix and position number of the part.
Profile	Adds the profile name of part, assembly, or cast unit main part.
Material	Adds the material of part, assembly, or cast unit main part.
Name	Adds the name of part, assembly, or cast unit main part.
Class	Adds the class of part, assembly, or cast unit main part.
Finish	Adds the finish of part, assembly, or cast unit main part.
Size	Adds the size of the part or the assembly or cast unit main part.
Length	Adds the length of the part or assembly, or of the cast unit main part. You can change the unit and format of the length.
Camber	Adds the camber of the part or the assembly or cast unit main part (if this user-defined part attribute is set).
Fittings (NS/FS)	Displays the near side/far side marks in the part mark. (only available in front views).

Element	Description
Face direction	Displays main compass direction (North, East, South, West) of the face, where the mark is added. The direction can only be shown if • the face is vertical • the direction is the same for all assemblies with the same assembly position number In other cases, the element produces no text to mark. Furthermore, face direction is not shown for columns in general arrangement drawings, if you have set **Mark always to center of column in GA drawings** to **Yes** in **File menu** --> **Settings** --> **Options** --> **Orientation marks** .
Gage of outstanding leg	Adds the hole gage. You can control the format of this option with the advance option .
Center-to-center distance	Adds center-to-center distance in the mark. You can control the format of this option with the advanced options XS_CENTER_TO_CENTER_DISTANCE_IN_ONE_PART_STRING and XS_CENTER_TO_CENTER_DISTANCE_IN_TWO_PARTS_STRING.
Rotation angle	Adds the rotation angle of a spiral beam in the mark. For other parts this element gives an empty value.

Bolt mark elements

You can define bolt mark options independently for site and shop bolts.

Below is a list of the elements specific to bolt marks.

Element	Description
Bolt length	Adds the length of the bolt. You can change the unit and format of the length.

Element	Description
Bolt diameter	Adds the bolt diameter. You can change the unit and format of the diameter.
Hole diameter	Adds the hole diameter. You can change the unit and format of the diameter.
Hole depth	Adds the hole depth
Material	Adds the bolt material grade.
Standard	Adds the bolt standard.
Short name	Adds the bolt's short name. This can be the commercial name of a specific bolt, for example.
Full name	Adds the complete name of the bolt. This name is visible in the dialog box list.
Assembly type	Adds the bolt assembly type.
Number of bolts	Adds the quantity of bolts.
Slot length (x) **Slot length (y)**	Adds the slot length in the x or y direction. You can change the unit and format of the length.
Slot length	Adds the slot length. You can change the unit and format of the length.
Slot height	Adds the slot height. You can change the unit and format of the height.
Size	Adds the hole size. You can change the unit and format of the size. You can also use certain advanced options (page 844) to define the contents of the bolt mark **Size** element in different types of drawings.
Countersunk	Adds countersink in the countersunk bolt marks.

Element	Description
Gage of outstanding leg	Adds hole gage. You can control the format of this element with the advanced option XS_GAGE_OF_OUTSTANDING_LEG_STRING.
Center-to-center distance	Adds center-to-center distance. You can control the format of this element with the advanced options XS_CENTER_TO_CENTER_DISTANCE_IN_ONE_PART_STRING and XS_CENTER_TO_CENTER_DISTANCE_IN_TWO_PARTS_STRING.

Reinforcement and neighbor reinforcement mark elements

You can define mark contents separately for single reinforcing bars, bar groups, and reinforcement meshes.

Below is a list of the elements you can include in all reinforcement and neighbor reinforcements marks.

Element	Description
Name	Adds the name of the bar or mesh.
Grade	Adds the material grade of the bar or mesh.
Layer	Adds the layer of the reinforcement.
Diameter	Adds the nominal diameter of the bar.
Class	Adds the class of the bar or mesh.
Length	Adds the total length of the bar. You can change the unit and format of the length.
Number	Adds the quantity of the bars.
Position	Adds the reinforcement position number.
Assembly position	Adds the position number of the associated rebar assembly (page 481).
Shape	Adds the shape of the bar or mesh.
Weight	Adds the weight of the bar or mesh.

Element	Description
cc	Adds the center-to-center spacing of the bars. The options are: • **cc** adds the spacing value if spacing does not vary • **cc min** adds the smallest spacing value of the bar group if spacing varies • **cc max** adds the largest spacing value of the bar group if spacing varies • **cc exact** lists all spacing values of the bar group • **cc target** lists all reinforcing bar target spacing values You can change the unit and format of the cc options.
Pullout picture	Adds a pull-out picture of a bar to the mark. For more information about pull-out pictures, see "Add pull-out pictures in automatic reinforcement marks" (page 848). You can also add pull-out pictures in a reinforcement mark in an open drawing.

Reinforcement and neighbor reinforcement mesh mark elements

You can define mark contents separately for reinforcement meshes.

The following elements are specific to the reinforcement and neighbor reinforcement mesh marks, other elements are the same as for reinforcement marks.

Element	Description
Size	Adds the nominal diameters of the mesh bars, dimensions of the mesh, and the spacings of the bars in the longitudinal and crossing directions.
Mesh length	Adds the length of the reinforcement mesh.
Mesh width	Adds the width of the reinforcement mesh.

Element	Description
cc	You can define cc individually for the longitudinal and crossing bars in the mesh. Adds the center-to-center spacing of the bars. The options are: • **cc longitudinal/cc crossing** adds the spacing value if spacing does not vary • **cc min longitudinal/cc min crossing** adds the smallest spacing value of the bar group if spacing varies • **cc max longitudinal/cc max crossing** adds the largest spacing value of the bar group if spacing varies • **cc exact longitudinal/cc exact crossing** lists all spacing values of the bar group • **cc target** lists all reinforcing bar target spacing values
Diameter longitudinal	Adds the diameter or size of the longitudinal bars.
Diameter cross	Adds the diameter or size of the crossing bars.

Elements in merged reinforcement marks

Some extra elements are available for merged reinforcement marks, in addition to basic reinforcement marks.

For more information about merging marks, see "Merge marks" (page 828).

Element	Description
Block prefix	Adds text or a value to the beginning of each repeated block. Opens a dialog box where you can enter the prefix. You can use the following variables as block prefixes: `%NUMBER%` includes the number of marks merged to the mark.

Element	Description
	%NUMBER_IN_PLANE% includes the number of marks merged in the plane of the drawing to the mark. %NUMBER_OUT_OF_PLANE% includes the number of marks merged in the depth direction of the drawing to the mark.
Single mark content	Adds the content of the single reinforcement mark selected on the **Content** tab to the mark.
Distances between groups	Adds the center-to-center distances between reinforcing bars or bar groups included in a merged mark. Opens a dialog box where you can enter the values.
Symbol separating blocks in mark	Adds a symbol between the blocks in the merged mark. Opens a dialog box where you can define the symbol. The elements that appear before this element on the mark content list generate a block.

Connection mark elements

In connection marks, you can show the connection code, name, number, and running number, the group they belong to, potential errors, and the related DSTV code.

Below is a list of elements specific to connection marks.

Element	Description
Code	Adds the connection code. This is a user-defined code you give for the connection in the connection dialog box. The code can be either a text string or a number.
Name	Adds the connection name, for example, Tube_splice.
DSTV code	Adds the DSTV code.
Connection number	Adds the number of the connection.
Running number	Adds the connection running number. All connections are automatically numbered with a running number.
Group	Adds the connection group.

Element	Description
Connection error	Adds the connection error. The numbers correspond to the connection symbol colors: • 1 = green • 2 = yellow • 3 = red symbol

Pour object mark elements

Element	Description
Material	Adds the defined pour material.
Pour number	Adds the identifier that groups the pour objects in the same group, for example, to be poured at the same time.
Pour type	Adds a property of the pour based on the name of a part.
Concrete mixture	Adds the defined concrete mixture.

Surface treatment mark elements

In surface treatment marks, you can show the name, material, Tekla Structures-specific name, and the code of the surface treatment.

Element	Description
Name	Adds the name defined in the **Name** box in the surface treatment properties in a model.
Material	Adds the surface treatment material.
Class	Adds the class of the surface treatment.
Code	Adds the code of the surface treatment option selected from the **Subtype** list in the in the surface treatment properties in a model. For example, if the subtype is `MF Magnesium Float`, the code is `MF`.
Surface treatment name	Adds the full name of the surface treatment option selected from the

Element	Description
	Subtype list in the in the surface treatment properties in a model. For example, if the subtype is `MF Magnesium Float`, the full name is `Magnesium Float`.

Section and detail mark elements

In section and detail marks, you can show the name of the section/detail, the name of the current drawing, and the name of the source drawing.

Element	Description
Section name/Detail name	Adds the name of the section or the detail (A, B, C, and so on).
Drawing name	Adds the name of the current drawing.
Source drawing name	Adds the name of the drawing where the view is.
Source drawing name when moved	Adds the name of the drawing where the view is. This is shown only if the view is not in the same drawing as the section/detail mark.

