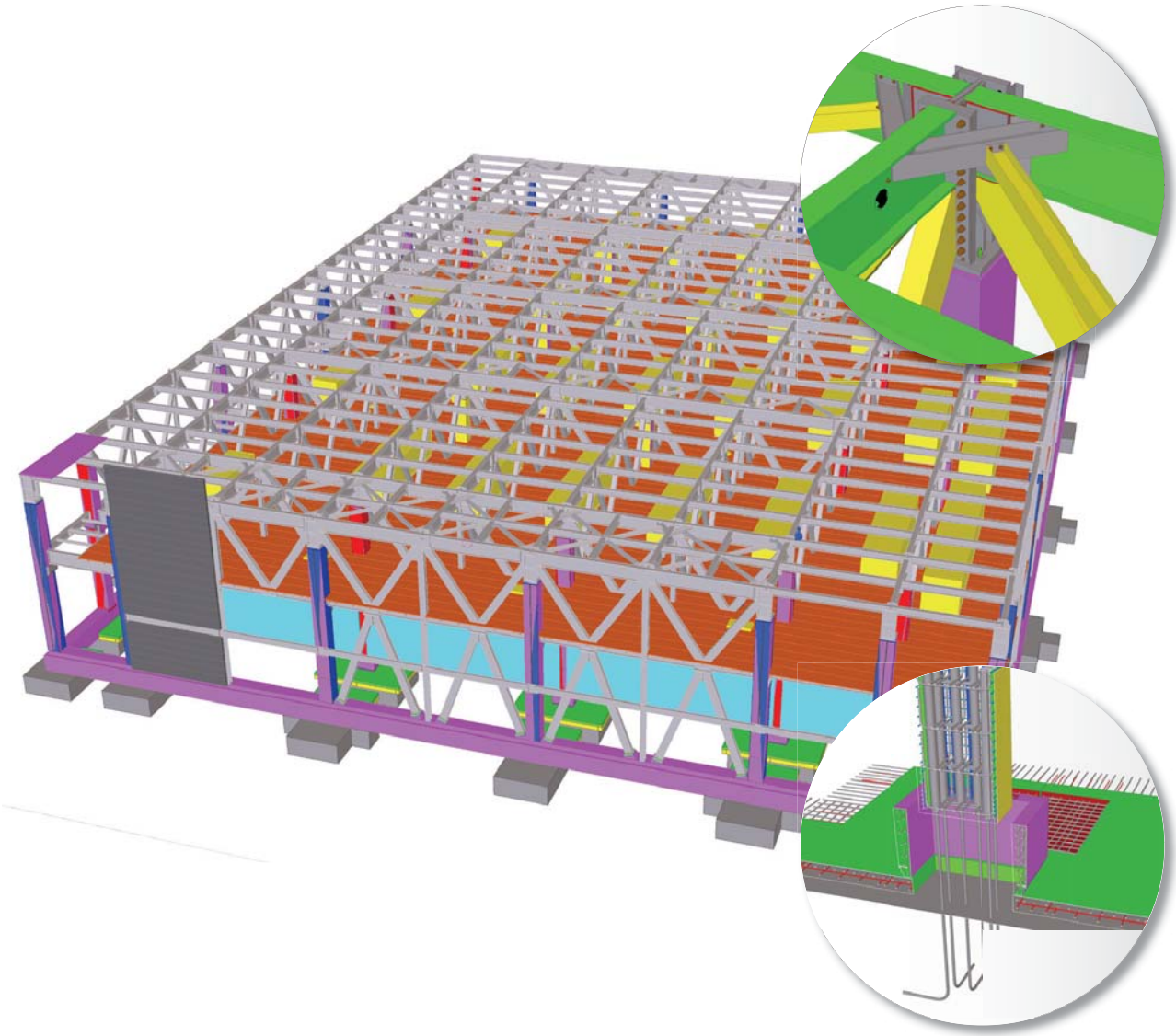
construsoft

TEKLA Structures

Full Detailing

Basic training Modeling



Productversion 13.1

May 2008

All rights reserved. No part of the contents of this manual may be reproduced or transmitted in any form or by any means wihout the permission of Construsoft Inc.

Construsoft Inc. is not responsible for any consequences as a result of using Tekla Structures.

© 2008 Tekla Corporation and its licensors. All rights reserved.

This Software Manual has been developed for use with the referenced Software. Use of the Software, and use of this Software Manual are governed by a License Agreement. Among other provisions, the License Agreement sets certain warranties for the Software and this Manual, disclaims other warranties, limits recoverable damages, defines permitted uses of the Software, and determines whether you are an authorized user of the Software. Please refer to the License Agreement for important obligations and applicable limitations and restrictions on your rights.

In addition, this Software Manual is protected by copyright law and by international treaties. Unauthorized reproduction, display, modification, or distribution of this Manual, or any portion of it, may result in severe civil and criminal penalties, and will be prosecuted to the full extent permitted by law.

Tekla, Tekla Structures, Xcity, Xengineer, Xpipe, Xpower, Xsteel, and Xstreet are either registered trademarks or trademarks of Tekla Corporation in the European Union, the United States, and/or other countries. Other product and company names mentioned in this Manual are or may be trademarks of their respective owners. By referring to a third-party product or brand, Tekla does not intend to suggest an affiliation with or endorsement by such third party and disclaims any such affiliation or endorsement, except where otherwise expressly stated.

Portions of this software:

2D DCM © 1989-2004 D-Cubed Limited. All rights reserved.

EPM toolkit © 1995-2004 EPM Technology a.s., Oslo, Norway. All rights reserved.

XML parser © 1999 The Apache Software Foundation. All rights reserved.

Project Data Control Library © 2006 - 2007 DlhSoft. All rights reserved.

DWGdirect, DGNdirect and OpenDWG Toolkit/Viewkit libraries © 1998-2005 Open Design Alliance. All rights reserved.

Analysis Engine included in Tekla Structures uses a program copyrighted by and is the property of Research Engineers International ©. All rights reserved.

FLEXnet Copyright © 2006 Macrovision Corporation and/or Macrovision Europe Ltd. All Rights Reserved. This product contains proprietary and confidential technology provided by and owned by Macrovision Europe Ltd., UK, and Macrovision Corporation of Santa Clara, California, U.S.A. Any use, copying, publication, distribution, display, modification, or transmission of such technology in whole or in part in any form or by any means without the prior express written permission of Macrovision Europe Ltd. and Macrovision Corporation is strictly prohibited. Except where expressly provided by Macrovision Europe Ltd. and Macrovision Corporation in writing, possession of this technology shall not be construed to confer any license or rights under any of Macrovision Europe Ltd. and Macrovision Corporation's intellectual property rights, whether by estoppel, implication, or otherwise.

Elements of the software described in this Manual may be the subject of pending patent applications in the European Union and/or other countries.

1 Tekla Structures 13.1 Modeling 1

1.1 Starting Tekla Structures 1
Noteboxes 1
Most common buttons 2
1.2 Creating a new model 3
1.3 Entering project properties 5
1.4 Opening a model 6
Saving a model with a different name 7
Open a multi-user model as a single-user model 7
Opening model from previous version 8
1.5 Views 8
General 8
Coordinate symbol 9
The toolbar View 9
Description of the icons 10
1.6 Creating a 3D view 11
1.7 Description view properties 12
1.8 Display 13
1.9 Creating grids 15
Creating grids 15
Creating a second grid 16
Creating a grid on a shifted work plane 17
Add single grid lines to an existing grid 17
1.10 Creating a top view 18
Creating a Top view at a level 19
Modify the Z co-ordinate in a plan view 19
1.11 Creating grid views 20
1.12 Create a view with 3 points 21
1.13 Creating the plane views along grid lines 22
1.14 Opening, closing and deleting views 24
1.15 Parts 25
Columns 25
Beams 28
Rollover highlight 28

Radius ... 31
1.16 Other profile types ... 32
Curved beam ... 32
Polybeam ... 32
Orthogonal beam ... 33
Twin profile ... 33
Contour plate ... 34
Modeling gratings and non-slip profiled plates ... 38
Profiles in drawings ... 42
Hide parts ... 43
Create a reference model ... 43
Updating reference models ... 48
Handling large reference models ... 48
Delete a reference model ... 48
1.17 Creating concrete parts ... 49
Concrete columns ... 49
Beams ... 51
Pad footings and Strip footings ... 52
Panels ... 52
Slabs ... 52
Parametric concrete profiles ... 53
Tab Deforming ... 54
1.18 Crossing selection ... 55
1.19 Snapping ... 56
Snap switches ... 56
Orthogonal snapping ... 57
Numeric snapping ... 57
Temporary reference points ... 58
Snap examples ... 59
1.20 Creating points ... 66
1.21 Detailing profiles ... 67
Fitting ... 67
Line cut ... 67
Part cut ... 68
Polygon cut ... 69

Batch jobs 71
1.22 Chamfer 72
1.23 Splitting and combining parts 73
Splitting 73
Combining 74
1.24 Placing bolts 75
Explanation of remainder of input fields in the dialog box Bolt properties 76
Slotted holes 77
Circular bolt patterns 78
Editing bolt patterns 78
Inserting a divided bolt pattern 79
Modeling bolts on cast-in embeds 80
1.25 Welds 81
1.26 Select switches 82
Explanation of the icons: 83
Selecting 85
Generating output 87
1.27 Work area 88
Fit work area by parts in selected views 88
Pick work area 89
Fit work area in selected views 89
Oversized work area in the model view 90
Hide the work area 90
1.28 Create clip planes 91
1.29 Cast-unit assemblies 93
1. Add to (concrete to concrete) 93
2. Add as sub-assembly (steel to concrete) 97
3. Weld (steel to steel) 100
4. Part Add (steel to steel) 102
Inquire Assembly for Cast Units 102
1.30 Modeling with concrete parts 103
Contact planes in cast units 103
Part Cut 104
Editing Cast Units 105
Pour side 106

Placing parts on a drawing 108
1.31 Work plane 110
Set work plane 110
Set work plane by 3 points 110
Set work plane to view plane 110
Set work plane to part top 111
1.32 Copy, translate and mirror 112
Mirroring an asymmetrical column 112
Copy 113
Copy / move with three points 116
Copy from model 117
Move 117
Rotate 118
Mirror 119
1.33 Inquire 120
Inquire object 120
Inquire assembly 121
Dimensions and angles 122
Center of gravity 122
1.34 General about custom components 123
Component types 123
Base plate detail 124
Availability of gebruikers componenten 125
1.35 Details, joints and components 126
Components, the main advantages 128
General function of details, joints and components 128
Changing conceptual components 129
Example: Base plate detail #1004 131
Storing component-settings 135
Create connection basis views 136
1.36 Concrete components 137
Battering (13) 137
Parts at beam (82) 138
Opening in wall (40) 140
Imbedded (1008) 143

Imbeds (8) 149
Mirroring elements containing components 150
Modeling stairs, balconies and platforms 150
1.37 Windbracings 152
Windbracings in a front plane 152
Windbracings in a straight roof plane 152
Windbracings in a skew roof plane 153
Shortening windbracings 156
Windbracing check 157
Shortening: how does it work? 158
Using Bolted gusset component 11 159
Examples 163
1.38 Phase manager 165
Phase representation in model views 167
Bolts in phases 168
1.39 Checking the model 169
Clash Check 169
Bolt Clash Check 170
Using Clash Check for Cast Units containing imbeds 170
Fly 172
Delete “Beep” from a model 172
Check with help of reports 173
1.40 Profile catalog 174
Editing profiles in the catalog 176
Removing profiles from the catalog 176
Adding profiles to the catalog 177
1.41 Using parametric user profiles 178
Selecting a parametric profile 179
Sketching parametric profiles 179
1.42 Material database 180
1.43 Hints & tips 182
Tekla Structures Help 182
Standard documents 185
Construsoft Xtranet 185
1.44 Component examples concrete 186

Concrete foundation (1030) 186
Precast found block (1028) 187
Concrete console (110) 188
Concrete console (111) 188
Wall wall teeth (12) 189
Column - beam (14) 190
Grating penetration (92) 190
Concrete beam - beam (112) 191
Anchor (10) 191
Column - 2 panels (12) 192
Parts at beam (82) 192
Tube in concrete column (83) 193
Embedded anchors (8) and Embedded (1008) 194
Seam anchors (31) 195
Opening in wall (40) 195
Foot plate for steel column (29) 196
Stairwells and elevation shafts (90) 196
Corbel connection (14) 197
Seating with dowel (75) 197
Two sided seating with dowel (76)) 197
Battering connection (13) 198
Concrete stairs (65) 198
1.45 Component examples steel 199
End plate (144) 199
Cranked beam (41) 200
Haunch (40) 201
Joining plates (14) 201
Base plate (1004) 202
Foot plate (1029) 202
Stiffeners (1003) 204
Stub (1011) 204
End plate (1002) 204
Stanchion weld (85) 205
Manlock column (1032) 206
Manlock beam (1033) 207

Array of objects (29) 208
1.46 Custom Component examples 209
1.47 Reserved shortcuts in Tekla Structures: 209
1.48 Notes: 212

1 Tekla Structures 13.1 Modeling

1.1 Starting Tekla Structures



Before starting Tekla Structures, log on to the system. Now double-click on the Tekla Structures icon. Tekla Structures will be started.

We use different typefaces for different items in this training:

- **Bold**, you have to click with the mouse.
- Any text in *Italic* appears on the monitor.

Noteboxes

We use several types of noteboxes, marked by different icons. Their functions are shown below:



A **Tip** might introduce a shortcut, or suggest alternative ways of doing things. A tip never contains information that is absolutely necessary.



A **Note** draws attention to details that you might easily overlook. It can also point you to other information in this training that you might find useful.



You should always read very **Important notes and warnings,** like this one. They will help you avoid making serious mistakes, or wasting your time.



This symbol indicates **advanced or highly technical information** that is usually of interest only to advanced or technically-oriented readers. You are never required to understand this kind of information.

Most common buttons

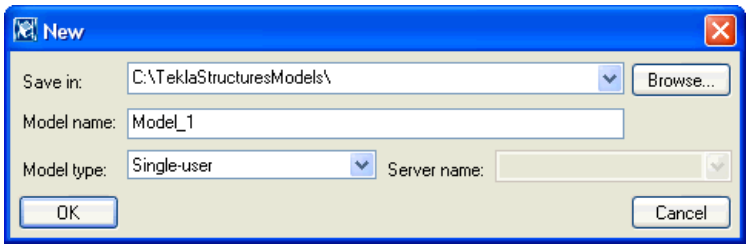
Most of the Tekla Structures dialog boxes contain common buttons. These buttons mean the same thing in all dialog boxes. The common buttons are:

Button	Description
Save	Stores the modifications of properties. Tekla Structures saves the properties in the file shown in the list box.
Load	Loads all previously stored properties to the dialog box. Tekla Structures also loads the properties of sub dialog boxes, even if they are not open. You can select the name of the file you want to use from the list box.
Save as	Stores the dialog box properties with the name given in the field. Tekla Structures stores the properties files in the model folder. The files also include the properties in sub dialog boxes. The file extensions are different for dialog boxes.
OK	Retains the properties in the dialog box and closes the dialog box. Tekla Structures uses these properties the next time you create an object of this type.
Apply	Stores the modifications of properties. Tekla Structures saves the properties in the file shown in the list box.
Modify	Modifies the selected object(s) using the properties in the dialog box, but does not retain the properties in the dialog box.
Get	Fills the dialog box with the properties of the selected object. When you select several objects, Tekla Structures takes the properties at random from one of the selected objects.
☑ / ☐	Toggles all the modify filter checkboxes in the dialog boxes on and off.
Cancel	Closes the dialog box without retaining the properties in the dialog box or modifying objects.

1.2 Creating a new model



In the pull-down menu go to **File** > **New...** or click on the icon **New**. The following dialog box appears:



Model name

Replace the name **New model** in the field **Model name**. Enter: **Model_1.**

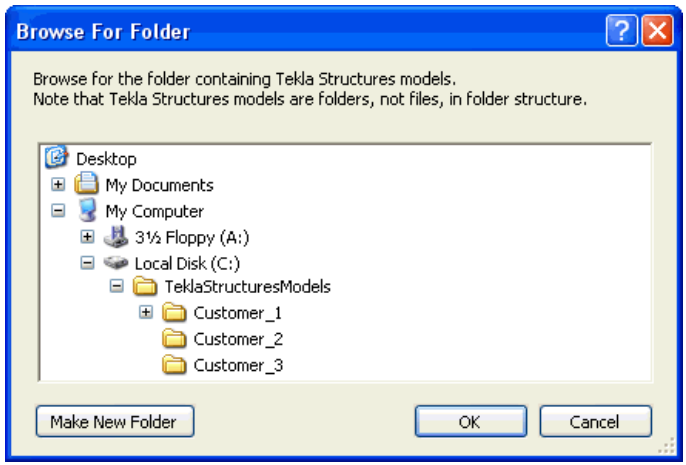


Do not use special characters (/ \ . ; : |) in model names!

Model location

Enter the location for the model in the field **Save in**, or accept the default location, *disk:\TeklaStructuresModels*.

Click the **Browse** button to open the **Browse for folder** dialog box, where you can browse for a folder or create a new folder. You can, for example, create new sub-directories to save the models per principal:



Single/multi-user

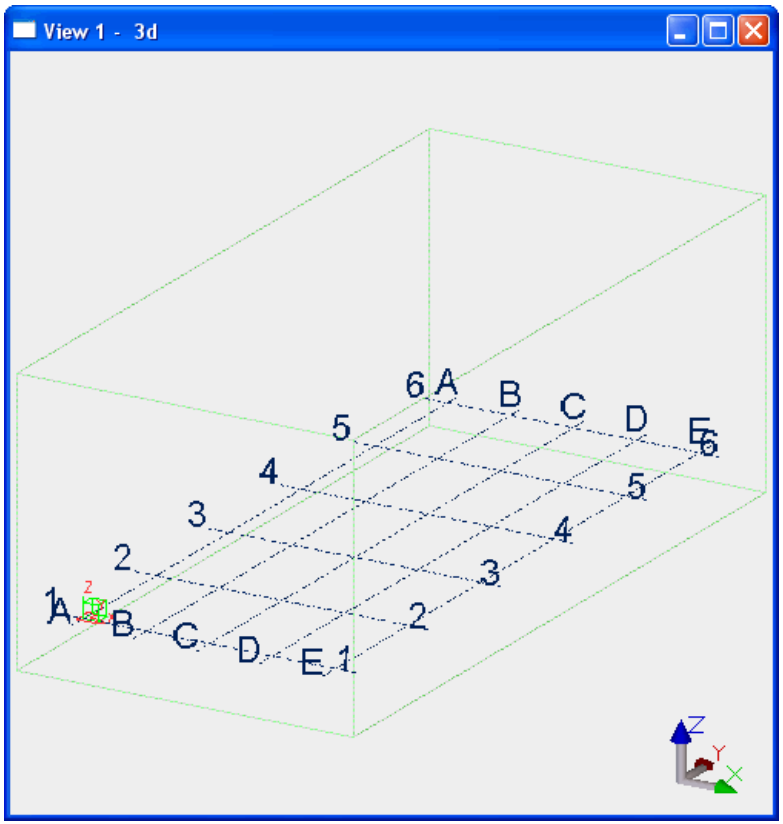
Use the **Model type** field to define whether the model is a single-user or a multi-user model, by default the model type is set to single-user. For a multi-user model, enter the name of the server in the **Server name** field.

See also

For more information about working in multi-user, see **Tekla Structures.pdf** in the folder: **disk:\TeklaStructures\version\environments\europe\pdf.**

Click <**OK**>

A 3D view is generated using the standard view properties:



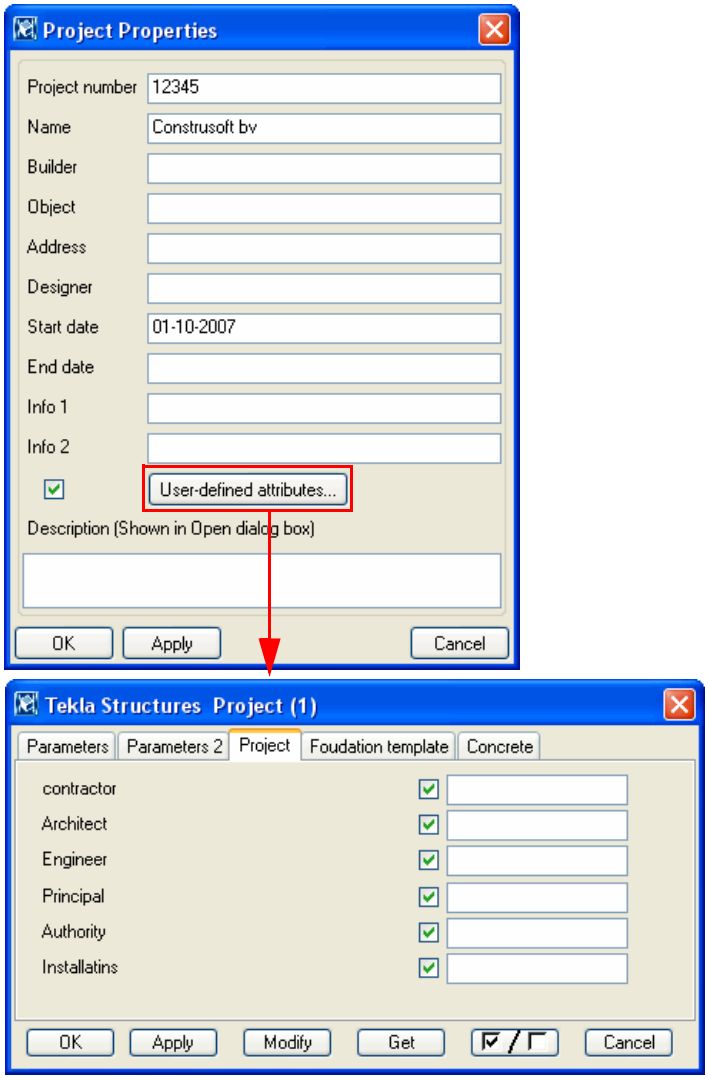


Save the model once to make sure that the model name is correct.

1.3 Entering project properties

Enter the project information at the beginning of a project to make reports and drawings to display the correct information automatically.

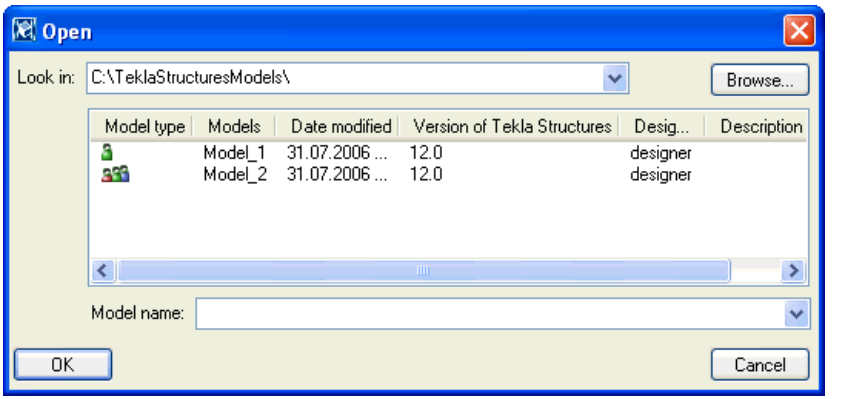
To enter the project information, click **Properties** > **Project...**. The Project properties dialog box appears. Tekla Structures displays this information in different reports and drawing titles.



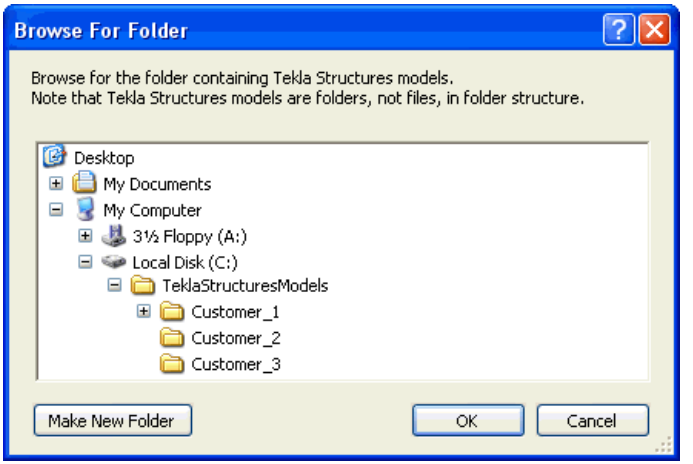
1.4 Opening a model



To open a model go to the pull-down menu **File > Open...** or click the icon. The dialog box **Open** appears:



Select a model in the list and click <**OK**> to open the model (or double-click on the model name in the dialog box) or click on the button **Browse...** to open the dialog box **Browse for folder** to browse for a model:



Select a folder and click <**OK**>. The dialog box **Open** displays the models which are in the selected folder. Now select a model in the list and click <**OK**> or double-click on the model name in the dialog box to open the model.



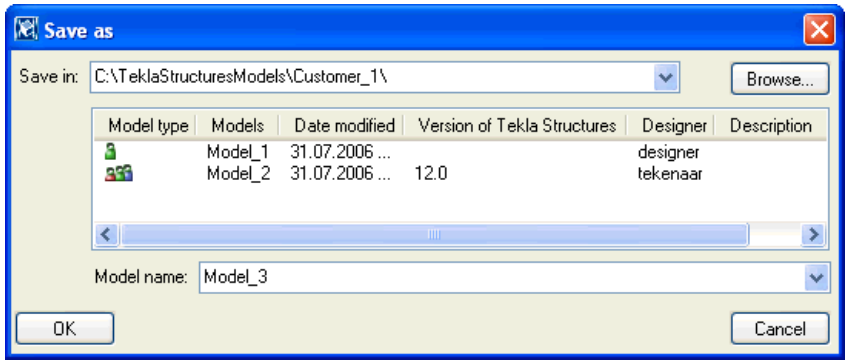
The information in the **Designer** and **Description** columns comes from the **Project properties** dialog box.

Last opened model

The 10 last opened models are automatically preserved so that you can open them easily in the dialog box **Open**. After restarting Tekla Structures the model name of the last opened model is already entered in the field **Model name**.

Saving a model with a different name

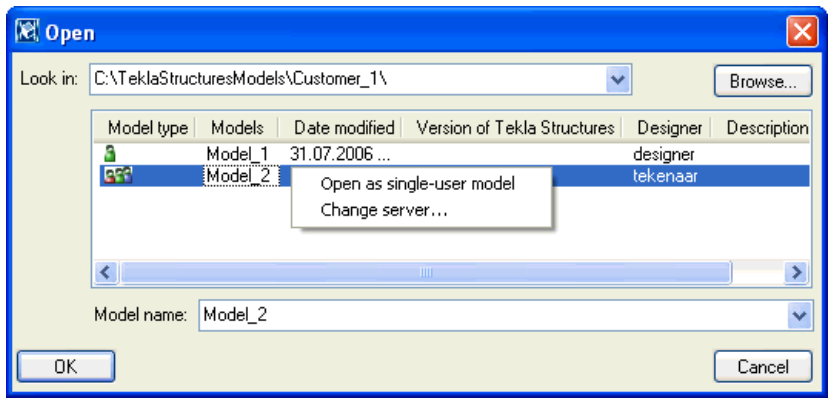
To save a model with a different name, click **File** > **Save as...** and enter a name in the **Model name** field. Click <**OK**>.



Open a multi-user model as a single-user model

Models are indicated in the row **Model type** as a single-user or a multi-user model.

Select a multi-user model in the **Open** dialog box and right-click, select the **Open as single-user model** option.

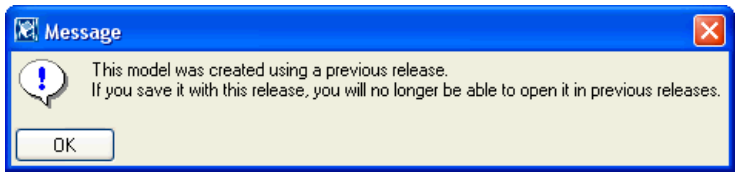




When you open a model or create a new model, Tekla Structures now lists the last saved models in the **Open** and **New model** dialog boxes in the field **Look in**.

Opening model from previous version

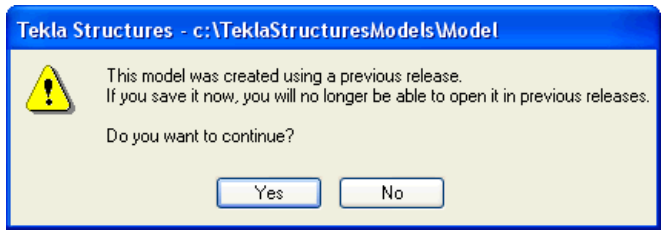
If a model is opened which is created in a previous version of Tekla Structures, a message dialog box appears. It states that saving the model in the current version will cause that the model can not be opened anymore in the version where is it initially created in:



Click <**OK**>. The model will be opened.

If you close Tekla Structures, or opening another model without saving the current model, it remains possible to open and edit the model in the version where it is initially created in.

If you click the button  , the following message will appear:



Now, if you click <**Yes**>, the model will be saved and it can **not** be opened and edited anymore in its original Tekla Structures version.

If you click <**No**>, the model will not be saved and it remains possible to open the model in its original version.

1.5 Views

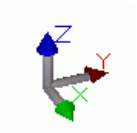
General

When you start modelling in Tekla Structures you first have to create views. A view represents a model from a point of view. There are several types of views you can create in Tekla Structures.

For example, you can create views:

- of the entire model (e.g. a 3D view)
- of selected part(s) and component(s) (e.g. a part basic view or a component basic view)
- along the grid lines (axis views)

Coordinate symbol

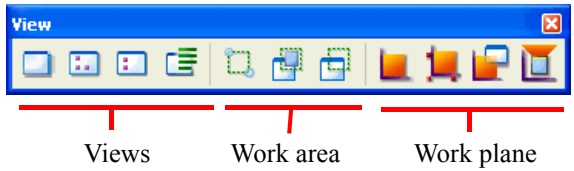


The coordinate symbol indicates the direction of the model. It is located in the bottom right corner of the model view. The coordinate symbol follows the work-plane.



The coordinate symbol is **only** visible in rendered views, not in wire views

The toolbar View



Description of the icons

	Create basic views, see **Creating a 3D view** (p. 11)
	Create a view with three points, see **Create a view with 3 points** (p. 21)
	Create a view with two point, see **Creating grid views** (p. 20)
	Open the list of named views, see **Opening, closing and deleting views** (p. 24)
	Pick work area, see **Pick work area** (p. 89)
	Fit work area by parts in selected views, see **Fit work area by parts in selected views** (p. 88)
	Fit work area in selected views, see **Fit work area in selected views** (p. 89)
	Sets the work plane parallel to xy, xz, or yz plane, see **Set work plane** (p. 110)
	Sets the work plane according to three picked points, see **Set work plane by 3 points** (p. 110)
	Sets the work plane to be equivalent to the view plane of a picked view, see **Set work plane to view plane** (p. 110)
	Sets the work plane on the front, top, back, or bottom plane of a picked part. The icon is for the top, see **Set work plane to part top** (p. 111)

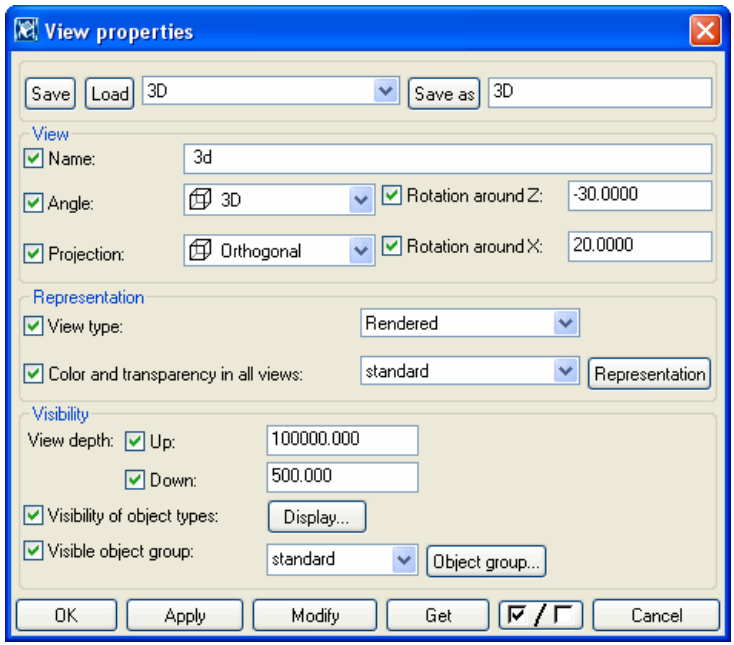
1.6 Creating a 3D view



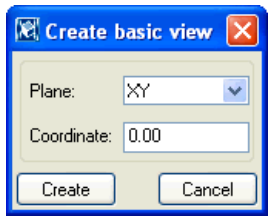
To create a 3D view, double-click on the icon **Create basic view**.

In the dialog box **View properties** select the setting **3D** in the list as shown below in the dialog box and click <**Load**>.

Now click <**Apply**>



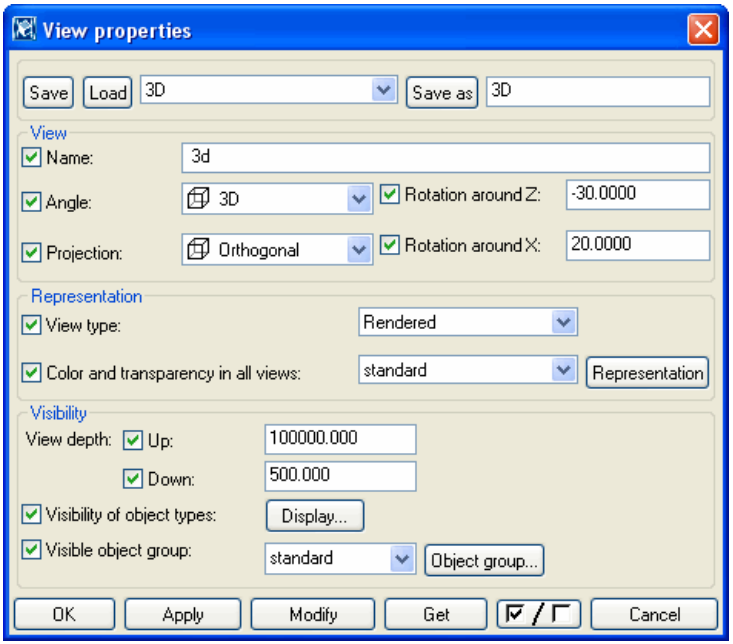
In the dialog box **Create basic view,** for the level coordinate enter **0.000**. Now click <**Create**> to create the 3D view.





The View depths **Up** and **Down** define the thickness of the displayed slice from the view plane in the model. The view plane level is the coordinate defined in the dialog box **Create basic view**.

1.7 Description view properties



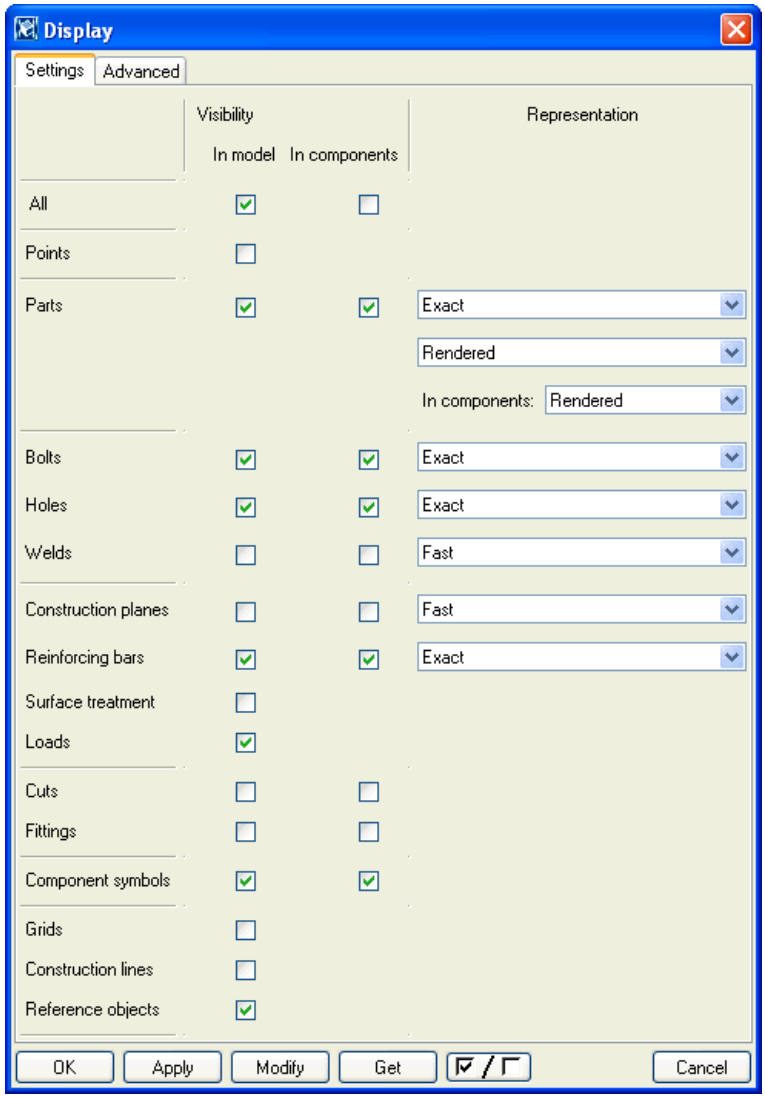
Field:	Description:
Name:	Name of the view
Angle:	Switch between Plane and 3D (shortcut <**Ctrl**> + **P**)
Projection:	Only available in a **Rendered** View type. The options are **Perspective** and **Orthogonal**
Rotation: Around Z:	Rotation around the z-axis. Standard is –30
Rotation: Around X:	Rotation around the x-axis. Standard is 20
View type:	Defines the appearance of the view. The options are: Wireframe and Rendered
Color and transparency:	See also **Phase representation in model views** (p. 167)
View depth: Up:	The view depth upwards from the view plane
View depth: Down:	The view depth downwards from the view plane
Visibility of object types:	See also **Display** (p. 13)
Visible object group:	See also **Select switches** (p. 82)

1.8 Display

This dialog box contains two tabs.

Settings

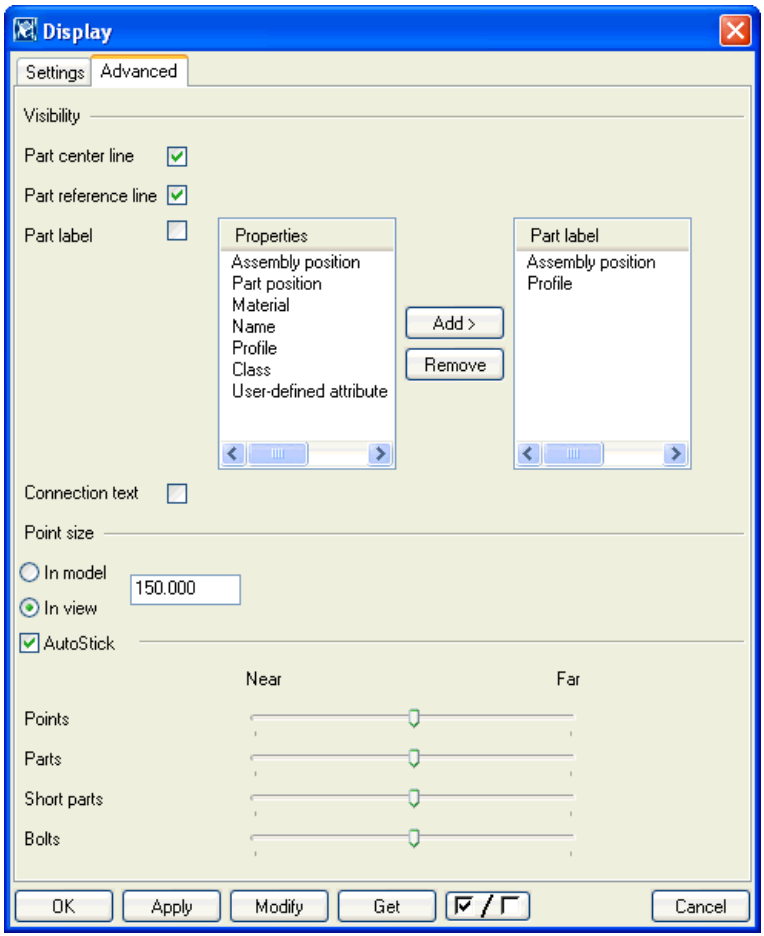
In this tab you can define the visibility for several objects. You can also define the representation of model parts and connection parts, you have 5 options: wire frame to rendered.



You can now select all the checkboxes at the same time on the **Display** dialog box by selecting **All** (**In model** and/or **In components**). The uppermost check-boxes are also automatically cleared if you clear any of the checkboxes under them.

Advanced

In this tab you can toggle the parts' center- and/or reference lines on or off. You can also define the point size and in the field **Part label** you have the option to display selected part properties, user-defined attributes, and template attributes:





Sometimes the work area in the model can be huge because, for example, points are positioned far away from the models' origin. To display these points and to delete them, change the **Point size** to 1000 or 1500 for instance.

Autostick

Autostick is a view setting when you are zooming in a model. By dragging the sliders you can define at which distance Tekla Structures displays objects as sticks instead of exact objects. This is only available in wire frame views.

1.9 Creating grids

In Tekla Structures you can create entire grids and single grid lines. You modify, copy, and move both.

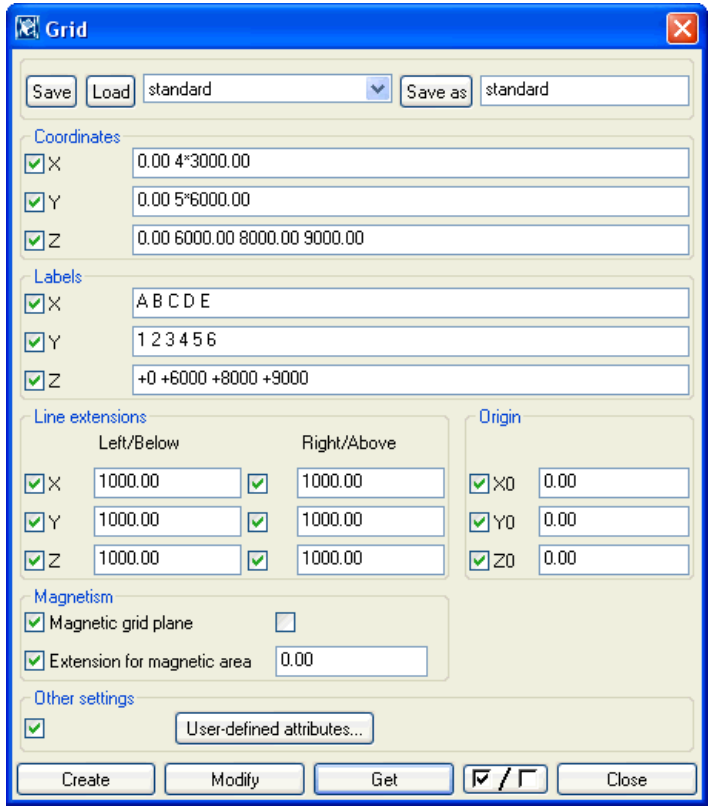
Creating grids



Click the icon **Create grid** in the toolbar **Points** or click **Points > Grid...**

You can adjust the grid properties by entering the X, Y en Z coordinates and the names of the labels of the grid lines:

- Coordinates in the X- and Y-direction are *relative* distances
- Coordinates in the Z-direction are *absolute* distances





Use a zero at the start to represent a grid on the 0,0 coordinate and spaces as separators for coordinates.

To create the grid, click <**Create**>. Click the button <**OK**> to close the window.

Initially, a small grid appears in the bottom left corner of the view. To enlarge the grid, click in the view to activate it. Then click **View > Fit work area.**

Magnetism

You can make use of the options **Magnetic grid plane** and **Extension for magnetic area** to link profiles to the grid to adjust the profiles automatically if the grid changes.

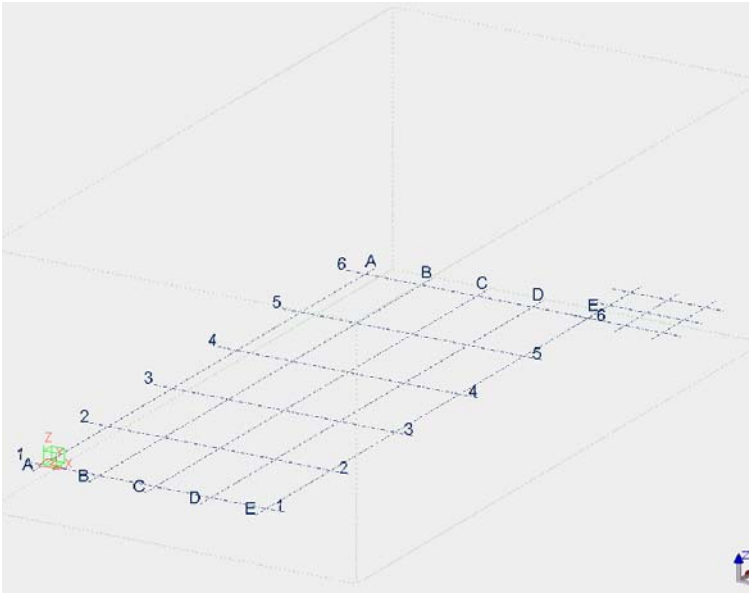
Other settings

You can "lock" the grid, if somebody else opens the model to modify the grid, a message appears.

Creating a second grid

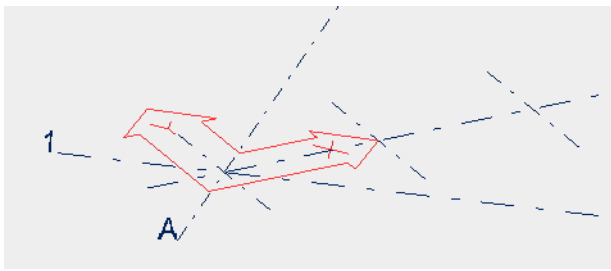
Click the icon **Create grid** in the toolbar **Points** or click **Points > Grid...**

To create a second grid, Tekla Structures prompts you to pick a point in a view, e.g. E-6. The coordinates of the picked point appear in the **Grid** dialog box as **X0, Y0** and **Z0**. Enter the coordinates and the labels and click <**Create>.**



Creating a grid on a shifted work plane

The current work plane defines the local coordinate system of a Tekla Structures model. You can shift the current work plane to any position by picking 3 points, which will be explained later on in this training. The current work plane is model-specific, so it is related to all views.

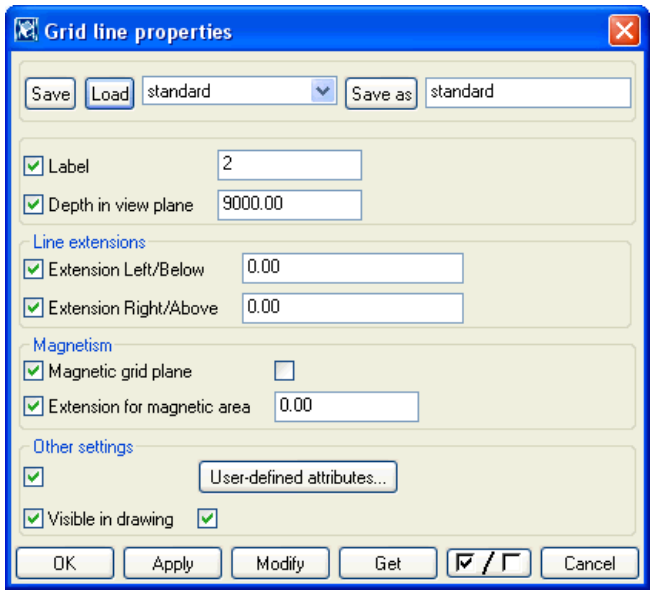


See also For more information about the workplane, see **Work plane** (p. 110)

Add single grid lines to an existing grid



Click the icon **Create grid line** in the toolbar **Points** or click **Points > Grid line.** The following dialog box appears:



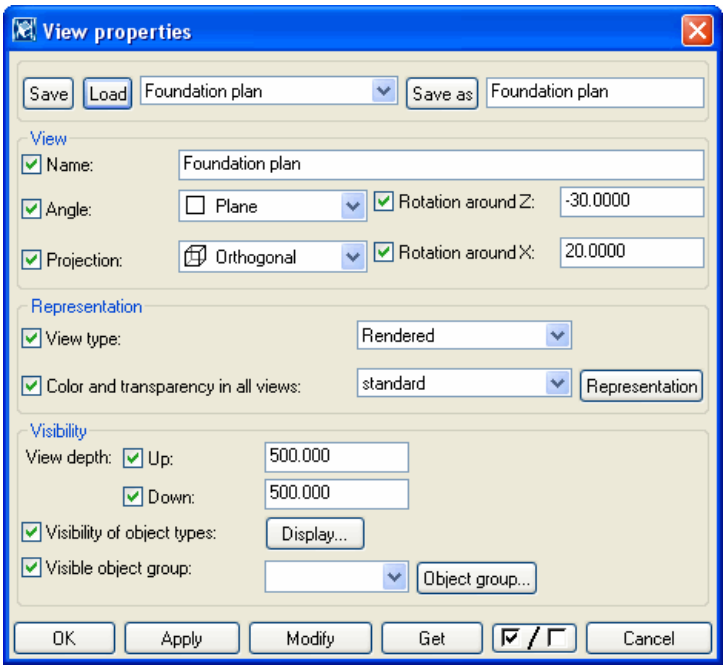
If necessary, adjust the grid properties. Now click <**Apply**> or <**OK**> and select the grid in the model view. Pick the first and the second position for the grid line. The single grid line is now added to the grid.

1.10 Creating a top view



To create a top view, double-click on the icon **Create basic view**.

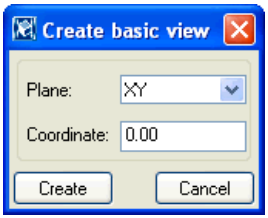
In the list box, select the setting **Foundation plan**. Click <**Load**>. The anchor plan view settings are defined. Click <**Apply**>.





The View depths **Up** and **Down** are changed to 500 which is the thickness of the displayed slice from the view plane in the model.

In the dialog box **Create basic view**, for the level co-ordinate enter “0.0”. Then click <**Create**> to create the view.

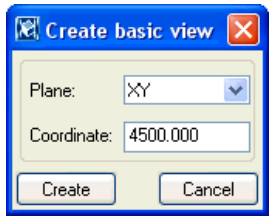


Creating a Top view at a level

Create a top view for a floor at a level of +4500.

Enter the name in the dialog box View properties, e.g. *Floor 4500+* and click <**Apply**>.

Enter co-ordinate “4500” in the dialog box **Create basic view** and click <**Create>.** Repeat this for a top view with a level of 10.000.



Modify the Z co-ordinate in a plan view

You can modify the Z co-ordinate in a plan view, so the view plane in the XY direction.

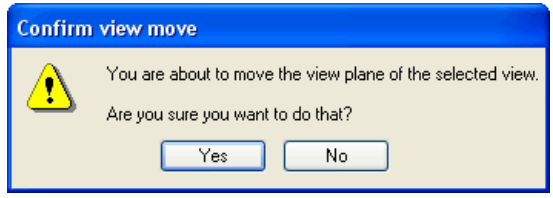
For this you select the view, a red colored frame appears at the edge of the view. The view is active now.



Click the right mouse button and select **Move** > **Translate...** in the popup-menu or click the icon as displayed on the left.

Enter the desired value in the dialog box **Move - translate**, i.e. 1700.

Now click <**Move>,** the following dialog box appears:



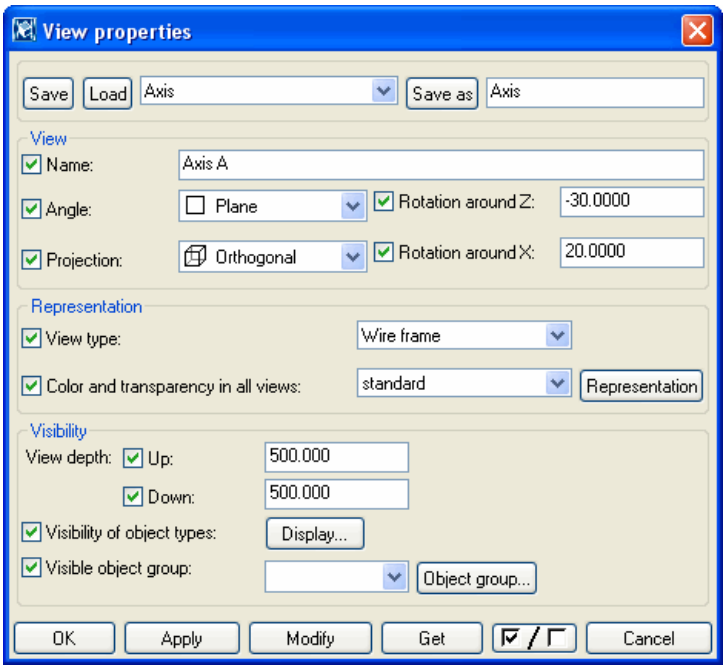
Click <**Yes**>.

Then double-click in the view to open the dialog box **View properties**. You can now change the view’s name, for example *Floor 6200+*.

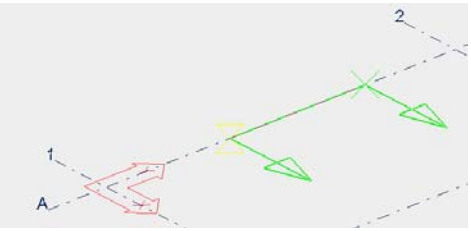
1.11 Creating grid views



Create the grid views from the grid lines **1** and **A**. Click the **Create view by two points** icon. **Load** the setting *Axis*. You don't have to click <**OK**>, <**Apply**> is sufficient. The name of the view is **AX A**. You can change the name before creating the view or afterwards.



Now pick the first and the second point on grid line A, two arrow symbols appear:



These symbols indicate the direction of the view.

Change in the dialog box **View properties** the name in **AX 1**. Click <**Apply**> and pick two points on grid line 1.

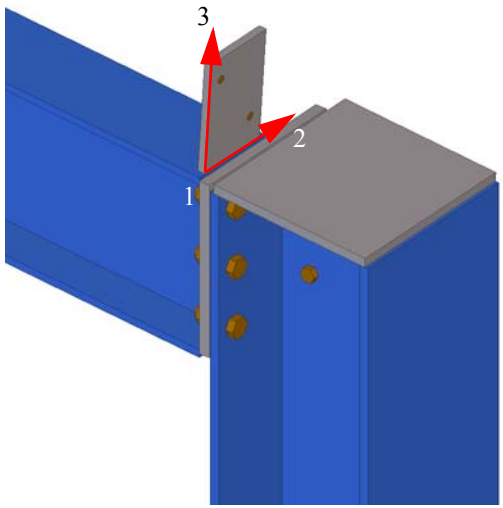
To change the name of the view afterwards, double-click the concerned view. Change the name and click <**Apply**> and <**OK**>.

1.12 Create a view with 3 points

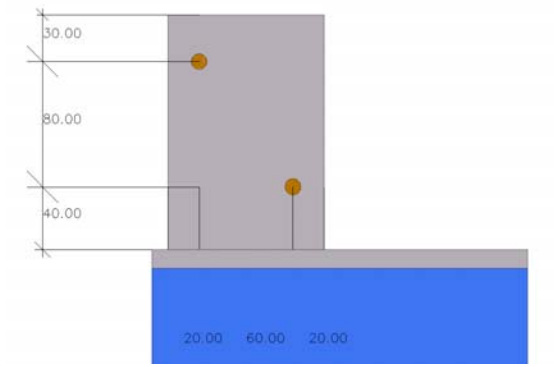


You can use this command to create e.g. views perpendicular to a part's plane.

- Double-click the icon **Create view by three points**.
- **Load** the setting *Axis*.



- Now pick three points at the stub, the first point you pick is the origin of the view plane, the second point indicates the direction of the X axis and the third point, the direction of the Y axis.

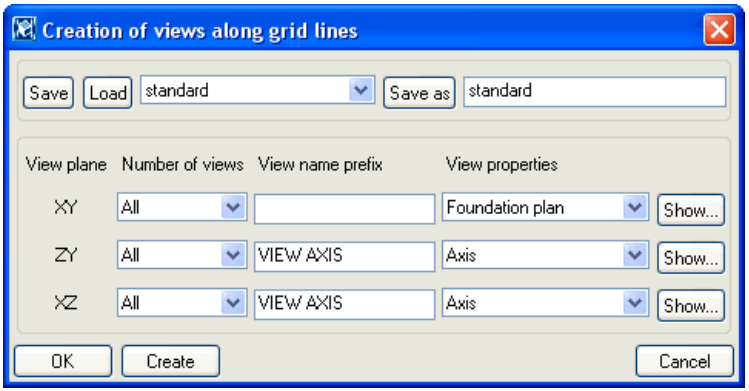


1.13 Creating the plane views along grid lines

Use this command to create all views along the grid lines of a selected grid. There are two options:

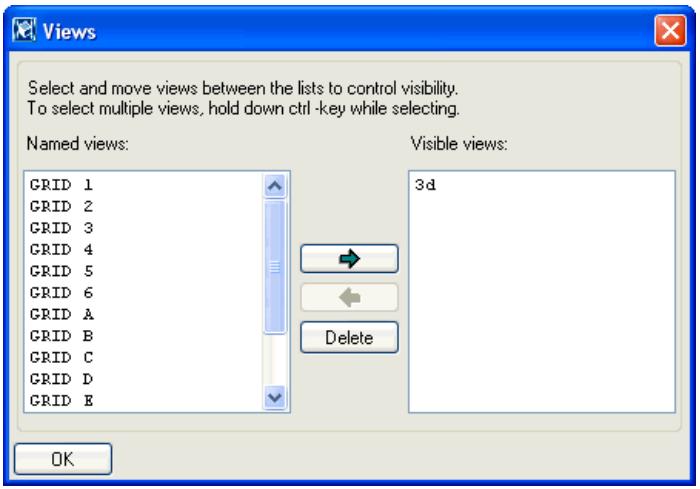
1. In the pull-down menu, click **View > Create view > Grid views...**
2. Select the grid, click right-mouse and select **Create view** > **Grid views...** in the contextmenu.

The following dialog box appears:

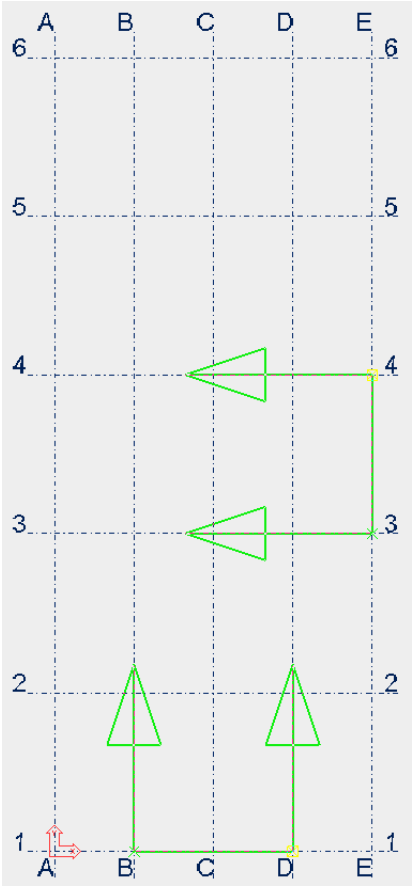


In this dialog box you can define several settings, e.g. the number of views, the view name prefixes, etc.

Select, if necessary, the grid in the model and click <**Create**>. The created views will not be visible until you move them to the visible views in the dialog box ***Views***.



All grid views direction is “into” the model:



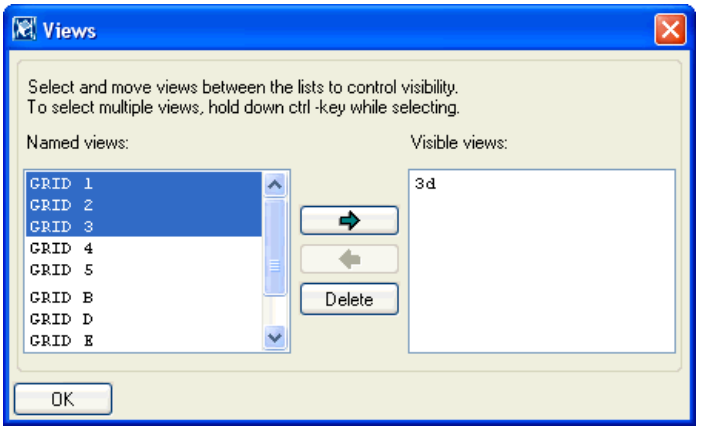
The direction of the last view (**AXIS 6** en **AXIS A**) is outside the model. You can delete these two views and create them manually.

See also For more information, see **Creating grid views** (p. 20)

1.14 Opening, closing and deleting views



To open, close and delete views, click the icon, click **View > Named views** or use shortcut <**Ctrl**> + **b** to display the dialog box **Views.**



To display or to hide views, select the view(s) and use the arrows between the lists. To select several views, hold down the left mouse button, then drag down.



Press <**Ctrl**> + <**Tab**> to switch between views.

Bringing a view on top

Now when you select a view in the **Visible views** list in the **Views** dialog box, the selected view moves to the top. If you have selected multiple views, all selected views will be moved up, with the first one on the list on top. Also, when you select a view in the model, it is automatically selected in the **Visible views** list.

To delete a named view, select the view and click **Delete.**



The number of existing named views is unlimited, the maximum of views on the screen is nine.

1.15 Parts

Parts are created by using the icons in the **Steel** toolbar.



As an alternative for the icons, the command's (column, beam, etc.) are also accessible through the pull-down menu **Parts**.



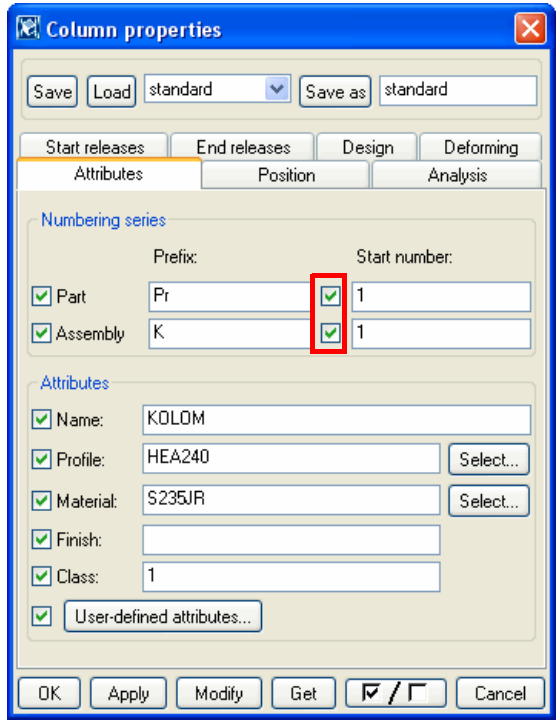
If you want to adjust the profile properties out of the pull-down menu before you create the profile, keep the <**Shift**> key pressed down and select one of the profiles in the pull-down menu; the dialog box will be opened.

Columns



To create columns, first double-click the **Column** icon.

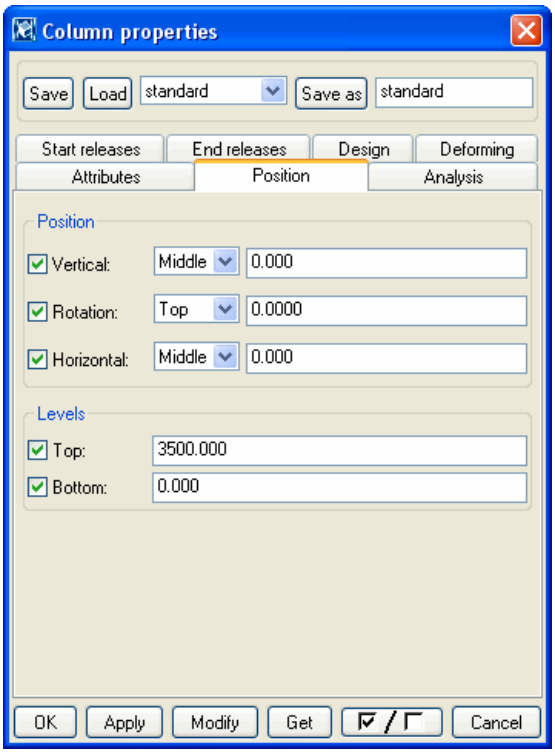
Fill in the dialog box **Column properties** as displayed below, next click <**Save>.**



Modifying numbering series

You can modify the profiles' numbering series by selecting the checkboxes separately for **Prefix** and **Start number** in the dialog box:

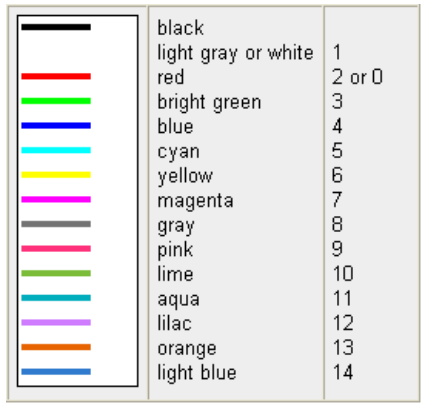
Because of this, you can modify start numbers for beams and columns, who have different prefixes, at once.



See also

For more information about overlaps in the numbering, see the file **Tekla Structures.pdf** in the folder **disk:\TeklaStructures\version\environments\europe\pdf**.

In the 3D view, pick the intersection of grid lines A-1 to create the column. Go to **Supplement 1** in the back of this manual and places the columns as indicated.

Explanation of the inputs in the Column dialog box	
Part	The Prefix of the part (Part number) is set to “Pr” and the Start Number is set to 1. After numbering the model, a possible Part number for this column can be Pr25. If the Start Number is set to “100”, the smallest number a part can get is Pr100
Assembly	The Assembly Prefix is set to “K” and the start number is set to 1. After numbering the model, a possible Assembly number can be K17 for the column
Name	The user-definable name of the column can be entered here
Profile	The profile is a HEA240. It is also possible to enter HE240A. Click **Select** to open the catalog containing all profiles.
Material	The material is set to “S235JR”. Also for the materials a catalog is present where a material can be picked
Finish	A user-definable finish can be entered here. This text can be displayed on drawings and reports
Class	To set the color of the parts in the model views: 
Vertical Position	The vertical position of the column, relative to it’s reference point. Possible options are Up, Middle and Down
Rotation	How the profile is rotated around it’s axis
Horizontal Position	The horizontal position of the column, relative to it’s reference point. Possible options are Left, Middle end Right
Bottom level	The position of the 1st end of the column. Input in millimeters
Top level	The position of the 2nd end of the column. Input in mm
User-defined attributes	This button opens a dialog box containing the User-defined attributes, use right mouse click to open.

Beams

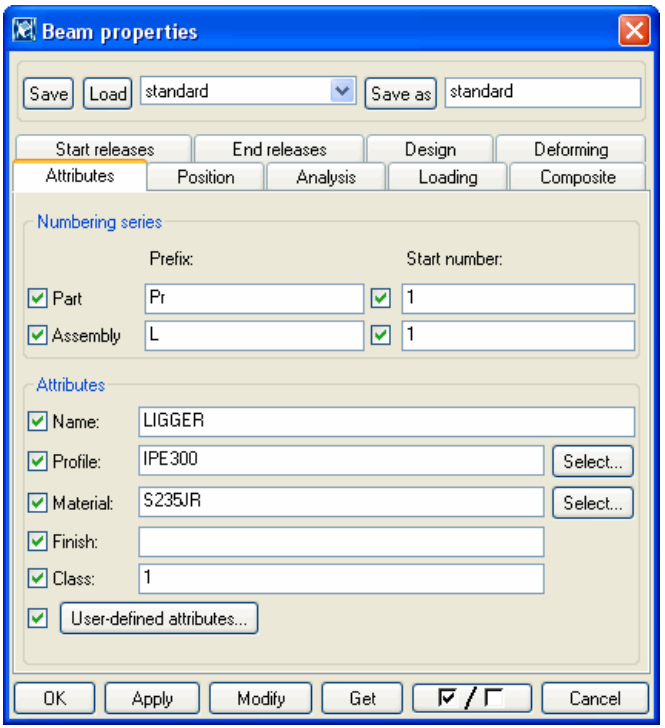


To insert a beam, first double-click the **Beam** icon.



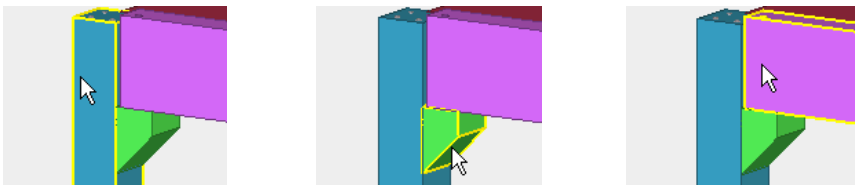
Beams, compression bars, windbracings, plates, tubes, etc. can all be placed with this command.

In broad outlines, the dialog box **Beam properties** corresponds to the dialog box **Column properties**. Only the options in tab **Position** vary.



Rollover highlight

The rollover highlight makes it easier to select objects, by highlighting each part in yellow as the pointer moves over it in rendered views.



Activate

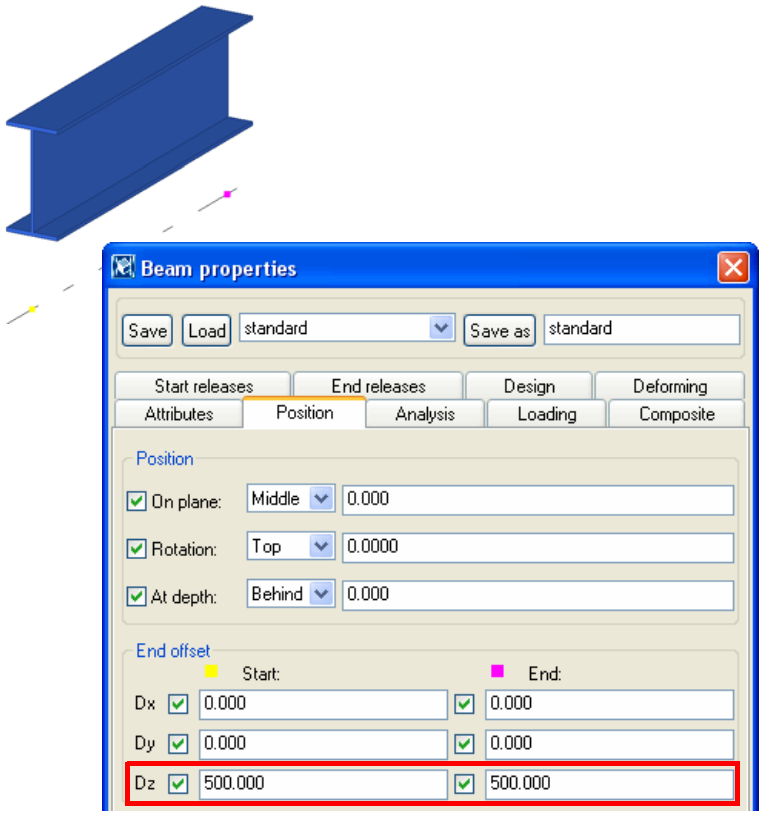
To switch the feature **Rollover highlight** on, click shortcut **H** or go to the pull-down menu **Setup** > **Rollover highlight.**

Moving start- and endpoints

When a part is selected, the 1st reference point will be highlighted in yellow and the 2nd reference point will be highlighted in magenta. Next, both ends can be moved.



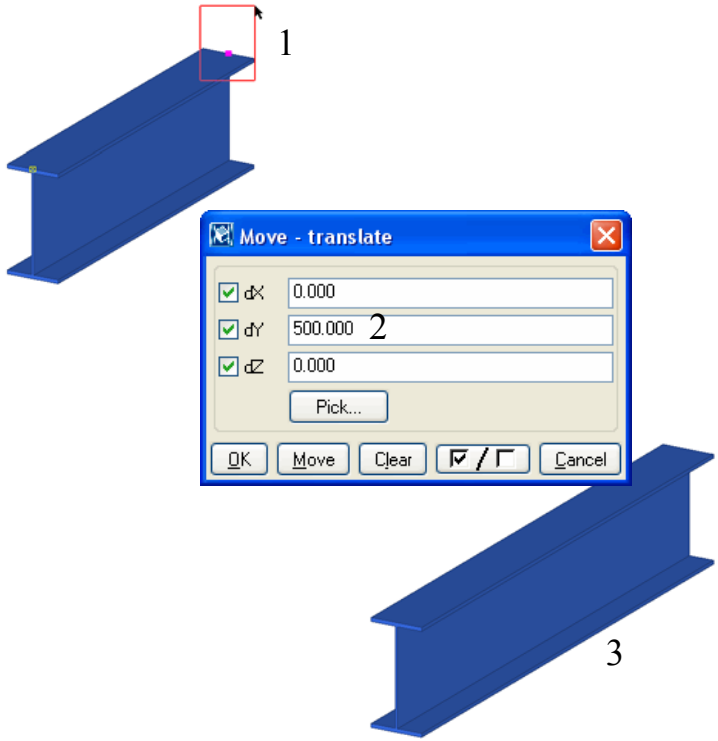
Preferably, don't use the Dx-, Dy- en Dz options in the **Position** tab of the dialog box for moving the profile-ends.



There are 2 better ways to do this:

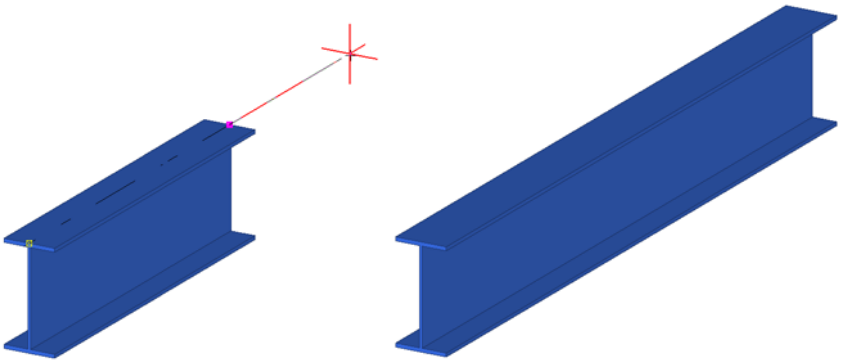
1. Select the profile which needs to be extended/ shortened. The reference points will be highlighted: the 1st reference point will be highlighted in *yellow*, the 2nd reference point will be highlighted in *magenta*.

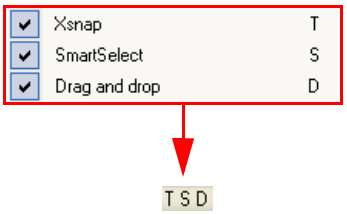
   Hold down the <**Alt**> key and select the parts again by clicking and dragging the mouse from left to right to define a window selection (1) and moving the point (2) with the **Move > Translate** command. The profile will be extended or shortened (3).



2. Select the profile which has to be extended/ shortened. The reference points will be highlighted: the 1st reference point will be highlighted in yellow, the 2nd reference point will be highlighted in *magenta*.

3. Now, by selecting a reference point and keeping the left mouse button pressed, the reference point can be moved **(Drag and drop).** The profile will than be extended or shortened.



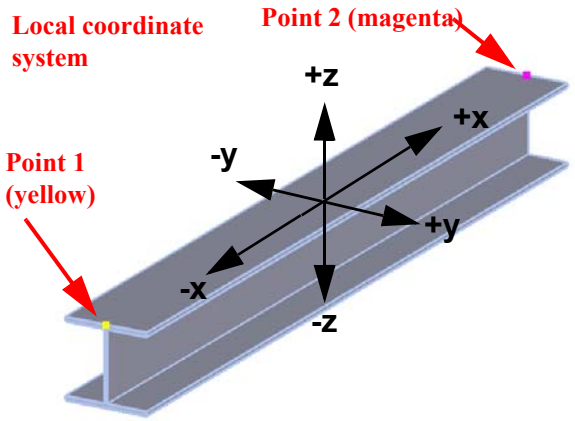


Remind that the *Drag and Drop* setting (**Setup > Drag and drop**) is checked.

In the Tekla Structures view, this will be displayed with a “D”-character in the status bar.

This also counts for the *XSnap* (T) and *SmartSelect* (S) commands

To know in which direction an profile-end is moved to, it is important to know how the local coordinate system of a profile is set up.



Radius

A straight-lined profile can easily be changed towards a bended or curved profile. This can be done by setting a value for “Radius”. In which plane the profile has to be bend (XY-plane or XZ-plane) can also be set.

No. of segments

If a radius is set, then *by number of segments* can be defined in how many straight lines the beam will be built up.

1.16 Other profile types

Curved beam



A curved beam is a beam which runs through 3 points. The dialog box corresponds with the **Beam properties** dialog box. The radius of the beam is automatically calculated on the basis of the 3 picked points.

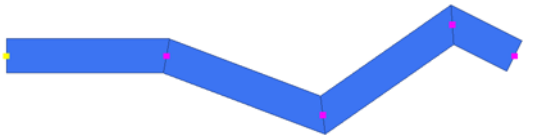


It is not possible to generate NC-files of curved beams. Use polybeams instead.

Polybeam



A polybeam is a profile which runs through several points.



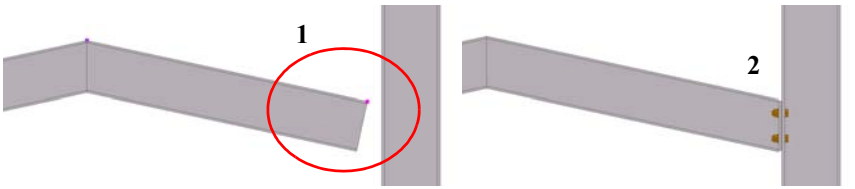
In addition, each bend has a handle. The purpose of this handle is to add a chamfer to a curved segment. After double-clicking the handle, the dialog box **Chamfer properties** appears. In here you can set the size and the shape of the chamfer.



Do not enter any values for the radius or the number of segments in the polybeam properties, it will effect the output in a negative way! For curved beams you can enter values for the radius or the number of segments.

NC files

While modeling polybeams, always model the polybeams a little bit shorter (1), to fit them afterwards or to connect them by using a component (2):

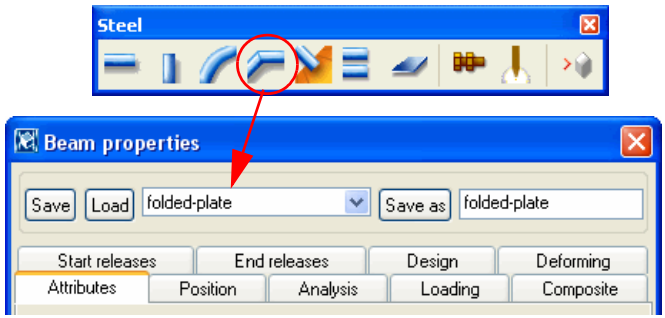


Only in this way you will have the correct information in the NC files.

See also

For more information, see **Chamfer** (p. 72).

First, a number of points have to be placed. The open end of the beam is always that end between the penultimate picked point and the last picked point (is start point). Also **Folded plates** have to be modelled as polybeam.



The shape of a folded plate can be changed by selecting and moving the chamfer-symbol. This can be done with the **Move** > **Translate** command or with **Drag and Drop**.

See also For more information, see **Moving start- and endpoints** (p. 29).



Pick at least 3 points you want the beam to go through and double-click the end point, or click the middle mouse button to finish picking

Orthogonal beam

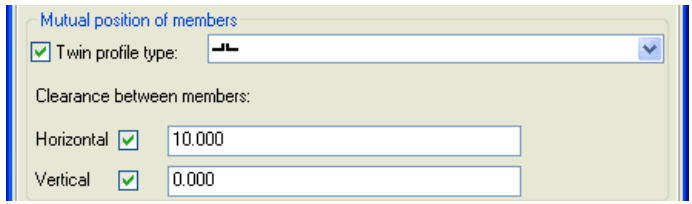


Typical for Orthogonal beams is that the profile always is placed perpendicular to the work plane. The dialog box corresponds with the **Column** dialog box.

Twin profile



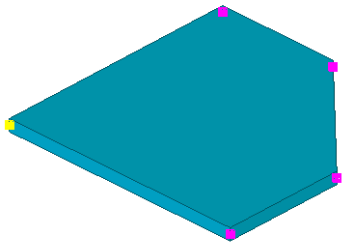
With the **Twin Profile** command two parallel profiles can be created. The dialog box corresponds mostly with the *Beam properties* dialog box, the only difference is that in the *Position* tab a frame is added in which the mutual position can be set, see image below:

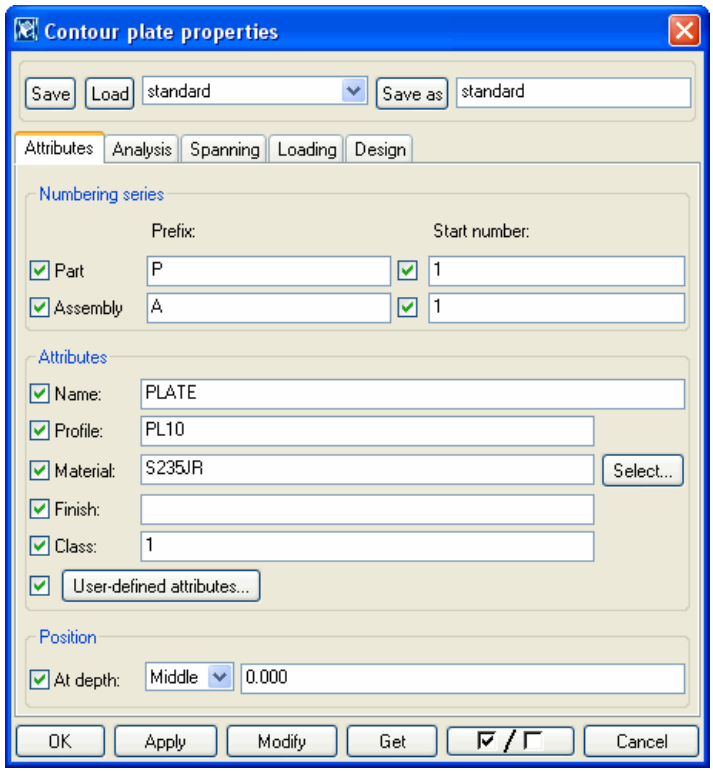


Contour plate



Contour plates are free-form shaped plates. The shape is determined by the points that are picked. In the profile notation, only the thickness of the plate may be defined (e.g. **PL20**); the shape is determined by the picked points.



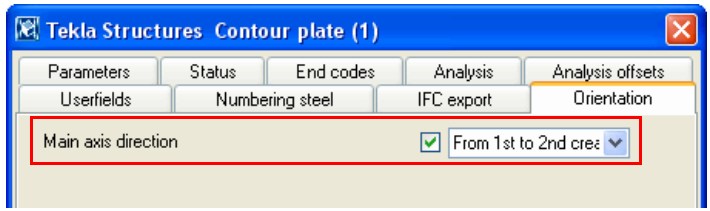


Contour plate orientation

You can set the contour plate main axis to follow the line created by the first and second points you pick. This enables you to manually define the plate orientation in drawings or reports.

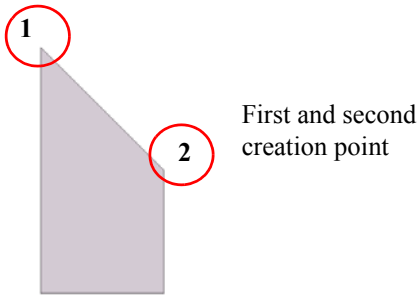
To define the contour plate orientation with first and second picked points:

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 dialog box
3. Click **User-defined-attributes** and go to the **Orientation** tab



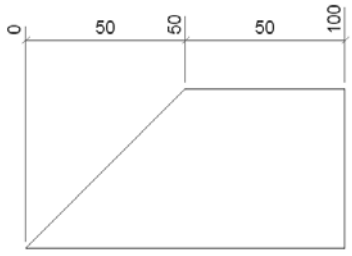
4. Select **From 1st to 2nd point** in the **Main axis direction** list box
5. Click **Modify**, and close the dialog box
6. Click **Tools > Numbering > Modified** to update numbering
7. To view the orientation of the plate, create a single-part drawing of the plate.

In the model

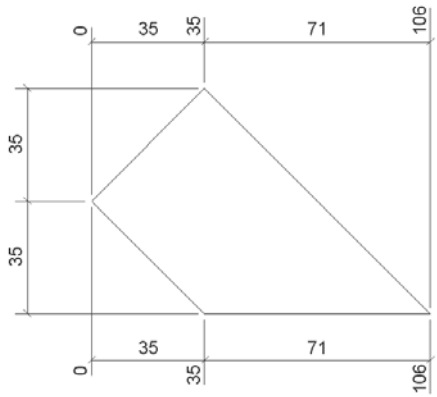


First and second creation point

In the drawing



Automatic

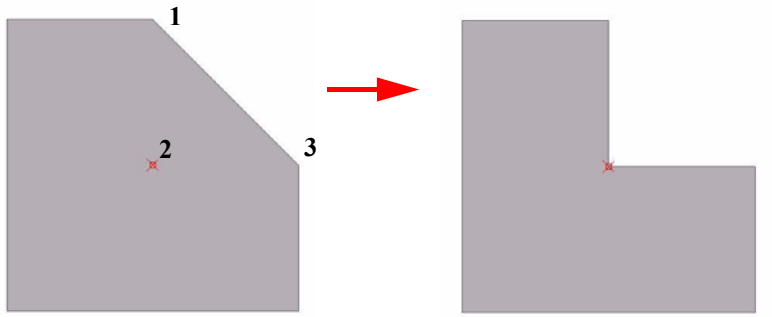


From 1st to 2nd point

Adding a corner

Adding a corner to a contour plate goes as follows:

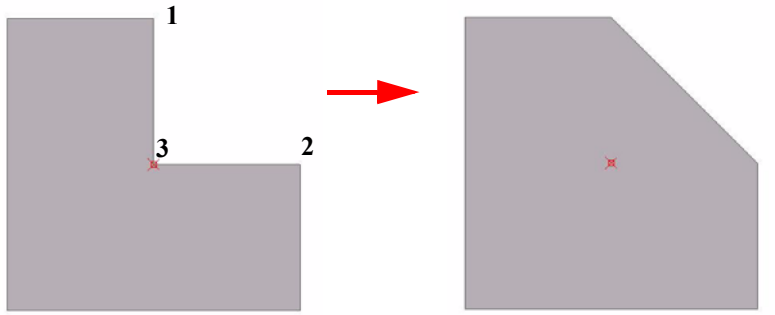
1. Select the contour plate
2. Go to **Edit** > **Polygon shape**
3. Pick the corner positions (1 - 2 - 3)



Removing a corner

Removing a contour plate corner goes as follows:

1. Select the contour plate
2. Go to **Edit** > **Polygon shape**
3. Pick the corner positions (1 - 2)
4. Pick the corner to be removed (3)

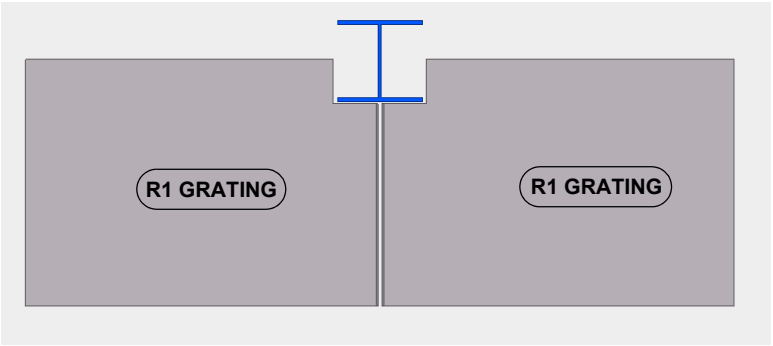


Modeling gratings and non-slip profiled plates

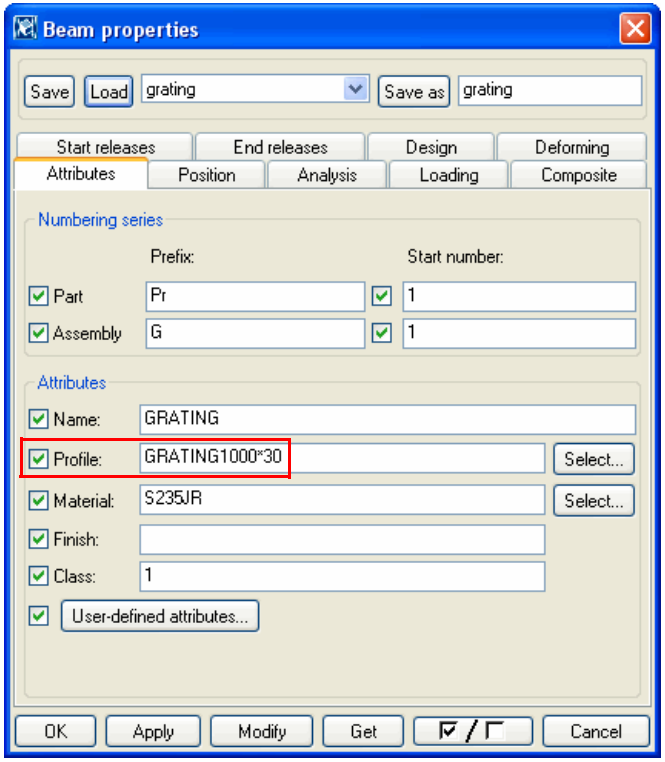
Gratings and non-slip profiled plates, created as a plate or as a contour plate in Tekla Structures, always get the same numbers. This, because plate top and bottom sides don't differ. Whereas you want different numbers for gratings and non-slip profiled plates.

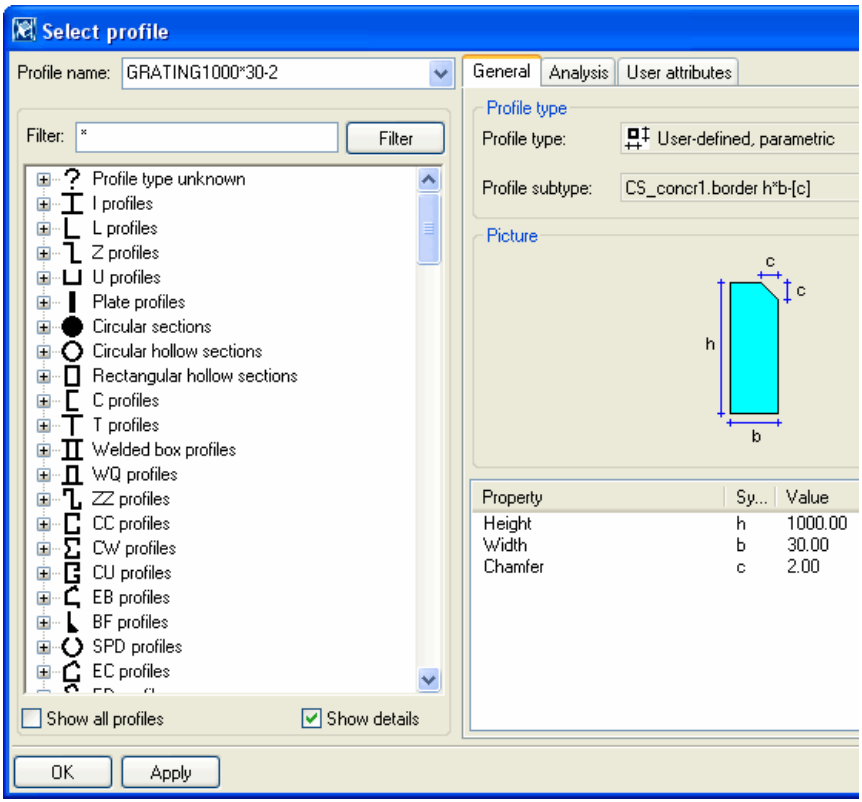
Example

Gratings created as plates:



To differ gratings in numbers, you can select the setting **GRATING** in the dialog box Beam properties:

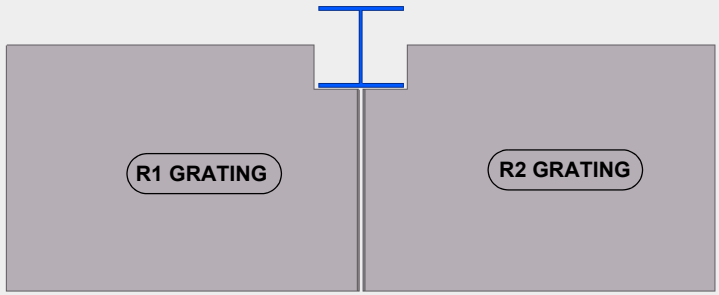






In case you are modeling gratings, the value for the chamfer (the value **c**) is set to 2 by default. However, this value can be left out.

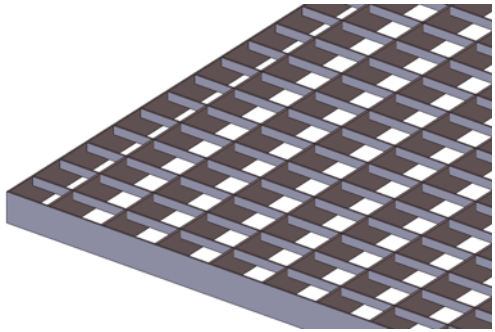
A parametric profile is used. On one side this profile contains a (small) facet. The number of the mirrored profile differs from the original profile number. The facet is now located on the other side.





You also have the possibility to create a grating in reality. For this, you can use component **Generation of profiles (48)**.

This component contains the setting **"grating"**. Load the setting and pick 3 points:

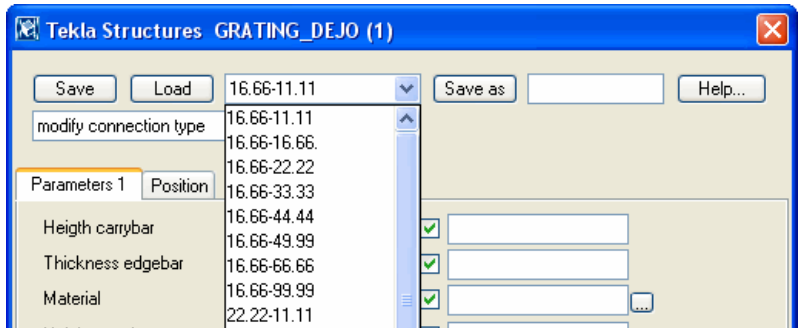


Weld all profiles or even better, put together all profiles!

Custom component Grating_dejo

You can also import custom component **Grating_dejo**. After that, the custom component is available in **all** new models directly:

1. In the Windows Explorer, go to the folder `...europe\custom-components\std` and extract the file `Grating_dejo.zip`, both `*.uel` file and accompanying component setting(s)
2. Start Tekla Structures and create a new model
3. Click icon **Find a component** in the **Components**-toolbar, or use shortkey <**Ctrl** + **F**> to open dialog *Components catalog* and set the picklist to **Custom** to display the Custom components
4. Double-click **GRATING_DEJO**, the custom component and the settings are now available:

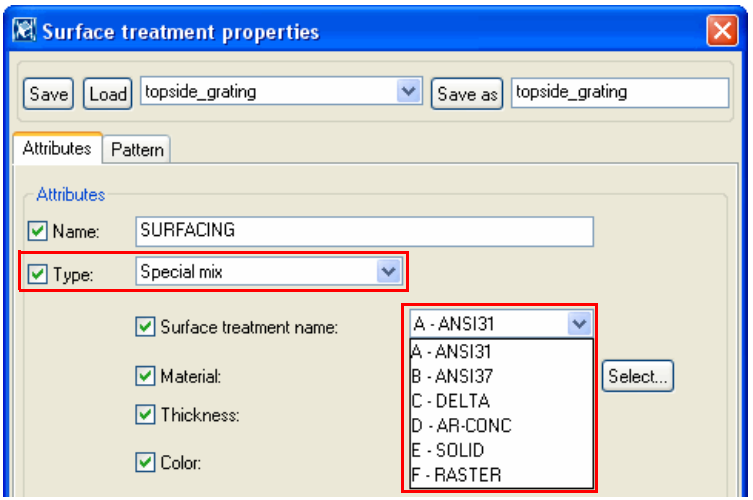


Gratings and non-slipped profiled plates in drawings

To represent gratings and non-slip profiled plates in drawings more clear, you can make use of surface treatments. First, model the gratings and non-slip profiled plates as described above.

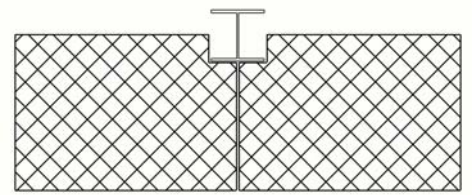
Step plan

1. In the Surface treatment properties, select the type **Special mix**, for **Surface treatment name,** select one of the following hatches:



A-ANSI31	B-ANSI37	C-DELTA
D-AR-CONC	E-SOLID	F-RASTER

2. Add one or more surface treatments
3. The surface treatment will be represented as follows in the drawing:



See also

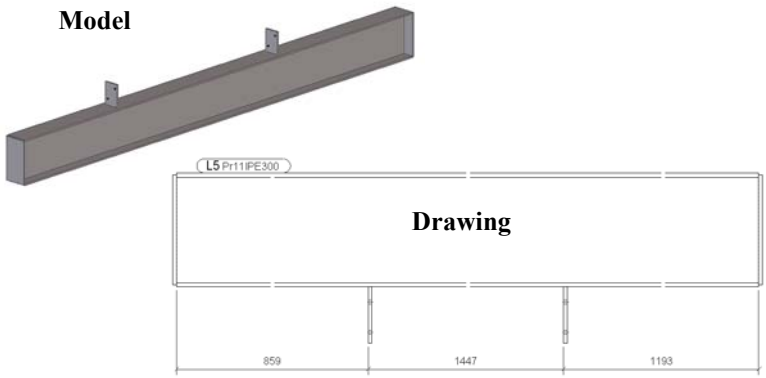
For more information about surface treatments, see **Tekla Structures.pdf** in the folder **disc:\TeklaStructures\version\environments\europe\pdf**.

Profiles in drawings

Profiles are placed on drawings according to their orientation in the model.

Example

It may happen that a beam with some purlin stubs will be displayed upside down on the drawing, i.e. with the stubs are displayed at the bottom of the beam.

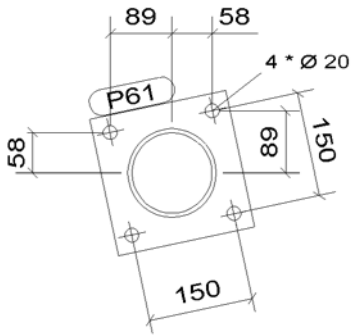


The reason is the orientation of the primary part (the beam). To correct this, change the *Rotation* of the beam 180 degrees.

You can prevent this by setting the component 1011 (U stub) to the direction **AUTO**. When you now generate this component in the model and the profile is orientated upside down, the stub will be generated at the bottom of the profile.

Example

For I- and UNP-section profiles, it is easy to determine the rotation of the profile. For tubes this is more difficult. It is handy to change the profile temporary to a rectangular hollow section profile. To do this, just change the notation of the profile from **PD** or **TUBE** to **RHS**.



An other work around for this problem is to set the component direction to AUTO.