View, section view and detail view label mark elements

In view labels, you can show the name of the view, section or detail, the view scale, drawing name, and source drawing name.

Element	Description
View name/Section name/Detail name	Adds the name of the view, section or the detail.
Scale	Adds the scale of the view.
Drawing name	Adds the name of the current drawing.
Source drawing name	Adds the name of the drawing where the view has originally been created.
Source drawing name when moved	Adds the name of the drawing where the view has originally been created. This is shown only when the view has been moved from its original drawing.

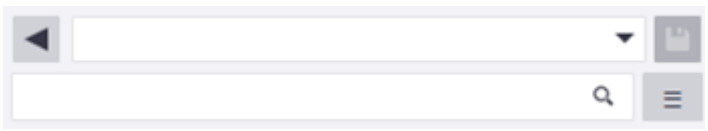
Drawing weld mark properties

In drawing weld mark properties, you can view and modify the properties of a weld mark that has been manually added in a drawing.

Unlike model weld marks, drawing weld marks do not have an associated physical weld in the model, and you can only add drawing weld marks in an open drawing. For more information about welds and weld marks in drawings, see Show welds in drawings (page 489).

To open the drawing weld mark properties, do one of the following in an open drawing:

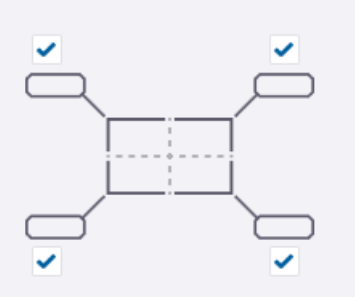
- Click a manually created drawing weld mark. If the property pane is not open, double-click the weld mark.
- Hold down **Shift** and click **Weld mark** on the **Annotations** tab.
- To open the drawing weld mark properties in a dialog box, go to **Quick Launch**, start typing "`weld mark`" and select **Weld mark properties** from the displayed list.

Option	Description
Save, load and search mark properties	 • To load existing mark properties, select the properties from the list. • To save the modified mark properties, enter a new name and click . • To search for a property, start typing the property name.
Mark	
Edge/Around	Indicate whether only one edge or the entire perimeter of a face should be welded. A circle in the weld symbol in drawings indicates that the **Around** option has been used.
Workshop/Site	Indicate where the weld should be created.
Stitch weld	Set this option to **Yes** to create a staggered, intermittent weld. Stitch welds are staggered on both sides of the welded part. Tekla Structures shows the weld type symbols as staggered in weld symbols. If you set this option to **No**, a non-staggered intermittent weld is created. To show the pitch in a weld mark, set **Pitch** to a value greater than 0.0.

Option	Description
Reference text	Additional information to appear in the weld symbol. For example, information about the weld specification or process.
Prefix	a= design throat thickness, s= penetration throat thickness, or z= leg length
Type	The type of the weld. The following weld types are available: Fillet weld Bevel groove single V butt Bevel groove bevel butt Square groove square butt Single V butt with broad root face Single bevel butt with broad root face U groove single U butt J groove butt Flare V groove Flare bevel groove Edge flange Corner flange Plug Bevel backing Spot Seam Slot Partial penetration single bevel butt plus fillet Partial penetration square groove plus fillet Melt through Steep flanked single V butt Steep flanked single bevel butt

Option	Description
	Edge ISO Surfacing Fold Inclined You can customize some of the weld type symbols (symbols 20 - 26), see Customize weld type symbols (page 489) for more information.
Size	The size of the weld. If you select a partial penetration weld as the weld type, you can enter two sizes. Partial penetration welds:
Angle	The angle of weld preparation, bevels, or groove. Tekla Structures displays the angle between the weld type symbol and the fill type contour symbol.
Contour	The fill type contour of a weld can be: None Flush Convex Concave
Finish	Tekla Structures displays the finish symbol above the weld type symbol in drawings. The options are: **G** Grind **M** Machine **C** Chip Finished weld Smooth transition
Effective throat	The weld size used in weld strength calculation.
Root opening	The space between the welded parts.
Length	The length of a regular weld depends on the length of the connection between the welded parts. You can set the exact length of a polygon weld by defining the start and end points of the weld.

Option	Description
Pitch	The center-to-center spacing of welds for non-continuous welds. Pitch is shown in the weld mark if the value is greater than 0.0. To create a non-continuous weld, define the center-to-center spacing and the pitch of the welds. Tekla Structures calculates the distance between the welds as the pitch minus the length of the weld. By default, Tekla Structures uses the – character to separate weld length and pitch, for example, 50–100. To change the separator to @, for example, set the advanced option XS_WELD_LENGTH_CC_SEPARATOR_CHAR 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. The middle button is blue when the values are linked. This means that if you change a value in either of the columns, the corresponding value in the other column also changes.
Font	In **Font color**, **Font**, and **Font height**, define the drawing weld mark font properties.
Background	Select **Opaque** to hide the part of the drawing that is covered by the mark. Select **Transparent** to show the part of the drawing that is covered by the mark, so that line work is visible, for example.
Leader line	
Line type, **Line color**	Define the drawing weld mark leader line type and color. Single line Double line
Placing	

Option	Description
Placing method	**Free** allows Tekla Structures to search for the first suitable location for the weld mark. **Fixed** allows you to place the weld mark in any location. When you use the option **Fixed**, the mark stays where it is even though you update the drawing, whereas with **Free**, Tekla Structures tries to find the optimal place for the weld mark.
Position	Define the areas where Tekla Structures searches for a position to place the weld mark. 
Distances	 **Distance s** is the empty margin that you want to leave around the mark. **Distance d min** is the minimum distance of the mark from the part.

Model weld mark properties in drawings

You can select the content that is displayed in model weld marks in drawings, modify the appearance and visibility of the model weld marks, and select how the weld marks are placed.

Model weld marks refer to the weld marks of the welds that have been created in the model. You can also add model weld marks manually in an open drawing. For more information about welds and weld marks in drawings, see Welds in drawings (page 489). For a list of drawing weld mark properties, see Drawing weld mark properties (page 991).

To display model weld mark properties in an open drawing, do one of the following:

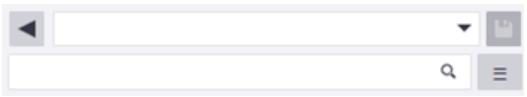
- Select **Weld mark** in the drawing property pane object list.
- Click a model weld mark in the drawing. If the property pane is not open, double-click the mark.
- To open weld mark properties in a dialog box, go to **Quick Launch**, start typing "`welding mark`" and select **Welding mark properties** from the displayed list.

On the drawing and view level, you can display weld mark properties in any of the following ways:

- On the **Drawings & reports** tab, click **Drawing properties**, select the drawing type, and go to weld mark properties directly by clicking the **Weld mark** button (GA drawings), or go to drawing **View properties** and click **Weld mark** in the options tree (other drawing types).
- In an open drawing, double-click the drawing background, and go to mark properties directly by clicking the **Weld mark** button (GA drawings), or go to **View properties** and click **Weld mark** in the options tree (other drawing types).
- In an open drawing, double-click a drawing view frame and click **Weld mark** in the options tree.

Weld mark properties in property pane

Most of the settings listed below are also available on drawing and view level, but the name of the setting might be slightly different.

Setting	Description
Save, load and search mark properties	 • To load existing mark properties, select the properties from the list. • To save the modified mark properties, enter a new name and click . • To search for a property, start typing the property name.
Mark	
Weld number **Edge/Around** **Workshop/Site** **Reference text**	Activate the eye buttons to show the information in model weld marks. These settings are common to above line and below line. If the settings are not activated, the related information is not shown in the model weld mark.

Setting	Description
	Weld number: Tekla Structures assigns a number to each weld when the weld is created. The maximum number of characters that can be shown for **Reference text** is 80, including one character for each line of text. This limitation comes from the model weld. On the drawing and view level, you need to select the needed check boxes or options.
Above line, **Below line**	If the eye buttons for the properties are not activated in the **Above line** or **Below line** column next to the following properties, the properties are not displayed in the model weld mark: **Prefix** **Type** **Size** **Angle** **Contour** **Finish** **Effective throat** **Root opening** **Length** **Pitch**
	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. The middle button is blue when the values are linked. This means that if you change a value in either of the

Setting	Description
	columns, the corresponding value in the other column also changes.
Font	
Font color, **Font**, **Font height**	Define the weld mark font properties.
Background	
Background	Select **Opaque** to hide the part of the drawing that is covered by the mark. Select **Transparent** to show the part of the drawing that is covered by the mark, so that line work is visible, for example.
Leader line	
Line type, **Line color**	Define the model weld mark leader line type and color: Single line Double line
Placing	
Position 	**Distance s** (**Search margin** on view and drawing level) is the empty margin that you want to leave around the mark. **Distance d min** (**Minimum distance** on view and drawing level) is the minimum distance of the mark from the part. Note that if you use high **Distance s** and **Distance d min** values, the mark placement may not work properly.

Additional model weld mark properties on view and drawing level

Weld mark properties on the drawing and view level contain some additional visibility and placing settings that are not available in the property pane.

Setting	Description
Welds **Welds in sub-assemblies** (assembly drawings)	**None** displays no welds in the drawing.
	Site displays only site welds in the drawing.
	Workshop displays only workshop welds in the drawing.

Setting	Description
	Both displays both site welds and workshop welds in the drawing.
Welds in hidden parts	Select how to show the weld marks for welds in hidden parts: • **None**: If the part is hidden, the weld mark is not drawn. • **Site**: If the part is hidden, weld marks for site welds are drawn. • **Workshop**: If the part is hidden, weld marks for workshop welds are drawn. • **Both**: Weld marks are always drawn for hidden parts.
Weld size limit	Enter a weld size to filter welds of that size or smaller out of the drawing. This is useful when you only want to show non-typical welds in a drawing. This option is also available under the **Common settings for all views** setting on the drawing level in single-part and assembly drawings. To set whether the weld size limit is an exact or minimum value, use the advanced option XS_WELD_FILTER_TYPE. To filter out a standard weld type, use the advanced option XS_OMITTED_WELD_TYPES.
Place...	**Placing**: • **free** allows Tekla Structures to search for the first suitable location for the mark. **fixed** allows you to place the mark in any location. • When you use the option **fixed**, the mark stays where it is even though you update the drawing, whereas with **free**, Tekla

Setting	Description
	Structures tries to find the optimal place for the mark. **Quarter** defines the areas Tekla Structures searches for a space to place the mark.

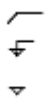
Level mark properties in drawings

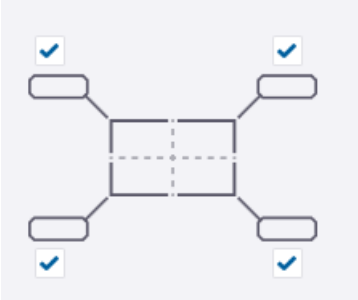
Use the settings in level mark properties to view and modify level mark contents, appearance, and placing.

To display level mark properties in an open drawing:

- Click an existing level mark in the drawing. If the property pane is not open, double-click the level mark.
- Hold down **Shift** and, on the **Annotations** ribbon tab, click **Level mark**.
- Open the object type list in the property pane and click **Level mark**.
- To open the level mark properties in a dialog box, in **Quick Launch**, start typing "`level mark properties`", and select **Level mark properties** in the displayed list.

Setting	Description
Save, load and search level mark properties	 • To load existing level mark properties, select the properties from the list. • To save the modified level mark properties, enter a new name and click . • To search for a propery, start typing the property name.
Mark	
Prefix	Display the text that you type in front of the mark.
Prefix for positive value	**+** displays a + character in front of the value.
Numeric value	Define if the numeric values are visible or hidden.
Postfix	Display the text that you type after the mark.
Font	

Setting	Description
Font color, **Font**, **Font height**	Define the level mark text font color, font type, and font height.
Frame	
Frame , **Frame color**	Define the frame type and color.
Background	
Background	To hide the part of the drawing that is covered by the level mark, select **Opaque**. To show the part of the drawing that is covered by the mark, select **Transparent**.
Numeric value	
Units	Define the units used in level mark dimensions. The available values are **automatic**, **mm**, **cm**, **m**, **foot - inch**, **inch** and **feet**.
Precision	Defines the precision of the level mark dimensions.
Format	Defines the format of the level mark dimensions. ### ###.[#] ###.# ###.[##] ###.## ###.[###] ###.### ### #/#
Digit grouping	Defines whether to use some grouping options to represent the level mark dimensions.
Leader line	
Leader line	Defines the type of the leader line. 

Setting	Description
Arrow, **Arrow height**, **Arrow length**	Defines the type of the leader line arrow, the arrow height, and the arrow length.
Placing	
Rotation	Define the level mark rotation.
Placing method	**Free** allows Tekla Structures to search for the first suitable location for the level mark. **Fixed** allows you to place the level mark in any location. When you use the option **Fixed**, the mark stays where it is even though you update the drawing, whereas with **Free**, Tekla Structures tries to find the optimal place for the mark.
Position	Define the areas where Tekla Structures searches for a position to place the mark. This setting is only available if the selected **Placing method** is **Free**. 
Distances	 **Distance s** (search margin) is the empty margin that you want to leave around the mark. **Distance d min** (minimum distance) is the minimum distance of the mark from the related object.

See also

Add level marks in drawings (page 342)

9.8 Part and neighbor part properties in drawings

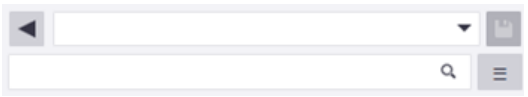
Use the settings in part properties or neighbor part properties to check and change the part or neighbor part properties. In the neighbor part properties, you can also control the visibility and appearance of neighbor part bolts.

To go to part or neighbor part properties:

- On the **Drawings & reports** tab, click **Drawing properties** , select the drawing type and navigate to the part or neighbor part properties.
- In an open drawing, double-click the drawing view frame, and click **Part** or **Neighbor part** in the options tree.
- In an open drawing, double-click the drawing background, and navigate to the part or neighbor part properties.
- Click a part in an open drawing. If the property pane is not open, double-click the part.
- In **Quick Launch**, start typing `part properties`, and then select **Drawing part properties** from the list.
- You can also access the part properties through the drawing property pane object list.

Also see the following support article: Neighboring parts in drawings.

The following table lists all settings on all property levels (drawing, view, object). All part properties do not contain all of the settings listed below.

Option	Description
Save, load and search properties in property pane	 • To load existing properties, select the properties from the list. • To save the modified properties, click . You can also save the properties with another name. • To search for a property, start typing the property name.
Visibility	Click the eye button to activate or deactivate the options.
Inner contours	Show the inner contours of a tube.

Option	Description
Hidden lines **Own hidden lines**	When **Hidden lines** is active, Tekla Structures shows hidden lines in secondary and neighbor parts. If **Own hidden lines** is active, Tekla Structures shows hidden lines in main parts.
Reference lines	Show reference lines. On the drawing and view level, you have the following options available for parts: • Select the **Main part**: **Beam**, **Plate** or **Polygon** check box to show reference lines in main parts. • Select the **Sec part**: **Beam**, **Plate** or **Polygon** check box to show reference lines in secondary parts.
Connecting side mark	Show connecting side marks (page 857).
Edge chamfers	Show edge chamfers (page 429).
Center line	Show center lines. Note that the center line is only shown for assembly main parts and not for secondary parts when looking from the cross section direction. If the part is viewed from the side, the center line is also shown for secondary parts. Note that the **Symbol offset** value affects the size of the center line cross. If the value is 0, the cross is not visible. On the drawing and view level, you have the following options available for parts: • Select the **Main part**: **Beam**, **Plate** or **Polygon** check box to show center lines in main parts. • Select the **Sec part**: **Beam**, **Plate** or **Polygon** check box to show center lines in secondary parts.
Orientation mark	Show orientation marks (page 857).

Option	Description
Connecting side mark	Show connecting side marks (page 857).
Pop-mark	Show pop-marks defined in NC settings.
Edge chamfers	Show edge chamfers (page 429).
Fillet edge	Show fillet edges (page 433).
Representation type	**Outline** shows parts as solid objects. **Exact** shows parts as solid objects. This option also draws the fillet edges and chamfers in the profile cross sections. For some profiles, **Outline** shows them as well. **Workshop form** draws round tube profiles as wrap-around templates. Note that **Workshop form** can only be used in single part drawings. **Symbol** draws parts as lines. Type the symbol offset in **Symbol offset**. **Symbol with partial profile** shows a partial profile of the part. You can also adjust the length of the partial profile (**Partial profile length**), and the offset of the partial profile from the middle point of the part (**Partial profile offset**). **Bounding box** draws parts as boxes surrounding the actual profiles. **Bounding box** is a good option to use with complex items with an extrema box containing many polygons that slow down drawings, because **Bounding box** makes drawings faster. **Base box** shows parts as boxes and uses the **h** and **b** values from the profile catalog as box dimensions.
Symbol offset	Define the distance of the end points of reference lines and center lines from the end points of the object. Note that the **Symbol offset** value affects the size of the center line cross. If the value is 0, the cross is not visible.