Hide parts

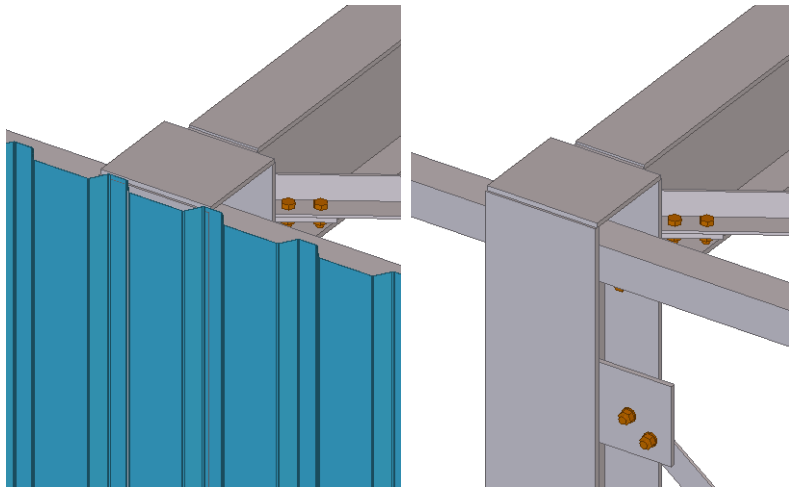
You can quickly hide selected parts from a view. For example, you could use this feature in complex connections, to temporarily hide parts in order to see the parts behind them:

1. Select the parts to hide
2. Right-click and select **Hide**

   The selected part is now invisible, by default, **Hide** changes parts into sticks.



To completely hide parts, combine the <**Shift**> key with the **Hide** command.

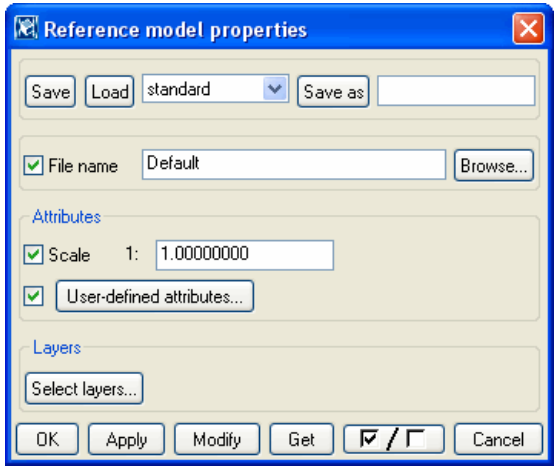


To make parts visible, use the command **Window** > **Redraw all** or right-click the sticks and select **Exact lines**.

Create a reference model



In Tekla Structures, reference models can be inserted, they serve as help (reference) to model the structure. To insert a reference model, double-click the icon opposite. The following dialog box will appear:



File name

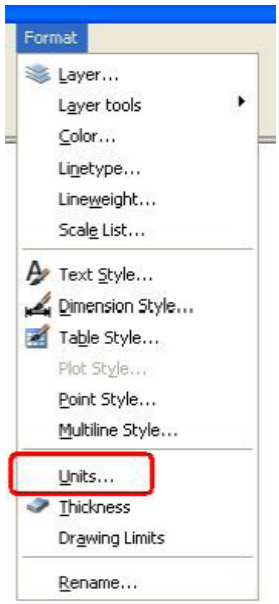
You can now select the reference model, for this click on the button <**Browse**>

Scale

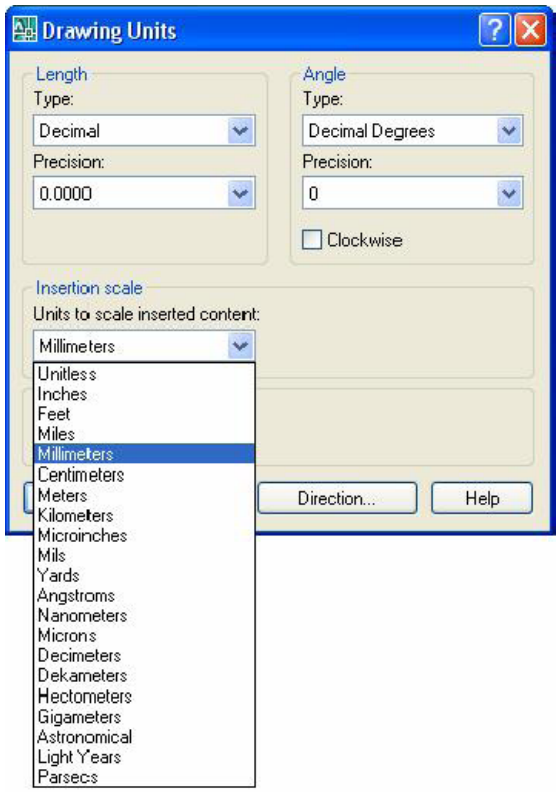
The value you enter for the scale, depends on how this DWG or DXF file is created in AutoCAD.

In the most cases those DWG or DXF files are not saved with a dedicated unit such as **mm** or **meters** so Tekla Structures will not define any units.

Instead of finding the correct scaling afterwards it will be of help to set the unit in the DWG or DXF file directly in AutoCAD. For this, in AutoCAD go to **Format** > **Units**:



Now specify in the dialog box **Drawing Units** the unit in the list box **Units to scale inserted content**:



With this the unit is defined while storing a DWG or DXF file, the unit will be recognized in Tekla Structures and the reference model will be scaled correctly.

XML files

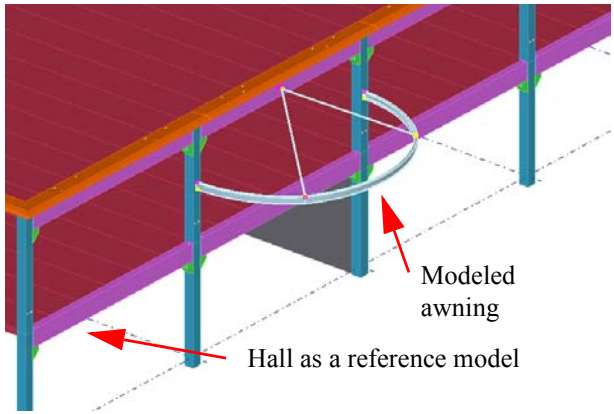
You can use DWG or DXF and also DGN (Microstation) files. You can also use webviewer models which are created in XML format.

Example

An existing model needs an addition, in this case an awning. First, save the existing model as a Webmodel. To do this, go to **File** > **Publish as Web page...**

In the model folder, in sub folder ..\PublicWeb, the file **Model.xml** is stored. This file will be used as a reference model.

Now delete all models from the current model and add the file **Model.xml** as a reference model.





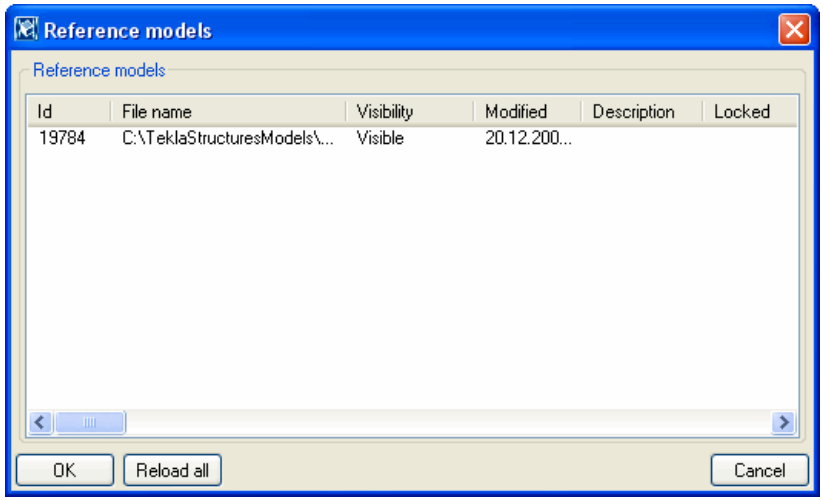
To be able to select a complete reference model, activate switch **Select components**, to select the reference model at the highest level.

To switch on or off one or more layers in the used reference model, make use of the button **Select layers...**.

Reference models will keep the size of Tekla Structures model files relatively small, because the reference file(s) are considered as single objects. It is possible to snap to vertexes from the reference model.

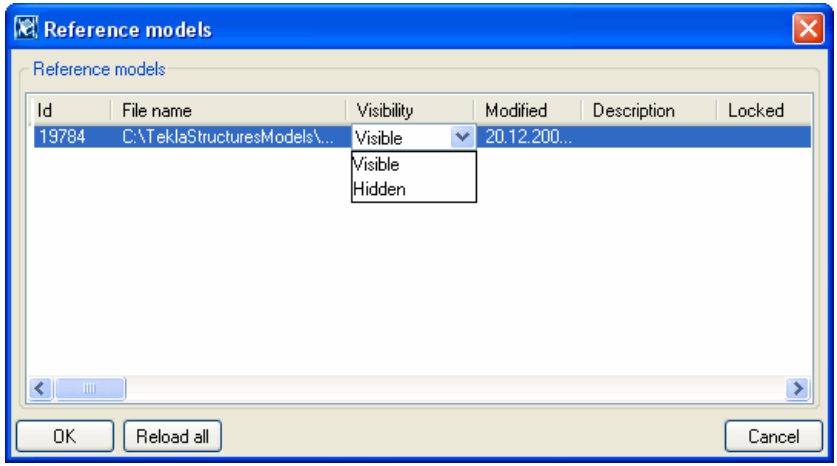
If there are several reference models in the Tekla Structures model, it can be useful to hide the reference models separately.

This can be done in dialog box *Reference models*. To open this dialog box, go to **File > Reference models**. The dialog box below will appear:



When selecting a line in this dialog box, the corresponding reference model will be highlighted in the model. The other way around, when selecting a reference model in the model will highlight the corresponding line in the dialog box.

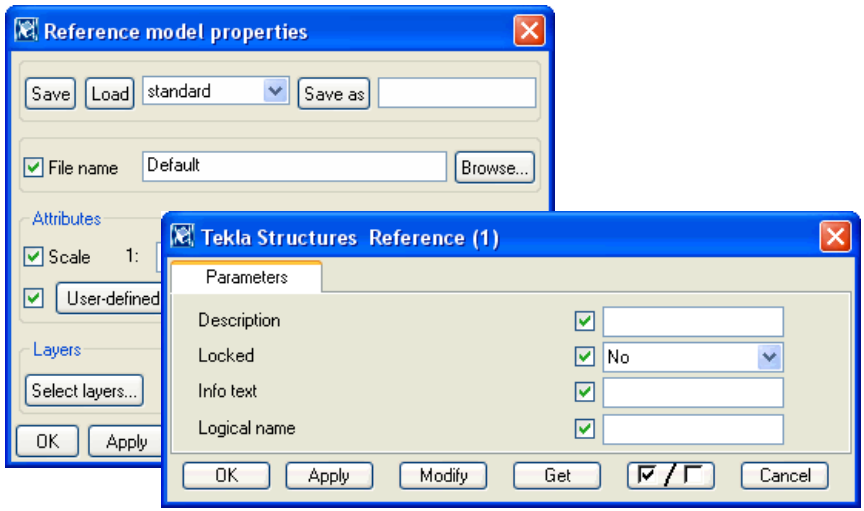
The visibility of several reference models can be set in column **Visibility**. By pressing the current setting in column **Visibility**, a list box will show up.



The options are **Visible** and **Hidden**.

If Tekla Structures is restarted, the reference model shall be invisible initially. To display this reference model again, check the option **Reference model** in **View Properties > Display**.

The displayed text in columns **Description, Name** and **Info text** in the dialog box above, can be entered in the User Defined Attributes of the reference model.



Updating reference models

You can easily update the reference models in your model in case there have been changes in the original reference models after you have brought them into Tekla Structures. To update all reference models:

- Click **File** > **Reference models...**
- Click <**Reload all**>.

Tekla Structures will go through all the reference models and re-generates the reference models having newer modified date.

Handling large reference models

Snapping to reference models

While snapping to high detailed, large reference models it may happen that a large number of snap points is found. This may result in a delay.

In such cases, it is advisable to switch off the select switches which are not necessary at that moment. For example, if an endpoint has to be found, it may be useful to switch off the other select switches.



It affects picking speed, because one dgn file is one solid object and it takes time to find correct picking point.

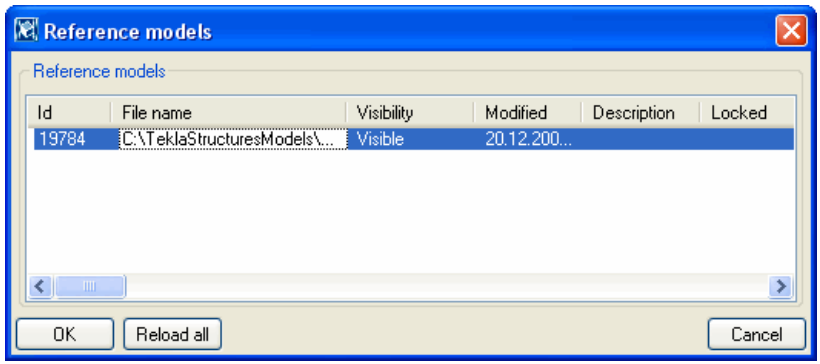
Split reference models

Split large reference models into smaller ones. Because of that only the parts you need in the model view are represented. This of course effects the speed.

Delete a reference model



To delete a reference model you open the dialog box **Reference models** by **File > Reference models...** Now select the model and click the icon **Delete**, or select the model in the model and click **Delete** or click <**Delete**> on the keyboard.



1.17 Creating concrete parts

The following commands are available in the **Concrete** toolbar:



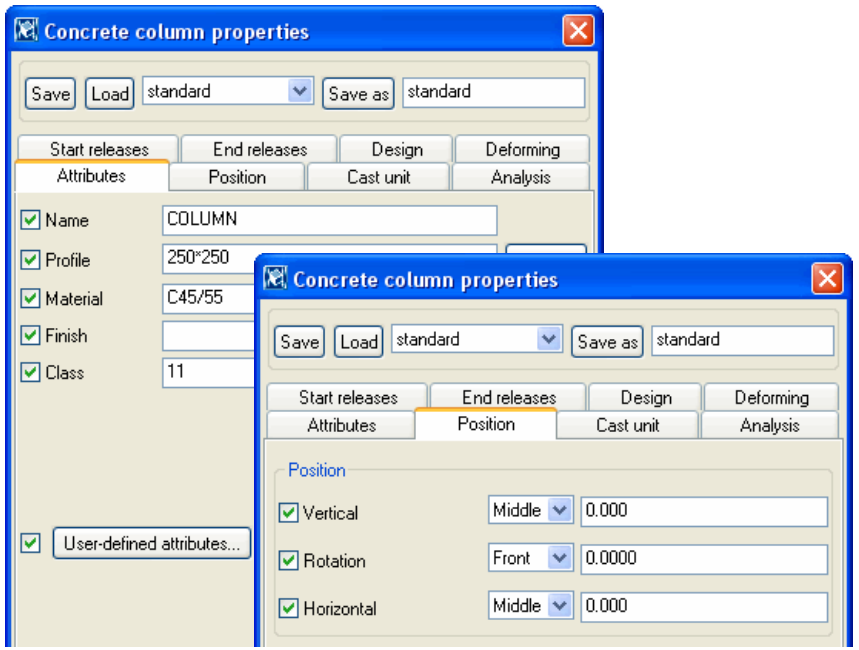
With concrete, almost any shape can be modeled. A big difference compared to Steel parts is that almost any concrete part is parametric, while steel parts are mostly library parts. Further on this chapter there is more information about parametric and library parts.

The commands for creating reinforcement and surface treatment will be explained in the training later.

Concrete columns



Double-click the icon **Create Concrete Column** to open the Concrete Column properties dialog box. The make-up of the dialog box, including the tabs and the working of the commands are roughly equal as steel parts.

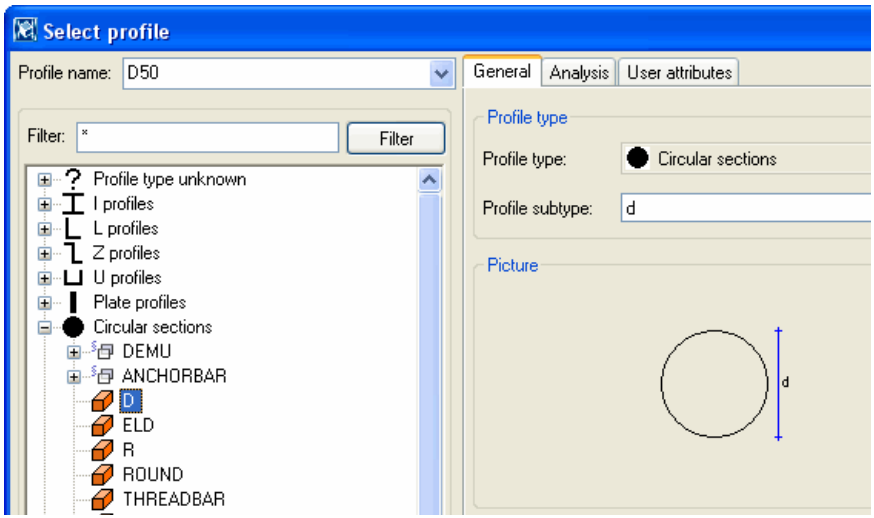


No Parts' Prefix is required for creating rectangular or square concrete columns. So, for a column with size 200x300, just enter 200*300 in the **Profile**-field. If the column needs to be rotated, then enter 300*200 or change the value for Rotation in **Position**-tab.

To place a column, close the dialog box with the <**OK**> button and pick a position in the model.

For other shapes than rectangular shapes, a Part Prefix is required. For example, for a round column with diameter 50 mm, 'D50' has to be entered.

To find out which prefix is required, the best way is to click the **...** button on the right of the input field; the parametric profile catalog will then be opened.



For Profile type, several shapes can be selected. In the picture above the Circular Section is selected.

In the Prefix list box, the available prefix' are listed. For circular sections the options are 'D' and 'R'. Click <**OK**>.

In the dialog box **Concrete column properties**, the diameter of the round profile can be entered/modified.



If the Part Prefix is already known, it's obviously easier to enter it in the input field.

In the **Cast Unit** tab, the numbering Prefix, Start number and Cast unit type (Cast in place or Precast) can be set.

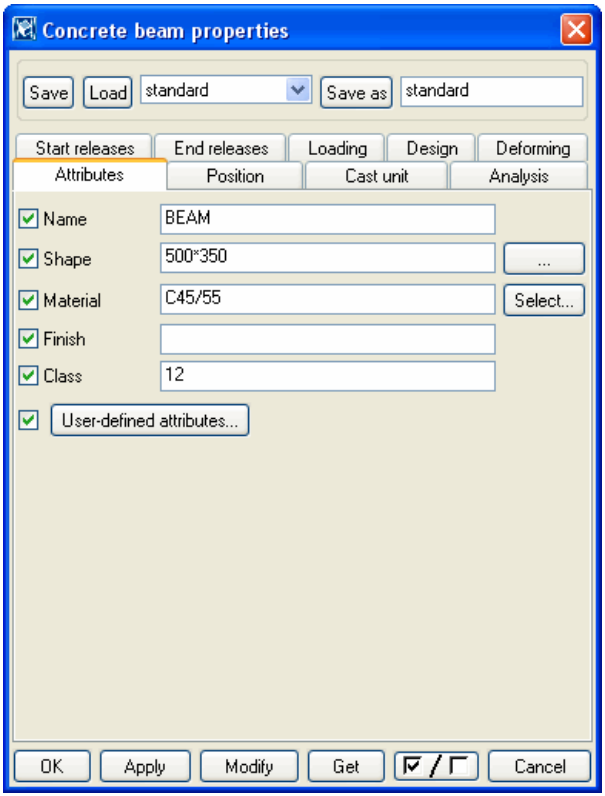
Exercise

Now create supplement A.

Beams



Double-click the icon **Create concrete beam** to open the dialog box to set the desired settings:



Concrete beams are placed by picking two points in the model and subsequently clicking the middle mouse button to close the command.

In case you want to pick more than two points, it is recommended to use the command **Create concrete polybeam**. You need to click the middle mouse button to close the command again.



Take into account while using polybeams that operations who are applied to the beam (line Cut, Fit, adding components, Part Cut and Polygon Cut), the entire profile is taken into account and that the local direction of the profile is determined by the direction from the first to the second point.

Lots of components use these direction so it may occur that components do not react on the polybeam as expected

Pad footings and Strip footings



The working of commands **Pad footings** and **Strip footings** are respectively the same as columns and beams. The footings are placed in the model by one mouse click, the Strip footings by 2 mouse clicks and subsequently the middle button click to close the command. The settings for **Pad footing** and **Strip footing** are related to various types of foundation techniques like foundation plates, foundation columns etc.

Panels



With command **Create concrete Panel**, panels can be created. The working is the same as for beams; 2 points need to be picked en then the middle button to close the command. As a matter of fact, a panel is a rectangular beam, i.e. with the dimensions 2600*200.

Slabs

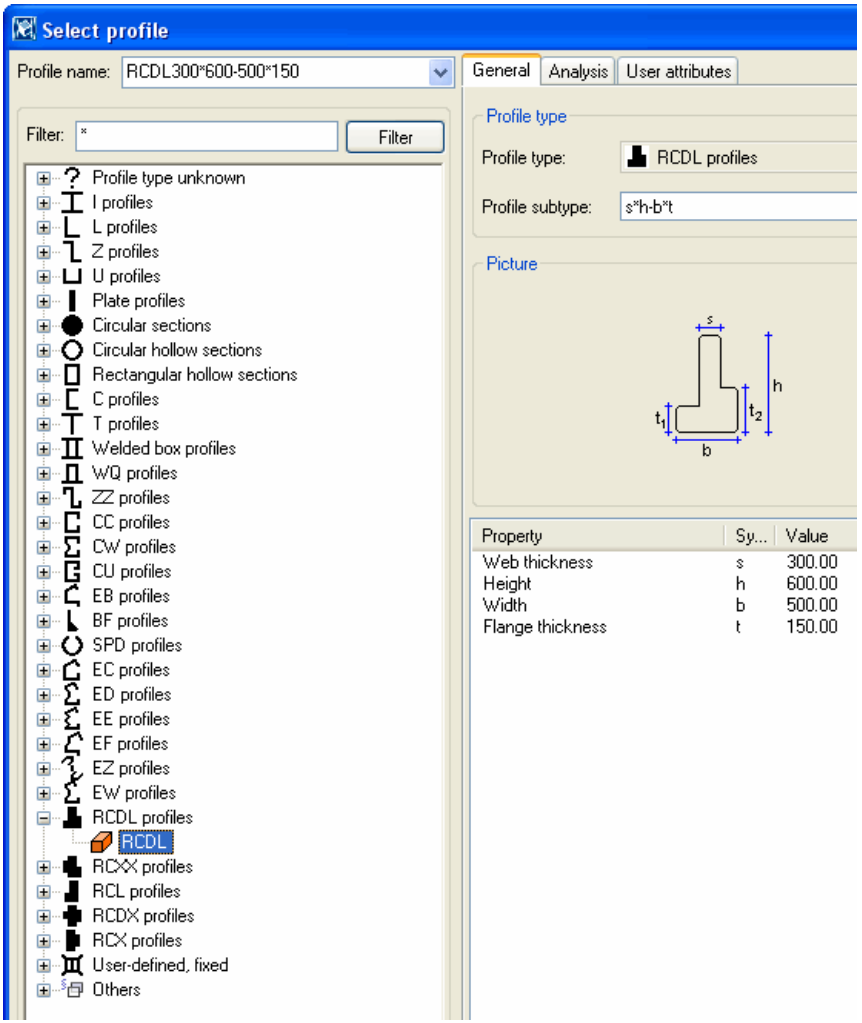


This command can create concrete contour plates. In the profile properties, in *Profile* the thickness off the slabs can be entered. Then the shape of the slab can be determined by picking points.

In practice this means you can model floors and walls with this command, but also panels (in section) if the desired shape is unavailable as Panel profile.

Parametric concrete profiles

There are various parametric concrete profiles available. These profiles can be used by any of the upper mentioned commands (except plates) These profiles are the so called RCDL, RCXX, RCL, RCDX and RCX profiles:



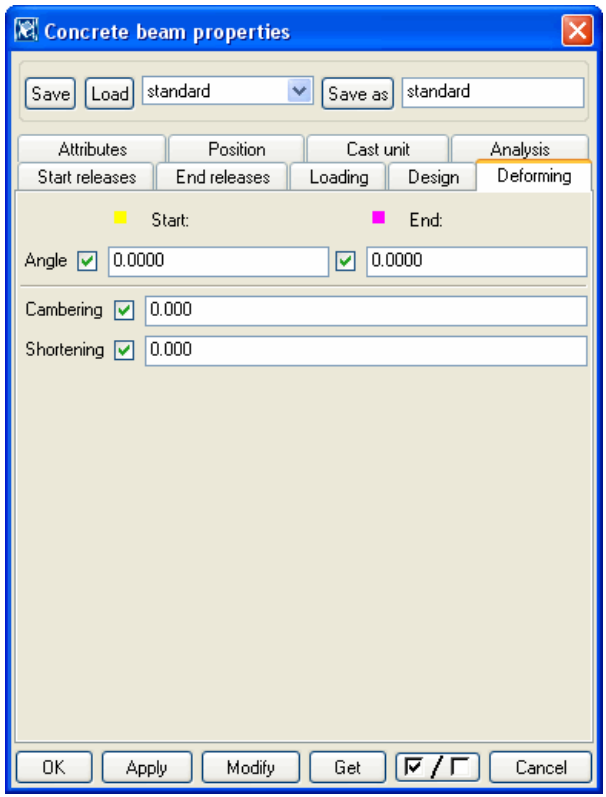
If these profile types are used frequently, it may be useful to add them to the profile library where these shapes are also predefined.

See also

For more information about adding profiles to the profile library, see **Tekla Structures.pdf** in the folder **disc:\TeklaStructures\version\environments\europe\pdf**.

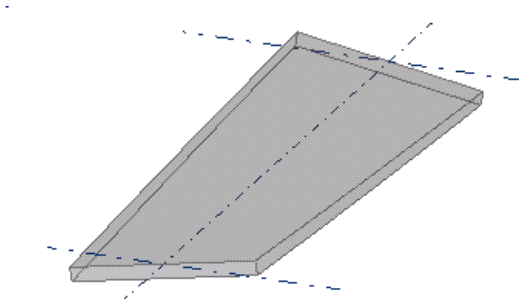
Tab Deforming

All dialog boxes for creating concrete profiles contain the tab **Deforming**. On this tab, you can enter a profile shortening or an angle for the start- or endpoint.





Cambering



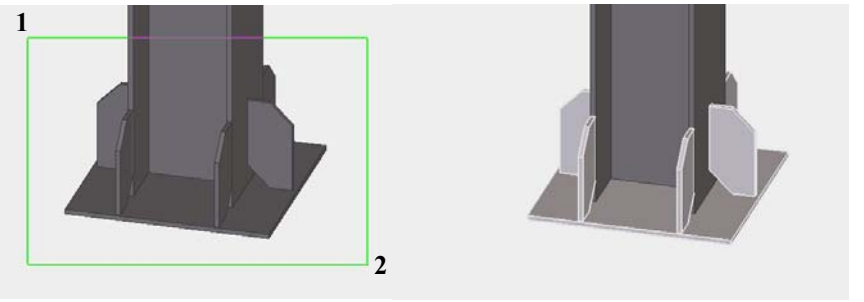
Angle for the start- and end-

1.18 Crossing selection

You can select parts in different ways when modeling or editing drawings. You can switch this option on and off in the pull-down menu **Setup > Crossing selection**.

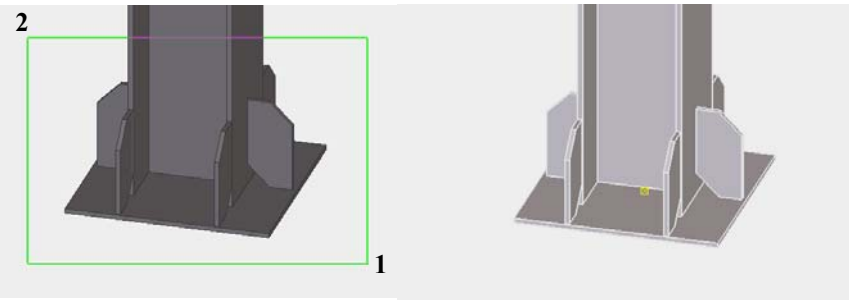
Crossing selection

Click and drag from left (1) to right (2) to select all parts **entirely** within the selection area!



Crossing selection

Click and drag from right (1) to left (2) to select all parts with any portion within the selection area!

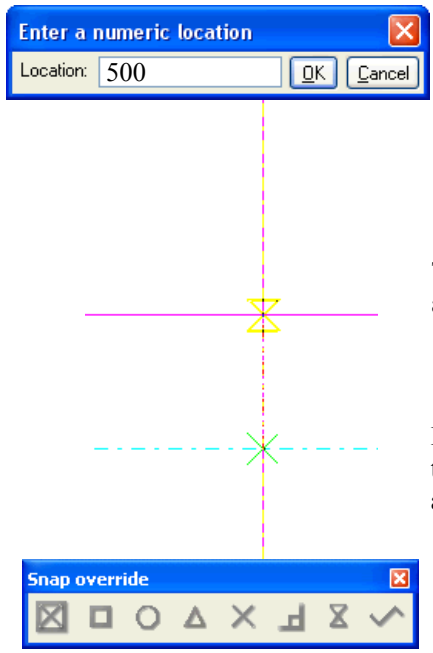


1.19 Snapping

The function **Xsnap** helps you to snap to the correct points by giving you visual cues. As you move the pointer over an object, Tekla Structures automatically highlights the points it suggests you snap to.

To toggle **Xsnap** on and off, use the keystroke **T** or go to the pull-down menu to **Setup** > **Xsnap.** By default, this function is toggled on.

Tekla Structures contains an extra set of snapping tools, for example toolbar **Enter a numeric location** you can use in combination with **Xsnap** to snap to specific locations.



Use to enter the location coordinates you want to snap to

Tekla Structures shows the available snap positions.

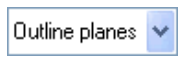
Hold down the <**Ctrl**> key to pick temporary reference points to use as an origin.

Use to temporarily override current snap switch settings.

Snap switches

You can use Snap switches in both Modelling- and Drawing mode:



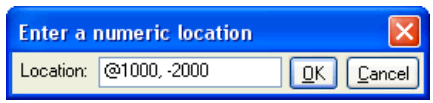
Icon	Positions to pick	Icon	Positions to pick
	Snap to points and grid intersections		Snap to nearest points
	Snap to end points		Snap to any position
	Snap to center points		Snap to reference lines / points
	Snap to mid points		Snap to geometry lines / points
	Snap to intersection point		For custom components
	Snap to perpendicu-lar points		

Orthogonal snapping

Use the shortcut O (**Setup > Ortho**) to snap to positions in orthogonal directions on the work plane (0, 45, 90, 135, 180 degrees, etc.).

Numeric snapping

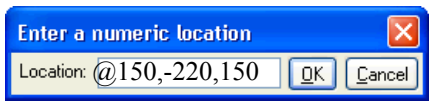
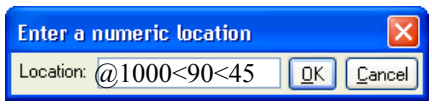
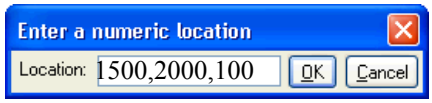
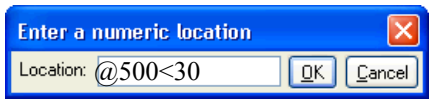
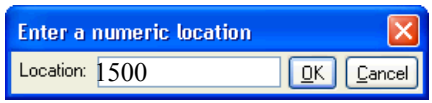
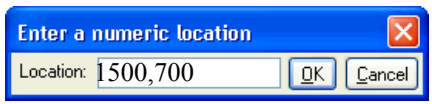
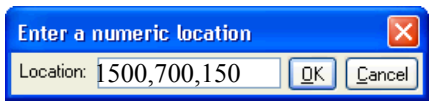
Use the **Enter a numeric location** toolbar to key in position coordinates you want to snap to.



To display the toolbar, initiate a command that requires you to pick positions, and then do one of the following:

- Start entering the coordinates using the keyboard.
- Click **Setup** > **Enter a numeric location** and select an option or press shortcut **R** and enter one or more values.

The following table explains the types of information you can enter:

Example	Description
	The x-, y- and z-coordinates of a position, separated by comma's
	A distance, an angle on the xy plane, and an angle from the xz plane separated by angle brackets Angles increase in the counter-clockwise direction.
	The coordinates based on the origin of the work plane.
	The coordinates relative to the last position picked, e.g. @1000,500 or @500<30
	A distance to an indicated direction.
	If you omit the last coordinate (z) or angle, Tekla Structures assumes that the value is 0.
	In drawings, Tekla Structures ignores the third coordinate.

After you enter the coordinates, press <**Enter**> or click <**OK>** to snap to the position.

Temporary reference points

You can create a temporary reference point to use as a local origin when snapping in models or drawings.

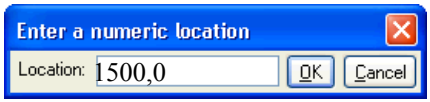
1. Initiate a command that requires you to pick positions
2. Hold down the <**Ctrl>** key and pick a position. A green cross indicates that this position is now a temporary reference point.
3. Use this temporary reference point with snap tools to define directions and distances.

Snap examples

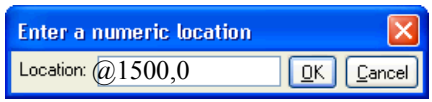
A beam with a certain length

1. Click **Create beam**
2. Pick the first point in between the grids A-1 and B-1 or enter the coordinate

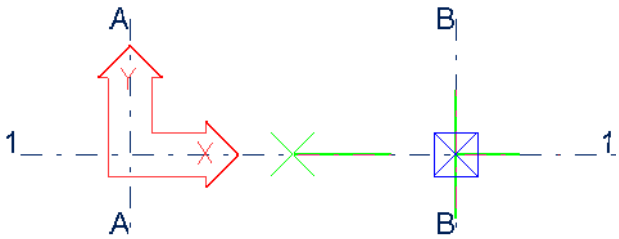
   Relative to the position of the **work plane:**



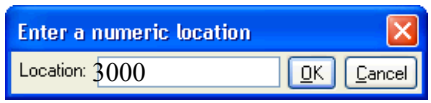
   Relative to the **last picked point** by using shortcut **R:**



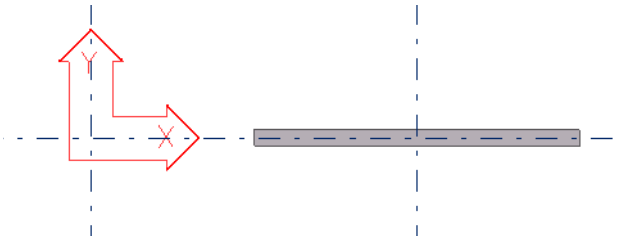
3. Move the mouse pointer in grid direction B-1:



4. Now pick the second point of the beam or enter the length of the beam



5. Press <**Enter**> or click <**OK**>, the beam will be created

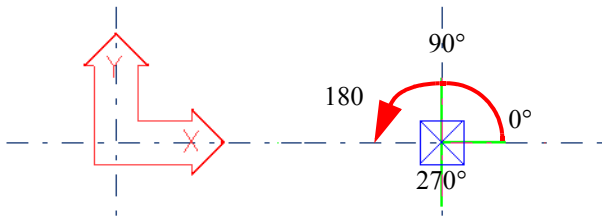


A shifted beam with a certain length

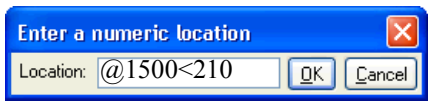
1. Click **Create beam**
2. Pick the first point or enter the coordinate relative to the position of the work plane or relative to the last picked point by using shortcut **R**



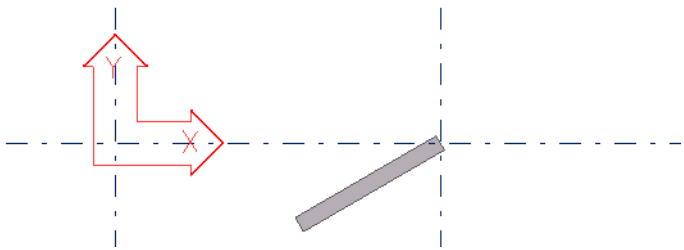
The direction of the X-axis of the work plane is always the 0°-axis!



3. Enter the length and the angle of the shifted beam relative to the work plane, use shortcut **R** and the symbol "<" as a separating character

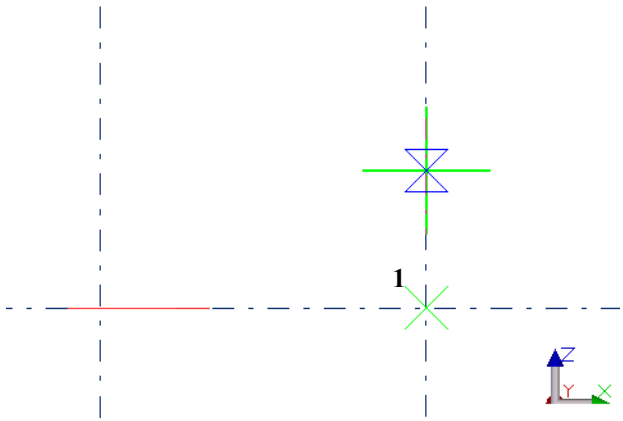


4. Press <**Enter**> or click <**OK**>, the beam will be created

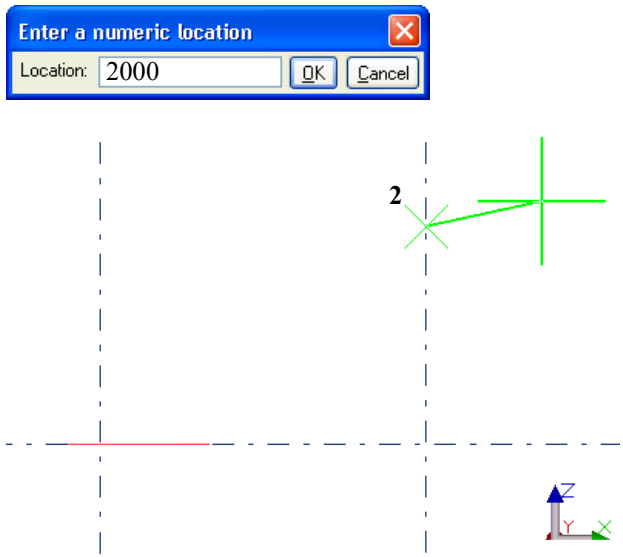


A beam using a temporary reference point

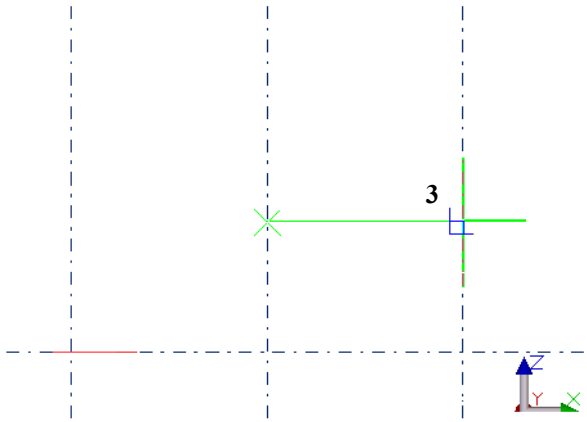
1. Click **Create beam**
2. Hold down the <**Ctrl**> key to pick a temporary reference point, a green colored cross, to use as an origin (1)

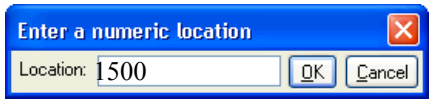


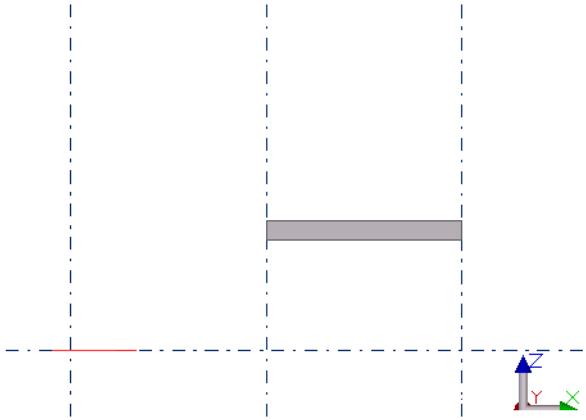
3. Move the mouse pointer in the direction of the start point of the beam and enter a value or pick the first point of the beam (2)



4. Move the mouse pointer in the direction of the second point of the beam and enter a value or pick a point (3)



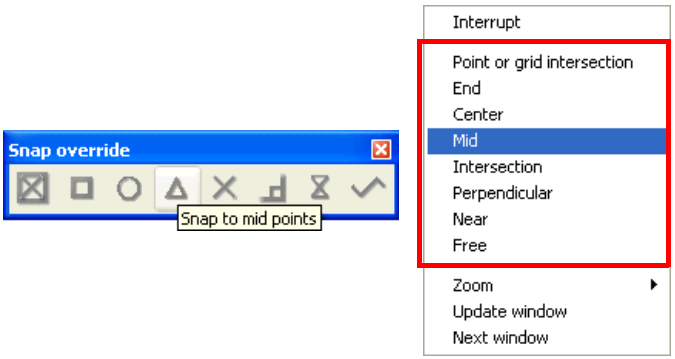




The beam will be created

A beam using temporarily override current snap switch settings

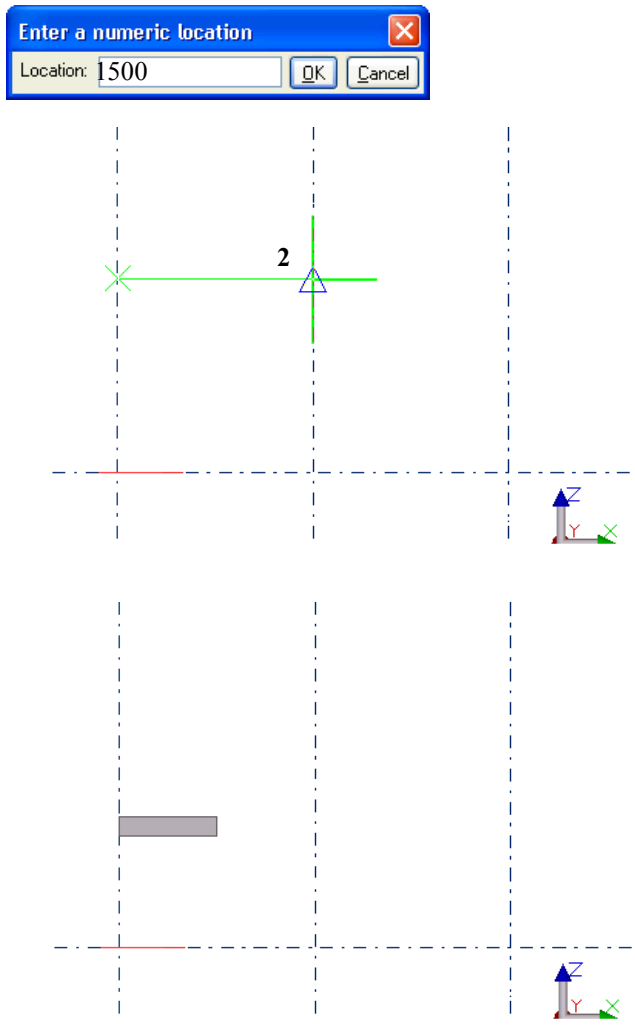
1. Click **Create beam**
2. Use a temporary reference point to pick the position of the beam at the centre of the grid line (1)



3. Click the right mouse button to select an option to temporary "overrule" the snap settings or use the toolbar **Snap override**



4. Move the mouse pointer in the direction of the second point of the beam and enter a value or pick a point (2)



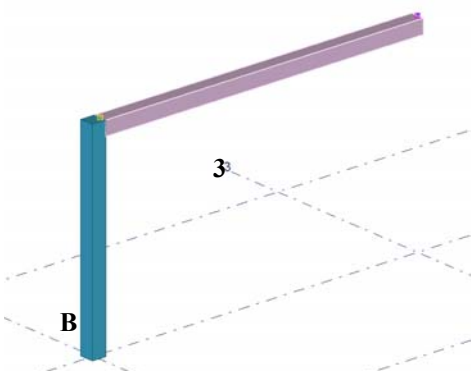
The beam will be created

Keys X, Y and Z

In Tekla Structures you can easily make use of the keys X, Y and Z to lock the mouse pointer in a specific direction.

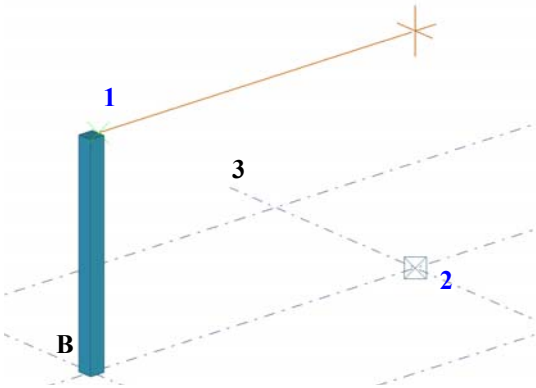
Example

You want to create a beam at a certain level:



Step plan

1. Click on the icon **Create beam** and pick the first position (1)



2. Snap at the first picked point and press the key **Z,** the z direction is now locked



You always need to snap on a certain point to be able to lock the mouse pointer!

3. Pick the second position (2) on grid B-3, now that the **Z** direction is locked, it automatically positions the second point of the beam at the locked **z** direction.

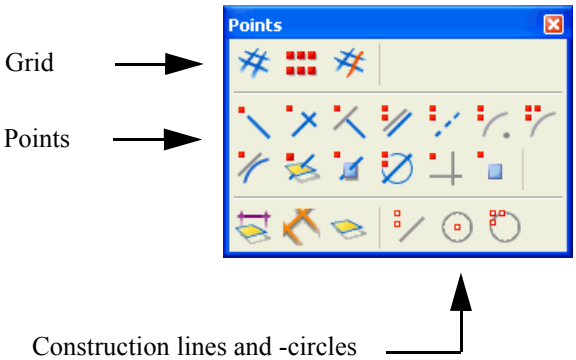
1.20 Creating points

To place an object where no lines or objects intersect, you need to pick points, of course it is prefered to make use of **Numeric snapping** as discribed in **Snapping** (p. 56).

Toolbar Points not visible by default

Because of that, the **Points** toolbar is not visible by default. Of course you can make the **Points** toolbar visible again. To do this, go to **Window** > **Toolbars** and check **Points**.

To create points you can use the commands in the toolbar **Points**:



It is also possible to create grids and construction lines and -circles.

See also

For more information about the properties and how to use points, grids and construction lines and -circles, see the Tekla Structures Online Help.



The properties dialog boxes from all icons in toolbars can be opened quickly by holding down the <**Shift**>-key while activating the command.

1.21 Detailing profiles

Fitting



A fitting operation fits a profile end to the selected fitting line. A fitting operation may occur only once to both ends of the profile. With fitting, the shortest part of the profile will be deleted automatically.





When shortening profiles, always use the FIT-command. When the Line-Cut command is used for shortening, the length values of profiles in NC files can be wrong!

See also For more information, see the document **TeklaStructures.pdf** in the folder **disk:\TeklaStructures\version\environments\europe\pdf**.

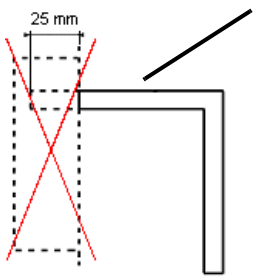
Line cut



An other way of shortening profiles is using the Line-cut command. Several line-cuts can be applied to one profile end or a combination of one fitting and one or more line-cuts can be applied. When using line-cut for shortening, you always have to select the side of the profile which has to be removed.



When profiles are shortened in longitudinal direction using the fit- or line-cut command, the profile notation shall not be changed in reports.



In the example an angle steel L100/10 needs to be changed towards L100/75/10.

WRONG: removing 25mm strip using Fitting or Line cut

RIGHT: In the properties dialog of the profile, change the profile towards L100/75/10

Part cut



With the **Part Cut** command, a cut can be made in a profile using another profile:

1. If necessary, position a profile through another profile
2. **Detailing > Part Cut**, or click the icon
3. Pick the profile to be cut
4. Pick the other profile (the cutting profile). The dark blue lines are the cutting lines. It is possible to double-click the cutting line and change the shape and size of it. It is even possible to delete the cutting profile. The cut will remain!

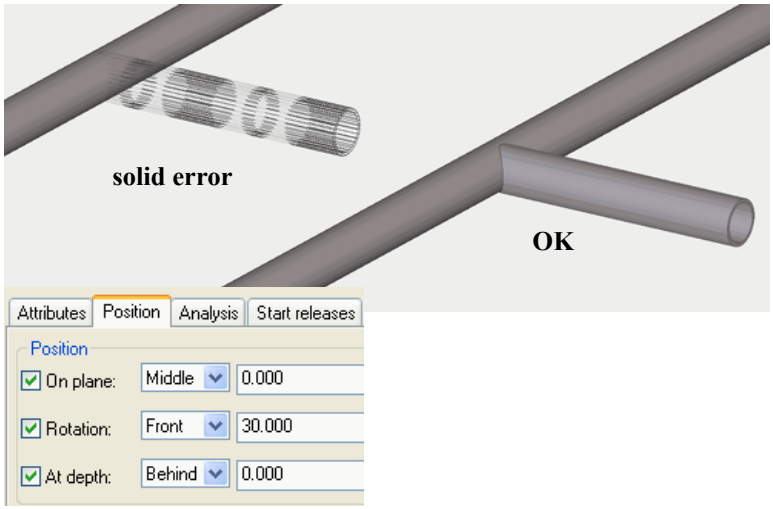


Circular holes can be made with the **Part Cut** command. Those type of holes are also dimensioned on drawings. However, when creating circular holes, it is better to use the **Bolt** command. For larger holes simply enlarge the tolerance of the bolts.

Solid error

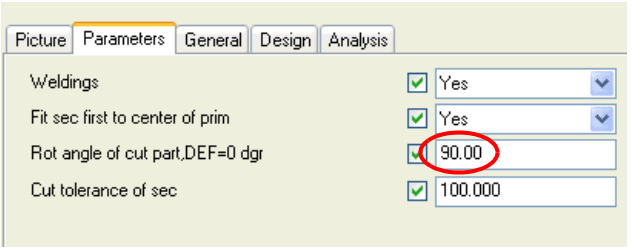
Occasionally, after using the Part Cut command, a *SOLID ERROR* can occur. This type of error occurs when Tekla Structures experiences difficulties in calculating the part cut operation. In those cases the cutted profile will be highlighted in a transparent color.

In most cases, the part cut can be made if an angular rotation is applied to the cutted profile (see picture).



A Solid Error can also occur when applying macro 23 (Round tube). To solve the problem, set a slight rotation for the macro.

A Solid Error always needs to be corrected. If you do not solve this, the lengths of profiles may be wrong and you are not able to create drawings.



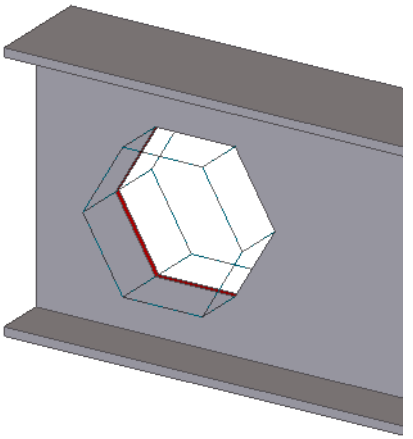
Polygon cut



With the **Polygon Cut** command, a random contour is cut out of a profile.

Step plan

1. Create points on a profile
2. **Detailing > Polygon Cut** or select the icon
3. Pick the profile to be cut
4. Define the shape by clicking points. Finish with the starting point.



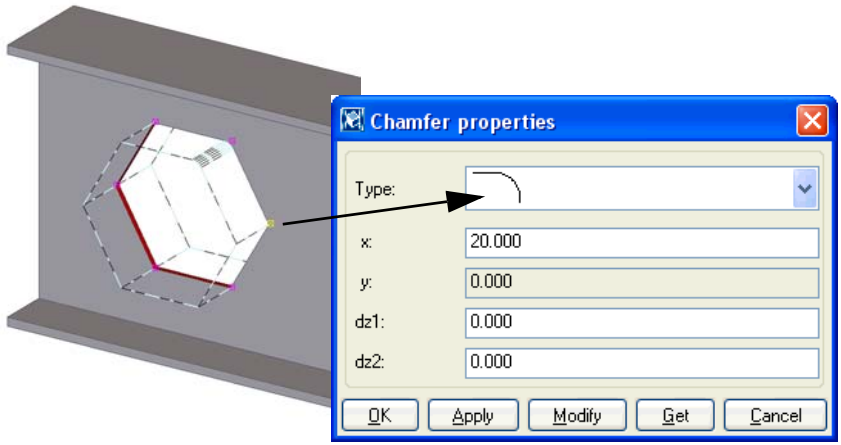
The dark blue lines which are visible, are the cutting lines, this cutting symbol can be used for copying.

The chamfer properties of the vertexes can be edited by double-clicking the handles.

See also

For more information, see **Chamfer** (p. 72).

This also goes for contour plates and poly profiles.

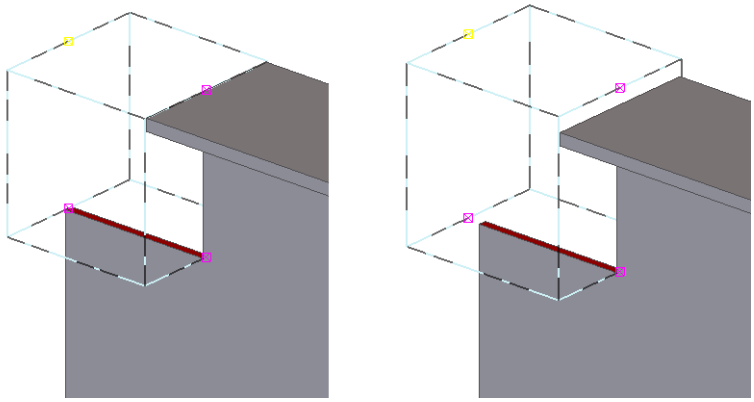


The shape of a contour can be modified by moving chamfers:

Step plan

1. Select a chamfer
2. Click the **Move** > **Translate** icon and enter a value
3. Click <**Move**> and <**OK**>
4. The contour will change

Do not let the cutting contour coincide with the contour of the cutted profile: unexpected behavior can occur. (see image below).



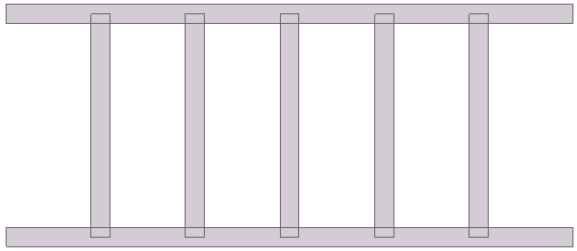
Not like this, But like this!

Batch jobs

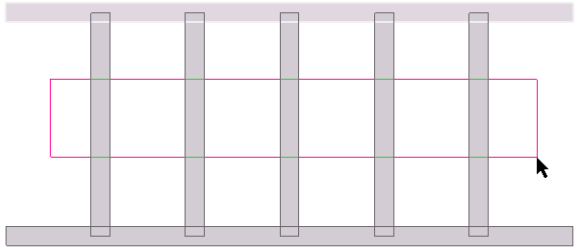
Mentioned jobs above can also be used for more profiles at once. E.g. welding and fitting in the following example:

Step plan

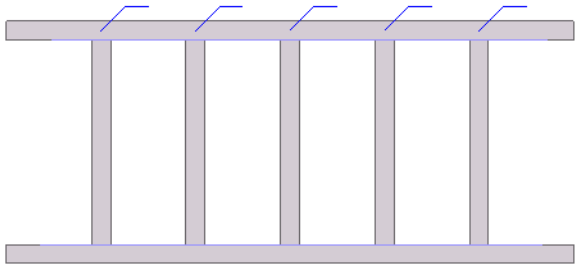
1. Select the command **Create weld**



2. Now select the main part and after that the secondary parts using a crossing.

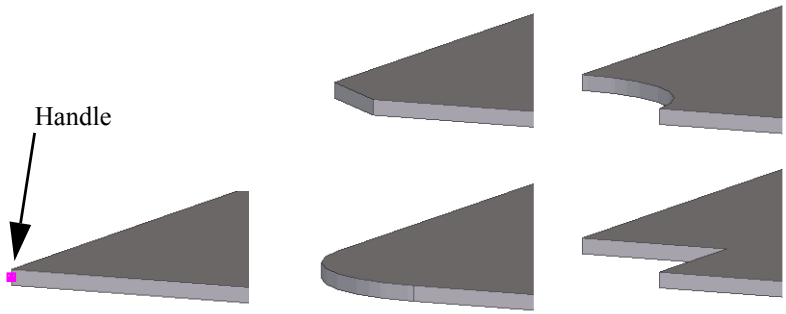


3. All 5 profiles are now welded at once. Now you can also fit all profiles at once.



1.22 Chamfer

You can modify the chamfers of contour plates and polybeams:



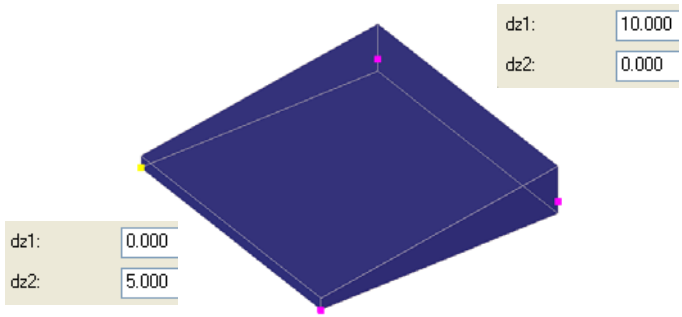
Default chamfer Modified chamfers

To do this you need to select a profiles' handle.

Select handles

1. Select the object
2. Hold down the <**Alt**> key and drag the mouse pointer from left to right to select a handle(s). Tekla Structures will highlight the profiles' handle(s). The handle of the first picked point will be yellow, the rest will be purple:
3. Double-click on the selected handle(s) or click right mouse button and select **Properties...,** the dialog box **Chamfer properties** will open
4. Select an option and enter the values, now click **Modify**

   Use the fields *dz1* and *dz2*, to give plates for example varying thicknesses. Moves the top surface (dz1) or the bottom surface (dz2) of the part corner in the part's local z direction.



1.23 Splitting and combining parts

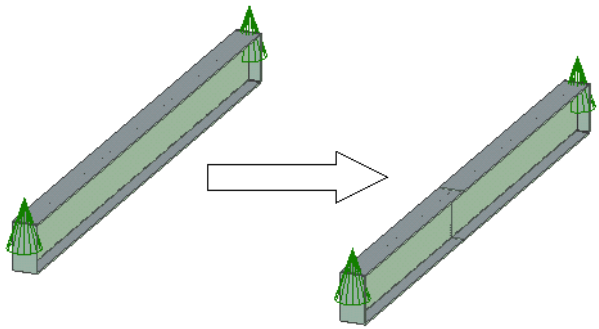
In Tekla Structures, parts can be splits and combined. This can also be achieved with component (1047), but this component has a disadvantage that already existing components connected to the profile will be deleted when using it.

Splitting



Go to **Edit > Split** of use the icon opposite.

Pick the part to be split, next pick the point for the dividing line. (create point first if necessary).



On the split spot, new start- and endpoints are placed.

If numbering already has been done, than the part- and assembly number is allocated to that part that was closest to the initial start point of the profile. The other part gets a new number, and drawing(s) have to made of it.

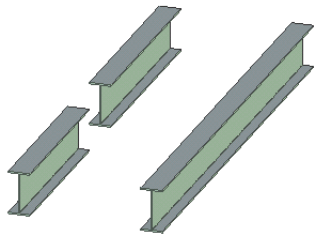


If the splitted part contains a bolt group and the part is splitted at the position of a bolt group, that bolt group also has to be splitted manual. There is no function for that!

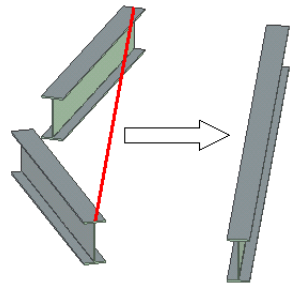
Combining



Go to **Edit > Combine** or use the icon. Next, pick two parts which have to be combined.



Space will be filled after combining



Largest distance between start-/endpoint

Profiles whose center lines are not in line with each other will be combined by taking the largest distance between the start-/ endpoints from both profiles.

All part properties (profile, class, user-defined attributes, etc.) from the first picked part are taken for the combined part.

Remind that in case of combining a Ø32mm round bar with a Ø16mm round bar, the new profile will receive the properties of the first selected profile.

1.24 Placing bolts



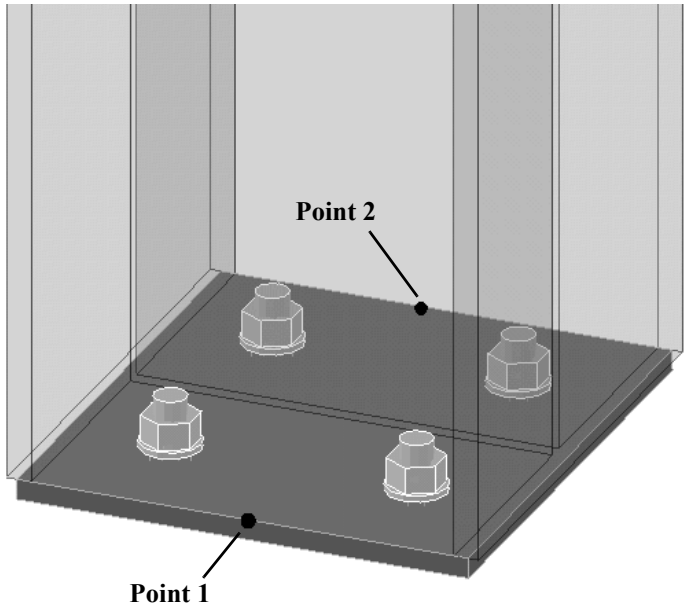
Bolt patterns are used to connect profiles. There are two points needed for placing the bolts. Bolts can penetrate one profile, but also a number of profiles.

To insert a bolt pattern, you always need to pick two points. These two points determine the (local) X-direction of the bolt pattern.

Step plan

1. Create two points, required for placing the bolt pattern
2. Double-click the **Bolts** icon
3. Pick a desired setting and click <**OK**>
4. Select the parts which have to be connected, terminate the selection with the **middle mouse button**. In the image below only the base plate has to be selected
5. Pick the start- and endpoint for placing the pattern. These points determine the **X-direction** for the pattern (important for the bolt distances)
6. The bolts will be inserted

In the image below, a bolt pattern is inserted in a base plate with the settings corresponding with the above dialog box. The two points which have to be picked before the bolts are placed, are indicated with **Point 1** and **Point 2**.



With the option **Start point** (in frame Offset from), you can move the bolt pattern in the x-, y- and z-direction. In the example above, the offset from start point in Dx is set to **35**, i.e. the bolt pattern will be moved 35 mm in the x-direction, compared to the start point (=point 1)

Explanation of remainder of input fields in the dialog box Bolt properties

Bolt size	Diameter of the bolt. Available diameters depend on the selected Bolt standard
Bolt standard	Standard (quality) of the bolt. Default set to 931-8.8
Bolt type	With this listbox can be chosen when the bolts have to be mounted; already in the workshop or on the site. Default setting is *Site.* When bolts are mounted to the parts in the workshop, use “Workshop”.
Thread in material	Defines if thread may be situated in fixed length
Cut length	Depth of search-area, usually about **3x** fixed length. If you want to force a bolt to be a certain length, enter a negative value for length (e.g. -60).
Extra length	Extra length for the bolt
Shape	The rough shape of the pattern. Options are: Array, Circle and XY-list
Bolt dist. X	Bolt distance(s) of the bolts in X-direction. One distance means 2 rows, 2 distances means three rows, etc.
Bolt dist. Y	Bolt distance(s) of the bolts in Y-direction
Tolerance	Clearance between bolt and hole
Slotted hole X	Size of slotted hole in X-direction. Works only in combination with the checkboxes above the picture in the dialog box
Slotted hole Y	Size of slotted hole in Y-direction. Works only in combination with the checkboxes above the picture in the dialog box
Bolt size	Diameter of the bolt. Available diameters depend om chosen Bolt standard



A bolt pattern can contain up to **100** bolts or holes as a maximum.

Slotted holes

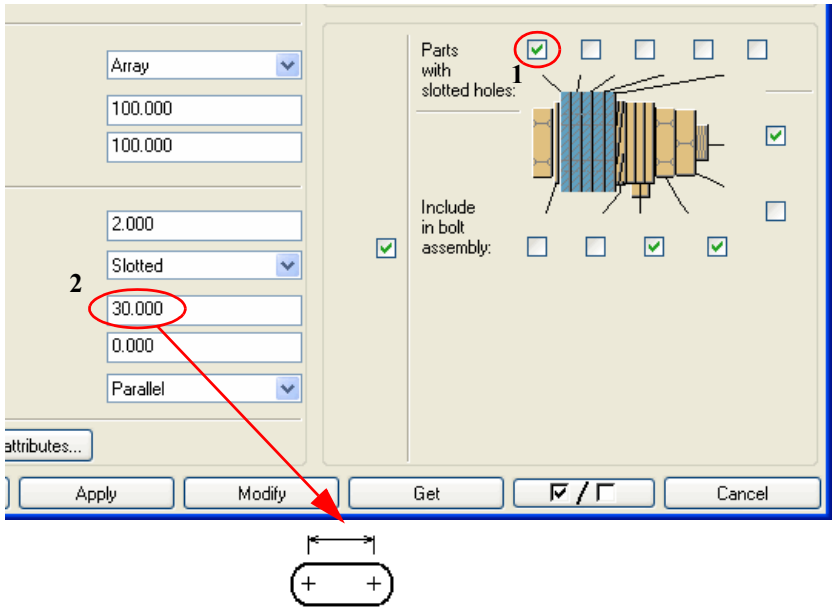
Besides circular holes, it is also possible to create slotted holes with the *Bolt*-command. Slotted holes can be set in two directions, X or Y. The length of the slotted holes is set in the input fields *Slotted hole X* and *Slotted holeY* (in the image below marked with “2”). This distance indicates the centre distance of the two outside circles.

Example

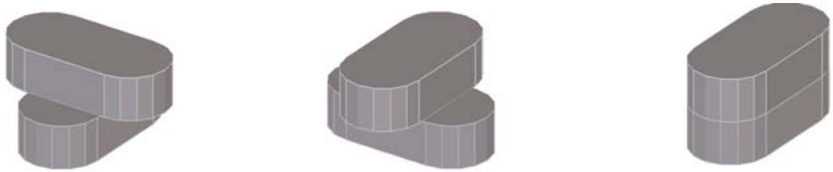
If a *Bolt size* of 16 mm is selected, with a *Tolerance* of 2mm, and a *Slotted hole* size of 30mm, a slotted hole is created with a maximum size of 48*18mm.

When a bolt (pattern) penetrates more than one profile, you can determine for each profile individually if slotted holes have to be created or not. This is determined by checking one or more checkboxes above the bolt picture (in the image below marked with “1”).

- Mark 1: select the profile which should contain the slotted hole (check). If no profiles are checked, no slotted holed will be created!
- Mark 2: set the size and direction of the slotted hole



Different types of holes:

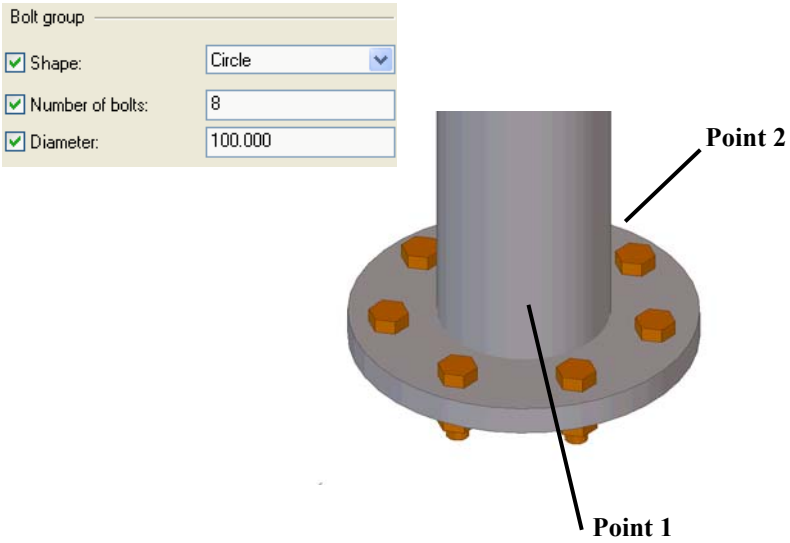


even **odd** **parallel**

Circular bolt patterns

By default, rectangular bolt patterns are created. This is determined by the setting of the **Shape**-listbox, which is set to **Array** by default.

A circular bolt pattern can be made by changing the **Shape**-listbox to **Circle**. The two inputs below the listbox will change to **Number** and **Diameter**.



Editing bolt patterns

In case bolts do not penetrate the correct profiles, than it is easy to edit this:

1. Select the bolt (pattern) and go to **Edit > Boltparts** in the pull-down menu or click the right mouse button en select **Boltparts** from the contextmenu

   After the bolt pattern is selected, the corresponding parts will be highlighted in different colors (red, yellow, white)

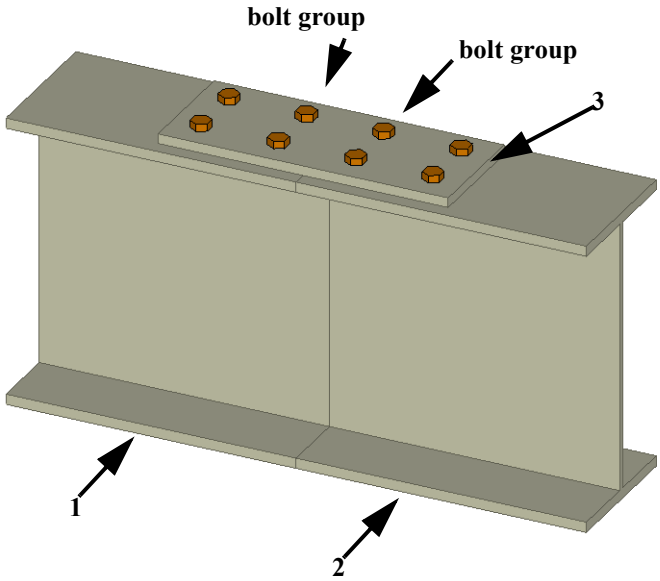
2. In the command bar will be asked to pick the connecting parts again

   The bolt length will automatically be adjusted to the new material thickness.

Inserting a divided bolt pattern

The **Bolt**-command is able to divide automatically a bolt pattern across multiple profiles. This can not be done afterwards, it has to be set beforehand.

In the example below, two beams (IPE300) are connected and joined together with a plate.



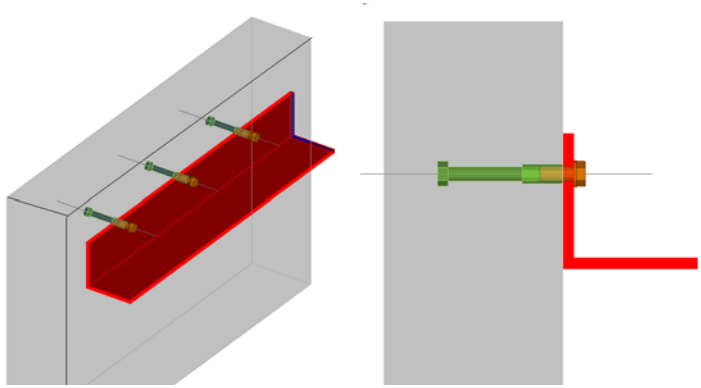
The properties of the bolts are entered in dialog box “Bolt properties”.

- In X-direction, 4 rows are set (Bolt dist. X = 3*100)
- Close the dialog box with <**Apply> and <OK>**
- Pick parts which have to be connected. First pick the beams, and than the plate (picking order: 1 –2 - 3; see example). Than pick the points and the bolt pattern will be created. The bolt pattern will automatically be divided across the two beams

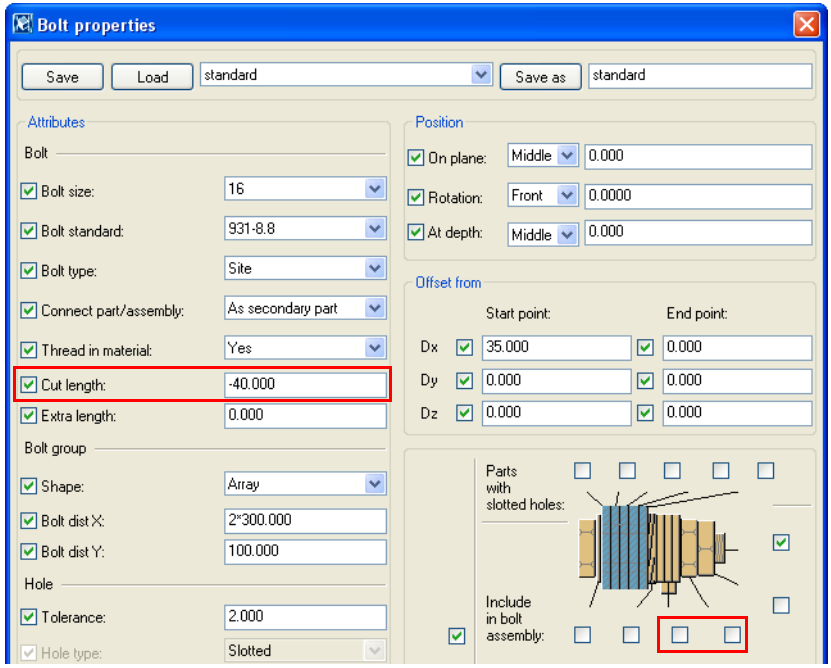
In the example above the bolt pattern is divided across two profiles. However, it is possible to divide bolt groups across several profiles.

Modeling bolts on cast-in embeds

You use the bolt-command to connect two (or more) parts to each other. The bolt length is calculated by means of a formula, using the material thicknesses of the connected parts and some other factors. You can also determine the bolt length manually.



Select, like in the example above, only the angle steel profile as the bolted part. Switch off the checkboxes for the nuts and the washers (possibly switch on the checkbox for the washer at the head of the bolt). For the cut length, enter a negative value:



1.25 Welds



This command is used to form assemblies. An assembly always consists of 1 main part and 1 or more secondary parts.

If the weld type and -size is vital, than the settings in the dialog box are important. In case of less important welds, i.e. a weld which is not necessary to be displayed on drawings, the settings are less vital. Only the weld size is important. This size has to be set smaller than the default weld size.

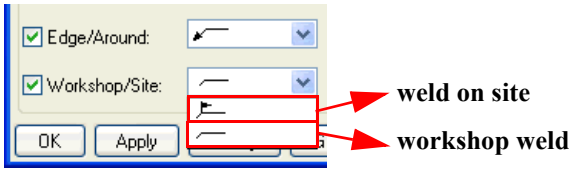
When applying welds, it is important that the **main part is picked first**, and than select the part(s) that have to be welded to that main part. The welding-order determines which part of the assembly will be the main part!

In most cases workshop welds will be selected. Of course it is also possible that welds on site are required, e.g. parts welded to an existing structure. In those cases in your model also select welds on site.



When you select welds on site, the parts are **not** welded actually. On the drawings, a flag will indicate the welds on site.

Do not use the commands for creating sub- and super assemblies to weld for example a stub to a beam.



Set new main part of assembly

In some cases, when you have welded for example a complex truss, it may occur that the assemblies' main part is not the correct main part anymore.

The assemblies' main part can be redefined very easily:

- Select the correct assemblies' main part
- Click the right mouse button and select **Set as new main part of assembly** in the contextmenu
- The selected profile is the main part of the assembly.

1.26 Select switches

The toolbar **Select switches** contains switches which control the selection of objects.

For example

If only the **Point** switch is active, Tekla Structures only selects points:



Dragging the mouse, i.e. holding down the left mouse button and drawing a rectangle with the mouse, selects all objects within that rectangular area and highlights them.



To select all objects in the model, you can make use of the command **Edit** > **Select all objects** or use shortkey <**Ctrl**> + **a.**



The command **Select all objects** takes the settings in the **Select switches** into account and also the setting of the select filter at that time, for example the column- or the beam filter.

The main advantage is that no objects will be forgotten when they are selected!



This command takes the filter settings of the view not into account! If, e.g. phase 1 is set, all the other objects in other phases will still be selected!

To select or deselect objects from the model manually, press the <**Shift**> or the <**Ctrl**> key while selecting the objects.

Key	Description
SHIFT	To add objects to the current selection
CTRL	To toggle the current selection on and off. Tekla Structures deselects the objects, which are already selected and selects those, which are not previously selected
ALT	To select the entire assembly containing that part
ALT Gr	SHIFT + ALT

The select switches are predefined to select all objects, accept single bolts (or holes) in a bolt group.

Explanation of the icons:

	Select all		Single bolts
	Connections		Reinforcing bars
	Parts		Loads
	Surface treatments		Planes
	Points		Distances
	Grid		Select component
	Grid lines		Select objects in components
	Welds		Select assemblies
	Cuts		Select objects in assemblies
	Views		Filter
	Bolts		Select filter dialog box

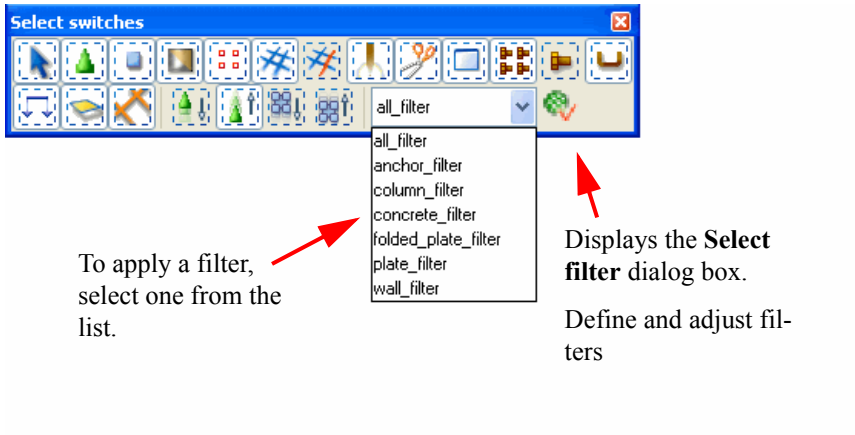


The switches **Reinforcing bars** and **Loads** are only in use with the Concrete and/or Analysis & Design add-ons to select reinforcing bars and loads.

Planes and **distances** are used for custom components.

Select filters

Tekla Structures contains several standard select filters. Using the Select Filter, you can select more specified objects. Using a filter, e.g. the **column_filter**, only columns will be selected in the model. The select filter list box lists the standard filters:



Defining your own filters

1. Use the **Display select filter** dialog button to open the **Select filter** dialog box.
2. Find a filter which is close to the settings you need.
3. Modify the settings, then enter a new name in the **Save as** field.
4. Click <**Save>**, then <**OK**> to exit.

   You can now select the new filter in the list.

Selecting

The icons in the **Select Switches** toolbar are used to define which objects can be selected:





If you activate switch **1**, you can select every level of an component, from the highest level downwards to loose parts in components.



If you activate switch **2**, the select order is opposite: from the lowest level (loose parts in components) to the highest level.

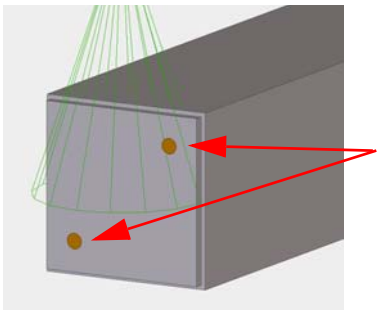


If you activate switch **3**, you can select every level of an assembly, from the highest level, through sub-assemblies, downwards to the loose parts.

You use this switch to copy objects correctly, by which all parts of an assembly are selected and copied. So with this switch you select **complete** assemblies.

Example

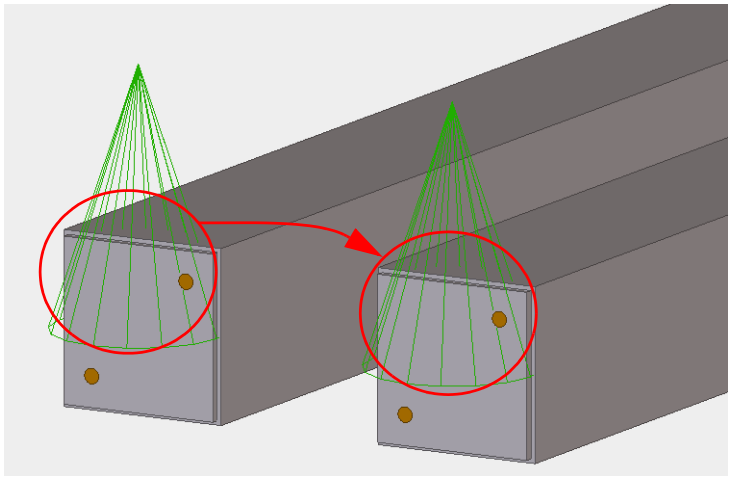
See the following situation:



A beam with macro End plate detail (1002).

Two zinc coating holes have been added apart from the macro, so added to the end plate manually.

Activate switch **3**. Select the beam and copy it:



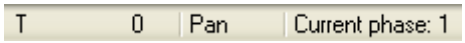
The selected assembly, including the manually modelled zinc coating holes are copied correctly.



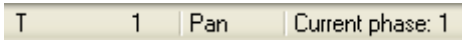
If you activate switch **4**, the select order is opposite:, from loose parts (lowest level) to complete assemblies (highest level).

The level of a selected assembly is displayed in the **Status bar**:

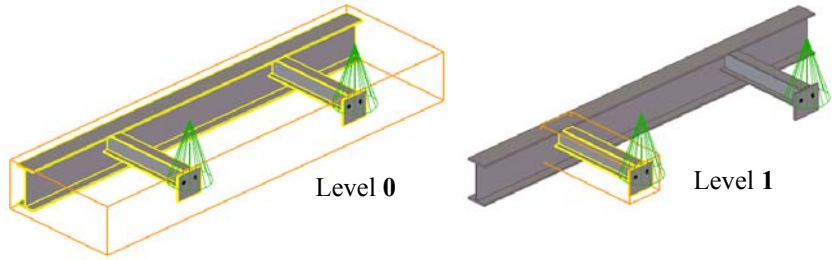
Level 0



Level 1



To shift to a specific level in an assembly, while the assembly is highlighted, hold down the <**Shift**> key and scroll the mouse wheel. The large assembly gets 0 (null), the smaller assembly gets 1, etcetera. You can continue to level 9:





The four switches above only work in rendered views, not in wire views.

Generating output



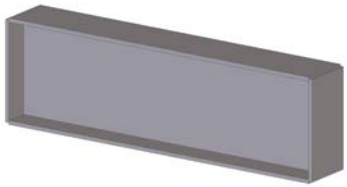
While generating reports, drawings, NC files or other types of output, you **always** make use of switch **2** to select all parts!



Switch **1** selects at the highest level, because of that, stiffeners or end plates, created by a component, will highlight but will not be selected!!!

Example

A beam + 2 end plates:





If you activate switch **1**, only the beam will appear on the list:

Profile	Part	Grade	Number	Length(mm)	Area(m²)	Weight(kg)
IPE300	Pr13	S235JR	1	980	1.14	41.4
				980	1.14	41.4
				Totaal:	1.14	41.4



If you activate switch **2**, both end plates and beam will appear on the list:

Profile	Part	Grade	Number	Length(mm)	Area(m²)	Weight(kg)
FLAT10*150	P7	S235JR	2	290	0.10	3.4
				580	0.19	6.8
IPE300	Pr13	S235JR	1	980	1.14	41.4
				980	1.14	41.4
				Totaal:	1.33	48.2

Super assemblies

You add ***sub-assemblies*** and parts to already existing assemblies. A sub-assembly is an assembly that is added to another assembly to form a hierarchical assembly. We use the term ***super assemblies*** to describe this type of assembly.

Using ***super assemblies*** is useful when you are modeling split trusses, brackets, railings or pipe racks, for example.

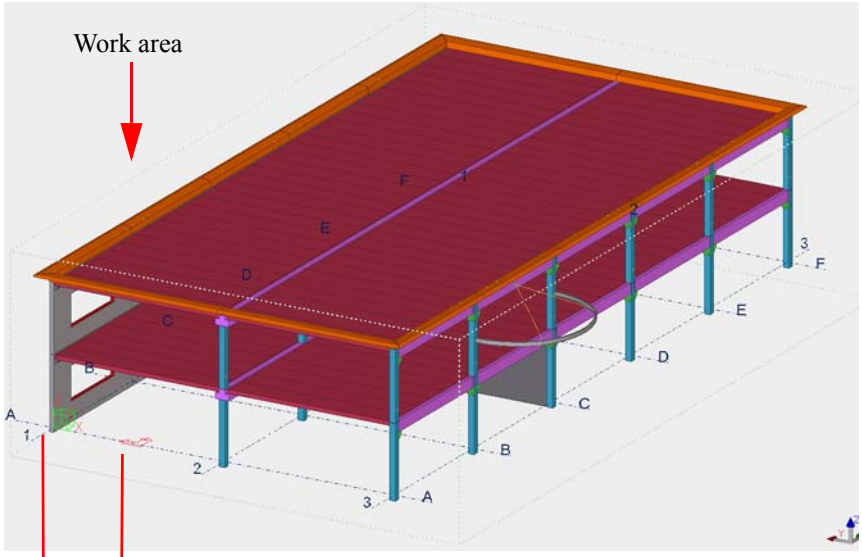
See also

For more information about super assemblies, see **Tekla Structures.pdf** in the folder:

disk:\TeklaStructures\version\environments\europe\pdf.

1.27 Work area

You can define the work area to one part for example but also to several parts or to the complete model or just a part of the model. Defining the work area makes it faster and easier to work with the model. Tekla Structures indicates the work area of a view using green, dashed lines.



Fit work area by parts in selected views



Use this icon to fit the work area by parts in selected views.

Select one or more parts and click on the icon **Fit work area by parts in selected views** or in the pull-down menu go to **View** > **Fit by parts**.

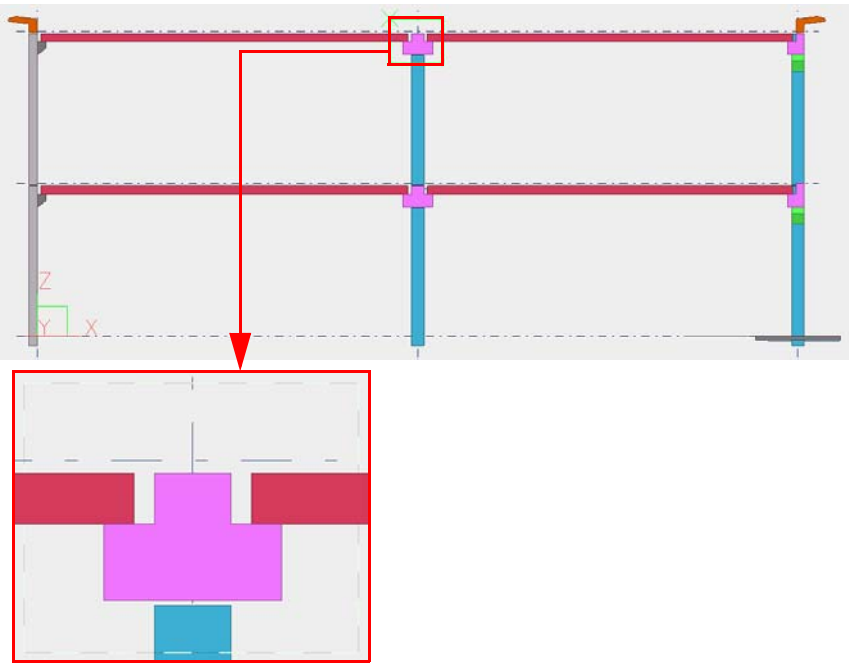
(Objects outside the work area still exist, but they are not visible).

Pick work area



Use this icon to define a work area to suit particular situations. You can shrink and expand the work area by picking the corner points of the new work area.

Click on the **Pick work area** icon or go to **View > Pick work area** and click two positions:



To size the work area to the grid:

1. Click the right mouse button and select **Fit work area** or
2. In the pull-down menu go to **View** > **Fit work area** > **All views** or **Selected views** or
3. Click on the icon **Fit work area in selected views.**

Fit work area in selected views



Use this icon to fit the work in selected views. The work area will size to the grid:

Click on the **Fit work area in selected views** icon or in the pull-down menu go to **View** > **Fit work area** > **Selected views/All views**.

Oversized work area in the model view

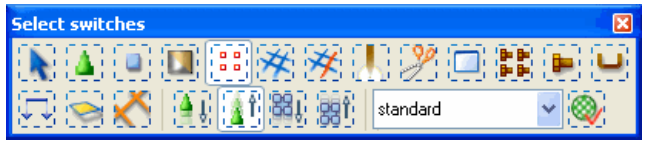
It may occur that the work area in a model view is oversized.

The reason could be that one or more points are positioned far away from the models' origin as a result of incorrect copying. Also reference models can be positioned far away from the models' origin.

In those cases, the command **Fit work area > All views** is not functioning anymore. To fit the work area to the model view, you need to do the following:

Points

1. Set the toolbar **Select switches** in a way to only select **points** and make sure the 2nd button is switched on:



2. Go to **Edit** > **Select all objects** or use <**Shift**> + A, all points will be selected
3. Now check the number of selected points in the status bar and press the button <**Delete**>
4. You will see that one or more points are deleted
5. Now fit the work area



Reference models

1. To be able to see the reference model in the model, go to **File** > **Reference models...**
2. Now select the reference model in question in the dialog box **Reference models**
3. Click the icon **Fit work area by parts in selected views**, the work area will be resized to the reference model



4. You can now move the reference model

Hide the work area

In Tekla Structures you can hide the work area. Because of this you have an even better view on the model.

Hidden

To hide the work area, keep pressed down the <**Ctrl**> and the <**Shift**> key and select in the pull-down menu **Window** > **Redraw all**. The work area is now hidden.

Visible

To display the work area, select in the pull-down menu **Window** > **Redraw all** or press shortkey **g**. The work area is visible again.

1.28 Create clip planes



In Tekla Structures you can create clip planes. You can create up to six clip planes in any rendered model view.

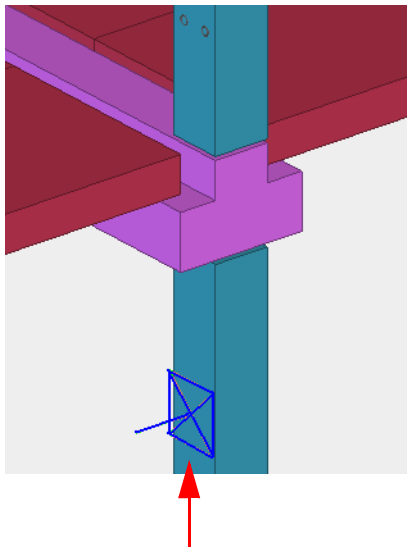
Step plan

1. Select a rendered model view
2. Right-click and select **Create clip plane** from the popup menu



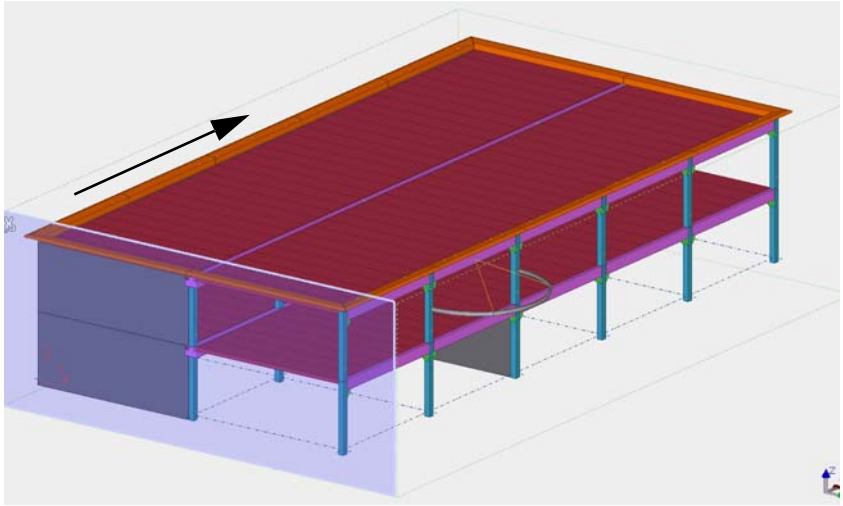
3. To select the position of the clip plane, click a plane
4. To finish, right-click and select **Interrupt**

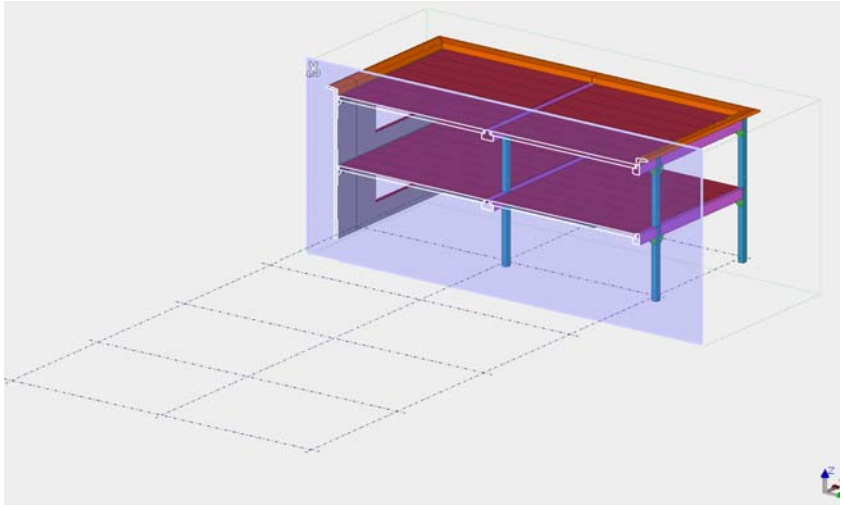
   The clip plane symbol appears in the model:



Move clip plane

To move the clip plane, click the symbol, and drag and drop it to a new location:





You can only move the clip plane in both plane directions.



You can change the location of the symbol by holding down the **<Shift>** key and moving the symbol.

Delete clip plane

To delete a clip plane click the clip plane symbol and click **Delete**.

1.29 Cast-unit assemblies

When you have modeled several separate parts from various material groups, and they should be combined to one element, then these separate parts can be connected to each other. The combined parts will be handled as one assembly, which will result in a cast-unit drawing which contains one assembly.

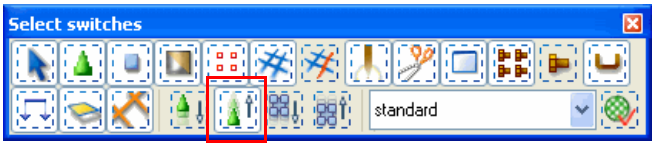
You can make use of the following options:

1. **Add to (concrete to concrete)**
2. **Add as sub-assembly (steel to concrete)**
3. **Weld (steel to steel)**
4. **Create part add (steel to steel)**

1. Add to (concrete to concrete)

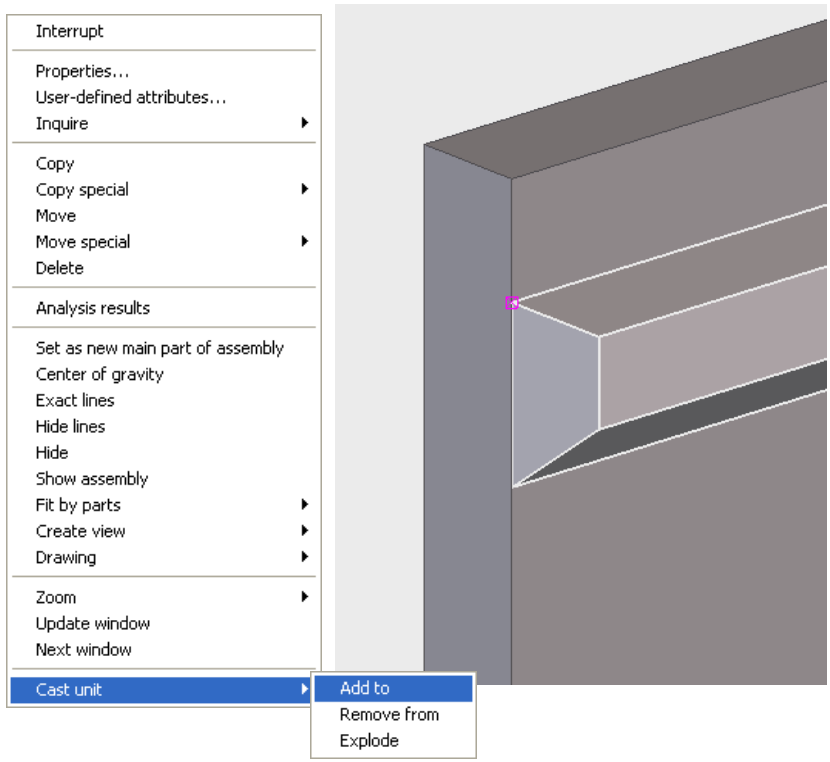
Step plan

1. Select the part that contains a material grade from the group Concrete that must be added by using **switch 2**:





2. Click the right mouse and select **Cast unit** > **Add to** in the pop-up menu, now select the part you want to add to the part:



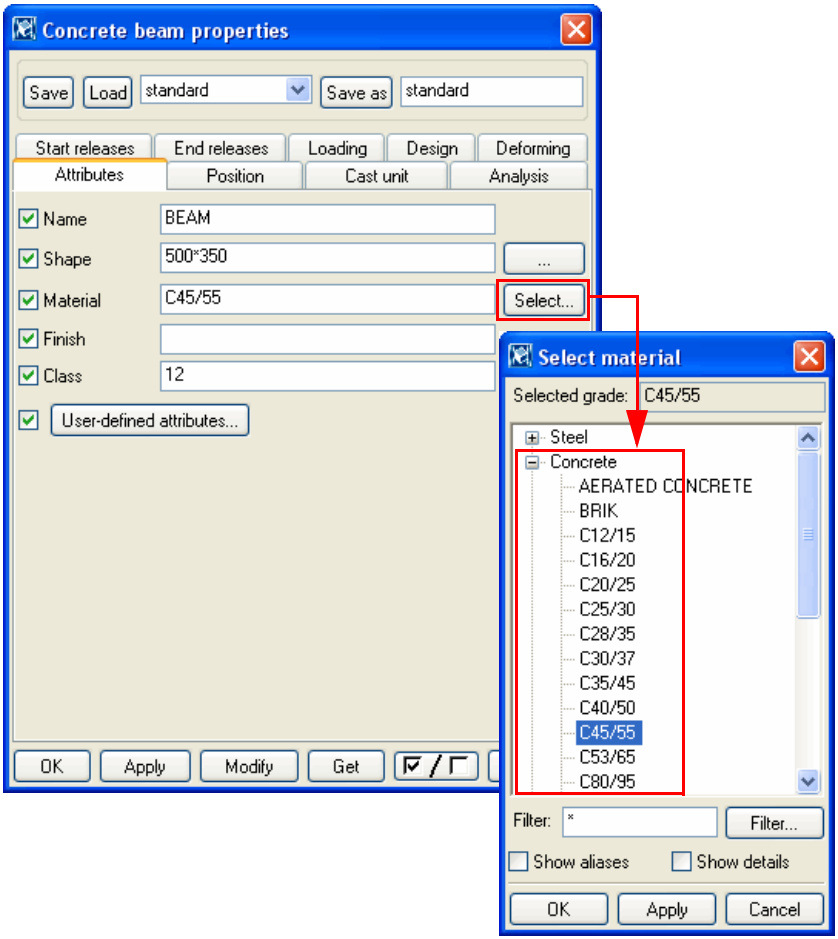


By using the icon **Create cast unit** in toolbar Create cast unit assemblies you can also create a cast unit that consist of one than just one concrete parts. Select the icon and all parts that should be included in the cast unit. Next click the middle mouse. The parts that consists of the **largest volume** determines the cast unit prefix. You can change this by selecting another part and clicking right mouse, next click **Set as new main part in assembly** in the pop-up menu**.**



If you want to add more than just one part at the same time to a precast cast unit, select the parts you want to add while keeping the <**Ctrl**> key pressed down.