Option	Description
Bolt representation	This setting is only available on the drawing and view level for neighbor parts. The options are: **solid** **exact solid** **Symbol** **Symbol2** **Symbol3** **DIN symbol** **User-defined symbol** created in Symbol Editor (page 371). The **DIN symbol** option corresponds to German standards (DIN). The only DIN symbols you can control are: • Symbol 24 for normal workshop bolts • Symbol 25 for normal site bolts • Symbol 26 for front countersunk site bolts • Symbol 27 for back countersunk site bolts • Symbol 28 for front countersunk workshop bolts • Symbol 29 for back countersunk workshop bolts • Symbol 30 for front countersunk holes • Symbol 31 for back countersunk holes
Symbol content	This setting is only available on the drawing and view level for neighbor parts. Select whether to include **hole** or **axis** in the symbol.
Neighbor parts	This setting is only available on the drawing and view level for neighbor parts.

Option	Description
	None does not show neighbor parts. **Connected parts** shows all parts connected to the model object. **Connecting parts** shows only the parts the model object is connected to. **All components** combines the **Connected parts** and the **Connecting parts** options. **By extreme** shows all parts within the boundaries of the main and secondary part.
Main/Secondary parts	This setting is only available on the drawing and view level for neighbor parts. **Main parts** shows only neighbor parts that make the main part of an assembly or a cast unit. **Secondary parts** shows only neighbor parts that are secondary parts of an assembly or a cast unit. **Both** shows both main and secondary parts.
Skew parts	This setting is only available on the drawing and view level for neighbor parts. **Yes** shows skew parts as neighbor parts in the drawing, **No** does not.
Bolts	This setting is only available on the drawing and view level for neighbor parts. **Yes** shows the bolts in the neighbor parts, **No** does not.
Visible lines	Set the color and type of visible lines.
Hidden lines	Set the color and type of hidden lines.
Center line	Set the **Color** of the center lines. You can change the color of the center lines on the drawing and view level only. You can adjust the line type of part center lines with the advanced option XS_CENTER_LINE_TYPE.

Option	Description
Reference lines	Set the color and type of reference lines.
Bolts: **Color**	This setting is only available on the drawing and view level for neighbor parts. Sets the color of the bolts in neighbor parts.
Fill	To add a hatch to outer part faces, use the **Part faces** area, and to add a hatch to cross sections in section views, use the **Sections** area.
Type	Define the hatch type to be used. Clicking the **...** button next to the list opens a preview of the hatch patterns (page 868). **Automatic** selects the hatch pattern automatically from the hatch pattern schema files. **None** uses no hatch.
Color	Define the color for the fill. You can select a predefined color or use **Special** color that is not converted to black in printouts.
Background	Define the background color for the fill. Background color selection is disabled for hardware hatches. Background color can be set for automatic hatches, but it has effect only if the automatic hatch is not defined for the material in the hatch pattern schema file (page 876).
Scale	**Automatic** scales and rotates the hatch automatically. **Custom** allows you to select scaling and rotation manually. **Scaling in X direction** and **Scaling in Y direction** define the scales in the X and Y direction.

Option	Description
	Keep X:Y scaling ratio retains the relative proportions in the hatch pattern. **Scaling angle** rotates the hatch. For example, **Angle** 0.0 is for horizontal and **Angle** 90.0 for vertical.

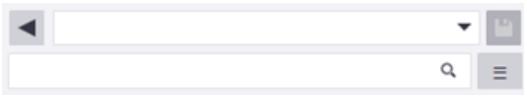
9.9 Bolt properties in drawings

Use the settings in bolt properties to check or change the bolt content and appearance.

To open bolt properties:

- On the **Drawings & reports** tab, click **Drawing properties**, select the drawing type, and navigate to bolt properties.
- To open view level bolt properties, in an open drawing, double-click the drawing view frame and select **Bolt** in the options tree.
- In an open drawing, double-click the drawing background, and navigate to bolt properties.
- Click a bolt in an open drawing. If the property pane is not open, double-click the bolt.
- Select **Bolt** in the property pane object list.
- In an open drawing, go to **Quick Launch**, start typing `bolt properties`, and select **Drawing bolt properties** in the list.

The following table lists all bolt settings on the view and drawing level and in the property pane. All bolt properties do not contain all of the settings listed below.

Option	Description
Save, load and search properties in property pane	 • To load existing properties, select the properties from the list. • To save the modified properties, click . You can also save the properties with another name. • To search for a property, start typing the property name.

Option	Description
Visibility (property pane) **Symbol content**	Select whether to show **Hole** and **Axis** symbols in the drawing.
Presentation type (property pane) **Solid/Symbol**	The options are: **Solid** **Exact solid** **Symbol** **Symbol2** **Symbol3** **DIN symbol** **User-defined symbol** **DIN symbol** corresponds to German standards (DIN). The only DIN symbols you can control are: • Symbol 24 for normal workshop bolts • Symbol 25 for normal site bolts • Symbol 26 for front countersunk site bolts • Symbol 27 for back countersunk site bolts • Symbol 28 for front countersunk workshop bolts • Symbol 29 for back countersunk workshop bolts • Symbol 30 for front countersunk holes • Symbol 31 for back countersunk holes **User-defined symbol** is a symbol that has been created in Symbol Editor.
Color	Change the bolt color.
Visibility of bolts	Control the visibility of bolts in main parts, secondary parts and sub-assemblies separately on the view level, and in GA drawings, also on the drawing level. **Visible** shows bolt group holes in main parts or

Option	Description
	secondary parts. **Not visible** hides the bolt group holes. In assembly drawings you can also define whether to show or hide bolt group holes in sub-assemblies. This setting is not available in the drawing property pane.

See also

Define drawing bolts (page 863)

Part and neighbor part properties in drawings (page 1004)

Define automatic bolt properties in drawings (page 863)

Mark elements (page 981)

9.10 Surface treatment properties in drawings

Use the settings in surface treatment properties to check or change the surface treatment drawing properties. You can show in drawings the surface treatment that you have added on parts in the model.

To open surface treatment properties in drawings:

- On the **Drawings & reports** tab, click **Drawing properties** , select the drawing type, and navigate to surface treatment properties.
- In an open drawing, double-click the drawing view frame and select **Surface treatment** in the options tree.
- In an open drawing, double-click the drawing background, and navigate to surface treatment properties.
- Click a surface treatment in an open drawing. If the property pane is not open, double-click the surface treatment.
- Select **Surface treatment** in the property pane object list.
- In an open drawing, go to **Quick Launch**, start typing `surface treatment`, and select **Drawing surface treatment properties** in the list.

The following table lists all surface treatment properties on the drawing and view level, and in the property pane.

Setting	Description
Save, load and search properties in property pane	

Setting	Description
	• To load existing properties, select the properties from the list. • To save the modified properties, click . You can also save the properties with another name. • To search for a property, start typing the property name.
Visibility	The following settings are only available on the drawing and view level: • **Visible** shows the surface treatment in the drawing. • **Not visible** does not show the surface treatment in the drawing. In the property pane, click the eye button to activate or deactivate the following visibility options: • **Pattern**: Define whether the surface treatment pattern is shown. • **Hidden lines**: Define whether hidden lines in secondary and neighboring parts are shown. • **Own hidden lines**: Define whether hidden lines in main parts are shown. These visibility options are available on the drawing and view level as check boxes.
Representation	Define the representation style of the surface treatment. The options are: **Outline** **Exact** **Workshop form** **Symbol** **Bounding box** **Base box**
Hidden lines	Define the color and the type of the hidden lines.

Setting	Description
Visible lines	Define the color and the type of the visible lines.

See also

Define drawing surface treatment (page 880)

Modify building object properties (page 427)

Surface treatment hatch pattern properties (surfacing.htc)

You can change the properties of the hatch patterns for each surface treatment type separately.

The properties of the hatch patterns are defined in the `surfacing.htc` file, located by default in `..\Tekla Structures\<version>\environments\common\system`. In addition to this file, the surface treatment code file `product_finishes.dat` is needed. It is located in the same folder.

If you create your own surface treatment hatch patterns in your company, you can store `surfacing.htc` and `product_finishes.dat` in the firm folder defined by the advanced option `XS_FIRM`.

Note:

NOTE When you edit a schema file, you must reopen the model to apply the changes.

The syntax of `surfacing.htc` file is:

```
Surfacing Type, Surfacing Code, Hatch name, Scale, [Color],
[Automatic Scaling and Rotation]
```

Example:

```
1,MF,ANSI31,0.7
1,SMF,ANSI32,0.7
1,WT,ANSI33,0.7
1,HT,ANSI34,0.7
1,LSB,AR-SAND,0.7
2,SM1,CROSS,1.0
2,SM2,CHECKERED,1.0
3,TS3,FBBRICKC,1.0
4,FP,ANSI31,1.0
4,UP,ANSI32,1.0
```

Option	Description
`Surface treatment type`	• 1 = concrete finish • 2 = special mix • 3 = tile surface • 4 = steel finish
`Surface treatment code`	This is the abbreviation used in drawings and reports, for example, MF for Magnesium Float. The `product_finishes.dat` file contains a full list of surface treatment codes.
`Hatch name`	You can check the hatch pattern names and related hatch patterns by going to the **Fill** tab in part properties, selecting a hatch pattern from the **Type** list and clicking the **...** button next to the list. The selected hatch pattern is marked with a red frame.
`Scale`	Scale is a numeric value that Tekla Structures uses to scale hatches.
`Color` (optional)	0=black (default) 1=white 2=red 3=green 4=blue 5=cyan 6=yellow 7=magenta 120= Special (use this color for gray shade) The hatch color defines the line width for the printer. If you do not define a color for a hatch in the `surfacing.htc` file, Tekla Structures uses the color defined on the **Appearance** tab in surface treatment properties. The **Visible lines** color and type is used for the front of the surface treatment, and the **Hidden lines** for the back.

Option	Description
`Automatic Scaling and Rotation` (optional)	1=true 0=false (default)

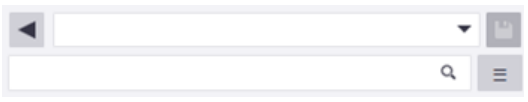
9.11 Reinforcement object properties in drawings

Use the options in reinforcement properties, rebar properties and rebar mesh properties to check and modify the rebar, rebar assembly, and rebar mesh visibility, appearance, and content. The neighbor reinforcement properties are the same as reinforcement properties.

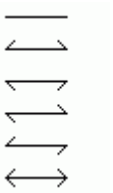
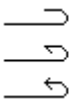
To open reinforcement properties:

- On the **Drawings & reports** tab, click **Drawing properties**, select the drawing type, and navigate to reinforcement properties.
- In an open drawing, double-click the drawing view frame and select **Reinforcement** in the options tree.
- In an open drawing, double-click the drawing background, and navigate to reinforcement properties.
- Click reinforcement in an open drawing. If the property pane is not open, double-click the reinforcement.
- Select **Rebar** or **Rebar mesh** in the property pane object list.
- In an open drawing, go to **Quick Launch**, start typing `reinforcement properties`, and select **Drawing reinforcement properties** in the list. If you want to open mesh properties instead, start typing `mesh properties`, and select **Drawing reinforcement mesh properties**.

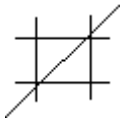
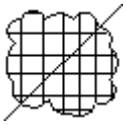
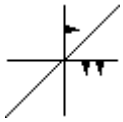
The following table lists all reinforcement properties on all property levels (drawing, view, object).

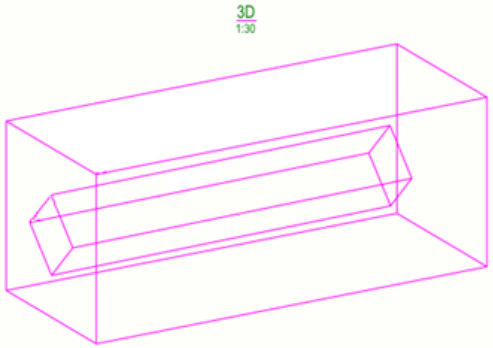
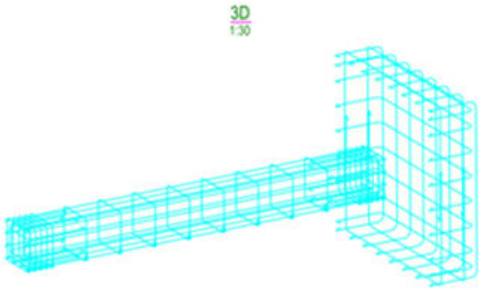
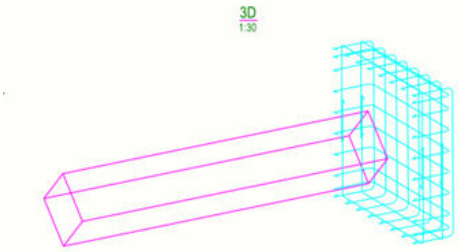
Option	Description
Save, load and search properties in property pane	 • To load existing properties, select the properties from the list. • To save the modified properties, click . You can also save the properties with another name. • To search for a property, start typing the property name.

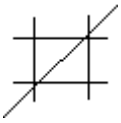
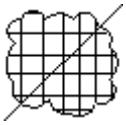
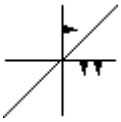
Option	Description
Visibility of lines	Click the eye button to activate or deactivate the options.
Lines behind parts (property pane) **Hide lines behind parts** (drawing and view level)	Hide the lines behind the parts. This is useful, for example, with lifting anchors where the reinforcing bar is partly outside the part. If you wish to show embeds while hiding reinforcement, use the XS_HIDDEN_LINES_UNHIDE_EMBEDDED advanced option.
Lines behind other rebars (property pane) **Hide lines behind other rebars** (drawing and view level)	Hide the lines behind other reinforcement bar lines.
Rebar content, visibility and representation:	
Visibility of all reinforcing bars	This setting is only available on the drawing and view level. **Visible** shows the bars. **Not visible** does not show the bars.
Representation type (property pane) **Representation** (drawing and view level)	**single line** draws a single line with radiused bends. **single line with filled ends** draws a single line for parallel bars and filled ends for perpendicular bars. **double lines** draws an outline of the bar with radiused bends. **double lines with filled ends** draws an outline of the bar with radiused bends and filled bar ends. **filled line** draws a solid bar with radiused bends. **stick** draws a single line without radiused bends.
Visibility of reinforcing bars in group	This setting is only available on the drawing and view level. **all** shows all bars in a group or mesh. **first bar** shows only the first bar in the group or mesh. **last bar** shows only the last bar in the group or mesh.

Option	Description
	first and last shows the first and last bar in the group or mesh. **bar in the middle of group** shows one bar in the middle of the group or mesh. **two bars in the middle of group** shows two bars in the middle of the group or mesh. **customized** indicates that you have specified the location of the only visible reinforcing bar. Only applies to bar groups.
Straight end (property pane) **Symbol at straight end** (drawing and view level)	Select symbols to be used at straight ends.  Reinforcing bar end symbols are always drawn in a solid line type, regardless of the selected reinforcement line type. Only applies to reinforcing bars.
Hooked end (property pane) **Symbol at hooked end** (drawing and view level)	Select symbols to be used at hooked ends.  Only applies to reinforcing bars. Reinforcing bar end symbols are always drawn in a solid line type, regardless of the selected reinforcement line type.
Mesh content, visibility and representation:	
Visibility of all meshes	This setting is only available on the drawing and view level.

Option	Description
	Visible shows the meshes in the drawing. **Not visible** does not show the meshes in the drawing.
Representation type (property pane) **Representation** (drawing and view level)	**single line** draws a single line with radiused bends. **single line with filled ends** draws a single line for parallel bars and filled ends for perpendicular bars. **double lines** draws an outline of the bar with radiused bends. **double lines with filled ends** draws an outline of the bar with radiused bends and filled bar ends. **filled line** draws a solid bar with radiused bends. **stick** draws a single line without radiused bends. **outline** shows the shape of the mesh using an outline rectangle or polygon, and a diagonal line. Applies only to reinforcement meshes. **outline (ignore holes)** ignores holes and draws over them. Only applies to reinforcement meshes.
Visible longitudinal bars (property pane) **Visibility of longitudinal wires** (drawing and view level) **Visible cross bars** (property pane) **Visibility of crossing bars** (drawing and view level)	**all** shows all bars in a group or mesh. **first bar** shows only the first bar in the group or mesh. **last bar** shows only the last bar in the group or mesh. **first and last** shows the first and last bar in the group or mesh. **bar in the middle of group** shows one bar in the middle of the group or mesh. **two bars in the middle of group** shows two bars in the middle of the group or mesh. **customized** indicates that you have specified the location of the only

Option	Description
	visible reinforcing bar. Only applies to meshes.
Mesh symbol	Define the mesh symbol to be used. The mesh symbol appears in the middle of the diagonal line. **Symbol 1**  **Symbol 2**  **Symbol 3** 
Mesh symbol size	Define the size of the mesh symbol.
Rebar and mesh line properties:	
Visible lines	Define the color and type of the visible lines. Reinforcing bar end symbols are always drawn in a solid line type, regardless of the selected reinforcement line type.
Hidden lines	Define the color and type of the hidden lines.
Rebar assembly properties:	
Visibility of rebar assemblies	This setting is only available on the view level. **Visible** shows the rebar assemblies. **Not visible** does not show the rebar assemblies.
Rebar **Representation**	This setting is only available on the view level. **Outline**: All sub-assemblies of the rebar assembly are shown as outline boxes.

Option	Description
	 **As individual rebar objects**: All rebar objects in the rebar assembly and sub-assemblies are visible.  • **Sub-assemblies as outline, highest level as individual rebar objects**: All sub-assemblies are shown as outline boxes and the highest level of the rebar assembly is shown as individual rebar objects. 