The command **Add to** is used to create a precast cast unit from parts, that contain a material grade from the group Concrete:



Use the command **Add to** to create a precast cast unit from parts, that contain a material grade from the group Concrete.

Added parts from a precast cat unit can still be edited and modified at part level.

Remove from

To remove part from a precast cast unit, select the part, click right mouse and select **Remove from.** Now select a part from the from the precast cast unit.



You can also use the icon **Remove from cast unit.**

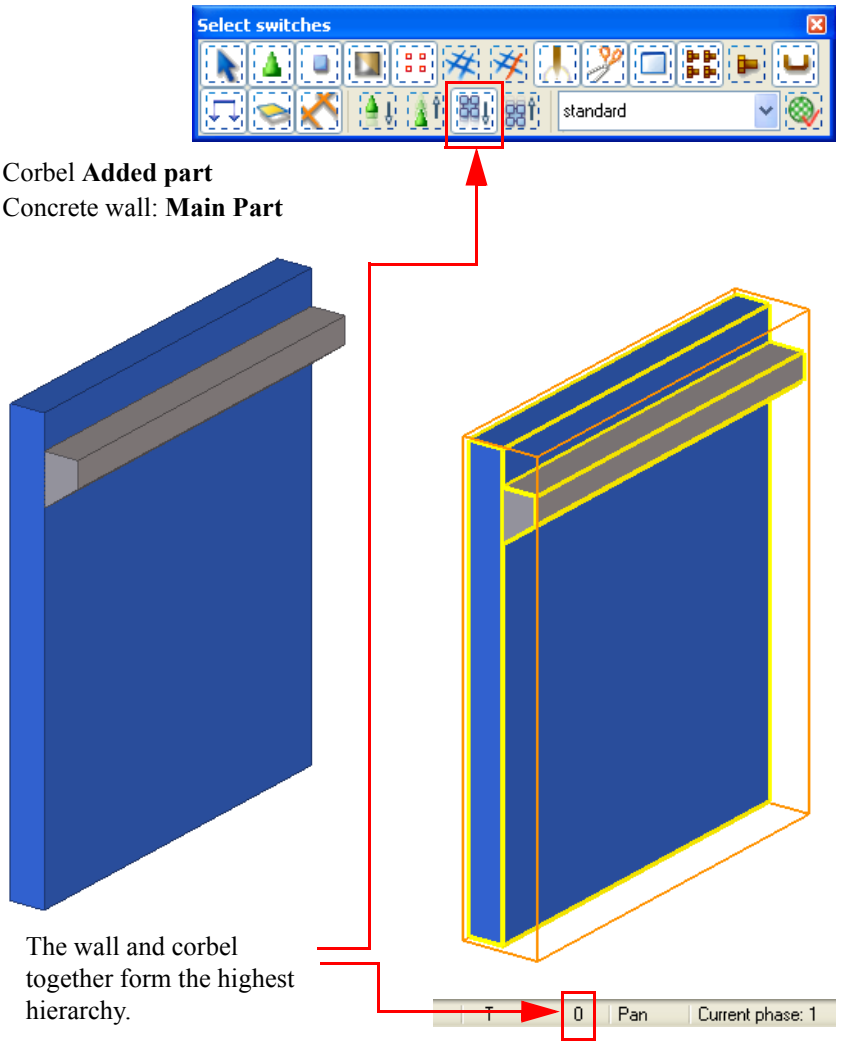
Explode

You can also explode a precast cast unit; in that case all added part will become loose parts. For this, select an added part, click right mouse and select **Cast unit > Explode.**



You can also use the icon **Explode cast unit.**

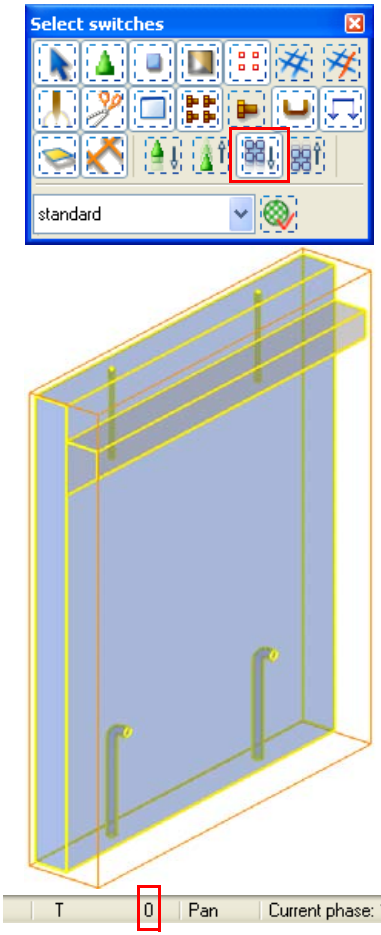
A cast unit of a concrete element can consist of multiple concrete parts. All concrete parts in the cast unit are in the highest hierarchy level:



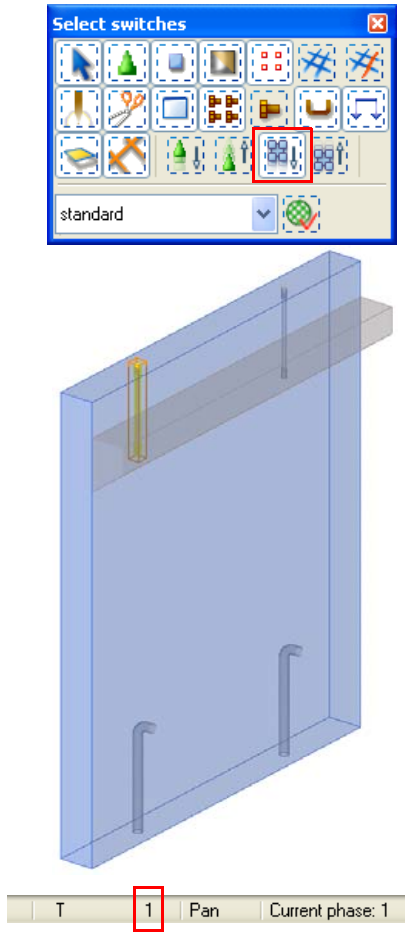
2. Add as sub-assembly (steel to concrete)

All other parts that should be added (so except the concrete shape) and included to the precast cast unit, must be added at a lower level, i.e. cast-in embeds such as boltanchors, liftanchors, pins, gains, welded plates, etc.

Similar parts must be added as a sub-assembly to create a desired difference in levels in the hierarchy.



The wall, corbel, starter bars and gains form the precast cast unit.

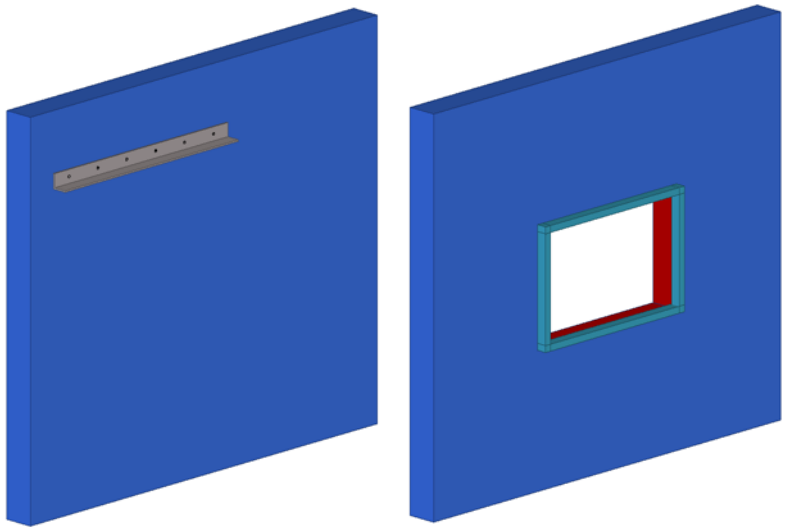


The starter bar is a sub-assembly at level 1.

You can not create drawings from welded (steel/timber) parts in the module PCD or RCD, you can in the module FUD. If it concerns cast-in embed, such as anchors, this is not a problem, those parts are repurchased parts and created already. It may occur that a steel beam or a steel assembly must be casted and separate drawings of the beams or assemblies should be generated.

Examples

An Angle-steel or a wooden window frame needs to be added to a wall:



If you use a FUD module (Full Detailing), you can create welds and single part and assembly drawings of steel parts.

If you use a PCD module (Precast Concrete Detailing), you can not create single part- and assembly drawings. In that case, you can create a Cast Unit drawing of steel parts.

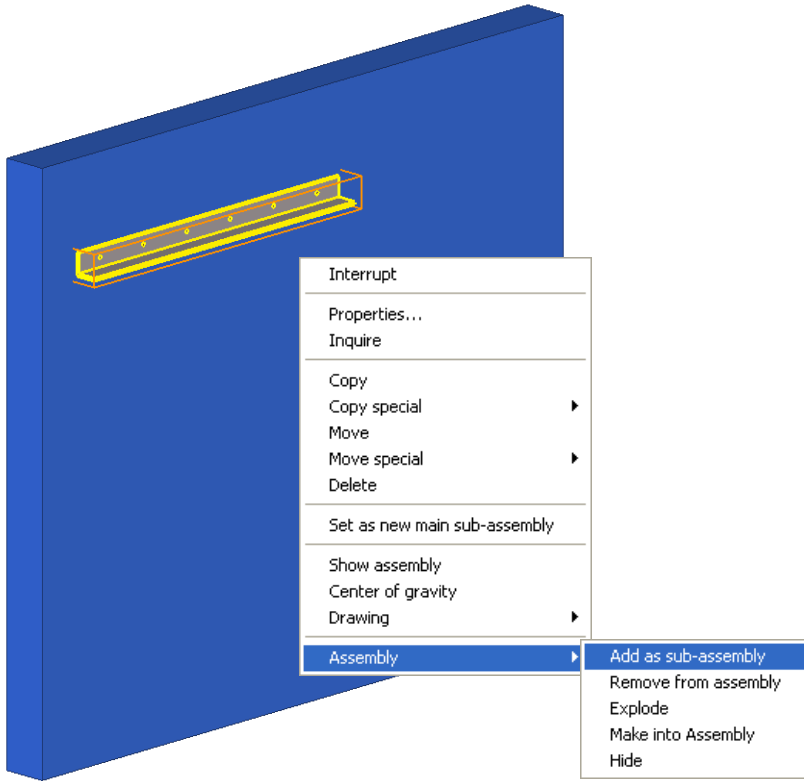
These steel parts must be added to the cast unit as a sub-assembly:

Step plan

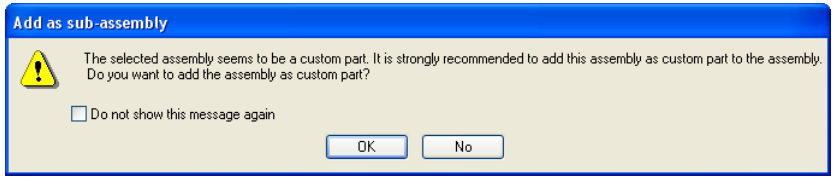
1. Select the sub-assembly using **switch 3**:



2. Next, click right mouse and select **Assembly** > **Add as sub-assembly**. Pick the cast unit:



If steel parts are added, the following message appears:



3. Click <**OK**>, the part will be added automatically as a sub-assembly.

Sub-assemblies can consist of one or multiple parts. In both cases the parts have to be added to the cast-unit as a **sub-assembly**. If the sub-assembly consists of multiple parts, than these (steel) assembly can be modelled in two ways: by **Welding** or by using the **Part Add** command.

3. Weld (steel to steel)

In case you want to cast in a steel assembly, that consists of more then one parts, to a precast cast unit, the steel secondary parts must be welded to the steel main part.



Click the icon **Create weld** and select the main part and the part(s) you want to weld.

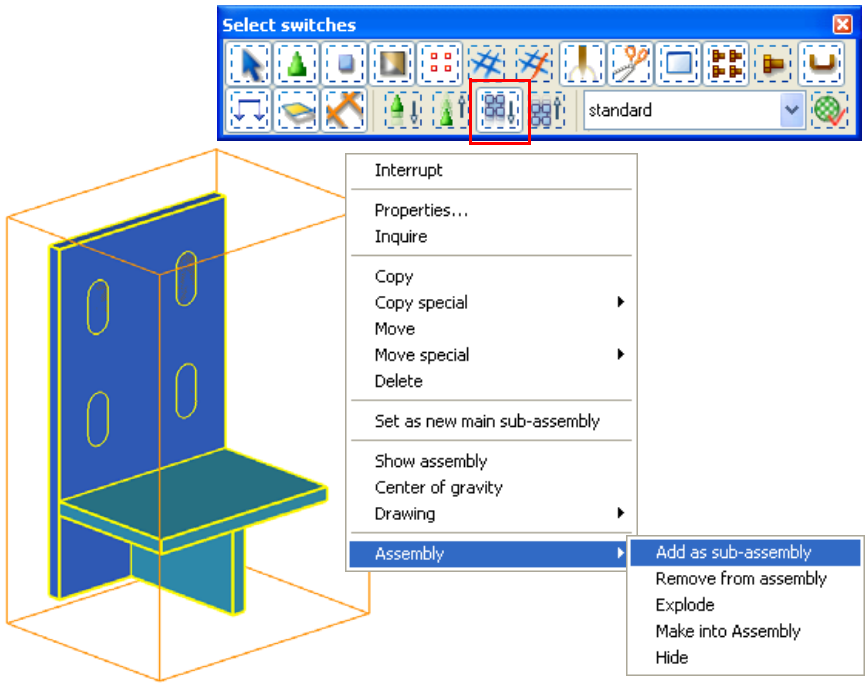
When applying welds, it is important that the **main part is picked first**, and than select the part(s) that have to be welded to that main part. The welding-order determines which part of the assembly will be the main part!

Set new main part of assembly

The assemblies' main part can be redefined very easily:

1. Select the correct assemblies' main part
2. Click the right mouse button and select **Set as new main part of assembly** in the menu

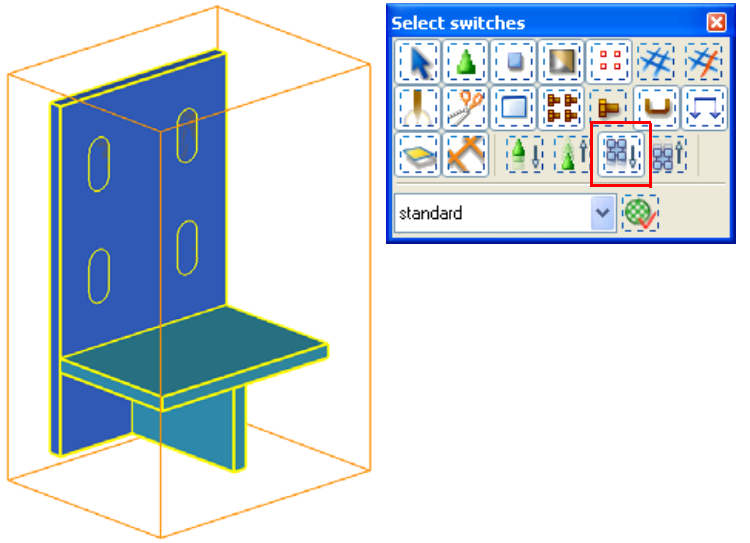
The selected profile is the main part of the assembly. The entire steel assembly must be added as a sub-assembly. Mind **switch 3** to be switched on:

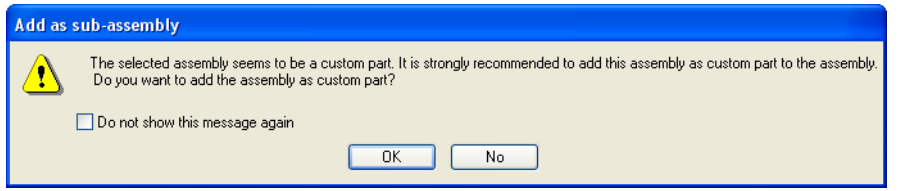




If you create a "Part" type custom component of a steel sub-assembly, make sure **switch 1** is switched on.

If you add a "steel" "Part" type custom component as a sub-assembly using **switch 3,** the following message appears:





By clicking <**OK**>, the steel sub-assembly will still be added as a component.



Never weld concrete parts to other concrete parts, or steel parts to concrete parts.

Only use the weld command to connect steel (sub)assemblies.

4. Part Add (steel to steel)

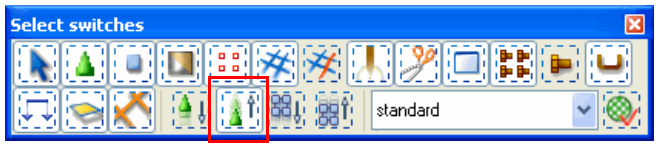
The **Part Add** command can be used to join multiple parts to one part. This action is permanent!

The added part can not be exploded or edited afterwards. Do not use this command to form cast units.

This command can be used to join purchase articles, such as a dowel-ins which is build from several parts; an advantage is that the joined part will get just one part mark on drawings.



Click the **Part Add** icon in the **Detailing** toolbar, select **switch 2** and pick part to be added to. Next, select the part to be added.





Do not use the command **Create part add** to combine concrete parts to one precast cast unit. For this, only use the command **Add to**.

Inquire Assembly for Cast Units



Activate command **Inquire** >**Assembly** and select a part of the cast unit. The main part (the largest volume which defines the assembly prefix) will be colored purple and all concrete additions will be cyan colored.

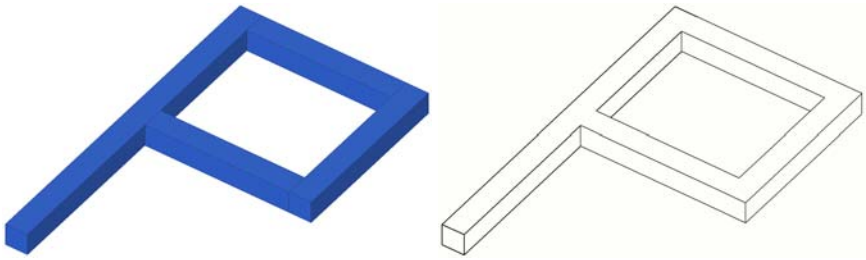
Imbeds having non-concrete material will be highlighted in yellow, and possibly parts welded to the imbeds will be highlighted in yellow as well.

Steel parts which are not added to a cast unit, will be highlighted in a red color. Parts welded to this steel part will be highlighted yellow.

1.30 Modeling with concrete parts

Contact planes in cast units

If cast unit edges touch each other, the coinciding lines will not be visible on cast unit drawings. However, these lines remain visible in the modeling environment:



Model: contact planes visible **Drawing**: no lines

For part's contact planes for parts which are not added using the Part Add command, a variable is available to define whether the lines are visible or not.

If you want the contact lines being visible on cast unit drawings, set the variable `XS_DRAW_CAST_PHASE_INTERNAL_LINES` in **Tools** > **Advanced options** > **Concrete detailing** to TRUE.

If you want to hide the contact plane lines, set the variable to FALSE.



Default settings for this variable are depending which module is in use. Go to **Advanced options** to find out which setting is being used.

If the planes don't (just) coincide, the part edges will be visible on drawings.

Copy variable `set XS_HIDDEN_LINES_CHECK_TOLERANCE=0.1` from file `env_global_default.ini` (located in folder *..\environments\country-independent*) to file `user.ini` (located in folder *..\nt\bin*) to enter a margin for contact lines being visible of being hidden on drawings.

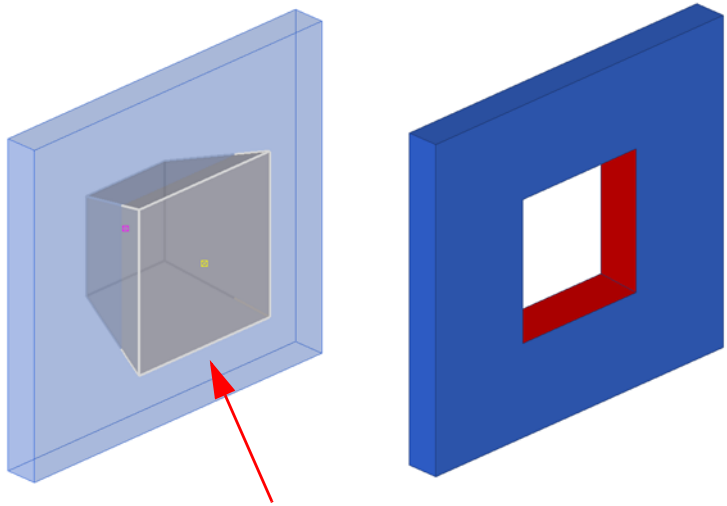
If parts are not added to each other (using the Part Add command), then the contact planes will always be visible on drawings.

Part Cut

To add a recess to a cast unit, you can use the **Polygon Cut** command. If the recess needs to be drafted (for releasing), this command in not suitable; in that case a recess-shaped parametric plate should be used and cutted out using the **Part Cut** command.

The original parametric plate may be deleted. The remaining negative volume can be moved, copied and modified afterwards.

Another option for creating cut-outs is the command **Detailing > Weld preparation**. The cut out is being created and the original plate will be removed also:



For example parametric profile
PL_V1000*1000-600-600-0-0

Editing Cast Units

The shape of a cast unit is determined by the used profile (width and height) and the length (except contour plates).

The length is determined from the picked points. These points can be moved afterwards and the length can also be modified by applying fittings (possibly by using components).

The height and width can be changed by editing the part properties.

For example, if the height of a wall needs to be modified, do this by editing the part properties. Do not apply fittings or polygon cuts; this may result in incorrect reports and bad functioning of some components.

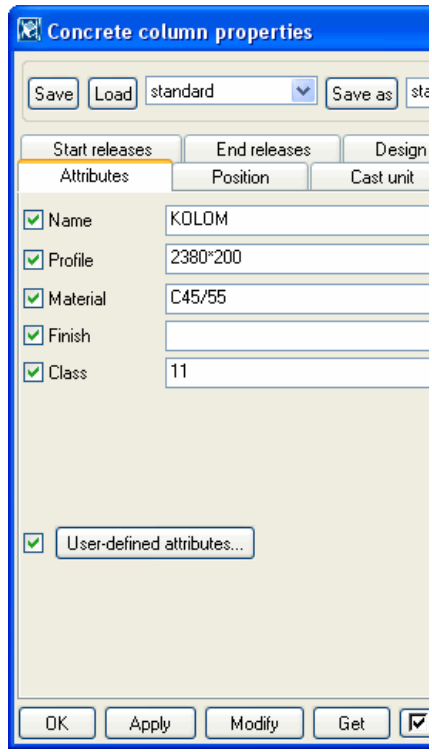


Never use fittings, Line cuts and/or polygon cuts om change the width of a cast unit.



Example: to change the wall width to 2380:

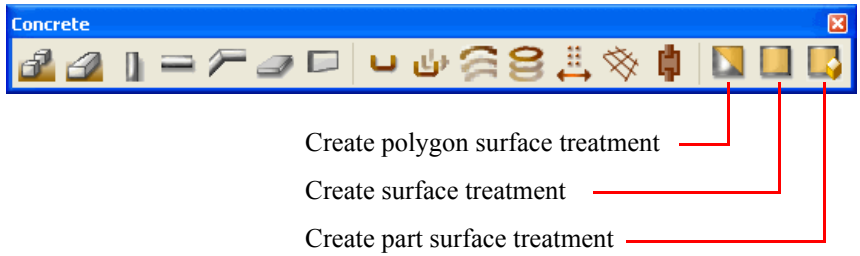
Do not use Line Cut, but modify the **Profile Properties**.



Pour side

Using **Surface treatments**, you can define the pour side of a cast unit. You can use the command for many purposes (e.g. brickwork indication) but this paragraph only describes the pour side.

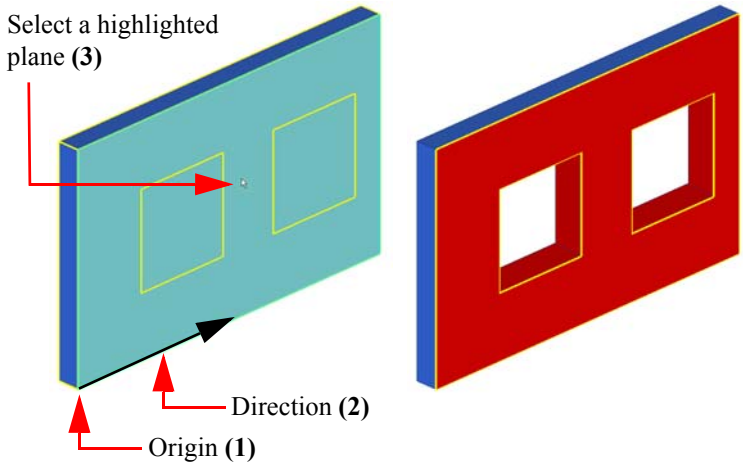
In the **Concrete** toolbar, 3 commands are available for applying surface treatments:



To indicate a side to be the pour side, double-click the icon **Create surface treatment.** Load the setting **pouring side** and click <**OK**>.

Select the origin and the surface direction. For the origin, choose the lower left corner of the pour side, for the direction pick a point in the part's linear direction.

Click the plane to be the pour side, move the mouse pointer over a part; you will see the parts planes to be highlighted:



Part Cuts and Polygon Cuts are taken into account when applying surface treatments. If the cuts are modified, the pour side will updated automatically. But if the length of the concrete part is being changed, the surface treatment will not be adapted automatically.

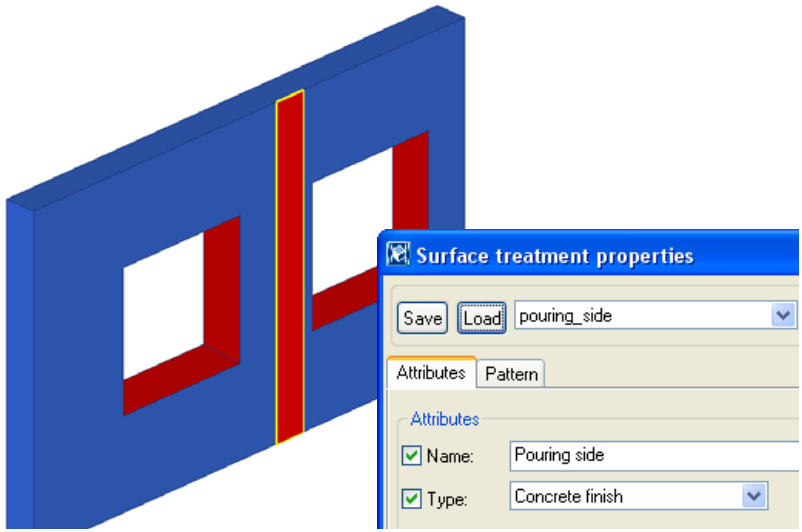
In that case, the surface treatment must be replaced, or the size must be changed by moving the handles.

Pour side affects numbering!

Using command **Create polygon surface treatment**, you can apply a surface treatment to a selected area. For example, if an area needs a rough finish for later pouring, this area can be defined using command **Create polygon surface treatment**.

Double-click the icon **Create polygon surface treatment**. For **Type** and **Surface treatment name**, you can pick a predefined finish. This picklist can be extended by modifying file `concrete_product_finishes.inp`. This file is located in the *system* folder. Select the desired finish.

Define the origin and the direction. Next, click the part the surface treatment is being applied to and pick a polygon.



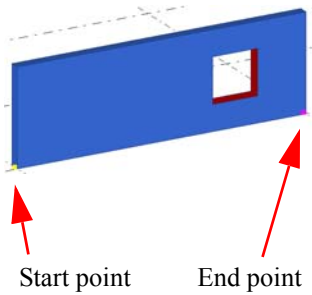
See also

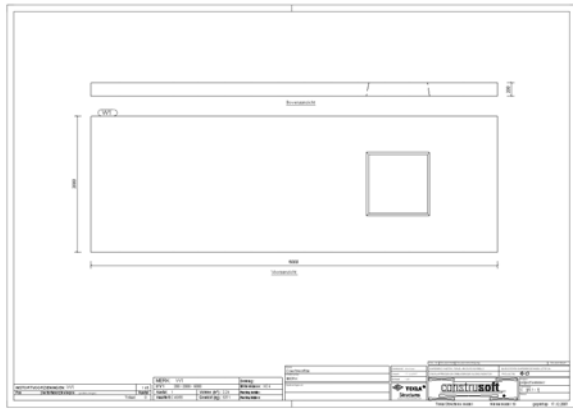
For more information about surface treatments, see **Tekla Structures.pdf** in folder **\\TeklaStructures\versie\environments\europe\pdf**.

Placing parts on a drawing

An applied surface treatment does not affect the parts orientation on a cast unit drawing. Especially for walls, it is common that the front side on a drawing is the pouring side, but also for other parts (beams, columns) it is useful to know how the part will be orientated on a drawing and to take the parts' orientation into account while modeling.

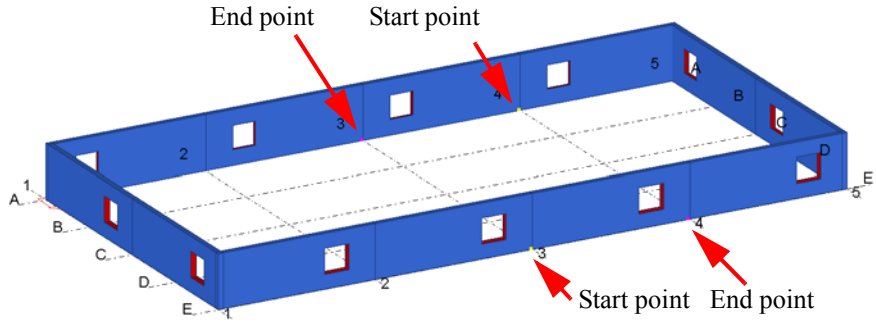
On cast unit drawings, parts are placed according to the parts' local direction. This means the direction between the parts' reference points (yellow/ magenta points). The right plane is will be the front view on the cast unit drawing.





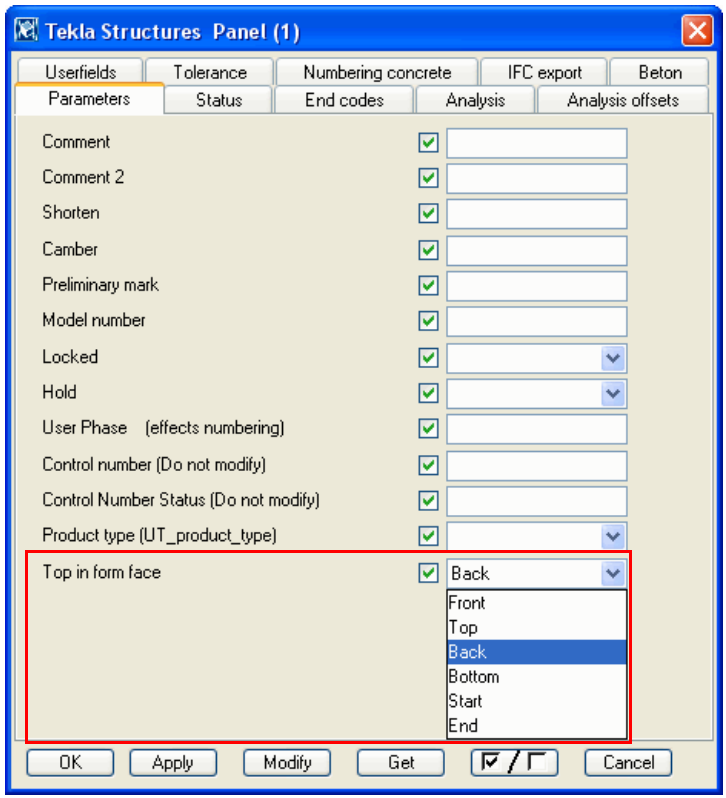
Right plane is front view

If the pour side needs to be on the outside of the building, you have to take this into consideration while modeling the building.

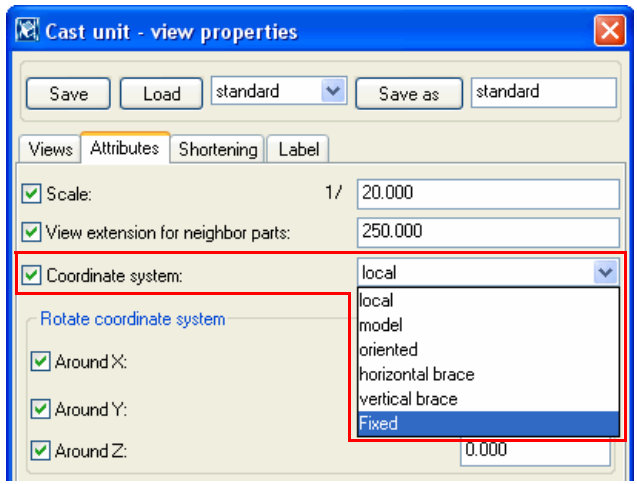


To switch the pouring side after a cast unit has been completed (the cast unit shape, imbeds and rebars are added and drawings are created), follow this step plan:

1. Modify the parts' User Defined Attributes; in tab **Parameters**, change the option "Top in Form face" to `Back`:



2. In dialog **Cast Unit properties**, go to **View** > tab **Attributes** and change "Coordinate system" to `Fixed`:



1.31 Work plane

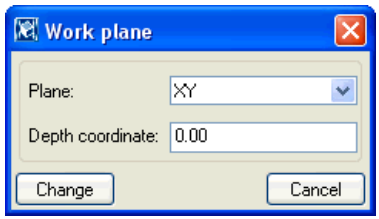
The red coordinate arrow symbol indicates the work plane, which is the current local coordinate system of the model. The arrow symbol shows the xy plane. The z direction follows the right-hand rule. The work plane is the same in all views.

You can set the work plane manually. This is handy while modeling sloped parts, i.e. rafters.

Set work plane



Use this icon to set the work plane parallel to the xy, xz, or yz direction. You can also move it to a new position. Double-click the icon or go to **View** > **Work plane** > **Work plane…**, the following dialog box appears:



Select a plane and/or a depth coordinate and click **Change**.

To set the work plane to be the same as the view plane of a selected view, use the icon **Set work plane to view plane** or go to **View** > **Work plane** > **To view plane.**

Set work plane by 3 points



Use this icon to set the work plane using three picked points. The first point you pick is the origin, the second defines the x direction and the third defines the y direction of the work plane. Click the icon or go to **View** > **Work plane** > **With three points.**

To set the work plane to be the same as the view plane of a selected view, use the icon **Set work plane to view plane** or go to **View** > **Work plane** > **To view plane.**

Set work plane to view plane



Use this icon to set the work plane to be the same as the view plane of a selected view. Click the icon or go to **View** > **Work plane** > **To view plane.**

Set work plane to part top



Use this icon to set the work plane parallel to the top plane of a picked part.

Click the icon or go to **View** > **Work plane** > **To part plane** > **Top.** Now select a profile.

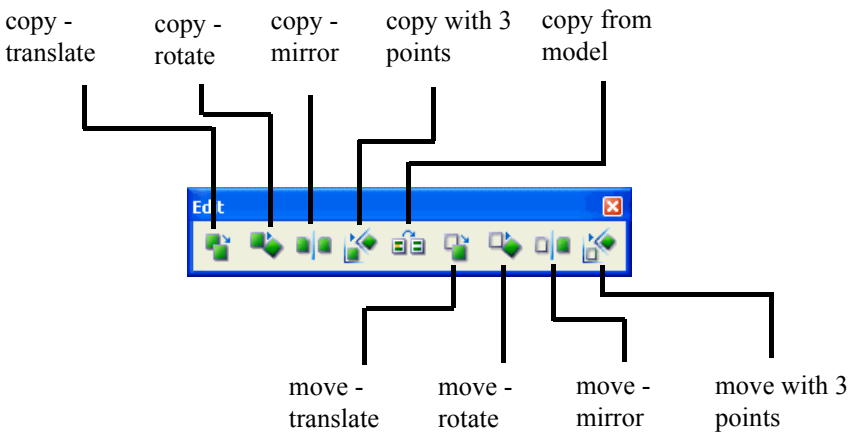
When a view is created, you initially work in the plane that belongs to that view. If you want to work in e.g. the roof plane, assuming that the roof is tapered, than you need to displace the work plane.

To set the work plane to be the same as the view plane of a selected view, again use the icon **Set work plane to view plane** or go to **View** > **Work plane** > **To view plane.**

1.32 Copy, translate and mirror

When copying beams, columns or other profiles, Tekla Structures automatically takes the accompanying joints, welds, fittings, line cuts and bolts into account. When copying parts, only the main parts have to be selected.

The icons for copying, moving and mirroring commands are located in the **Edit** toolbar.



When moving parts, the bolts and welds have to be part of the selection.



With the **copy**-, **translate**- or **mirror**-commands, data from a-symmetrical bolt groups and components will not be mirrored. Only the position of the macro will be mirrored.

Mirroring an asymmetrical column

When an asymmetrical column (for example UNP240) is mirrored, the mirrored profile will get beam properties. However, the original column will retain it's column properties.

To give the mirrored column the appropriate properties, do the following:

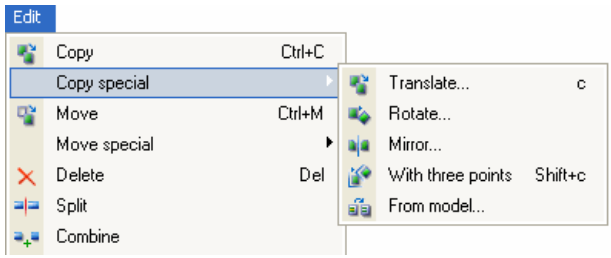
1. Select the mirrored column
2. Double-click the Orthogonal beam icon
3. Click <**Get>, <Modify>, <Apply>** and **<OK>**

Now double-click the mirrored column and you will see that the column has the correct properties.

Copy

To copy objects, Tekla Structures includes the following options:

- **Copy special > Translate** (shortcut **C**) to copy selected objects and move them to an entered distance from it's original
- **Copy** (shortcut **Ctrl + C**) to pick multiple locations to easily create multiple copies of the selected objects to picked locations



Copy special > Translate

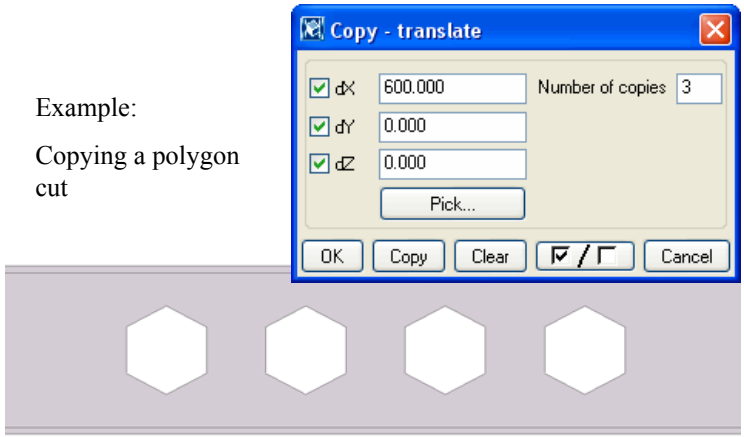
To copy objects to a new destination:



1. Select the object(s) to copy
2. Click **Edit** > **Copy special**, the following dialog box below appears, now there are two options:
   - either enter the distance(s) in the dialog box
   - or pick an origin and a destination point to select the translation distance with the mouse. The distance between the points will be calculated automatically and displayed in the dialog box.
3. Click <**Copy**>, the parts will be copied.

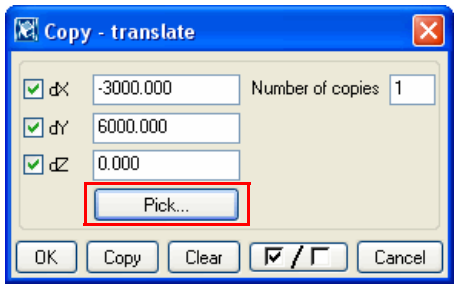
Example:

Copying a polygon cut



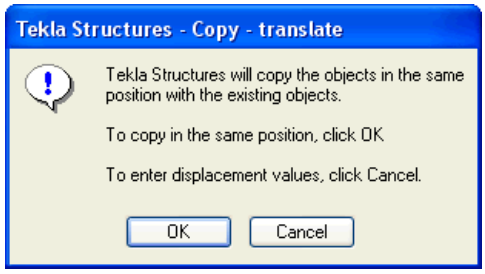
Pick

The **Copy - translate** and **Move - translate** dialog boxes contain a new **Pick** button. When you want to pick a new destination point for copying or moving, and you already have the dialog box open, but the command is not active anymore, you do not have to start the command again, just click the **Pick** button.



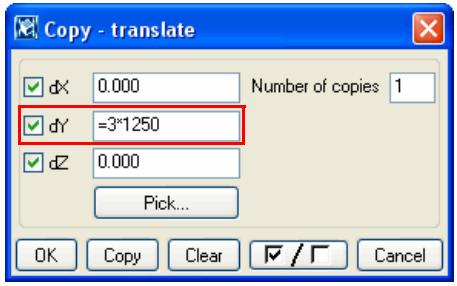
Warning

When you try to copy an object to its current position, and the fields are set to zero, Tekla Structures displays the following warning:



Use formulas in Copy and Move

To copy a part to three spans away, enter 3* and the size of the span in the field dY. The formula must begin with the equals sign (=):

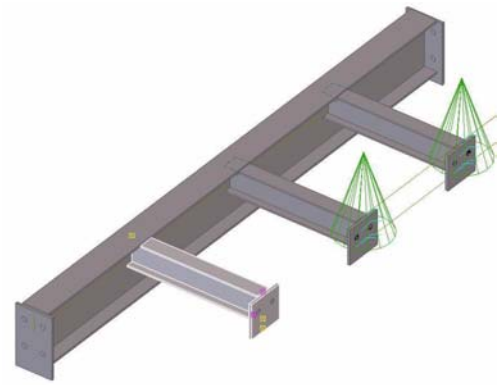


Copy

To copy objects to a new destination:

1. Select the objects to copy.
2. Click **Edit > Copy**.
3. Pick the origin for copying.
4. Pick the destination point. The objects are copied immediately.
5. You can continue and pick other destination points, and the objects are copied to all these locations.

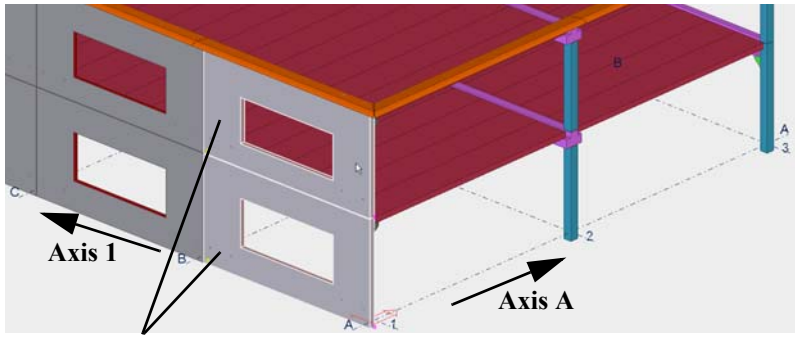
   If you click **Edit > Undo**, the latest copy operation is undone, but the **Copy** command remains active.
6. When all copies have been made, right-click and select **Interrupt** from the pop-up menu.



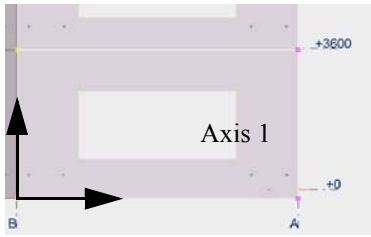
Copy / move with three points



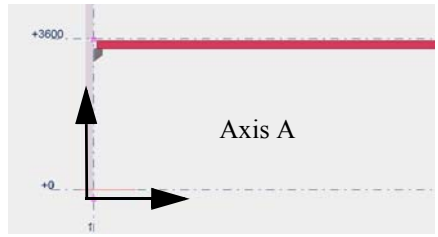
Use these commands to copy or move the selected objects from one plane (Axis 1) to another plane (Axis C) and possibly a 3rd plane, etc.



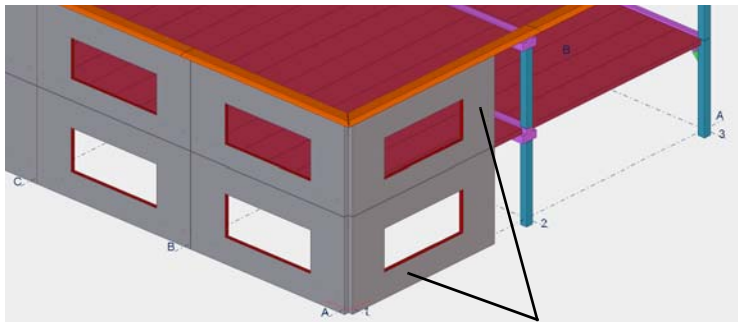
Select objects



Define Origin and position



Define Origin and position



Copied objects



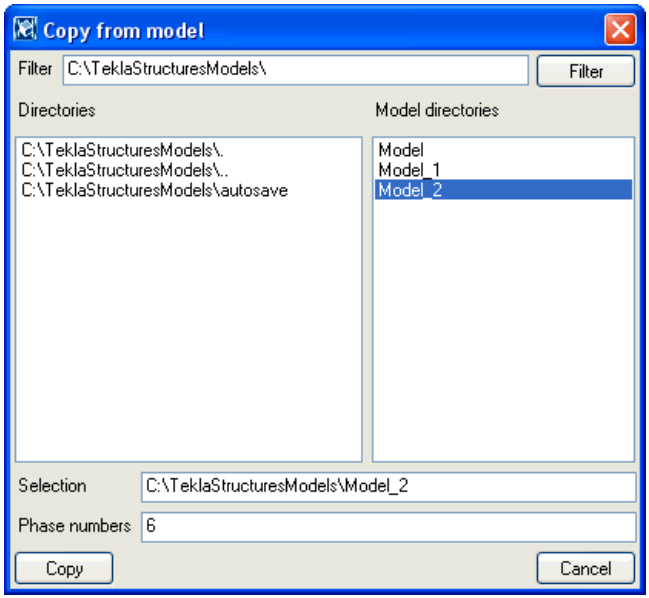
If the initial situation does not match the new situation, some or all macros have to be inserted again!

Copy from model



In the dialog box **Copy from model** you can copy phase numbers from an existing model to the current model. You can use this command also when two or more people are working in the same model.

In the dialog box **Copy from model** you have to define the model and it's phase number(s) which have to be copied to the current model.



When inserting several phases, the different phase numbers have to be separated by a space character.



Drawings can not be imported into the model!

Move



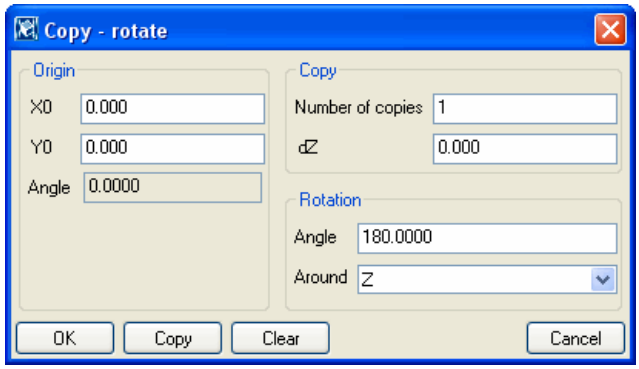
The **Move** commands work the same way as the Copy-commands, the only difference is that the objects are moved (i.e. the original object does not remain).

Rotate



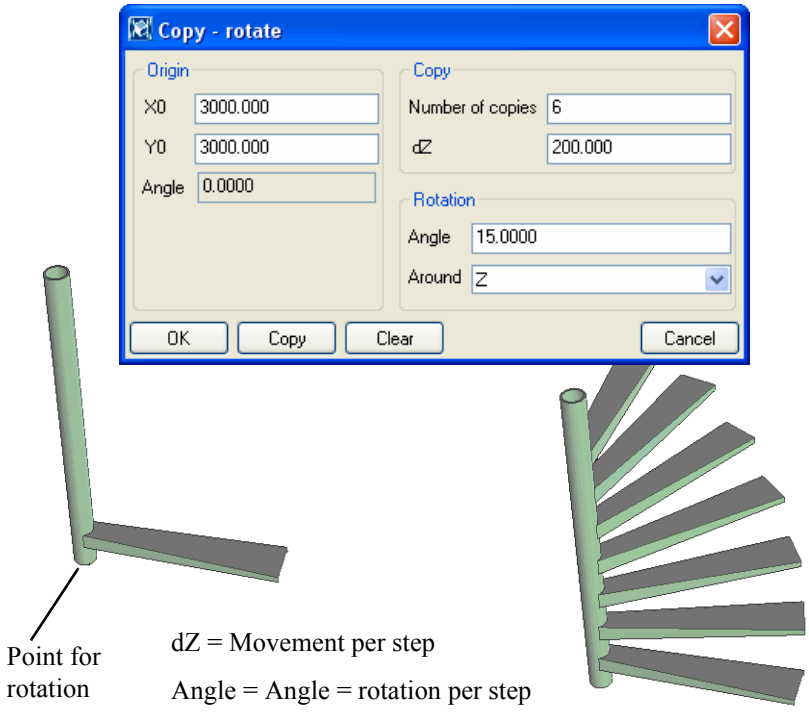
With the commands **Copy > Rotate** and **Move > Rotate** it is possible to rotate objects around a certain axis.

1. Select the object(s) to be rotated
2. Click the **Copy** > **Rotate** icon. The dialog box below appears:



3. Enter the values in the dialog box or modify the setting, then click <**Copy**>

Example



Mirror



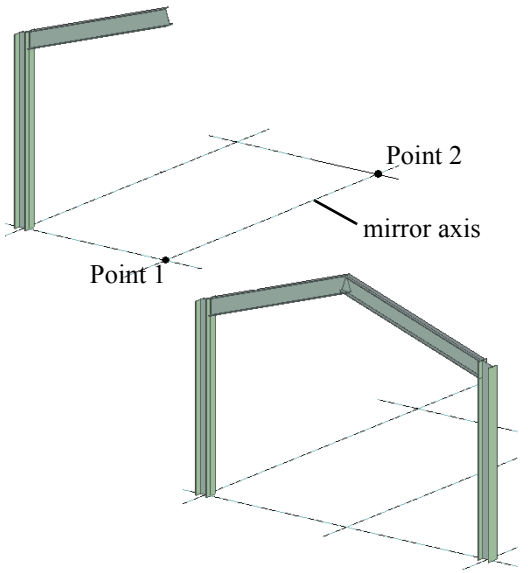
With the commands **Copy/ Mirror** and **Move/ Mirror**, objects can be mirrored through a defined axis.

1. Select the objects to be mirrored. (the main parts only, macros will be automatically taken into account!)
2. Click the icon **Copy/ Mirror** or **Move/ Mirror**
3. Select the line which will be used for mirroring the object(s)
4. Click <**Copy**>, the objects will be mirrored



When mirroring objects, be sure that the work plane is always equal to the defined mirror line

Example





Always modify asymmetric macro's: the direction has to be changed!



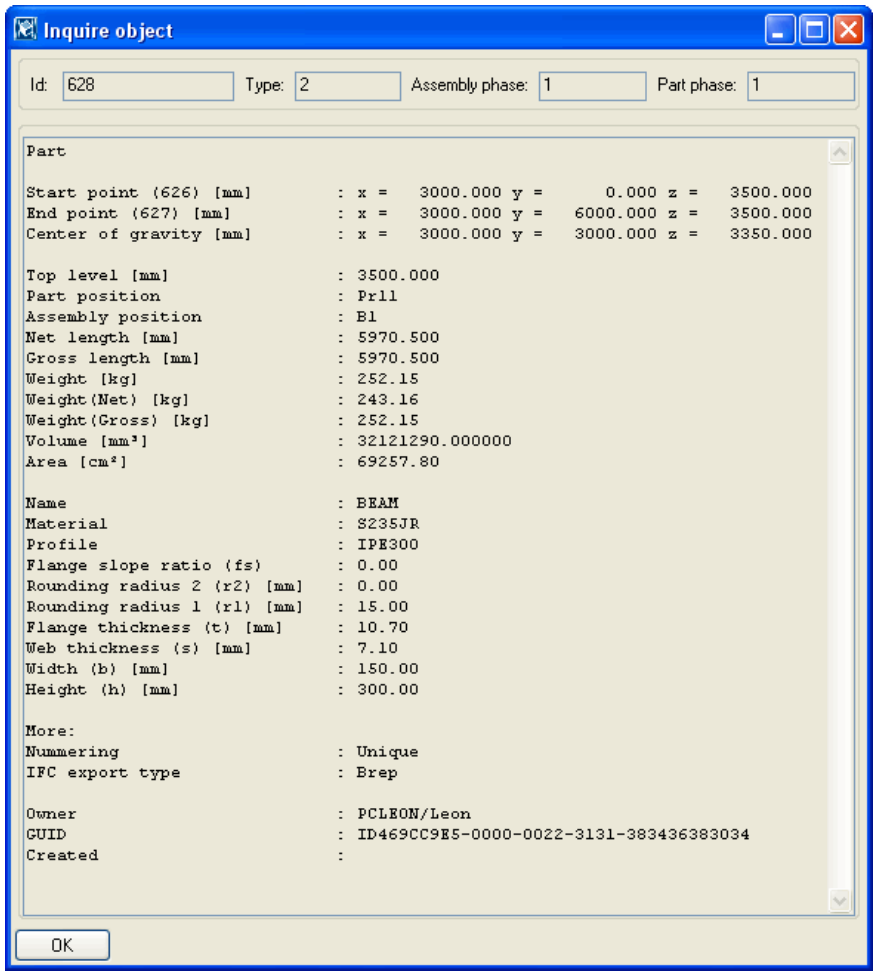
The command **Rotate** is always preferred to the command **Mirror**!

1.33 Inquire

Inquire object



The command **Inquire > Object** will show all relevant information of the object (part, bolt, macro, weld, etc.) in the **Inquire Object** dialog box. Which information is displayed depends on the object that is selected. The selected object and **all** parts of the object will be highlighted.



An other method to ask which parts belong to a bolt pattern is to select the bolts, right mouse button and select **Bolt Parts** from the contextmenu.

Using this method, it is also possible to change the profiles which are part of the bolt connection!

Inquire assembly



This command shows which part is the main part of an assembly en which are the secondary parts. The profile highlighted in **red** always is the main part, the secondary part(s) are highlighted in **yellow**.



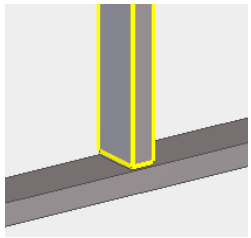
During modelling, you better not use the red and yellow color as a profile color.

Material from the “CONCRETE”-category will be highlighted in **magenta**, possibly added parts will highlight in **light blue** instead of red or yellow.

This command is also accessible from the pop-up menu (a part has to be selected first).

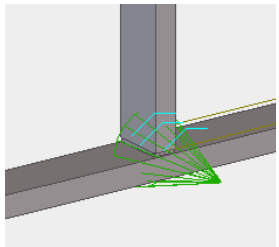
Displaying all parts’ objects

In some cases it may be useful to see all parts’ objects, for example to examine if a part is welded incorrect:



Step plan

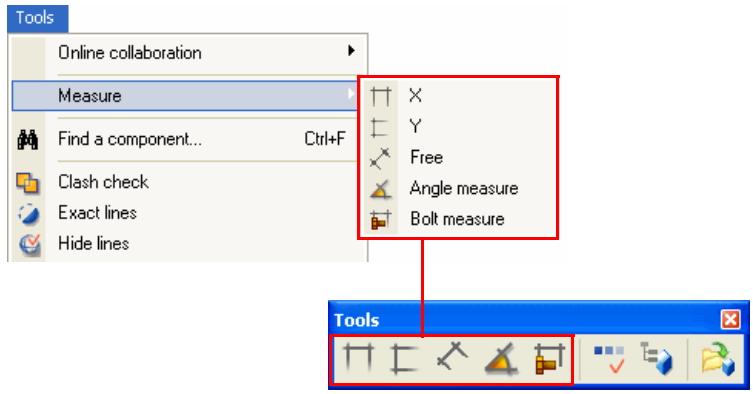
1. Select the part
2. Click <**Delete**>
3. Press <**Ctrl**> + **Z** or click the <**Undo**> icon



The part is now displayed, including all accompanying objects, such as components, welds, fittings, etc.

Dimensions and angles

While modeling it can be very handy to check dimensions or angles. Go to **Tools > Measure** or click one of the icons:

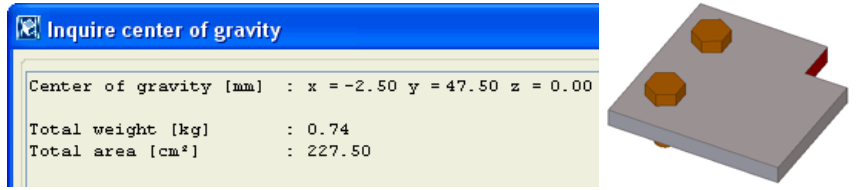


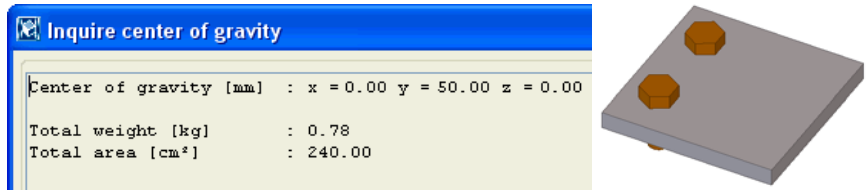
Center of gravity

To inquire about an objects´ center of gravity (assembly or part) you must select an object, now select **Inquire > Center of gravity** or click the right mouse button and select **Center of gravity.**



When calculating the center of gravity of an object in Tekla Structures, bolts and welds are **not** taken into account. Polygon- and part cuttings, just like fittings and line-cuts **are** taken into account.





1.34 General about custom components

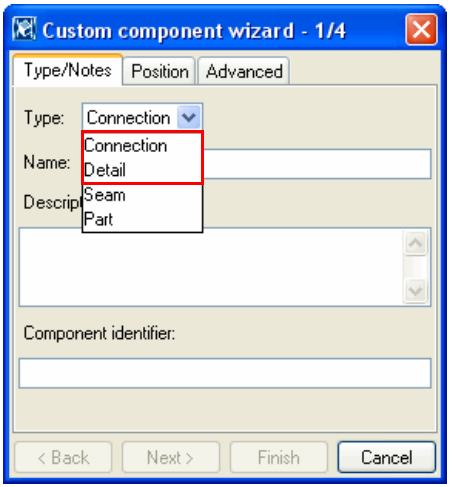
With the provided components, almost all connection types can be created. However, in some cases the available components are not sufficient. This means that the connections have to be created manually.

As this can be quite time-consuming, Tekla Structures provides the possibility to create connections and details, so-called gebruikers componenten.

You can tool up these gebruikers componenten with parameters, which means that a gebruikers component can be used in several situations!

Component types

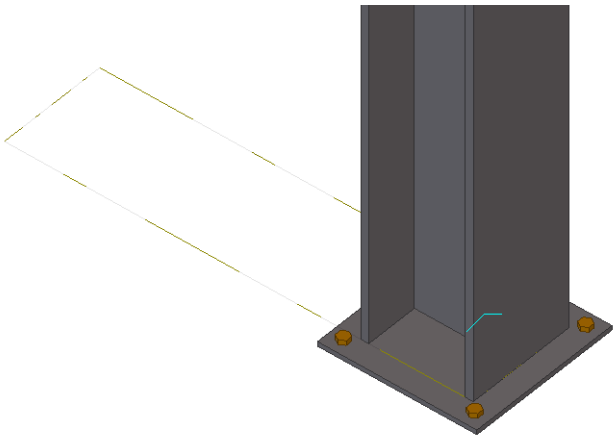
While creating a gebruikers component, you can choose the gebruikers component type:



For proper functioning of the gebruikers componenten, the connection type is vital. Below you see the difference between the following types:

- **Connection**: A **connection** between 2 or more profiles is created.
- **Detail**: For creating details, a **profile** and a **insertion point** have to be selected. Examples are a stub detail, stiffener detail or a base plate detail.

Base plate detail

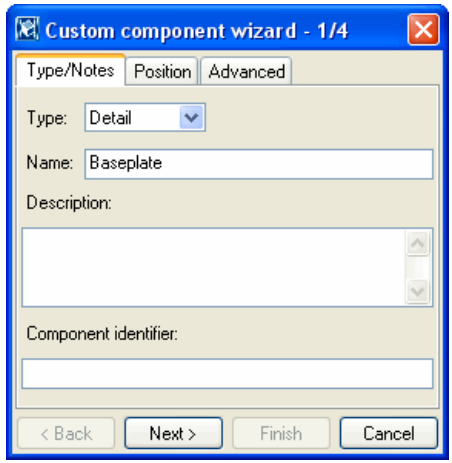


Step plan

1. Create a base plate manually as displayed in the picture above
2. Select all parts that will be included in the gebruikers component (so in this example the base plate, the weld, the fitting line and the bolt group)



3. Go to **Detailing** > **Define gebruikers component...**
4. In the dialog box, set the **Type** to **Detail** and enter the gebruikers component's name:



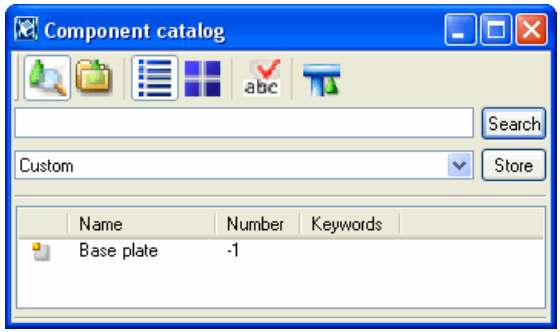
Follow the steps from the wizard, the gebruikers component will be created.

Availability of gebruikers componenten

The gebruikers component Base plate is now available in the current model:



1. Click the icon **Find a component** on the toolbar Components or use the shortcut <**Ctrl + F**> to open the dialog box **Component catalog**
2. Set the listbox to `Custom` to be able to select the gebruikers component



New models

To be able to use the gebruikers component in all **new** models, you need to export the gebruikers component:

1. Select the gebruikers component in the dialog box **Component catalog**, click the right mouse button and select **Export....**
2. Save the gebruikers component by using the name `base plate.uel` in the folder *...\environments\europe\components_sketches\std_sde* (this depends on the module that you use)
3. Restart Tekla Structures and create a new model, the gebruikers component Base plate is now available

Existing models

To be able to use the gebruikers component in all **existing** models:

1. Click the icon **Find a component** to open the dialog box **Component catalog**
2. Click the right mouse button and select **Import....**
3. Select the file `base plate.uel` in the folder

   ...\environments\europe\components_sketches\std_sde
4. Click <**OK**>, the gebruikers component Base plate is now available

All available gebruikers componenten

The folder `..\europe\components_sketches\std_sde` contains all available gebruikers componenten.

1.35 Details, joints and components

Recurrent actions can be speed up by using details, joints and components. Note that details, joints and components are a tool to create the desired connection. A suited component is not available for every situation. In those cases, it may be useful to use details, joints and components, then explode these components and modify the connection objects. Afterwards, it is possible to create a (parametric) custom component of the connection.

With *details, joints and components,* you can easily add parts and connections to the model.

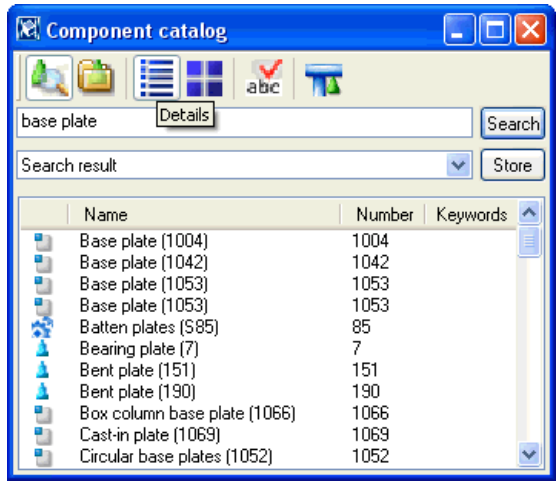
3 types of connections are available:

- **Detail***:* To add parts to a profile on a certain position (e.g. a stub). So, a detail doesn't connect two or more profiles.
- **Joint***:* To connect 2 or more parts (e.g. column - beam connection)
- **Component***:* To create an object (e.g. stairs)



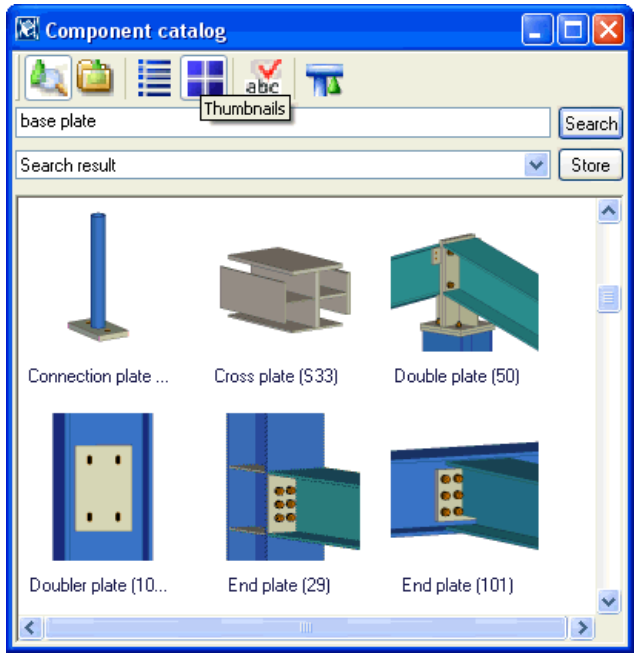
Use the icon **Find a component** on the toolbar **Components** or the shortcut <**Ctrl** + **F**> to open the dialog box *Component catalog* in which you can search for components. You can also create a list of often used components (favorites).





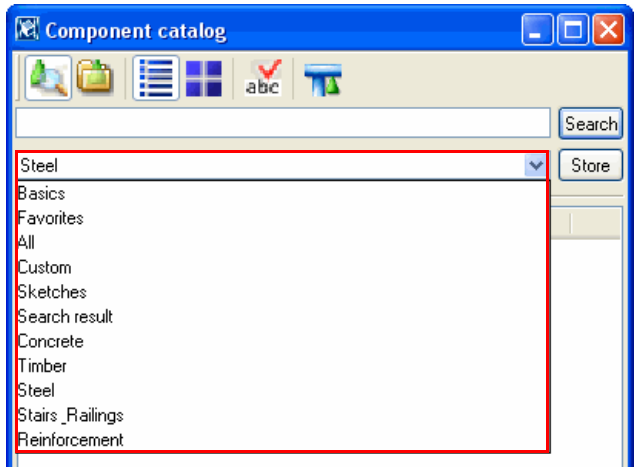
This example shows the results for the search term "**base plate**".

You can set the view to **Details** or to **Thumbnails**



In the listbox you can select several default component groups.

In the list, the accompanying components will appear, i.e. steel.



See also

For more information about the Component catalog, see **Tekla Structures.pdf** in the folder **disk:\TeklaStructures\version\environments\europe\pdf**.

By pressing the *arrow buttons*, you can page through the available component pages. A component is available for most types of connections.

The term “component” is used in this manual, also joints and detail are meant by this.

Components, the main advantages

- The settings of a component can be saved to a file. After that, these settings can be used in all other projects.
- When modifying a profile, all details/ joints attached to that profile are updated automatically to the new situation.
- When multiplying profiles, like, copying or mirroring, all detail and/ or components are also copied. The copied or mirrored details/components are identical as the original ones. So, exactly the same platen, bolts, welds, etc.

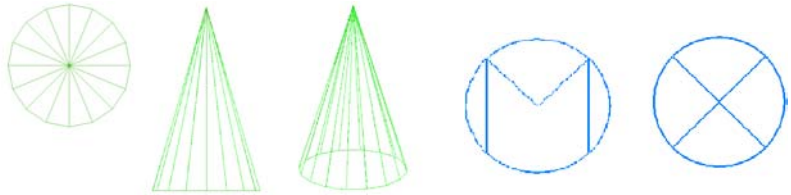
General function of details, joints and components

When applying details and joints, several profiles have to present. In the *Picture*-tab of the component, the main profiles which have to be present are displayed in blue color. The parts which can be created by the component are displayed in yellow.

After selecting the main profile, Tekla Structures shall ask to pick a point (in case of a detail) or to pick another profile (in case of a joint). For example, when applying a base plate component, you only have one profile (the column), so Tekla Structures will ask pick a point for determining the level of the base plate. When applying a connection between e.g. a column and a beam, you obviously have more than one profile (a main part and a secondary part). In case of a complete object, like a stair, no parts are present before applying the component. It is the component that creates all parts. Components are placed between two picked points.

A detail or joint always has to be placed with a specific direction. The possibilities are: +X, -X, +Y, -Y, +Z and -Z. Furthermore, there is an **Auto** option. By setting the component to this option, Tekla Structures automatically searches the most suitable direction for that particular component. The direction of the component is indicated with an arrow. Every component has a preferred direction, in which the component works best. This preferred direction can be found in the *Picture*-tab of the component, or by pressing the *Help*-button in the component dialog box.

For components, a different symbol is used (see picture):



Detail or **Joint** **Component**

Changing conceptual components

When a model is created in the modules Standard Design or Reinforced Concrete Detailing, and you open such a model in the module Steel Detailing, this model may include so-called conceptual components. The draftsman then takes the conceptual components and creates fully-detailed components that include all the information needed for fabrication.

The Tekla Structures configuration you have determines which components you can create:

Type	Symbol	Configuration	Description
Conceptual	 Rectan-gular	Standard Design Reinforced Concrete Detailing	Conceptual component does not create assembly, cast unit, or reinforcing bars. The dialog box is the same as in detailed component, but does not contain fields for part and assembly positions.
Detailed	 Round	Steel Detailing Precast Concrete Detailing Full Detailing	Fully detailed component contains part and assembly numbering fields and creates needed assemblies, cast units and reinforcing bars.

Changing component type

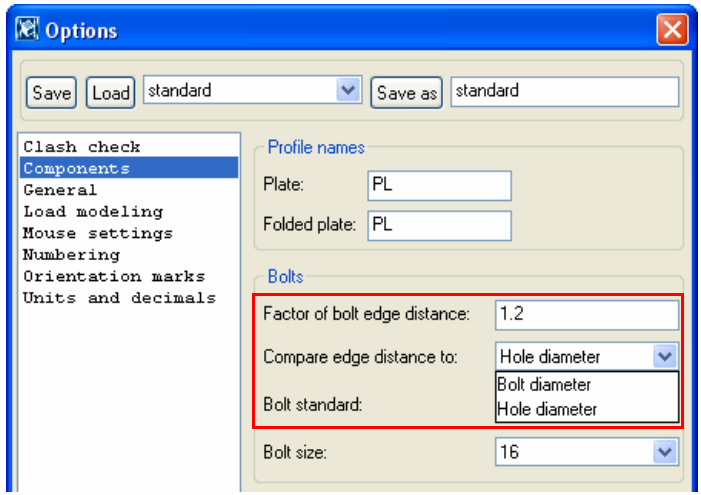
To change a conceptual component to a detailed component:

1. Select the component symbol.
2. Click **Tools** > **Change to detailing component**

Changing part properties, such as changing the size of the main part, will not automatically change the component type. So someone who is using the Steel Detailing configuration can change the model without automatically changing conceptual components back to detailed ones.

draftsmanDetail- and joint symbols can take three colors; green, yellow and red. These colors have the following meaning:

- **Green*:*** The bolt edge distances are sufficient; Tekla Structures checks if the edge distances are sufficient compare to the value entered in the *Options* dialog box **Tools > Options...**



The component symbol is always displayed in blue color.



To check if bolts can be mounted, you can use the Clash Check option for that

- **Yellow*:*** The edge distance(s) are not sufficient
- **Red*:*** Tekla Structures can not calculate a solution for the joint. This can have several causes, for example wrong direction of the joint, wrong work plane, wrong detail or joint used

When applying a component between 2 or more profiles, Tekla Structures asks first to pick the main profile, and after that a secondary profile (or more than 1 secondary profiles) has to be picked. When all necessary profiles are picked, press the middle mouse button. After this the joint will be created.

Example: Base plate detail #1004



Double-click the icon or use shortcut <**Ctrl** + **F**> to search for the component.

Create the component

1. Load setting called **standard**
2. Click **<Apply>** and **<OK>**
3. Pick the column
4. Pick a point. The base plate detail will be placed under the column.

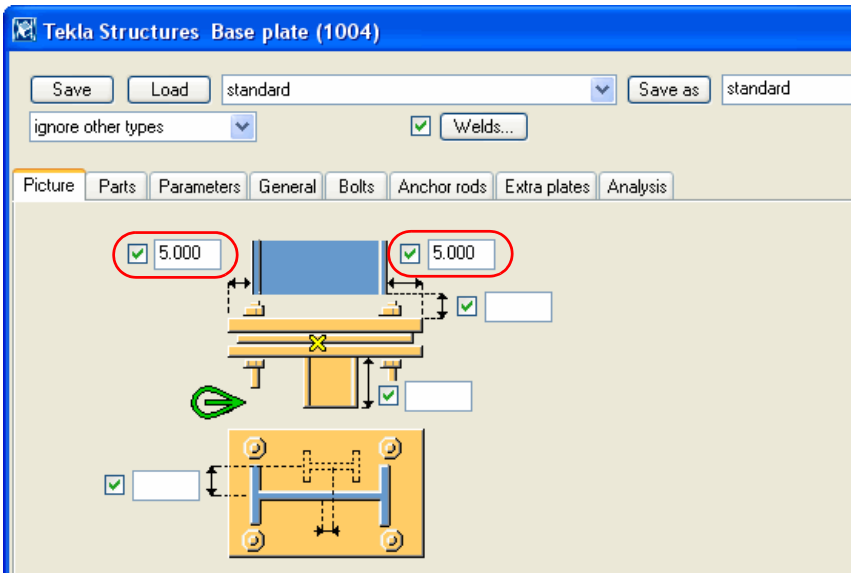
Edit the component

1. Double-click the base plate detail symbol.
2. The dialog box will be opened.

The main settings of this detail are:

Picture tab

The base plate edges from column edges can be set here. Furthermore some input fields are present for a (possible) additional beam.



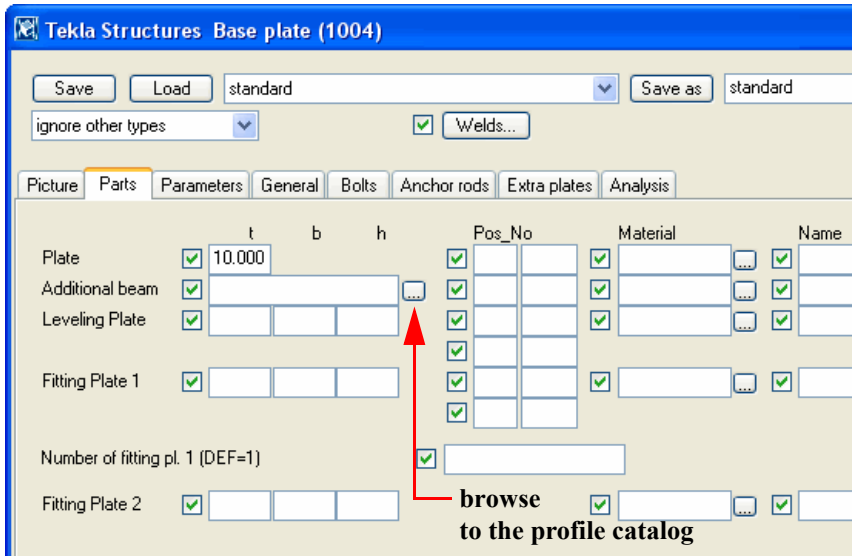


The up direction of a connection or a detail indicates how the connection is rotated around the secondary part, relative to the current work plane. If there are no secondary parts, Tekla Structures rotates the connection around the primary part. The options are: auto, +x, -x, +y, -y, +z, -z.

The **Picture** tab in the dialog box shows the up direction Tekla Structures will use. Tekla Structures also indicates the up direction in the connection symbol.

Parts tab

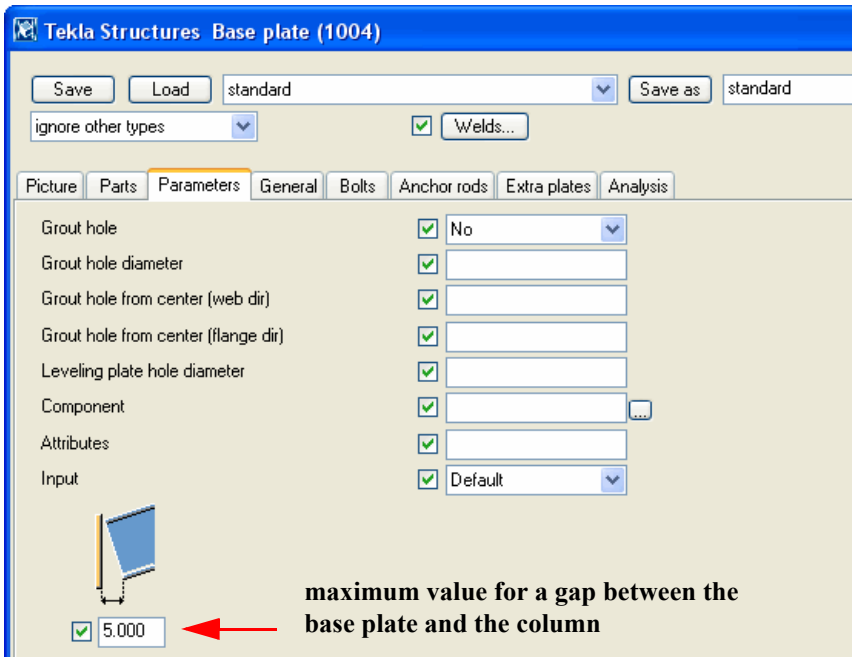
The thickness of the base plate can be set here. It is also possible to set the presence of a additional beam and a spacer plate.



Parameters tab

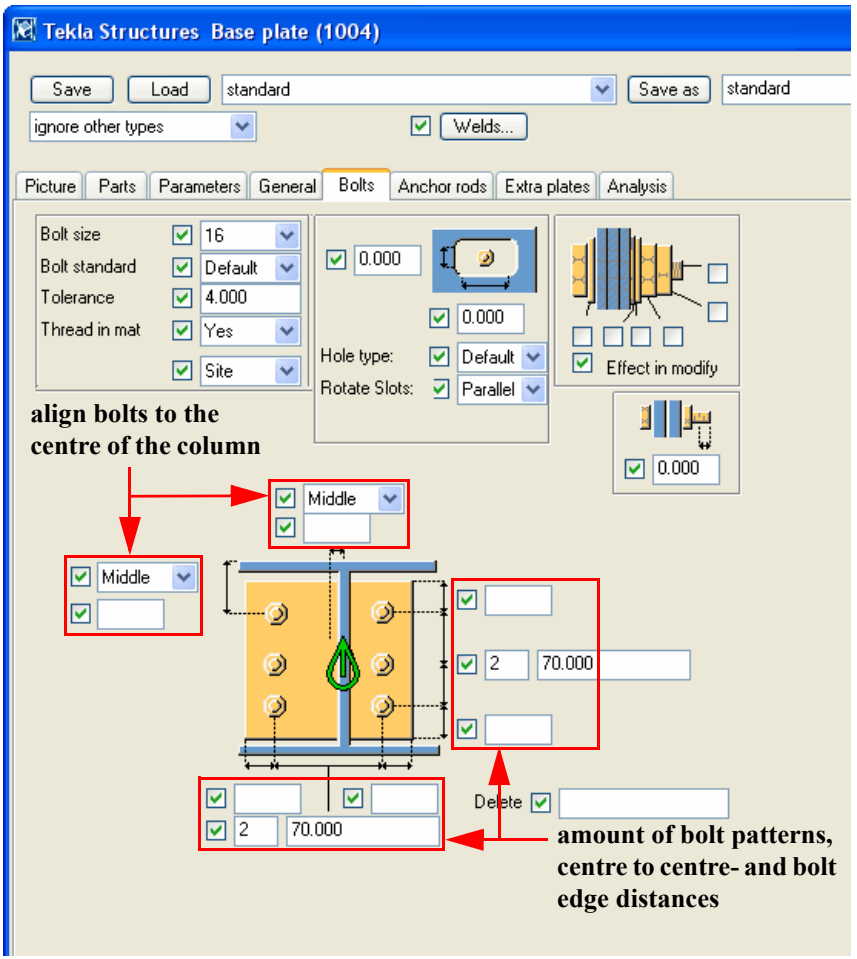
This tab contains some input fields for adding a grout hole in the base plate.

Furthermore it is possible to enter a maximum value for a gap between the base plate and the column (to achieve that the column can be shortened with a square angle).



Bolts tab

In this tab all parameters for the bolts-/ anchor-pattern can be set.



You can enter the **Bolt size** and the **Bolt standard**. If <*Default*> is selected, Tekla Structures will automatically take the standard settings as entered in **Setup > Options**.

At **Tolerance**, enter the clearance for the bolt compared to the hole. In this case the size of the hole will be 16 mm + 4 mm = 20 mm.

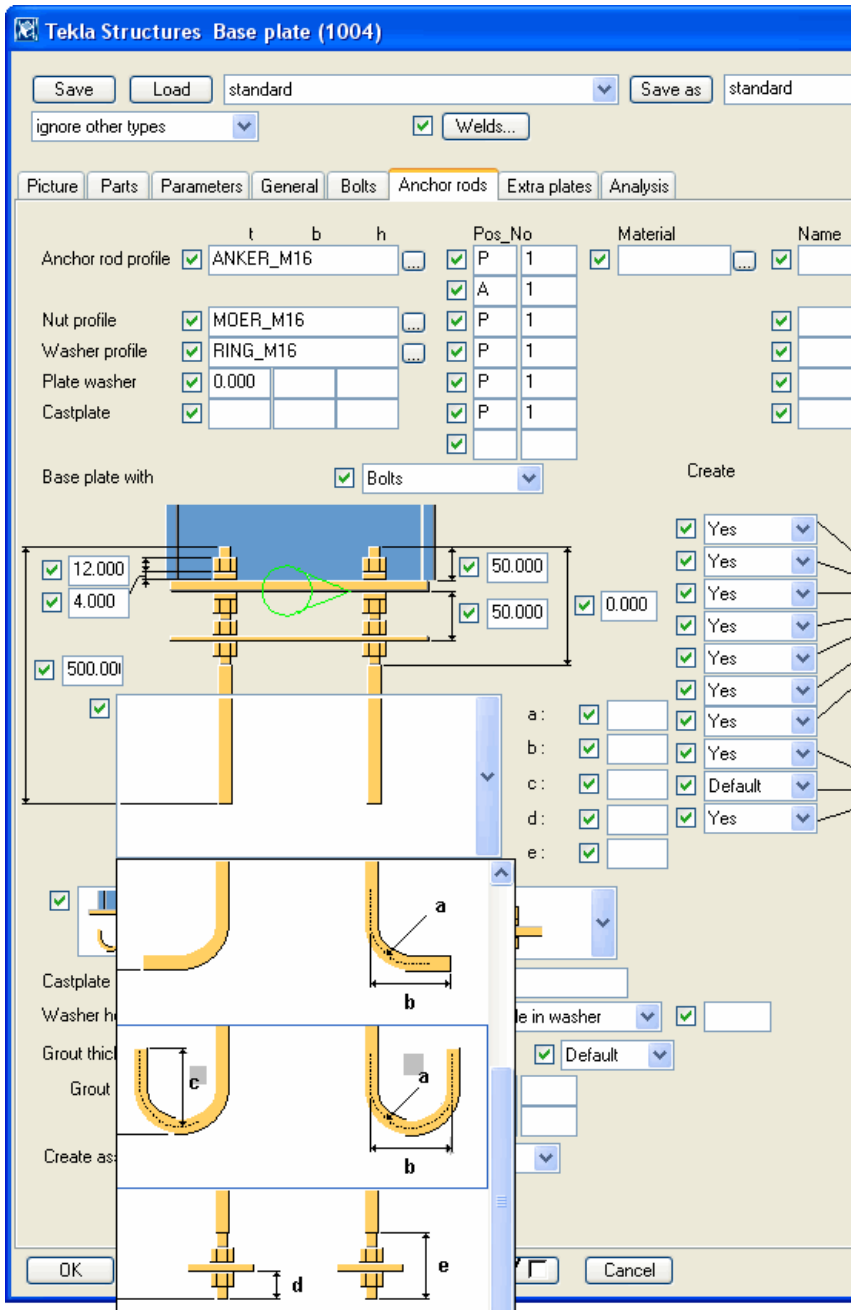
The alignment of the bolt patterns (for both vertical and horizontal) is set to *Middle*. The other options are for vertical *Up* and *Down* and for horizontal *Left* and *Right*. These settings indicate the alignment of the bolts. In this case the bolts are aligned from the center of the column.

Below and right from the picture, the number of bolt rows can be entered with the corresponding centre distances and edge distances. In the example, for both vertical and horizontal 2 bolt rows are placed with a center distance of 70 mm in both directions.

Also the base plate is welded to the column and the column will be fitted to the base plate. Click the **Welds...** button to see how the base plate is welded to the column. The column will be fitted to the topside of the base plate.

Anchor tab

In here *rods* can be set whether bolts or anchors have to be placed.



If *Anchors* is selected, the type of anchor, nut and washers can be entered. There is also the possibility of using a cast plate. Also the shape of the anchors can be selected from a listbox.

Exercise

Remove the existing baseplates from the model. Be sure that the associated fittings are removed too. Now, rebuild the baseplates with use of component 1004.

The base plate will be placed under the column including all welds, bolts and holes. The edge-distance of the base plate for welding can be set in two ways: in relation to the profile (e.g. 5mm inwards of the edge of the column), or in relation to the bolts (e.g. 30mm from center of hole).



When connecting tube profiles, it is recommended not to enter plate edges from tube edges (for welding) in the Picture-tab, instead set the plate edges with bolt edge distances. This to get rounded-off strip metal dimensions.

Storing component-settings

A setting is created for a certain column. If you want to save this setting, do the following:

1. Double-click the component symbol; the dialog box will be opened
2. In the input field right from the <**Save as>** button, type an easy recognizable name, e.g. *HEA140* (no space characters!)
3. Click **<Save as>** once

A file is made from this setting and it is stored in the folder of your current Tekla Structures model. At this time, the setting is only useful in the current model.

If you want to use this setting in all models, i.e. both new and old models, than this file needs to be copied to the folder **ts**:

disk:\TeklaStructures\version\environments\europe\ts

In Tekla Structures you can open the model folder very fast without opening the Explorer first and to search for the model folder. This counts for the modelling- and the drawing editor:

- In the pull-down menu, go to **Tools** > **Open model folder** or
- Press shortcut “4” on your keyboard (note: not on the numeric part)

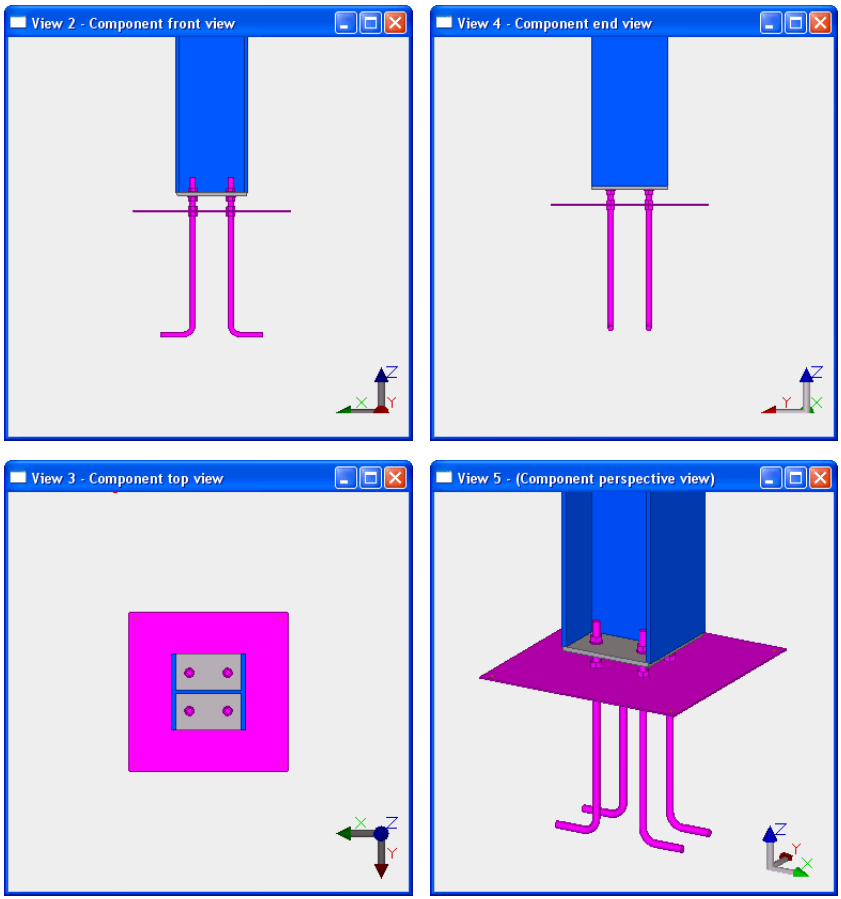
The file can be recognized by a number. For example, the base plate detail has no.1004. The file’ extension is *j1004*.

Create connection basis views

To check a component and its dimensions, it can be very handy to create so-called component basic views in Tekla Structures:

Select a component, click right mouse button and go to **Create view** > **Component basic views**.

Tekla Structures will create 4 views to display the connection in different views.



1.36 Concrete components

Concrete parts often need further editing. This can be done using commands like fit, line cut, split, part cut and polygon cut, part add, etc. but also using components.

In the following paragraphs some concrete components are described. The most common concrete components are located on component pages CS3 and CS4; but also components from other pages can be used.

Not every component detail will be described; possibilities, options and picking order are clearly indicated. Several concrete component examples are located in the back of this training, as well as some steel components. Some often used components are described in the following paragraphs.

Battering (13)

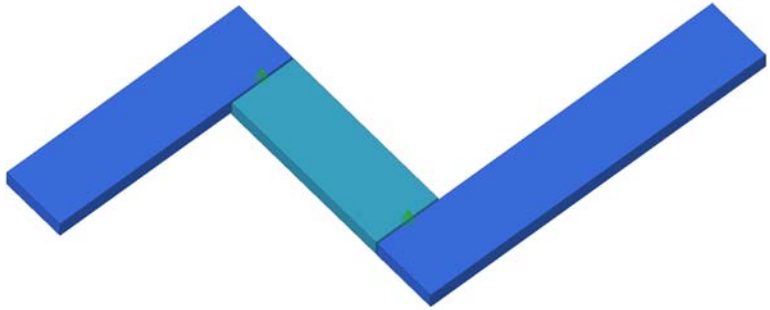
For panels in a row one can create a joint between them by filling in the offset values (tab position) Of course you can model the panels with the real length (no offset). Advantage of doing this is that you can't make mistakes when snapping on the panel.

With component **Dividing part (1047)** dividing part one can divide panels in a row with a joint-thickness.

Joints between panels which are perpendicular connected can be made with component **Battering (13)**. This component can create a continuous joint.



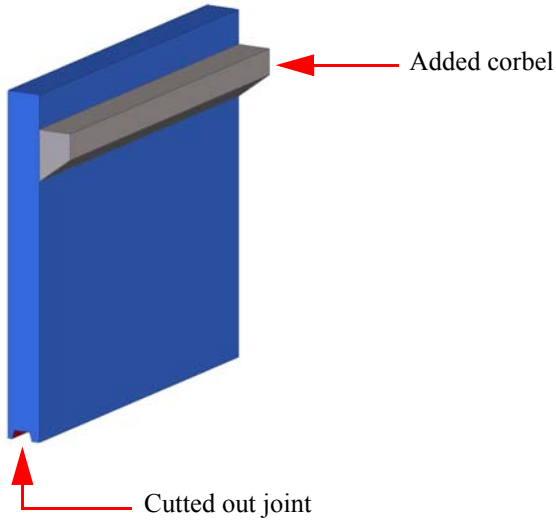
This component can also be used for connecting e.g. foundation beams or foundation strips to each other. In such situations the joint width will be ‘0’.



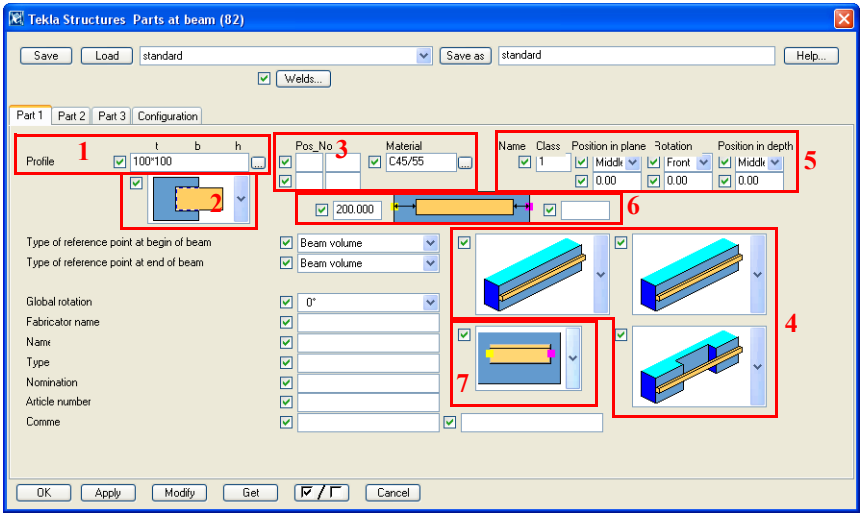
Because the end planes ‘touch’ each other, a continuous slab will be displayed on the drawing.

Parts at beam (82)

With component **Parts at beam (82)**, parts can be added or cut out. With this you can for example cut out a joint or add a corbel to a panel.



Select the component, load a setting (or define your own) and select the part you want to apply to component to.



Description of buttons and fields:

1 **Profile**

Define the profile to be added or to cut out

2 **Method of adding**

For concrete to concrete: Add (cyan)

3 **Material and Numbering settings**

Select a concrete material in case of a concrete part (contact planes on drawing!)

4 **Orientation to the main part**

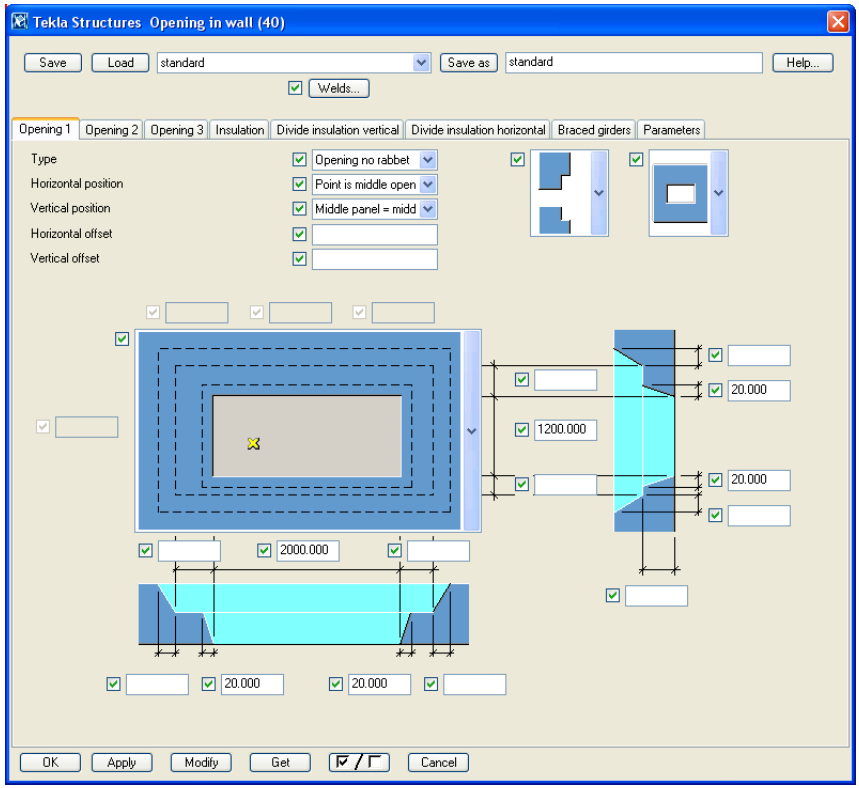
5 **Orientation of the added part/ cut out**

6 **Added part's offset from main part**

7 **Swapping the start and endpoint of the added part**

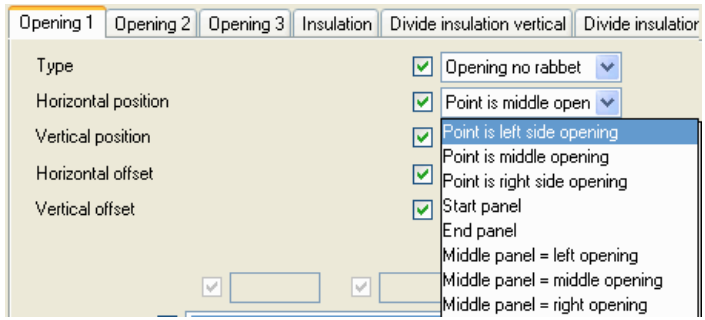
It is possible to add multiple parts or cut outs. To do this, use the tabs **Part 2** and **Part 3**.