Option	Description
Visible lines	Define the color and type of the visible lines in rebar assemblies. Reinforcing bar end symbols are always drawn in a solid line type, regardless of the selected reinforcement line type.
Mesh symbol	This setting is only available on the view level. Define the mesh symbol to be used in rebar assemblies. The mesh symbol appears in the middle of the diagonal line. **Symbol 1**  **Symbol 2**  **Symbol 3** 
Mesh symbol size	This setting is only available on the view level. Defines the size of the mesh symbol in rebar assemblies.

Additional ways of modifying reinforcement

In addition to the settings in reinforcement properties, you can modify the reinforcement in any of the following ways:

- Increase the size of the bend and end symbols (in drawing units) using the advanced options XS_REBAR_BEND_MARK_SYMBOL_MIN_SIZE and XS_REBAR_END_SYMBOL_MIN_SIZE in **File menu** --> **Settings** --> **Advanced options** --> **Concrete detailing**.
- Use the advanced option XS_REBAR_REVERSE_END_SYMBOLS in **File menu** --> **Settings** --> **Advanced options** --> **Concrete detailing** to change the end symbols to a different direction.

- Modify the reinforcing bar bending schedule, rounding of bar dimensions, symbols for meshes, strands, and unbonding, and appearance of reinforcement pull-outs in the `rebar_config.inp (page 1023)` file.

See also

Show rebar assemblies in drawings (page 481)

Define drawing reinforcement and meshes (page 885)

Part and neighbor part properties in drawings (page 1004)

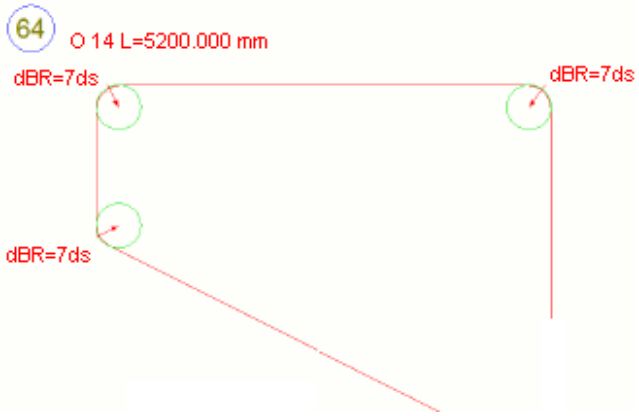
Reinforcement settings for drawings (rebar_config.inp)

Tekla Structures uses the settings in the `rebar_config.inp` file in the system folder (XS_SYSTEM), firm folder or in the project folder to define the following reinforcement-related issues in drawings:

- Selected area-specific reinforcing bar bending schedule
- Rounding of bar dimensions
- Available symbols for meshes, strands, and unbonding
- Appearance of reinforcement pull-outs

The entries in the `rebar_config.inp` are listed and described below:

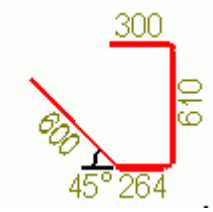
Entry	Description
`MergeOneFormat`	No longer used. Define these properties in the drawing properties.
`MergeTwoOrMoreFormats`	
`MergeAndFormat`	
`LeaderLinetype`	
`DimensionMarkSpacingSeparator`	`="/"` Affects the separator in reinforcement marks.
`ExactDimensionMarkSpacingSeparator`	`=" + "` Separator between different exact spacing values in reinforcement marks.
`ExactDimensionMarkPcsSeparator`	`="*"` Separator between the number of bars and their exact spacing value in reinforcement mark.
`BendingAngleTolerance`	Set a tolerance value for the angle. Angles that differ from the set tolerance value less than

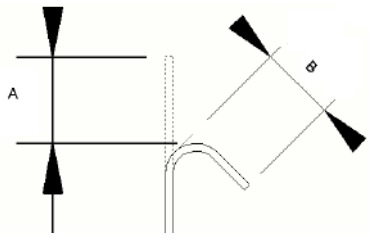
Entry	Description
	the tolerance are recognized and lead to a correct bending shape. Enter the tolerance value as a radians, not as a degree. The default value is 0.001 radians, which is 0.0573 in degrees. This applies to all bending shapes.
BentRebarTolerance	Set a tolerance value. Depending on the value, slightly curved reinforcing bars get straight shape. If reinforcing bar diameter is 20 mm and radius is 200 m then value 20/200000 = 0.0001. This variable defines correct curved reinforcing bar in case of long reinforcing bars to get a correct shape for the bar. This option is used in comparing the relation of the reinforcing bar diameter and radius. If the relation is smaller than BentRebarTolerance, the reinforcing bar is of bend_type_1, otherwise it is of bend_type_34.
PullOutBendingRadiusAsMultiplier	Set to 1 to show the pullout bending radii using multiplier instead of mm. 
GroupBarMark	No longer in use.
MarkingDimAttributes	No longer in use.
ScheduleCountry	Defines which bending schedule is used. Affects bending shapes in templates and reports. The available schedules are FIN, SWE, UK, US. When you number the model, the bending shape for the bar is given according to this

Entry	Description
	information. For example, in the Default environment, the bending shapes are letters A, B, C, and so on.
ScheduleDimensionRoundingDirection ScheduleTotalLengthRoundingDirection	Options: • "UP": rounds bar dimensions up • "DOWN": rounds bar dimensions down • "NEAREST": rounds bar dimensions either up or down
ScheduleDimensionRoundingAccuracy	Sets the rounding accuracy for bar dimensions. Default is 1 mm. Tekla Structures rounds individual bar dimensions up or down according to the option you select for ScheduleDimensionRoundingDirection.
ScheduleTotalLengthRoundingAccuracy	Sets the rounding accuracy for the total bar length. Default is 10 mm. Tekla Structures rounds individual bar dimensions up or down according to the option you select for ScheduleTotalLengthRoundingDirection.
BentSymbolFile	Points to the symbol file that contains the available reinforcing bar bent symbols. By default, points to the file bent.sym, which in the default environment is located in the folder ..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\symbols.
MeshSymbolFile	Points to the mesh symbol file that contains the available mesh symbols. Affects the available reinforcement mesh symbols in drawings. By default, points to the mesh.sym file in the ...\Tekla Structures\<version>\environments\common\symbols folder.
StrandSymbolFile	Points to the strand symbol file that contains the available strand symbols. Affects drawings. By default, points to the strand.sym file in the ...\Tekla Structures\<version>\environments\common\symbols folder.

Entry	Description
UnbondingSymbolFile	Points to the unbonding symbol file that contains the available unbonding symbols.
RebarMeshSize	Template for rebar mesh size. ="%CC_DIAMETER_LONG%/%CC_DIAMETER_CROSS%-%CC_PITCHING_LONG%/%CC_PITCHING_CROSS%-%LENGTH%/%WIDTH%"
PullOutDimensionFormat	Defines the format for displaying the dimensions. The format follows the dimension properties format. Options: • 0 = ### • 1 = ###[.#] • 2 = ###.# • 3 = ###[.##] • 4 = ###.## • 5 = ###[.###] • 6 = ###.### • 7 = ### #/# • 8 = ###/##.###
PullOutDimensionPrecision	Sets the level of precision. The precision is calculated using the following formula: 1/value = precision. In metric systems, you may want to use the values 1, 10, and 100, and in imperial systems, the values 2, 4, 8, 16 and 32, for example.
PullOutDimensionUnit	Defines the units to use. Options: • 0 = auto • 1 = mm • 2 = cm • 3 = m • 4 = inch • 5 = foot and inch

Entry	Description
PullOutColor	Sets the color for the pull-outs in reinforcement marks. Options: • 1 = black • 2 = red • 3 = bright green • 4 = blue • 5 = cyan • 6 = yellow • 7 = magenta
PullOutVisibleLineType	Sets the line type for reinforcing bar shape in pull-outs. Options: 1 = 2 = 3 = 4 = 5 = 6 = 7 =
PullOutRepresentation	Sets the representation type. Options: • 0 = single • 1 = double • 2 = filled • 3 = stick
PullOutAngleColor	Sets the color for the angle in pull-outs. Options: • 1 = black • 2 = red

Entry	Description
	• 3 = green • 4 = blue • 5 = cyan • 6 = yellow • 7 = magenta • 8 = brown • 9 = green • 10 = dark blue • 11 = forest green • 12 = orange • 13 = gray 
PullOutAngleLineType	Sets the line type for angle lines in pull-outs. Options: • 1 = —————— • 2 = - - - - - • 3 = — — — • 4 = — - — - • 5 = - - - - - - - - • 6 = — .. — .. • 7 = — - - - -
PullOutLeaderLineMinLength	Sets a minimum length for the small leader lines that point to the dimension text. The default value is 10 mm. To switch leader lines off completely, use a large value.
PullOutShowDuplicateDims	Defines whether duplicate dimensions are showed multiple times for one bar. Options:

Entry	Description
	• `0` = duplicate dimensions are not shown (default) • `1` = equal and parallel dimensions are shown, but similar hook dimensions are not shown • `2` = equal and parallel dimensions are not shown, but both hook dimensions are shown • `3` = all dimensions are shown • `4` = hook dimensions are not shown • `5` = hook dimensions, or equal and parallel dimensions are not shown
`PullOutShowUSHookDims`	Defines if the US/NA style of dimension will be shown for hooks over 90 degrees. Options: • 0 = shows European hook dimension (= leg length, default) • 1 = shows US hook dimension (= straight length) for hooks >90 degrees See the image below for the difference between the US/NA (A) and European (B) hook dimension. 