Opening in wall (40)

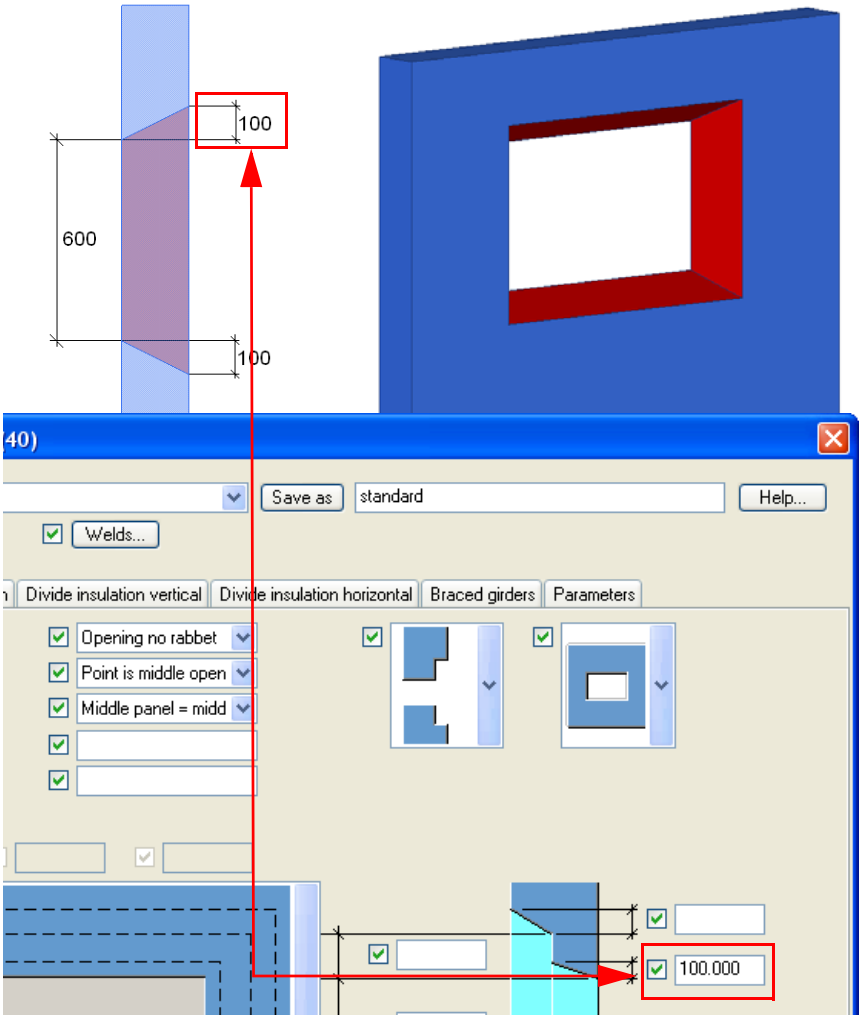


The component **Opening in wall (40)** can create an opening in a concrete wall. The size and position of the opening can be adjusted in many ways.

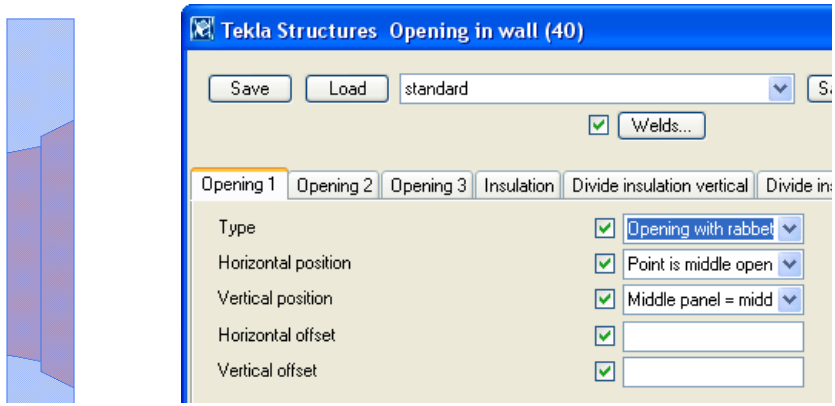
The opening's position cab be determined by a point or by entering the opening's coordinates. See tab **Opening 1**.



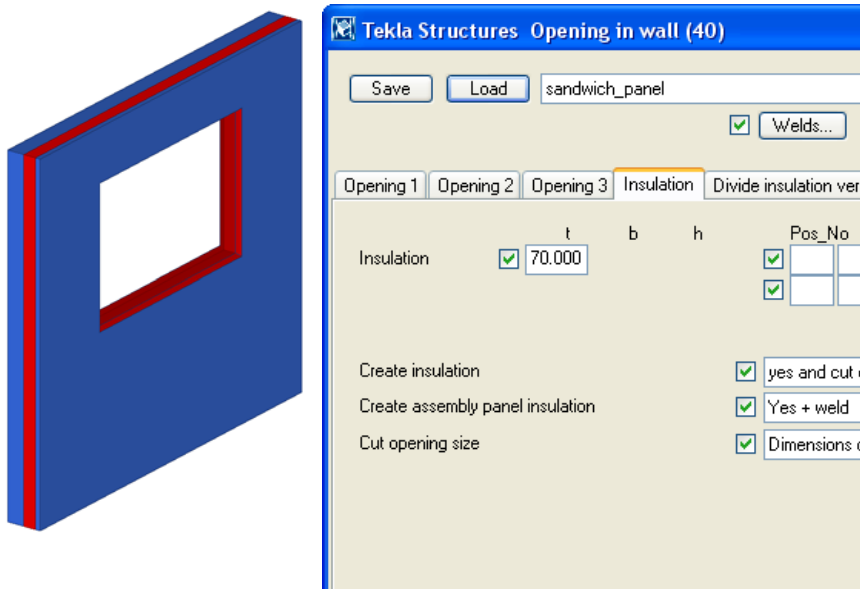
Clearances can be added by entering values in tab **Opening 1**.


100
600
100
40)
Save as
standard
Help...
Welds...
Divide insulation vertical
Divide insulation horizontal
Braced girders
Parameters
Opening no rabbet
Point is middle open
Middle panel = midd
100.000

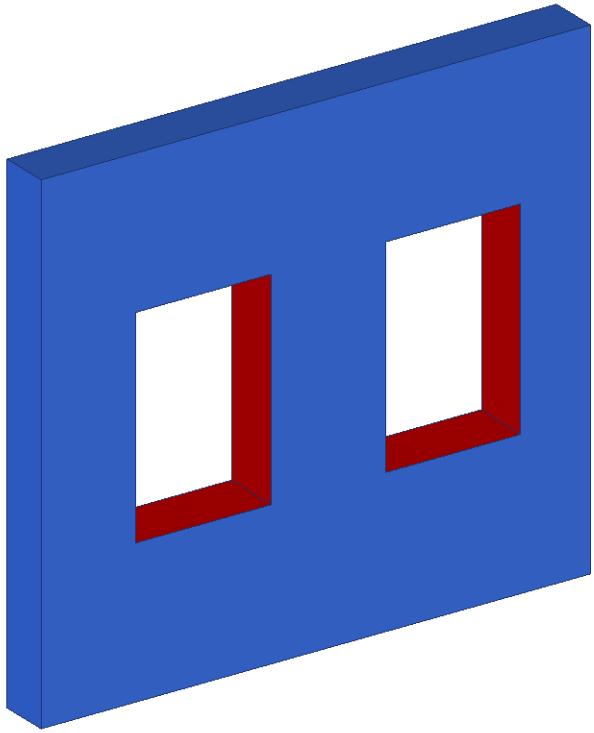
Rabbets can be created by choosing the **Opening with rabbet**-option in the Type picklist and entering the rabbet depth, possibly in combination with a clearance.



Insulation can be added to the wall using the **Insulation** tab. A thickness and offset from the outer contour can be set to create sandwich panels.

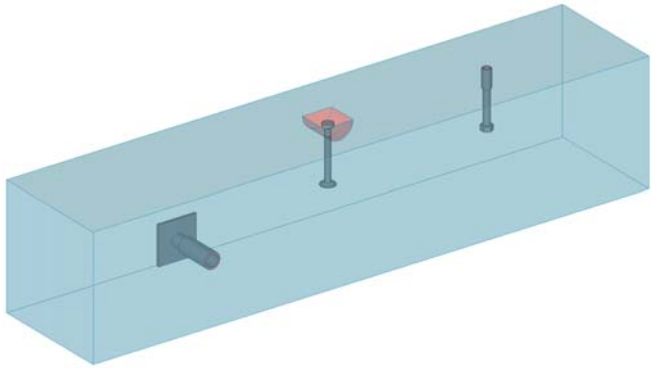


It is possible to create multiple openings at once. To do this, enter the desired values in tabs **Opening 1**, **Opening 2** and **Opening 3.**

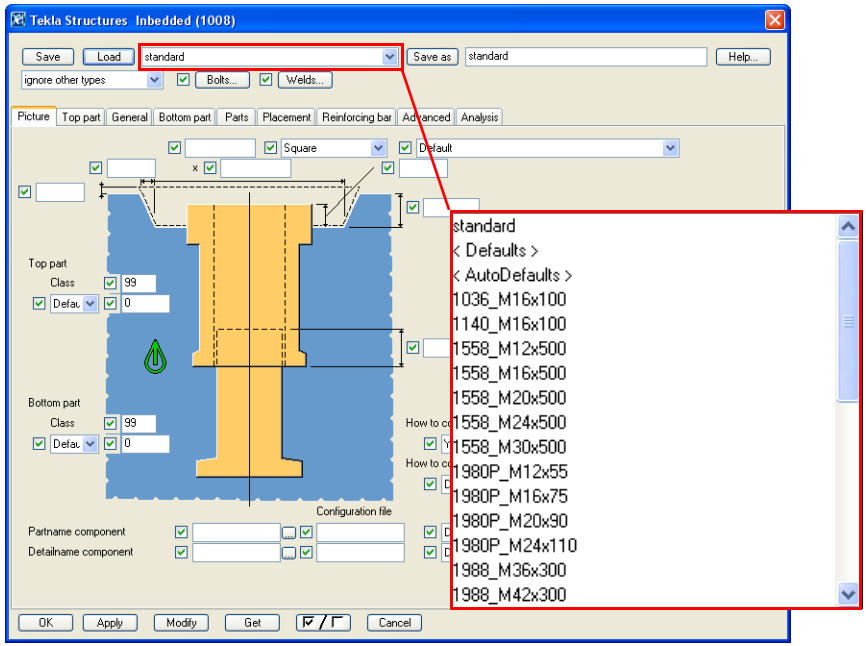


Imbedded (1008)

With component **Imbedded (1008)** on CS4 page you can add imbeds to concrete parts.



Double-click the component to open the dialogs. Several pre-defined settings can be loaded.

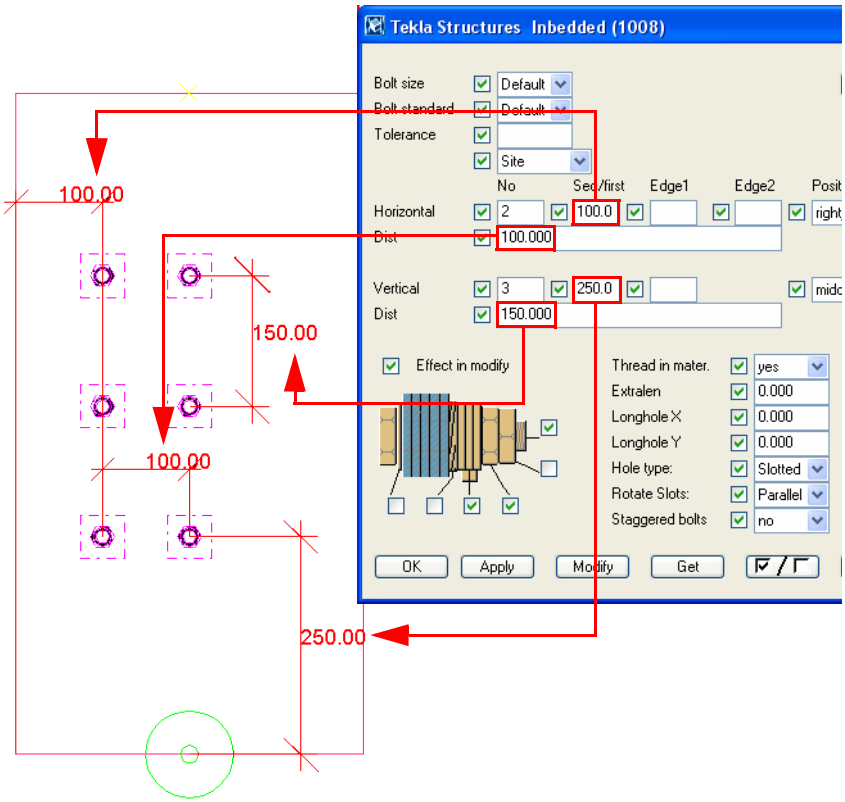


Load a setting - or change the current settings in the dialog - and click <**OK**>.

Click the part to which the imbeds will be added and subsequently a position.

For each operation/group multiple imbeds (from de same type) can be placed.

At button **Bolts**, the number of bolts and bolt distances (x- and y-direction) can be set. Also the position of the imbedded-group (left/right) and the distance to the edge can be set.

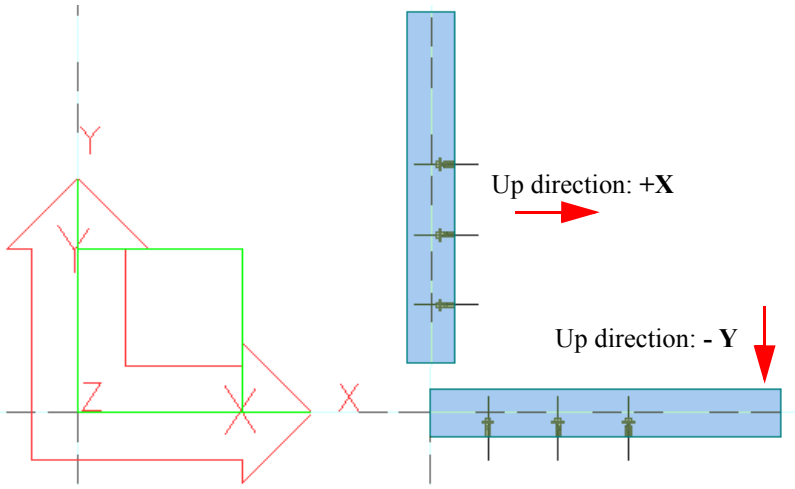


The distance to the edge in the x-direction of the component (dimension ‘250’ in picture above) can also be determined by holding down the <**Ctrl**> key while selecting the element and then entering ‘250’.

Using the *Up. direction* on the **General** tab, you can adjust the positioning. These directions are according to the position of the work plane.

Basically the direction is set to **Auto** which comes down to the **+Z** direction. The anchors are being placed on the upper surface of the concrete element. **-Z** counts for the bottom side.

It becomes more difficult when anchors need to be positioned on the side of the panel because then the direction of the component depends on the position (rotation) of the panel compared to the work plane. See next picture for more clearness:

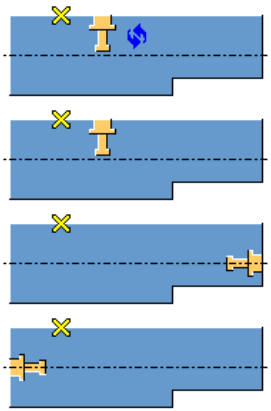


The plane where the anchor will be placed (the opening) depends on the direction of the work plane. In the picture above, two identical panels are placed, but the component's setting is still different.

Of course it is possible to trace the orientation of a part in comparison with the work plane.

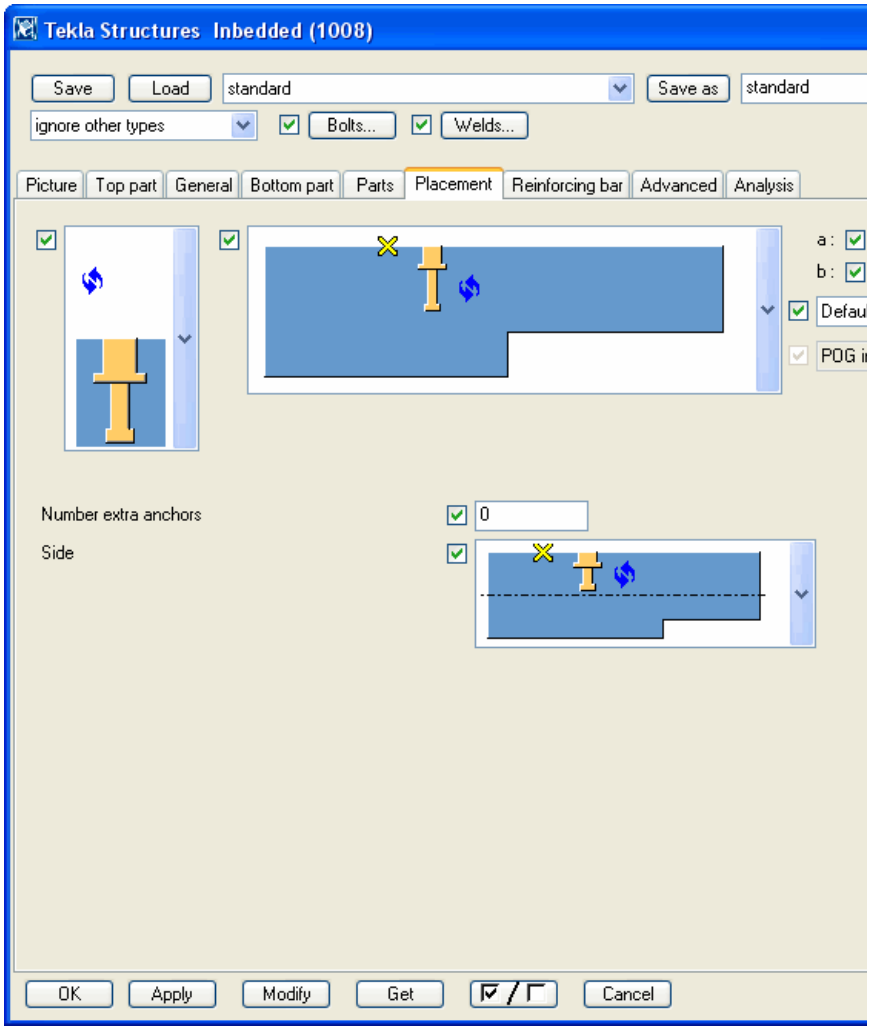
It might be easier to place the work plane at the local X-axis of the part each time the component will be used. That way the settings and orientation in the component are always the same.

For placing the imbeds on front-, back-, top- and bottom side, you need to adjust the component's direction in the **General** tab. Should the anchors be placed on the parts heads, you can do this in the Side-picklist in the **Placement** tab.



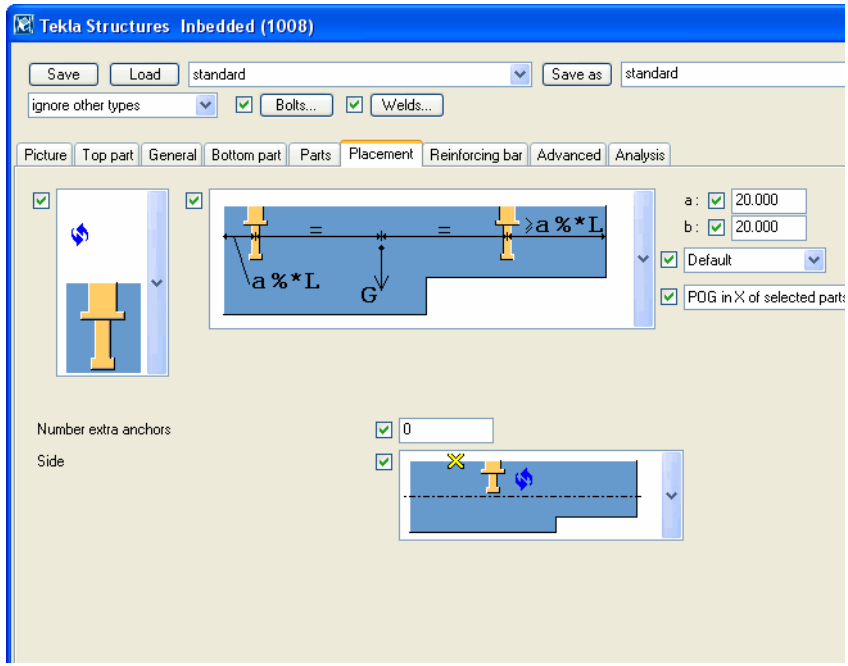
Multiple imbeds in one go

Multiple groups of imbeds can be created in one go. Just enter the number of groups in tab **Placement**, see example in the picture below:



Lifting anchors

Component **Imbeds (1008)** can create lifting anchors. In tab **Placement**, an option can be selected which makes Tekla Structures automatically finds the centre of gravity of the concrete element and subsequently positions the lifting anchors in the correct position. In the picture below the setting is selected where the anchors are placed on 20% of the C.O.G. length.



You can choose for one, two or more anchors. If 2 anchors are selected, the gravity of the parts will be calculated and the anchors will be positioned at a given percentage of this. When selecting multiple anchors (>2), the intermediate distances will be equally divided.

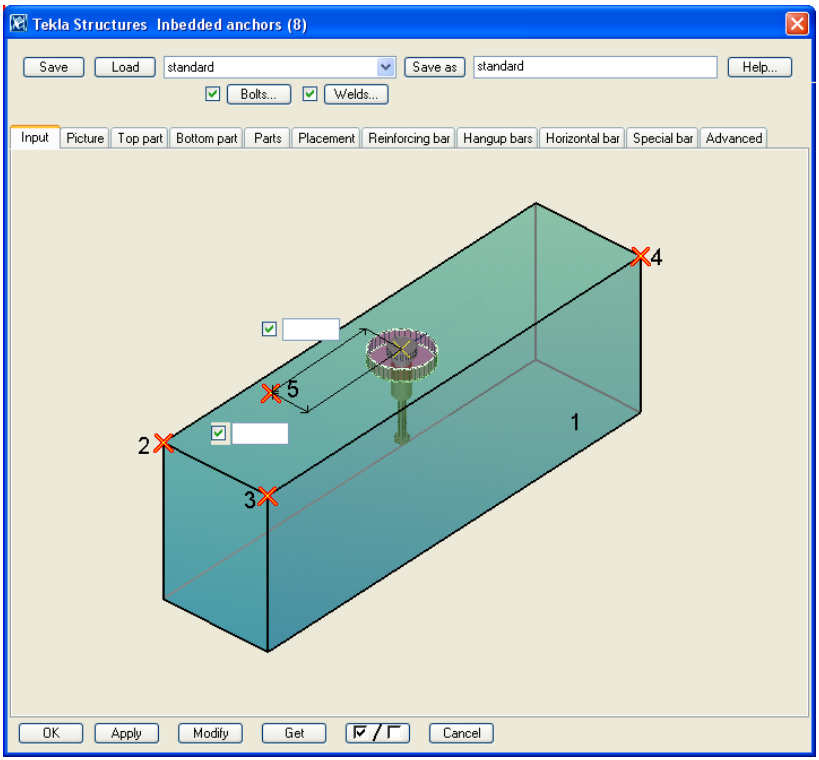
Adding imbeds

If an anchor or imbed has to be modelled which is not available in the standard settings, then this anchor/ imbed can be added. In tab **Parts**, several cross sections can be defined. In tab **Top part**, the length of the parts can be set.

You can also define the settings, like the way the anchor has to be added (welds, adding shapes, etc.) and various comments, article number, fabricator and related information which possibly can be displayed on drawings and in reports.

Imbeds (8)

Component **Imbeds (8)** is similar to component **Imbeds (1008)**, the only difference is the way of positioning the imbeds. A tab *Input* is available to define the imbed location by picking a number of points. The *General* tab -also necessary for placing the embeds in component **Imbeds (1008)** - is not present in this component.



Select the component and the part. Next, pick three points, these 3 points will form a plane where the imbed will be positioned. Finally, click the part's position. Possible, you can set an offset (two directions) from the last picked point.

As the plane is determined by 3 picked points, this makes the component suitable for placing imbeds in skew planes or in contour plate profiles.

Exercise Now start with exercise D

Exercise Now start with exercise E (check the examples in the back first).

Mirroring elements containing components

If parts containing (many) components are mirrored, the result may be disappointing. The part itself will be mirrored correctly, as well as the components' positions, but the components are then being recalculated, which may cause components which are not good mirror images. In such cases it may be better to explode the components prior to mirroring.

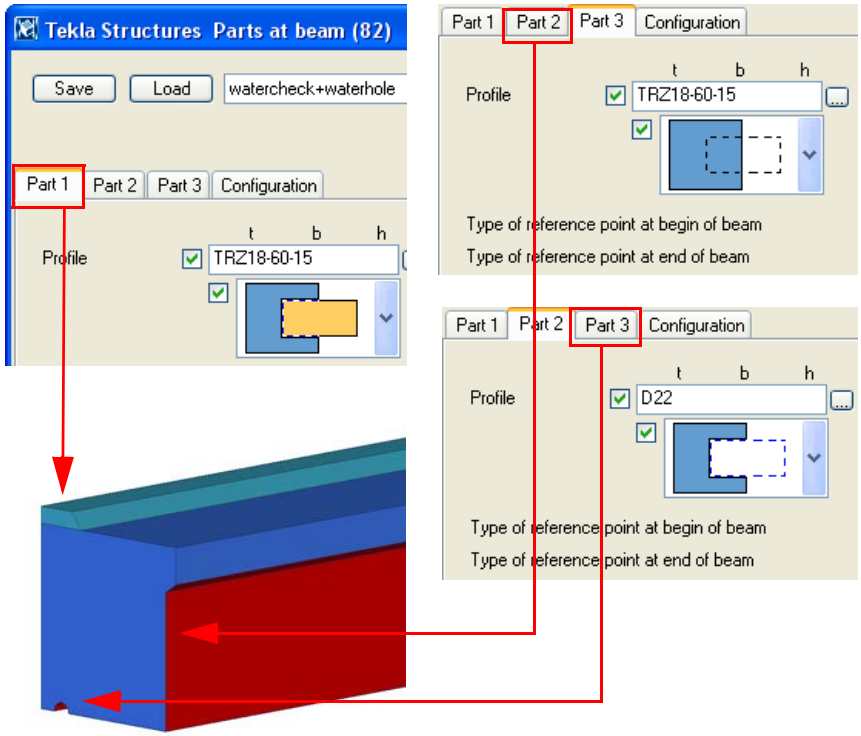
If you don't want to explode components, you always have to check the mirrored components and adjust them if necessary.

Modeling stairs, balconies and platforms

Modeling more complex prefab cast units, such as stairs, balconies, platforms and gallery plates needs to be done in "layers". For example, a gallery plate consists of a normal plate or beam, and successively some parts are cutted out (gutter, throatings) or added (stand ups).

By using materials from the concrete-group, the seams in the model will be invisible on drawings.

For better working, it is better to start with a thin base and building the element, instead of starting with a thick base and adding multiple cuts. By using the tabs in component **Parts at beam (82)**, often-used sections can be saved for later use.





For connecting the edges of balconies/platforms/gallery plates you can use component **Timber part cut (44)**. First, select the main part and next the part you want to cut. This component remains active; if the main parts' shape changes, the secondary part' cut will be regenerated automatically.

1.37 Windbracings

Modelling windbracings is a bit more difficult than modelling columns or beams. Especially when windbracings have to be modelled in a skew roof plane. First we are going to model a bracing in a front plane, after that in a roof plane.

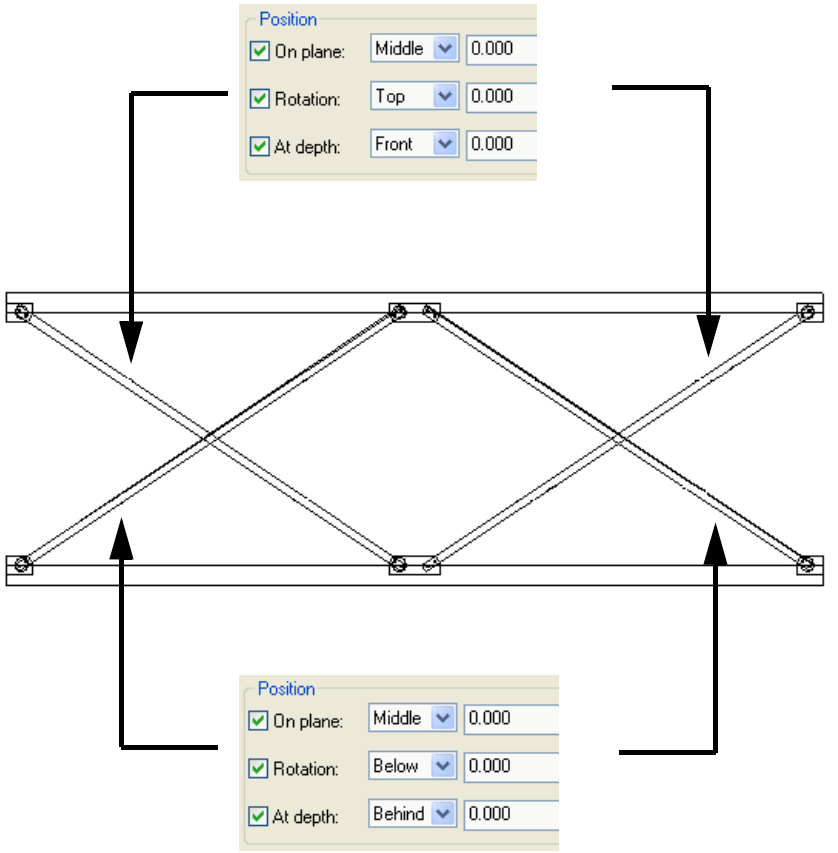
Windbracings in a front plane

- Double-click the **beam** icon and load the setting “bracing”.
- By default, by “Profile” an angle steel profile is loaded. In case of windbracings in front planes, often flat bracings are used, therefore change the profile to e.g. FL10*60.
- At “Position in plane”, enter the values: “Right” and “5”. The windbracing will be positioned on the right side with an extra distance of 5 mm.
- Click <**Apply**> and <**OK**>
- Using the beam command, place a windbracing profile from the left bottom side towards the right upper side.
- Position the windbracings in the centre of the columns
- Select a macro (e.g. macro #11, page 4) and connect the braces to the columns.

Windbracings in a straight roof plane

In case the bracings have to be positioned in a straight roof plane, in general the same order can be kept as described in the above paragraph “Windbracings in a front plane”. Only the used profile type is usually a different; mostly angle steel profiles are used because of their better resistance against deflection. To let the bracings cross each other in a proper way, the values for ‘Position in plane’ have to be increased. For example, for a profile L60/6, the displacement has to be set to “30”, or the “position in depth” has to bet set to “front” or “behind”. Usually, in case of crossing angle steel windbracings, the bracings are positioned with the closed side towards each other. That means that one bracing has to be rotated 180 degrees, by changing the rotation from “below” to “above”.

If two windbracings are connected to one gusset plate, the thickness of the gusset plate has to be taken into account when setting the clearance for the bracings.



Example where the bracings just touch each other

Windbracings in a skew roof plane



To insert windbracings in a skew roof plane, it is recommended to move the work plane in a way that the work plane is parallel to the roof plane. This can be done with the icon opposite (*Set work plane to part top*).

By pressing the icon and than picking a roof beam, the work plane will be set equal to the slope of the roof beams.

If the work plane is equal to the roof beams, a view can be made which is equal to the skew roof plane.

There are two ways to do this:

Option 1

- Set the work plane equal to the roof beam
- Go to **View > Create view > To work plane**

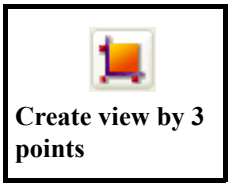
A new view is created in which you look perpendicular to the roof plane.

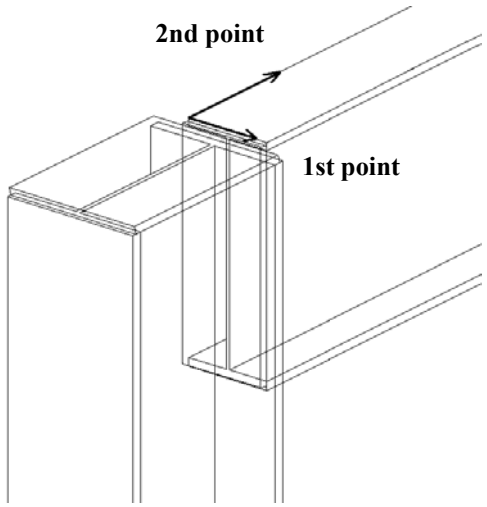
Remind that the view that was just created has the same properties as the current view. The name of the view is placed between parenthesis, that means it is a temporary view which will not be stored when closing the view. If you want the view to be stored, you will have to change the name of the view into a unique name:

- Double-click the view
- Change the name of the view
- Click <**Modify**> and <**OK>**

Option 2

- Double-click command "Create view by three points" to create a new view
- Select the setting 'foundation plan' (concerning the display depth of +/- 500mm) and change the name of the view in e.g. *roof plane*
- Pick 2 points on the roof beam to define the plane of the new view





If a view is created according one of the above methods, the following actions can be done:

- If necessary, insert reference points on the room-beam for positioning the windbracings
- Load setting 'bracing' and position the windbracings
- Change the position of the bracings in that way they do not intersect each other.

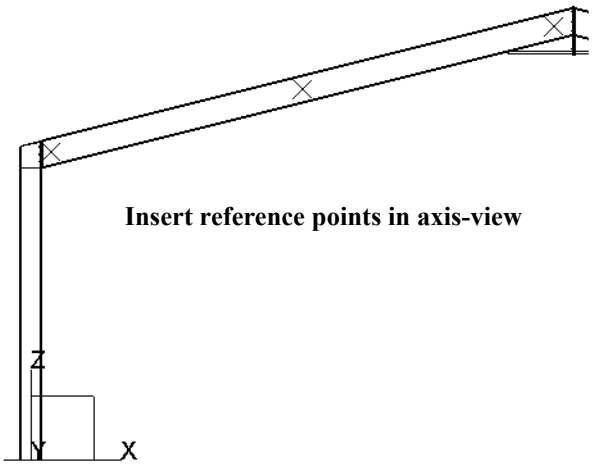
See also

For more information, see **Windbracings in a straight roof plane** (p. 152).

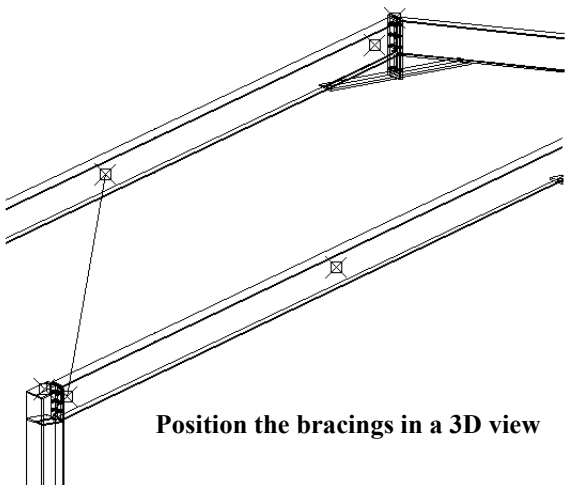
- Connect the bracings to the roof-beams with e.g. Gusset macro #11 (page 4)

An other possibility to model the windbracings without creating roof plane view(s) is the following method:

- In an Axis-view, insert reference points on the roof-beam
- Copy the reference points to the next grid line
- In a 3d view, switch on the ‘points’-display and set the work plane equal to the roof plane
- Model the windbracings in the 3D view by snapping to the reference points



Insert reference points in axis-view

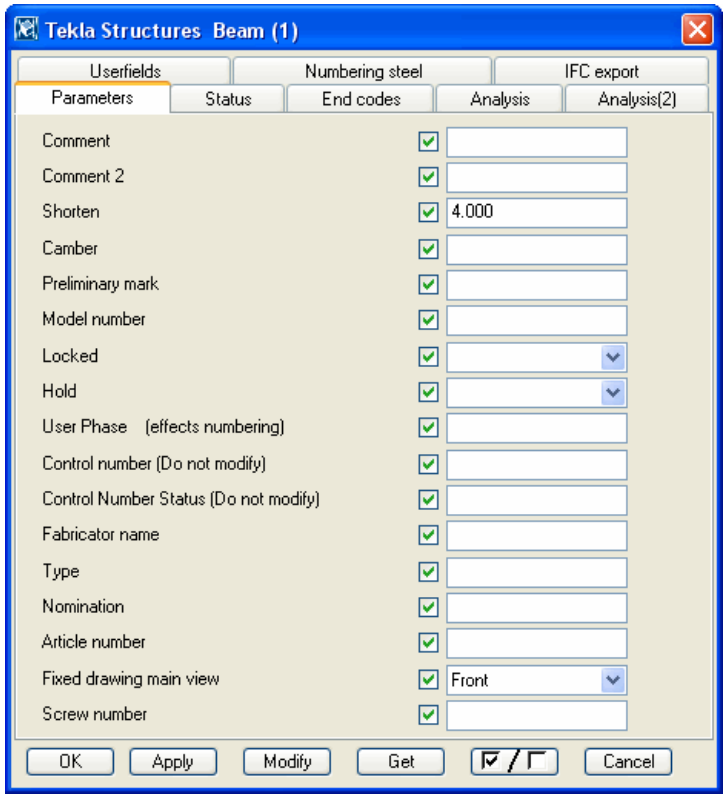


Position the bracings in a 3D view

The actions needed to avoid intersecting bracings are the same as described the previous paragraph.

Shortening windbracings

Shortening of windbracings works by setting a value for input field ‘Shorten’. Click on the **User Defined attributes** button to open the dialog box, see picture below:





After entering a value for Shortening, press <**Modify**> and <**Cancel**> or <**Modify**> and <**Close Window**>.

In case you press <**Apply**> and <**OK**>, it is possible that this shortening value is given to new placed beams, which can cause beams turning out too short.

Apply shortening **only** after finishing the model. This way you are sure no other beams will get a shortening value!

Windbracing check

There are two options to check if shortening of windbracings has been carried out well:

Reports

You can use the reports **UDA** (User Defined Attributes) and **id_part_list**. On these reports, in column **Shorten** is displayed which profiles are shortened.

```
                                                    Date: 05.05.2004
                                                    Time: 14:54:25
-------------------------------------------------------------------------
  Pos       Profile       Number Comment  Shorten Cambering Prel. mark
-------------------------------------------------------------------------
  P3        L80/8            1               4.0
  Prl       HEA180           1
-------------------------------------------------------------------------
                  Total     3   parts
-------------------------------------------------------------------------
```

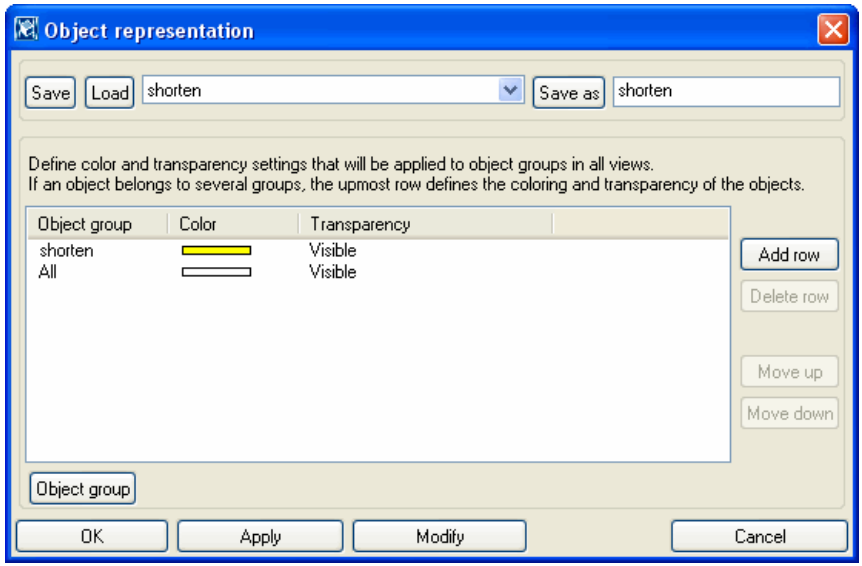


Note when entering values in the **Value** field (for example **Shorten**):

There is a difference in entering value “**0**” or entering **nothing**!

View representation

You can use the view representation to check the shortening of profiles. Go to **Setup** > **Object representation..**. The following dialog box appears:



Select the setting **Shorten** in the listbox. Click <**Load**> and <**Modify**>.

All views will display the profiles which have a value greater than "0" for shortening, in Yellow.

Shortening: how does it work?

Example

A windbracing has a length of 6000 mm and is shortened 4mm. The length of the bracing should become 5996 mm. Both ends of the bracing have holes, bolt distance 60mm, edge distance 40 mm. Without shortening, the distances would be: 40-60-5800-60-40.

What does Tekla Structures do? First, the total length after shortening is divided by the initial length of the bracing. In this example, that means 5996 / 6000 = 0.9993.

Next, each individual distance is multiplied with this factor.

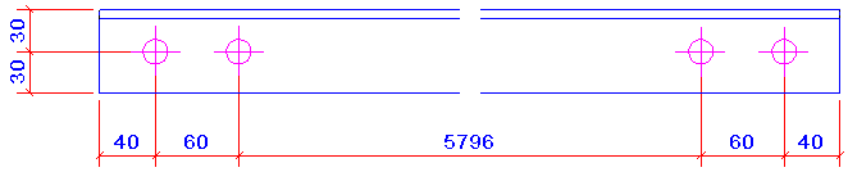
0.9993*40 = 39.973 mm

0.9993*60 = 59.960 mm

0.9993*5800 = 5796.133 mm

By default, dimensions are rounded off on millimeters. So the dimensioning will be: 40-60-5796-60-40. The shortening value of 4 mm will be applied to the largest distance, i.e. the distance between the two bolt groups.

In this case, no problem in dimensioning will arise. But in case the bracing is shorter, or the *Shortening*-value is bigger, (round off) problems can arise.



Example

A windbracing, length 2000mm, shortening value 20 mm.

What happens in Tekla Structures? First, the total length after shortening is divided by the initial length of the bracing: 1980/2000 = 0.99.

Next, each individual distance is multiplied with this factor.

0.99*40 = 39.6 mm

0.99*60 = 59.4 mm

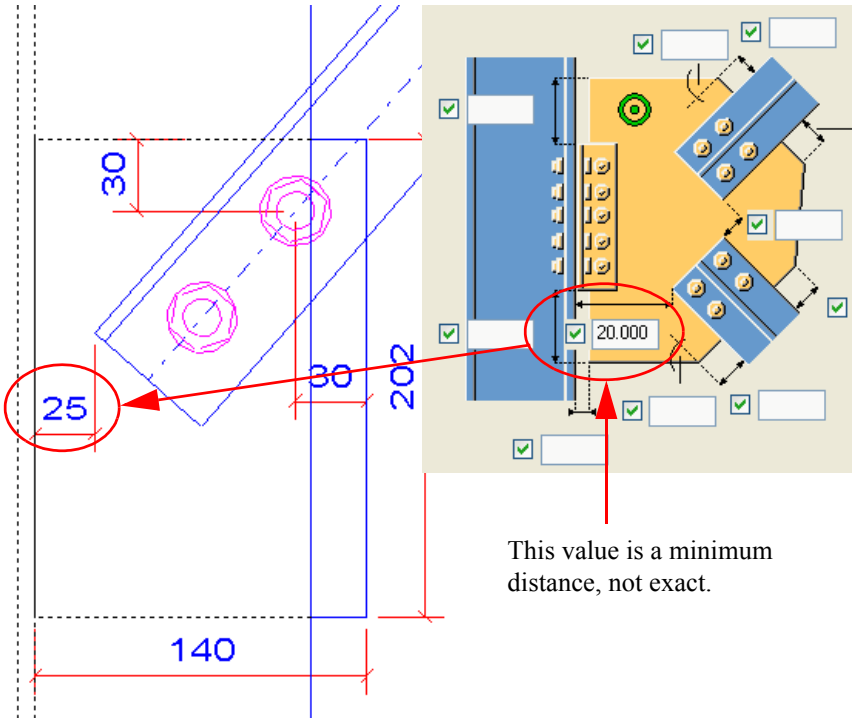
0.99*1800 = 1782 mm

By default, the dimensioning in Tekla Structures is truncated to integers. So, the dimensioning values will be 40-59-1782-59-40. The shortening of 20 mm is not only applied to the biggest distance, but also to the bolt distances!.



Using Bolted gusset component 11

Gusset

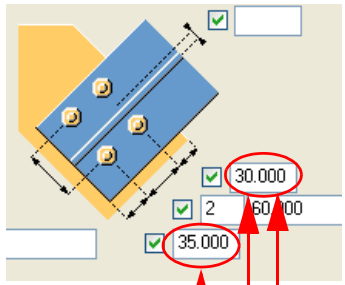


This value is a minimum distance, not exact.

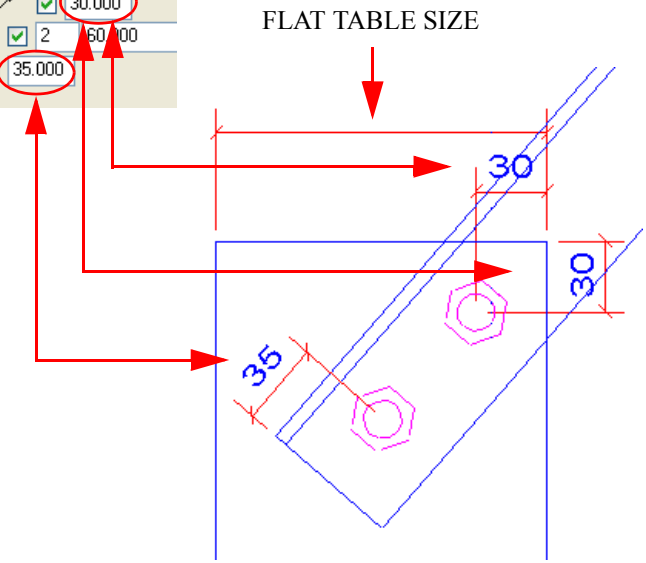


In the tab Gusset, do not enter a value for **b** and **h**. According to the flat table, a size is selected.

After that you can eventually change the height.



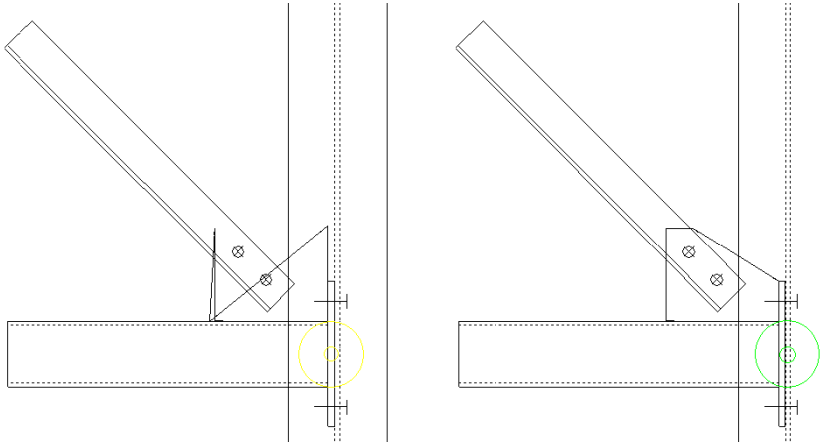
Enter a value in tab **Bracebolts 1, 2** and/or **3**.

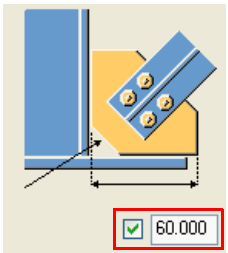


Tab Gusset



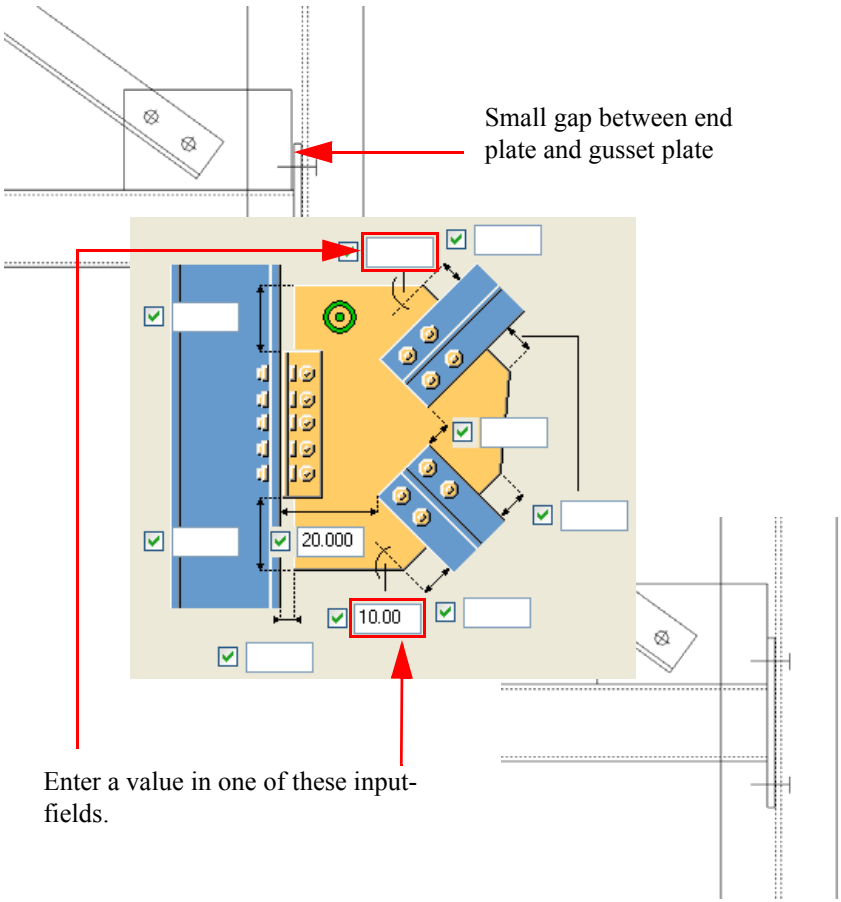
Connecting a bracing to a compression bar using a gusset plate, see image below:





In the **Picture**-tab; enter a value in this input field

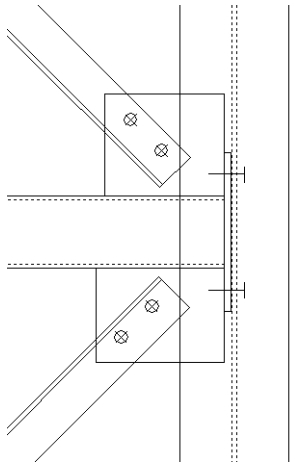
Gusset does not recognize end plate



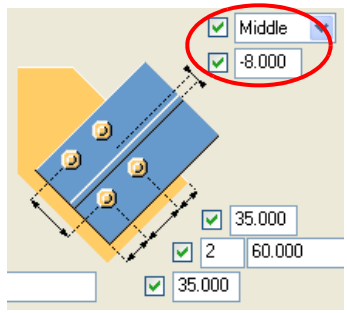
Asymmetrical bolt group

In case bolts are not exactly positioned in the center of the bracing, it may happen that after a mirror operation, the bolts are not in the correct position. This can be fixed by setting up the macro according to the pictures below.

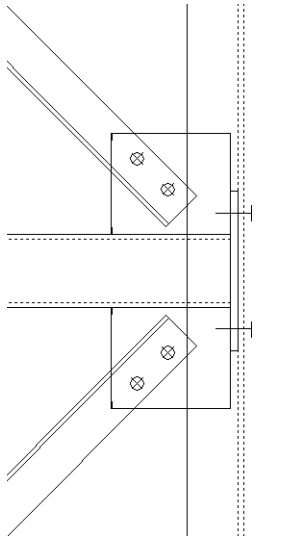
Tab **Brace bolts 1**:



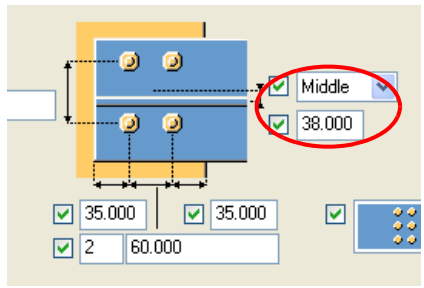
Before mirroring



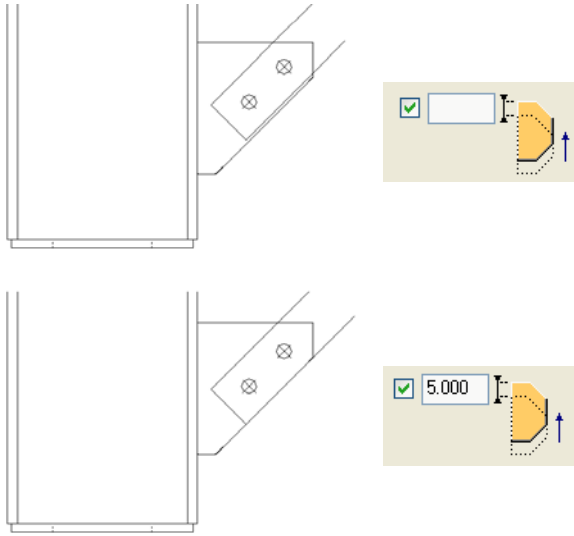
Tab **Brace bolts 2**:



After mirroring



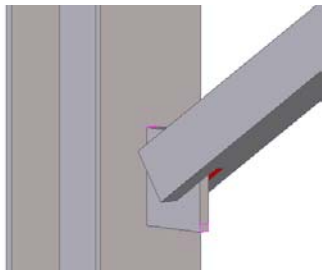
Moving gusset plate



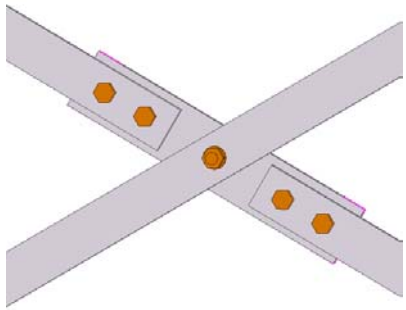
Examples

Below some examples of default settings you could use:

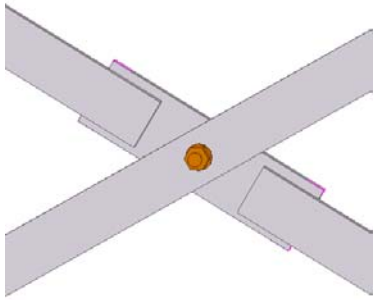
Welded connection



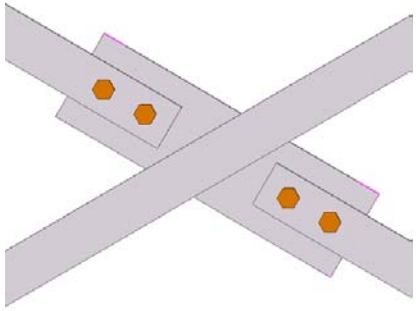
Cross bolted



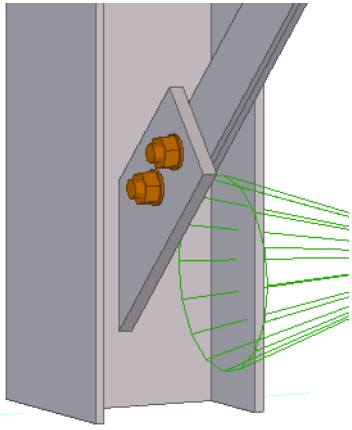
Cross welded



Cross welded bolted



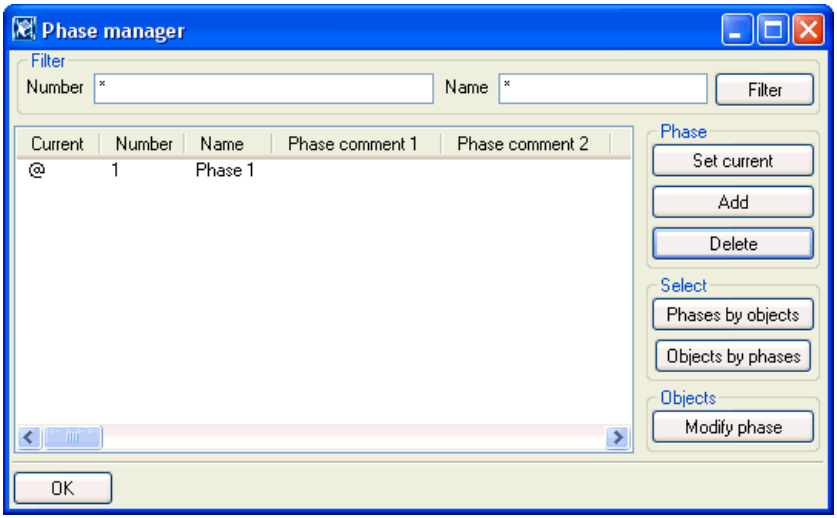
Flat bar



1.38 Phase manager

If a model is very big (or will become very big), it can be useful to split the model into several phases. Only profiles can be accommodated into phases; grids and points can not put into phases!

To open the Phase manager, go to **Properties > Phase number...** or go to **Setup > Phase number...**or press shortcut **F:**



Current phase: 1, Fase 1

In the Tekla Structures status bar, except the phase number, also the phase name is displayed now.

Description of the dialog box

Filter

Text boxes to filter for both phase numbers as well as phase names.

Set current

With this button you determine to which phase new inserted objects will belong. The current phase is indicated by an @-character. To switch phases, first select a phase, than press the button **Set Current**. Also double-clicking a phase is an option to set another phase to be the current phase.

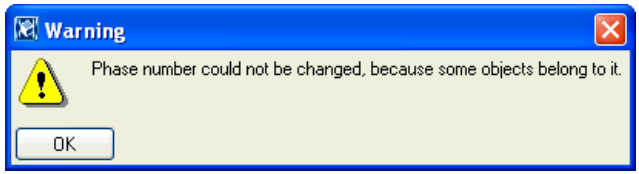
Add

With this button new phases can be added. The numbering automatically continues (phase 1 - phase 2 - phase 3 -.....).

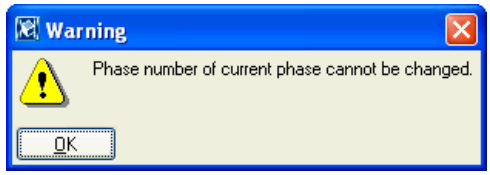
By clicking the input fields **Number**, **Name**, **Phase comment1** or **Phase comment2** once, the text can be edited.

Number	This input field can contain numbers only
Name	This input field can contain letters as well as numbers
Phase comment 1	Can contain may letters as well as numbers
Phase comment 2	Can contain may letters as well as numbers

The phase number can not be changed as long there are objects in that phase. The following warning will appear:



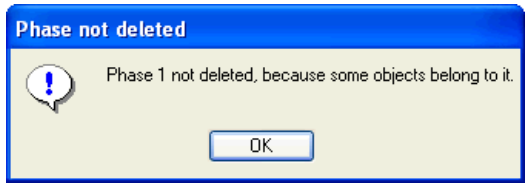
If the phase you want to change is the current phase, the next warning will be displayed:



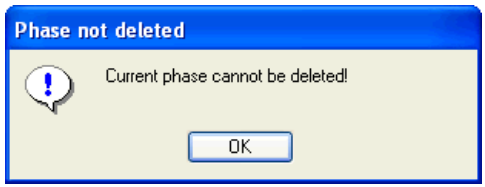
However, the name of the phase can be changed at all times.

Delete

With this button, a selected phase can be deleted from the model. A phase can not be deleted if the phase still contains objects. The warning below will appear on the screen:



or:



A phase can only be deleted if all objects are accommodated into another phase.

Phases by objects

With this button can be found out to which phase(s) the selected object(s) belong. If more than one object is selected which belong to different phases, than all phases will be displayed.

Objects by phases

With this button can be found out which objects belong to the selected phase. The object will be highlighted in the model.

Modify phase

With this button it is possible to modify the phase for one or more objects. Method: Select the objects in the model. In the Phase manager, select the phase to objects should be moved to. Than click the button **Modify phase.**

Phase representation in model views



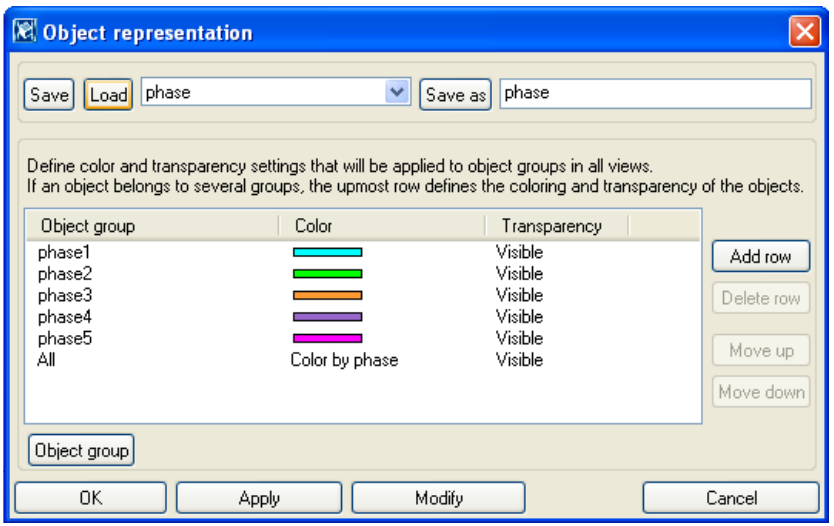
You can represent the phases in model views by using the **Object Representation manager**.

The Object Representation manager sets the color and visibility of objects in model views. These objects are grouped by a set of rules and conditions to represent these objects by these rules in model views.

Changing the settings in the Object Representation manager affect **ALL** model views.

Step plan

Go to **Setup** > **Object representation...** or click the icon, the object representation manager will open:



Select the setting **phase** in the listbox and click <**Modify**> and <**OK**>.

Profiles modelled in phase 1 are represented in cyan, profiles modelled in phase 2 are represented in green, etc.

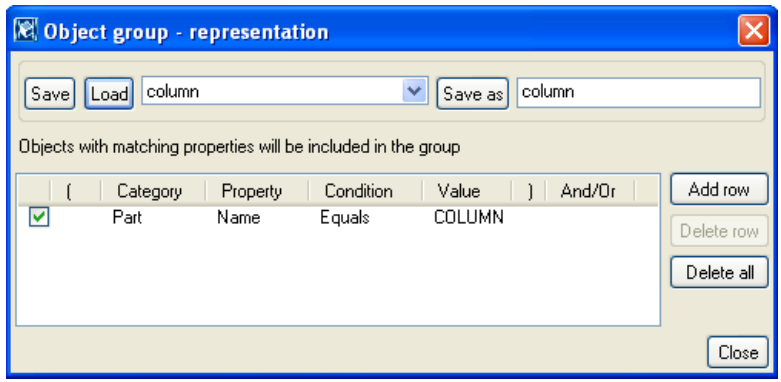
If you want to select only the "steel" in the model, make use of the default setting **only steel** so that only profiles which have material from the group **steel** will be displayed.

By default, several standard settings are included but you can also create object representation settings by yourself.

Example: We will create a new Object Group where all columns will be represented in blue - 70% transparency.

Step plan

1. In dialog box **Object Representation**, click <**Add row**>. A second Object group will be added, this group initially has the same properties as the first Object group
2. Click <**All**>. A picklist will appear containing a number of options. These are the default options for Object groups. There is no option for hiding columns, so pick **<Create new group...>**, dialog box **Object group - representation** appears
3. Leave the **Category**-option to Part
4. Leave **Property** to **Name**, we will use the part name to select columns
5. Leave **Condition** to **Equal to**
6. For **Value**, enter COLUMN (note: Capitals!). Define the dialog as follows:



7. Save the setting as **column** to be able to re-use the new Object condition in other models and click <**Close**>.

Bolts in phases

Bolts can also be divided into phases. By selecting all bolts belonging to a certain phase, very easily a bolt list can be generated of all bolts of that phase. While modelling bolts are automatically divided in the current phase.

When dividing bolts in phases afterwards, you must select bolts manually and divide them in the specific phase.

1.39 Checking the model

After completing of the model, the next step is generating drawings. However, to prevent errors in the model, it is useful to perform a global check. This check can be done on several ways: visually, using the *Clash check* command, check with help of reports and drawings. The available methods for checking the model are:

Clash Check	Check if objects intersect each other
Fly	To 'fly' through the model (in rendered views with perspective projection only)
Beep	Check incorrect welds/fittings
Dynamic zoom	Checking the model with the help of a zoom view (available in wireframe views only)
Reports	Check with help of assembly- and part numbers, numbers, etc.

Clash Check

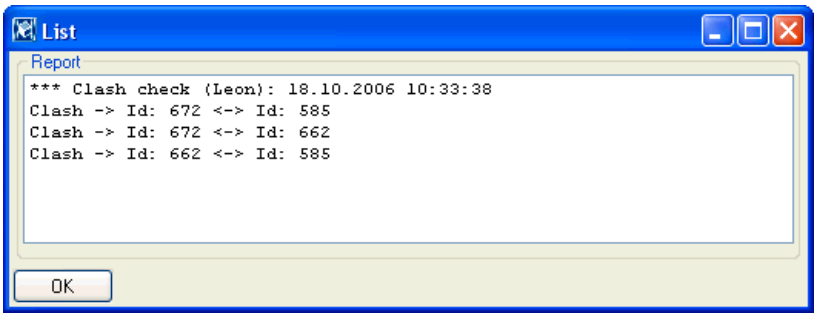


With the **Clash check**-command (located in the **Tools** menu), objects (profiles, bolts, etc.) can be checked on colliding each other. If clashes are found, a sound signal is given (beep) and the clashing object will be highlighted in yellow.

Step plan

1. Select the objects which you want to check for clashes
2. Select the **Clash Check** command, located in menu **Extra**. The command can also be activated from the contextmenu.

If clashes occur, the clashing objects will highlight in yellow and a list will be displayed in which the ID-numbers of the clashing objects are represented:



While selecting the line in the list, the accompanying object(s) will highlight in the model. If it is hard to find these objects, fit the work area by the (selected) parts.



Click on the icon or go to **View** > **Fit by parts**.

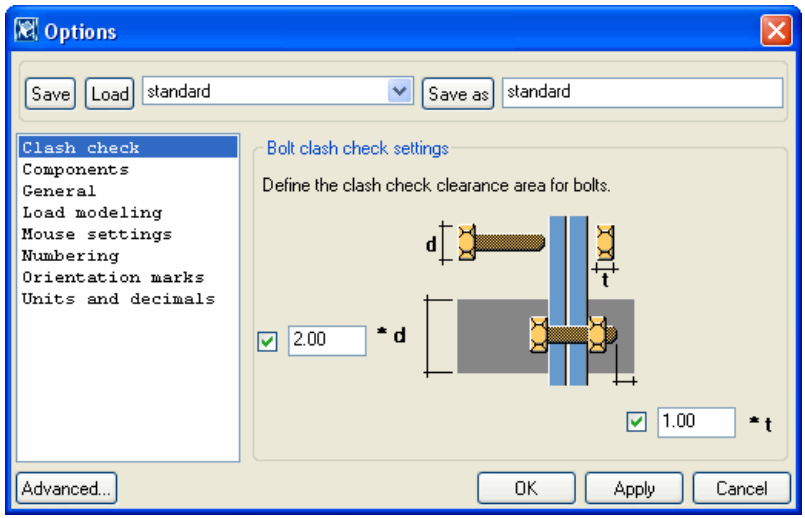


In case a large amount of objects are highlighted, it can be difficult to determine the exact location of the clash.

It that cases, it is preferable to decrease the amount of selected objects.

Bolt Clash Check

Bolts can also taken into account in the Clash Check operation. There will be checked if bolts collide with profiles, and if there is enough space to mount the bolt. The minimum clearance for the bolt can be set in the category `Clash check` in the dialog box **Options** in the pull-down menu **Tools > Options**, see image below:

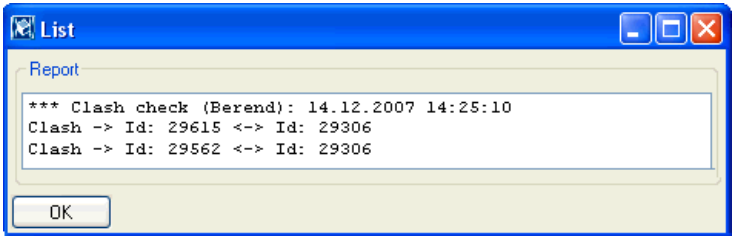


To remove the clash-color from the objects, click in the window once.

Using Clash Check for Cast Units containing imbeds

The **Clash Check** command is being used to find parts that collide. To do this, select 2 or more parts and select **Clash check** from the contextmenu (or go to **Tools > Clash Check**).

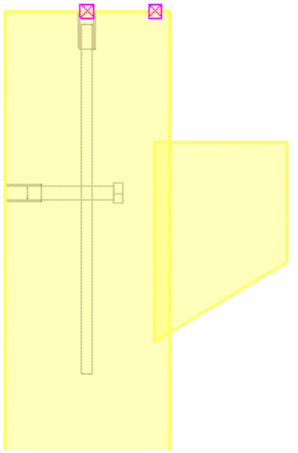
Colliding parts will highlight in yellow. Also a report will be displayed listing the ID-numbers of the colliding parts. Select a line in the report to highlight the corresponding part in the model view:





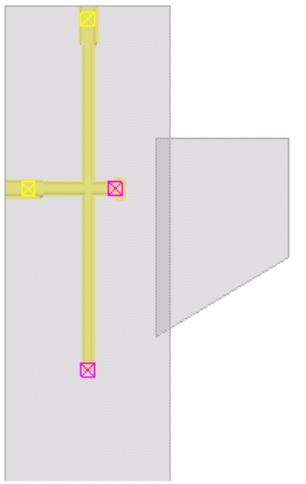
Small colliding parts may be difficult to locate, even if they are highlighted. To locate them, you fit the work area by using the command **Fit the work area by parts in selected views**.

Imbeds will collide with the cast unit. To prevent this, two clash check select filters are available:



Select filter **clash_check_element:** only cast units will be selected.

Imbeds are excluded.



Select filter **clash-check-embedded.** Only imbeds are selected, cast units are excluded.

The Clash Check command can also be used to check for collisions between imbeds and rebars. To do this, use the select filter **clash-check-reinforcement-embedded**.



No collisions will be reported between rebars and concrete parts (i.e. material from the Concrete-group).

Fly

With the **Fly** command, you can ‘travel’ through the model. The flying route can be changed by moving the mouse. The more the mouse pointer is moved from the point of departure, the faster the picture is moved. The flying speed can be altered with the scroll wheel.



The Fly command only works in rendered views with Projection set to **Perspective**.



To start flying, select the **Fly** command in the pull-down menu **Window** or click the icon. When the Fly command is started, the mouse pointer changes, see picture below.



As soon as the mouse pointer is being moved, an arrow is included to the mouse pointer to indicate the flying direction. To stop the Fly process, press the **Interrupt**-command or the **Esc**-key.

Delete “Beep” from a model

A “beep” (sound signal) indicates incorrect applied fittings or welds.

Models containing incorrect fittings or welds will “beep” when model views are opened. It is recommended to remove these “beeps”.

This can be done by removing profiles (one by one) and then pressing the <**Undo**>-button. The profile and all accompanying objects will return to the model including a sound signal when a profile is welded or fitted incorrect.

Check with help of reports

You can output the information contained in models in reports. These could be lists of drawings, bolts, parts, etc. Reports can contain information from selected parts or the entire model and therefore report are very handy to check the model. You can, among other things, generate reports of:

- Bolt-, nut- and washerlists, materiallists
- ID-lists, the main feature of these lists is that the lines in the lists which contain profiles can be selected. Subsequently, the accompanying profile will highlight in the model. This can be useful in case a profile is hard to locate.
- User defined attributes list, to check if the correct profiles are shortened

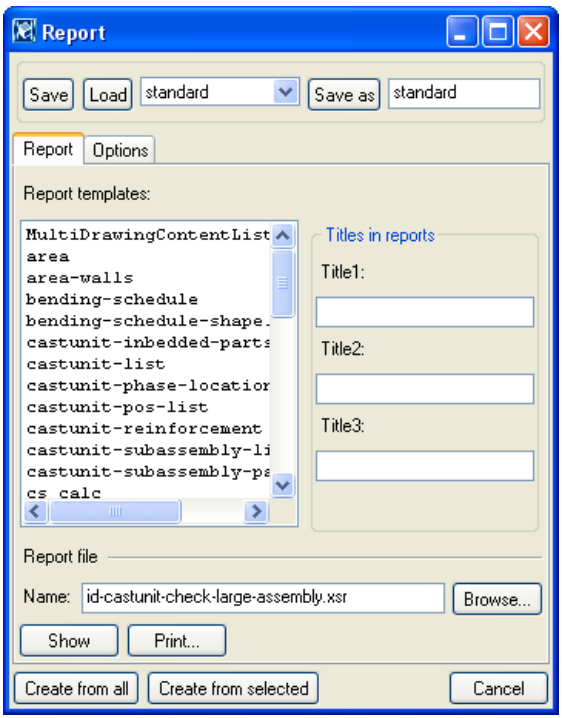
See also

For information, see **Tekla Structures.pdf** in the folder **disk:\TeklaStructures\version\environments\europe\pdf**.



Reports can be generated as follows:

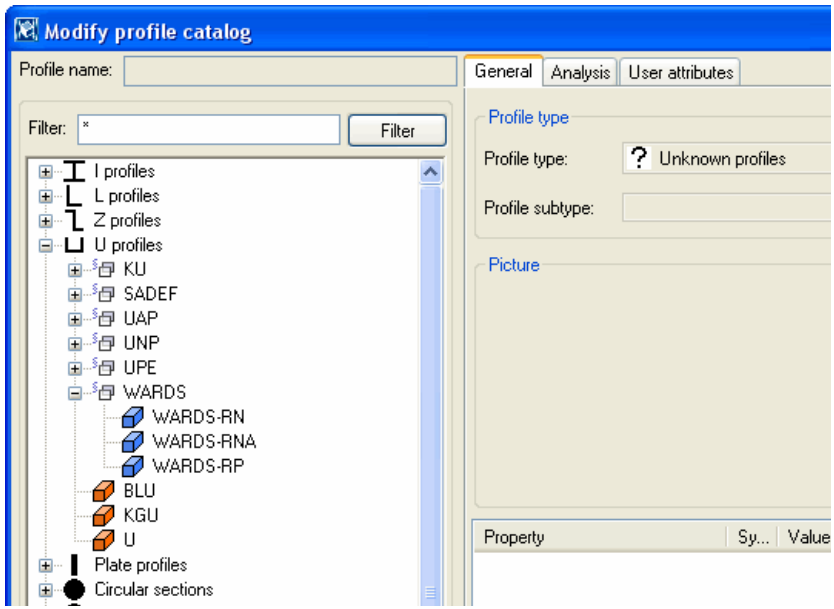
1. Go to Tools > Numbering > Full
2. Go to **File > Reports** or click the icon opposite. Select a report and click “Create from all” or “Create from selected”. Click **Show** to display the report on the screen.



1.40 Profile catalog



Go to **File > Catalog > Profile > Modify...** or click the icon opposite to open the profile catalog, the following dialog box appears:



Tree structure

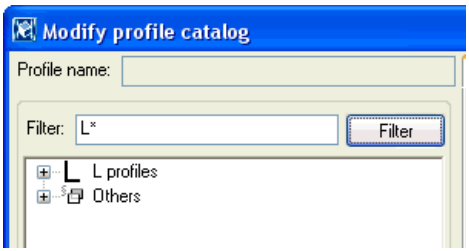
In Tekla Structures the profiles are displayed in a so-called tree structure. Profiles are grouped according to rules such as profile type (e.g. I profiles) and profile sub-type (e.g. HEA. Different icons denote profile types, rules and profiles within the tree:

Icon	Used to show
	Profile type rule. Different icons show different types.
	Rule
	Library profile
	Parametric profile

The Profile catalog is setup in a way that profiles can be located and selected easily.

Filter

The input field *Filter* filters profile types. For example, when entering ***L**** and pressing the *Filter* button, only profile names starting with "**L**" will be displayed.



You can now select the needed profile, the name and details of the selected profile are displayed in the dialog box.

Profile type

The available profile cross-sections.

Profile properties

The tab *General* displays information about the profile type and dimensions. The tab *Analysis* shows information about the properties of the profile and is used for construction-analysis, like the design-check while calculating connections. The tab *User attributes* is for viewing and assigning User attributes to profiles.

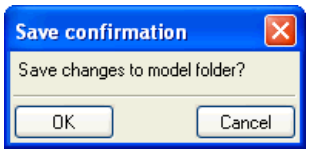
Height and width values *h* and *b* have a special meaning in the “User defined profiles”. Those values have to match exactly with the real values of the cross-section of the profile, otherwise problems may occur when applying macros. The height and width values are also taken into account when applying Right/middle/left offsets.

Update

Stores all modifications to the memory. Only if the <**OK**> button is pressed, the modifications to the catalog are stored permanently.

OK

Stores the modifications to the catalog and saves it on the hard disk. A dialog box will appear to confirm the save action. A modified profile catalog will be saved in the model folder and will be available only for that current model. To make the modified catalog available for all models, copy the catalog file (the file **profdb.bin**) to the folder **ts**.



Cancel

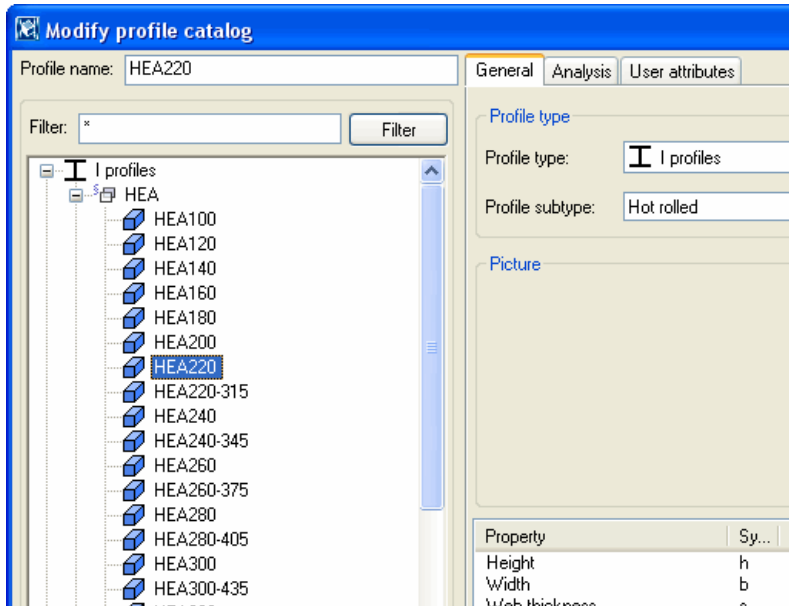
The <**Cancel>** button is used for closing the dialog box without saving modifications., modifications carried through formerly by pressing <**Update**> will not be saved.

Editing profiles in the catalog

With this function, parameters from existing profiles can be edited.

Step plan

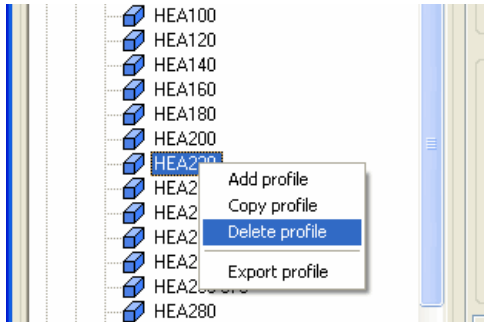
1. In the column “Value” the dimensions can be changed by clicking the input field
2. When all changes have been carried out, click <**Update**> and <**OK**>.



Removing profiles from the catalog

Step plan

1. Select the profile which has to be deleted and click the Right mouse button
2. Select the option **Delete profile**
3. By pressing <**OK**> and confirm the removal, the profile will be removed from the catalog.

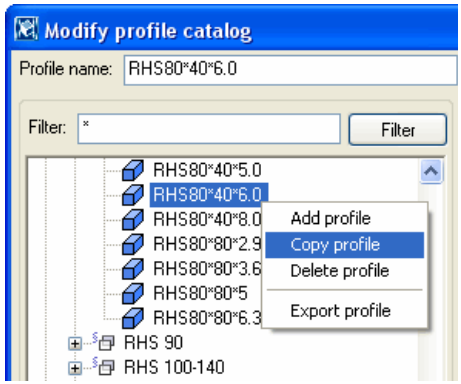


Adding profiles to the catalog

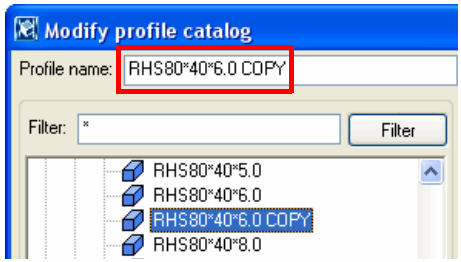
Adding a box girder profile RHS80*40*6.3.

Step plan

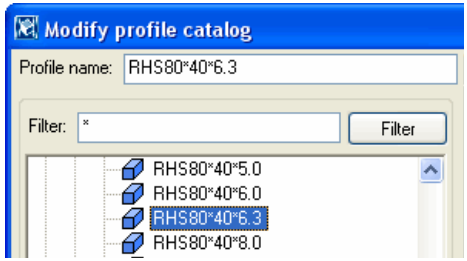
1. Select an existing profile which matches best with the new profile and click the right mouse button
2. Select the option **Copy profile**



The copied box girder profile is added with the name RHS80*40*6 COPY



3. You can now edit the profiles' name and enter the correct values in the "Value"
4. Click <**Update**> and <**OK**>

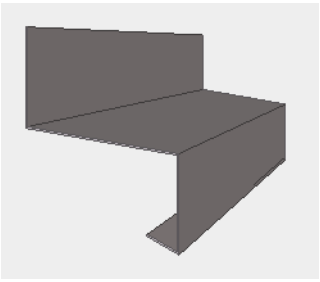


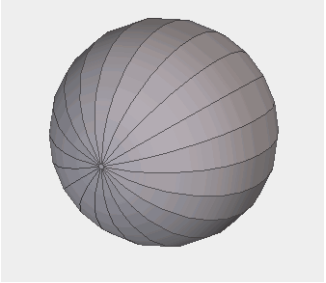
See also

For information about the profile database, see **Tekla Structures.pdf** in the folder **disk:\TeklaStructures\version\environments\europe\pdf** or check the Tekla Structures Online Help.

1.41 Using parametric user profiles

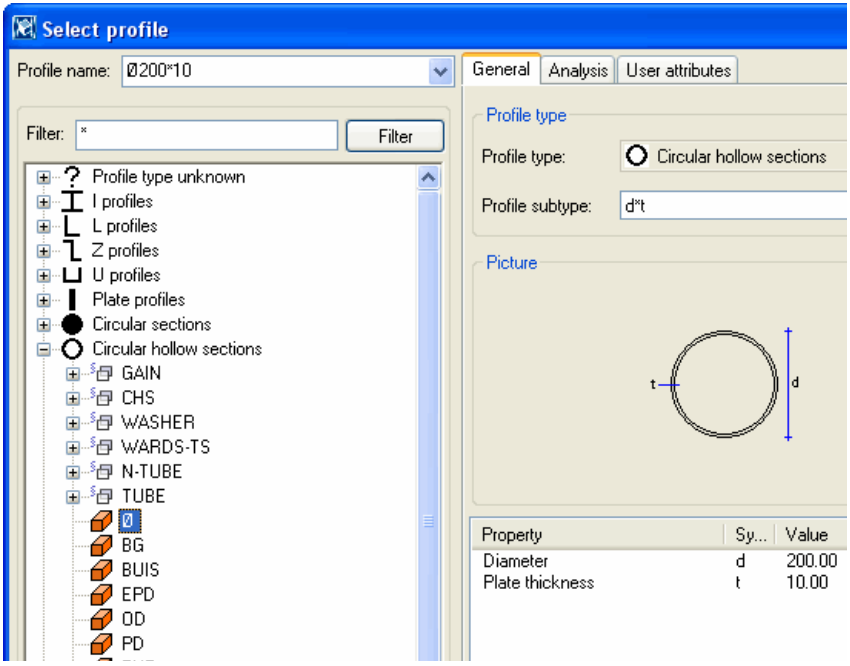
A lot of parametric user defined profiles are available in Tekla Structures. For example a gutter profile or a ball as shown in the images opposite:





Parametric user profiles are not polybeams; they can not be unfolded nor a flat-pattern can be created!

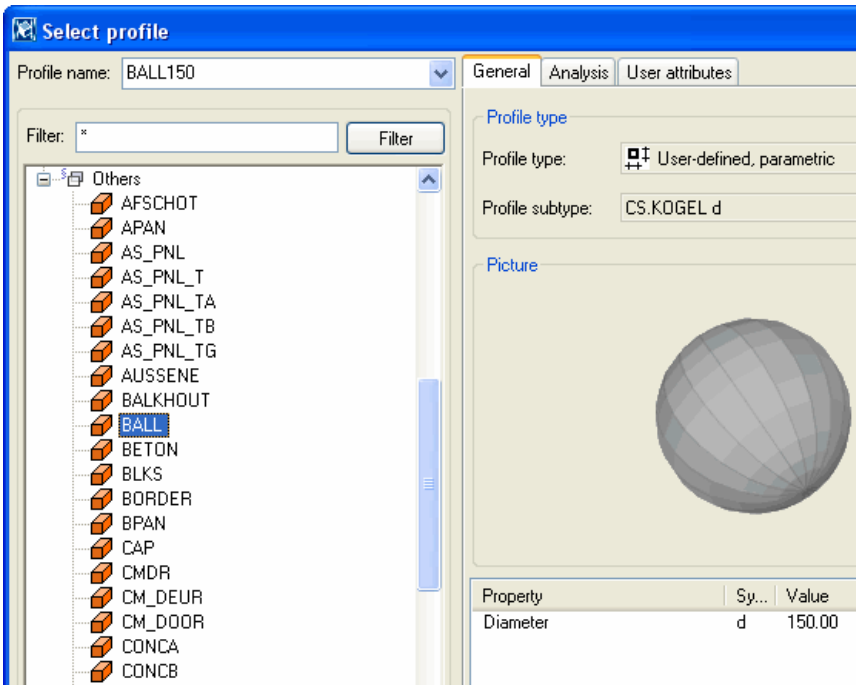
The library profiles also contain lots of parametric profiles:



In this chapter we will only focus on parametric user defined profiles.

Selecting a parametric profile

To select a parametric user profile, go to the folder **Others** in the profile catalog:



Select a parametric user profile and enter the correct values, now click <**Apply**> and <**OK**>.



To check which parametric user profiles are available, use the arrows **Up** and **Down** on your keyboard.

See also For more information about parametric profiles and a list with examples, see the Tekla Structures Help File.

Sketching parametric profiles

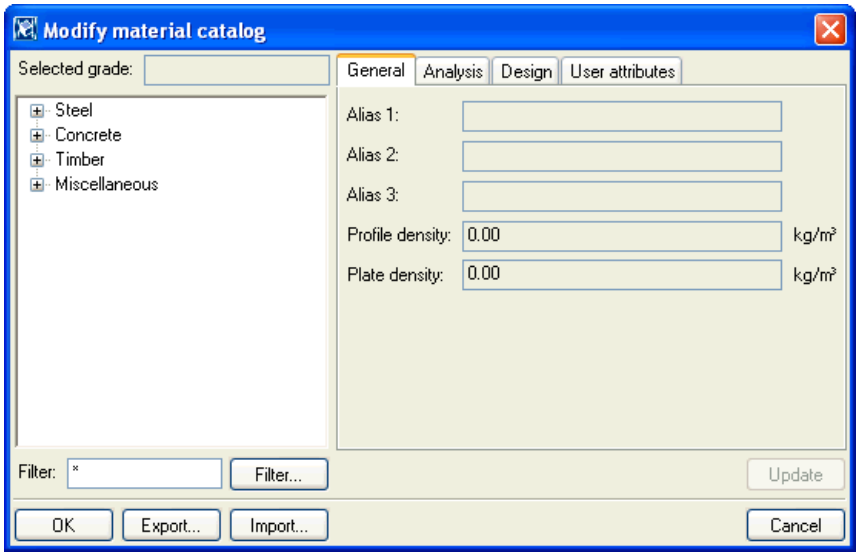
Tekla Structures includes a cross section Sketch Editor, which you can use to create your own profile cross sections. Sketched cross sections are parametric, so you can change their dimensions each time you use them in a model.

See also For information about the Sketch Editor, see **Tekla Structures.pdf** in the folder **disk:\TeklaStructures\version\environments\europe\pdf**.

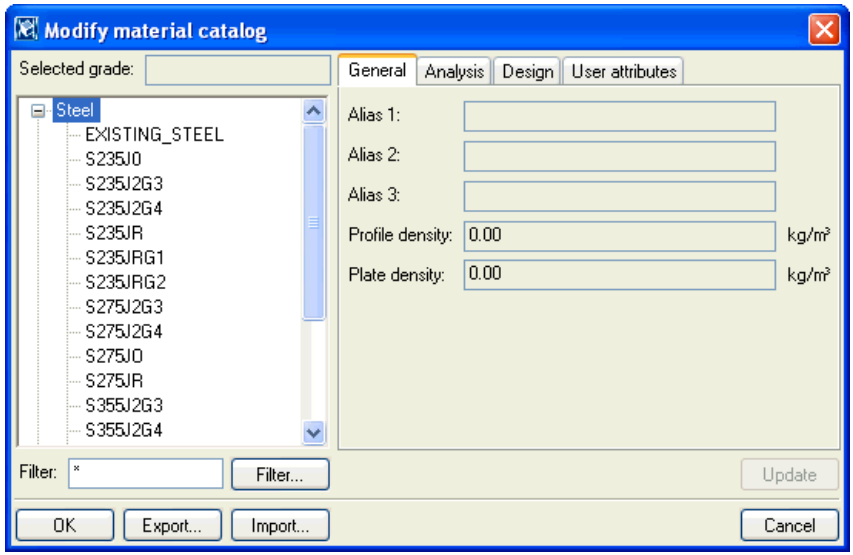
1.42 Material database



To open the dialog box, go to **File > Catalog > Materials > Modify...** or click the icon.



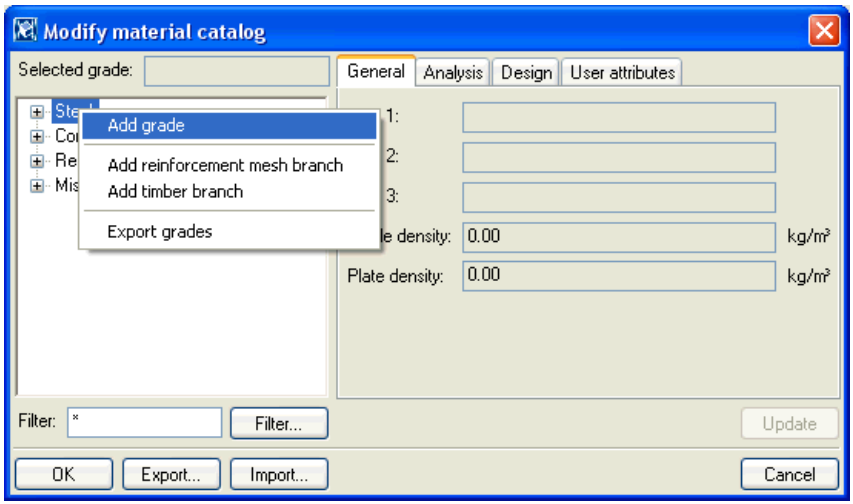
All grades are grouped. That means, steel grades are listed in the “Steel”-group, concrete grades are in the “Concrete”-group, etc.



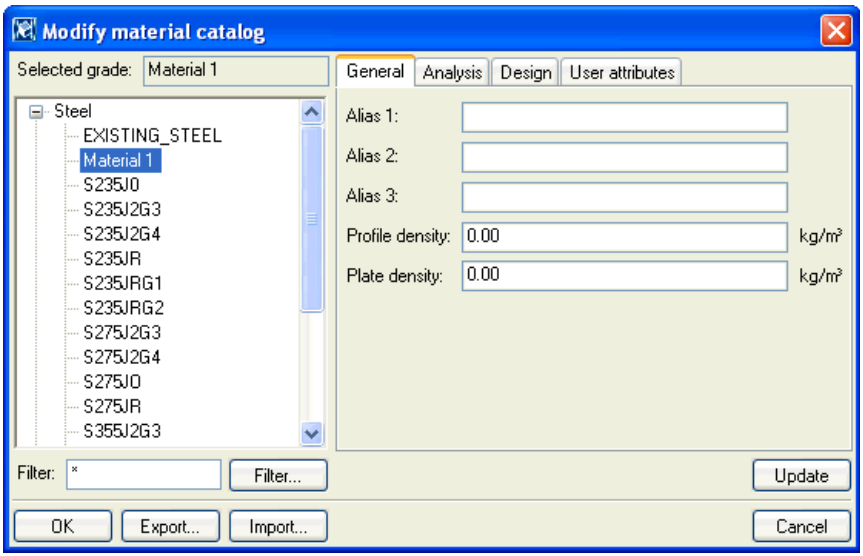
To add a steel grade:

Step plan

Select steel and click right mouse button, now select **Add Grade**



The default name of the new added grade is **Material 1**. To rename the grade, click on the grade name.





Don't use spaces in the grades' name. It is better to use, for example, an underscore-sign (_).

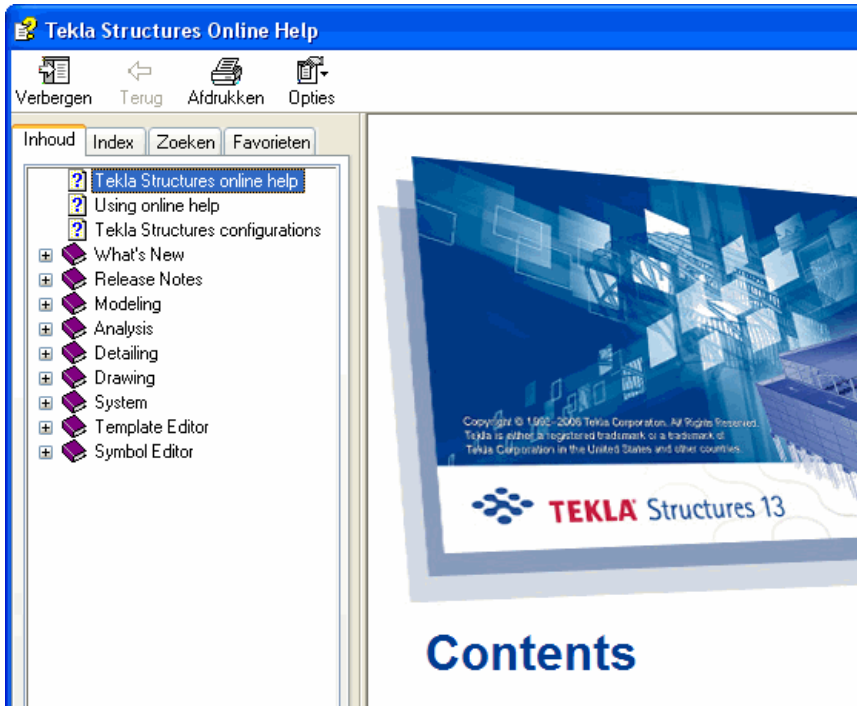
All materials from the **Concrete** group will highlight in light blue color when asking for the assembly information through **Inquire** > **Assembly**.

If you want to use the yellow and red colors for the concrete grade, delete the grades from the **Concrete** group and add them to the **Miscellaneous** group.

1.43 Hints & tips

Tekla Structures Help

The Tekla Structures Help system is task oriented and easy to use. The set-up is very Windows oriented. To open the Help dialog box, press F1 or go to **Help** > **Tekla Structures Help F1**.

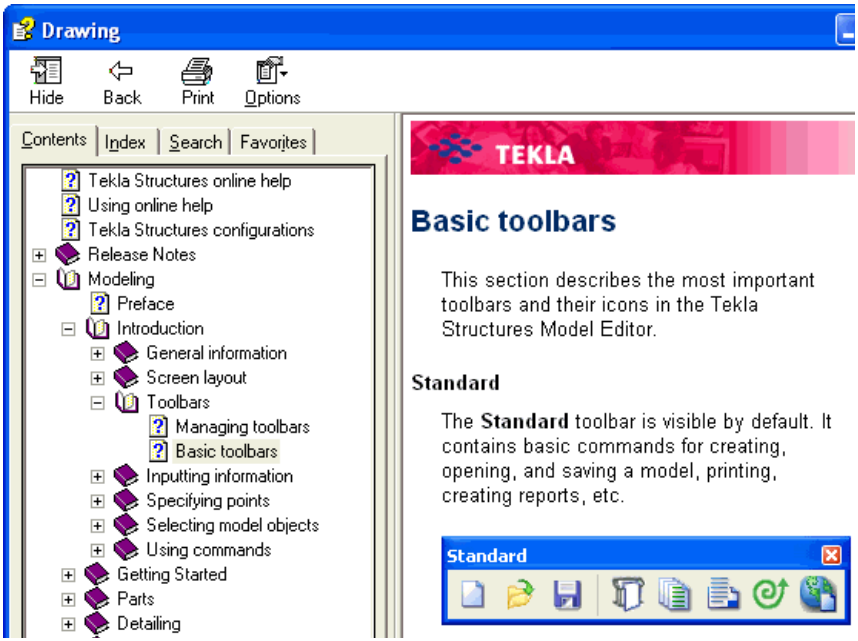


Tab Contents

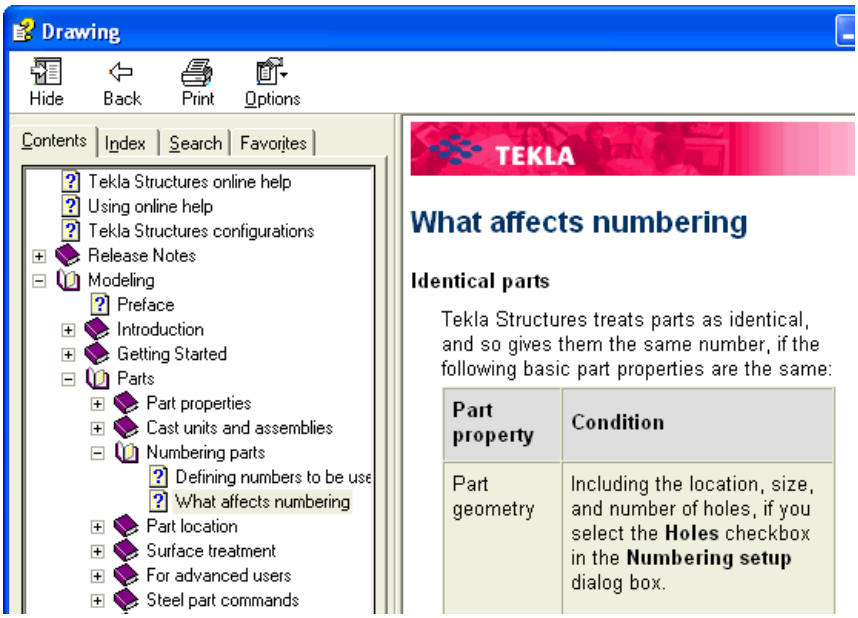
The online help system screen is divided in two parts.

The left part is used for look-up and are the topics arranged, the right side displays the contents of the topics, very often an example is added. The left side of the online help system is very clear organized, it starts with some explanation about how the help system works. After that, 9 main groups appear.

By pressing the '+' symbol in front of a main group, the contents of that group will be opened. Main group **Modeling** for example is divided into several sub groups.



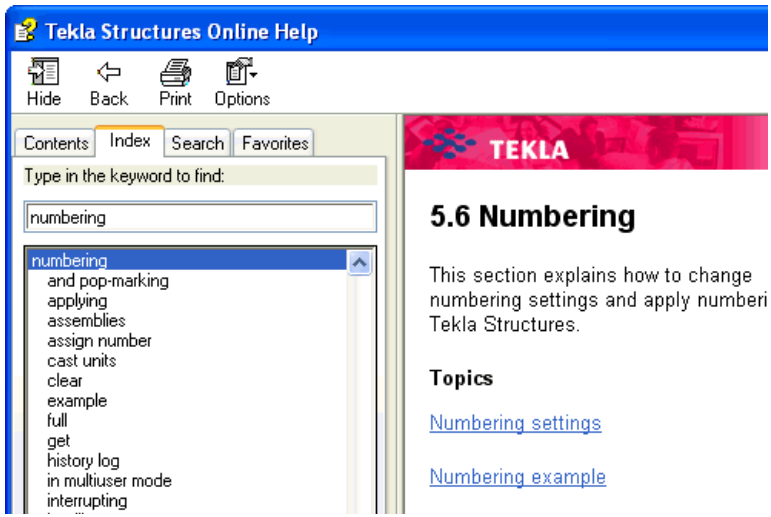
In this way you can page through the topics where you want to have more information about. For example, you want to know which settings affect the numbering of the model?



By clicking a topic, detailed information about the topic will be displayed on the right half of the dialog box.

Tab Index

Sometimes, it can be difficult to gain quickly the information you are looking for. Therefore an **Index**-tab is added to the system.

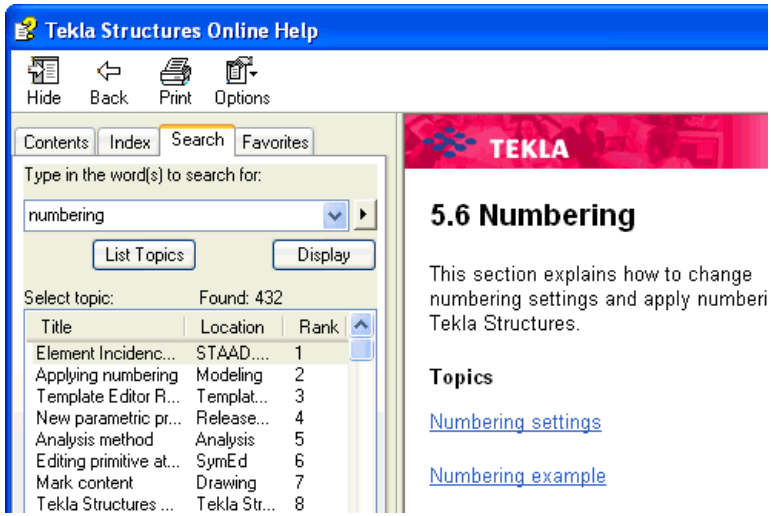


In the Index-tab, a keyword can be entered. All relevant topics about this keyword are showed in alphabetical order. By double-clicking the topic, the detailed information is shown on the right side of the dialog box.

(Selecting the topic and pressing the Display-button has the same effect)

Tab Search

Also a Search function is assimilated in the Help system.



In tab **Search**, enter a keyword and press **List Topics**.

All topics which contain the keyword are displayed. Double-click the topic to view detailed information, or select the topic and press the **Display**-button.

It is also possible to use wild cards, with help of the *- and ?-characters. For example, by entering **num***, there will be searched for all words starting with num, that means **number, numbering, numbers, etc.** will be found. Entering **pla?e** will show all topics containing the words **plane, place** and **plate**.

Also “Boolean operators” (AND, +, OR, NOT, -, NEAR) can be used in the Search function.

For example: search for topics containing the words “number” and “name”: search for **number and name** *(number + name)*.

Tab Favorites

If you have reached a topic which you think you want to refer to often, than it is possible to add this topic to your ‘Favorites’. To do this, go to tab **Favorites** and click the <**Add>** button at the bottom of the dialog box.

Standard documents

There is a standard document which gives extra clarification about several topics. The documents’ name is **TeklaStructures.pdf** and is stored in the folder: **disk:\TeklaStructures\version\environments\europe\pdf.**

Construsoft Xtranet

On the Construsoft website (**www.construsoft.com**), all customers can log on to the Xtranet section. All customers can submit a request for this. After submitting this request you will receive an e-mail to indicate that you have been added to the database. Now you will have access to the Xtranet section; in this section are several useful documents and files available for download. You will also be kept informed to new downloads and changes.

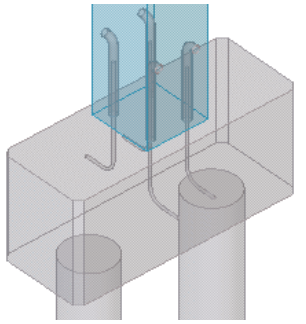
Also information about Frequently Asked Questions will be available. So log on frequently!!



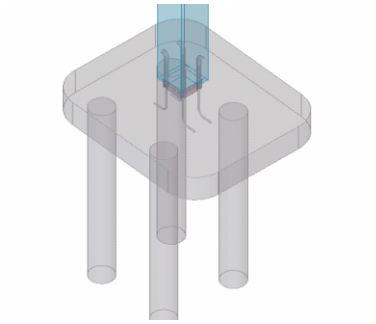
1.44 Component examples concrete

Below you will find a lot of very often used components:

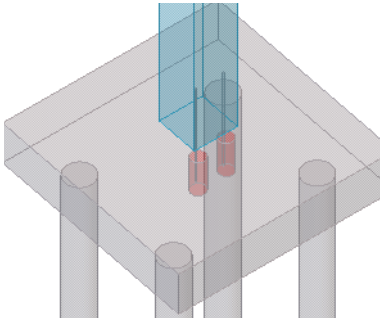
Concrete foundation (1030)



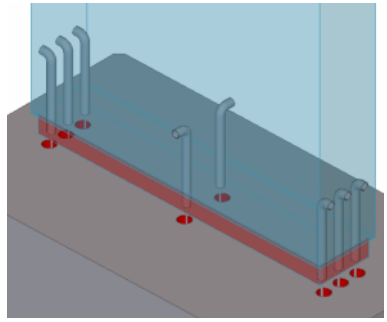
footing plate with 2 posts



eccentric column

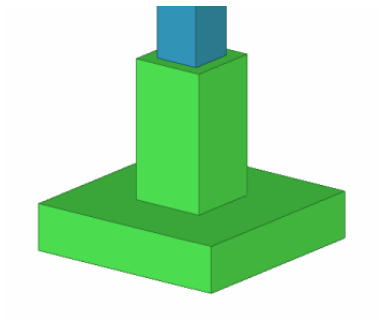


column + starter bars

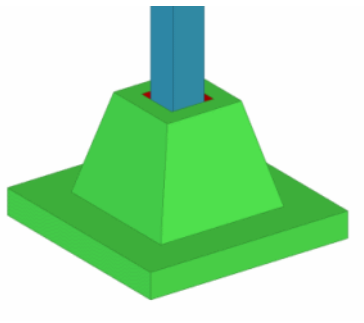


gains

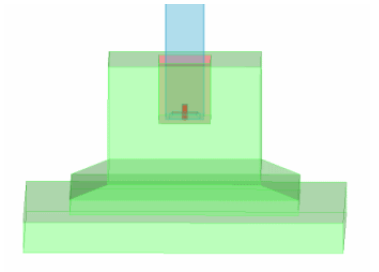
Precast found block (1028)



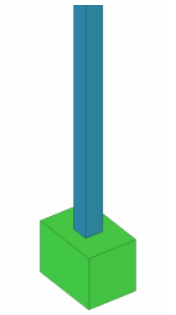
foundation block



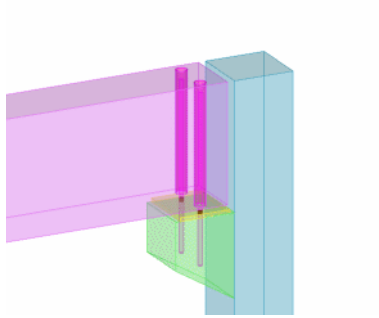
sloped planes and recess



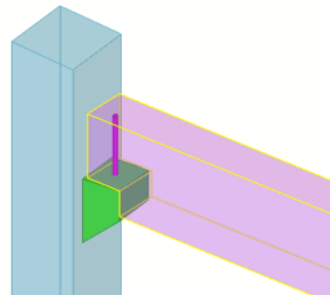
connected to a column



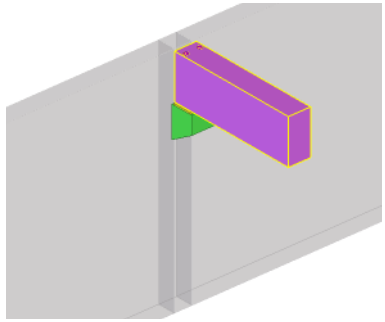
Concrete console (110)



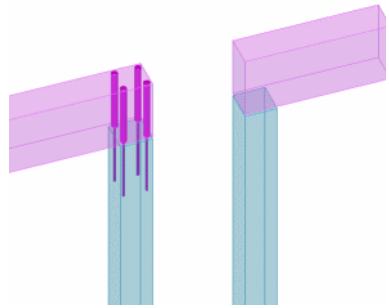
default console



intermediate sloped connection

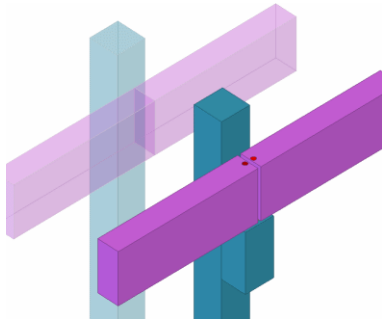


console in panel

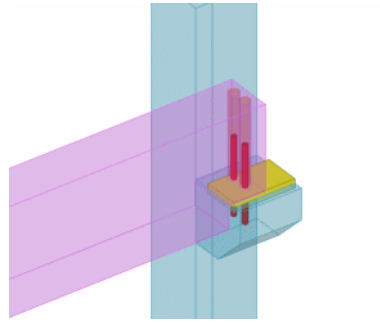


corbel connection

Concrete console (111)

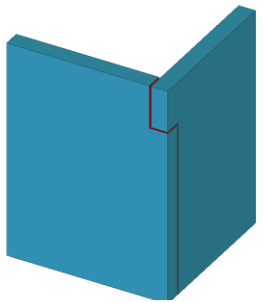


concrete console

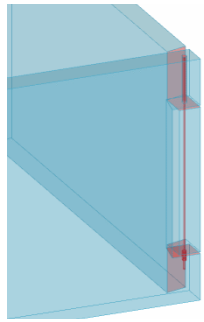


intermediate console + anchors

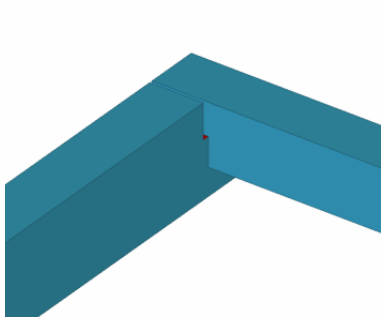
Wall wall teeth (12)



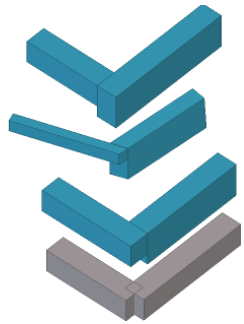
wall wall teeth panel



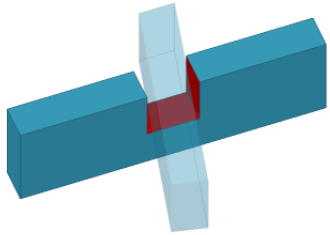
sloped angle + connection



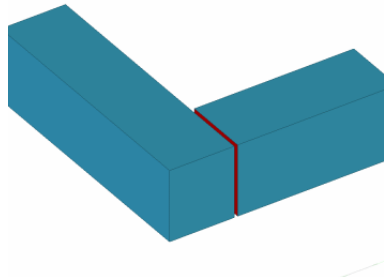
2 beams, prefab connection



beams, cast in place connection

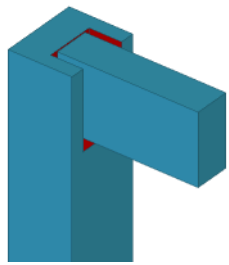
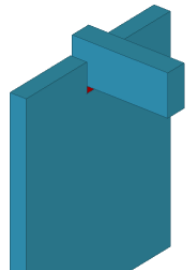
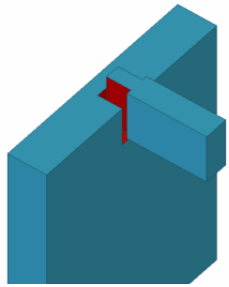
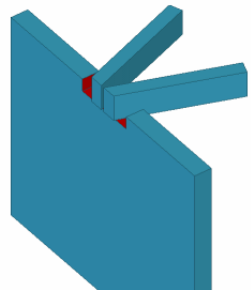


continuous beam precast

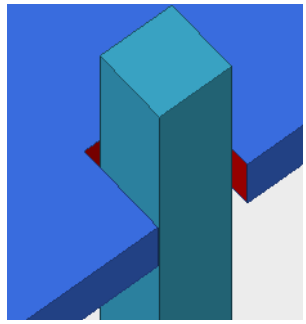


connection + seam

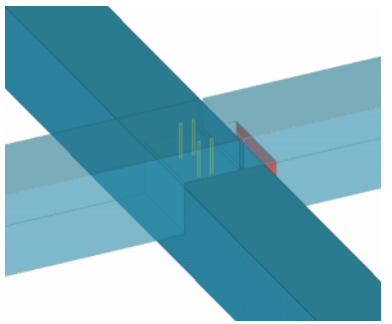
Column - beam (14)

 column beam connection	 connection panel beam
 recesses	 sloped connection

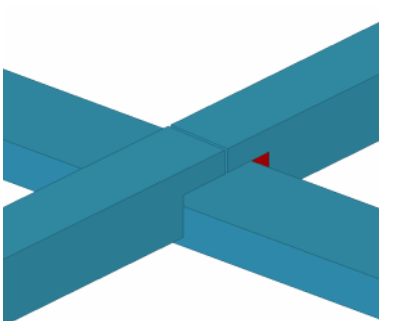
Grating penetration (92)

 Column penetrates floor	

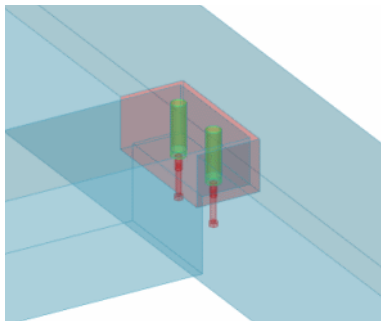
Concrete beam - beam (112)



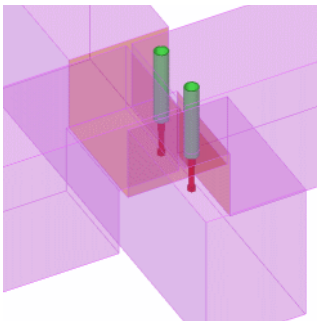
connection + anchors



sloped connection

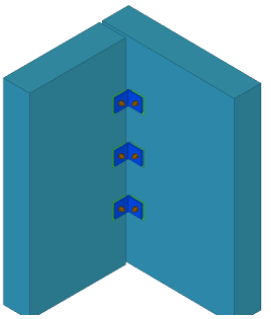


anchors+ gain single

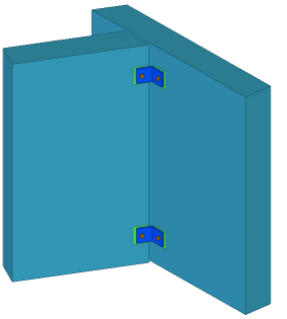


anchors + gain double

Anchor (10)

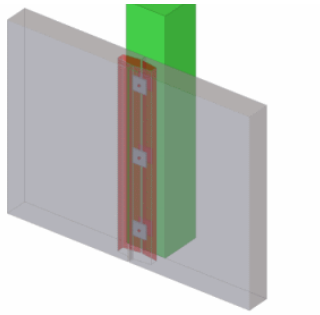


default connection

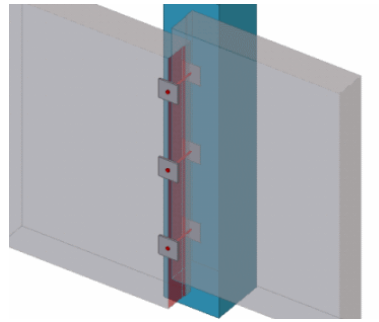


sloped connection

Column - 2 panels (12)

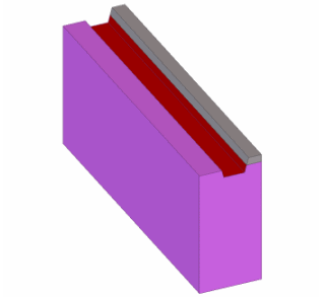


recess in panels

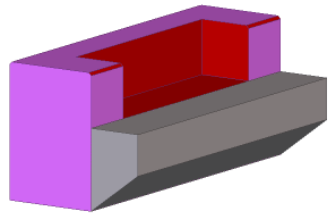


panel connection

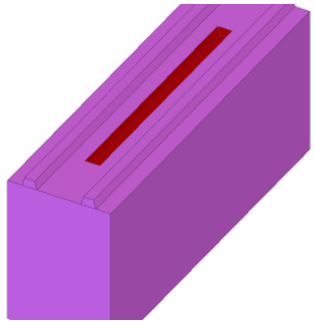
Parts at beam (82)



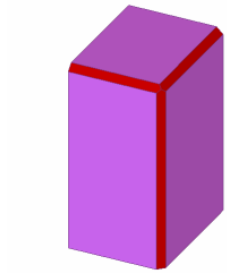
gutter + elevation



support beam, trenching, chamfering

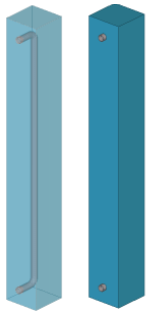
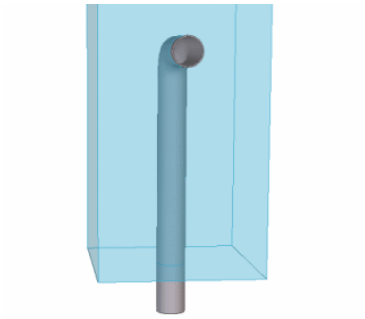


striations + trenching

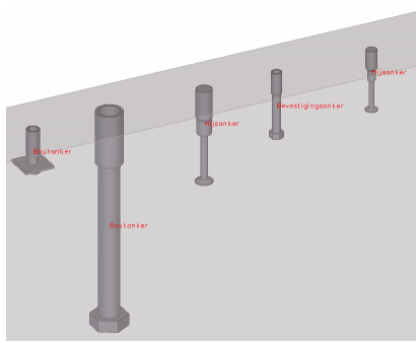


chamfering 3x

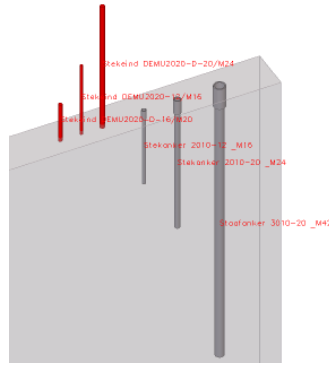
Tube in concrete column (83)

	
tube in column	supply situation

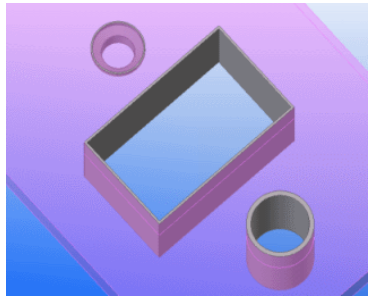
Embedded anchors (8) and Embedded (1008)



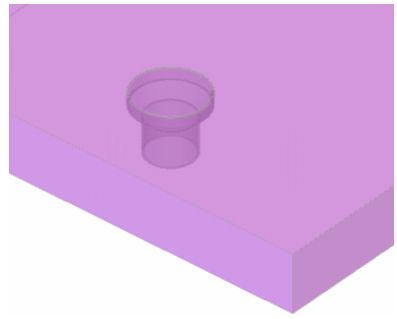
Lifting- and bolt anchors

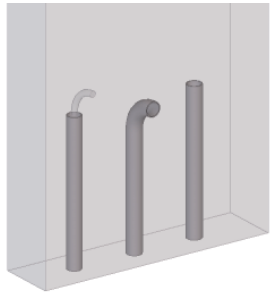


anchor bars

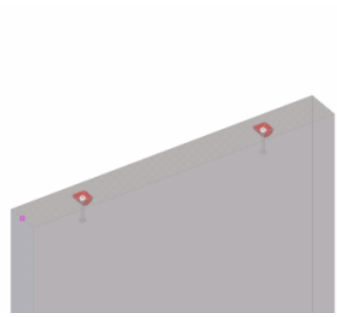


socket rectangle and tube



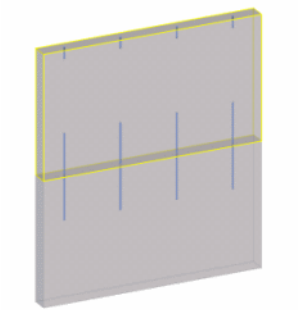


gains

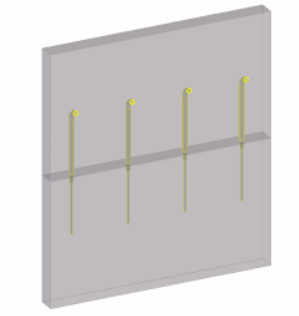


several anchors

Seam anchors (31)

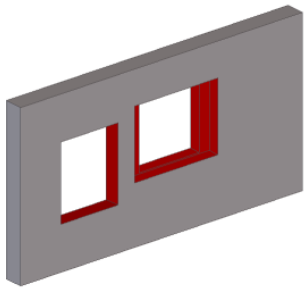


connection with anchors

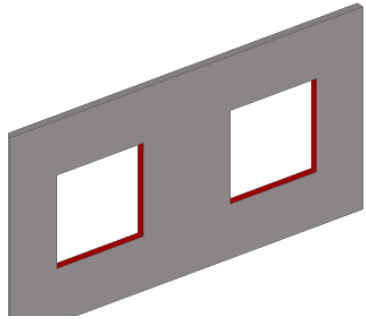


connection with gains

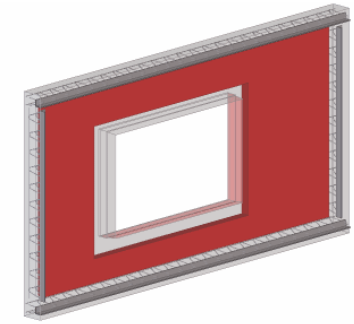
Opening in wall (40)



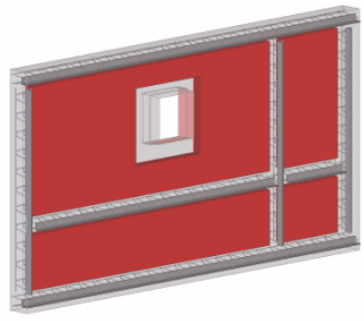
define rabbet



several openings

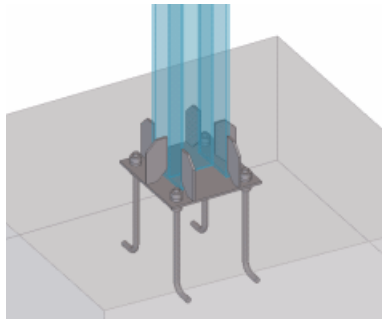
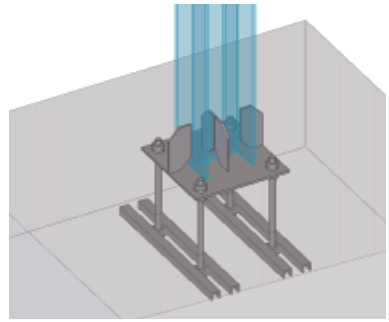


adding insulation

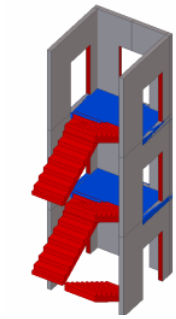
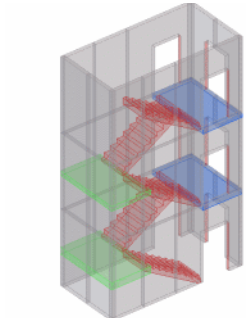


braced girders

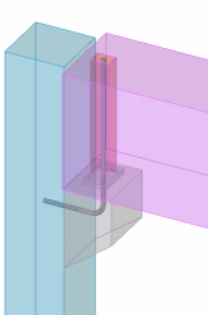
Foot plate for steel column (29)

	
foot plate + stiffeners	extra profiles

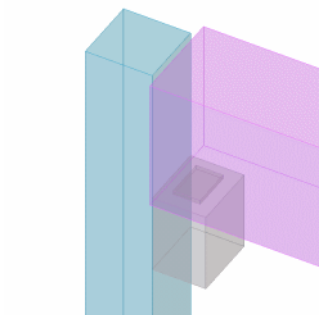
Stairwells and elevation shafts (90)

	
default	

Corbel connection (14)

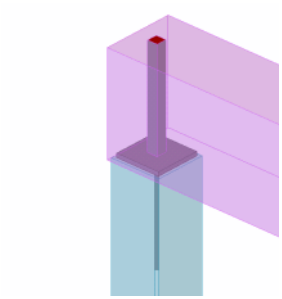


default

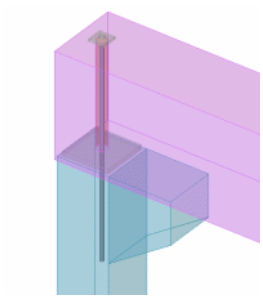


plate

Seating with dowel (75)

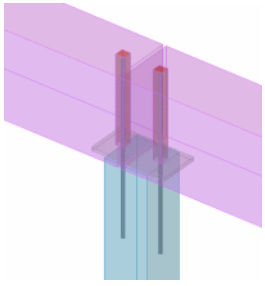


default

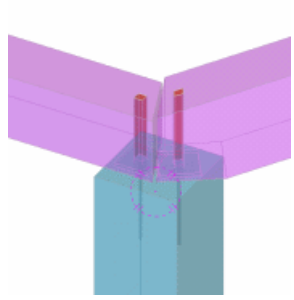


corbel

Two sided seating with dowel (76))

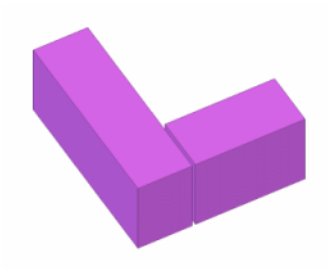
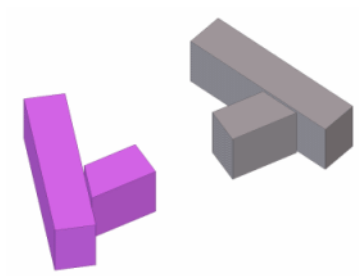


default

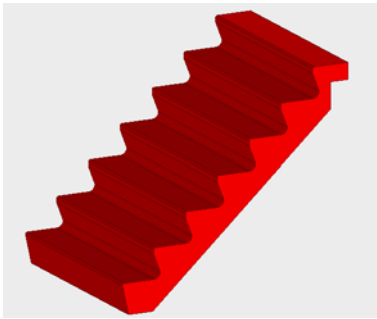
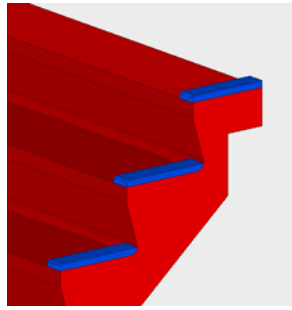
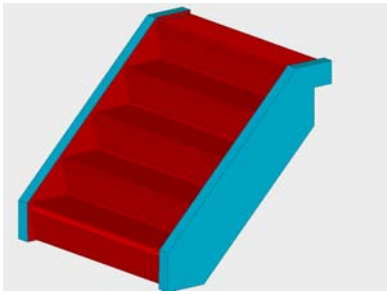
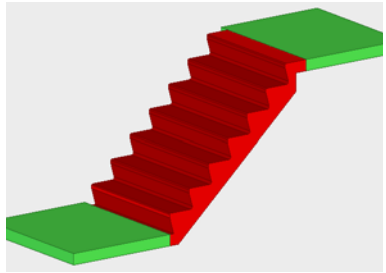


sloped angle

Battering connection (13)

	
default	sloped angle, mirrored shape

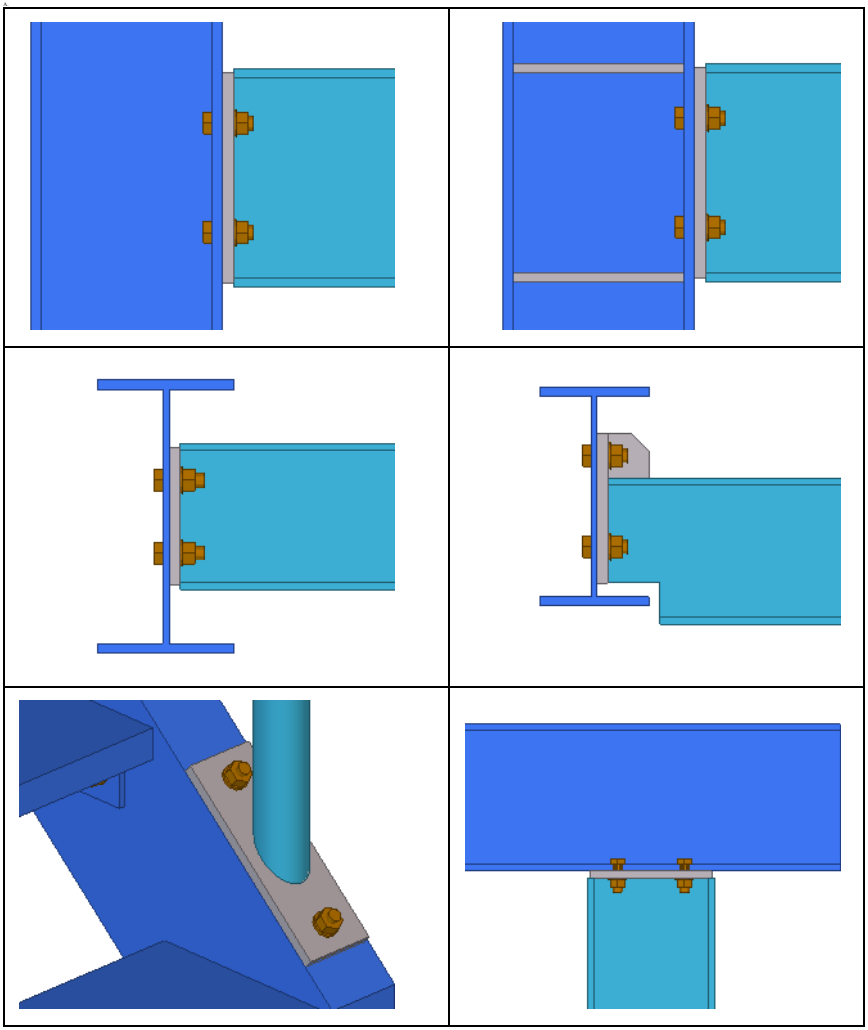
Concrete stairs (65)

	
default	
	
includes stringer	includes landing

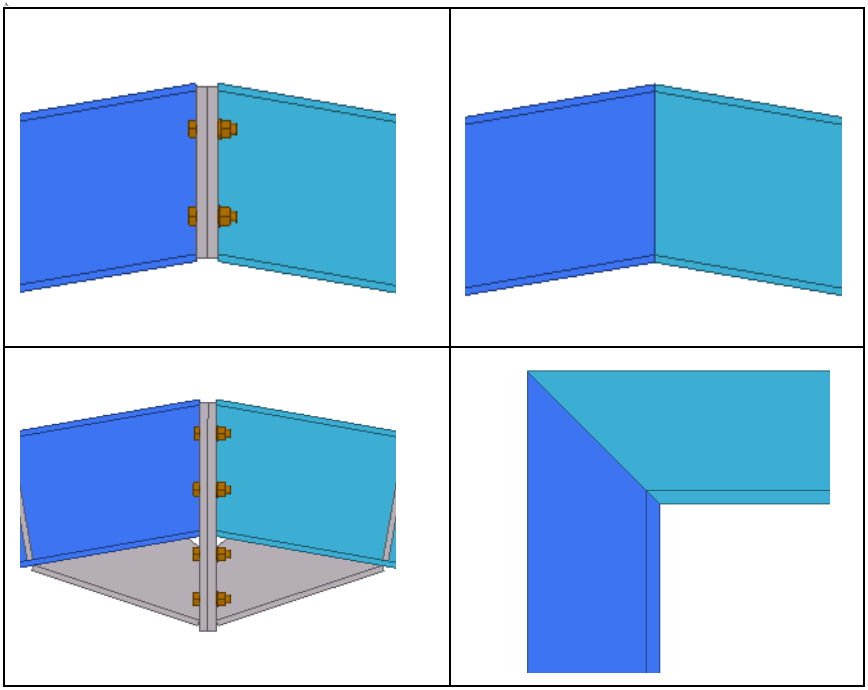
1.45 Component examples steel

Below some examples of very often used components:

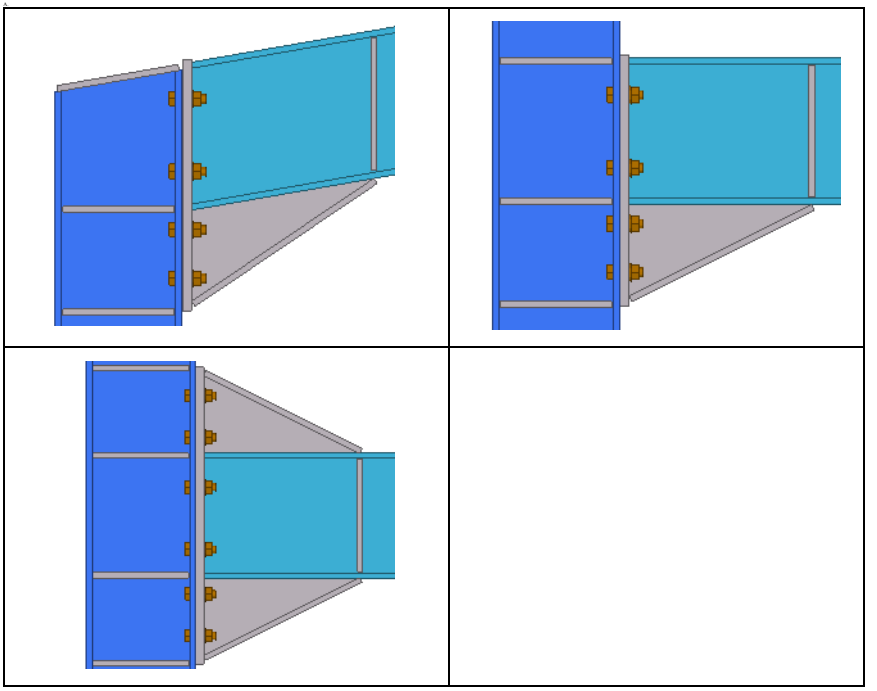
End plate (144)



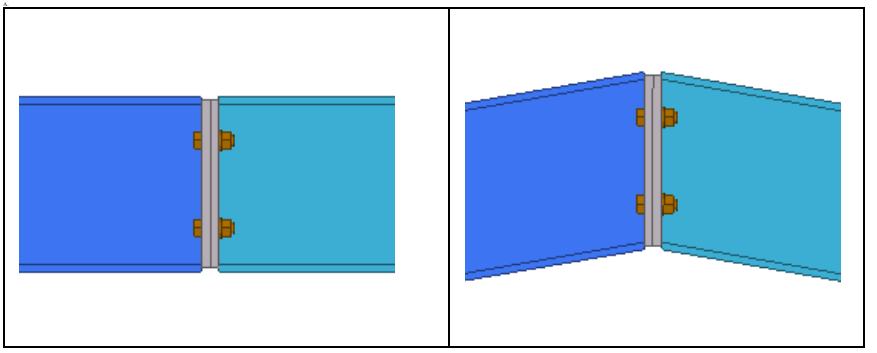
Cranked beam (41)



Haunch (40)

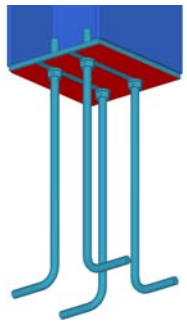


Joining plates (14)



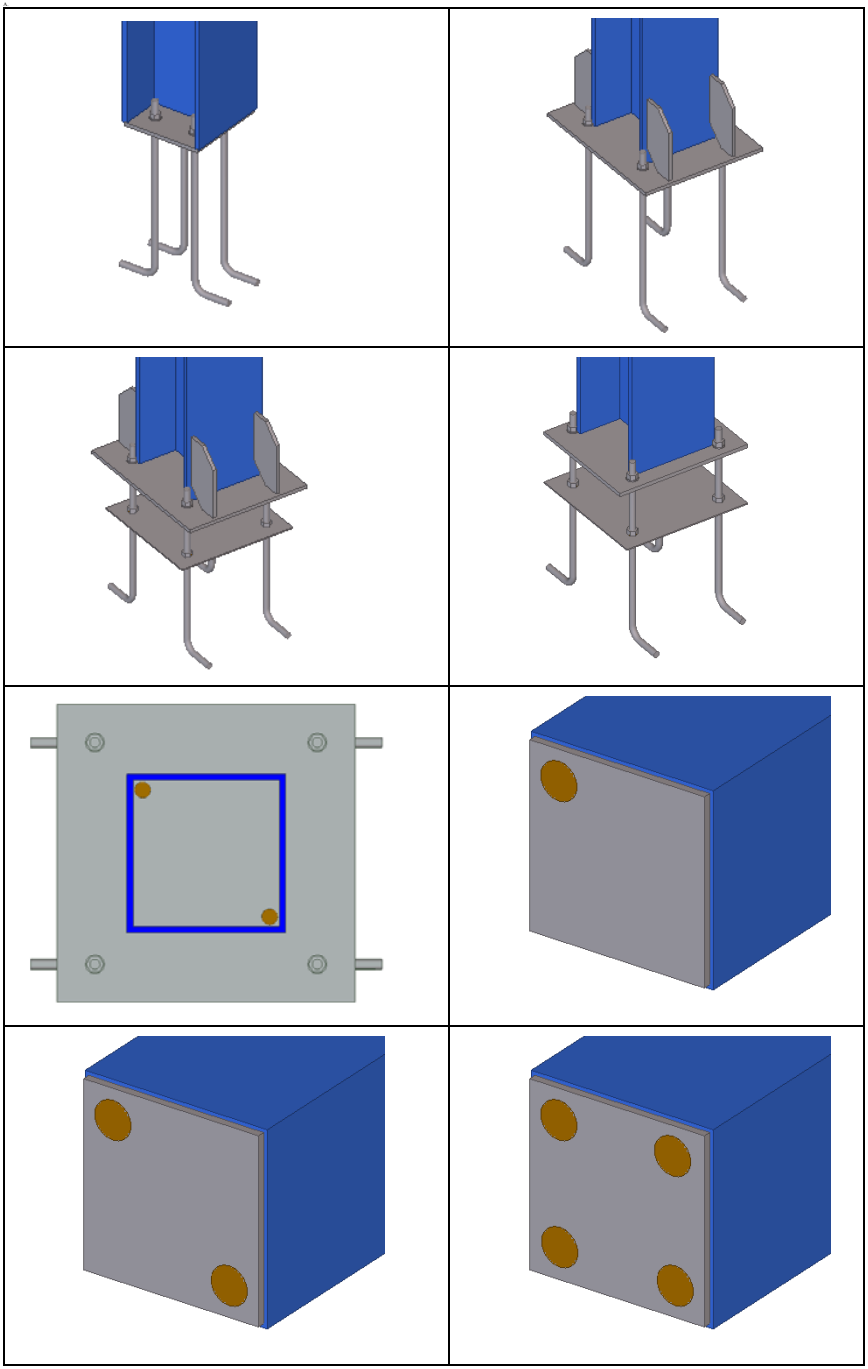
Base plate (1004)

This component includes an option to create one or more filler plates.

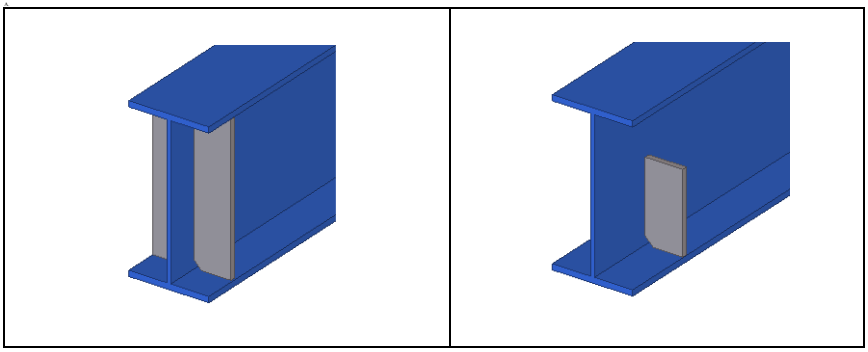


Foot plate (1029)

With the “standard” setting, a base plate is created with a landing plate including nail holes. A base plate only is also an option (landing plate t=0) with or without grout holes. Also additional stiffeners can be added.

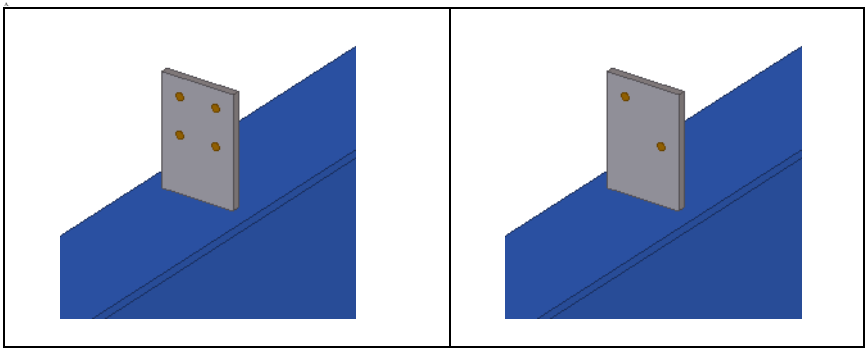


Stiffeners (1003)

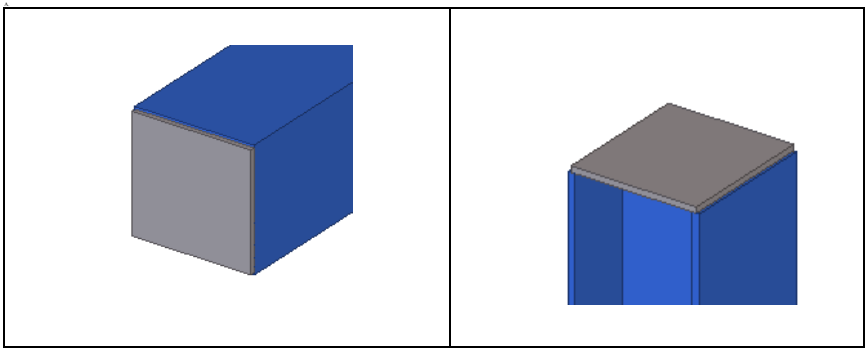


Stub (1011)

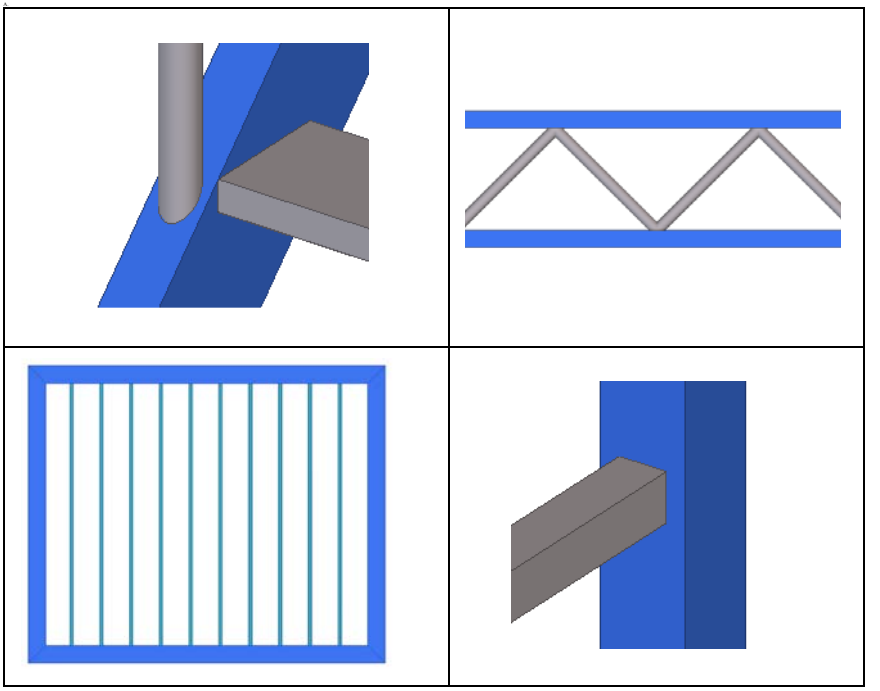
Make use of the bolt delete option on the tab **Bolts** to determine which holes should be included in the stub.



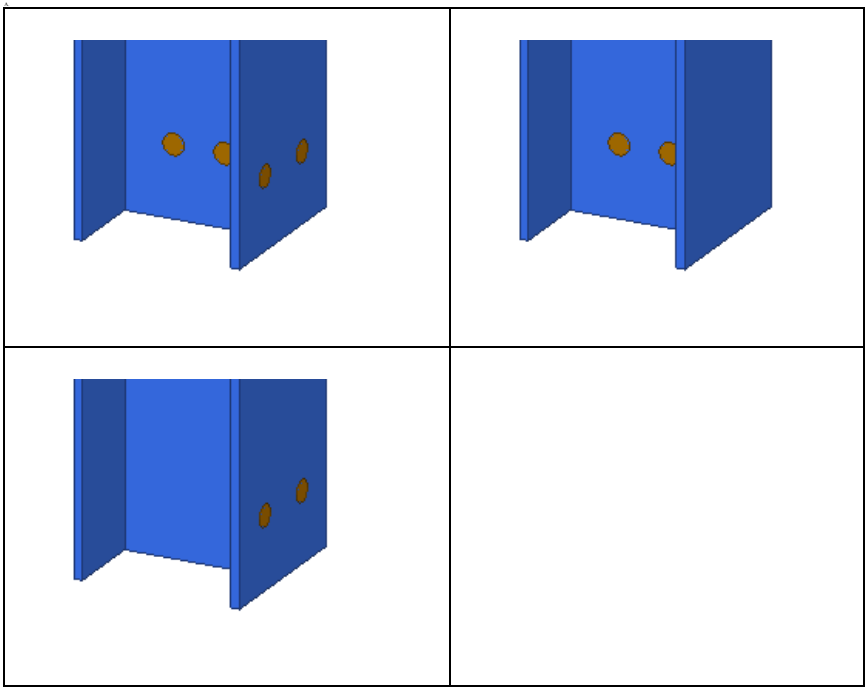
End plate (1002)



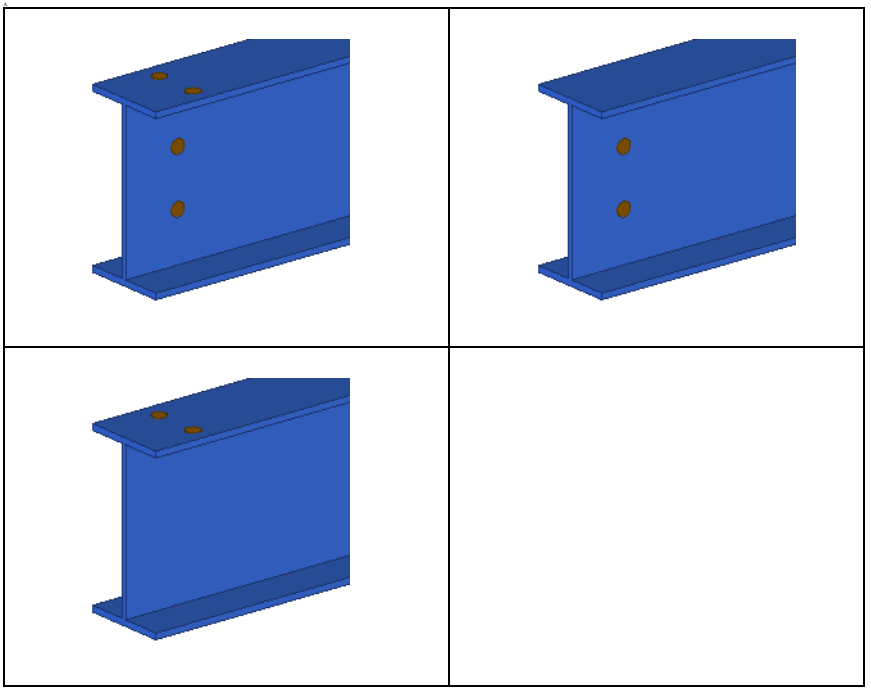
Stanchion weld (85)



Manlock column (1032)

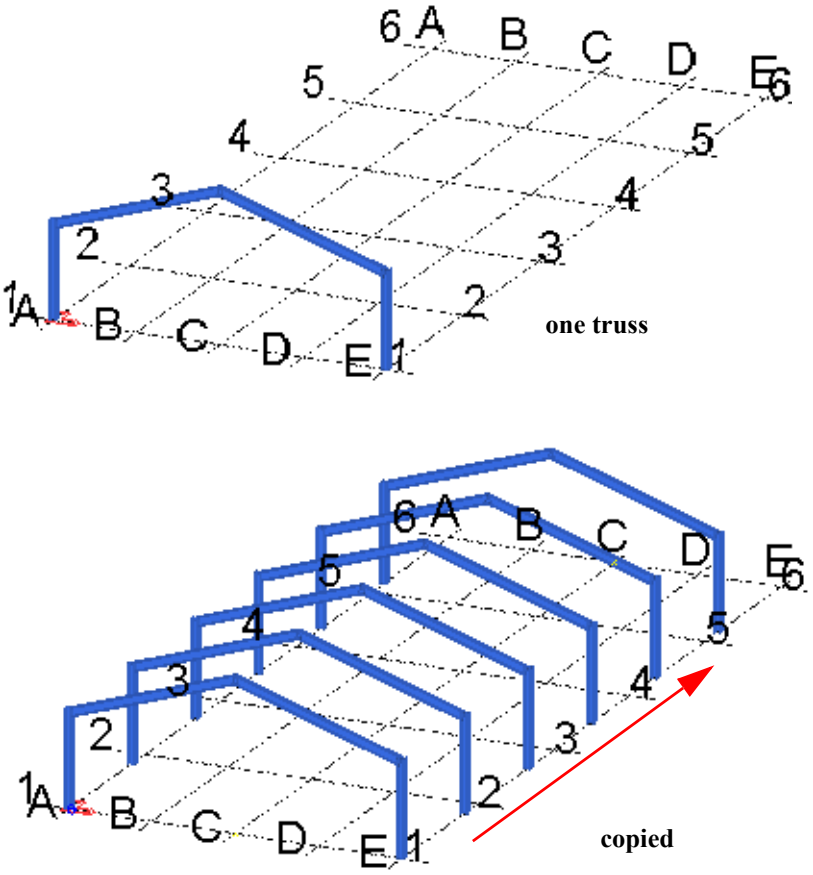


Manlock beam (1033)

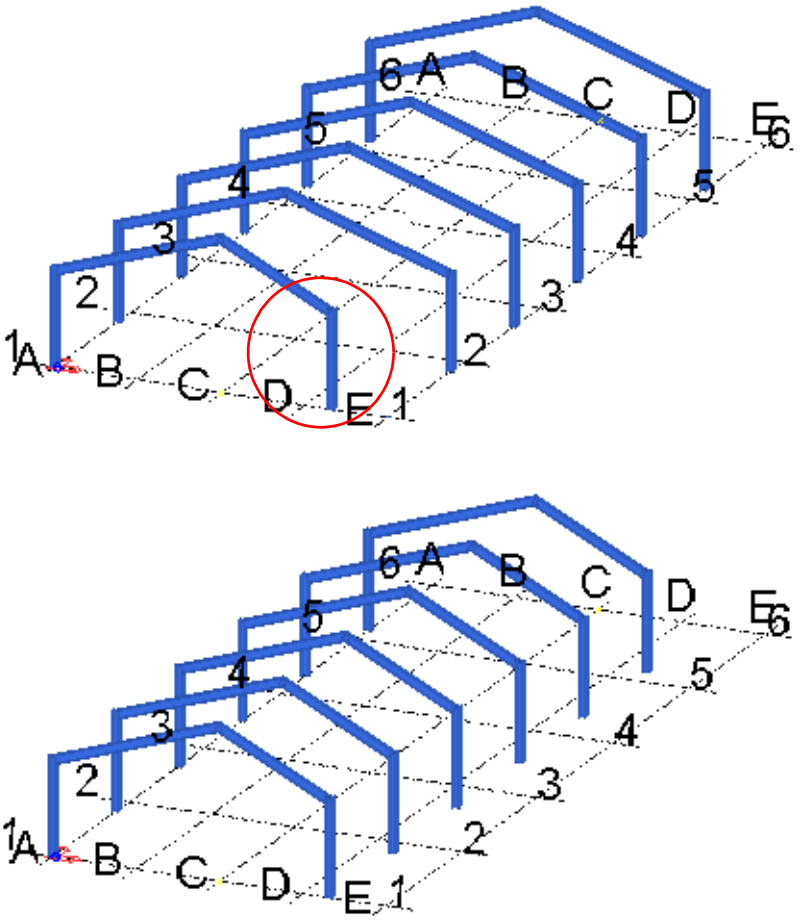


Array of objects (29)

Use this component to easily copy model objects.



If you modify the original objects, a column for example, Tekla Structures also changes the copied objects.



1.46 Custom Component examples

Various custom components are available. How they work, importing them, creating and adding parameters, please read the training course Custom Components. This course contains several examples.

1.47 Reserved shortcuts in Tekla Structures:

	Command		Command
a	Customizing toolbars	**m**	Move > Translate
c	Copy > Translate	**n**	Numbering > Full
e	Open model folder	**o**	Ortho
f	Phasemanager	**q**	Create clip plane
g	Redraw all	**s**	Smart Select
h	Rollover highlight	**t**	Xsnap
i	Inquire assembly	**u**	Update all
k	Inquire object	**v**	Pick rotation point
l	Drawing list	**w**	Wizard

	Command		Command
1	Create single-part drawing	**3**	Create GA drawing
2	Create assembly drawing	**4**	Create cast unit drawing

	Command		Command
Enter	Repeat last command	**Alt + p**	Print
Alt + Enter	Properties	**Ctrl + c**	Undo last polygon pick
Home	Zoom original	**Space bar**	Finish polygon input

	Command		Command
Ctrl + b	Named views	**Ctrl + m**	Move
Ctrl + c	Copy multi	**Ctrl + p**	2D / 3D
Ctrl + f	Find a component	**Ctrl + w**	Weld

	Command		Command
Ctrl + a	Select all objects	**Shift + k**	Column properties
Shift + b	Bolt	**Shift + l**	Beam properties
Shift + c	Copy > With 3 points	**Shift + m**	Move > With 3 points
Shift + f	Fitting	**Shift + z**	Line cut
Shift + h	Construction line		

Drawing Editor

	Command		Command
a	Customizing toolbars	**l**	Drawing list
b	Black and white drawing	**m**	Move objects
c	Copy objects many times	**o**	Ortho
e	Open model folder	**s**	Smart Select
f	Phasemanager	**t**	Xsnap
j	Close drawing	**u**	Update all
o	Inquire > Object	**z**	Zoom in

	Command
Ctrl + b	Open named view list
Alt + Enter	Properties
Home	Zoom original
Enter	Repeat last command
Ctrl + Page Up	Open previous drawing
Ctrl + Page Down	Open next drawing
Shift + A	Associative symbol

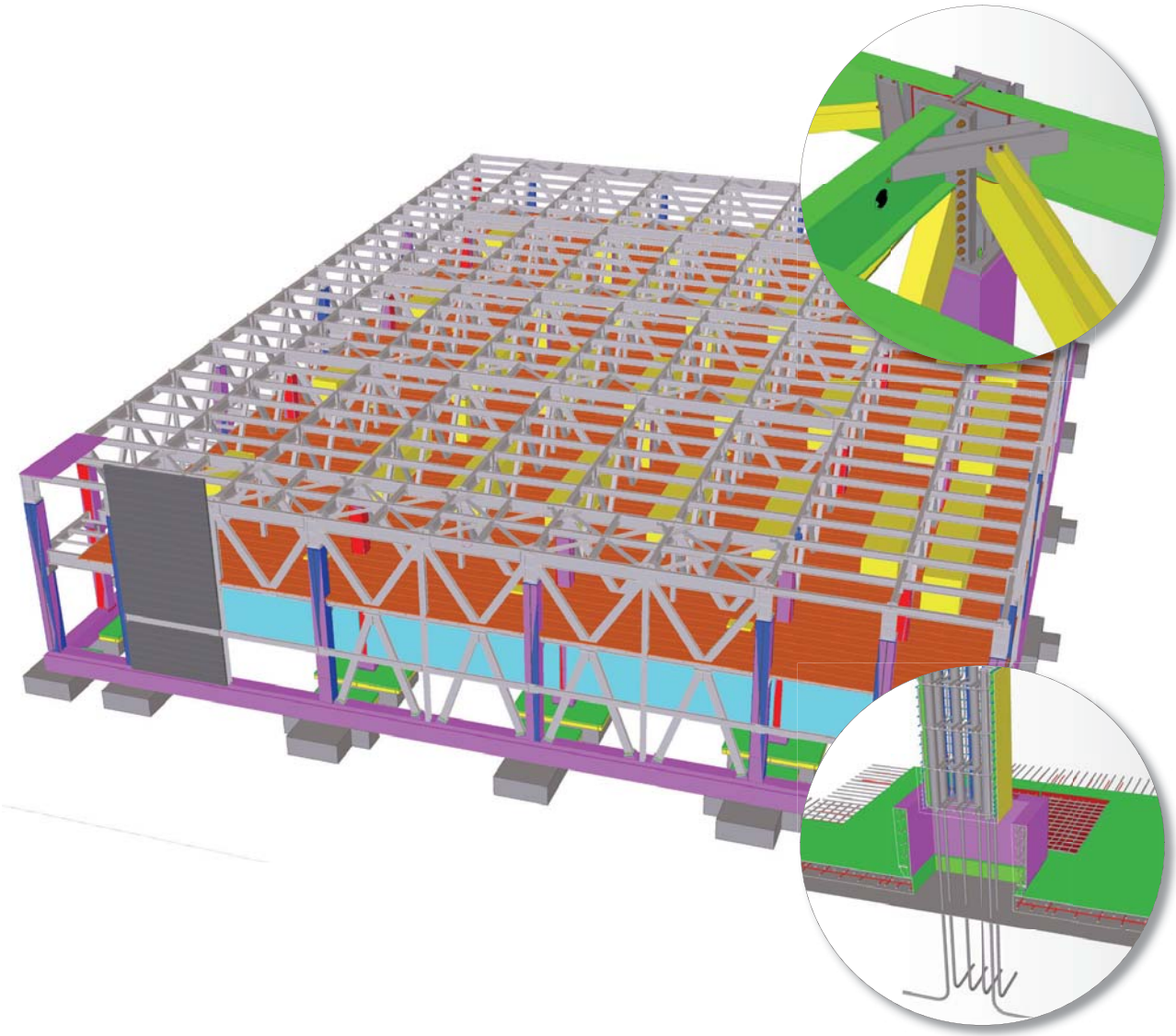
1.48 Notes:

construsoft

TEKLA Structures

Full Detailing

Basic training Drawings



Productversion 13.1

May 2008

All rights reserved. No part of the contents of this manual may be reproduced or transmitted in any form or by any means wihout the permission of Construsoft Inc.

Construsoft Inc. is not responsible for any consequences as a result of using Tekla Structures.

© 2008 Tekla Corporation and its licensors. All rights reserved.

This Software Manual has been developed for use with the referenced Software. Use of the Software, and use of this Software Manual are governed by a License Agreement. Among other provisions, the License Agreement sets certain warranties for the Software and this Manual, disclaims other warranties, limits recoverable damages, defines permitted uses of the Software, and determines whether you are an authorized user of the Software. Please refer to the License Agreement for important obligations and applicable limitations and restrictions on your rights.

In addition, this Software Manual is protected by copyright law and by international treaties. Unauthorized reproduction, display, modification, or distribution of this Manual, or any portion of it, may result in severe civil and criminal penalties, and will be prosecuted to the full extent permitted by law.

Tekla, Tekla Structures, Xcity, Xengineer, Xpipe, Xpower, Xsteel, and Xstreet are either registered trademarks or trademarks of Tekla Corporation in the European Union, the United States, and/or other countries. Other product and company names mentioned in this Manual are or may be trademarks of their respective owners. By referring to a third-party product or brand, Tekla does not intend to suggest an affiliation with or endorsement by such third party and disclaims any such affiliation or endorsement, except where otherwise expressly stated.

Portions of this software:

2D DCM © 1989-2004 D-Cubed Limited. All rights reserved.

EPM toolkit © 1995-2004 EPM Technology a.s., Oslo, Norway. All rights reserved.

XML parser © 1999 The Apache Software Foundation. All rights reserved.

Project Data Control Library © 2006 - 2007 DlhSoft. All rights reserved.

DWGdirect, DGNdirect and OpenDWG Toolkit/Viewkit libraries © 1998-2005 Open Design Alliance. All rights reserved.

Analysis Engine included in Tekla Structures uses a program copyrighted by and is the property of Research Engineers International ©. All rights reserved.

FLEXnet Copyright © 2006 Macrovision Corporation and/or Macrovision Europe Ltd. All Rights Reserved. This product contains proprietary and confidential technology provided by and owned by Macrovision Europe Ltd., UK, and Macrovision Corporation of Santa Clara, California, U.S.A. Any use, copying, publication, distribution, display, modification, or transmission of such technology in whole or in part in any form or by any means without the prior express written permission of Macrovision Europe Ltd. and Macrovision Corporation is strictly prohibited. Except where expressly provided by Macrovision Europe Ltd. and Macrovision Corporation in writing, possession of this technology shall not be construed to confer any license or rights under any of Macrovision Europe Ltd. and Macrovision Corporation's intellectual property rights, whether by estoppel, implication, or otherwise.

Elements of the software described in this Manual may be the subject of pending patent applications in the European Union and/or other countries.

2 Tekla Structures 13.1 Drawings 1

2.1 Introduction 1
2.2 Numbering parts 3
Compare assemblies / parts 5
Assign numbers 5
Family numbering 8
2.3 Step plan for creating a General Arrangement drawing 11
GA-drawing with beforehand selected model view(s) 11
Copy model view to the GA-drawing 13
Move views 16
Three levels of editing drawings 18
Representing bolts on GA drawings 18
Part marks on GA drawings 19
Associative notes 21
Creating a detail view 22
Move section- or detail views from one to another drawing 23
Foundation plan 26
Modifying drawing grids 37
Only grid line marks visible 38
Editing grid line marks 39
Dimensioning GA-drawings 40
Add/remove dimension points 44
Fixed dimensions 45
Dimension line tags 46
2.4 Save drawings 50
2.5 Step plan for generating Single part drawings 51
2.6 Step plan for generating Assembly drawings 54
2.7 Step plan for generating Cast unit drawings 57
Option Cast unit definition method 60
Option Sheet number 61
Surface treatment and surface treatment mark 62
2.8 Adding graphical objects 64
Add a rectangle on a random location 64
Adding a rectangle using fixed coordinates 65

Adding text 66
Adding a text file 68
Adding a symbol 70
Inserting a DWG/DXF file 72
2.9 Lengthen, shorten, split and divide graphical objects 74
Cutting 74
Extending 74
Splitting 75
Divide 75
2.10 Step plan for creating a Multi drawing 76
Add views to an existing multi drawing 77
Auto-sizing multi drawings 78
2.11 Storing Drawing settings 79
2.12 Applying a new drawing setting to an existing drawing 80
2.13 Wizard 81
Explanation of lines 83
Summary creating a new Wizard 84
2.14 Drawing list 85
Selecting drawings 86
Selecting drawings from predefined sets 86
Searching and using search groups 88
2.15 Drawing revisions 92
2.16 Associative drawings 94
Example assembly drawing 95
Example General arrangement drawing 96
2.17 How to handle model modifications 97
Associativity regarding to ID-numbers 101
Description of the status symbols and -text in the drawing list 102
2.18 Cloning drawings 103
Clone from drawing 104
Clone from Drawing Template 105
2.19 Dimensioning curved beams 106
2.20 Hatching 107
Hatching automatically 107
Hatching manually 108

Insulation 112
Accentuate cross-sections 113
2.21 Printing drawings 114
Printing non up-to-date drawings 117
Default Line thicknesses 117
2.22 Layout and Table layout 119
Setup the Layout and table layout 119
Modifying the layout 121
Use dwg/dxf files as a template 124
DWG/DXF 126
Table Layout when using Auto Sizing 127
2.23 Editing templates 128
2.24 Reports 129
2.25 Printing reports 131
2.26 Saw cut angles 132
2.27 Cannot open a drawing anymore 132
2.28 Classifier 133
Classifier principle 133
“Phase” properties 135
“Profile” properties 138
“Material” properties 142
"Supplier" properties 146
Exercise 154
Classifier for assembly- and cast unit drawings 157
2.29 Original drawing 161
2.30 Explanation of file extensions 164
2.31 Appendix 165
Concrete related reports 165
Concrete related templates 177
Steel related reports 181
Steel related templates 205
2.32 Notes: 213

2 Tekla Structures 13.1 Drawings

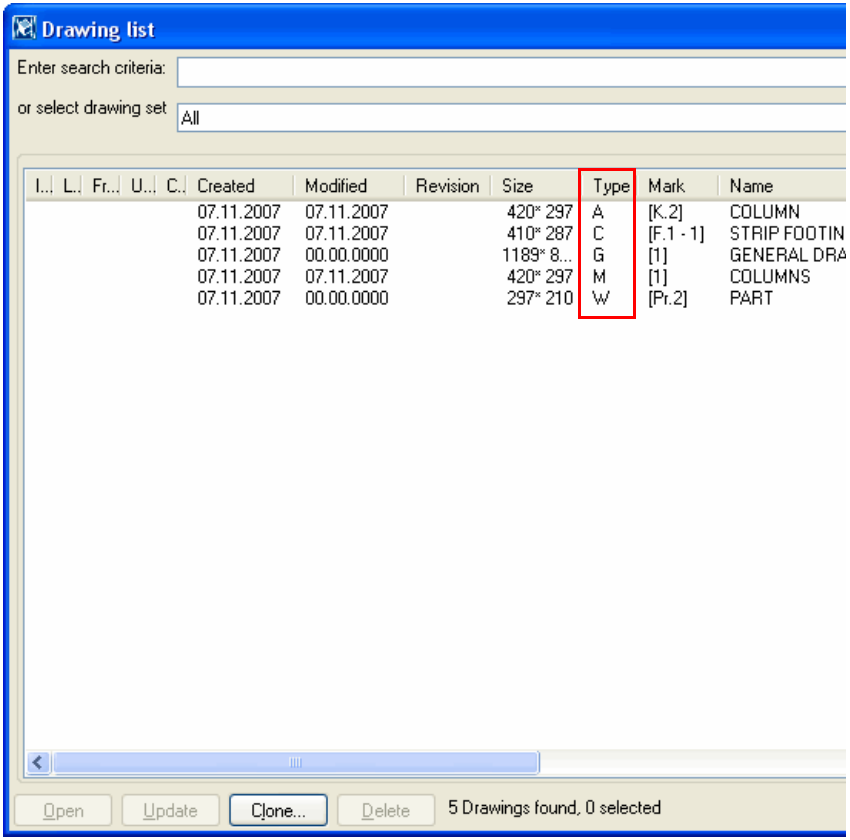
2.1 Introduction

There are 5 drawing types that can be generated by Tekla Structures:

- single part drawings, per part (W = workshop)
- assembly drawings (A = assembly)
- general arrangement drawings (G = general)
- multi drawings (M = multi)
- cast unit drawings (C = cast unit)

This depends on the module that you use:

	Steel Detailing (STD)	**Standard design (SDE)**	**Precast concrete detailing (PCD)**	**Full Detailing (FUD)**	**Reinforced Concrete Detailing (RCD)**
Single part drawings	*			*	
Assembly drawings	*			*	
General arr. drawings	*	*	*	*	*
Multi drawings	*		*	*	
Cast unit drawings			*	*	


Drawing list
Enter search criteria:
or select drawing set
All
I... L.. Fr... U... C.. Created Modified Revision Size Type Mark Name
07.11.2007 07.11.2007 420* 297 A [K.2] COLUMN
07.11.2007 07.11.2007 410* 287 C [F.1 - 1] STRIP FOOTIN
07.11.2007 00.00.0000 1189* 8... G [1] GENERAL DRA
07.11.2007 07.11.2007 420* 297 M [1] COLUMNS
07.11.2007 00.00.0000 297* 210 W [Pr.2] PART
Open
Update
Clone...
Delete
5 Drawings found, 0 selected

2.2 Numbering parts



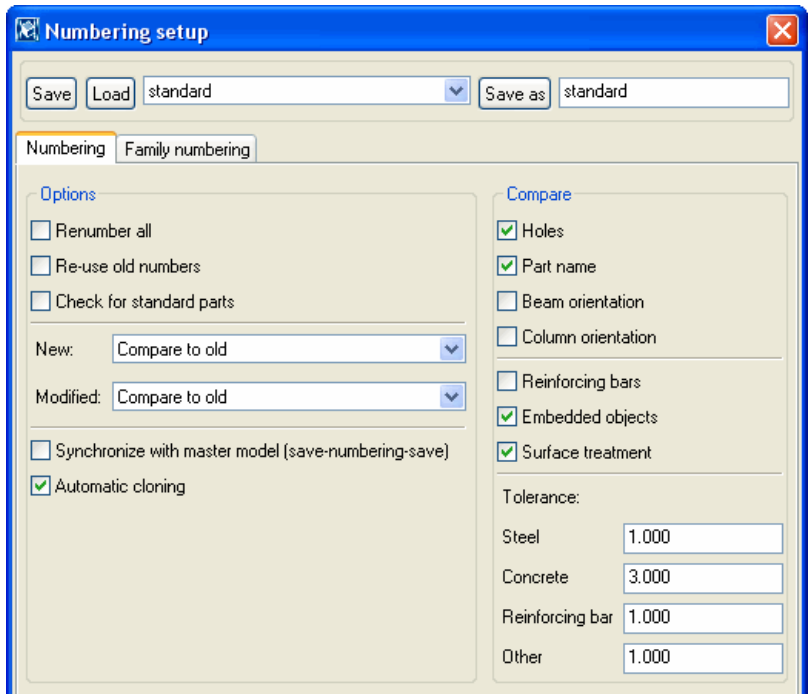
Before drawings can be made, all parts in the model have to get numbers.

Go to **Tools** > **Numbering**, there is the option between **Full** or **Modified**

- **Full***:* Numbers are allocated to all parts and assemblies
- **Modified***:* Numbers are allocated to all modified parts and assemblies in the model. Previous numbered parts and assemblies which are not modified keep their numbers!!

How the numbering is carried out can be set in the dialog box **Numbering Setup**. To open the dialog box, go to **Setup** > **Numbering**.

The image below appears.





Using the default setting is effective is most cases.

If numbering is carried out with **Numbering Full** or **Numbering Modified**, Tekla Structures checks all parts for modifications:

- Existing parts remain their number (if unmodified)

- New added parts are compared with existing parts:
  - New parts which are equal to existing parts receive the same existing number
  - New parts which do not already have a number, receive a new number

Renumber all	All parts receive a new number
Re-use old numbers	Numbers which are not in use anymore (due to model modifications), are re-used for new parts
Check for standard parts	With this option, it is possible to make that a certain part (e.g. a base plate) gets the same number in every project. See also document “Use of Standard Parts”
New	Numbers of new parts are compared with numbers of existing parts
Modified	When modifications have been done, compare numbers with the numbers of existing parts
Holes	Defines if amount and size of holes may affect the numbering
Part name	Defines if part name may affect the numbering
Beam orientation	Defines if the orientation of equal beams may affect the numbering
Column orientation	Defines if the orientation of equal columns may affect the numbering
Tolerance	Part can get the same number if they do not differ more than as set in this input field.

It is recommended to use **Renumber all** once before generating drawings. This to avoid gaps in the numbering.

Size	Type	Mark	Name
420* 297	C	[C.3 - 1]	CASTUNIT
420* 297	C	[C.4 - 1]	CASTUNIT
420* 297	C	[C.5 - 1]	CASTUNIT
420* 297	C	[C.6 - 1]	CASTUNIT
420* 297	C	[C.7 - 1]	CASTUNIT

Without **Renumber all**

Size	Type	Mark	Name
420* 297	C	[C.1 - 1]	CASTUNIT
420* 297	C	[C.2 - 1]	CASTUNIT
420* 297	C	[C.3 - 1]	CASTUNIT
420* 297	C	[C.4 - 1]	CASTUNIT
420* 297	C	[C.5 - 1]	CASTUNIT

With **Renumber all**

Compare assemblies / parts



With this command, two assemblies (or parts) can be compared to each other.

Select two parts in the model which you want to compare. Next, click **Tools** > **Compare** > **Parts**.



Select two parts of two different assemblies to compare two different assemblies. Next, click **Tools** > **Compare** > **Assemblies**.

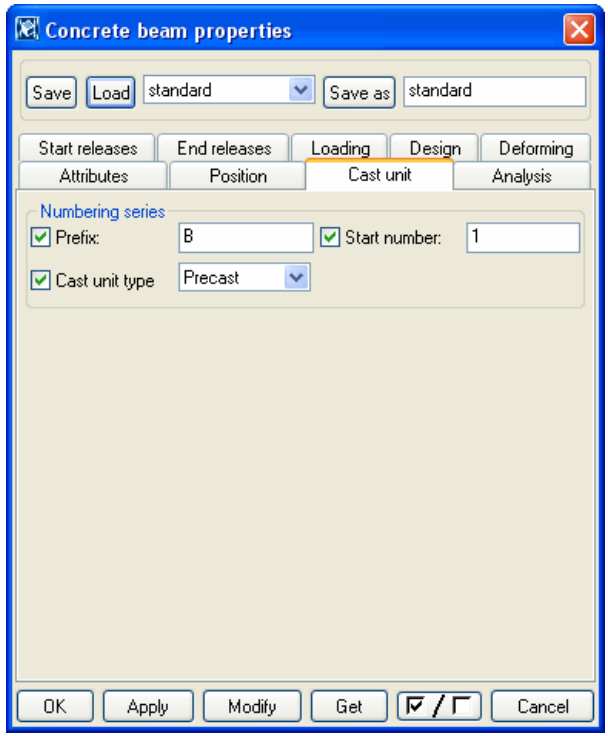
The results are displayed on the statusbar.

Assign numbers

In Tekla Structures you can assign the **part number** and/or the **assembly number**. This, because for example customers, require specific numbers.

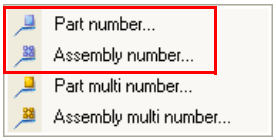
Example

You want to assign an assembly number to a concrete beam, for instance B400. This concrete beam includes the following properties:

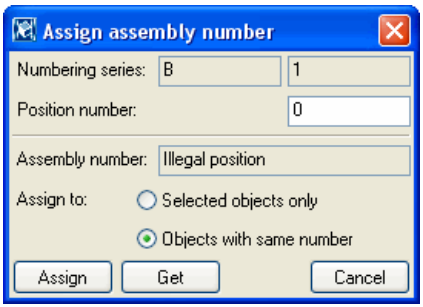


Step plan

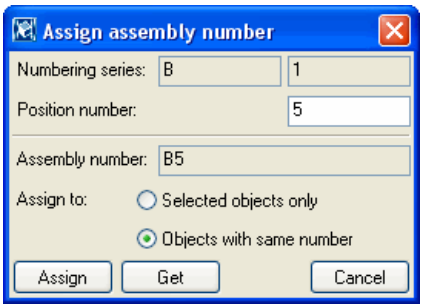
1. Select a concrete beamGo to **Tools** > **Numbering** > **Assign number** and click one of these options, in this case click Assembly number:



2. The following dialog box appears:

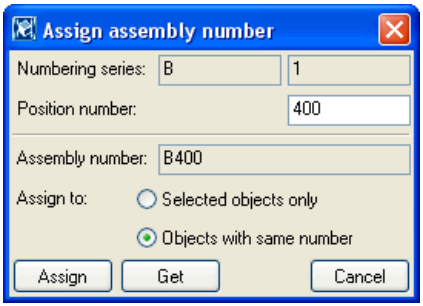


Model not numbered

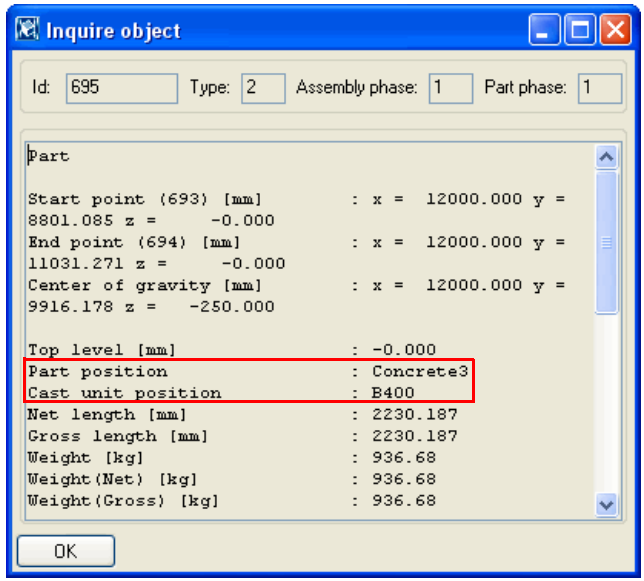


Model numbered

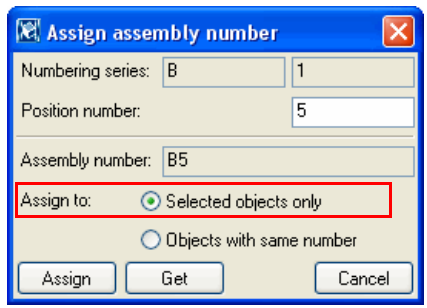
3. Enter you number you want to assign and click <**Assign**>:



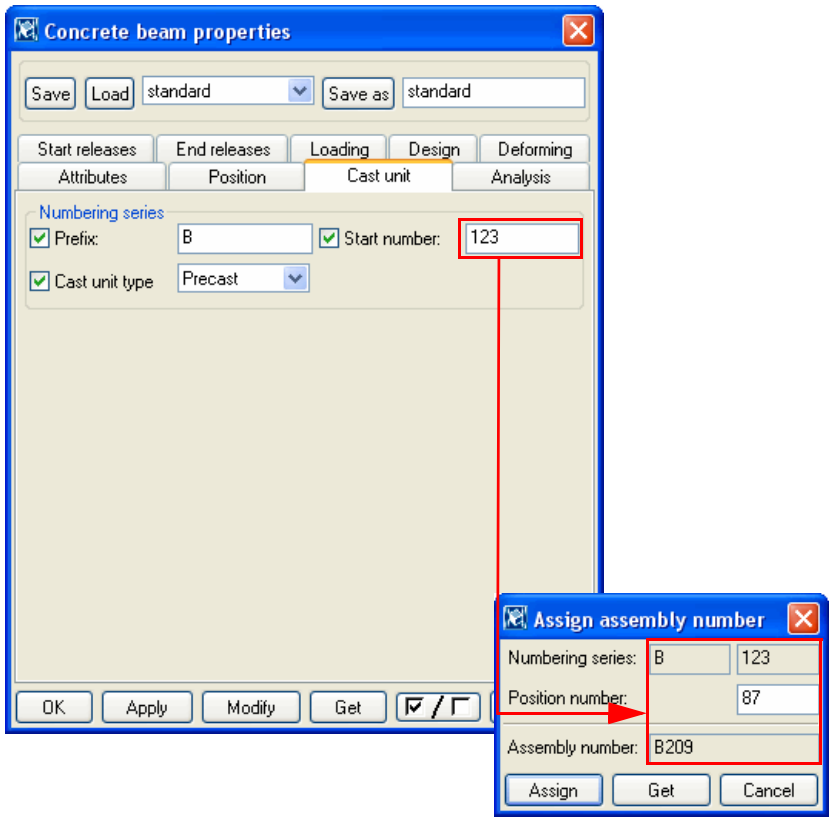
4. Go to **Inquire** > **Object** and select the concrete beam, you will se that the number has been assigned, it will be assigned to all identical concrete beams automatically:



You can also change the assembly number of any one part without affecting others, even if they are similar to the part, whose number you are changing. To do this, select the **Selected objects only** radio button in the dialog box:



Warning When you assign numbers by yourself, the start number always must be set to 1. If not, the start number and the position number you assign will be added up and decreased by 1, see the following example:



See also For more information about assigning numbers, see the Tekla Structures Online Help.

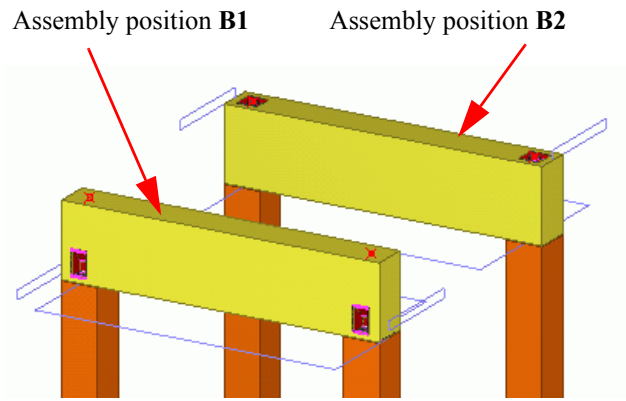
Family numbering

With family numbering you can group objects within the same numbering series into different "families". When you use family numbering, the cast unit position numbers consist of **family number** and **qualifier**, for example **B1 - 1**.

Inhere, the family number is **B1** and the qualifier is **1**.

- The **family number** is the same for assemblies and cast units which match the criteria you define in the **Numbering setup** dialog box.
- Assemblies or cast units which have the same family number but different exact geometry or materials get unique **qualifier** numbers.

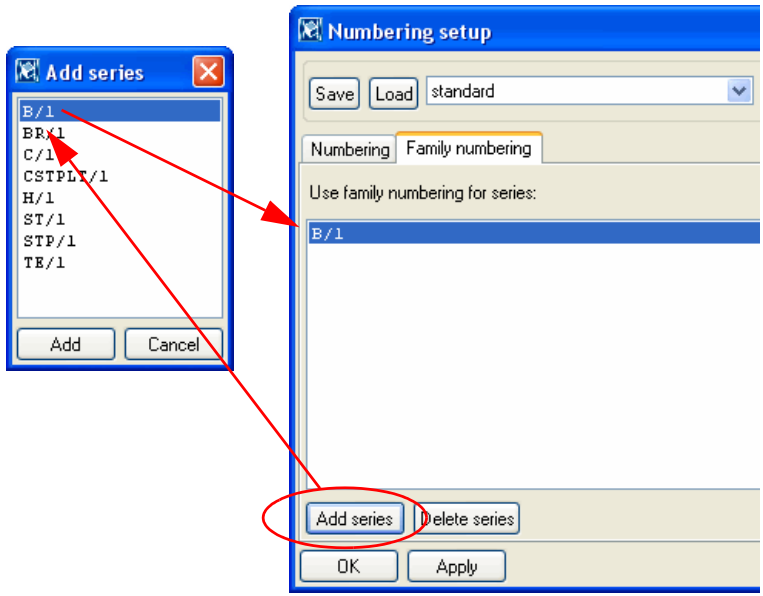
Example Two beams have both numbering series prefix **B**:



The beams are otherwise similar, but the connections used are different.

To assign family numbering for the series:

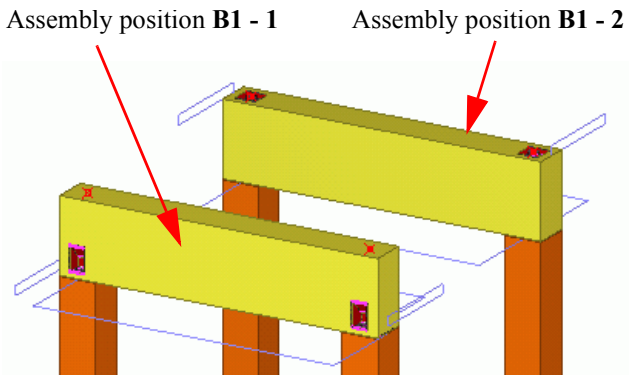
1. Click **Setup** > **Numbering** to open the **Numbering setup** dialog box.
2. On the **Family numbering** tab, click **Add series**, to open the **Add series** dialog box, which lists all the assembly and cast unit numbering series in the model.
3. Select numbering series `B/1` and click **Add**. The numbering series appears in the family numbering field.



4. Use the **Compare** section of the dialog box to define what to compare for the numbering series.Define the compare criteria for each numbering series separately.

5. Click <**Apply**> and <**OK**> and update the numbering in the model.
6. Tekla Structures assigns the beams family numbering.

Tekla Structures considers beams different according to the compare criteria you defined, and they get different family numbers:





Do not change the family numbering settings while working in a project. If you change the family numbering settings, you first need to clear the numbering to modify the model (numbering) correct.

2.3 Step plan for creating a General Arrangement drawing

A general arrangement drawing (GA drawing or layout plan) is used for displaying model views, including foundation plans and details of the model. A general arrangement drawing is recognizable in the drawing list with a **G** in the column **Type**. This **G**-character stands for **G**eneral arrangement drawing.

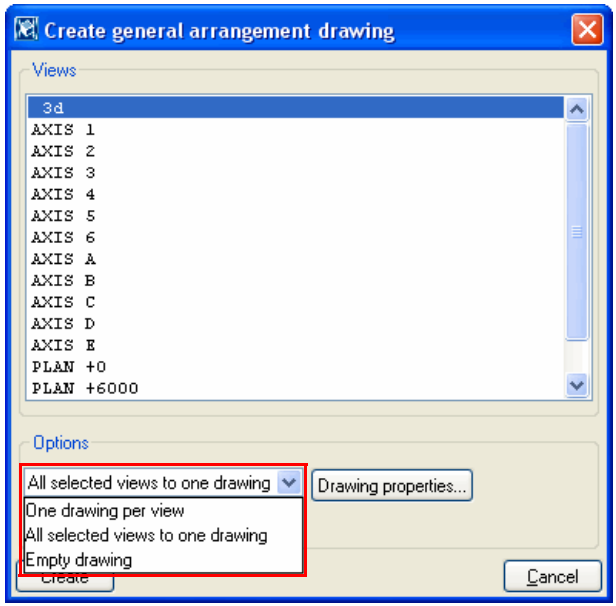
There are two methods of adding model views to a GA-drawing:

- One method is selecting model views before creating the drawing, see **GA-drawing with beforehand selected model view(s)** (p. 11)
- It is also possible to start with an empty drawing and add model views afterwards, see **Copy model view to the GA-drawing** (p. 13)

GA-drawing with beforehand selected model view(s)

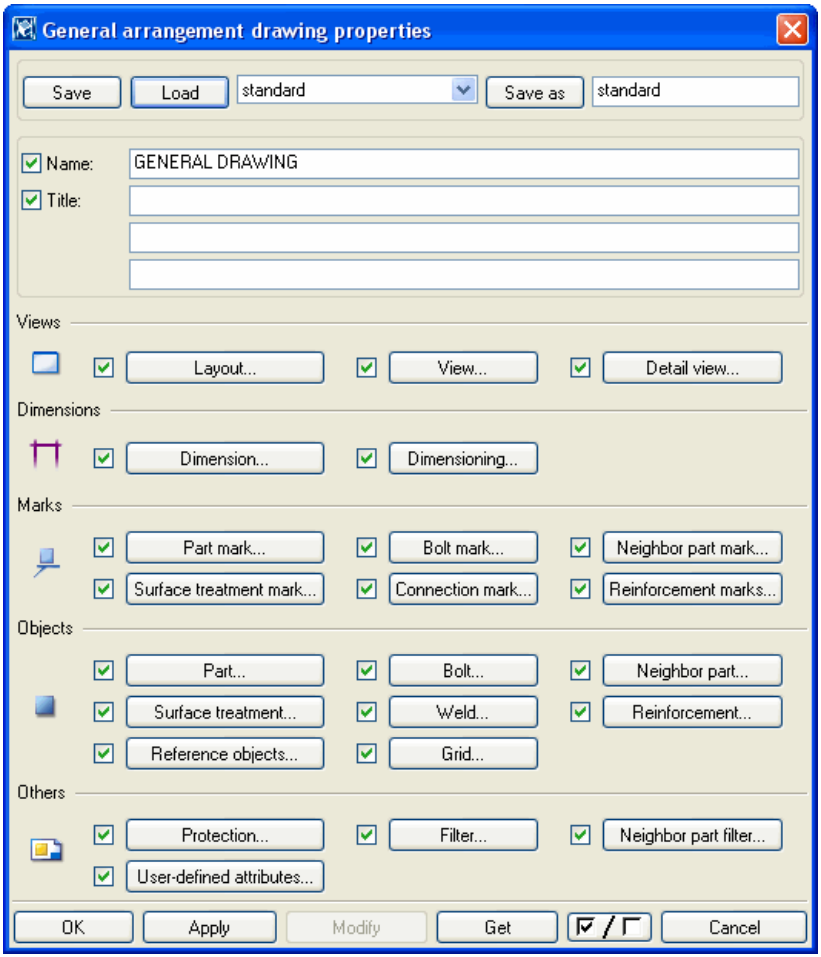


1. In the pull-down menu go to **Drawing** > **General Arrangement drawing...**, click on the icon on the left or click shortcut “3”, the dialog box **Create general arrangement drawing** appears:



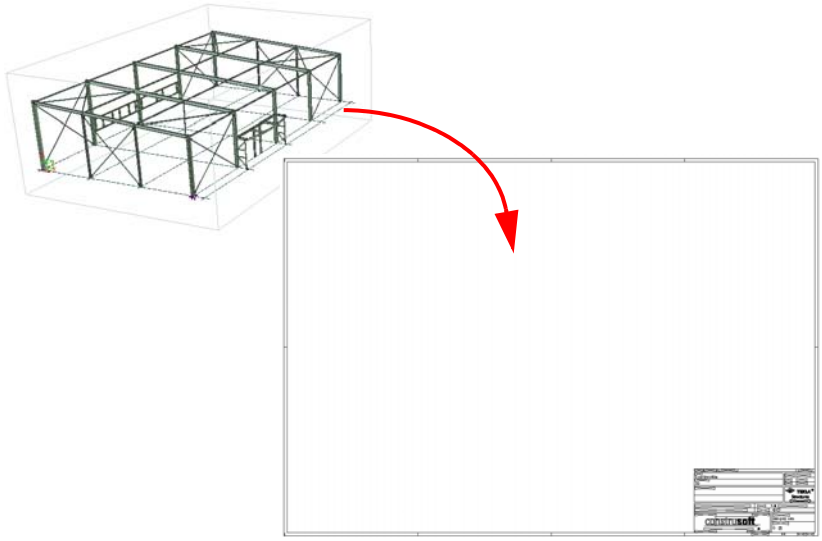
2. In the image above, only a 3D view is selected. In the Options-listbox, choose for **All selected views to one drawing**

3. By pressing the **Drawing properties**-button, the properties of the drawing (e.g. name) can be set (For example, with the **Grid**-button a sub dialog box will be opened where can be set if grids have to be displayed in the view)



4. By checking the option **Open drawing**, the drawing will be opened automatically after the drawing has been created
5. Finally, click <**Create**>
6. Tekla Structures creates the GA drawing and displays it.
7. To change the drawing properties, go to **Properties** > **Drawing...** or double-click on the drawing (not on the blue frame, but outside of it)

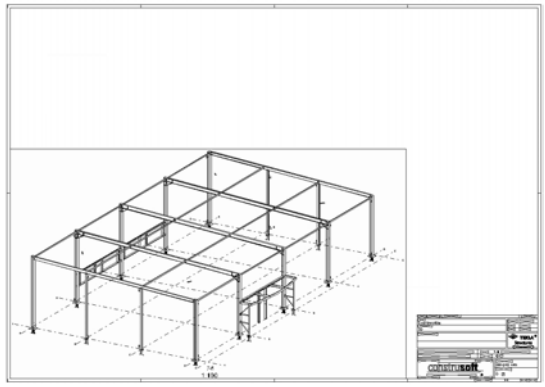
Copy model view to the GA-drawing



1. Open a GA-drawing
2. Position the drawing besides the 3D model view
3. Double-click the icon **Create view from model view**. The dialog **View properties** shall appear



4. Select setting **3D**. Next, press <**Load**>
5. Click <**Apply**> and <**OK**>
6. Select the 3D model view, the following shall appear on the screen:

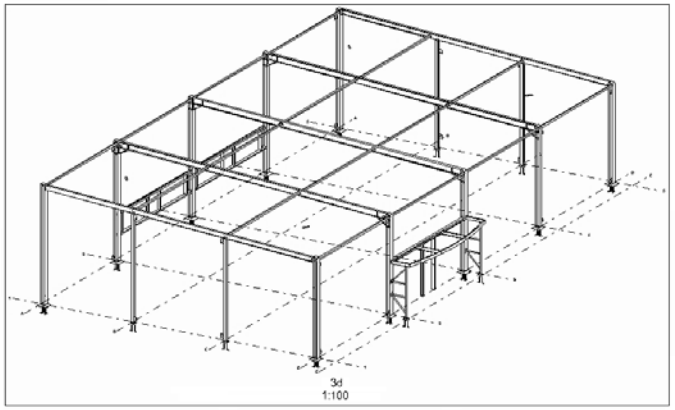




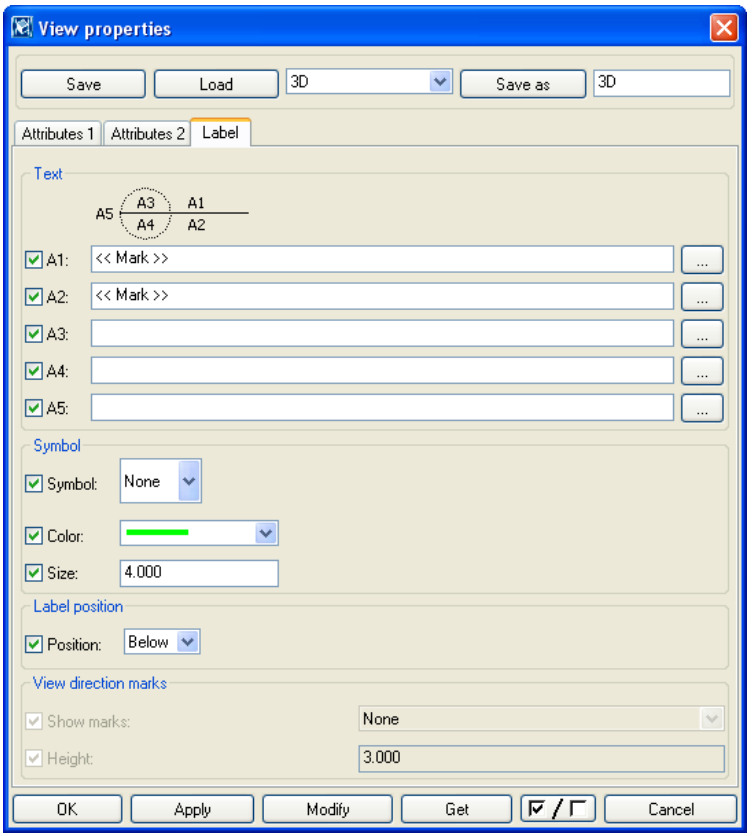
7. In the pull-down menu go to **Edit** > **Place views**, click on the icon on the left or click the right mouse button and select **Place views** so that the view will be aligned centered on the drawing.

General drawing view labels

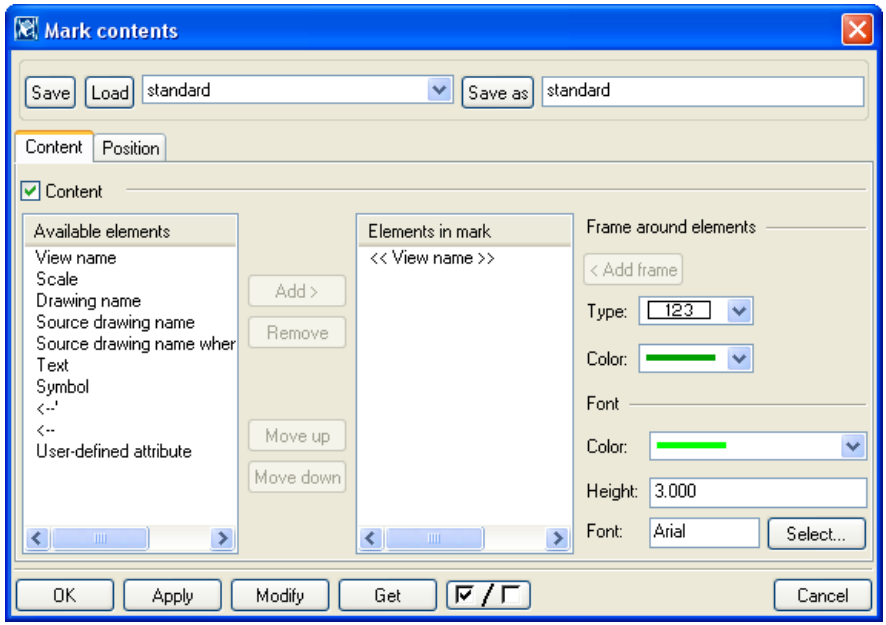
You have several options to control the creation and appearance of drawing view labels and the information they contain. By default, the view name and the view scale appear:



1. Double-click the view in a drawing to open the **View properties** dialog box:



2. Click the **Label** tab, and use it to define the appearance of the view label, to define the contents of the view label, click the **...** buttons next to text fields to open the **Mark contents** dialog box:



3. Click <**Modify**> and <**OK**>

Move views

You have several options to move views:

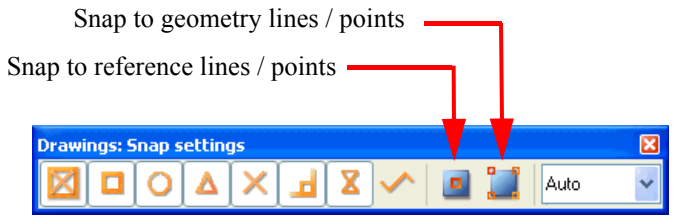
1. You can move views in a drawing, see also **Move a view in a drawing** (p. 16)
2. You can move views from one to another drawing, see also **Move views to another drawing** (p. 17)

Move a view in a drawing

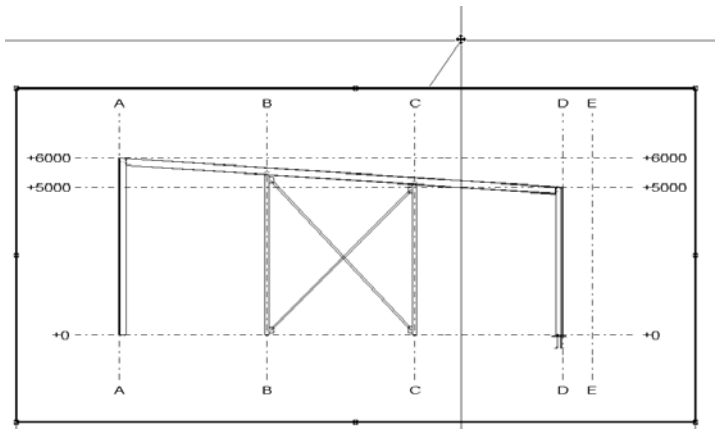
Rough

1. Select the view to be moved
2. Hold down the <**Shift**> key and point the mouse to the blue frame from the view, than hold down the left mouse button
3. Now the view can be moved by moving the mouse
4. If the view is on the desired position, release the left mouse button and release the <**Shift**>-key

To prevent the view from jumping in any arbitrary direction while moving, switch off temporary the icons **Snap to reference lines / points** and **Snap to geometry lines / points** on the toolbar **Drawings: Snap settings**.