9.12 Pour object and pour break properties in drawings

Use the settings in pour object properties and pour break properties in general arrangement drawings to control the visibility and appearance of the pour objects and pour breaks.

NOTE The options and functionality related to pours are only available if pours are enabled in the model. You can enable pours in a model by setting the XS_ENABLE_POUR_MANAGEMENT advanced option to

TRUE. In the Default environment, pours are only enabled in the (Concrete) Contractor role.

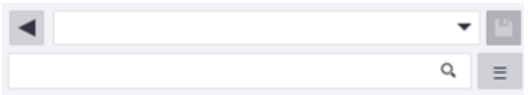
WARNING If the pours are enabled in the model, do not disable the pours using XS_ENABLE_POUR_MANAGEMENT, especially in the middle of the project. This may cause problems if you have drawings containing pours, and if you are sharing your model. The pours and pour breaks in the model and in the drawings may get invalid, and you may lose all pour-related modeling work.

Pour object properties

To open pour object properties in GA drawings:

- On the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing** and then click **Pour object...**
- In an open drawing, double-click the drawing background and click **Pour object...**
- In an open drawing, double-click the drawing view frame, and select **Pour object** in the options tree.
- Click a pour object in an open drawing. If the property pane is not open, double-click the pour object.
- Select **Pour object** in the property pane object list.
- In an open drawing, go to **Quick Launch**, start typing `pour object properties`, and select **Drawing pour object properties** in the list.

The following table lists all pour object properties on all property levels (drawing, view, object).

Option	Description
Save, load and search properties in property pane	 • To load existing properties, select the properties from the list. • To save the modified properties, click . You can also save the properties with another name. • To search for a property, start typing the property name.
Visibility	In the property pane, click the eye button to activate or deactivate the following options:

Option	Description
	Hidden lines: Define whether pour object hidden lines are shown. **Own hidden lines**: Define whether pour object own hidden lines are shown. **Edge chamfers**: Define whether edge chamfers (page 429) are shown. **Fillet edges**: Define whether fillet edges (page 433) are shown. The visibility options are available on the drawing and view level as check boxes.
Visible lines	Select the color and type of the visible pour object lines.
Hidden lines	Select the color and type of the hidden pour object lines.
Fill To add a hatch for outer pour faces, use the **Part faces**/**Pour faces** options, and to add a hatch for cross sections in section views, use the **Sections** options.	
Type	Define the fill type. To open a preview to the hatch patterns, click the ... button next to the pattern list. The **Automatic** option selects the fill type automatically from the hatch pattern schema files. The **None** option leaves hatching out.
Color	Defines the color for the fill. You can select a predefined color or use **Special** color that is not converted to black in printouts.
Background	Defines the background color for the fill. The **Background** selection is disabled for hardware hatches, such as hardware_SOLID or hardware_LINES. The background color can be set for automatic hatches, but it has effect only if automatic hatch is not defined for the material in the hatch pattern schema file.

Option	Description
Scale	Note that the option names are slightly different on the view and drawing level. **Automatic** scales and rotates the fill automatically. **Custom** allows you to select scaling and rotation manually. **Scaling in X direction** and **Scaling in Y direction** define the scales in the X and Y direction. **Keep ratio of x and y** retains the relative proportions in the fill. **Scaling angle** rotates the fill. For example, **Scaling angle** 0.0 is for horizontal rotation and **Scaling angle** 90.0 for vertical rotation.

Pour break properties

To open pour break properties in GA drawings:

- On the **Drawings & reports** tab, click **Drawing properties** --> **GA drawing** and then click **Pour breaks...**
- In an open drawing, double-click the drawing background and click **Pour breaks...**
- In an open drawing, double-click the drawing view frame, and select **Pour break** in the options tree.
- Click a pour break in an open drawing. If the property pane is not open, double-click the pour break.
- Select **Pour break** in the property pane object list.
- In an open drawing, go to **Quick Launch**, start typing `pour break properties`, and select **Drawing pour break properties** in the list.

Option	Description
Save, load and search properties in property pane	 • To load existing properties, select the properties from the list. • To save the modified properties, click . You can also save the properties with another name.

Option	Description
	• To search for a property, start typing the property name.
Visibility	The following settings are only available on the drawing and view level: • **Visible**: Pour breaks are visible. • **Not visible**: Pour breaks are not visible. In the property pane, click the eye button next **Hidden lines** to show or hide the pour break hidden lines. This option is available on the drawing and view level as a check box.
Visible lines	Select the color and type of the visible pour break lines.
Hidden lines	Select the color and type of the hidden pour break lines.

See also

Show pours in drawings (page 485)

9.13 Placement properties for marks, notes, dimensions, texts and symbols in drawings

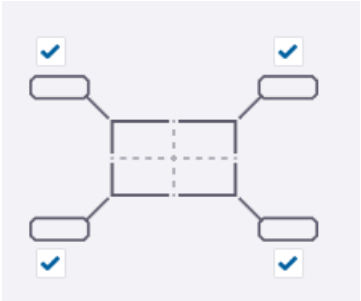
Use the placing settings for annotation objects (dimensions, marks, notes, texts, and symbols) to control how the objects are placed in a drawing.

You can set automatic placing properties for dimensions and marks before you create a drawing. In an open drawing, you can define placing properties for annotation objects that you create manually, and modify the placing properties of the annotation objects that you already have in your drawing.

- In the property pane, select an annotation object in the object list. The placing properties are located in the **Placing** section of the annotation object properties.
- On the drawing and view level, open the annotation object properties first and then click the **Place...** button.

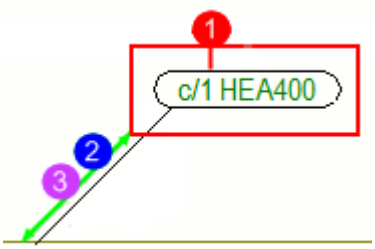
Setting	Description
Placing section in property pane or **Part mark placing** properties on the drawing and view level	

Setting	Description
Short dimensions	Define the text location of short dimensions: : Text is inside the dimension lines. : Text is outside the dimension line.
Dimension line spacing	Define the space between parallel dimension lines. In manually created dimensions, this setting works only if dimension **Placing** is set to **Free**.
Alignment	Select where you want to align the mark, text, or note: **Left** **Center** **Right** **Leader line** The **Leader line** option is available in following mark types: bolt marks, connection marks, part marks, neighbor part marks, pour object marks, reinforcement marks, neighbor reinforcement marks, merged reinforcement marks and surface treatment marks.
Rotation	Define the mark rotation.
Placing method	**Free** allowsTekla Structures to search for the first suitable location for the annotation object. This also affects the direction of the dimensions. **Fixed** allows you to place the annotation object in any location. When you use the option **Fixed**, the annotation object stays where it is even though you update the drawing, whereas with **Free**, Tekla Structures tries to find the optimal place for the annotation object.

Setting	Description
Negative direction, **Positive direction** **Direction** (drawing and object level)	Define the side of the dimensioned object on which Tekla Structures places the dimensions. This setting affects the **free** setting. This setting is not available in GA drawing dimension properties.
Distance s (property pane) **Search margin** (drawing and view level)	 Define the empty margin that you want to have around the the annotation objects. Note that if you use a high **Search margin** value, the mark placement does not work properly.
Distance d min (property pane) **Minimum distance** (drawing and view level)	 Define the minimum distance of the annotation object from the part. Note that if you specify a high value, the annotation object placement does not work properly.
Distance d max (property pane) **Maximum distance** (drawing and view level)	 Define the maximum distance of the annotation object from the part.
Position (**Quarter** on the drawing and view level)	Define the areas where Tekla Structures searches for a position to place the annotation object. 

Setting	Description
	This setting is available for marks, notes, and texts. Weld placement depends on the welding direction. Welds can only be placed in certain sectors, so this setting is not available for model weld marks. However, this option is available for manually added weld marks in the final drawing.