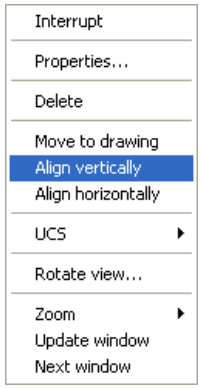


When a view is resized instead of moved, a **handle** highlights, zoom in a little bit more, see following picture:



Exact

1. Select the view to be moved
2. Click the right mouse button once
3. From the context menu, choose **Align vertically** or **Align horizontally**

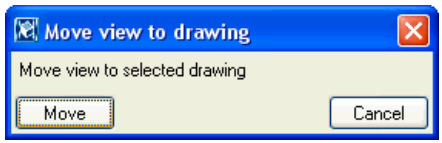


4. Pick two points (e.g. grid intersection)
5. The view will be moved

Move views to another drawing

You can move views to another drawing in the drawing list. To move drawing views to another drawing:

1. In the **Drawings** dialog box, open the drawing.
2. Select the drawing view you want to move, right-click and select **Move to drawing** from the popup menu.
3. Select the target drawing from the drawing list.
4. Click **Move** in the **Move view to drawing** dialog box.



5. Tekla Structures moves the view to the target drawing and creates references between the source and target drawings.

See also

To move section and detail views to another drawing, see also **Move section- or detail views from one to another drawing** (p. 23)

Three levels of editing drawings

You can edit drawings on three levels:

Drawing properties

On the highest level you can change the drawing properties, which changes all the objects in the drawing, for example Assembly drawing properties.



Whenever possible, you should modify drawings by changing the drawing properties. These modifications also remain when the drawing is recreated due to a model change, for example.

View properties

The second level is to edit on the view level, where you modify selected drawing views. This is the blue frame around a view.

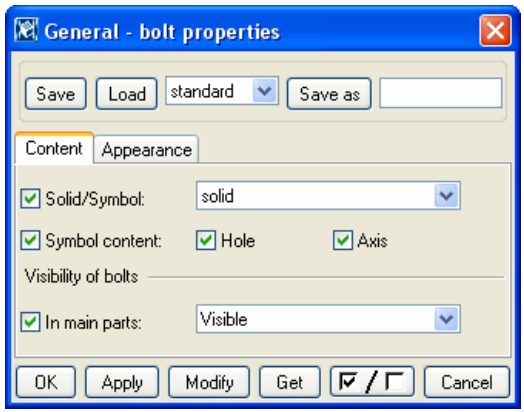
Object properties

The third and lowest level is to edit on the object level, where you change individual objects in a drawing. This means changing single objects like dimensions, symbols, part lines, labels, etc.

Representing bolts on GA drawings

It is recommended on GA drawings, when you use a very small scale (e.g. 1:100), to switch off the bolts.

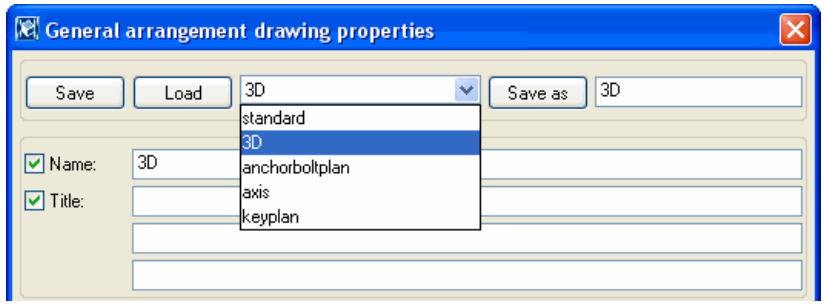
Because of this, GA drawings will open faster and you can work faster in the Drawing Editor.



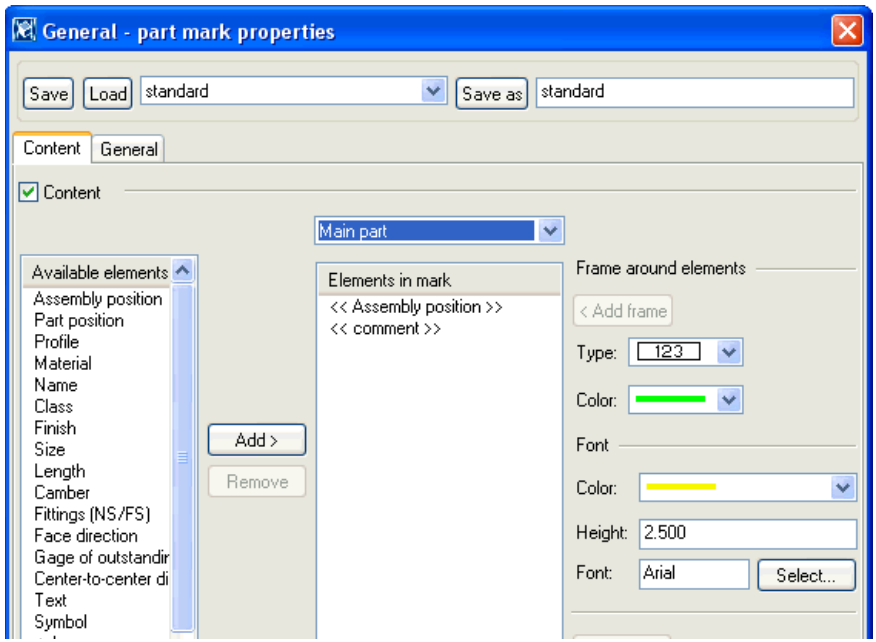
Part marks on GA drawings

Step plan

1. Go to **Drawing** > **General arrangement drawing...** and select the views you want to add on a GA-drawing.
2. Click on **Drawing properties**



3. Load the drawing setting **3D** click the button <**Part mark...**>, the following dialog box appears:



4. For all the main parts, the Assembly position and the comment will be displayed in the part marks, for the secondary parts no marks will be displayed
5. When you do want part marks for the secondary parts, you can make use of associative notes, see **Associative notes** (p. 21)

Dragging the base point of a leader line

You can drag the base point of a leader line in part marks and reinforcement marks:

- Select the part- or reinforcement mark
- Select the base point and drag it to any position by using the left mouse button

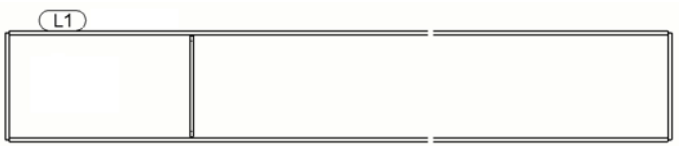
Using part marks as an orientation mark

You can make use of part marks on GA drawings as an orientation mark, they indicate the assembly direction of the parts and the assemblies.

The main advantage is that the position of the labels on both single part- and assembly drawings is always equal to the position on GA drawings.

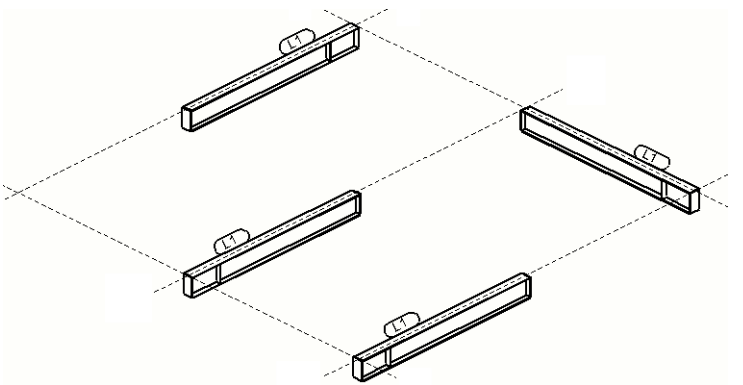
Assembly drawing

The part mark is positioned at the stiffeners' side:



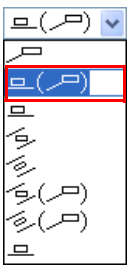
GA drawing

The marks are positioned at the same side, irrespective of the orientation of the beams:



Condition

Make use of the correct option for the position of the labels:



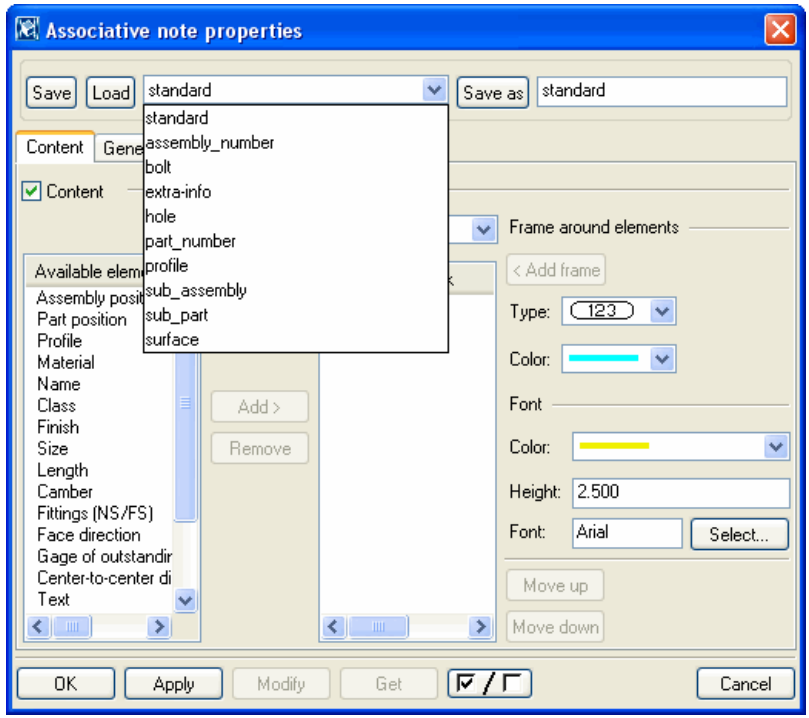
1. Go to **Properties > General arrangement drawing...** and click the button **Part mark..**
2. Select the following type of leader line in the listbox in the tab **General**

Associative notes

You can use associative notes in drawings as an extra mark such as secondary part marks. The notes will be updated automatically when the object changes in the model.

Step plan

1. Open a GA-drawing and go to **Properties** > **Associative note...**



2. Select the setting **part_number** in the list box
3. Click <**Load**> and <**OK**>
4. Click the right mouse button and select **Associative note** or go to **Create** > **Associative note** > **With header line**, **Without header line** or **Along line** in the pull-down menu.
5. Pick the object and the position
6. The associative note is created

Creating a detail view

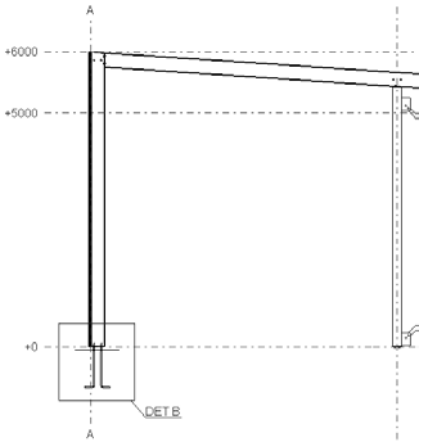
There are several ways of creating a detail view:

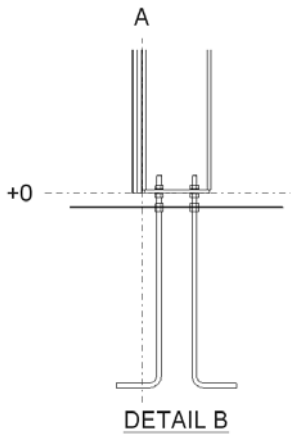
Create a detail view

In the pull-down menu, go to **Create** > **Detail view** or click the icon on the **Drawings: View** toolbar.

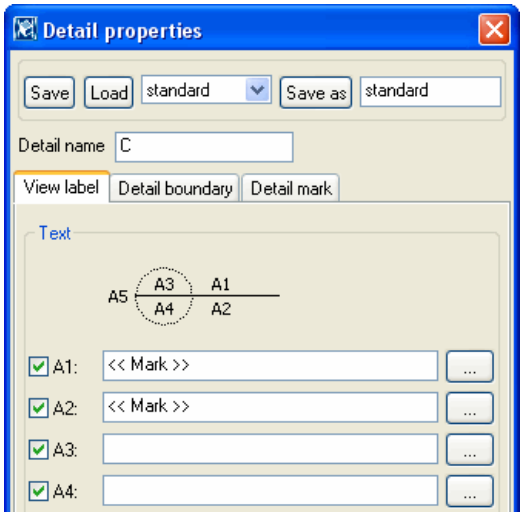


1. Pick the first corner of the detail view.
2. Pick the second corner.
3. Pick a the position of the detail symbol.
4. Tekla Structures creates the detail view and adds a detail symbol to the original view:



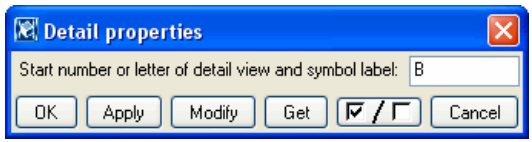


To modify the label properties, double-click the detail symbol:



To modify the detail views' start number or letter:

- In the drawing properties dialog box, click **Detail view..** to open the following dialog box.

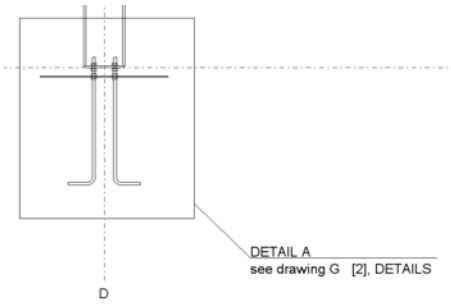


- Modify the properties and click <**Modify>** and <**OK**>.

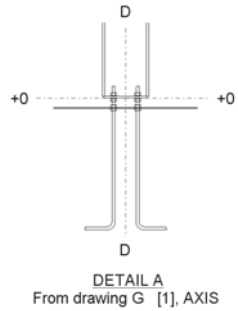
Move section- or detail views from one to another drawing

When you move section views (or detail views) to another drawing, the source drawing and target drawing will contain references to each other:

Drawing G1 contains the detail view:

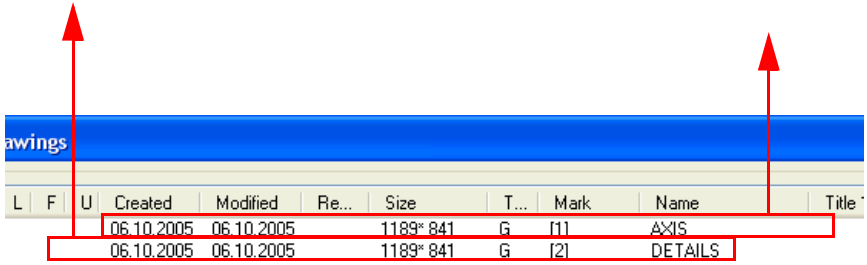


Target drawing G2:



The detail view contains a reference to the target drawing G2

The detail view contains a reference to the source drawing G1



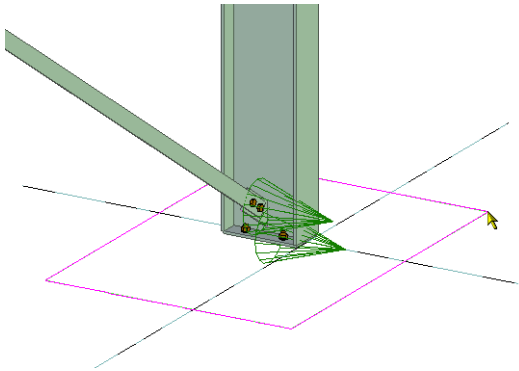
See also To move section- and detail views, see also **Move views** (p. 16)

Create view from model by area

1. Open the drawing where you want to add a detail view to.
2. Double-click on the icon **Create view from model by area.** The dialog box **View properties** appears



3. Select the setting “detail”. Then press <**Load**>
4. Click **<Apply>** and <**OK**>
5. Draw a frame around the bottom of column on grid-intersection A-2 (see image). A small view is created in the down-left side of the drawing.





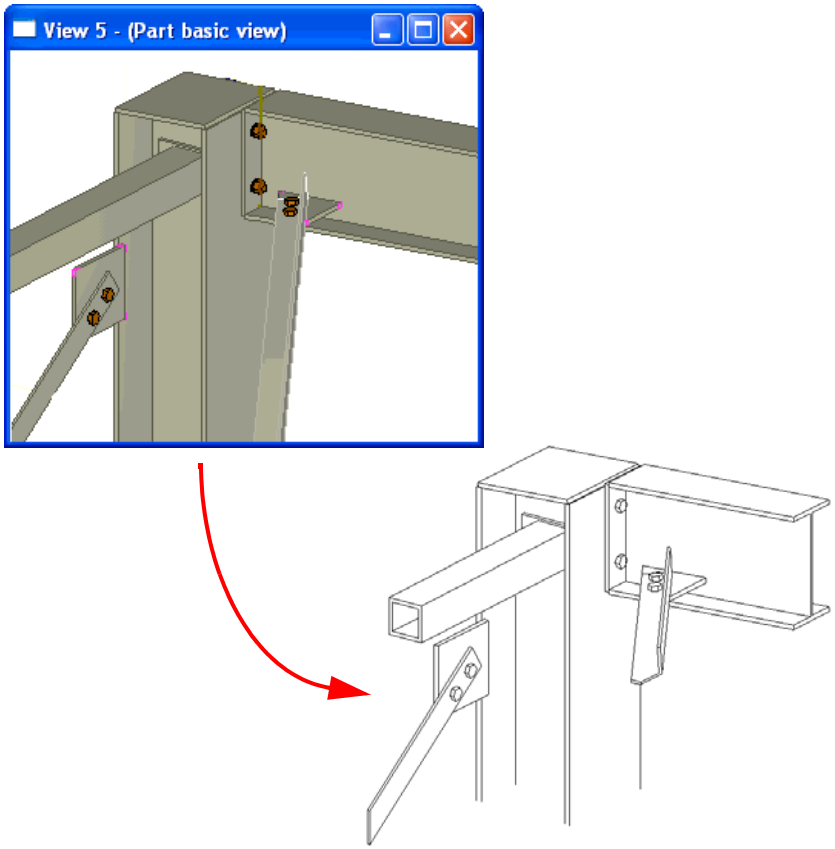
6. In the pull-down menu, go to **Edit** > **Place views**, click on the icon on the left or click the right mouse button and select **Place views**



This option only works properly if the detail is equal to grid level; if a detail has to be made on a higher level, e.g. +5000mm, than it is better to use option #2 (see below)

Create view...

1. In the Model Editor (so close the drawing in case it is still open), select a component symbol, click on the right mouse button and select **Create view > Connection basic view** or **Create View > Part basic View**
2. Open the GA-drawing and double-click icon **Create view from model view**
3. Select setting “detail” and press **<Load>**
4. Click the model view
5. A connection view is placed on the GA-drawing (see picture below)



The advantage of this method is that details can be made from any point of view

Foundation plan

General

There are several ways to generate a foundation plan:

- An anchor bolt plan with only one scale and broken grid lines
- An anchor bolt plan with two combined scales and no broken grid lines, you can also set a separate scale for detail views

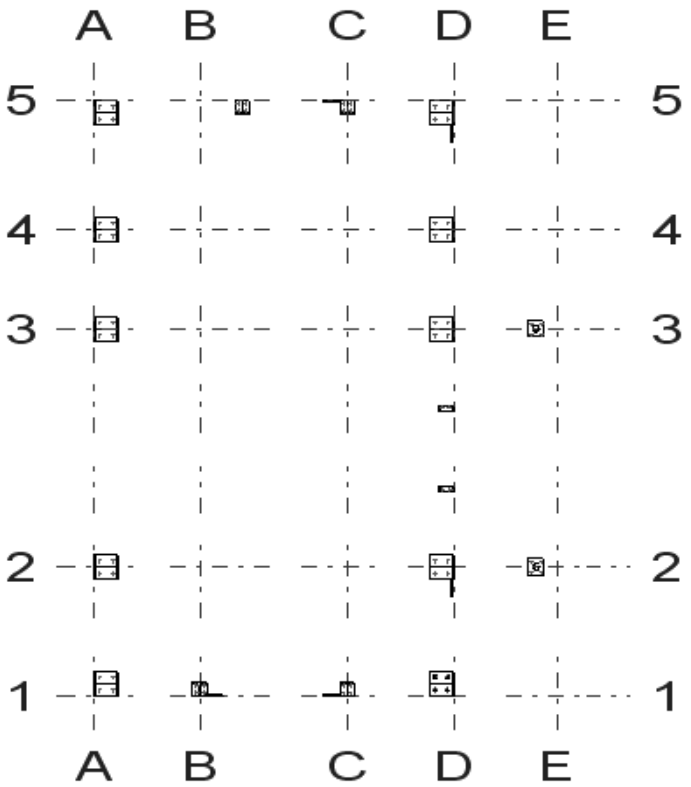
An anchor bolt plan with one scale

Step plan

1. Create a model view on base plate level (perspective off). Set the values for the View depth **Up** and **Down** (10 mm and 100 mm respectively).
2. Go to **Drawing** > **General arrangement drawing** and select a model view
3. Select the desired setting, e.g. **foundation-plan**, check the option **Open drawing** and click **<Create>**
4. The foundation plan drawing is being generated and will be opened afterwards



5. In the pull-down menu, in the Drawing Editor, go to **Edit** > **Place views**, click on the icon on the left or click the right mouse button and select **Place views**

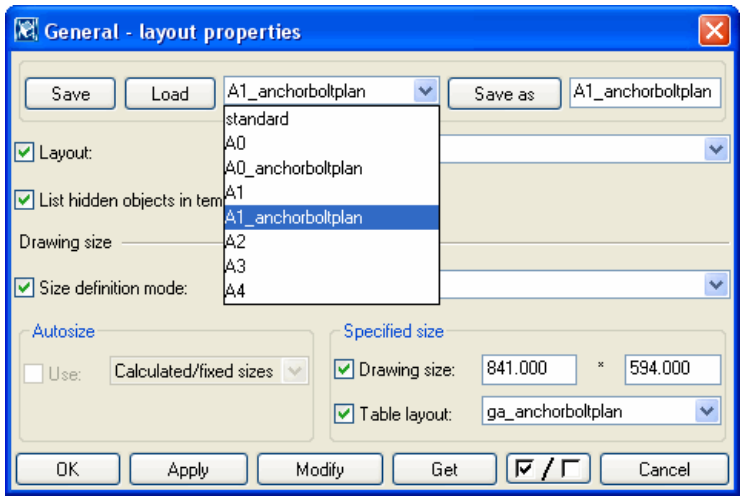


In case the foundation plan view does not fit properly on the drawing, some adjustments can be made:

Drawing size is to small

Increase the drawing size:

- Go to the GA-drawings' properties and click the button <**Layout**>.

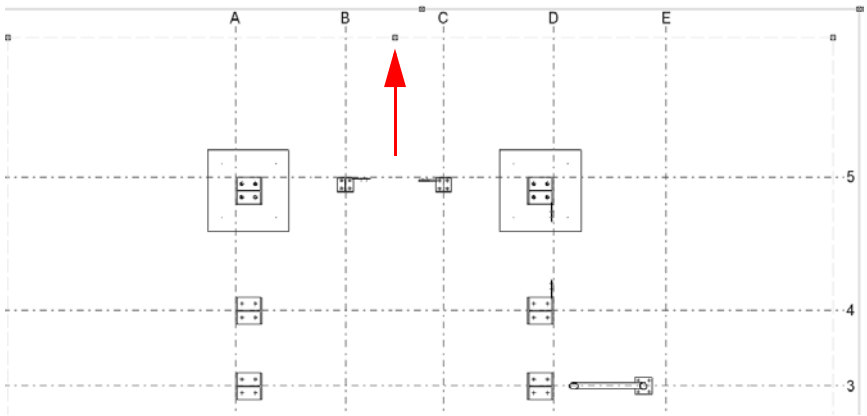


- Select the desired size, next click <**Load**> and <**Modify**>

The line extension for the grid lines is to large

In the view properties, the values for **Xmin**, **Xmax**, **Ymin** and **Ymax** were set to large.

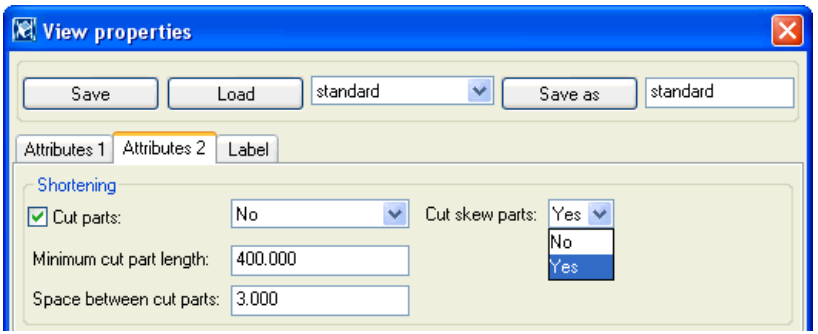
Select the view and drag the grips of the inner view plane (dashed lines) along the x and y axis of the view:



Of course you can also change the values for **Xmin**, **Xmax**, **Ymin** and **Ymax** in the view properties.

Grid lines are not broken on the spot of the skew profiles (windbracings).

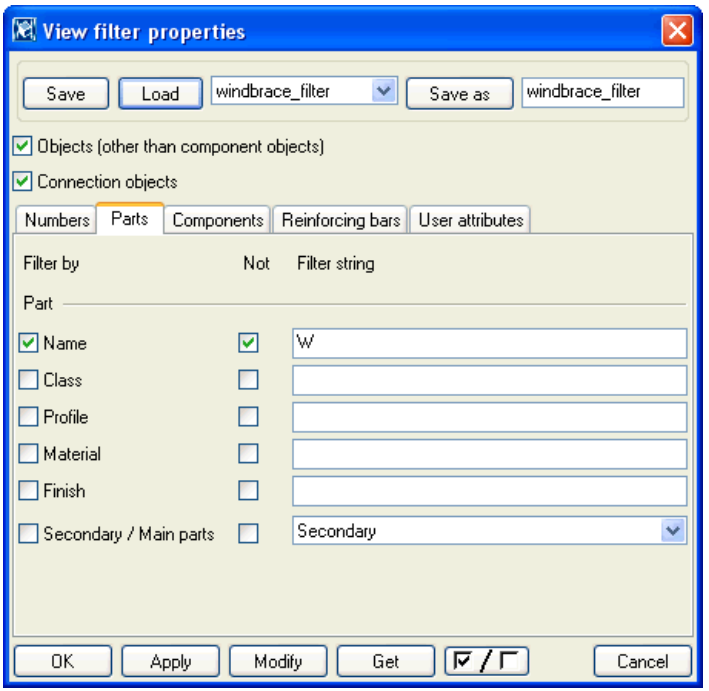
Open the view properties and switch on the option **Cut skew parts** on the tab **Attributes 2**:



Uneven cuts

The windbracings are (partially) displayed, therefore uneven cuts.

Open the view properties, click on the button <**Filter**> and filter the windbracing out of the view, see picture below:

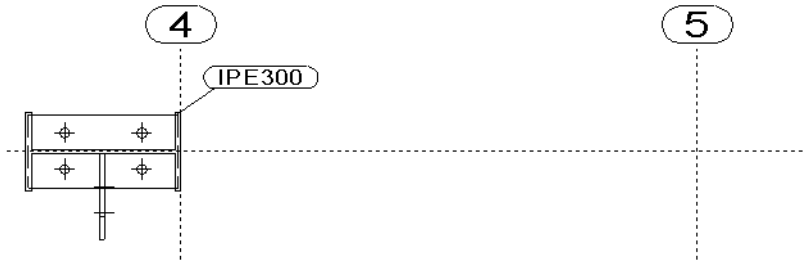


Some grid lines are not displayed on the drawing

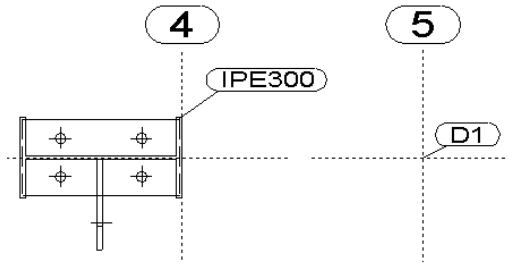
Select the view and drag the grips of the inner view plane or change the values for **Xmin**, **Xmax**, **Ymin** and **Ymax** in the view properties.

Grid is not cut when there is no contact with profiles

On **Axis 5** in the model, no profiles are present. On a foundation plan drawing, these grid lines will not be cut by default. See image below.



To make the grid lines to be cut, a profile has to be placed on it in the model. In the example below, a small profile is placed (Ø1, lg.10mm) so that it will not be conspicuous on the drawing. A part mark will be added to the profile by default, but this mark can be removed easily afterwards.

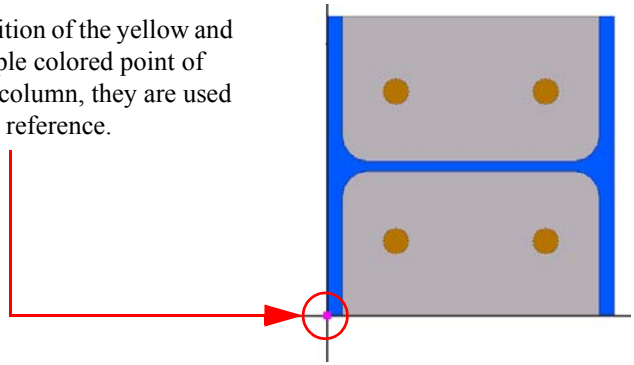


Important Note

In case you create an anchor bolt plan with two combined scales, the part in the view which represents the enlarged part, refers to the part in the original view.

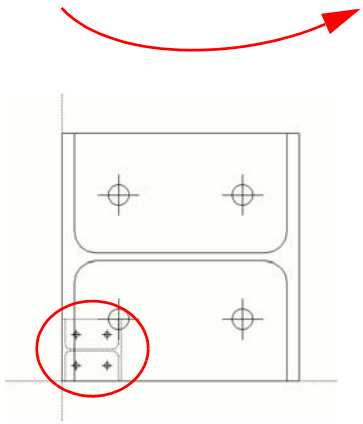
In the model

Position of the yellow and purple colored point of the column, they are used as a reference.



In the drawing

The view which represents the enlarged part is positioned to the original view



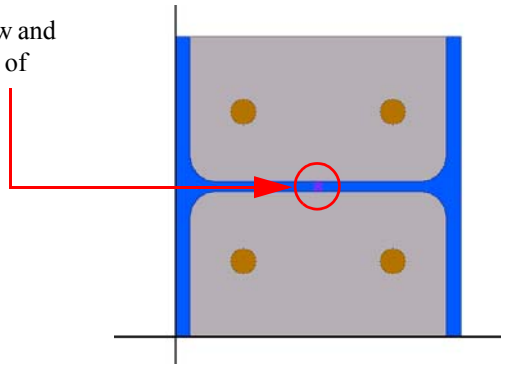
The parts in the original view are hidden

to finally only display the view which includes the enlarged parts.

You will see that column is positioned **correct** to the grid because the yellow and the purple colored point are in the correct position. In case they are positioned for instance in the centre of the column:

In the model

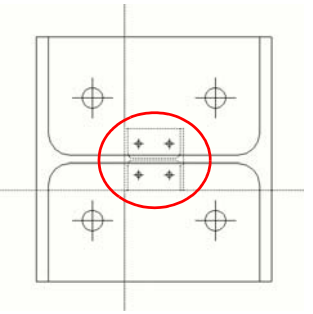
Position of the yellow and purple colored point of the column.



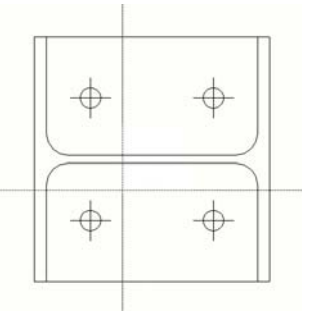
In the drawing



The view which represents the enlarged part is positioned to the original view



The parts in the original view are hidden



to finally only display the view which includes the enlarged parts.

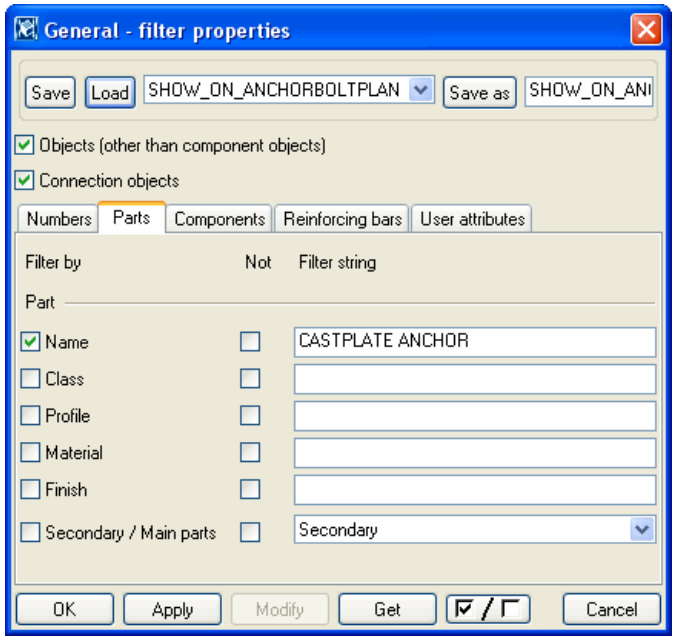


The position of the view which includes the enlarged parts is **not** correct!

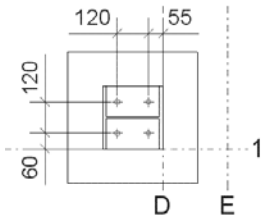
Showing additional parts in an anchor bolt plan

You can add parts that are not usually part of a column assembly to an anchor bolt plan, such as anchor rods or cast plates. In the following example you will make anchor rods and cast plates visible in an anchor bolt plan.

1. Define the name of the GA drawing filter. In this case we will set the variable XS_ANCHOR_BOLT_PLAN_ADDITIONAL_PARTS_FILTER in **Tools** > **Advanced options** > **Drawing properties** to SHOW_ON_ANCHORBOLTPLAN
2. Go to **Properties** > **General arrangement drawing...**
3. Press the **Filter...** button in the GA drawing properties dialog box
4. On the **Parts** tab in the **General - filter properties** dialog box, enter CASTPLATE and ANCHOR in the **Name** field

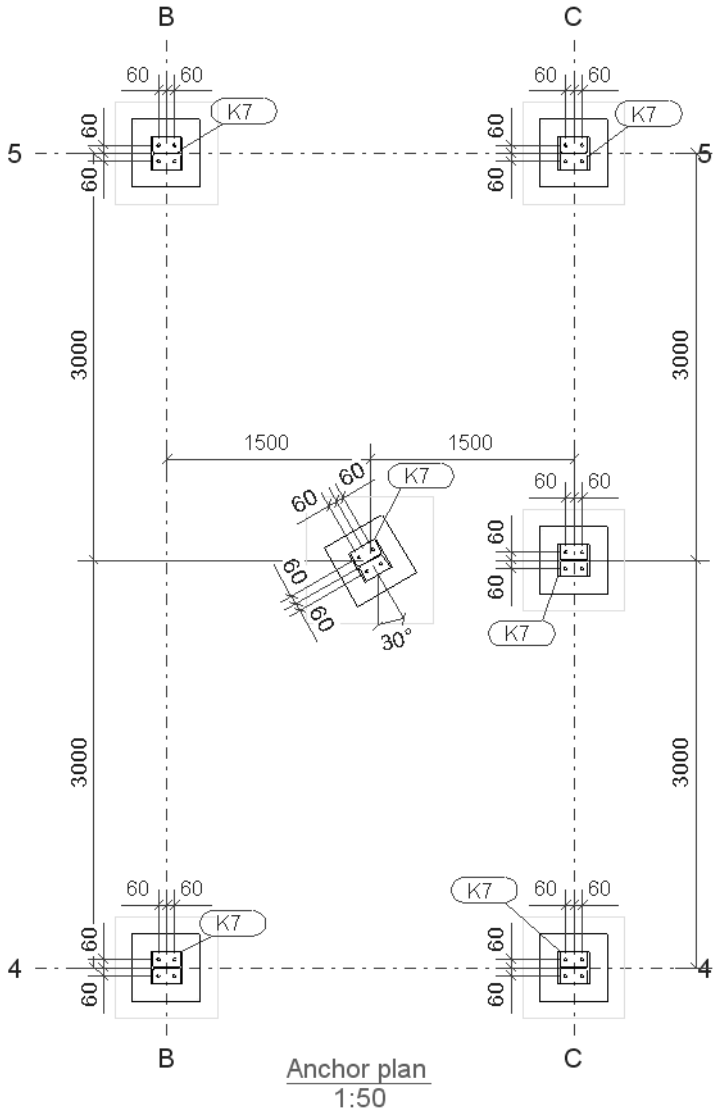


5. Save the filter as SHOW_ON_ANCHORBOLTPLAN. Now when you create an anchor bolt plan, both the anchor rods and the cast plates are shown in the drawing,



Automatic dimensioning

In case you create an anchor bolt plan with two combined scales by which the column reference point does not fall on the grid line, Tekla Structures dimensions its reference point automatically relative to the grid lines. If the column is rotated relative to the drawing, the rotation is automatically dimensioned as well.

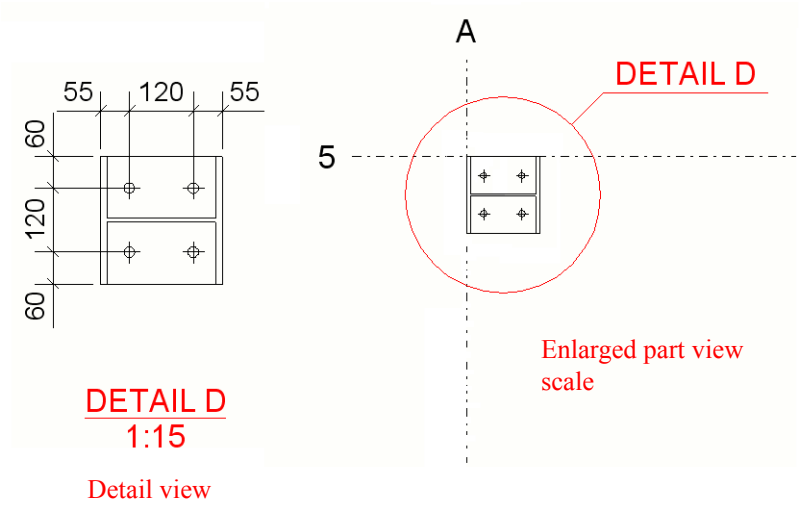


See also

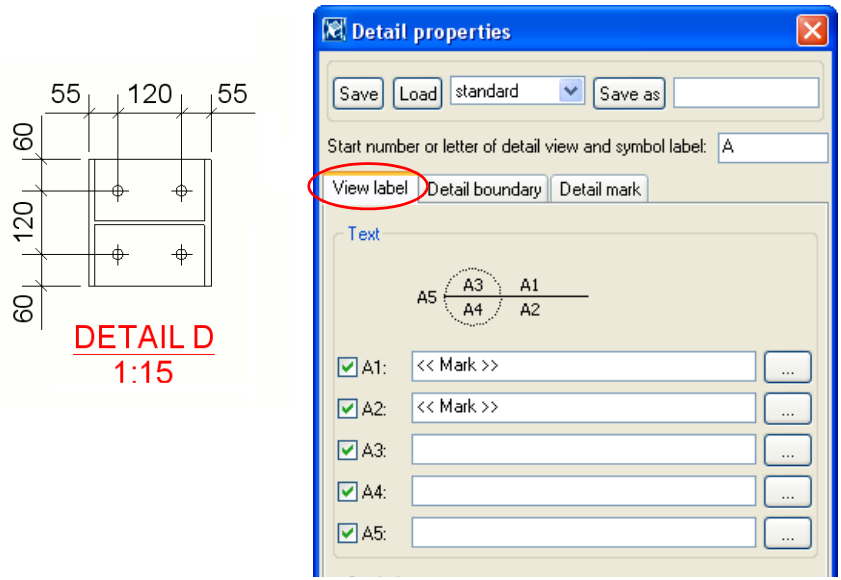
For more information about automatic dimensioning of general arrangement drawings, see **Help > Learning Center > What's New**.

Labels in detail views

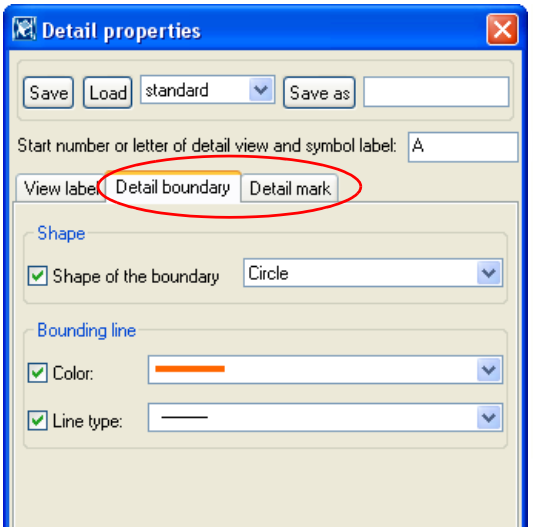
You can set the appearance of the views which includes the enlarged parts and detail views **before** you create the drawing:

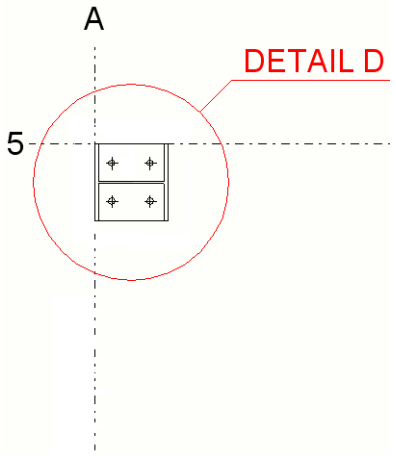


1. In de Model Editor, go to **Properties > General arrangement drawing...** to open the dialog box **General arrangement drawing properties**.
2. Click **Detail view...** to open dialog box **Detail properties**
3. In tab **View label** you can set the detail view label



4. In tabs **Detail boundary** and **Detail mark** you can set the views which includes the enlarged parts:





See also For more information, go to **Help > Learning center > What's New**.

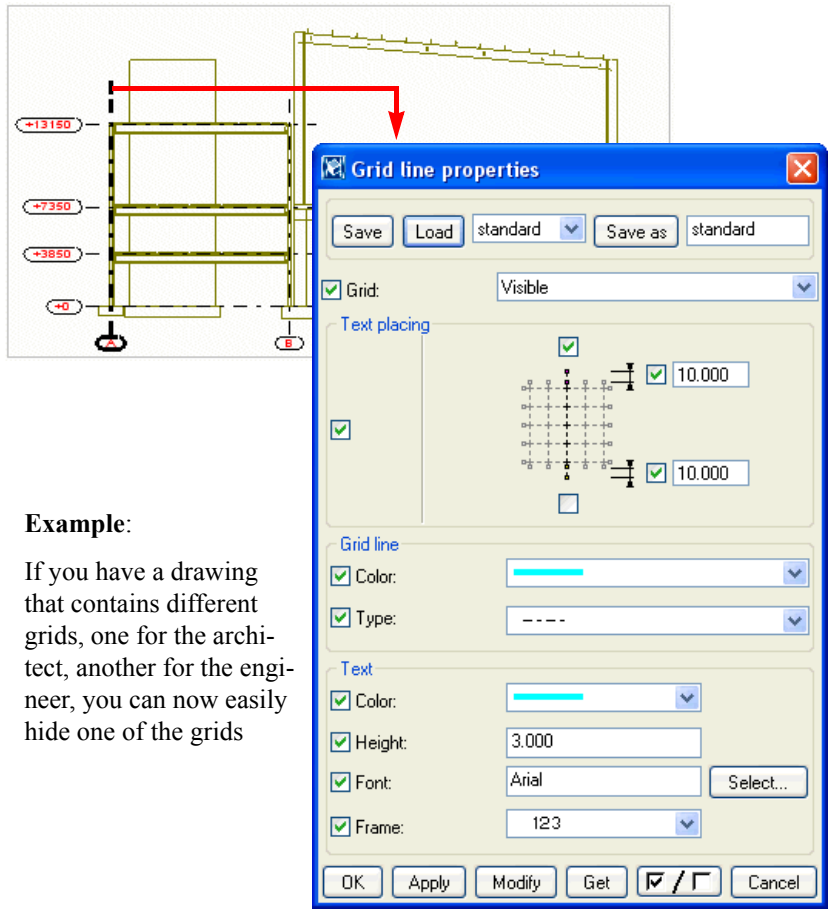
Modifying drawing grids

Two select switches in the toolbar **Drawings:Select** allow you to select grids or individual grid lines in drawings:



Double-click a grid or grid line to change its properties. This is useful when you want to:

- Not print grids or grid lines (use the background color of the drawing for grid lines and text to do this).
- Change a line type.
- Change a label (font, size, frame).
- Define a label location. You can now define different locations for the labels at each end of a grid or grid line.

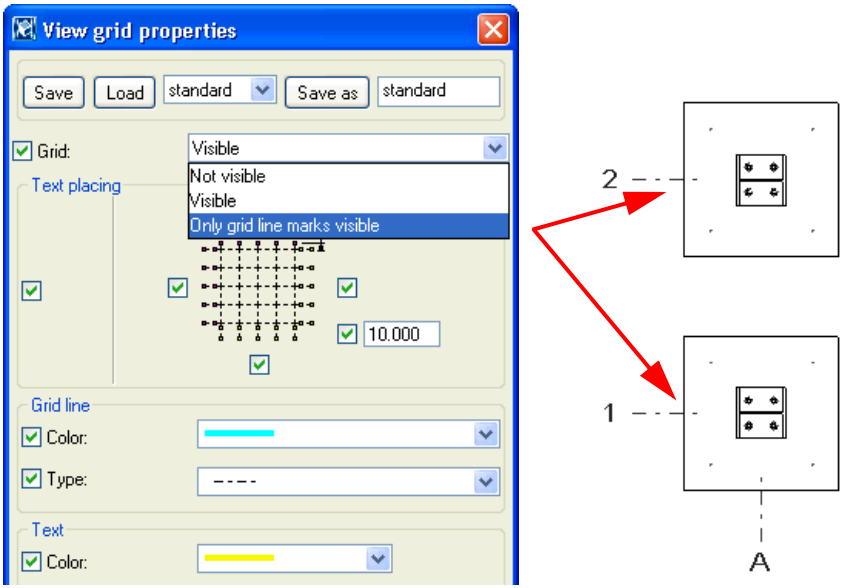


Example:

If you have a drawing that contains different grids, one for the architect, another for the engineer, you can now easily hide one of the grids

Only grid line marks visible

Use this option to only display grid line marks, that means no grid lines



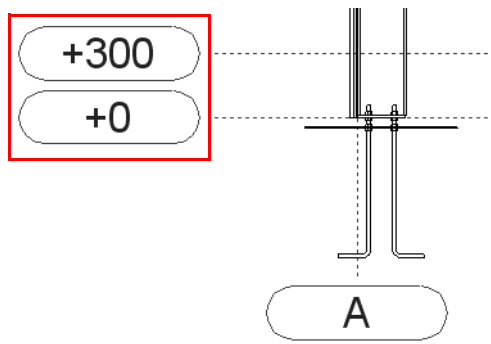
Grid label frame size

You can now define the frame size for the grid labels. This is useful for example when you want all the grid label frames to be the same size no matter whether there are one or two digits in the frame. This gives the grid a consistent appearance.

To define the grid label frame size, use the variable `XS_DRAWING_GRID_LABEL_FRAME_FIXED_WIDTH` in **Tools > Advanced options > Drawing properties**.

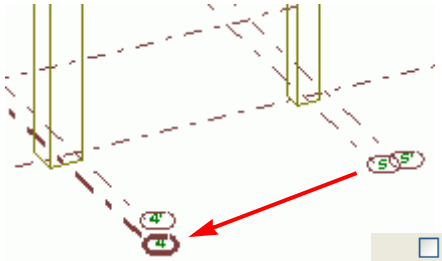
The default value is zero, which indicates that the width of the grid label frame depends on the width of the grid label.

To increase or decrease the width of the frame, enter the desired value in millimeters, in this example, value 12 has been used:



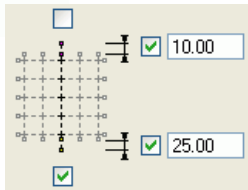
Editing grid line marks

If the marks of two grids overlap, simply extend the grid lines of one grid to move its marks:



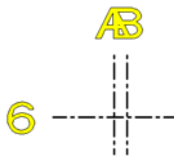
By default, the grid marks overlap.

Changing the length of one grid line makes each mark distinct.

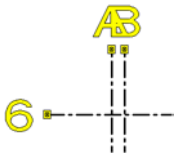


Also the grid line marks can be moved:

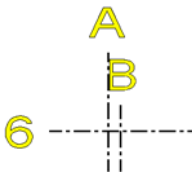
- Set toolbar **Drawings:Select** to only select grid lines
- Now select a grid line; the grips will highlight
- Select a highlighted grip and move it to any direction



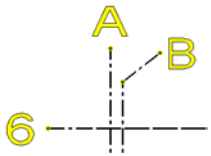
Standard situation: Grid line marks overlap



Select a grid line; the grips will highlight



Select a grip and move vertical



Select a grip and move horizontal

Dimensioning GA-drawings

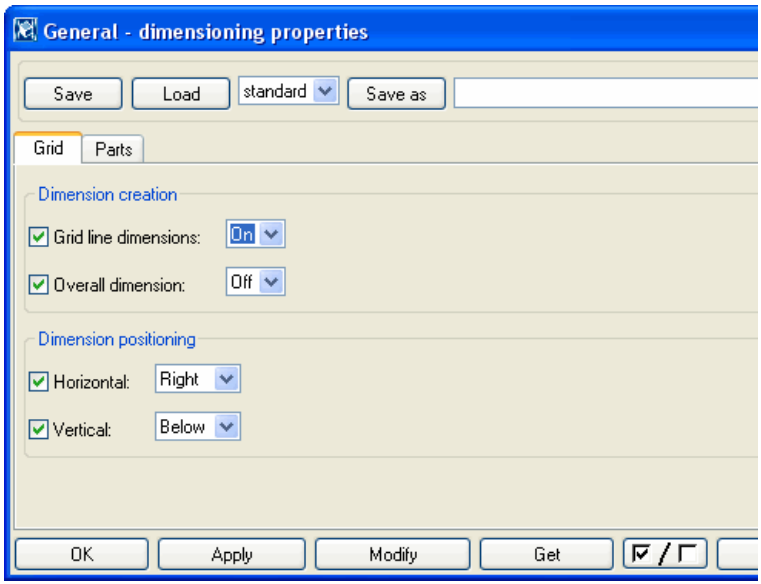
Dimensioning grid lines

Manual:

- Go to **Properties** > **Dimension**
- Load setting “grid” an click <**Apply**> and <**OK**>
- Go to **Create** > **Dimension** > **Grid**. Click the foundation plan view. The grid lines will be dimensioned in X and Y direction.

Automatic:

- Click on the button **Dimensioning...** in the dialog box **General arrangement drawing properties** to open the dialog box **General - dimensioning properties**
- Click on the **Grid** tab to set the grid dimension properties



- Select the options in the listbox to create grid line dimensions and/or overall dimensions and use the options to control where Tekla Structures positions horizontal- and vertical grid line dimensions in the horizontal- and vertical direction

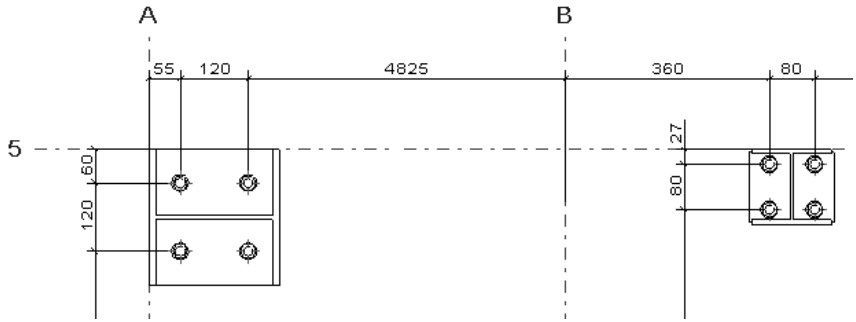
See also

For more information about automatic dimensioning of general arrangement drawings, see **Help > Learning Center > What’s New**.

Dimensioning anchor bars

If you use anchor bars while modelling you have the possibility to create dimensions automatically on general arrangement drawings. It makes use of semi-automatic GA dimensioning. This is a special dimensioning type you only use for GA drawings and it contains several dimensioning options:

- Which points and parts Tekla Structures dimensions
- How Tekla Structures combines dimensions
- Whether to use the reference line of the part or the center line as a dimension point





Mind that this way of dimensioning can be used for anchor bars because they are created as profiles; this option does not work for bolt holes!

Step plan:

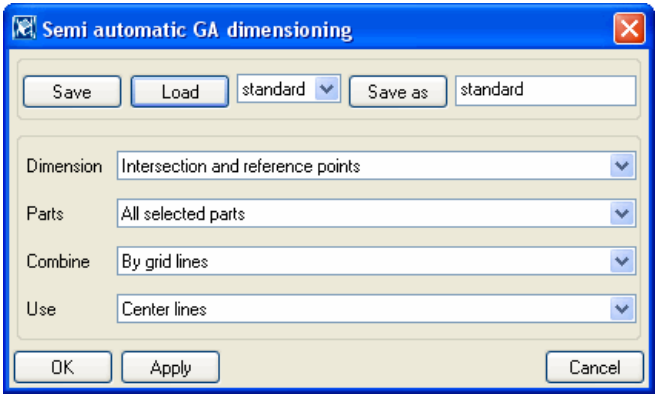
- Open a GA drawing (foundation plan)
- Go to **Properties** > **Dimension...**
- Load the setting you want to use and click <**Apply**> and <**OK**>



- Go to **Setup** > **Select filter...** or click the icon on the left and load the setting *anchorbar_filter.*
- Select in the toolbar **Drawings:Select** the icon *Select drawing parts:*



- Now select the anchor bars on the drawing you want to dimension
- Go to **Setup** > **GA dimensioning...** and set this dialog box as follows:



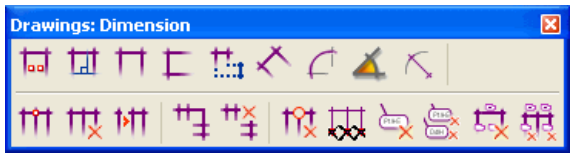
Field	Description
Dimension	Defines which points Tekla Structures uses for dimensioning. **Intersection points** only creates dimensions for points where two parts intersect. **Intersection and reference points** creates dimensions for points where two parts intersect, and to the reference points of the selected parts.
Part	Defines whether Tekla Structures dimensions all the selected parts or only main parts
Combine	Defines how Tekla Structures combines dimensions, see also Tekla Structures Help (F1)
Uses	Defines whether Tekla Structures uses the reference line or the center line as a dimension point.

- Click <**Apply**> and **<OK>**
- Click the right mouse button and choose one of the following options: **Dimension Parts X**, **Dimension Parts Y** or **Dimension Parts XY**

The anchor bars will be dimensioned automatically.

Dimensioning profiles and holes

Dimensioning profiles and holes can be done with the icons in the **Dimension** toolbar.



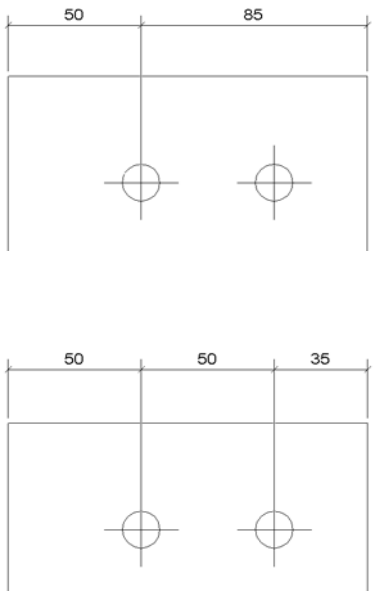
	Parallel dimension		Add dimension point
	Perpendicular dimension		Remove dimension point
	Dimension in X direction		Combine dimension lines
	Dimension in Y direction		Link dimension lines
	Ortho dimension		Unlink dimension lines
	Free dimension		Remove dimension point circle
	Curved radial dimension		Remove all dimension point circles
	Angle dimension		Remove mark change cloud
	Radius dimension		Remove all mark change clouds
			Remove dimension point cloud
			Remove all dimension point clouds

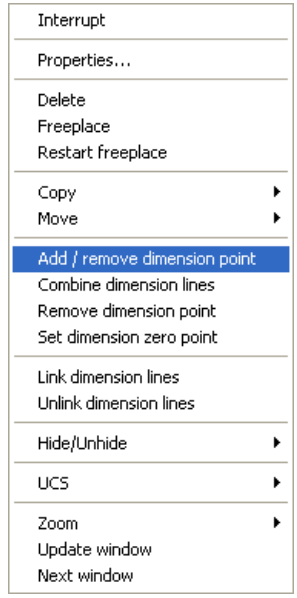
See also

For more information about dimensioning, see the Tekla Structures Help (F1).

Add/remove dimension points

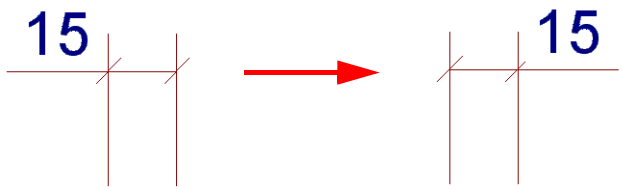
By means of the combination **Shift**-button and the function **Add/remove dimension point** you can remove dimensions. So you don't have to switch between the two functions anymore.





Change the side of outside dimensions

When measuring short dimensions in Tekla Structures and when the location of the dimension text is set to outside of the dimension lines, you can now select on which side of the extension line the dimension text is placed.



This makes drawing editing easier and faster. To change the side of outside dimensions:

1. Click **Edit > Flip outside dimension**.
2. Select the dimension.

Tekla Structures changes the side of the dimension text.

Fixed dimensions

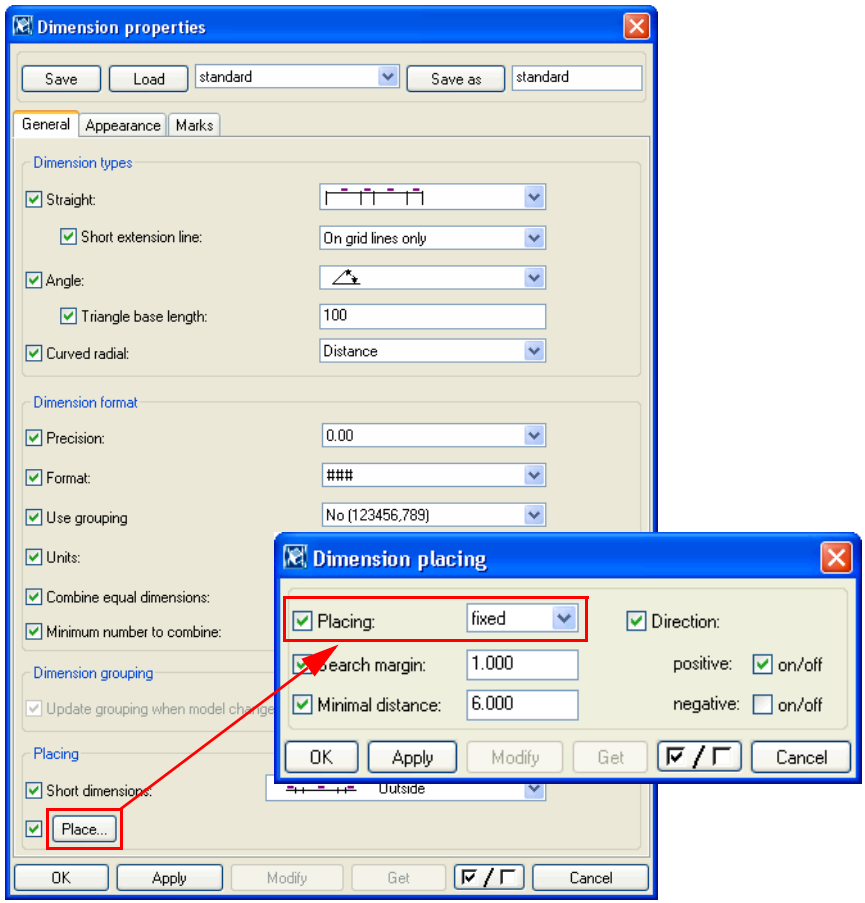
It is recommended to fix dimensions on GA drawings. Because of that they will not jump to any position on the drawing anymore when for example another view is added, by the way, this also applies for the marks!

Step plan

1. Create all the dimensions on the drawing
2. Set the toolbar **Drawings:Select** as follows:



3. Select all dimensions
4. Hold down the <**Shift**> key and double-click a random dimension, the following dialog box appears:



5. Now click the button <**Place**> and choose **Fixed** in the list box
6. All dimensions are now fixed.

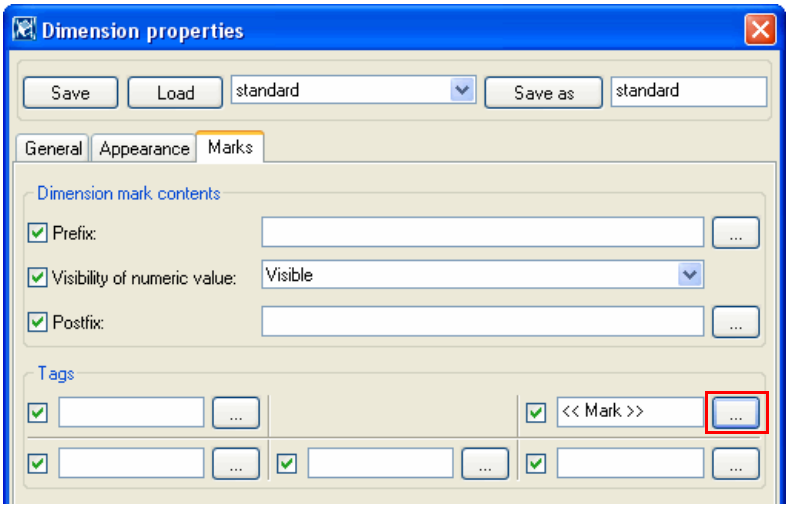
Dimension line tags

You can add dimension line tags to dimension lines. Dimension tags are fully associated to the objects the dimension belongs to and they are available for all dimension types. This extra information in the drawings will be automatically updated with any changes made in the model.

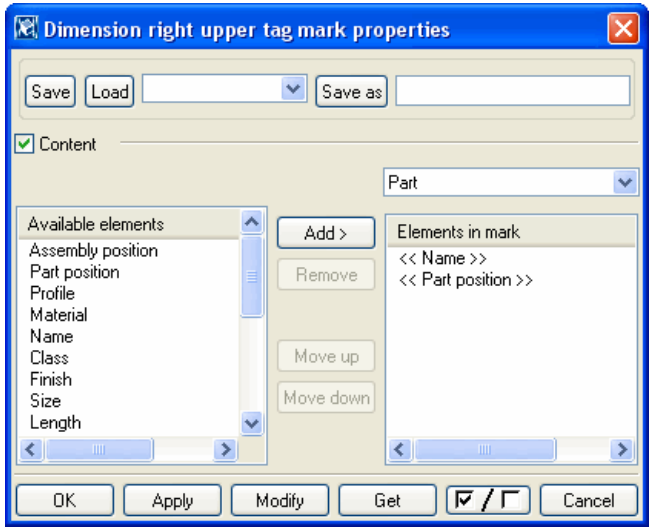
How to use

To add associative marks as text into dimensions tags:

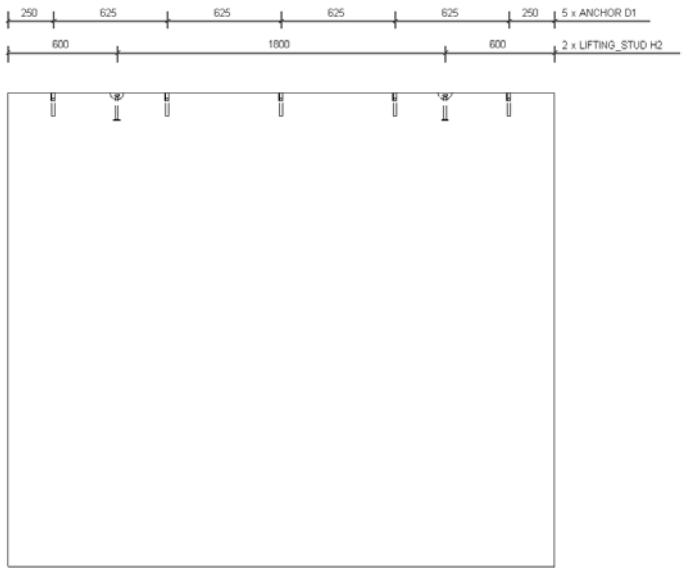
- Double-click the dimension line to open the **Dimension properties** dialog box.
- On the **Marks** tab, press the **...** button next to the text field of the tag where you want to add the associative mark to. For example, to add the associative mark to the upper right tag, select the button marked in the image below



- In the dimension mark properties dialog box, select the desired mark element in the list of available elements and click the <**Add**> button

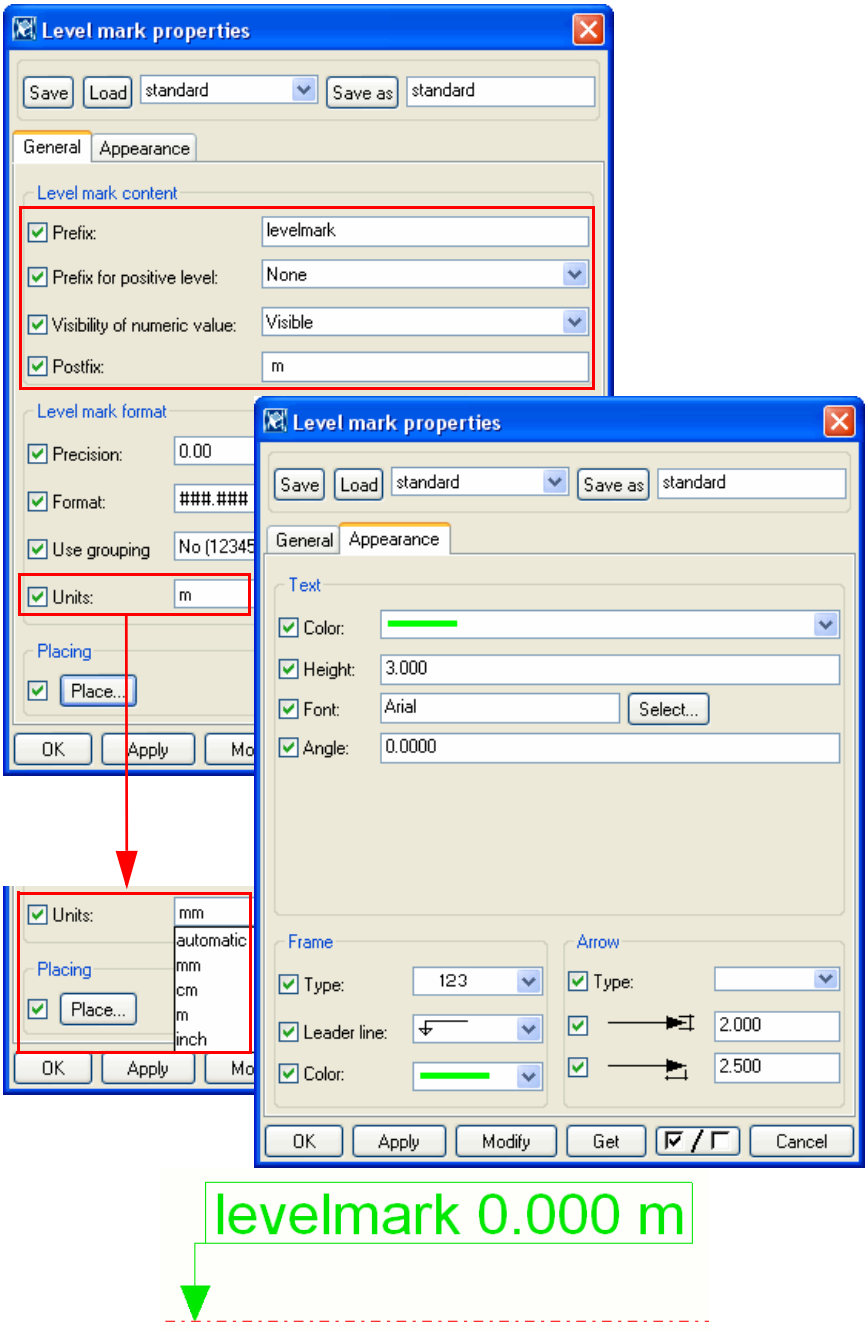


- Click <**Modify**>
- Tekla Structures adds the associative mark as text into the dimension tag:



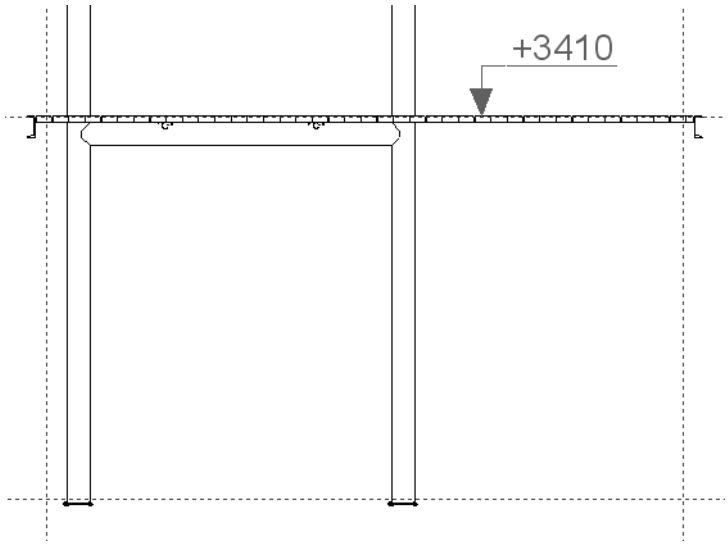
Level marks

In Tekla Structures you can use level marks and you have several options for level marks such as prefixes, postfixes and units.



Set a datum point for elevations

Tekla Structures determines the level marks in drawings using a reference point. These level marks can be manipulated in drawings manually. If you have created a level mark of 3610 mm in a drawing, you can enter a **Datum level** of 200, for example. The result is a level mark of 3410 mm.



Changing the reference point

1. In the Drawing Editor, open dialog box **Drawing properties** and click on the button **View**; or double-click a view go to the tab **Attributes 2**.
2. Select the option **Specified** in the listbox **Datum point for elevations:**

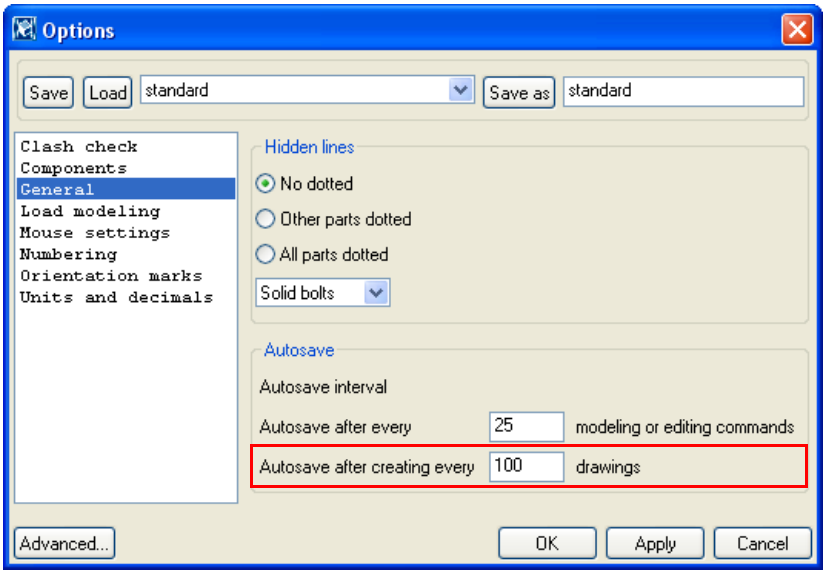


3. Enter a value in the field **Datum level**.

2.4 Save drawings

General

In the dialog box **Options** (**Tools** > **Options**) in de Model Editor, a value is entered for **Autosave after creating every 100 drawings**. So while generating drawings, making use of the Wizard, after 100 drawings an **Autosave** file is created.



This function **only** works for generating drawings, making use of the Wizard, **not** for editing drawings in the Drawing Editor.

Save drawings

When editing drawings you must save the drawing by clicking the icon.



Now both drawing (*.dg-file) and model (the *.db1 and the *.db2-file) will be saved, so frequently save the drawing!



When you regularly cannot open a drawing anymore, copy the variable

set XS_DELETE_UNNECESSARY_DG_FILES=TRUE

from the file `env_europe.ini` (in the folder *..\environments\europe*) to the file `user.ini` (in the folder *..\nt\bin*) so that Tekla Structures will not delete *.dg-files anymore.

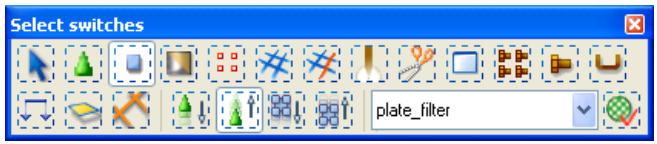
Lots of *.dg-files are stored now in the folder **drawings**, you can solve this issue by setting the variable to FALSE.

Now open the model once and close the model, the unnecessary *.dg-file are deleted.

2.5 Step plan for generating Single part drawings

Step 1

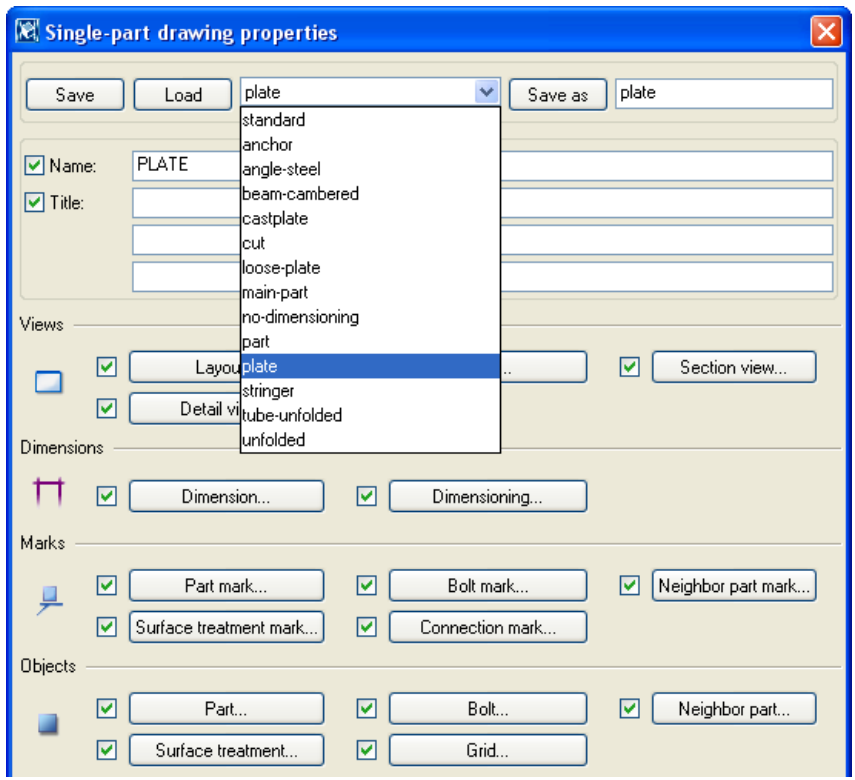
- Model numbering: Go to **Tools** > **Numbering** > **Full**. Basically, numbering the model one time is sufficient. After modifying the model, numbering has to be repeated!
- Set the toolbar **Select switches** in a way that only the icons **Select parts** and **button 2** are switched on:



- Select the **plate_filter** from the listbox and click in the model once (left mouse button) to lock the **plate_filter**
- Select the entire model, go to **Edit** > **Select all objects**. (The plate filter makes that only plates will be selected)

Step 2

- Go to **Properties** > **Single-part drawing…**
- Load the setting **plate**.



- Click <**OK**>



- In the pull-menu go to **Drawing** > **Single-part drawing**, click on the icon on the left or click shortcut “1”, the drawings will be generated now



- Open the drawing list by way of **Drawing** > **List** (or with the icon) and open a drawing (by double-clicking a drawing or by selecting the drawing and pressing the *Open* button)

<Ctrl> + <Shift>



To automatically open a drawing after creating (or updating it), press down the <**Ctrl**> + <**Shift**> buttons. In the pull-down menu go to **Drawing** > **Single-part drawing** or **Assembly drawing** or press the right-mouse button and select **Drawing** > **Single-part drawing** or **Assembly drawing** in the pop-up menu.

Tekla Structures now:

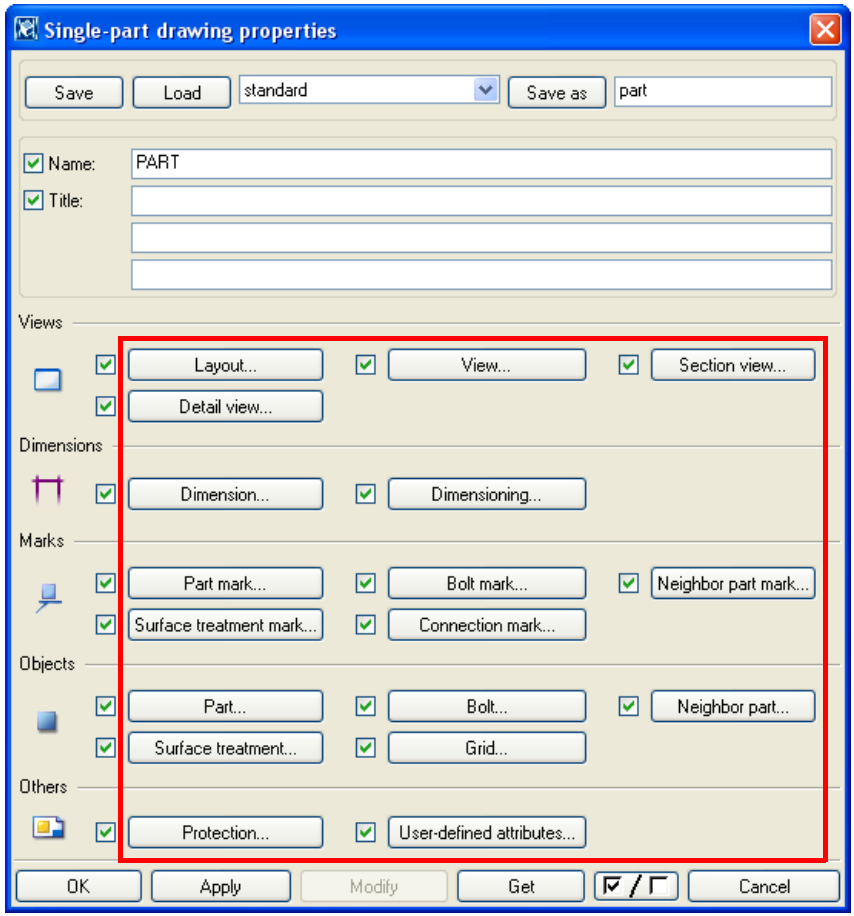
- Creates a drawing or updates an existing drawing
- Opens a drawing automatically

- The single-part drawings are recognizable in the drawing list by a “W” in the column “Type”. This “W” character stands for “*W*orkshop drawing”
- Go to **File** > **Close drawing** > **No**. After all, the drawing has not been changed, so it is not necessary to save the drawing

Exercise Select or create a filter and generate Single part drawings of all parts.

So far, you know how to create the single part drawings. But maybe even more important is how to create the right settings for your single part drawings!

- Open the single part drawing of the base plate on grid intersection D-4
- Go to **Properties** > **Drawing** or choose **Properties...** from the con-textmenu, the following dialog box appears:



All sub dialog boxes can now be changed. After the changes to the sub-dialogs have been carried out, press <**Apply**> and <**OK**> when leaving the dialog box. Then, in the main dialog box, press <**Modify**>, now you can discover what has been changed to the drawing due to the changes in the settings.

If you apply changes to a single part- (or an assembly drawing) manually (adding dimensions, etc.), and the drawing needs to be regenerated, all these manual modifications will be lost! If this drawing needs to be re-generated, making use of the desired setting, the setting needs to be loaded first. Now click <**Modify**>.

See also

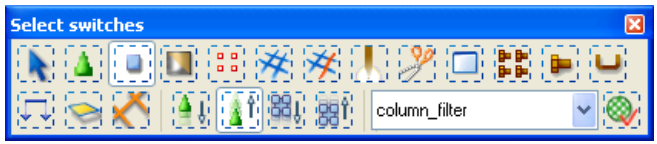
For more information about settings in the sub-dialogs, see the **Tekla Structures Online Help.**

2.6 Step plan for generating Assembly drawings

Example Assembly drawings of all columns.

Step 1

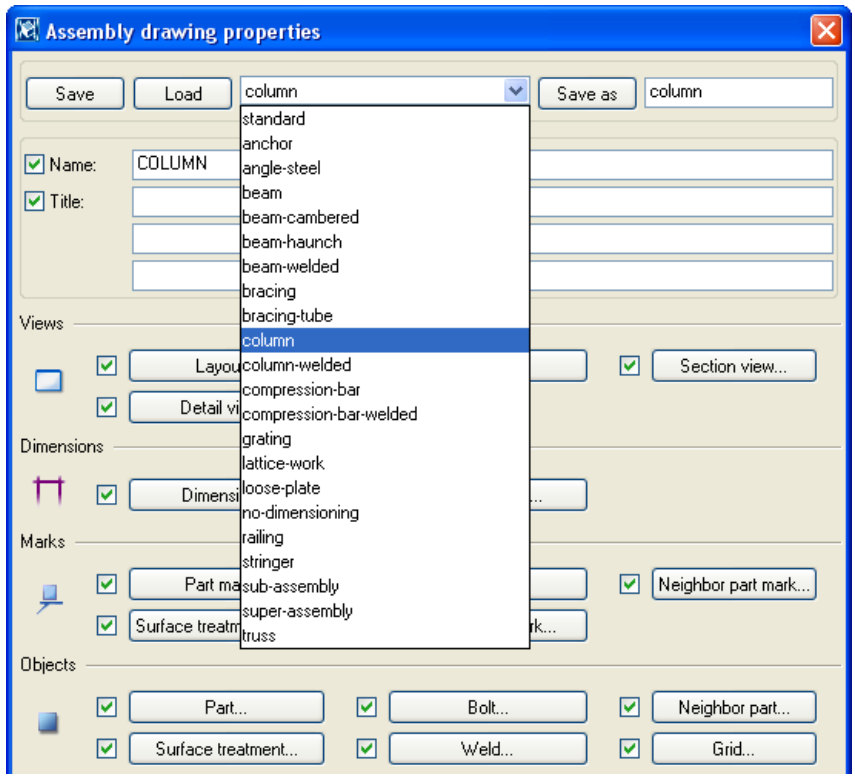
- Set the toolbar **Select switches** in a way that only the icons **Select parts** and **button 2** are switched on:



- Select the **column_filter** from the listbox and click in the model once (left mouse button) to lock the **column_filter**
- Select the entire model, go to **Edit** > **Select all objects**. (The column filter makes that only columns will be selected)

Step 2

- Go to **Properties** > **Assembly drawing...**
- Select setting **column**, now press the **Load**-button.



- Click <**OK**>



- In the pull-menu go to **Drawing** > **Assembly drawing**, click on the icon on the left or click shortcut “2”, the drawings will be generated now



- Open the drawing list by way of **Drawing** > **List** (or with the icon) and open a drawing (by double-clicking a drawing or by selecting the drawing and pressing the <**Open**> button)
- The drawing will be opened. The assembly drawings are recognizable in the drawing list by an “A” in the column “Type”. This “A”-character stands for “*A*ssembly drawing
- Go to **File** > **Close drawing** > **No**. After all, the drawing has not been changed, so it is not necessary to save the drawing

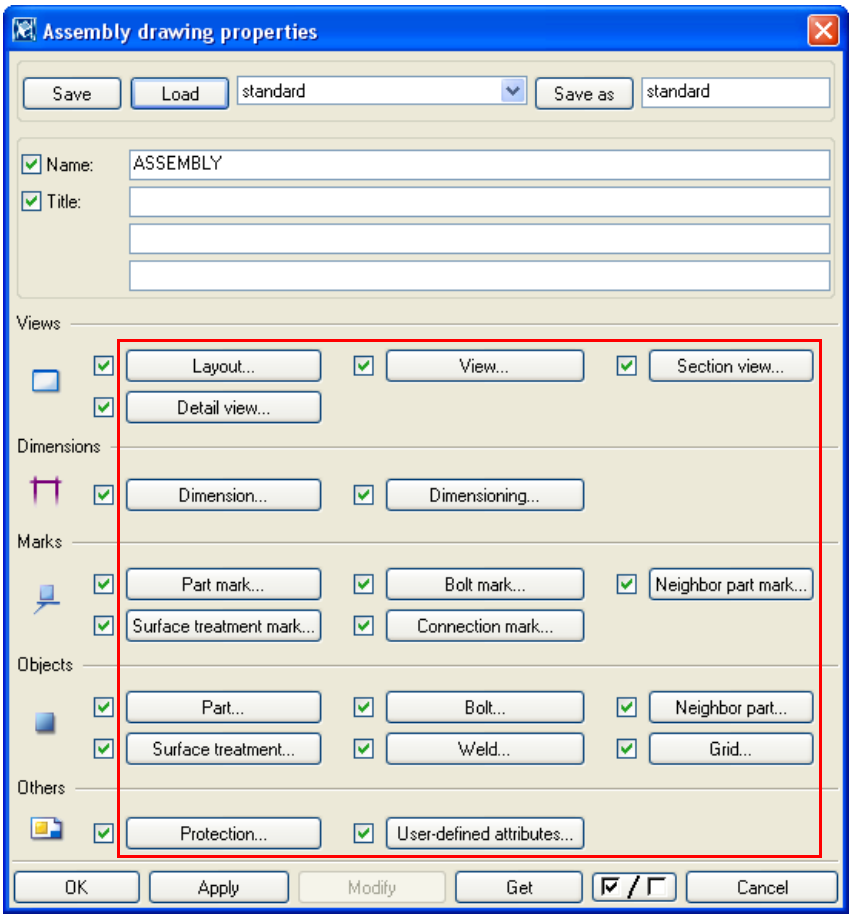
Exercise

Create assembly drawings for Columns, Beams, Box girders, Windbracings

Choose or create a proper filter and generate the assembly drawings!

So far, you know how to create the assembly drawings. But maybe even more important is how to create the right settings for your assembly drawings!

- Open the assembly drawing from the column located on grid intersection A-4
- Go to **Properties** > **Drawing** or choose **Properties...** from the contextmenu, the following dialog box appears:



All sub dialog boxes can now be modified. By pressing the <**Modify**>-button each time when a change is made in one of the sub dialog boxes, you can see what has been changed in the drawing.

See also For more information about the sub-dialogs, see the **Tekla Structures Help**.

Exercise Create assembly drawings of the columns like you usually create a drawing. Save this setting and copy the setting file **.ad*-file to folder: **disk:\TeklaStructures\version\environments\europe\ts.**

Do the same for the beams, the windbracings and the box girders!

2.7 Step plan for generating Cast unit drawings

Step 1

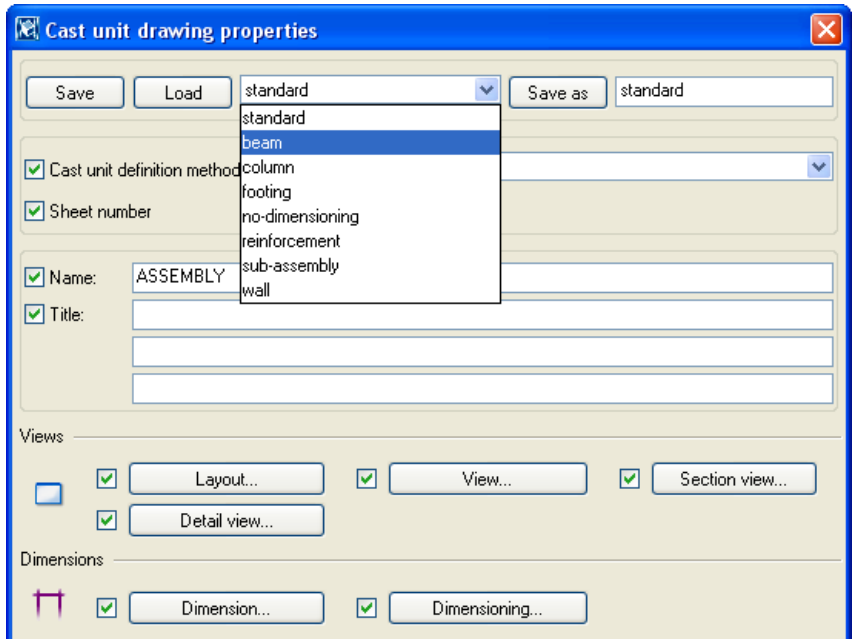
- Model numbering: Go to **Tools** > **Numbering** > **Full**. Basically, numbering the model one time is sufficient. After modifying the model, numbering has to be repeated!
- Set the toolbar **Select switches** in a way that only the icons **Select parts** and **button 2** are switched on:



- Select the **beam_filter** from the listbox and click in the model once (left mouse button) to lock the **beam_filter**
- Select the entire model, go to **Edit** > **Select all objects**. (The beam filter makes that only concrete beams will be selected)

Step 2

- Go to **Properties** > **Cast unit drawing…**
- Load the setting **beam.**



- Click <**OK**>



- In the pull-menu go to **Drawing** > **Cast-unit drawing**, click on the icon on the left or click shortcut “4”, the cast unit drawings will be generated now



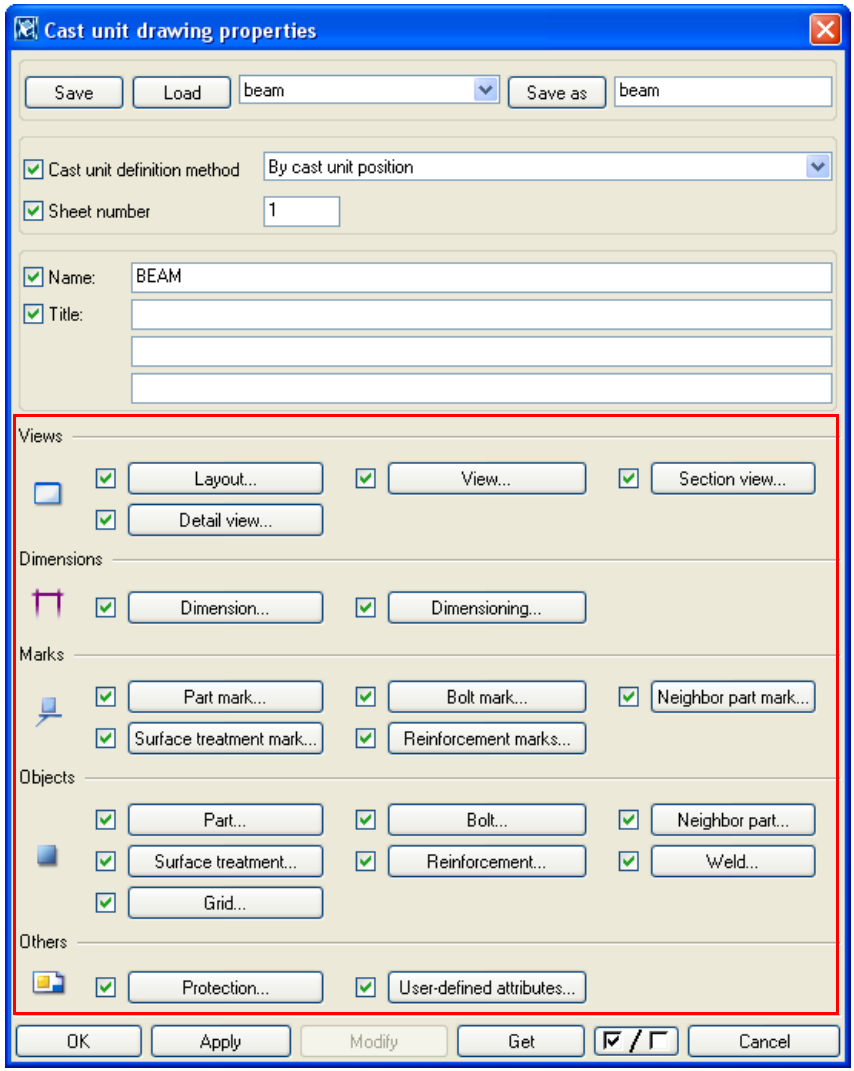
- Open the drawing list by way of **Drawing** > **List** (or with the icon) and open a drawing (by double-clicking a drawing or by selecting the drawing and pressing the *Open* button)
- The cast-unit drawings are recognizable in the drawing list by a “C” in the column “Type”. This “C” character stands for “*C*ast unit drawing”
- Go to **File** > **Close drawing** > **No**. After all, the drawing has not been changed, so it is not necessary to save the drawing.

Exercise

Create cast unit drawings for the cast unit parts and select or create a filter for those parts.

So far, you know how to create the cast unit drawings. But maybe even more important is how to create the right settings for your cast unit drawings!

- Open the cast unit drawing of the concrete beam
- Go to **Properties** > **Drawing** or select **Properties...** from the context-menu, the following dialog box appears:



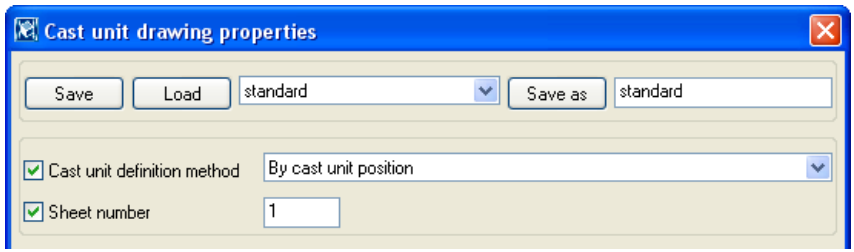
All sub dialog boxes can now be changed. After the changes to the sub-dialogs have been carried out, press <**Apply**> and <**OK**> when leaving the dialog box. Then, in the main dialog box, press <**Modify**>, now you can discover what has been changed to the drawing due to the changes in the settings.

If you apply changes to a cast unit drawing manually (adding dimensions, etc.), and the drawing needs to be regenerated, all these manual modifications will be lost! If this drawing needs to be re-generated, making use of the desired setting, the setting needs to be loaded first. Now click <**Modify**>.

See also For more information about settings in the sub-dialogs, see the **Tekla Structures Online Help.**

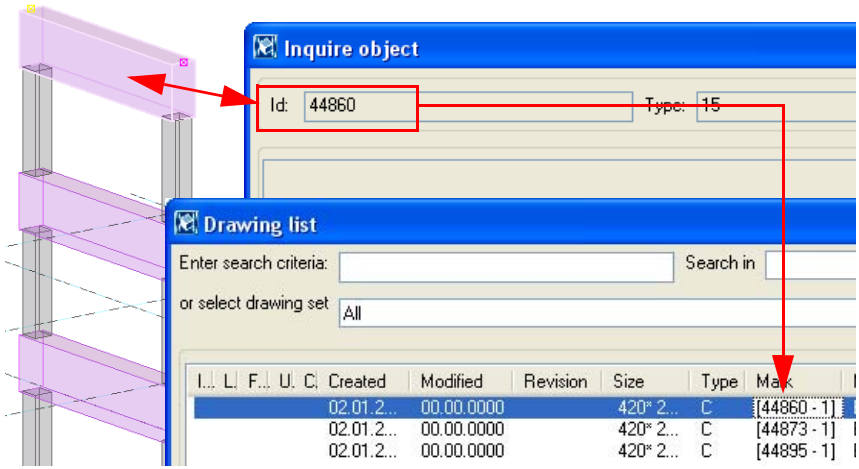
Option Cast unit definition method

In the Cast unit drawing properties you can set the **Cast unit definition method** and the **Sheet number**. You can only set this before creating cast unit drawings. So the properties need to be set in the model editor. In the pull-down menu, go to **Properties > Cast-unit drawing:**



By cast unit position In case the option **By cast unit position** is set, a drawing will be created from each unit. If there are more identical units, one of those units will serve as the base unit for the drawing. This is the common method for creating cast unit drawings. In this document, we consider all drawing to be created by using this option.

By cast unit ID Each part in the model includes a unique ID-number. You can set to create a drawing per ID-number. The ID-number determines the mark of the drawing:



From identical units, several drawings are created. The reason for this method is to detail some cast unit connections. This means that the cast units are identical but the position of the imbeds, for example, can differ. It could also be handy to create a drawing per ID-number in case that grids or elevation marks should be displayed on the drawings.

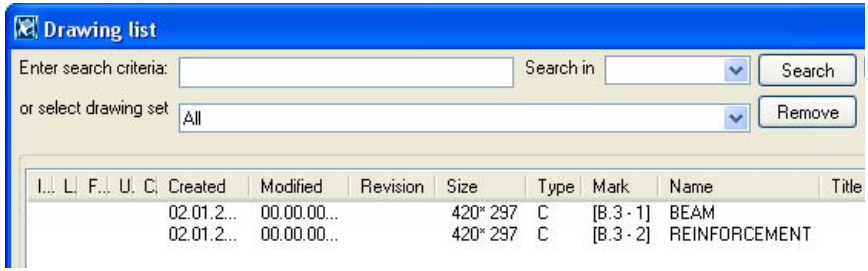
Option Sheet number

Per unit you can create several drawings to avoid an overload of information on one drawing. Generally, the carpenter who is making the cast mould, for example, is not interested in the reinforcement.

Suppose that from a unit the cast unit drawing already exists and that you also want to create a reinforcement drawing:

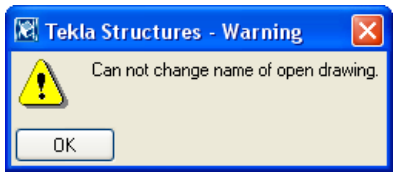
Step plan

- Open in the Model Editor the cast unit drawing properties via **Properties** > **Cast-unit drawing**.
- Enter for **Sheet number** '2' ('1' is the existing "shape" drawing)
- Load the existing setting and/or modify the setting and click **<Apply>** and <**OK>**
- Next select the unit and create the cast unit drawing:



The mark of the unit is displayed in the drawing list in the column **Name**, followed by the sheet number. In the example above: **B.3-1** and **B.3-2.**

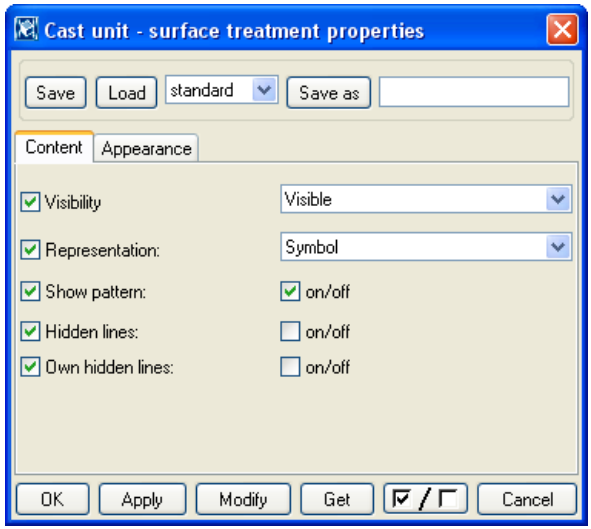
The cast unit drawing setting **Reinforcement** includes sheet number 2 by default. If you load this setting while having opened a cast unit drawing with sheet number 1 and you change the drawing, the following message will appear:



The drawing will be changed but the name of the drawing will not change. If you want to change the name, select the drawing from the drawing list. Now click the right mouse, select Properties in the pop-up menu, load the setting **Reinforcement** and click <**Modify**>.

Surface treatment and surface treatment mark

While modeling you can add one or more surface treatments to a plane of an element or to a part of a plane of an element. This surface treatment(s) can be displayed on a drawing. Open the drawing properties and click the button **Surface treatment.**

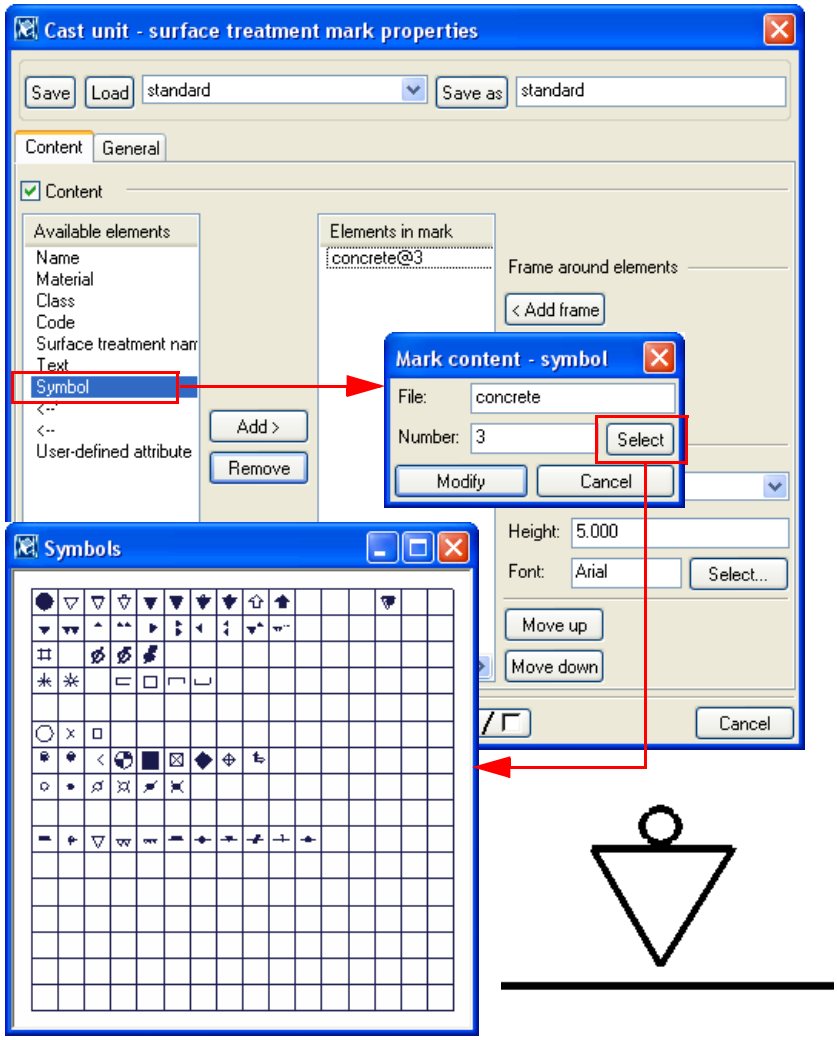


Of course you can set the common settings. Switch on the checkbox at **Show pattern** to display the pattern on the drawing. In case the surface treatment is used to indicate the cast unit elements' pouring-side, it can be handy to use the same color, at the tab **Appearance**, as the appearance of the part itself.

See also

For more information about using surface treatments, see **Tekla Structures.pdf** in the folder: **disk:\TeklaStructures\version\environments\europe\pdf.**

At the button **Surface treatment mark** you can define the mark content and the mark properties.



This mark can consist of the surface treatments' name or any text, for example. In the example above a symbol from the symbol file **concrete.sym** is used**.**

See also

For more information about defining marks and general mark properties, see the **Tekla Structures Online Help**.

2.8 Adding graphical objects

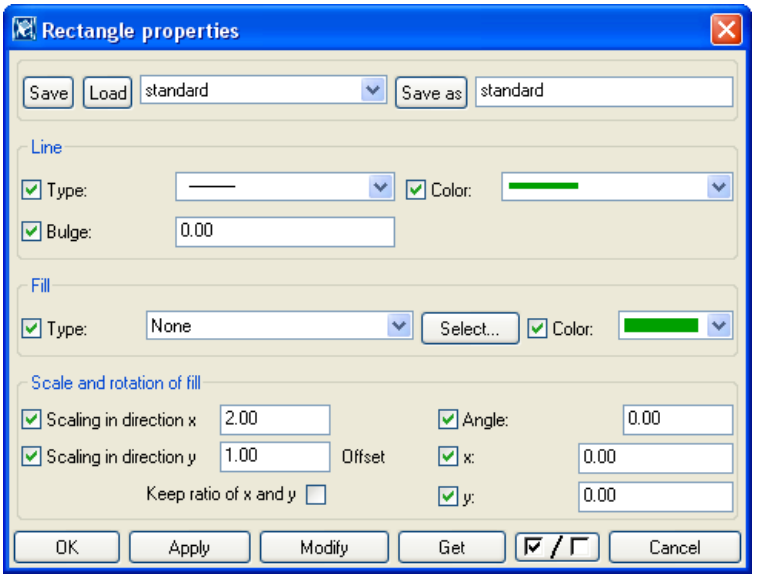
In Tekla Structures it is possible to add graphical objects such as AutoCAD drawings (DWG/DXF) and also symbols, text fields, dimension lines etc. to Tekla Structures drawings. Graphical objects can be placed on random locations, or with help of fixed coordinates.

Add a rectangle on a random location

1. Open a drawing
2. Go to **Properties** > **Rectangle...** or double click the icon in toolbar **Drawing: Drawing**



3. The following dialog box appears.

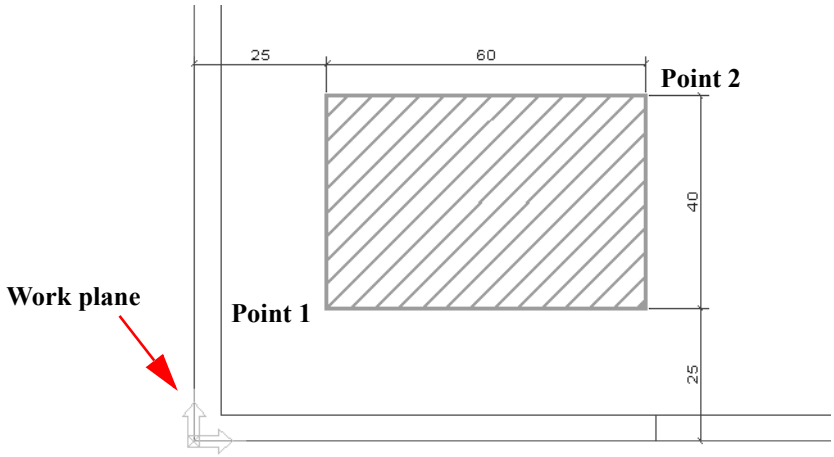


4. In frame **Line**, the line type, color and bulge can be set. In frame **Fill**, a hatch type and color can be set
5. Click <**Apply**> or **<OK>** and go to **Create** > **Rectangle...**
6. Make sure the all **Snap settings** buttons are pressed, so that the rectangle can be made on any random position

7. Now pick two points and the rectangle will be placed
8. Double-click the rectangle to change its properties
9. To resize the rectangle, select the rectangle and drag a vertex or midpoint.
10. By holding down the <**Shift**>-key, the rectangle can be moved

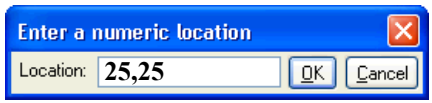
Adding a rectangle using fixed coordinates

Suppose you want to add the following rectangle:

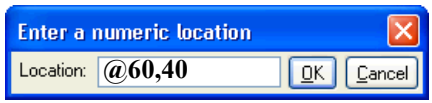


Step plan

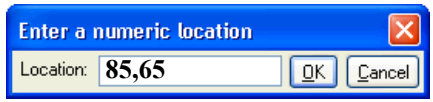
1. Go to **Create** > **Rectangle...** or click the icon
2. Enter the coordinates of the first point (1) relative to the position of the work plane



3. Enter the coordinates of the second point (2), relative to the last picked point using short key “R”



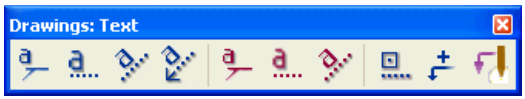
or



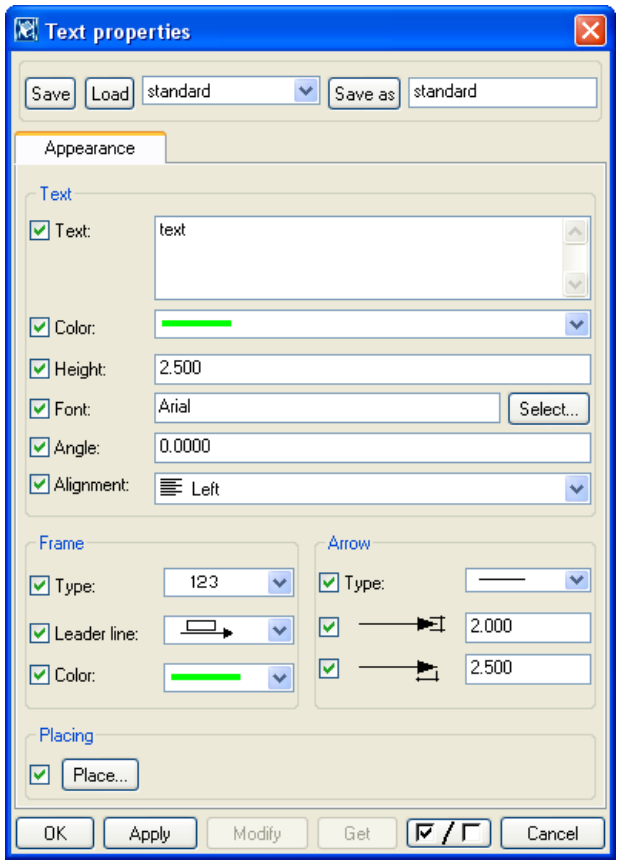
Adding text

Text can be added to drawing in two ways. Text strings can be added using the icons in the **Drawing: Text** toolbar. It is also possible to add a text by inserting a text file.

1. Open a drawing and go to **Properties** > **Text...** or double-click the icon in the toolbar **Drawings: Text**



2. There are several icons to add text (with or without a leader line, text along a line, etc.)
3. The dialog below will appear:



4. In frame **Text**, the text string can be entered. The options **Color** through **Angle** determine the representation of the text string

5. Also a frame and arrow can be added to the text string.
6. Click <**Apply**> or <**OK**> and go to **Create** > **Text...**
7. Make sure that all buttons in the **Snap settings** toolbar are pressed, so that the text string can be placed at any position in the drawing
8. Pick a point in the drawing, the text string will be placed
9. Double-click the text string to change the properties
10. To move the text string, select the string and drag it with the left mouse button

Using symbols in text

When using text in drawings you have the possibility to add symbols to this text (e.g. weld symbols).

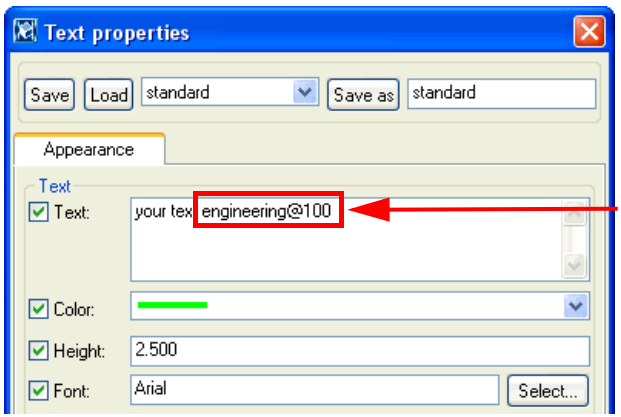


Step plan

1. Add a text to a drawing
2. To know what symbol files (**.sym*) there are and which numbers belong to the symbols, go to **Tools** > **Symbols** or click on the icon. The Symbol Editor opens.
3. In Tekla Structures by default several symbol files are available. These files are stored in

drive:\TeklaStructures\version\environments\country-independent\symbols

4. Enter de name of the **.sym* file and the number of the symbol
5. Double-click on the text, the following dialog box appears:



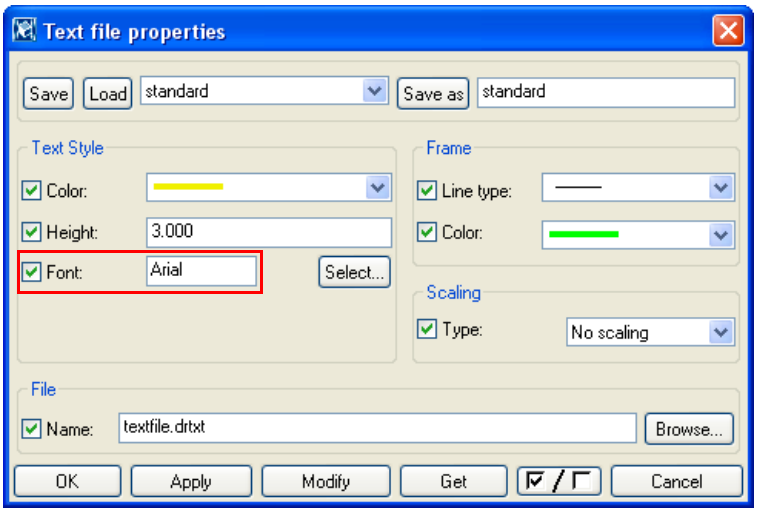
Enter here:

The name of the **.sym* file @ the number of the symbol

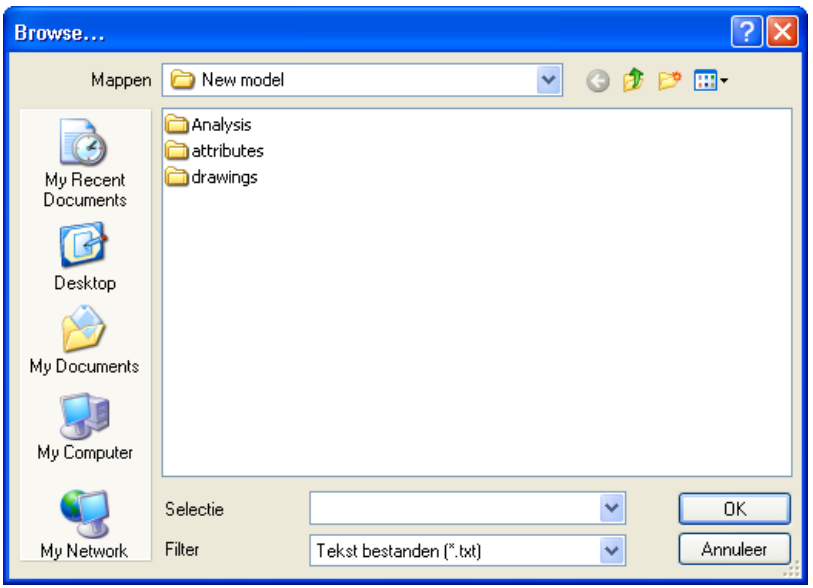
Adding a text file

Step plan

1. Open a drawing and go to **Properties** > **Text file...**
2. The dialog box below will appear:



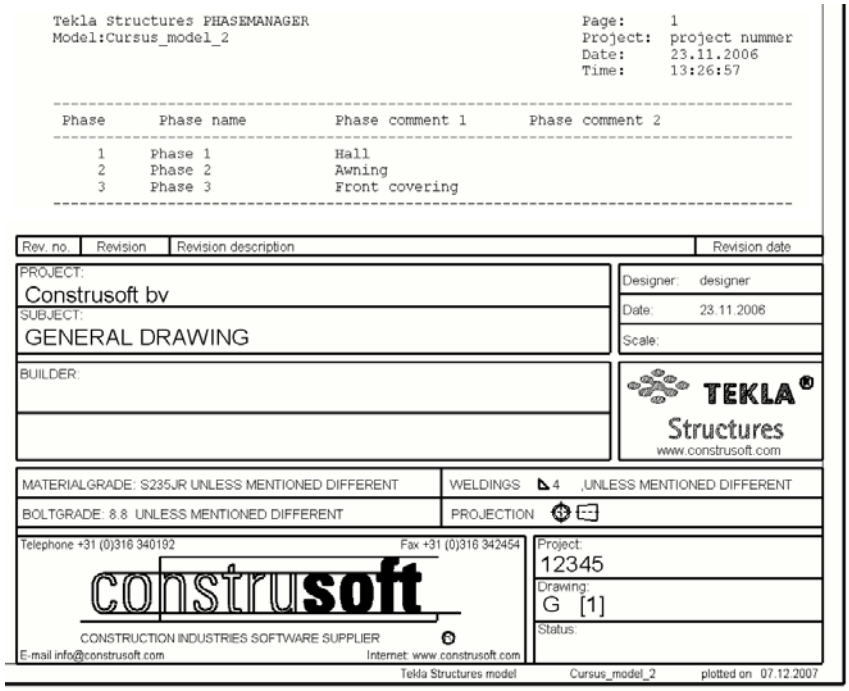
3. You can edit the text file properties in **Text style**, **Frame** and **Scaling**, the preferred font type is “Courier New”.
4. In **File** you can look up for the file you want to add to the drawing, click **<Browse>,** the following dialog box appears:



5. Look up the file you want to add to the drawing.
6. Click <**OK**> and <**Apply**> and **<OK>**
7. Make sure that all buttons in the **Snap settings** toolbar are switched on, so that the text file can be placed at any position in the drawing
8. Go to **Create** > **Text file**
9. Pick a random point in the drawing; the text will be added
10. The properties of the text file can be edited by double clicking the frame. To modify the text content, double-click the text content. A word processor linked to the file extension will be opened.
11. To move the text, select the frame around the text and drag the frame.



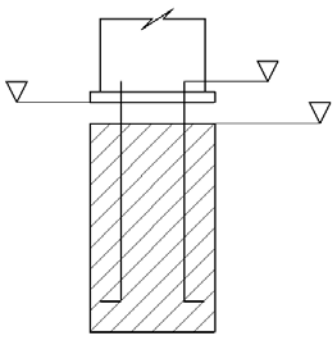
Tekla Structures Reports can also placed on drawings. To do this, the **filter** in the Browse-dialog has to be set to “All files (*.*)”. This because reports have the extension ***.xsr** and will not be found otherwise. Set the text font to “Courier New”.



Adding a symbol

In the Symbol Editor (SymEd), Tekla Structures symbols can be created and edited.

An example of a symbol can be a foundation detail, see image below:



Creating a symbol

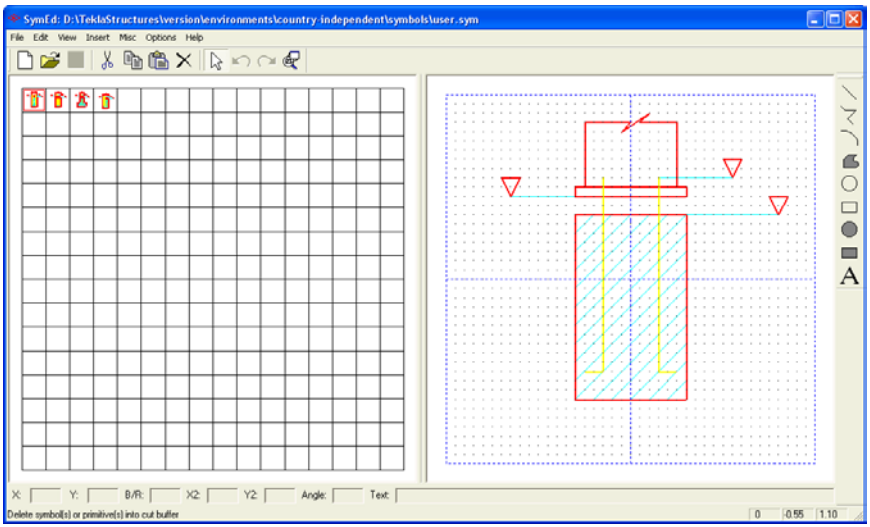
1. Go to **Tools** > **Symbols** or click on the icon to open the Symbol Editor (SymEd), accessible in both model- and drawing environment.



By default, some symbol libraries are available in Tekla Structures. Symbol libraries have the extension ***.sym** and are located in

disk:\TeklaStructures\version\environments\country-independent\symbols

2. When double-clicking file **user.sym**, the following symbols appear:

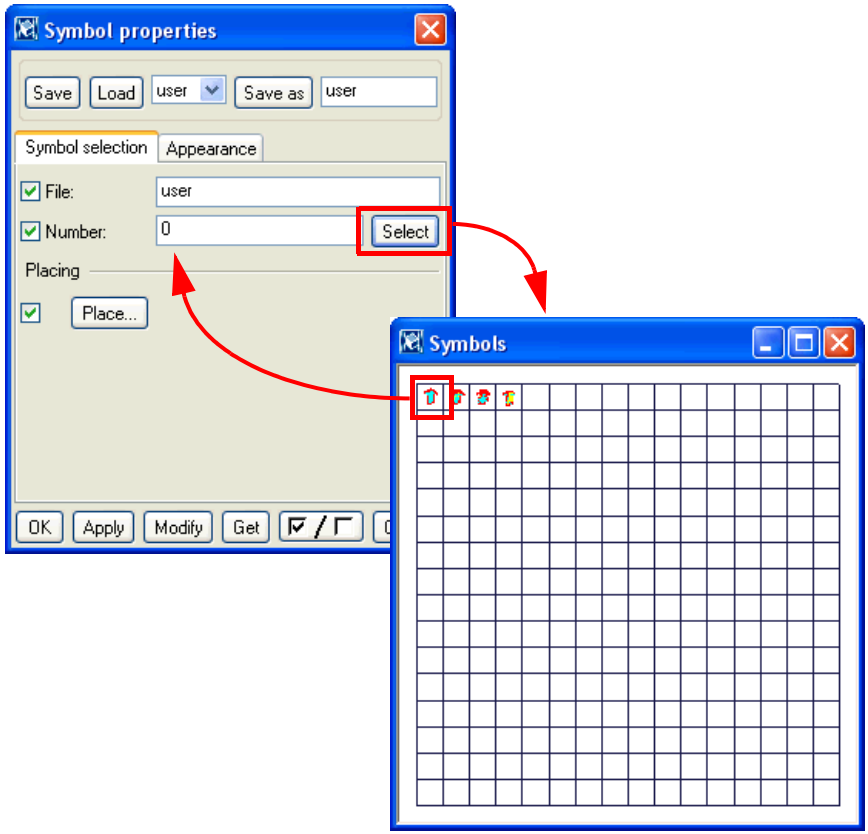


3. On the left part of the dialog are the current symbols in the “user”-library. A library file can contain 16 x 16 = 256 symbols. A symbol can be opened by double-clicking. The symbol will appear on the right side and can be edited.

An existing symbol can be copied to an empty box, so that it can serve as a basis for a new symbol.

Inserting a symbol

1. Open a drawing
2. Go to **Properties** > **Symbol...** the dialog below appears

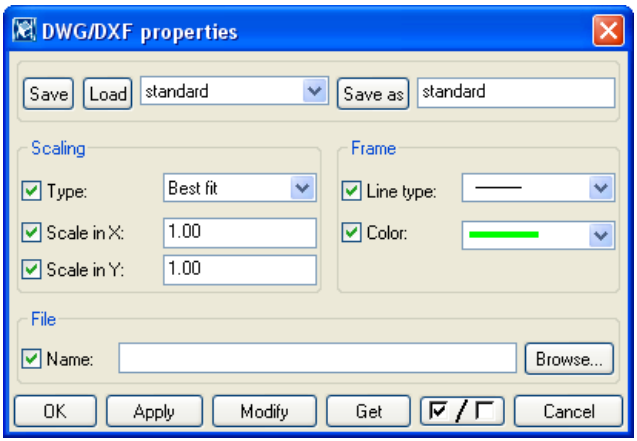


3. At input field **File**, the name of the file has to be entered (in this example: user), this name has to match the filename in the folder **disk:\TeklaStructures\version\environments\country-independent\symbols**, but without entering the extension ***.sym***
4. Click <**Select**>, the next dialog appear
5. Click a symbol you want to insert. The number of the symbol will automatically be filled in input field “Number”
6. Possibly, change the other properties in the dialog, like tab **Appearance**
7. Close the dialog, click **<Apply>** and **<OK>**
8. Go to **Create** > **Symbol** and select one of the options. Make sure that all icons are switched on at the toolbar **Snap settings** to add the symbol on any position you want

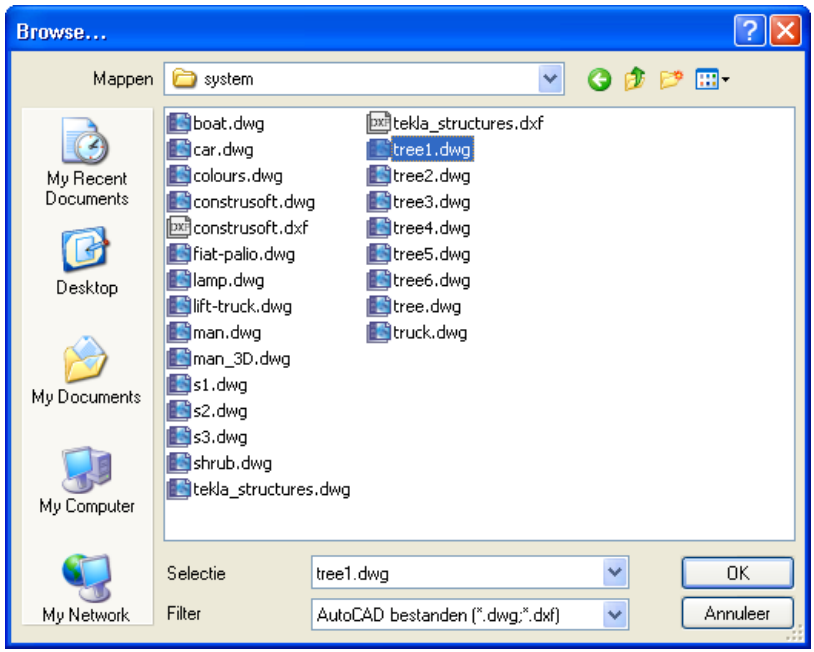
9. Pick a point on the drawing, the symbol will be placed on that position
10. The properties of the symbol can be changed by double clicking the frame around the symbol or double clicking a line of the symbol
11. To move the symbol, select the symbol and drag it to another position

Inserting a DWG/DXF file

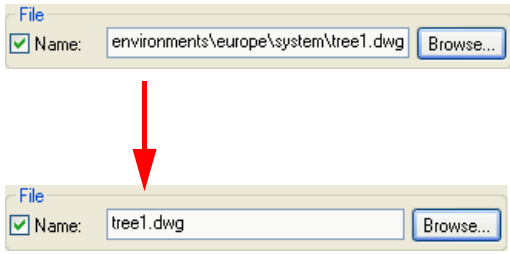
1. Open a drawing and go to **Properties** > **DWG/DXF..**
2. The dialog below appears:



3. You can edit the properties in **Scaling** and **Frame**
4. At **File**, you can browse to locate the DWG/DXF file in the Explorer, then click <**Browse**>



5. Select the file you want to insert and click <**OK**>
6. Copy or move the file to the model folder and remove the path. This way no problems in finding the file can occur, for example when the model is send to a third party like a customer



7. Go to **Create** > **DWG/DXF**
8. Make sure that all buttons in the **Snap settings** toolbar are switched on, to position the dxf/dwg-file at any position in the drawing and click two points
9. The properties of the file can be changed by double clicking the frame
10. To move the dxf/dwg, select the file and drag it to another position

2.9 Lengthen, shorten, split and divide graphical objects



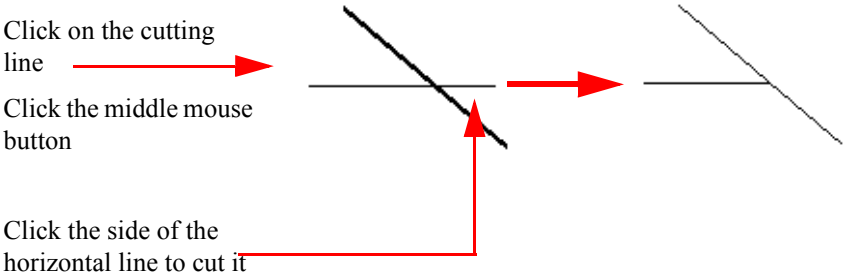
In Tekla Structures use the functions on the toolbar **Drawing: Edit** or in the pull-down menu **Edit** > **Trim**, **Split** and **Divide** to modify graphical objects relative to a boundary. The boundary does not have to be a line. For example, it can be a part, arc or rectangle.

Cutting



To cut a line:

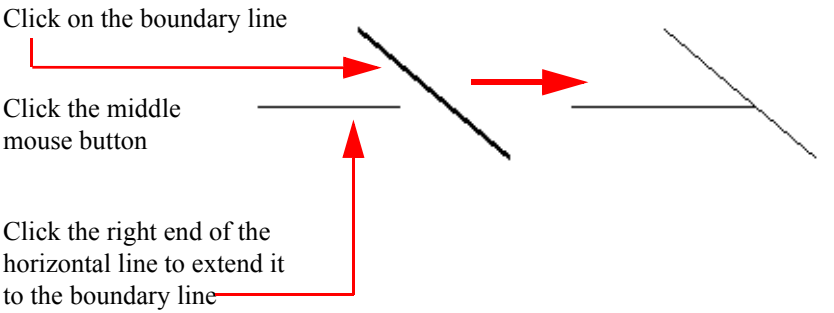
Click **Edit** > **Trim**



Extending

To extend a line:

Click **Edit** > **Trim**



Splitting



To split a line:

Select the line



Click **Edit** > **Split**

Pick a point to indicate the cutting location



Tekla Structures splits the line into two lines

Divide



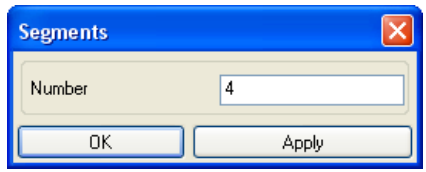
To divide a line into four:

Select the line



Click **Edit** > **Divide.** The dialog box **Segments** appears:

Enter “4” and click **OK**



Tekla Structuresdivides the line into four equal lines



2.10 Step plan for creating a Multi drawing



Multi drawings are drawings with a number of single part-, assembly- or cast-unit drawings put together. There are several methods of generating Multi drawings.

For all methods counts that the properties for the Multi drawing have to be set first.

- Go to **Properties** > **Multi drawing**
- Load the setting “plates” and click <**Apply**> and <**OK>**

Start with an empty drawing

1. Go to **Drawing** > **Multi-Drawing** > **Multi-Drawing**

An empty drawing is generated. Follow the instructions to fill the blank sheet:

2. Open the multi drawing from the drawing list
3. Select the drawing(s) which you want to copy to the multi drawing
4. Go to **Create** > **Link drawing**. The views are placed on top of each other on the drawing.
5. The views again can be re-arranged with command **Edit** > **Place views**. It can happen that some views do not fit into the drawing. In that case, modify the drawing size or decrease the amount of views (the highest numbers).
6. A (possible) part list is not updated right away. This is done when the drawing is saved and closed.

Select drawings which you want to collect on a multi drawing

1. From the **Drawing list**, select the drawings which you want to copy on the multi drawing
2. Go to **Drawing** > **Multi drawing** > **Selected drawings** or click the icon above, the multi drawing will be generated now. On this drawing are copied the drawings you selected in the drawing list

Select parts in the model which you want to collect on a multi drawing

1. Select parts in the model which you want to collect on a multi drawing
2. Go to **Drawing** > **Multi drawing** > **Selected parts** > **Single-part drawings**, the multi drawing(s) are generated. On this drawing are copied views from the to the model parts associated drawing views

The multi drawings are recognizable in the drawing list with a **M**. The **M**-character stands for ***M*ulti**.

The name of the drawing can be changed afterwards. After changing the name, save the drawing and close it.

Add views to an existing multi drawing

As a result of auto-generating multi drawings, it can happen that the final multi drawing contains a single part or assembly only. With a little effort it could be possible to add this part/ assembly on the penultimate multi drawing. The procedure how to achieve this:

1. Open de last multi drawing
2. Remember which assembly- or part numbers are placed on this drawing
3. Open the penultimate multi drawing
4. Open the drawing list
5. Select the assembly-, single part- or cast-unit drawing(s) which have to be linked to the opened multi drawing
6. Go to **Create** > **Link drawing**, the selected drawing(s) are placed on top of each other on the down-left side of the drawing
7. Go to **Edit** > **Place views**, all views will be re-arranged

See also

If the views do not fit properly on the multi drawing, they can be moved manually, see **Move views** (p. 16)



Never use a assembly-, single part- or cast-unit drawing for collecting purposes! (using **Link drawing**). Lists on the drawing will not be filled properly!

When collecting drawings, always use a Multi drawing!

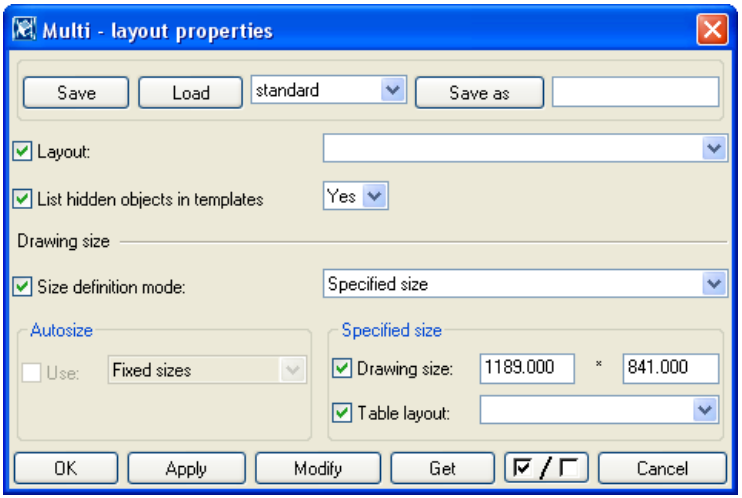
If you want to collect more then one part on a drawing, always use a Multi drawing to do this!

Auto-sizing multi drawings

In the layout dialog for multi drawings, among other things, the drawing size can be specified.

If listbox **Size definition mode** is set to **Fixed size**, a drawing size can be specified.

If **Size definition mode** is set to **Autosize**, Tekla Structures will determine a size (max. **A0**). Tekla Structures can not autosize if the listbox **Layout** is left blank, see picture below:



2.11 Storing Drawing settings

If the drawing setting is changed, then this setting can be saved so that it can be used in other Tekla Structures projects.

- Be sure that all sub dialog boxes are checked, if not, check them with the button 
- Click the <**Apply**> button once
- Now there are two ways of saving the setting as a file:
  - By clicking the <**Save**> button. The current setting in the "settings" listbox (right next to the <**Load**> button) will be overwritten.
  - Enter a unique name in the input field right next to the <**Save as**>-button, then click the <**Save as**>-button.



Do not use spaces in setting names!

The file that has been created will be stored in the model folder, in the **Attributes**-folder. Currently, the setting can only be used in the active project.

If you want to use the setting in other projects too, you need to copy the setting file to the Tekla Structures system folder:

disk:\TeklaStructures\version\environments\europe\ts

File extensions of drawing settings

Single part drawing	***.wd**
Assembly drawing	***.ad**
GA-drawing	***.gd**
Multi drawing	***.md**
Cast unit drawing	***.cud**

2.12 Applying a new drawing setting to an existing drawing

It can happen that you created a new drawing setting and you want this new setting apply to existing drawings. It is not necessary to delete and recreate all drawings.

Step plan

- Go to **Properties** > **Cast unit- or assembly drawing** and load the new setting. Let the *Cast unit- or assemblydrawing Properties*-dialog stay open.
- In the drawing list, select the drawings you would like to change.
- In the **Cast unit- or assembly drawing Properties**-dialog, click <**Modify**>.

The selected drawings will now be modified according to the new drawing setting.

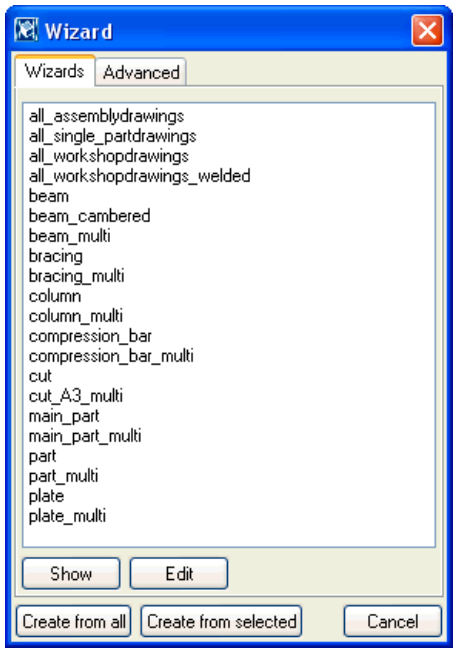
This procedure also works for Single Part drawings.

2.13 Wizard



The **Wizard** is used to generate single part-, assembly- and cast unit drawings and multi single part- and multi assembly drawings automatically. However, GA-drawings have to be made manually.

To generate drawings with the Wizard, go to **File** > **Wizard**, click the icon opposite, or use shortcut **W**, the following dialog box will appear:



The wizard-files are small programs. In a wizard file can be set which drawings have to be made and with which settings they have to be made.

Step plan

1. Be sure that the model is fully numbered
2. Select the parts where drawings have to be generated from
3. Open the Wizard dialog box with the icon, via **File** > **Wizard** or the shortcut
4. Select a wizard and click on the <**Create from selected**>-button once
5. Generation of drawings will start

or

1. Open the Wizard dialog with the icon, via **File** > **Wizard** or a shortcut
2. Select a Wizard and click on the <**Create from all>**-button once
3. Generation of drawings will start



It can be useful (especially for bigger models) to increase the value for "Autosave" to i.e. 1.000.000 in tab *General* in the pull-down menu **Tools** > **Options**

Generating drawing will go faster because less Autosave files are created!

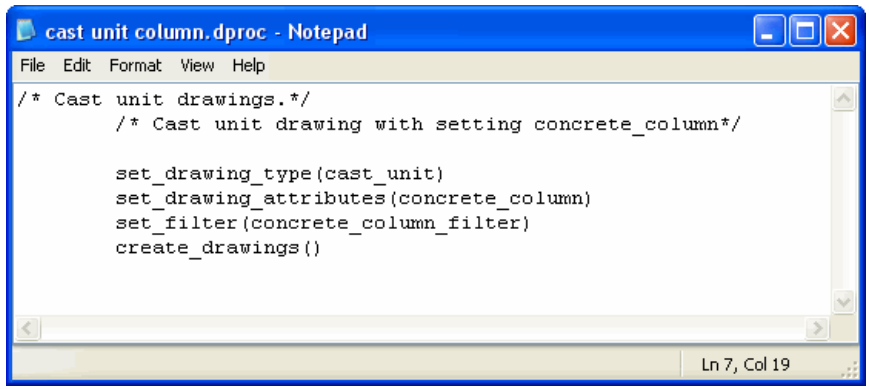
Let's take a look at such a Wizard-file.

Example

- Select the **cast unit column**-wizard in the dialog box and click <**Edit>**



If the message ***Could not open wizard file for editing…*** appears, then first link the extension of wizard files to a Text editor, e.g. Notepad. Wizard files have the extension ***.dproc**



- The Wizard file can now be viewed, edited and saved.

Explanation of lines

The text between the slash-characters (/ * *) will not be used. These are only for more clarification for the user.

1st line: Here is set which drawing type will be generated. The options are: **cast_unit** (cast unit drawing) or **multi_cast_unit** (multi drawing with cast unit parts)

2nd line: Here is set which drawing setting will be used for generating the drawings.

If **cast_unit** is set in the 1st line, then the Wizard will search for a setting file with the extension **cud**.

For multi-drawings a file with the extension ***.md** will be used. In the example above, the setting **cast_unit.cud** will be used.

3rd line: Here is set which filter is going to be used. The filter makes that the right parts are selected.

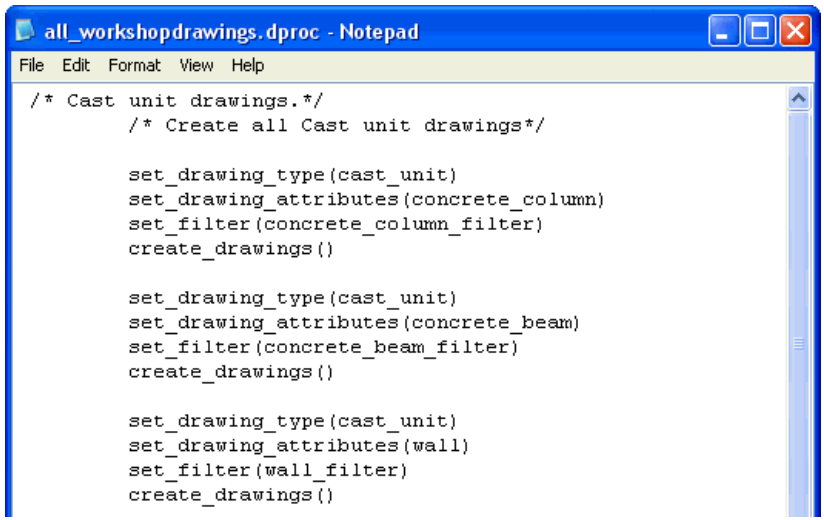
Filters can be created and edited in **Setup** > **Select filter**. Filter files have the extension ***.msf**.

4th line: Starts the generation of drawings. This line is always the same.

Summary creating a new Wizard

- Create a new drawing setting
- Open the Windows Explorer
- Copy the new file from the model-folder to the folder **disk:\TeklaStructures\version\environments\europe\ts**
- Open an existing wizard file (***.dproc**; also in system folder) in a Notepad
- Edit the text, save the file with an unique filename
- The new Wizard-file is added to the list with available Wizards

Below there is a section of the wizard named **all_cast unitdrawings**. This wizard is set up in a way that drawings are generated from all parts, no parts can be left out by this wizard.



Exercise

Create a wizard named **tube** that fulfils the following conditions:

- the wizard must generate single-part drawings
- the contour lines of the parts on the drawings must have a green color, the dimension lines must have a red color
- the Wizard should only create drawings of parts with the name “TUBE”.

Of course, this setting should be available in other projects, so the wizard file must be copied to the folder:

disk:\TeklaStructures\version\environments\europe\ts

2.14 Drawing list

General

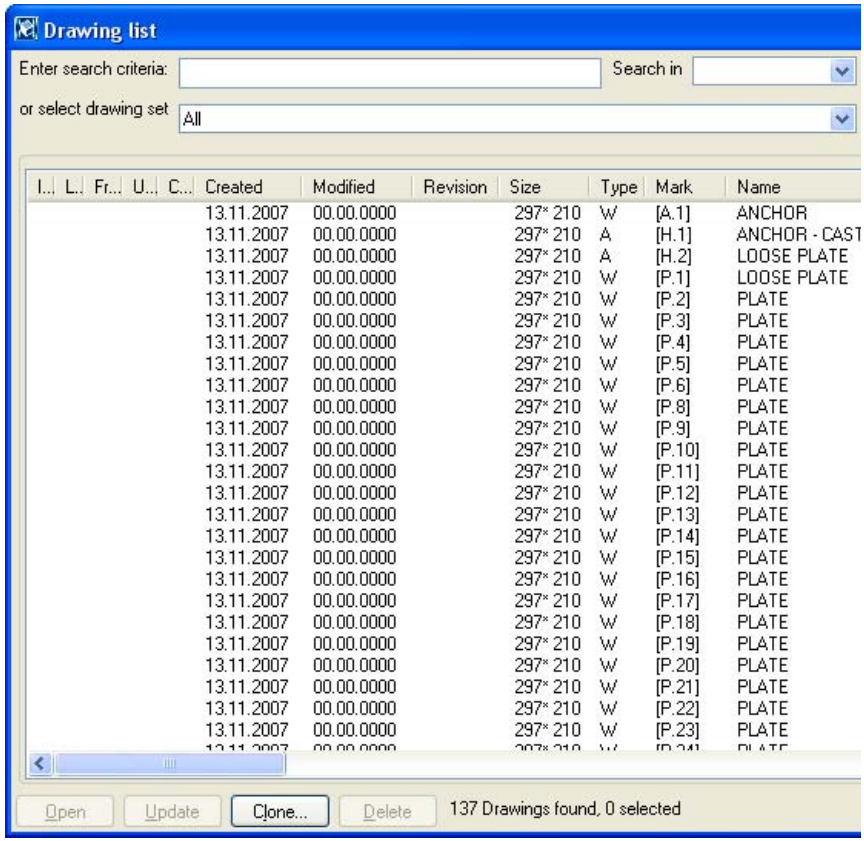
In the drawing list you can open, update, edit, freeze, lock, clone, and delete drawings. You can also sort, select, and display drawings according to different criteria, and print lists of certain drawings, for example.

You can also use the drawing list to find the connections between the drawings and the parts in the model.

The buttons on the right of the **Drawing list** dialog box are used to control the drawings in the dialog box.



The dialog box can be opened by clicking the icon or via **Drawing** > **Drawing list:**



Selecting drawings

Selecting all drawings

You can easily select all drawings displayed in the drawing list at once, for example, to make a report by pressing <**Ctrl**> + **A** when the drawing list is selected, or by right-clicking the list and selecting **Select all** from the pop-up menu.

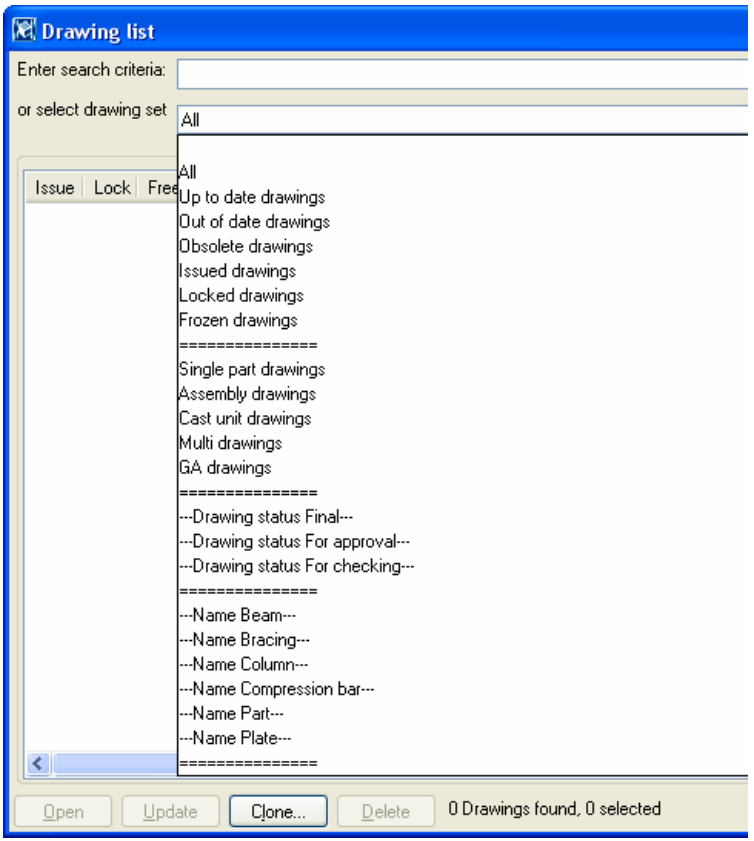
Select multiple drawings

To select multiple drawings in the drawing list:

- Select a drawing
- Hold down the <**Shift**> key
- Select the last drawing

Selecting drawings from predefined sets

You can now select to display drawings from various predefined sets. To do this, select an option from the **or select drawing set** dropdown list.



For example, if you wish to see general arrangement drawings only, select **GA drawings** from the or select drawing set dropdown list.

Drawing status

You can use the input field **Drawing status** in the user defined attributes in the drawing.

For this, variable **DRAWING.USERDEFINED.DR_STATUS** is used.

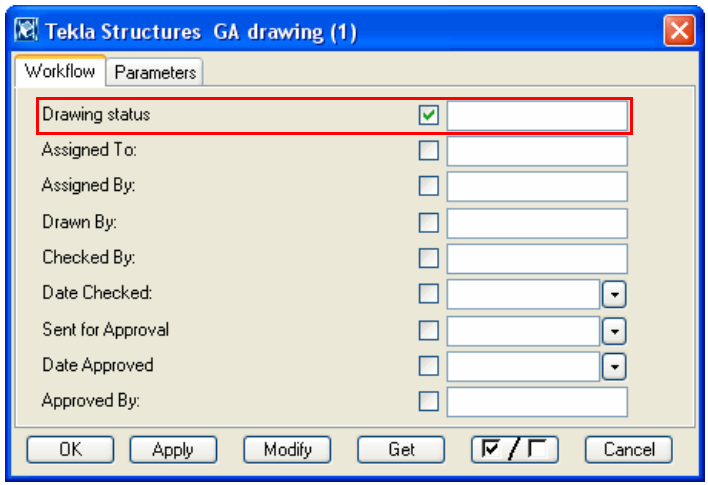
You can use this variable to indicate which drawings, for example, have been send to an architect: you can enter any random text.

The status information is displayed in the column **Drawing status** in the drawing list, you can select on this.

You need to add select filters to the drawing list by yourself, by default the select filters `Final`, `For checking` en `For approval` are added to the drawing list to display and select drawings.

To assign a status to one or more drawings, select one or more drawing(s), i.e. several GA drawings and open the **User defined attributes**.

For this, click on the right mouse or use the shortcut <**Alt**> + **U**, the following dialog box appears:



Make use of the button ☑/☐ to switch off all checkboxes in one go, now check the field **Drawing status**.

Enter a status in the field **Drawing status** in the **User defined attribute** for the selected drawing(s), for example `For checking`.

The entered status information is **not** case sensitive but per select filter a condition is used:

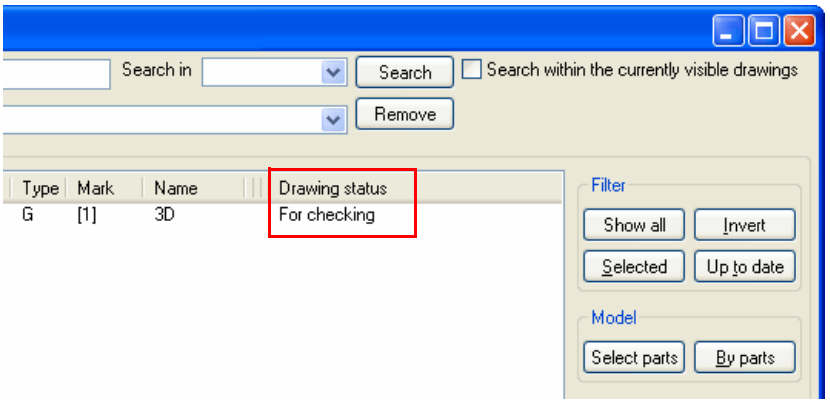
For checking So the select filter `For checking` must at least contain the text **check**. This means that you can also enter **forchecking** but not **For Ch.**.

Final The select filter `Final` must at least contain the text **final**.

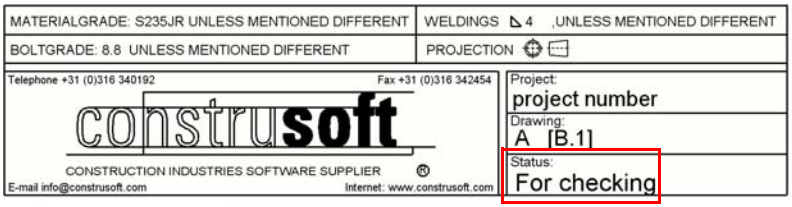
For approval The select filter `For approval` must at least contain the text **approval**.

Click <**Modify**> and <**OK**>.

The status information will be displayed in the column **Drawing status** in the drawing list, you can select on this by using the select filter `For checking`, for example:



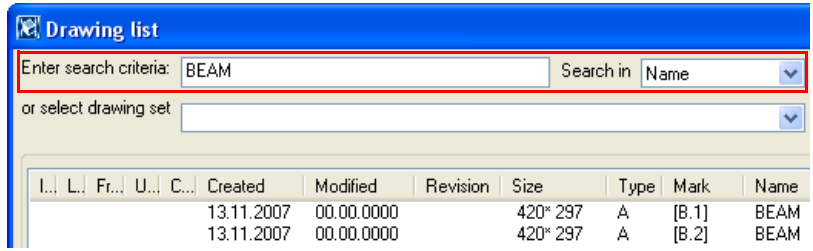
Template_A In the drawing, the drawing status will be displayed in **template_A** automatically:



Searching and using search groups

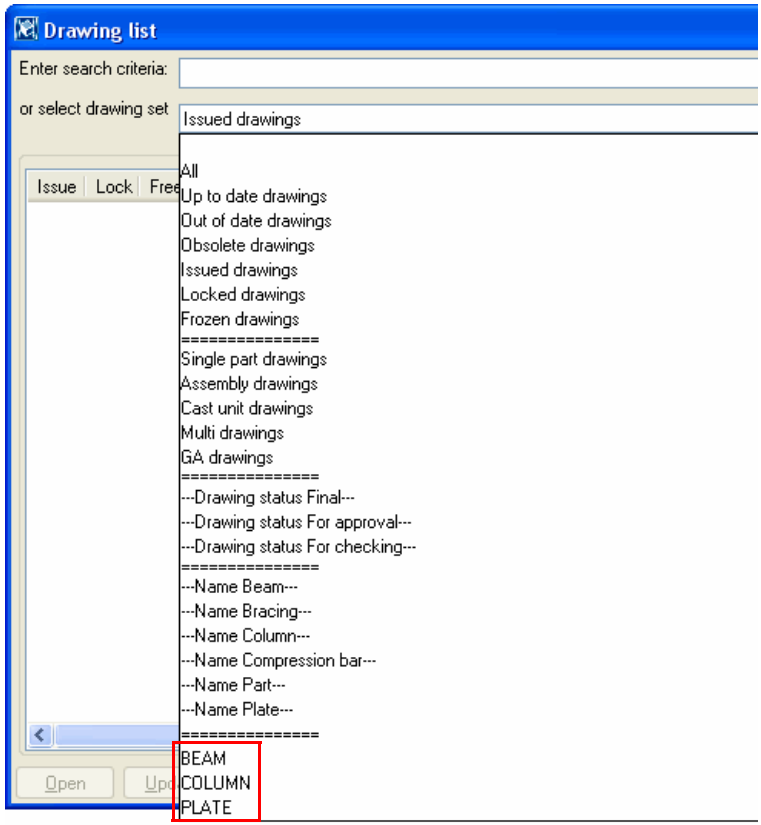
Search You can use searching to find drawings quickly. You can now quickly find the drawings you need using text-based search by entering your criteria in the **Enter search criteria** field. You can limit the searches by columns by entering the name of the column in the **Search in** field.

To limit the search to the currently visible drawings, select the option **Search within the currently visible drawings**.



Save the search results

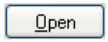
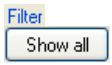
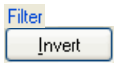
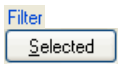
You can save the searches you have made. To do this, click the <**Store**> button and enter a name in the **Store search result** dialog box. The searches are saved in the DrawingListSearches in the model folder. After you have saved the search and re-opened the dialog box **Drawing list**, it shows among the predefined drawing sets.

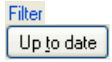
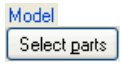
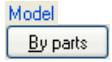
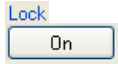
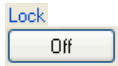
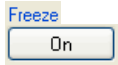
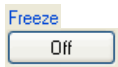
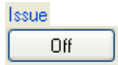
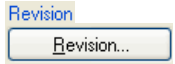




Only searches can be saved. The button <**Store**> only appears in the dialog box **Drawing list** after a search command.

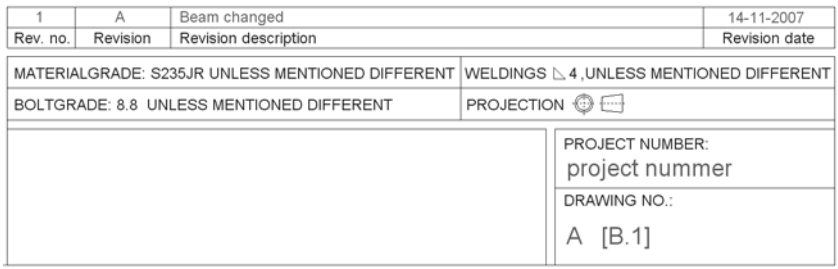
Explanation of buttons in the Drawing list dialog box

Button	Description
	Opens a selected drawing from the drawing list. Double-clicking a drawing has the same effect. After opening a drawing, Tekla Structures switches to the drawing environment. Only one drawing can be open at a time. If more than one drawing is selected, the **Open**-button turns gray and can not be used.
	Updates the drawing after model modifications. Frozen drawings can be updated also. Locked drawings can not be updated.
	See **Cloning drawings** (p. 103)
	To delete selected drawings.“Locked” drawings can not be deleted!
	Button to find drawings. Enter your criteria in the **Enter search criteria** field. You can limit the searches by columns by entering the name of the column in the **Search in** field. To limit the search to the currently visible drawings, select the option **Search within the currently visible drawings**.
	Button to delete the search results.
	Button to save the searches you have made. Click the <**Store**> button and enter a name in the **Store search result** dialog box. After you have saved the search and **re-opened** the dialog box **Drawing list**, it shows among the predefined drawing sets.
	Button to list all existing drawings
	Button to invert the contents of the current list. First a drawing selection has to be isolated with the **Filter** > **Selected** or **Filter** > **Up to date**-buttons
	Button to isolate selected drawings. The other drawings can be displayed with the **Invert**-button

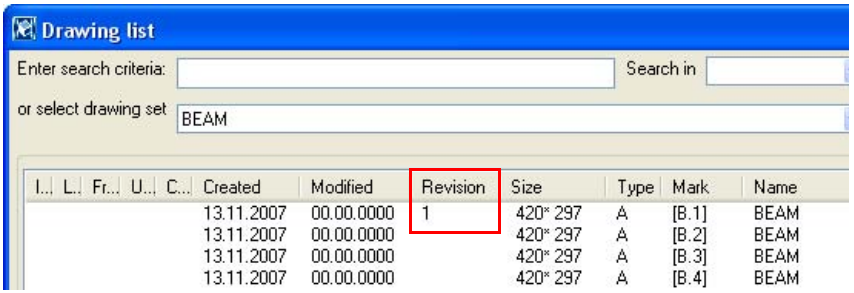
Button	Description
	Button to isolate drawings which need to be updated. These drawings can then be displayed by clicking the **Invert**-button.
	Button to highlight parts in the model which match with the selected drawings in the drawing list.
	Button to filter drawing in the drawing list which match with the selected parts in the model.
	Button to lock drawings. When a drawing is locked, it is impossible to open the drawing. De “update”-marks (N and P) can also be put in front of locked drawings. Locked drawings can not be deleted, but they can be printed.
	Unlocks the selected drawing
	See **Associative drawings** (p. 94)
	See **Associative drawings** (p. 94)
	Button to give drawings an Issue status. An Issue-flag does not prevent the user from plotting, deleting or opening the drawing. It must be interpreted as a reminder, for example for already plotted drawings.
	Removes “I”-flag
	Allocating a revision to a drawing (or multiple drawings). Procedure; first select drawing(s) which need the same revision. Press the *Revision*-button. Fill in the dialog, at least the input fields *Mark*, *Date* and *Text* have to be filled in. Next, press the *Create*-button once! The number of revision that have been applied to a drawing can be seen in the drawing list (column “Rev”.)

2.15 Drawing revisions

When you revise drawings, you can attach information to them about the changes you have made. Tekla Structures displays the information about the revision number, the revision mark and the description on the drawing:



The revision number is also shown on the drawing list:





You can choose whether Tekla Structures shows the revision numbers or the revision marks on the drawing list. By default, it shows the revision numbers.

You can set the variable `XS_SHOW_REVISION_MARK_ON_DRAWING_LIST` to **1** in **Tools** > **Advanced options** > **Drawing properties** to see the revision marks.

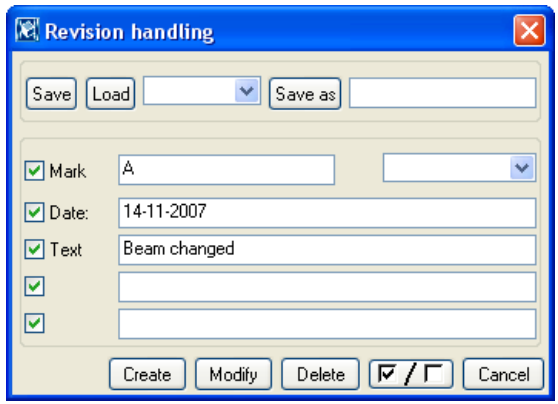
The reports **drawing_revision_list** and **drawing_last_revision_list** show the drawings revision date.



To attach the same revision information to several drawings simultaneously, select multiple drawings from the drawing list.

Creating revisions

- In the Drawings dialog box, select the drawing(s) to revise
- Click <**Revision>,** the **Revision handling** dialog box appears:



- You must enter a mark and can choose to enter a date for the revision. You can also add 3 lines of Text about the changes, then click **<Create>**

Changes revisions

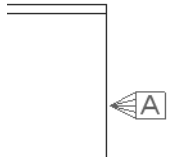
- In the Drawings dialog box, select the drawing(s) to revise
- Click <**Revision>**
- In the **Revision handling** dialog box, select the number of the revision in the list box next to the **Mark** field
- Change the revision information and click <**Modify>**

Deleting revisions

- In the **Revision handling** dialog box, select the revision number in the list box next to the **Mark** field.
- Click <**Delete>**. When you delete a revision, Tekla Structures automatically adjusts the remaining revision numbers for that drawing. The revision marks do not change.

Revision marks on drawings

- Click **Properties** > **Revision mark...** to open the **Revision mark properties** dialog box.
- Enter the mark, date, and information on the changes.
- Modify the other revision mark properties as required.
- Click **<OK>** to save the properties and exit the **Revision mark properties** dialog box.
- Click **Create** > **Revision mark** and select an option.



- Repeat these steps to add revision marks elsewhere in the drawing.

2.16 Associative drawings

What is associativity?

All drawing types in Tekla Structures are associative, if they are created manually or automatically.

This means that drawing objects (labels, dimensions, welds, views, cut symbols, angle dimensions, level marks and associative notes) are linked to model objects. Because of that, drawing objects are updated automatically if the model changes!

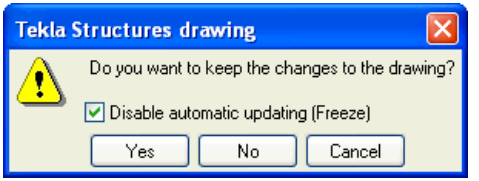


In drawings, a nail symbol indicates an object to be associative. This symbol will not be printed. To hide associative symbols, click **Setup** > **Associative symbol** or use shortcut <**Shift** > + **A**.

Freeze

If you modify and save a drawing, Tekla Structures ask if you want to save the changes. Click <**Yes**>.

To prevent that a drawing will be updated automatically, check the option **Disable automatic updating**:



This will **freeze** the drawing. Freezing drawings causes drawing objects to be not associative anymore:



Drawing not frozen --> Associative --> Updated automatically

Drawing frozen --> Not associative --> Not updated automatically

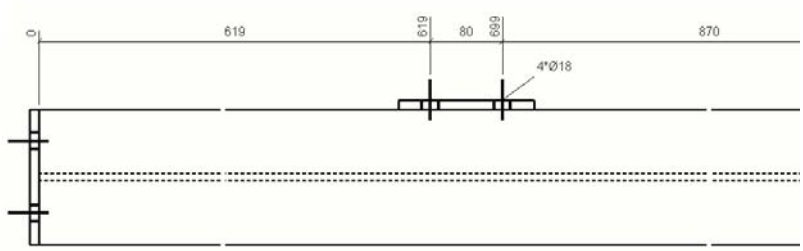
Manual changes

In Tekla Structures, manual modifications will **always** be kept, no matter if the drawing is frozen or not.

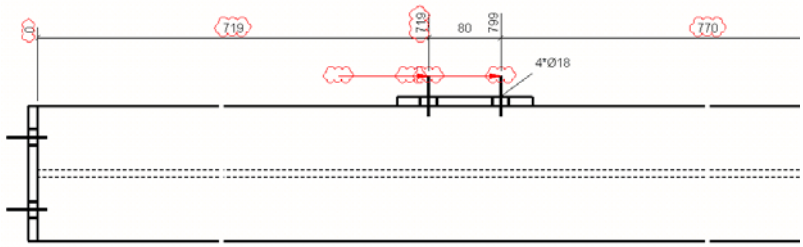
Example assembly drawing

Drawing is not frozen

Before modification:

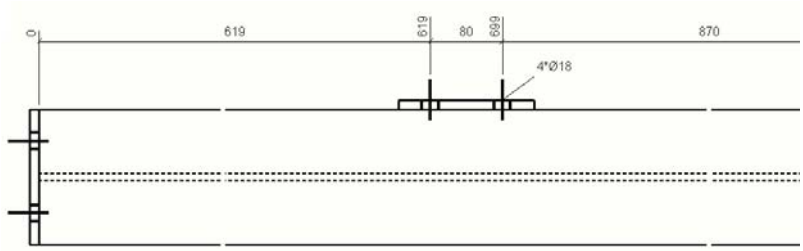


After modification:

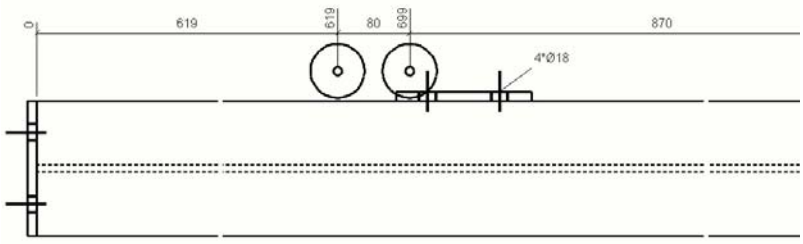


Drawing is frozen

Before modification:



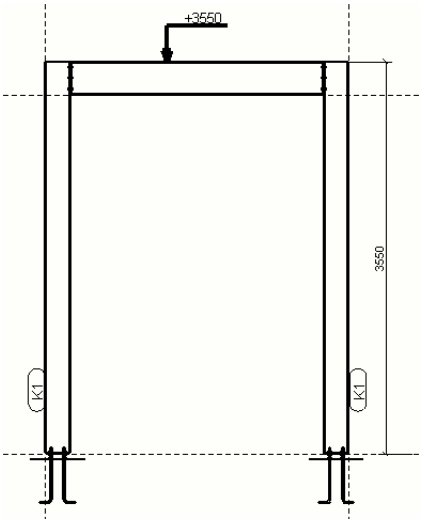
After modification:



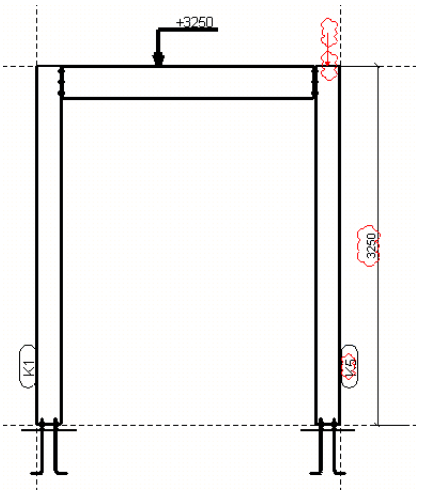
Example General arrangement drawing

Drawing is not frozen

Before modification:

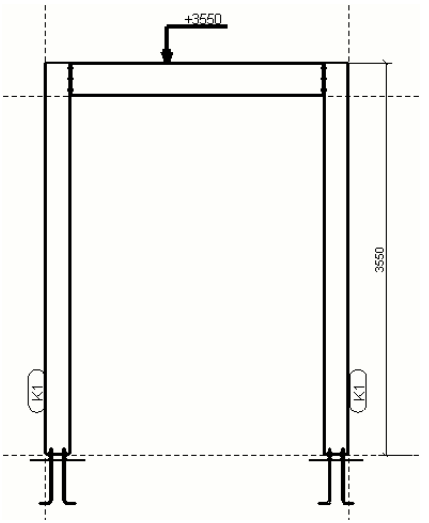


After modification:

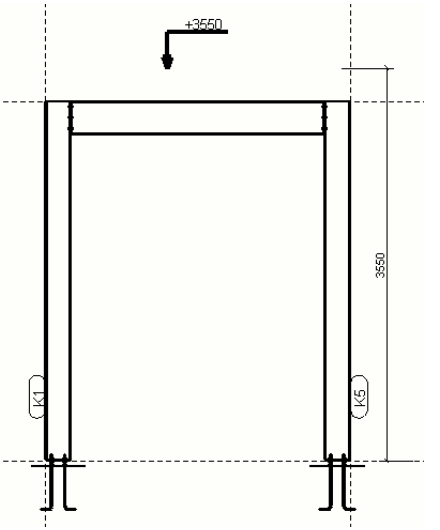


Drawing is frozen

Before modification:



After modification:



You see the level mark and dimension line are not updated, also the view size has not been changed.

2.17 How to handle model modifications

After modifications in the model, one or more drawing may need an update:

Step plan

1. Renumber the model, go to **Tools** > **Numbering** > **Full**
2. Open the drawing list; affected drawings get a symbol in the column **Up to date**
3. Click the **<Show all>** button, all drawings which be displayed
4. Click the **<Up to date>** button, all up-to-date drawings which be displayed
5. Next, click **<Invert>**, the drawing list now consists only of all drawings which need to be updated. The drawing list now consists only of drawings with a symbol in the column **Up to date**
6. Possibly, assign a revision to the updated drawing(s)
7. Click the button <**Update>**, the selected drawings will be updated now

A modified part has

- or retained it's part number. This can happen only if the part is a unique part which only exists once in the model
- or has received a new part number; the parts which were identical (but which are not modified) will remain their part number! This new part will get a new part number. A drawing has to be made of this new part!

A modified assembly has

- or retained it's assembly number. This can happen only if the assembly is a unique assembly which only exists once in the model
- or has received a new assembly number; the assemblies which were identical, will remain their assembly number! This new assembly will get a new assembly number. A drawing has to be made of this new assembly.

8. Click the button **<Show all**> and <**Invert>** in the drawing list, the drawing list does not show any drawings at all currently
9. Open the Wizard and click **all_workshopdrawings**, now click <**Create from all**>

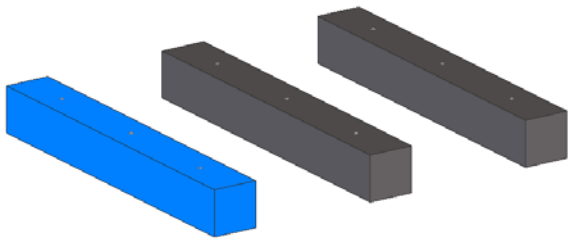
Tekla Structures recognizes which drawings are already present and will **only** create drawings of new parts/assemblies. These new drawings will be displayed in the drawing list

New drawings can be identified by the date in the column **Created** in the drawing list). In case there are more drawings with the same date, and it is difficult to recognize which drawings are the newest, a log file can give solution. (Go to **Tools** > **Display Log file** > **Drawing history log**). This log shows all actions in the current project which have to do with drawings.

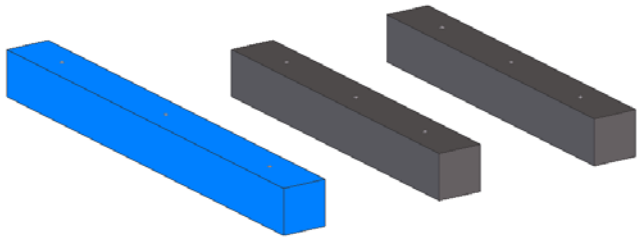
Example

When the original part of a drawing has changed and moved to another serie, the existing drawing will not be deleted, but automatically assigned to another part of that serie. When the modified part is moved to a serie without drawings, the original drawing will be cloned.

1. You have 3 identical parts in the serie **B1** and you have created a drawing **B1** from one part (The part is actually linked to that drawing):



2. You now modify this part so that this part is no longer equal to the other parts:



3. After numbering, drawing **B1** gets an symbol, which indicates that the part from which the original drawing was created, is deleted.
4. When you update the drawing, the second part in the serie will be linked to the drawing and the an symbol will disappear.
5. What happens to the modified part? If no other identical parts have a linked drawing in the model, drawing **B1** will be cloned to display the changes in the modified part and the drawing gets an symbol in the drawing list.

6. When there are other identical parts in the model and also have a drawing, this drawing gets an  symbol. This means that the drawing is up to date, but the number of parts needs to be updated.

Indicate changes in drawings

In changed drawings, modified dimensions and dimension lines are indicated through clouds as a change symbol if you update associative drawings (single part-, assembly- and general arrangement drawings).

These clouds only appear in **not** frozen drawings (associate) drawings.

These clouds are not printed.

Printing clouds

Clouds are displayed in the color **Red** on your drawings, because of that the clouds are automatically included in printed drawings. You can change the color using the variable XS_DRAWING_CHANGE_HIGHLIGHT_COLOR in **Tools** > **Advanced options...** > **Drawing properties** and enter the color (RED, BLACK, GREEN, BLUE, GREY).

If you use the color **magenta**, the symbols are displayed on the screen but they will **not** appear in the printed drawings.



To remove clouds from part labels, go to **Edit** > **Remove mark change symbol** > **Single change symbol** or **All**, or click an icon.



To remove clouds from dimensions, go to **Edit** > **Remove dimension point change symbol** > **Single change symbol** or **All** of click an icon.

The 4 mentioned icons for removing clouds are assimilated in toolbar **Drawings-Dimension** and are only available if you execute the correct Registry file.



See also

For more information about executing the registry file, see **Tekla Structures.pdf** in folder: **disc:\TeklaStructures\version\environments\europe\pdf.**

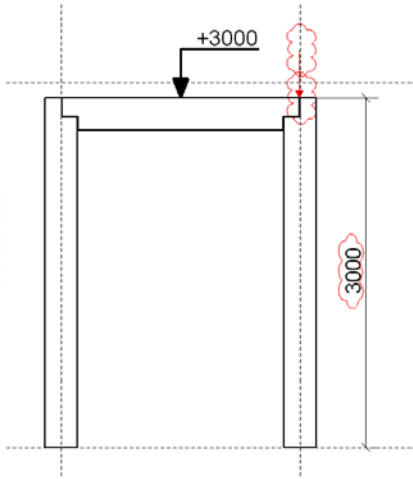
Updating a frozen drawing

When a general arrangement drawing is frozen, you can **unfreeze** it, and then re-open it to automatically update the drawing with recent changes.

To re-enable automatic updating in a frozen drawing:

1. Select the drawing in the Drawing list
2. Right mouse click and select **Freeze** > **Off**.

Now, clouds are placed to indicate the modifications



Associativity regarding to ID-numbers

The associativity on drawings in linked to ID-numbers. In case you move a part, and it's accompanying ID-number, this change will be noticed on the associative (not frozen) drawing and the drawing (dimension line, mark, etc.) will be edited.

In case you add some parts to the model, the ID-numbers of these parts are unknown in the drawing, because of that no dimension lines will be added for those parts. Depending on the situation, the drawing can be recreated or the dimension line can be added manually.

In case of changes, some components will be recalculated. It looks like some parts of the component change, in fact existing parts will be deleted and new parts will be added, because of that new ID-numbers are distributed and the link on the drawing is broken. You can now recreate the drawing, add the dimension lines manually or freeze the drawing (the dimension line still exists, but needs to be modified).

In case you only want to move anchor bolts, for example, it is better to explode the component and to move the anchor bolts, the ID-numbers and the associativity will remain on the drawing.

Description of the status symbols and -text in the drawing list

For the status symbols and the additional text for describing the drawing status, see the following table:

Symbol	Column	Status text
	**Up to date**	**Parts modified**, parts have been added or deleted, or part properties have changed: the drawing is not updated **Quantity increased or decreased**, the actual drawing is up to date, but the number of identical parts has changed **Original parts deleted**, the part where the drawing is initially created from, is deleted from the model
	**Up to date**	All relevant objects on the drawing are deleted from the model
	**Up to date**	**Linked drawing changed** **Original parts are deleted, the others exist** **Drawing updated**
	**Up to date**	The drawing is a cloned drawing
	**Issue**	The drawing has been issued. An Issue symbol can be assigned to selected drawings, e.g. when they are already sent to the workshop. So, an *Issue* symbol can be considered as a reminder
	**Issue**	The issued drawing has been edited or otherwise changed
	**Lock**	Drawing is locked, i.e. it can not be opened
	**Freeze**	The drawing is frozen

2.18 Cloning drawings

Modified and finalized drawings can be cloned by use of the function **cloning drawings**.You should consider cloning drawings when:

- There are several similar parts or assemblies in the model
- You need to produce single-part or assembly drawings of similar parts or assemblies
- The drawings need a lot of manual editing

Then you only need to modify the cloned drawings where the parts or assemblies differ. After cloning a drawing, you should always check it to ensure the dimensions and view sizes are correct.

Preferably make use of a drawing from which the part or assembly is most similar to the part or assembly you want to clone to reduce modifying.



Make sure that **no** drawing exists of the part or assembly you want to clone!!

See also

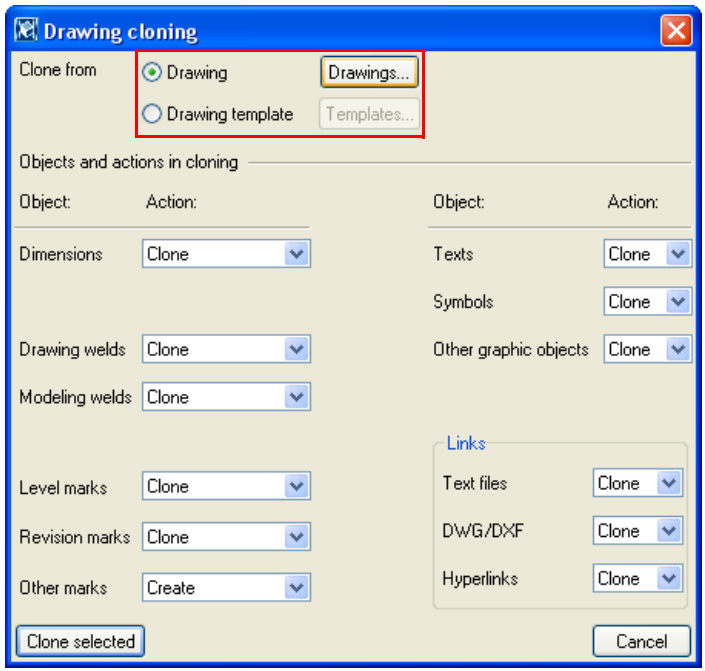
For cloning there are two options:

- You can make use of drawings from an **existing model**, see **Clone from drawing** (p. 104)
- You can make use of a **standard model**, a so-called. drawing template, see **Clone from Drawing Template** (p. 105)

Clone from drawing

Step plan

1. Finalize, save, and close the drawing you want to clone.
2. In the model, select the parts or assemblies to include in the drawing.
3. In the Model Editor, click **Drawing** > **List**... to open the drawing list, select the finalized drawing on the drawing list and click <**Clone**>.



4. Select the option **Clone from** > **Drawing** in the **Drawing cloning** dialog box.
5. Use the **Objects and actions in cloning** options to define the drawing objects to use in the new drawing and click <**Clone selected**>.

Clone from Drawing Template

You can clone drawings by using so-called drawing templates. For this purpose you must define the location of the drawing template library.

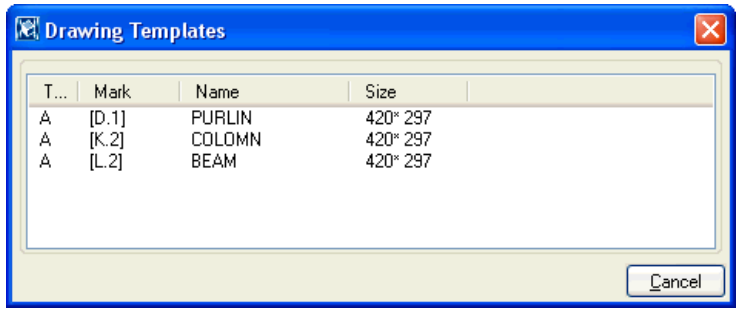
Copy the variable `XS_DRAWING_TEMPLATES_LIBRARY` from the file `env_europe.ini` (in the folder ..*environments**europe*) to the file `user.ini` (in the folder ..*nt**bin*) and point to point to the model folder that contains the template drawings, for example:

```
set
XS_DRAWING_TEMPLATES_LIBRARY=C:\TeklaStructuresModels
\template_model
```

where `template_model` is the modelname.

Step plan

1. Create a template drawing in the folder defined by the variable `XS_DRAWING_TEMPLATES_LIBRARY`.
2. To open the **Drawing cloning** dialog box, click **Drawing** > **Clone drawing...**, or click <**Clone**> in the drawing list dialog box.
3. Use the **Objects and actions in cloning** options to define the drawing objects to use in the new drawing.
4. Select the **Clone from** > **Drawing template** option and click the <**Templates**> button:
5. In the **Drawing templates** dialog box, select the template drawing.



6. In the model, select the part to create the drawing from.
7. Go to the **Drawing cloning** dialog box and click <**Clone selected**>.



Make sure that the drawings in the drawing template are **not** locked!!

See also

For more information, see the **Tekla Structures Online Help**.

2.19 Dimensioning curved beams



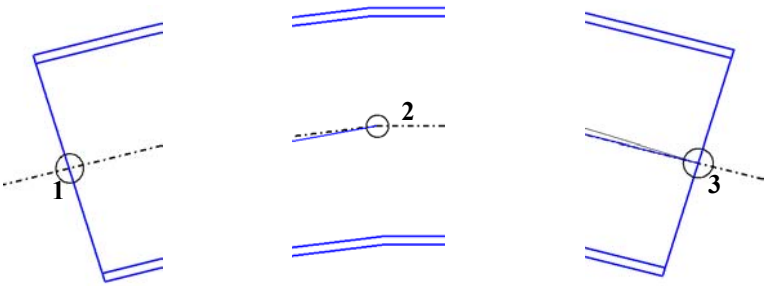
Step plan for radius dimensioning

To create a correct radius- and/or radial dimensions, placing these dimension types has to be done in a particular way.

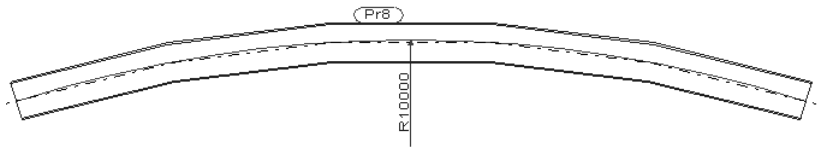
1. Double-click in the drawing on the curved beam and add a center line**.** Close with **<Apply**> and **<OK>**
2. Toggle on <u>only</u> the three icons on the left in toolbar **Snap settings**



3. Click the **Radius dimension** icon, or go to **Create > Dimension > Radius*)***. Now, pick 3 points on the center line (preferably de two outermost points and a point in the middle of the beam):



4. Pick a random position on the center line. The dimension will be placed.



(The second picked point is the point where two center lines of two segments intersect: this is the point which determines the exact radius.)



For better preciseness, check **Xsnap** in pull down menu **Setup** (or press **T**).

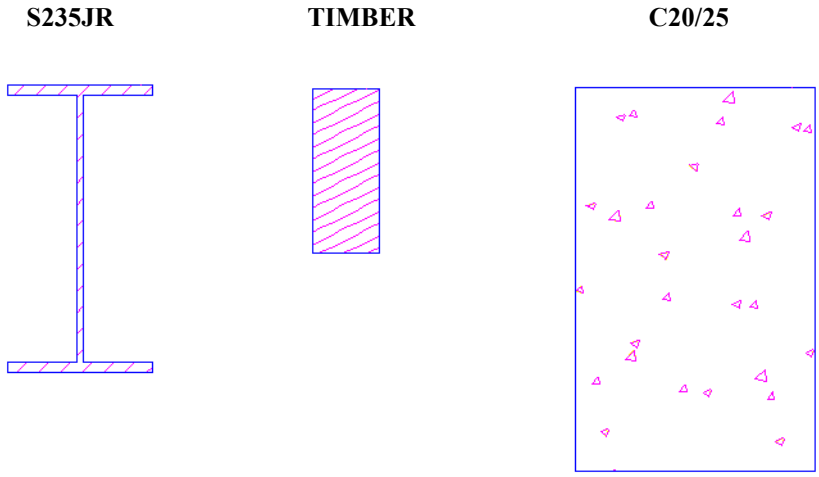
For creating curved radial dimension, after picking three points on the center line of the curved beam and picking two points on the beam to which have to be dimensioned, always close picking points by pressing the middle mouse button.

2.20 Hatching

In Tekla Structures you can hatch automatically and manually.

Hatching automatically

In Tekla Structures, profile sections can be hatched automatically with a specified hatch type. Also the scale and hatch color can be specified.



By default, this feature is disabled and profile-sections will thus not be hatched.

See also

For more information about hatching automatically, see **Tekla Structures.pdf** in the folder: **disc:\TeklaStructures\version\environments\europe\pdf** and the **Tekla Structures Online Help**.

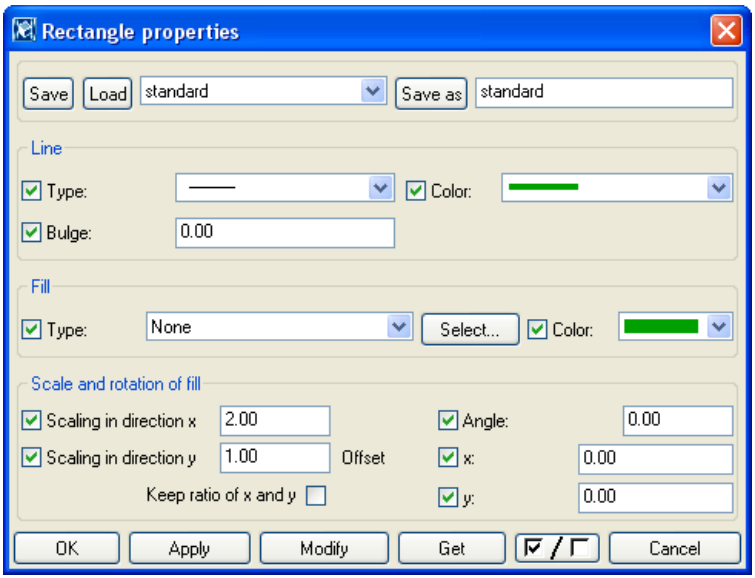
Hatching manually

In Tekla Structures you can hatch both graphical objects (such as rectangular, circles, clouds, etc.) and profile cross sections and part faces as well.

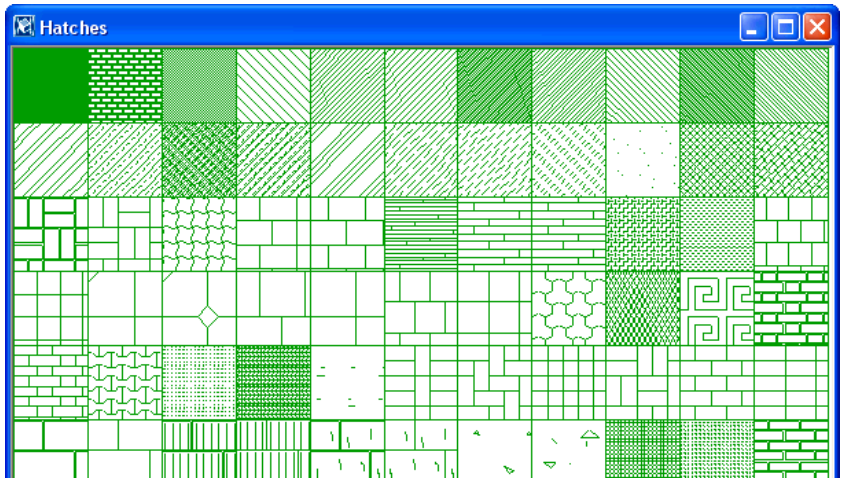
Hatching graphical objects

Step plan

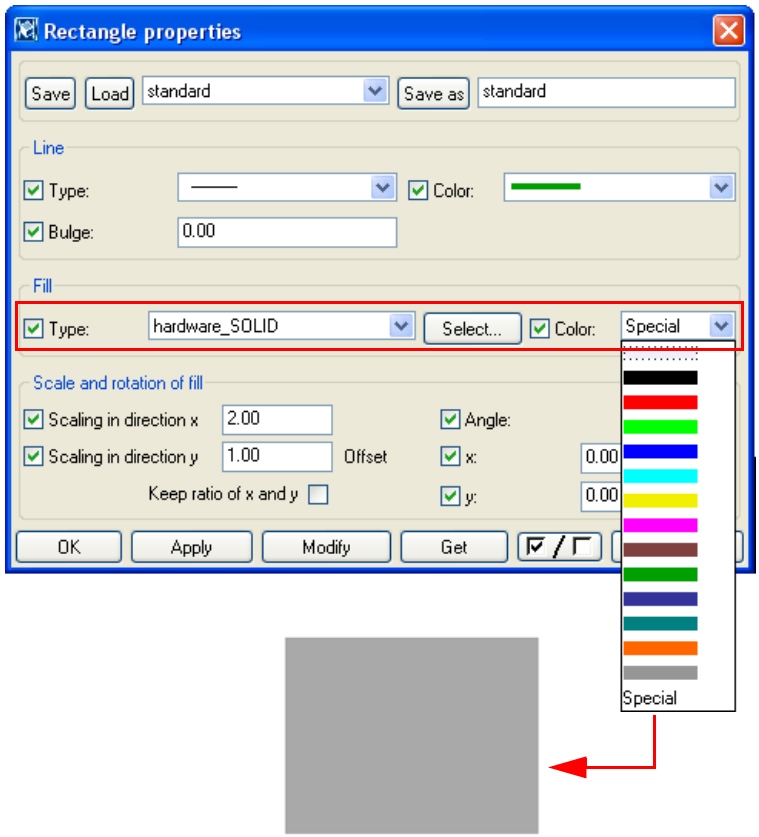
- Double-click on a graphical object, e.g. a rectangular object, the following dialog box appears:



- In the listbox **Type** you can select the pattern or you click on the button <**Select**> to open the dialog box **Hatches** in which all available patterns will be displayed



- Double-click on the pattern you want to use, the dialog box **Hatches** closes automatically.
- If you want to use a gray scale, select the option **hardware_SOLID** as a hatch pattern and **Special** as a color:



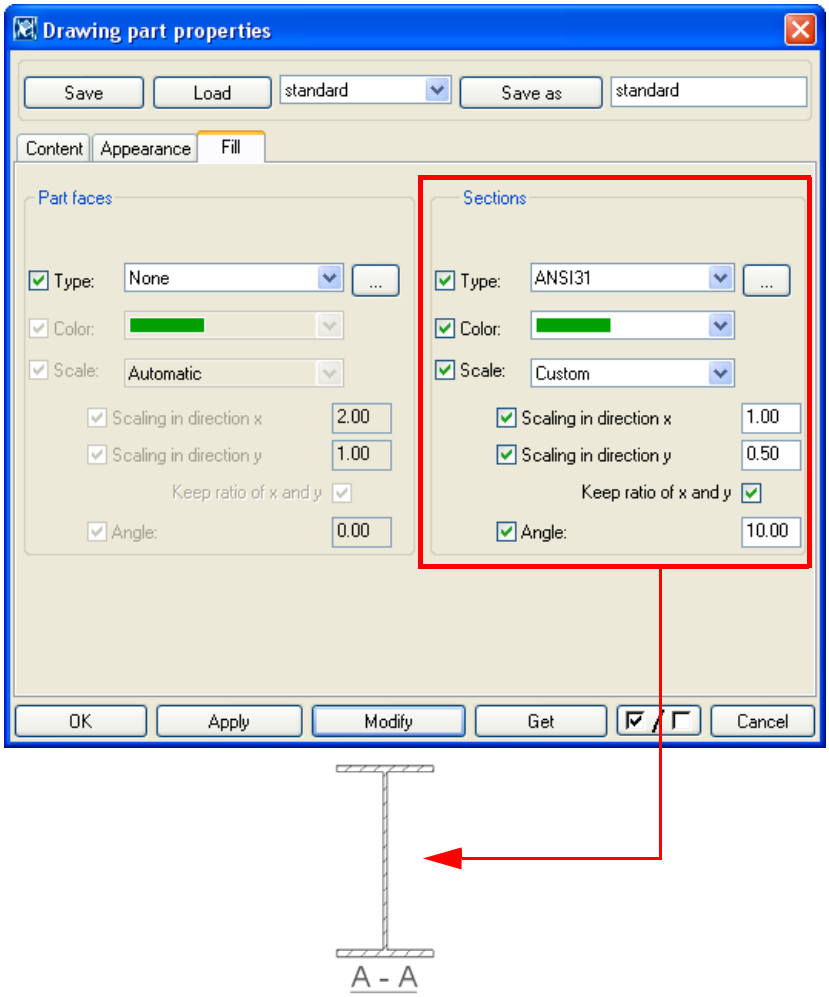
- The color will be plotted as a gray scale, depending on the selected printer settings. The hatch color is defined using RGB (Red Green Blue) values on a scale of **0** to **255**.
- Click <**Modify**> and <**OK**>

Profile section view hatching

In Tekla Structures you can define different hatching properties for parts in section views. Per profile you can edit the current hatching, switch it off or add one per part.

Step plan

- Open a drawing and double-click on a profile cross section, the following dialog box appears:

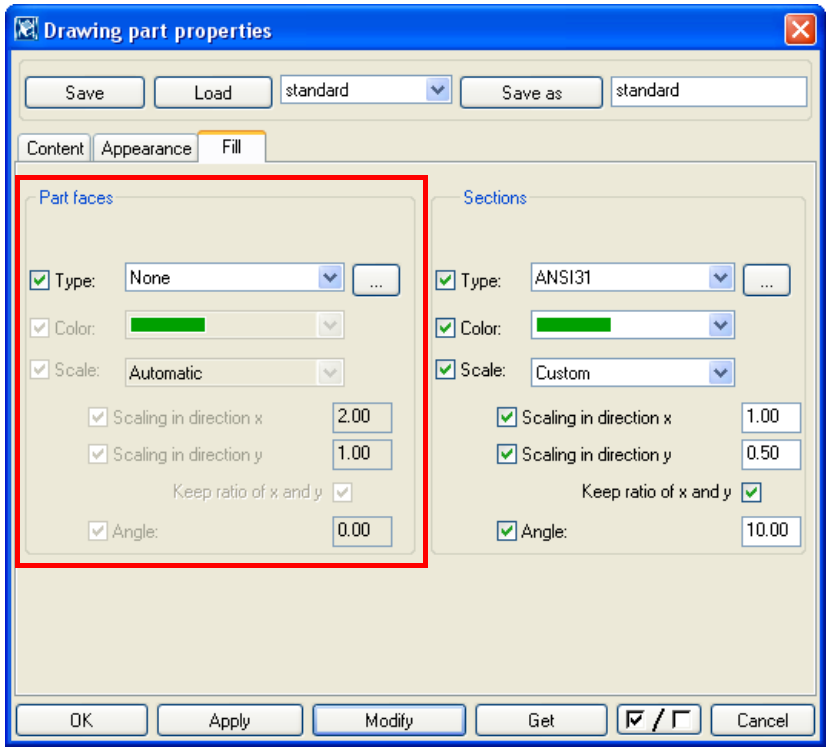


- Go to the tab **Fill** and select a hatch pattern in the listbox
- Click on <**Modify**> and <**OK**>

Hatching part faces

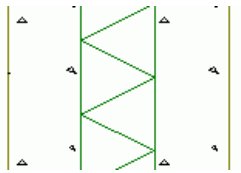
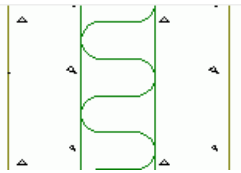
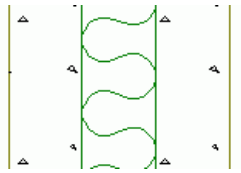
You also have the option to use hatching also on normal part faces to show different materials, for example.

To define the hatching properties for part faces, use the options in the **Part faces** section on the **Fill** tab in the part properties dialog box.

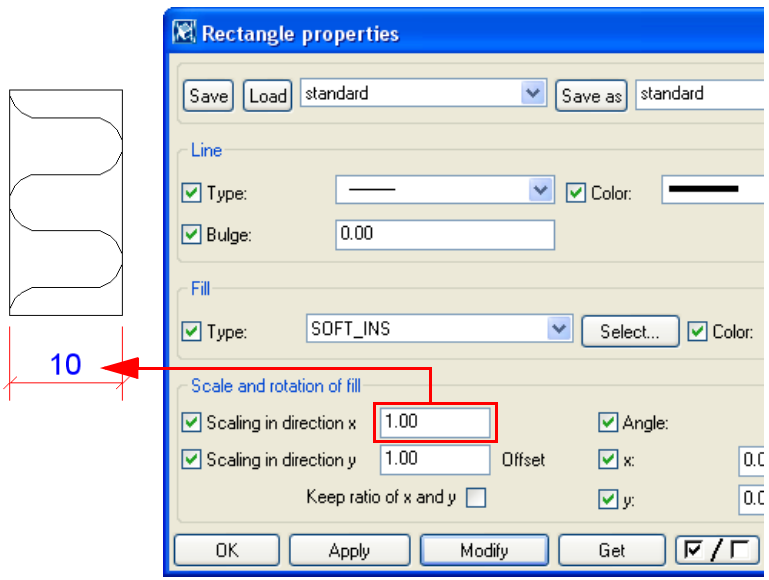


Insulation

You can use the following fill types to represent insulation

Hatch name	Pattern
HARD_INS1	
SOFT_INS	
SOFT_INS2	

Use the option **Scaling in direction x** to modify the thickness of the insulation. Scale 1.0 corresponds to 10 mm, 0.5 to 5 mm, etc.



Accentuate cross-sections

Cast unit shapes in cross-sections can be accentuated on drawings. For this, the variable XS_SECTION_LINE_COLOR is set to the color PURPLE (value 7).

This means that - assuming that the purple color is set as a thicker line type in the plotter catalog - the cross-section shapes on the drawings are printed thicker automatically.

In case you don’t want this, or you want to define another color, set the variable XS_SECTION_LINE_COLOR in **Tools** > **Advanced options** > **Hatching**.



The purple color is only visible on the drawing when you drag the middle mouse on the drawing.

2.21 Printing drawings

Introduction

Tekla Structures prints drawings to both selected print devices and to file (pdf or plt), from either the Drawing Editor or the drawing list in the Model Editor. You can also export drawings to dwg/dxf files.