The following image illustrates the **Distance s** (**Search margin**), **Distance d min** (**Minimum distance**), and **Distance d max** (**Maximum distance**) of a mark:



(1) Distance s

(2) Distance d min

(3) Distance d max

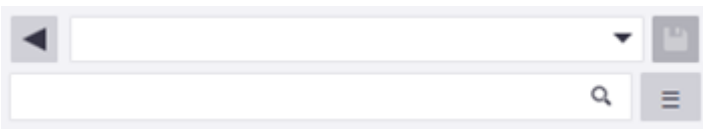
9.14 Model weld properties in drawings

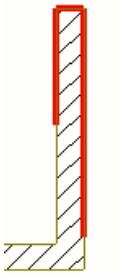
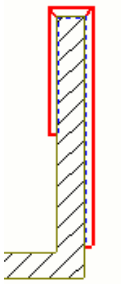
You can select which model welds are visible in drawings and drawing views, and set the weld color and line type.

To open weld properties in drawings:

- On the **Drawings & reports** tab, click **Drawing properties**, select the drawing type, and navigate to weld properties.
- In an open drawing, double-click the drawing view frame, and click **Weld** in the options tree.
- In an open drawing, double-click the drawing background, and navigate to weld properties.
- Click a weld in an open drawing. If the property pane is not open, double-click the weld.
- In **Quick Launch**, start typing `weld properties`, and then select **Drawing weld properties** from the list.

The following table lists all weld settings on all property levels (drawing, view, object). All weld properties do not contain all of the settings listed below.

Setting	Description
Save, load and search properties in property pane	 • To load existing properties, select the properties from the list. • To save the modified properties, click . You can also save the properties with another name. • To search for a property, start typing the property name.
Welds **Welds in sub-assemblies**	These settings are only available on the drawing (GA drawings) and view level. **Not visible** does not display any welds in the selected view or drawing. **Site weld visible** displays only site welds in the view/drawing. **Workshop weld visible** displays only workshop welds in the view or drawing **Both visible** displays both site welds and workshop welds in the view or drawing.
Weld size limit	This setting is only available on the drawing (GA drawings) and view level. Enter a weld size limit to filter welds of that size and smaller out of the drawing. This is useful when you only want to show non-typical welds in a drawing. To set whether the weld size is an exact or minimum value, use the advanced option XS_WELD_FILTER_TYPE. To filter out a standard weld type, use the advanced option XS_OMITTED_WELD_TYPE. The weld size limit can also be set for all views in a drawing (page 906).
Visibility	In the property pane, click the eye button to activate or deactivate the visibility options. Select whether to show **Hidden lines** or **Own hidden lines**.

Setting	Description
Representation	Select **Path** or **Outline**. **Path**  **Outline**  Weld solids are shown in drawings in the following cases: • Weld solids are shown in drawings for those weld types that have real solid support. Welds that have no real solid support are shown in the model with a hexagonal placeholder, and in drawings weld solids are not shown. • Welds that have custom cross-sections are also supported.
Visible lines	Set the color and type of the visible weld lines.
Hidden lines	Set the color and type of the hidden weld lines.

9.15 Drawing sketch object properties

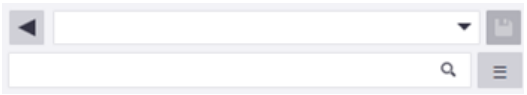
You can draw many kinds of sketch objects (graphical objects) in your drawings (lines, rectangles, polylines, polygons, arcs, circles, clouds) and use the sketch objects for highlighting particular spots of interest, for example. In sketch object properties, you can control the appearance of the sketch objects.

To open the properties of a sketch object, do one of the following:

- Go to the **Drawing** tab, hold down **Shift** and click any of the sketch object commands.

- To open sketch object properties in an open drawing, click a sketch object. If the property pane is not open, double-click the object.
- Select the sketch object in the property pane object list.
- You can also open sketch object properties through **Quick Launch**. For example, start typing `polygon properties`, and then select **Drawing polygon properties** in the list.
- Note that the **Pattern line** command has a property dialog box of its own (page 394).

The available settings vary depending on the sketch object type. All sketch object settings are described below.

Setting	Description
Save, load and search properties in property pane	 • To load existing properties, select the properties from the list. • To save the modified properties, click . You can also save the properties with another name. • To search for a property, start typing the property name.
Line	
Line type/Type	Define the line type of the object.
Line color/Color	Define the color of the object lines.
Bulge/Bulge for all lines	Allowed values 0 - 1. The bulge factor defines the curvature of the curved segments of objects using the following calculation:

Setting	Description
	Arc height = Line length * Bulge factor Changing the bulge factor of a polyline or polygon modifies all the segments of that object.
Radius	Define the radius of arcs and circles.
Arrow	
Arrow	Define the end arrow type for lines.
Position	Define the arrow position: **None**, **Left**, **Right**, **Left and right**.
Arrow height	Define the arrow height.
Arrow length	Define the arrow length.
Fill: You can use hatches (fills) also in sketch objects like rectangles, polygons, clouds, and circles.	
Type	Define the fill type to be used in a sketch object. Clicking **...** shows the available fill types. Note that all settings are not available for automatic or hardware hatches.
Color	Define the color of the fill. This setting is disabled for automatic hatches.
Background	Define the background color of the fill. This setting is disabled for hardware hatches.
Angle	Rotate the fill. Angle 0.0 is for horizontal and 90.0 for vertical. This setting is disabled for hardware hatches and automatic hatches.
X offset for hatch **Y offset for hatch**	Move the fill pattern inside the object in the X and Y direction by the specified value. This setting is disabled for hardware hatches and automatic hatches.
Scaling in X direction **Scaling in Y direction**	Define the fill scales in the x and y direction. This setting is disabled for hardware hatches and automatic hatches.

Setting	Description
Keep ratio of x and y	Retain the relative proportions in the hatch pattern. This setting is disabled for hardware hatches and automatic hatches.
Position	
Behind model objects	Select to place the sketch object behind model objects. This setting is not available for all sketch objects.

See also

Draw and modify sketch objects in drawings (page 388)

Create and add pattern lines in drawings (page 394)

Hide part face areas and outlines with cover-up tools (page 413)

9.16 Drawing grid and grid line properties

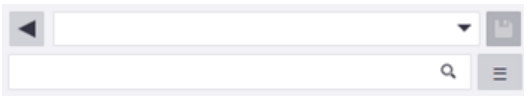
Use the grid and grid line properties to check and modify the grid settings in drawings. For example, you can modify the grid visibility, label placing, line and label colors, and label font.

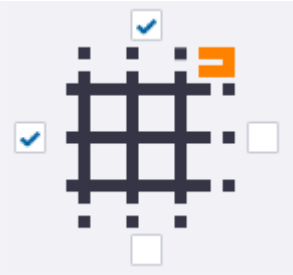
To open the grid or grid line properties in drawings:

- To show view-specific grid properties: In an open drawing, double-click a drawing view frame and click **Grid** in the options tree.
- To show drawing-specific grid properties in a general arrangement drawing: In an open drawing, double-click the drawing background, and click **Grid...**.
- To show the grid or grid line properties in an open drawing, click the grid or the grid line. If the property pane is not open, double-click the grid or grid line. Ensure that you have the correct selection switch active: To modify grids, use the selection switch **Select grid** , and to modify grid lines, use **Select grid line** .
- You can also open grid or grid line properties by selecting **Grid** or **Grid line** in the property pane object list.

- You can also show grid or grid line properties through **Quick Launch**: Go to **Quick Launch**, type `grid`, and select **Drawing grid properties** or **Drawing grid line properties** in the list.

The following table describes all grid properties that are available on the drawing and view level, and in the property pane:

Option	Description
Save, load and search properties in property pane	 • To load existing properties, select the properties from the list. • To save the modified properties, click . You can also save the properties with another name. • To search for a property, start typing the property name.
Grid lines	
Visibility (property pane) **Grid** (drawing and view level)	**Visible** shows the grids. **Only grid labels visible** shows only the grid label and a short section of the grid line. **Not visible** does not show the grids. This option is only available on the drawing and view level.
Extension length	Define the distance between the end of the grid line and the label text. On the drawing and view level, there is a box for entering the value in the **Text placing** section.
Line color, **Line type** (property pane) **Color**, **Type** (drawing and view level)	Define the grid line color and type.
Grid labels	
Font color, **Font**, **Font height**, **Frame type** (property pane) **Color**, **Height**, **Font** , **Frame** (drawing and view level)	Define the text color, font, font height, and frame type for the grid labels. To show more font options, click the **...** button.

Option	Description
Placing (property pane) **Text placing** (drawing and view level)	Select the locations where to show the grid labels by selecting the check boxes.  On the drawing and view level, you can enter the length of the grid line extension here.

Other ways to adjust grids in drawings

Additionally, XS_DRAWING_GRID_LABEL_FRAME_FIXED_WIDTH, XS_DRAWING_GRID_LABEL_FRAME_LINE_WIDTH_FACTOR, XS_GRID_TEXT_FONT, and XS_DEFAULT_FONT allow you to adjust the grid labels further. You can also customize grid labels (page 516).

See also

Define drawing grids (page 891)

Show grids in drawings (page 514)

Modify grids and grid lines in drawings (page 514)

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