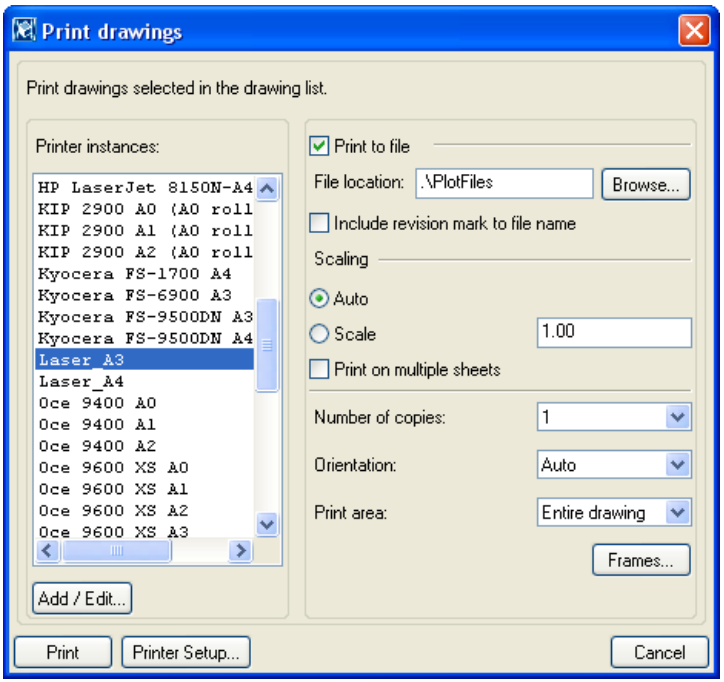
See also

For more information about exporting drawings to dwg/dxf files, see **Tekla Structures.pdf** in the folder disc:\TeklaStructures\version\environments\europe\pdf.



The procedure for plotting and printing Tekla Structures drawings is as follows:

Select the drawing(s) in the drawing list which you want to print. Next, click the icon opposite, or go to **File > Print drawings...**. The following dialog appears:

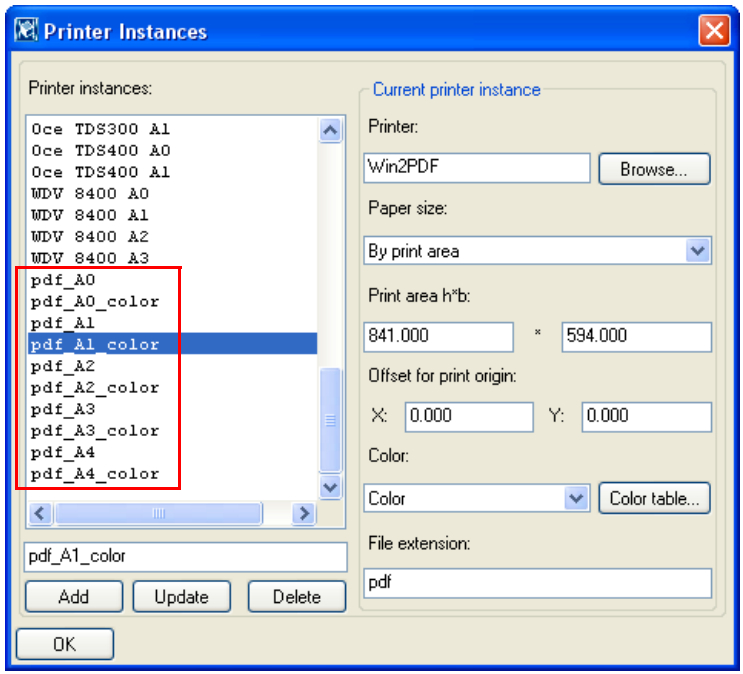


Choose a printing device and press <**Print**>.

With the **Auto scale** option enabled, the drawing shall be scaled automatically to the paper size. A fixed scale can be specified also. You can also set the number of copies.

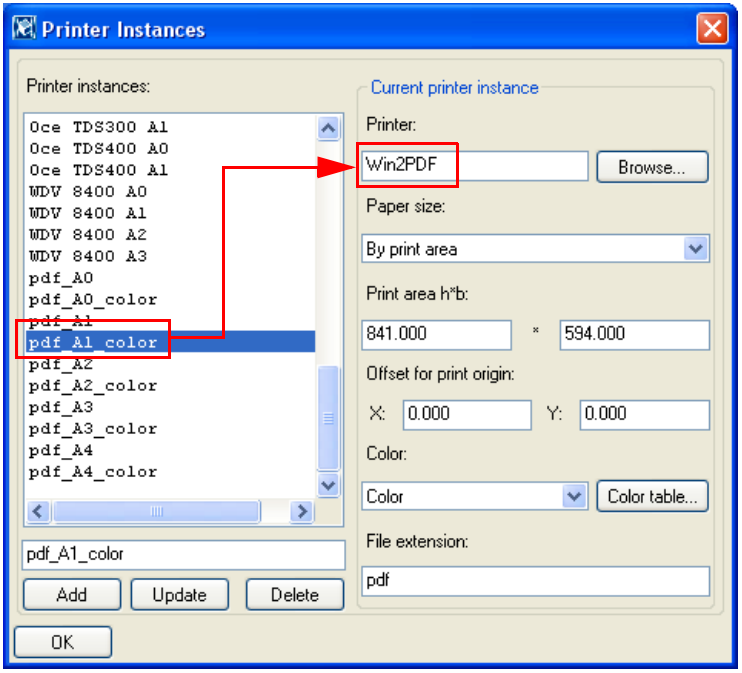
Print to PDF

To print drawings to PDF files (in color or in black/white), you can use one of the following devices:

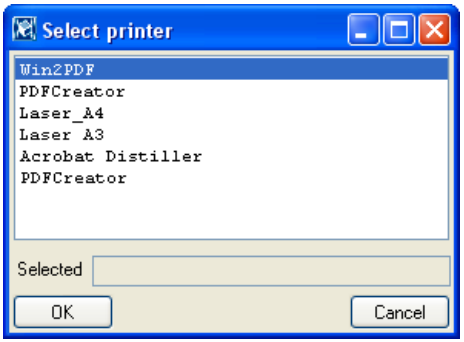


Before this, you first need to install a suitable printer, i.e. `Win2Pdf` or `Pdfcreator` in Windows.

After that you can link that specific printer in the dialog box **Plotter instances** in Tekla Structures. Click <**Add/Edit**>, the following dialog box appears:



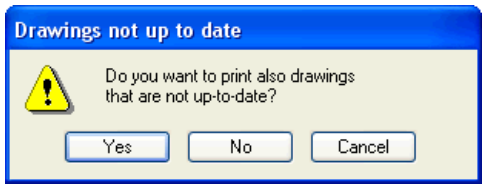
Click <**Browse**> and select the desired printer:



Click <**OK**>, <**Update**> and <**OK**>.

Printing non up-to-date drawings

When you start a print job for a drawing which isn't up to date, Tekla Structures will give a warning:

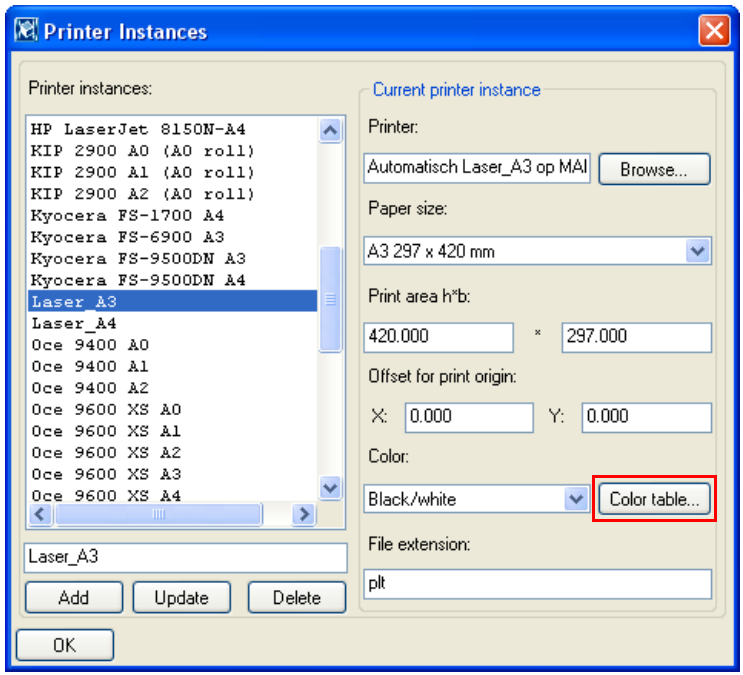


This is an extra protection to avoid that non up-to-date drawings are printed and come into circulation.

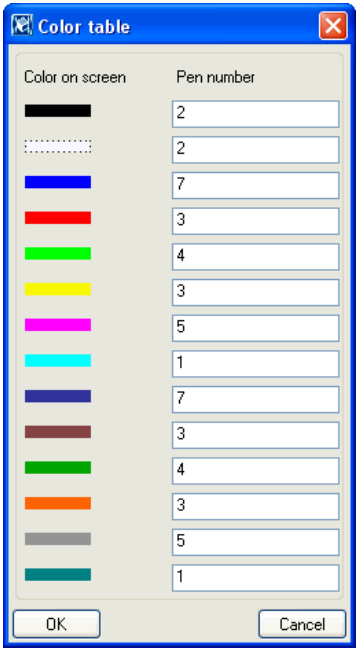
Default Line thicknesses



The line-thicknesses can be specified for each individual printing device. Go to **File** > **Catalog** > **Printers** > **Modify...** or click the icon:



Select a printer or a plotter and click <**Color table**>



The entered number corresponds to the line thickness in tenths of millimeters.

Example

- Red = 3
- Line thickness = 0.3 mm



In case you want to make use of line thicknesses such as 0,35 mm, enter the value **35** in the dialog box **Color table** and enter the value **0.01** for the variable `XS_BASE_LINE_WIDTH` in **Tools** > **Advanced options** > **Printing**.

The default value is **0.1**.

2.22 Layout and Table layout

The *Layout* and accompanying table layout have to do with the tables and lists which can be put on drawings.

Setup the Layout and table layout

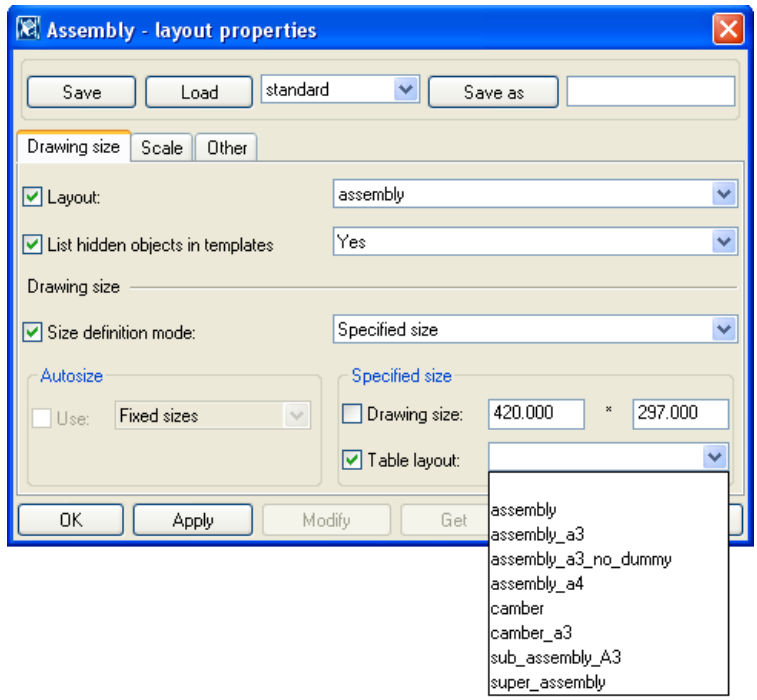
For each drawing type (*single part*, *assembly*, *GA*), different tables are used. This is controlled with the Layout and accompanying Table Layout.

There are three different layouts available:

- assembly
- single
- GA

For each layout, several table layouts can be selected. The used templates are the same, only the size (scale) of the templates differs. This is set because for smaller sized drawings you usually prefer smaller templates than for big size drawings.

The following picture shows the dialog **Layout properties** for Assembly drawings:



If **assembly** is selected for Layout, than the choices for the accompanying Table layout will be:

- **assembly (1:1)**
- **assembly_a3 (1:2)**
- **assembly_a3_no_dummy (1:2)**
- **assembly_a4 (1:2)**
- **camber (1:1)**
- **camber_A3 (1:1)**
- **sub_assembly_A3 (1:2)**
- **superassembly (1:1)**

If the listbox Table layout is left blank, no templates will be displayed on the drawing!

In the table below are the available layouts with accompanying table layouts listed:

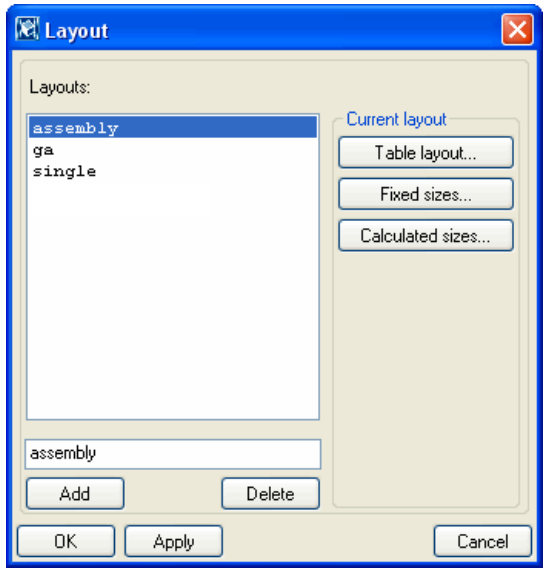
Layout	Table layout	Template scale	Drawing size
Assembly	assembly	1:1	A0 through A2
	assembly_a3	1:2	A3
	assembly_a3_nd	1:2	A3
	assembly_a4	1:2	A4
	camber	1:1	A0 through A2
	camber_a4	1:2	A4
	sub_assembly_a3	1:2	A3 and A4
	superassembly	1:1	A0 through A2
Single	plate	1:1	A3 and A4
	single	1:2	A0 through A2
	single_a3	1:2	A3 and A4
GA	ga	1:1	A0 through A2
	ga_a3	1:2	A3 and A4
	ga_foundationplan	1:1	A0 through A2
	ga_concrete	1:1	A0 through A2

Modifying the layout

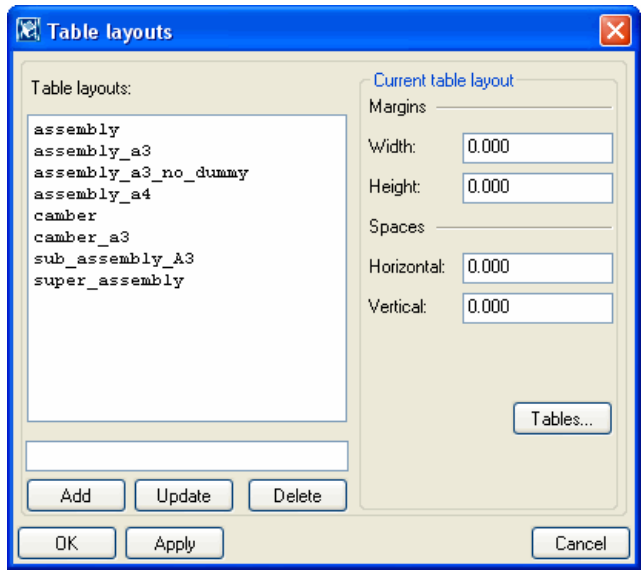
To modify the layout and table layout, go to **Properties** > **Layout.** The dialog box below shall appear. In the list are the available layouts.

Example

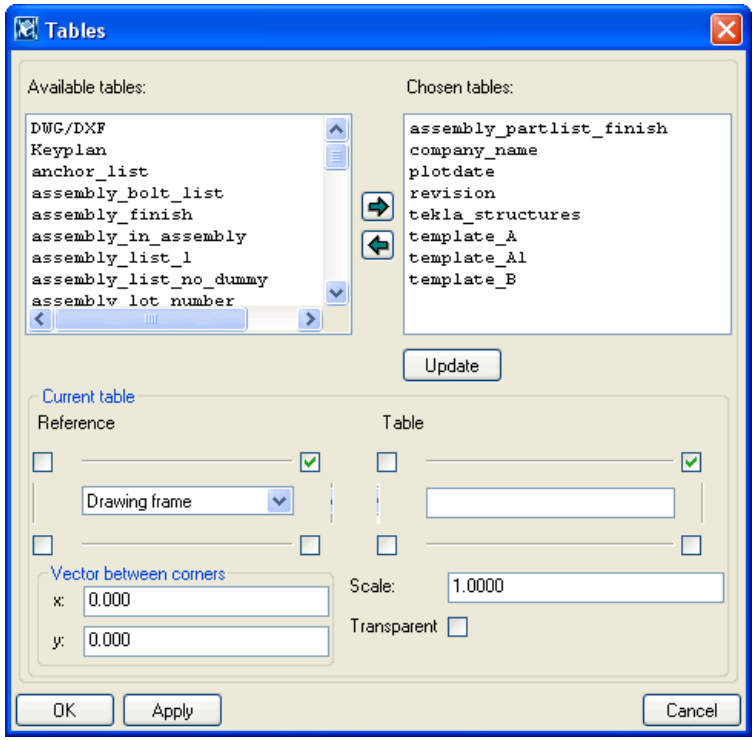
For layout **assembly**, we will add a template (**assembly_partlist_finish**) to both table layouts:



1. Select **assembly**. Than click <**Table layout**> once. The following dialog box appear:



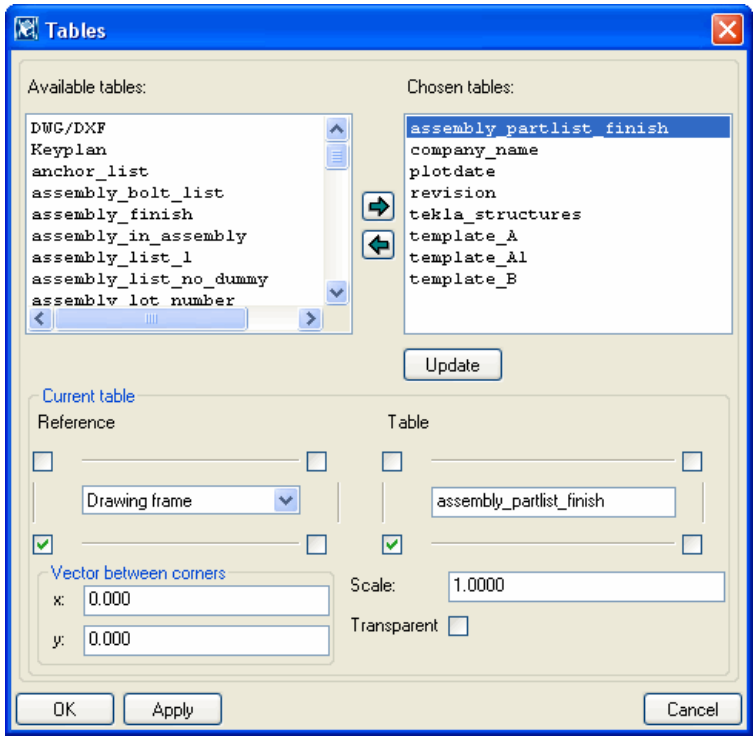
2. Select **assembly**. Next, click the button <**Tables**>. The dialog box below appears.



Chosen tables shows the displayed tables which will be positioned on the drawing (if this layout will be used). We will add table **Assembly_partlist_finish** from the list with **Available tables** to the list with **Chosen tables**.

The position of the table should be the left-down side of the drawing.

3. In the left column, select **Assembly_partlist_finish**, than click the <**Add**> button.
4. **Assembly_partlist_finish** is now added to the list with **Chosen tables**.



5. In the right column, select table **Assembly_partlist_finish**. This table is now displayed in field **Table**.

Now the selected table has to positioned on a fixed location. This can be set with help of the ‘corner checks’ at **Reference** and **Table**.

6. At **Table**, place a check at the down-left side of the table. Make sure that no other corners of the table are checked. Make sure that from the **Reference** listbox, *Drawing Frame* is selected.

7. Place a check in the down-left side of the table. We are modifying table layout called **assembly**, the scales of the tables in this layout have to be set to 1:1. So, in this case we can leave the value for **Scale** untouched.

8. To store the setting, click the <**Update**> button. If all tables are set to the desired position, close the dialog with <**Apply**> and <**OK**>.

9. You will return to the previous dialog, where you can adapt the other table layout, **assembly_a3**. Follow the same procedure as above, but mind for the different scale (1:2).

10. If all table layouts are set properly, close all dialogs with <**Apply**> and <**OK**>. The settings are saved into a **.lay* file, which are stored in the **attributes** folder of the model folder.

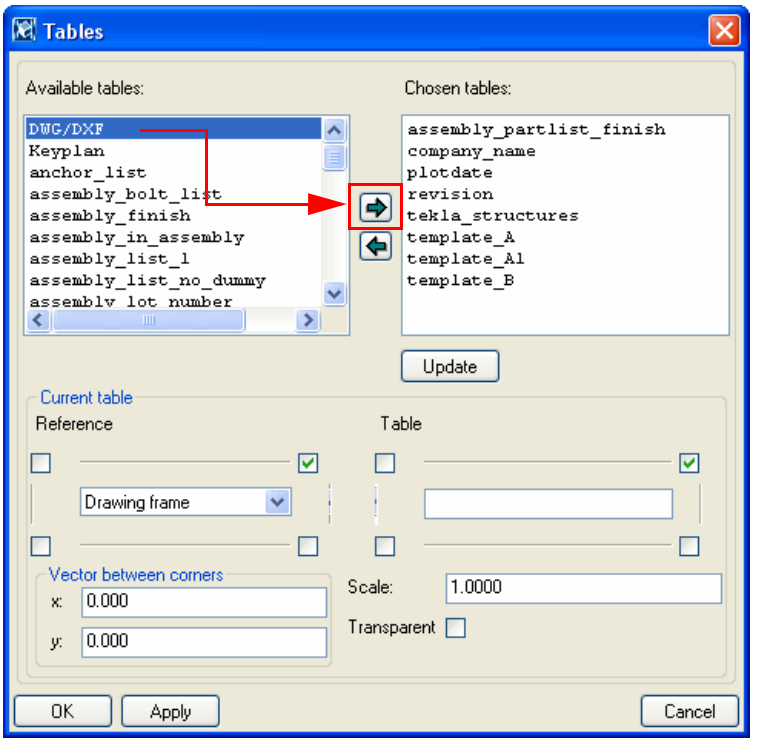
Use dwg/dxf files as a template

When you have a company logo created as a dwg or dxf file, you can add this file directly to a template.

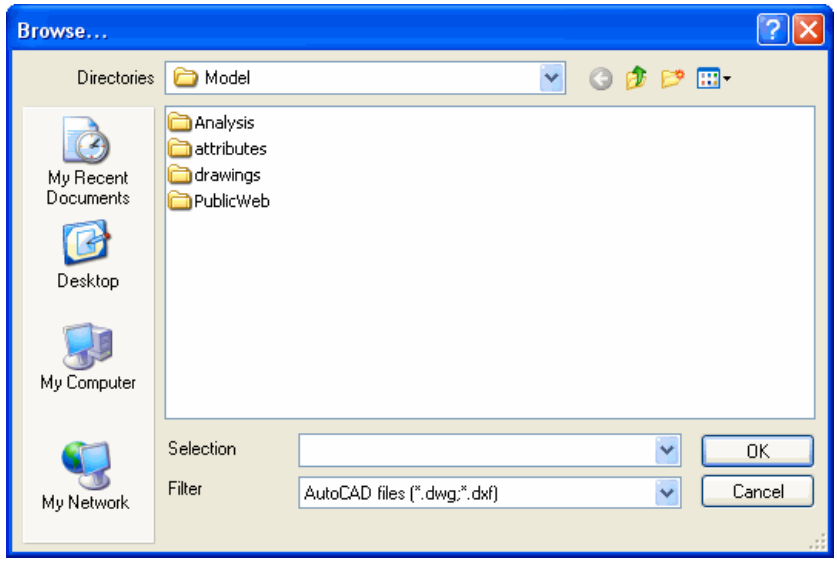
Also third party drawings (principals/architects) can be provided with the right template quick and easily.

Step plan

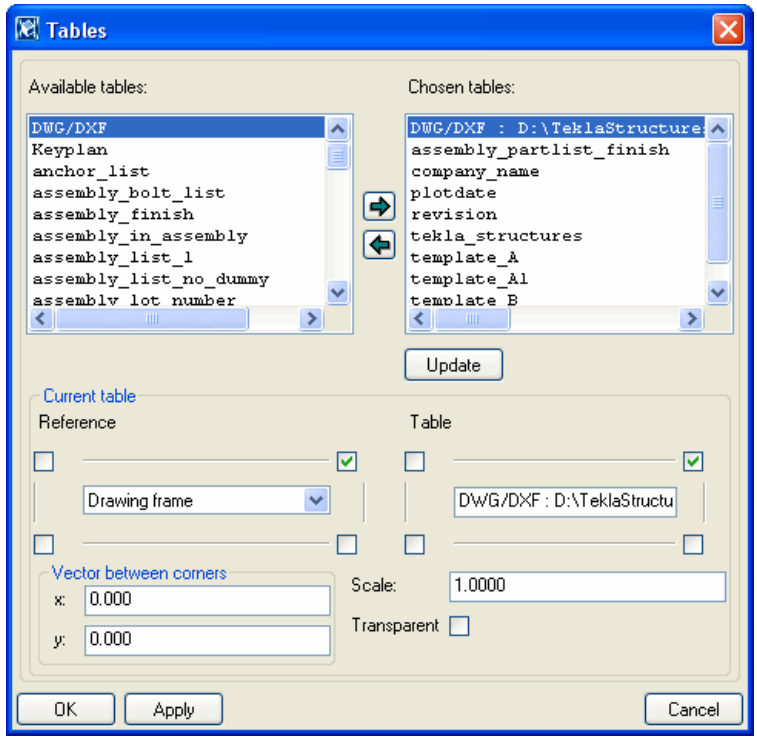
1. Go to **Properties** > **Layout** and select the template you want to add the dwg/dxf file to.
2. In **Available tables** select the option <**DWG/DXF**> and add this template to the **Chosen tables** using the green arrow in the middle.



3. A browse dialog appears, where the dxf/dwg file can be located:



4. The dwg/dxf file is now added to the **Chosen tables**. Next, the dxf/dwg file can be positioned on the drawing sheet, using the corner checks.



5. Make sure that the logo is created with the correct scale. The logo also must be positioned on the coordinate origin (0,0).

DWG/DXF

It was already possible to add a key plan-dwg/dxf file to Tekla Structures drawings. It is also possible to add regular dwg/dxf files to drawing layouts. The procedure of adding the dwg/dxf is the same as for the key plan.

Step plan

In the model environment, go to **Properties** > **Layout** and select the table where you want to add a dwg/dxf to. In the “Available tables”, select DWG/DXF and click the ‘right-arrow’ button. A browse-dialog appears, where the dxf/dwg file can be located. Next, the dxf/dwg file can be positioned on the drawing sheet, using the corner checks.

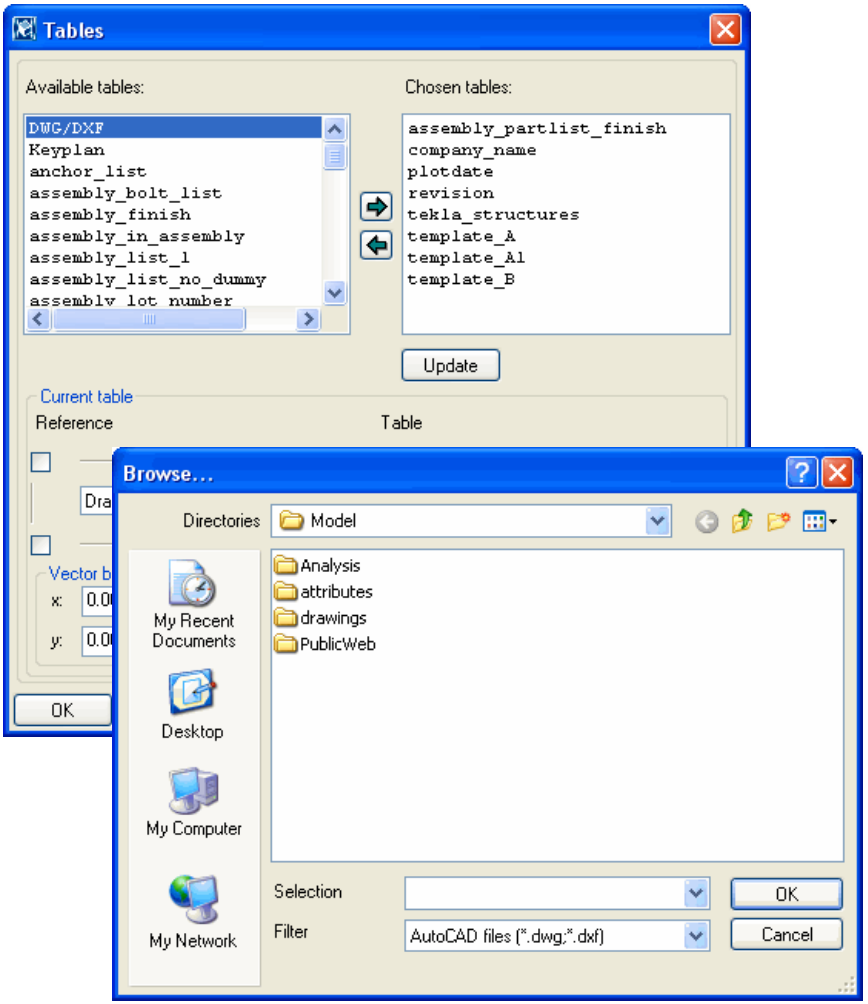
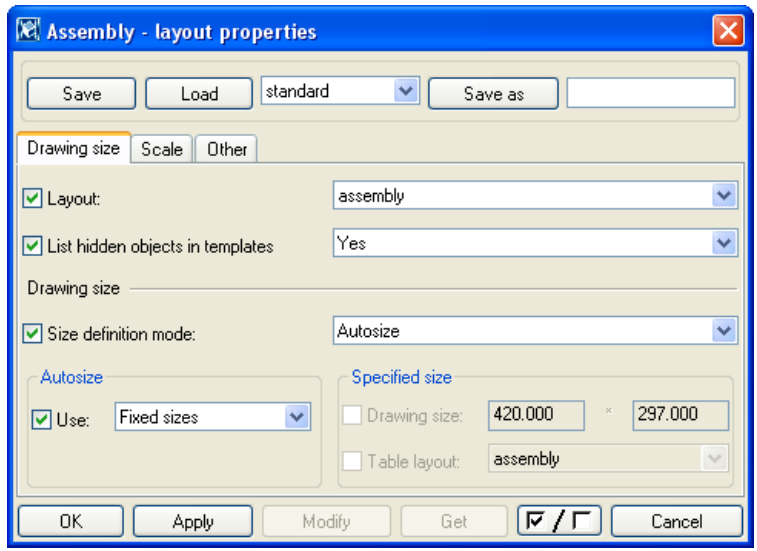


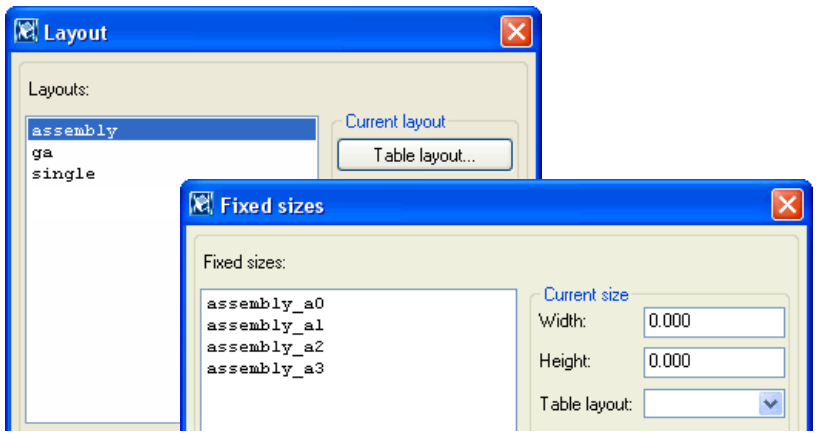
Table Layout when using Auto Sizing

If in the Layout properties, the **Size definition mode** is set to **Auto Size**, a table layout can not be selected. See picture:



Nevertheless, a table layout will be used. Which table layout is used, is also set in the Layout properties. (**Properties** > **Layout)**.

In the 1st dialog, select a layout, e.g. **assembly**. Next, click **Fixed Sizes,** dialog **Fixed sizes** open in which you set what sizes may be chosen and which table layout is connected to which size. In this example, a drawing with layout **assembly** and **AutoSize** enabled, never can choose a A4 sheet size because that size is not set in the list with **Fixed Sizes**. However, the A4 size can be added of course.



2.23 Editing templates

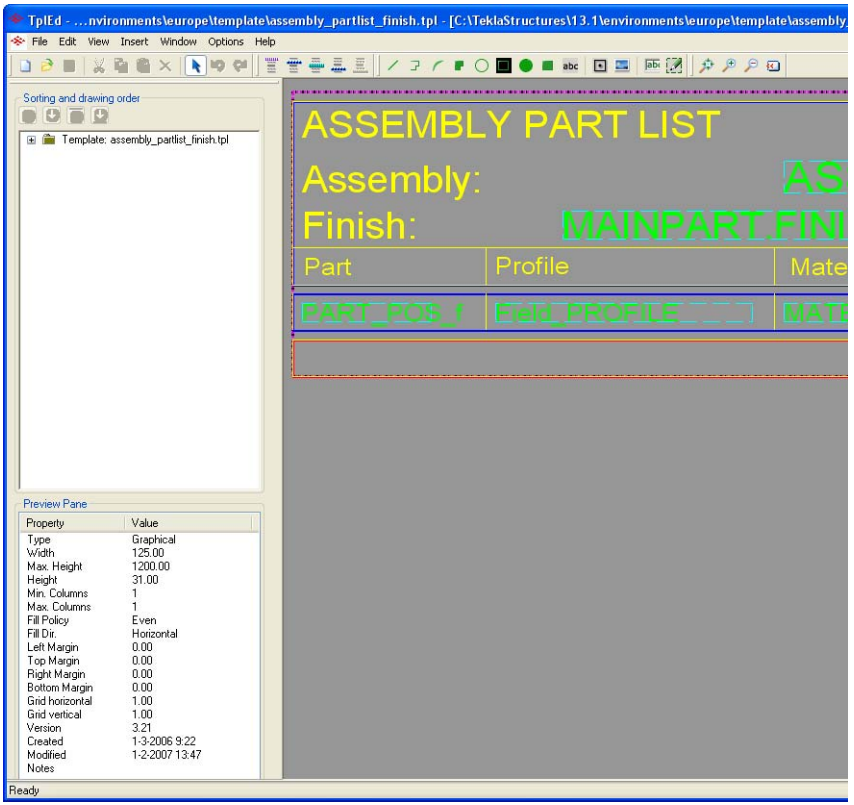


Templates can be created and edited in the Template Editor. Open this editor in the pull-down menu **Tools** > **Templates...** or click the icon:

Start > **Programs** > **TeklaStructures version > Tools > Template Editor 3.3 > Template editor enu Europe.**

To open an existing template:

- Go to **File** > **Open..**
- In the list, select for example **assembly_partlist_finish.tpl** and click <**OK**>
- The template will be displayed and can de edited now:



See also

For more information about the Template editor, see **Tekla Structures.pdf** in the folder: **disc:\TeklaStructures\version\environments\europe\pdf**.

2.24 Reports

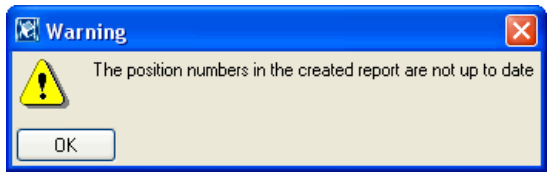


Step plan

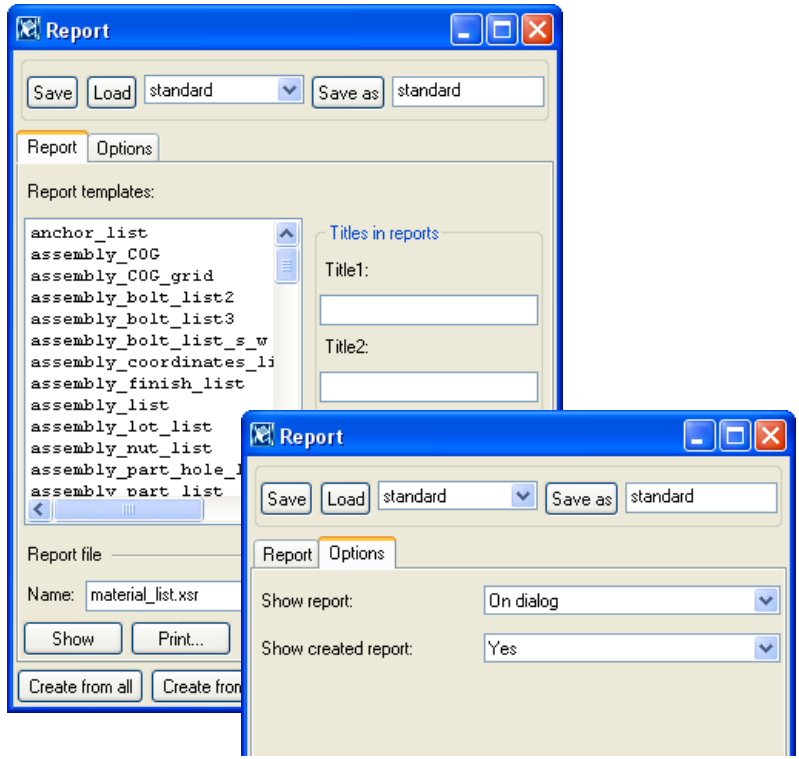
To generate reports, go to **File** > **Report** or click on the icon, the dialog box **Report** appears:

1. Select the desired report, now there are two options:
   - Create a report of all parts in the model, click the button <**Create from all**>
   - Create a report of selected parts. First make a selection of the model. Next, click <**Create from selected**>

If the numbering is not up-to-date, the following dialog appears:



Re-number the model.



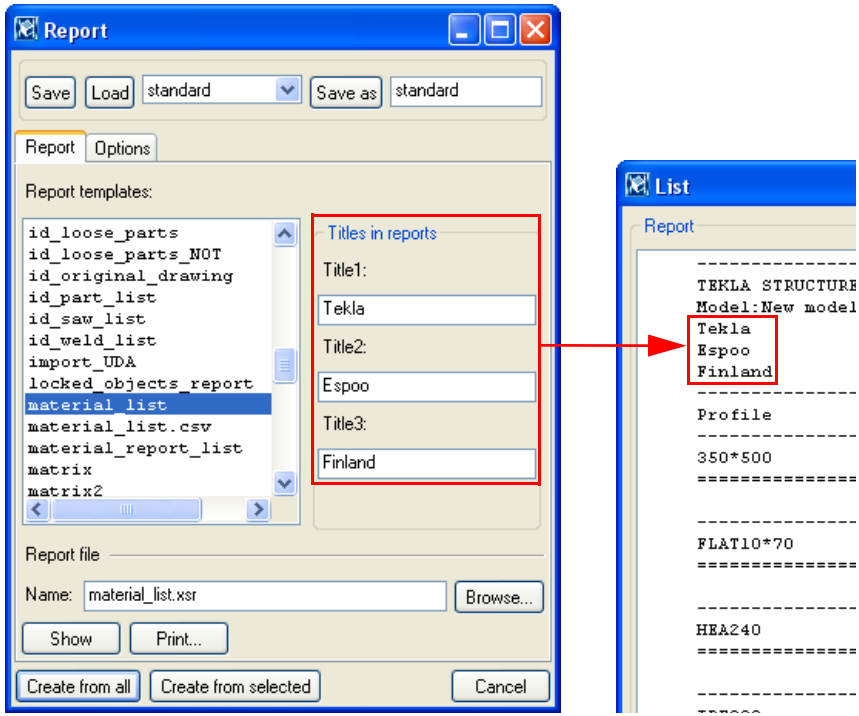
2. Next you choose if, and how the report must be displayed

3. Before this, go to the tab **Options** and select **Yes** or **No** in the listbox **Show created report.**
4. If the option **Show report** in the tab **Options** is set to **On dialog** and **Show created report** is set to **Yes**, than the report will be displayed in a dialog automatically after generation of the report has been completed.

To add more information to reports, you can make use of the fields **Title 1, Title 2** and **Title 3**.

Tekla Structures provides a large number of reports. In case only a few of them are used, it can be useful to move the unnecessarily reports to another folder. This makes that the reports are not listed anymore in the Report-dialog.

Example



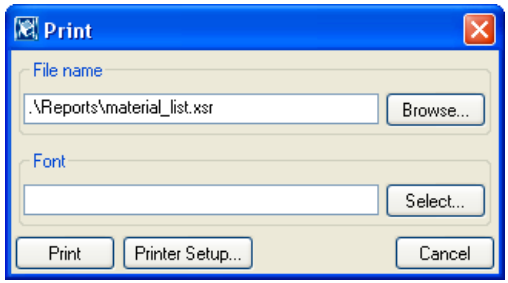
By default, the name of the saved report file is equal to the name of the report as it is shown in the list. However, you can edit the filename before generating the report. This can be useful when using the same report more than once in a model.

Reports are saved in the folder **Reports** in the model folder.

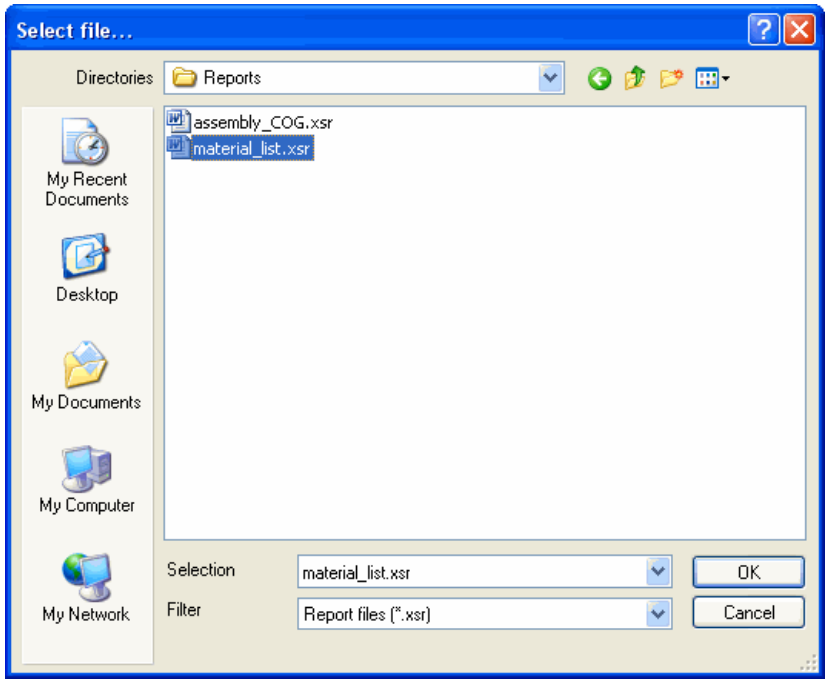
2.25 Printing reports



Printing reports can be done by clicking the button **<Print>** in the Report dialog box:



The report can be located and selected by clicking the <**Browse>** button. The default folder is the current model folder.



The report font can be set also in this dialog box. By default, the report is being printed to the Default printer. If you want to print to another printing device, than press the <**Printer setup**> button and select another device.

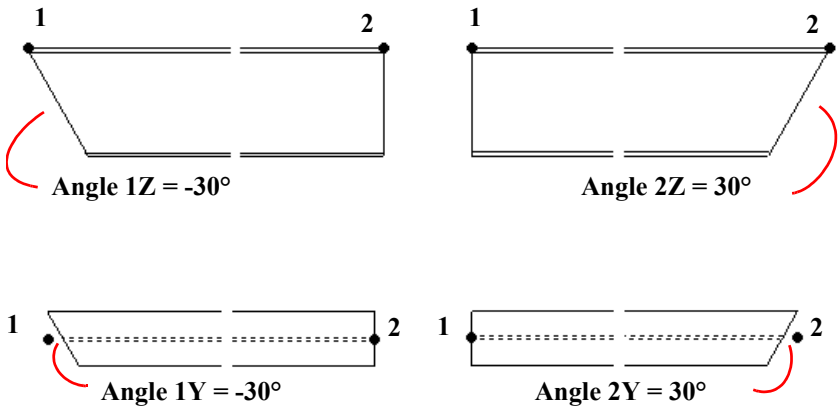
2.26 Saw cut angles

On some reports are the saw cut angles mentioned.

In the figure below is displayed which saw angle corresponds to which report parameter. The angles always determined according to the local coordinate system.

See also For more information, see **Moving start- and endpoints** (p. 29) in the Modeling Manual.

Example The local direction between reference points (from point 1 to point 2) is always the positive X-direction.



2.27 Cannot open a drawing anymore

Sometimes one or more drawings can not open any more. This is the cause of:

- A *.dg file (in the folder **drawings**) is deleted from the model folder. Never delete these files, these are drawings.
- When Tekla Structures is opened twice and the same model is opened twice.

Example In a model one or more drawings are modified, saved and closed. Now Tekla Structures is closed and the model is saved. After that the “other” model is saved and Tekla Structures is closed.

If you open the model now, you cannot open one or more drawings any more.

2.28 Classifier

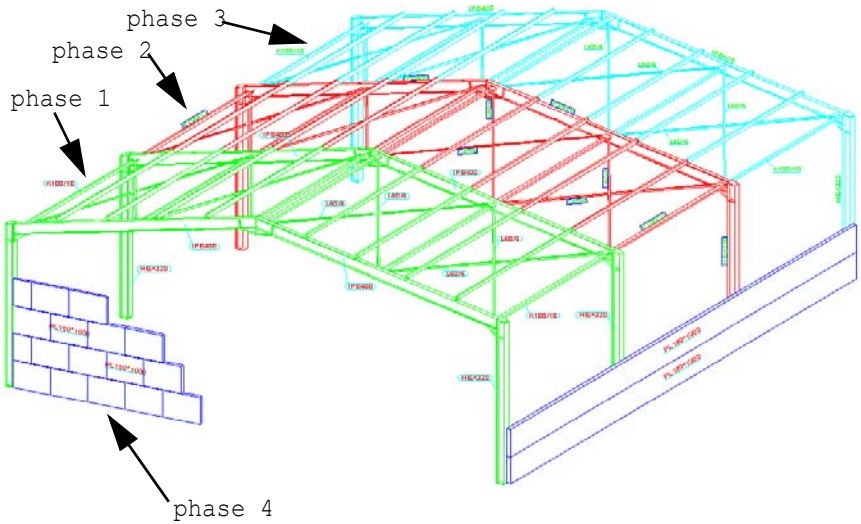
Classifier principle

The classifier enables you to create different drawing styles.You can use the classifier to control the drawing properties for different types of model objects, for example:

- Different phases in separate colors
- Different parts in separate colors
- Different material grades in separate colors

Except colors, you can also apply this to the position of part marks for each separate profile, phase and material grade.

The picture below shows an example how to use the classifier. The phases appear in separate colors. Also the position of the part marks differs for each phase.

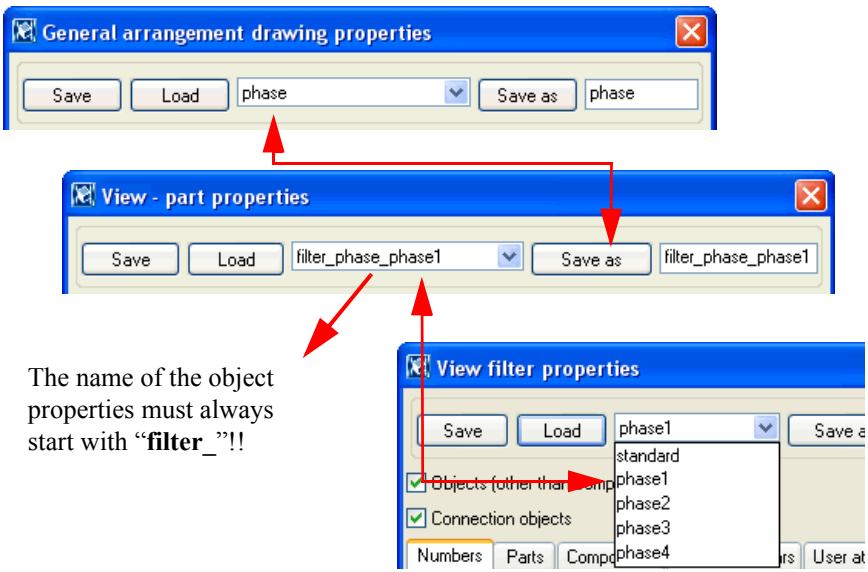


The classifier works by using a specific naming convention for saved properties. We create them in steps. Once created, you can use them, of course, again and again. It enables you to create drawings containing a specific style every time!

The classifier uses the following properties:

- general arrangement drawing properties
- view filter properties
- object properties, for example; part, part mark, bolt and bolt mark

The classifier works by using a specific naming convention for saved properties (see picture below).



The name of the object properties must always start with "**filter_**"!!

Make sure that the names that you use for the properties match to each other!

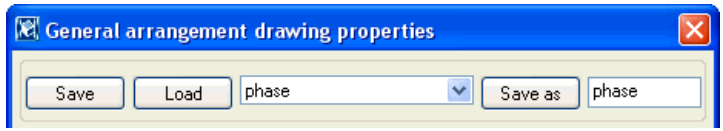
On the basis of several examples we will explain how the classifier works.

“Phase” properties

You now learn how to create a drawing in which different phases appear in different colors.

Define drawing properties

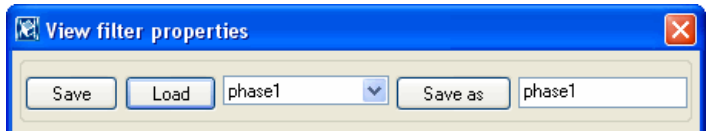
1. Go to **> Properties > General arrangement drawing.** Create a general arrangement drawing property, enter the name *“phase”* in the text box and click <**Save as**>.



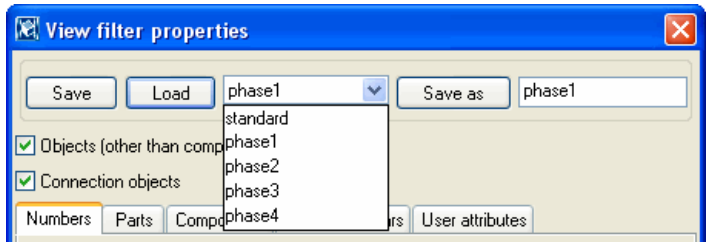
Creating view filters

2. Create view filters to define the specific group of model objects, whose appearance you want to define. In this example every phase appears in a corresponding color on a drawing. So for each phase we create a view filter, one view filter for phase 1, one view filter for phase 2, etc.

3. Open any drawing and double-click on a view. Click on the button **Filter.** In the dialog box **View filter properties** go to the tab **Numbers** and define the setting so that phase 1 is filtered (see picture below). Enter the name ***phase 1*** in the text box and click <**Save as**>.



4. Repeat the steps for all phases you want to define. In this example: ***phase 2***, ***phase 3*** and ***phase 4***. Click <**Cancel**> to close the dialog box!



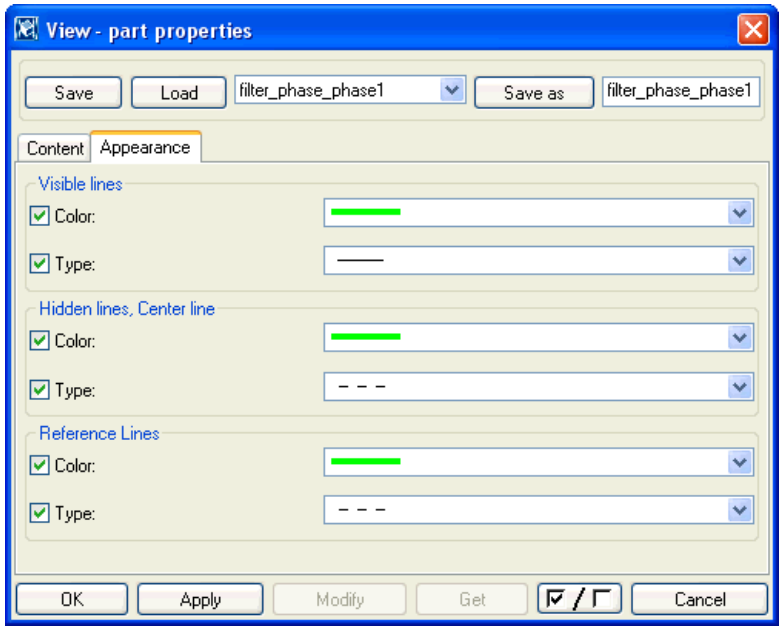


Do not click <**Apply**>, as this will apply the current filter to the entire drawing and the Classifier will not work! Just click <**Cancel>.**

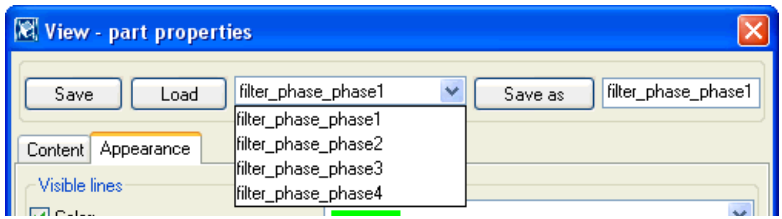
Always use view filters when you use the Drawing Classifier. The Drawing Classifier does not recognize select filters!

Creating drawing object property files

5. For each phase you can define the corresponding color who appears on a drawing. In this example phase 1, 2, 3 and 4. Double-click on a view and click on the button **Part.** Define the setting for phase 1. Enter the name ***filter_phase_phase1*** in the text box and click <**Save as**>. (see picture below).

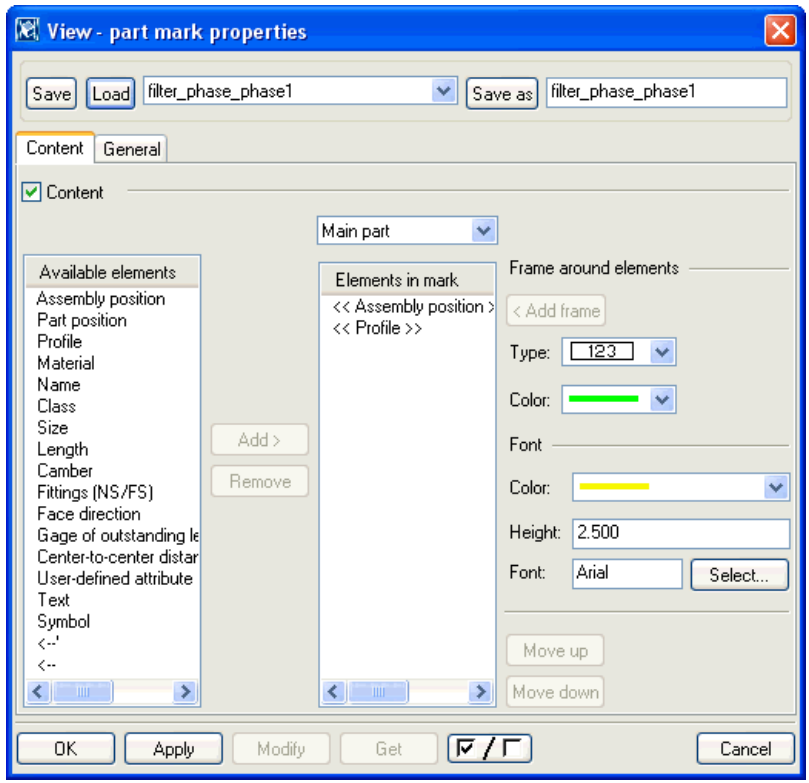


6. Repeat these steps for all phases you want to define. In this example: ***filter_phase_phase2***, ***filter_phase_phase3*** and ***filter_phase_phase4***. Click <**Cancel**> to close the dialog box!

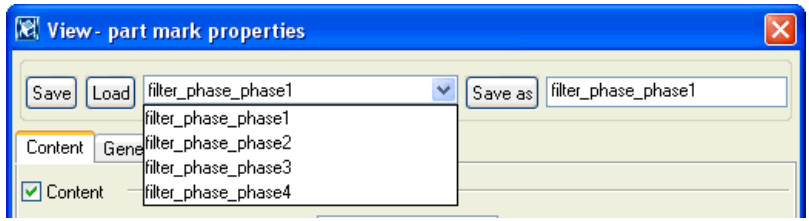


7. For all phases a specific color is defined. Of course it is possible to define part mark property files for each phase, or to change the appearance. Follow the steps we used for defining the part colors for each phase.

8. So now you can define the appearance and position of part mark property files for each phase. In this example phase 1, 2, 3 and 4. Double-click on a view and click on the button **Part mark.** Define the setting for phase 1. Enter the name ***filter_phase_phase1*** in the text box and click <**Save as**>. (see picture below).

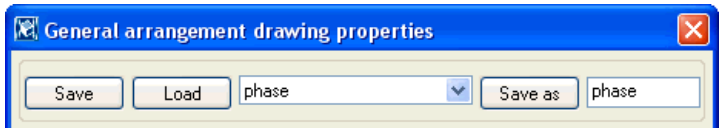


9. Repeat the steps for all phases you want to define. In this example: ***filter_phase_phase2***, ***filter_phase_phase3*** and ***filter_phase_phase4***. Click <**Cancel**> to close the dialog box!

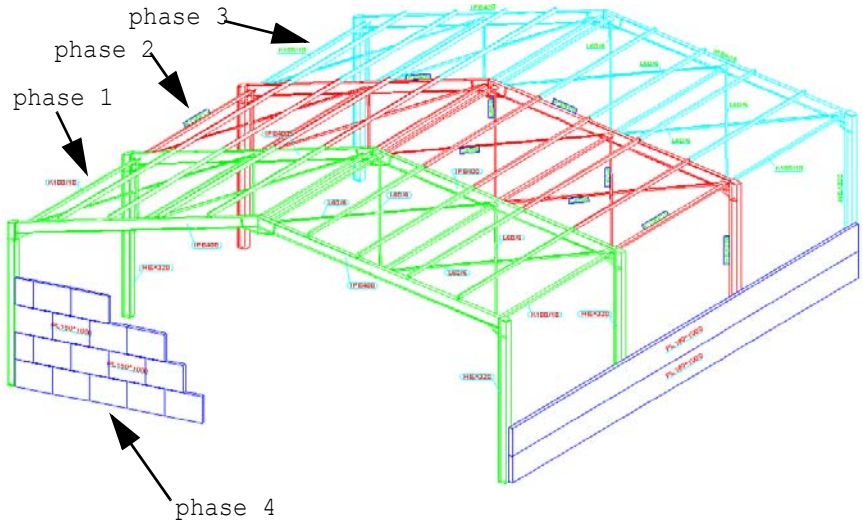


Creating drawings

10. Go to **> Properties > General arrangement drawing...** Click the down arrow in the Load text box and select the ***phase*** drawing properties you created. Click <**Load**> (see picture below). Now create the general arrangement drawing.



You will see that a drawing is created in which different phases appear in different colors.

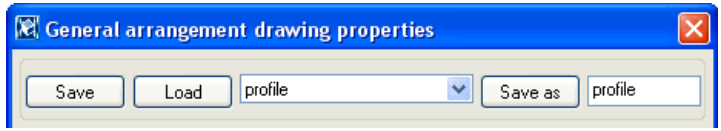


"Profile" properties

You now learn how to create a drawing in which different parts for each part name appear in different colors. So all columns in a specific color, all beams in a specific color.

Define drawing properties

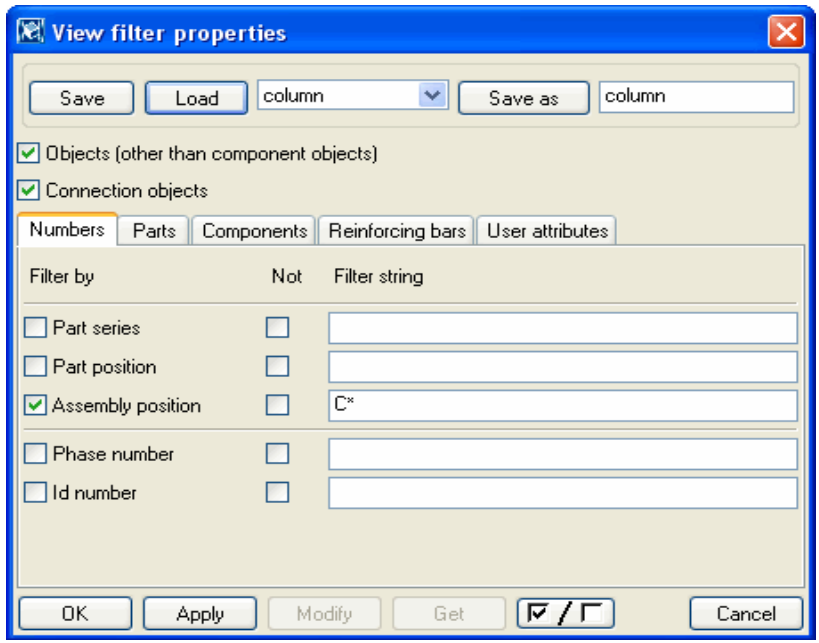
1. Go to **> Properties > General arrangement drawing.** Create a general arrangement drawing property, enter the name “profile” in the text box and click <**Save as**>.



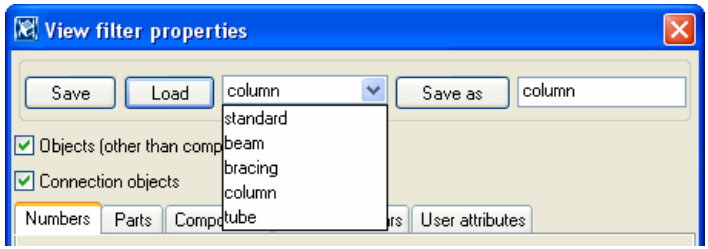
Creating view filters

2. Create view filters to define the specific group of model objects, whose appearance you want to define. In this example every type of profile appears in a corresponding color on a drawing. So for each profile we create a view filter, one view filter for columns, one view filter for beams, etc.

3. Open any drawing and double-click on a view. Click on the button **Filter.** In the dialog box **View filter properties** go to the tab **Numbers,** and in the filter string text box next to Filter by Assembly position enter “**C***”. (for beams B*). In the text box next to the <**Save as**> button, enter “column” and click <**Save as**>. (see picture below)

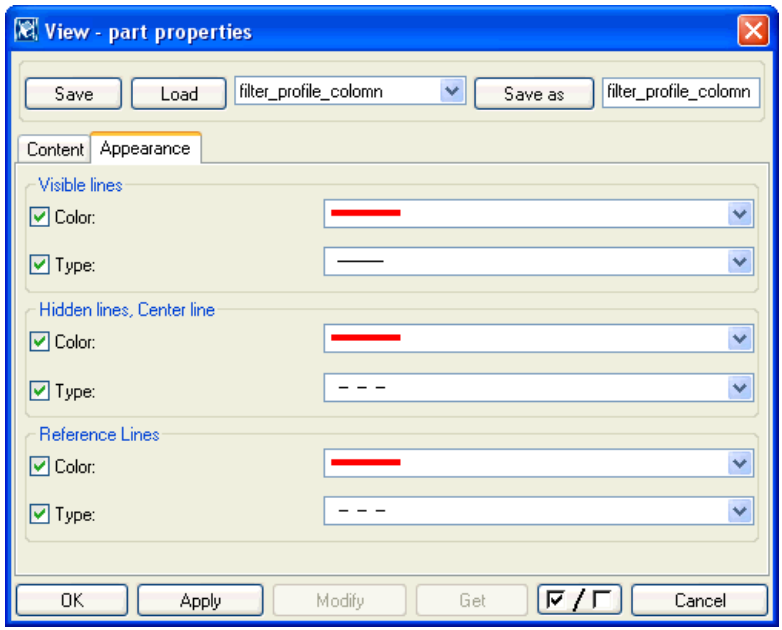


4. Repeat the steps for all profiles you want to define. In this example: ***beam***, ***bracing*** and ***tube***. Click <**Cancel**> to close the dialog box!

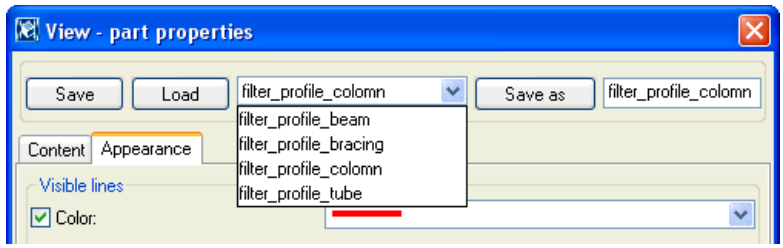


Creating drawing object property files

5. For each profile you can define the corresponding color who appears on a drawing. In this example column, beam, tube and bracing. Double-click on a view and click on the button **Part.** Define the setting for columns. Enter the name ***filter_profile_column*** in the text box and click <**Save as**>. (see picture below).

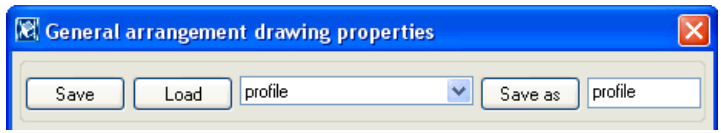


6. Repeat the steps for all profiles you want to define. In this example: ***filter_profile_tube***, ***filter_profile_bracing*** and ***filter_profile_beam***. Click <**Cancel**> to close the dialog box!

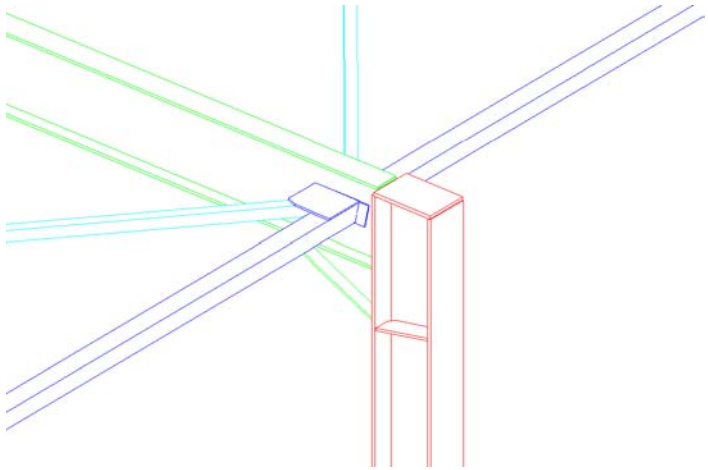


Creating drawings

7. Go to **> Properties > General arrangement drawing...** Click the down arrow in the <**Load>** text box and select the ***profile*** drawing properties you created. Click <**Load>** (see picture below). Now create the general arrangement drawing.



You will see that a drawing is created in which different types of profiles appear in different colors.

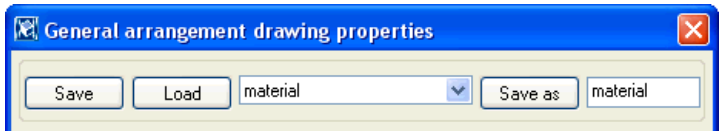


“Material” properties

You now learn how to create a drawing in which different types of material grades appear in different colors.

Define drawing properties

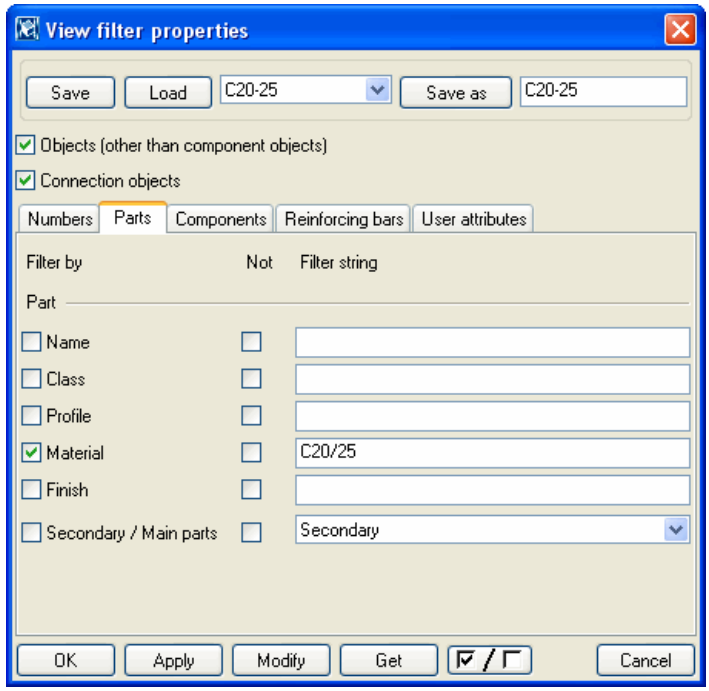
1. Go to **> Properties > General arrangement drawing.** Create a general arrangement drawing property, enter the name “material” in the text box and click <**Save as**>.



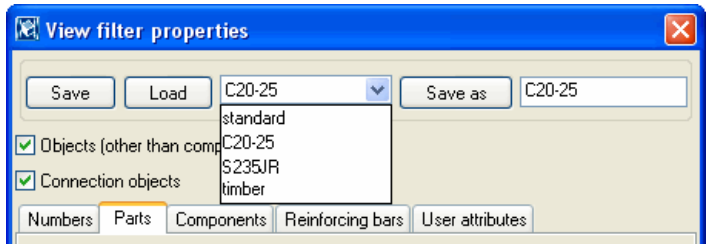
Creating view filters

2. Create view filters to define the specific group of model objects, whose appearance you want to define. In this example every material grade appears in a corresponding color on a drawing. So for each grade we create a view filter.

3. Open any drawing and double-click on a view. Click on the button **Filter.** In the dialog box **View filter properties** go to the tab **Parts,** and in the filter string text box next to Part Material enter "**C20-25**". In the text box next to the <**Save as**> button, enter "C20-25" and click <**Save as**>. (see picture below):

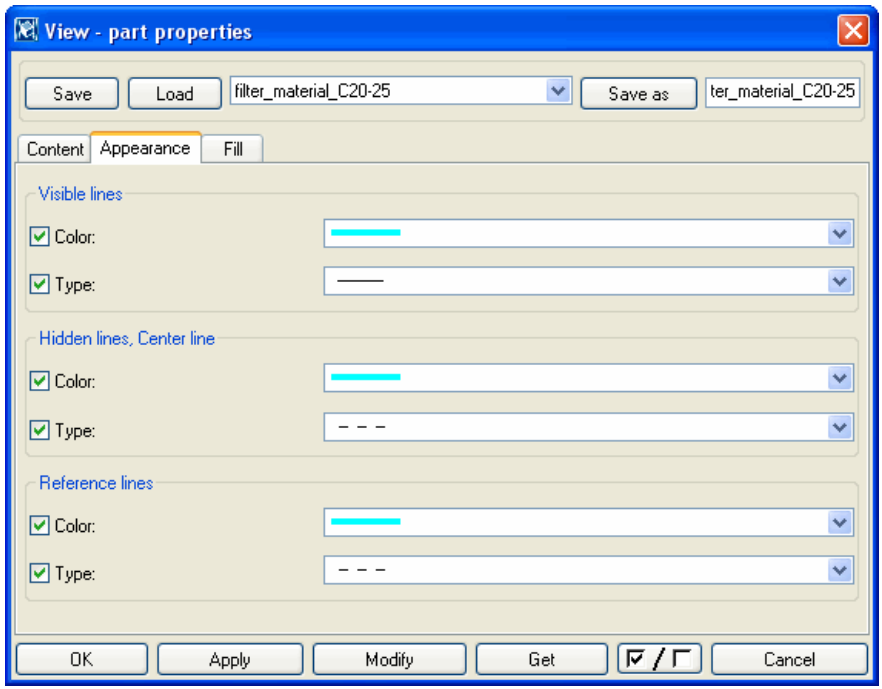


4. Repeat the steps for all material grades you want to define. In this example: ***timber***, ***S235JR***. Click <**Cancel**> to close the dialog box!

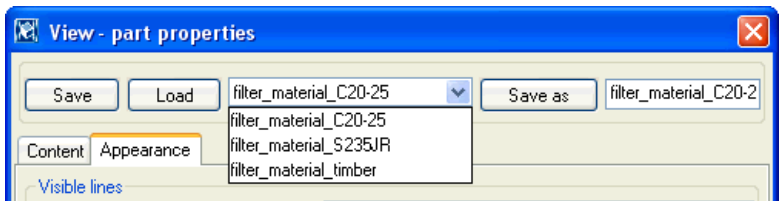


Creating drawing object property files

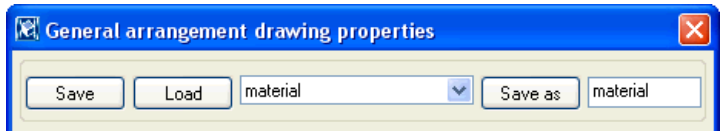
5. For each material grade you can define the corresponding color who appears on a drawing. In this example C20-25, timber and S235JR. Double-click on a view and click on the button **Part.** Define the setting for material grade C20-25. Enter the name ***filter_material_C20-25*** in the text box and click <**Save as**>. (see picture below).



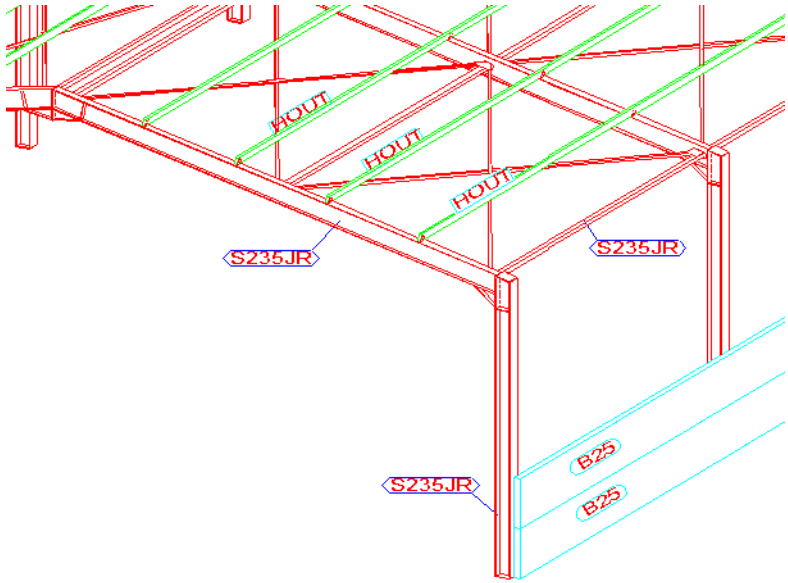
6. Repeat the steps for all material grades you want to define. In this example: ***filter_material_timber*** and ***filter_material_S235JR***. Click <**Cancel**> to close the dialog box!



7. Go to **> Properties > General arrangement drawing...** Click the down arrow in the <**Load**> text box and select the ***material*** drawing properties you created. Click <**Load**> (see picture below). Now create the general arrangement drawing.

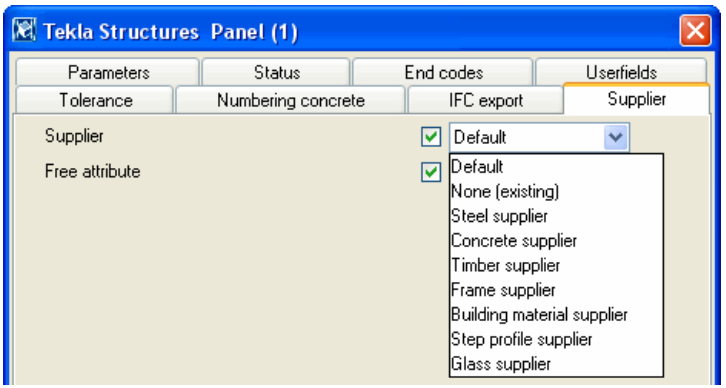


You will see that a drawing is created in which different types of material grades appear in different colors.



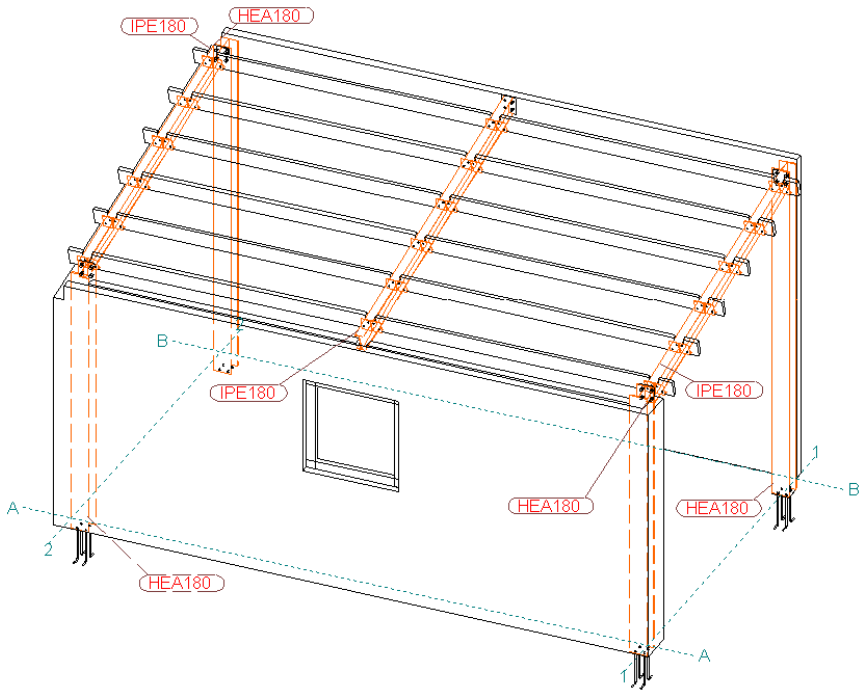
"Supplier" properties

You now learn how to create a GA drawing in which user-defined attributes are used. In this case the user-defined attributes *Steel supplier*.



All parts with the user-defined attribute *Steel supplier* appear on the drawing in Orange, also the **Hidden lines** and the **Own hidden lines** will be displayed. Besides, all marks are displayed.

The other parts will appear on the drawing in White. The Hidden lines, the Own hidden lines and the marks will not be displayed:

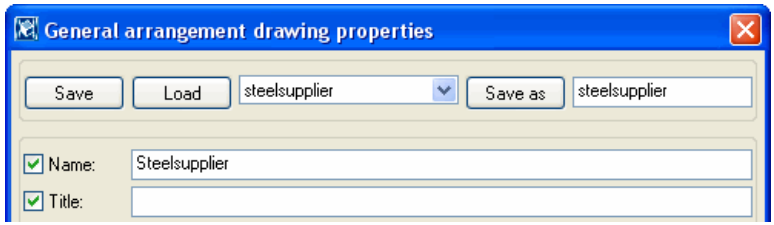


Open the model `Supplier`, you can download the model from the Construsoft Xtranet.

This model contains parts in which the user-defined attributes on the tab **Supplier** are set to, for example, *Steel supplier* or *Glass supplier*.

Define drawing properties

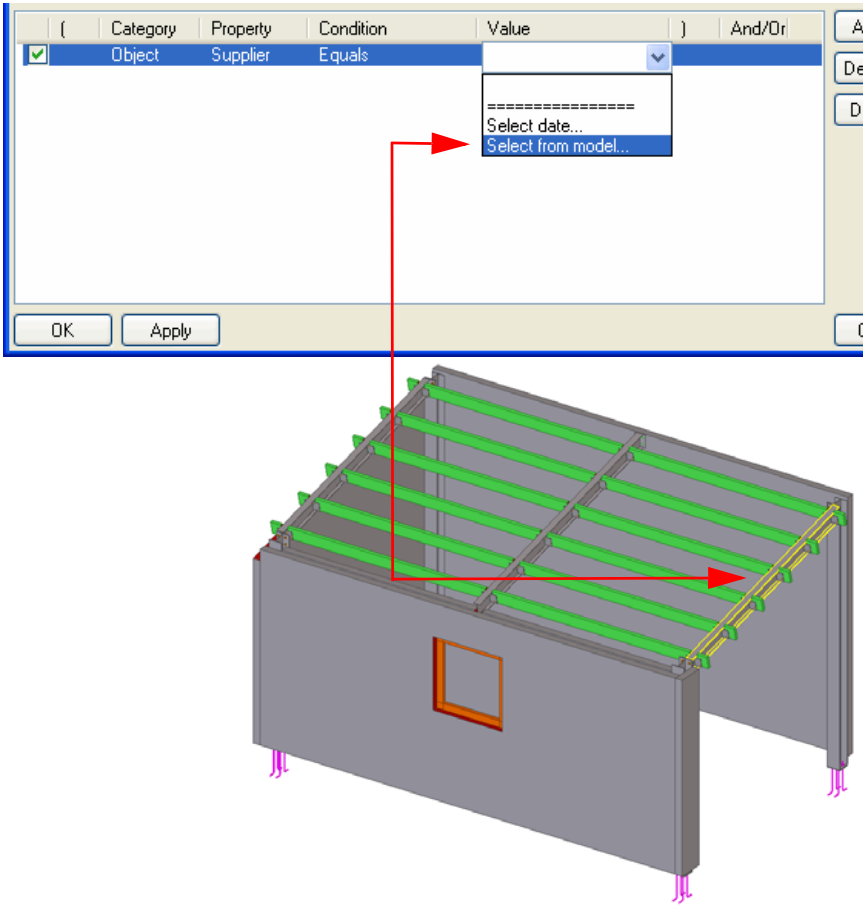
1. Go to **> Properties > General arrangement drawing.** Create a general arrangement drawing property, enter the name “steelsupplier” in the text box and click <**Save as**>.



Creating view filters

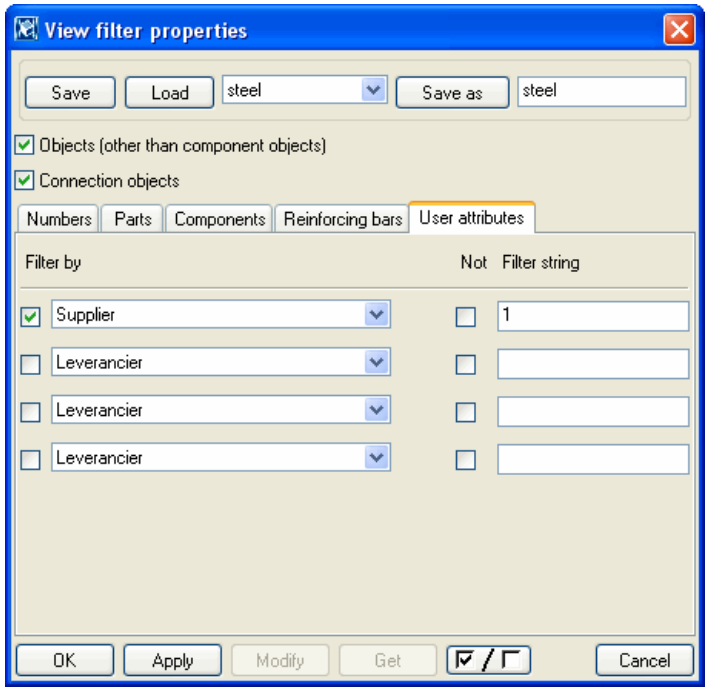
First we have to find out in the model what we need to enter in the view filter in the drawing:

2. The view filters define the specific group of model objects, whose appearance you want to define. In this example all parts containing the user-defined attribute *Steel supplier* appear on a drawing in a specific color.
3. Click **Setup** > **Select filter...** to open the dialog box **Object group- select filter**
4. To find out easily what to enter for the filter properties, go to the column **Category** and select **Object**
5. In the column **Property** select the option **Supplier** in the list box
6. In the column **Condition** select the option **Equals**
7. In the column **Value** select the option **Select from model...** to select the concerning part in the model.

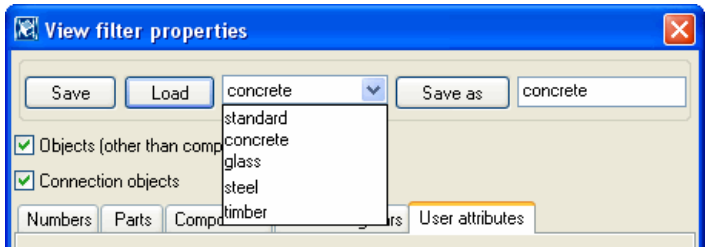


8. Click on a steel part, in the column **Value** the digit **1** appears. You can filter on this.

9. Open any drawing and double-click on a view. Click on the button **Filter.** In the dialog box **View filter properties** go to the tab **User attributes** and define the setting. Save this setting ***steel*** in <**Save as**> as follows:



10. Repeat the steps if necessary for the other suppliers you want to define. Click <**Cancel**> to close the dialog box!





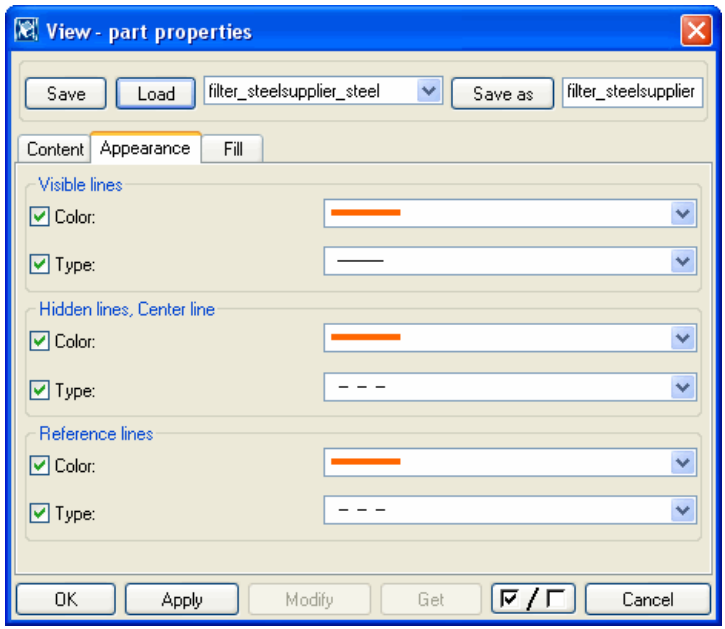
Do not click <**Apply**>, as this will apply the current filter to the entire drawing and the Classifier will not work!



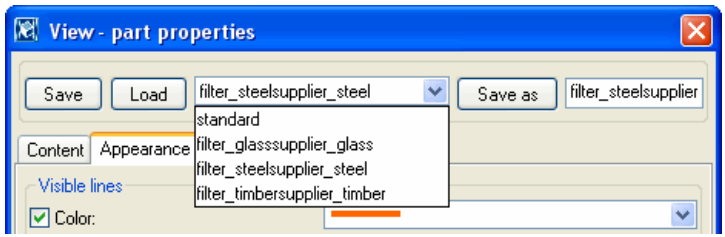
Always use view filters when you use the Drawing Classifier. The Drawing Classifier does not recognize select filters!

Creating drawing object property files

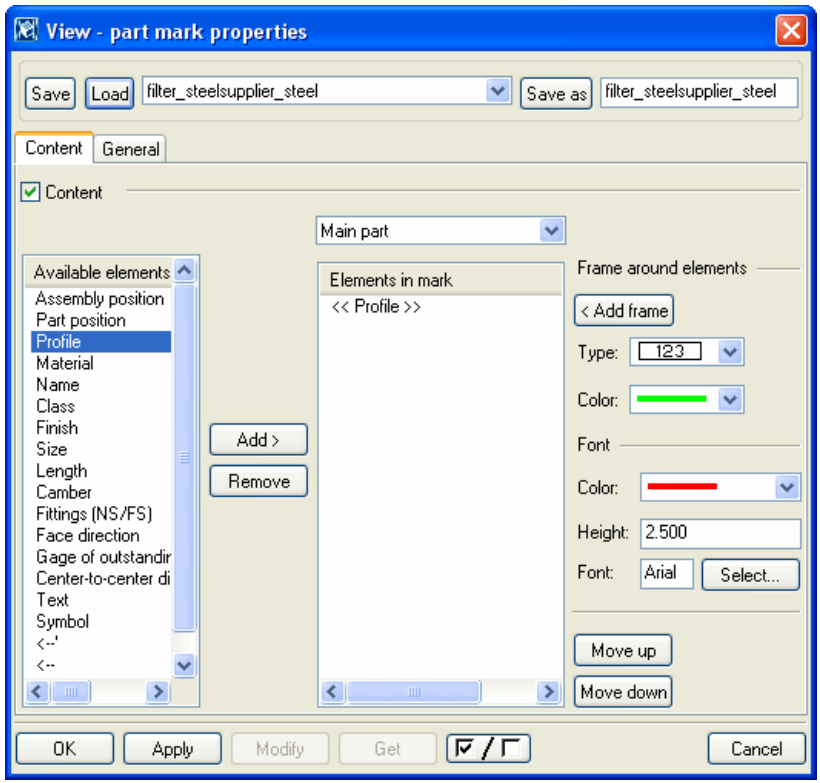
11. For user-defined attribute *Steel supplier* you can define the corresponding color who appears on a drawing. Double-click on a view and click on the button **Part.** Define the setting for steel. Enter the name ***filter_steelsupplier_steel*** in the text box and click <**Save as**>. (see picture below).



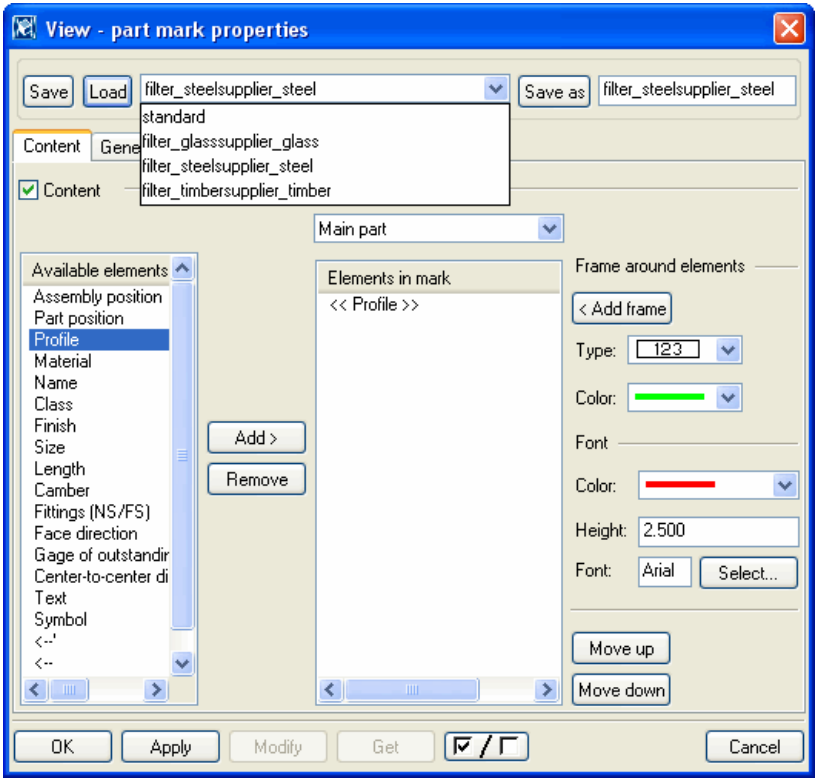
12. Repeat the steps for the user-defined attribute values you want to define. In this example: ***filter_timbersupplier_timber*** and ***filter_glasssupplier_glass***. Click <**Cancel**> to close the dialog box!



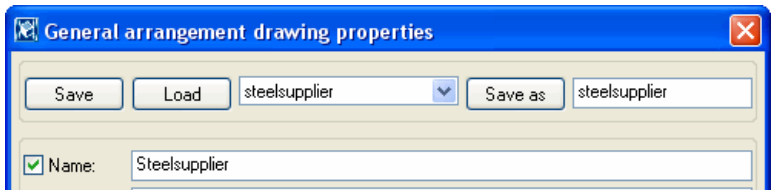
13. You can also define the part marks. Double-click on a view and click on the button **Part marks.** Define the setting for steel. Enter the name ***filter_steelsupplier_steel*** in the text box and click <**Save as**>. (see picture below).



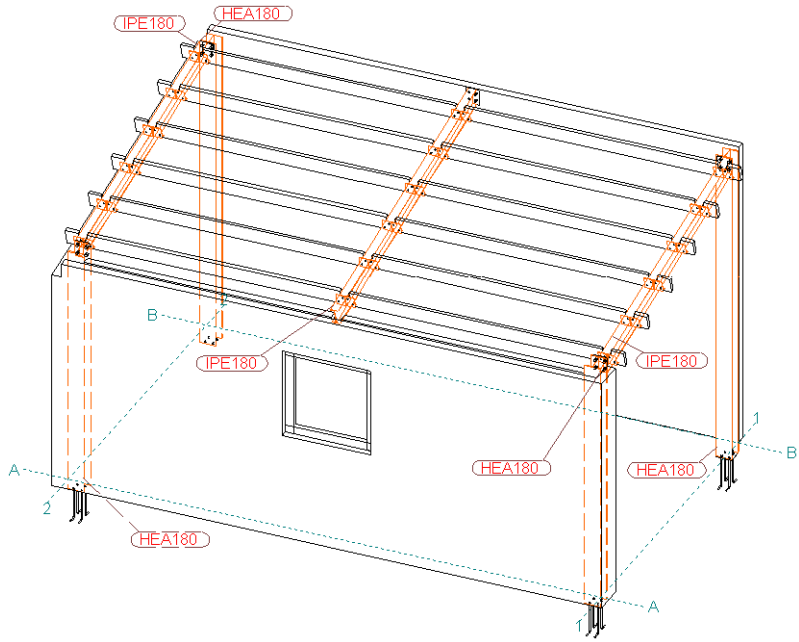
14. Repeat the steps for other user-defined attribute values you want to define. Click <**Cancel**> to close the dialog box!


View - part mark properties
Save
Load
filter_steelsupplier_steel
standard
filter_glasssupplier_glass
filter_steelsupplier_steel
filter_timbersupplier_timber
Save as
filter_steelsupplier_steel
Content
Gene
Content
Main part
Available elements
Assembly position
Part position
Profile
Material
Name
Class
Finish
Size
Length
Camber
Fittings (NS/FS)
Face direction
Gage of outstandir
Center-to-center di
Text
Symbol
<--'
<--
Add >
Remove
Elements in mark
<< Profile >>
Frame around elements
< Add frame
Type:
123
Color:
Font
Color:
Height:
2.500
Font:
Arial
Select...
Move up
Move down
OK
Apply
Modify
Get
Cancel

15. Go to **> Properties > General arrangement drawing...** Click the down arrow in the <**Load>** text box and select the ***Steelsupplier*** drawing properties you created. Click <**Load>** (see picture below). Now create the general arrangement drawing.



You will see that all parts with the user-defined attribute *Steel supplier* appear on the drawing in Orange, also the Hidden lines, the Own hidden lines and the marks are displayed.

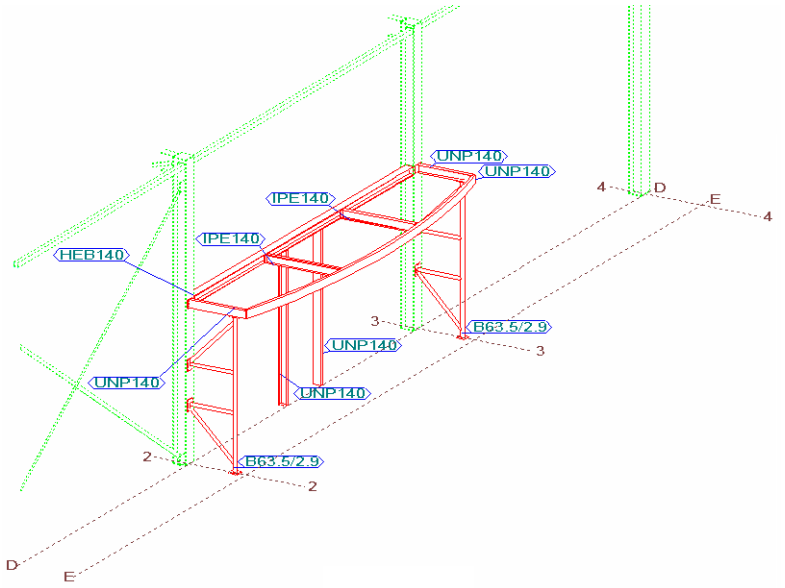


Exercise

It may occur that an existing project needs some additions, for example an awning. On a GA-drawing the following has to be set:

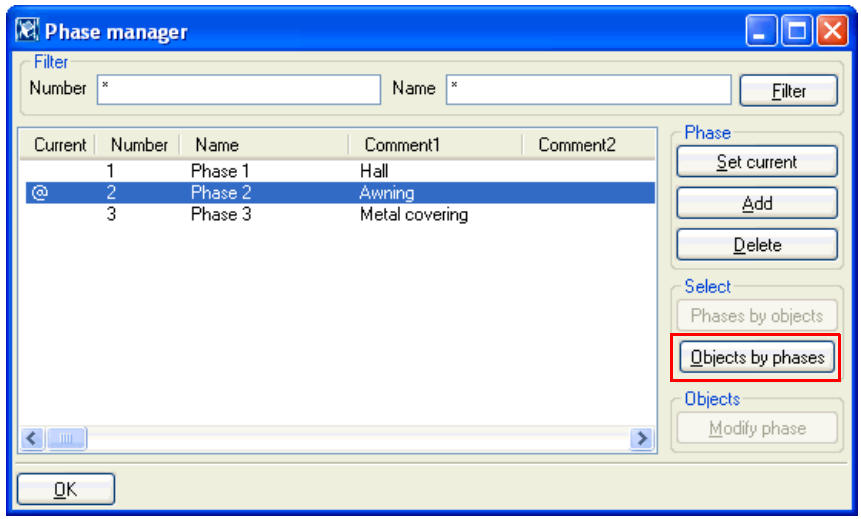
- Only for the awning, part marks have to be added, not for the existing structure
- The parts of the existing structure are displayed as dotted lines, the part lines of the awning are displayed as full lines.

For this exercise you can make use of model *Hal_1*. Create a GA drawing and make use of a classifier setting:



Step plan

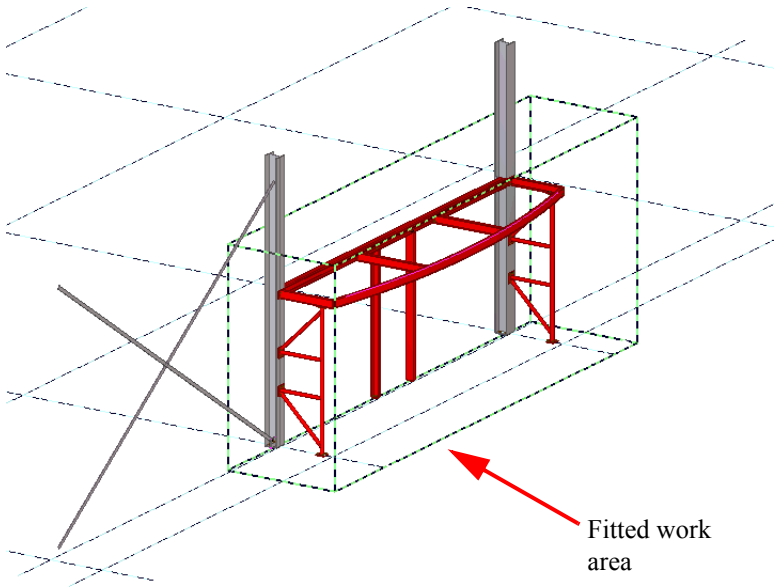
1. Make sure the awning is in a unique phase in the model (e.g. phase2)
2. Open the phase manager



3. Select phase 2 and click <*Objects by phase*>, all parts who belong to the awning will highlight in the model view

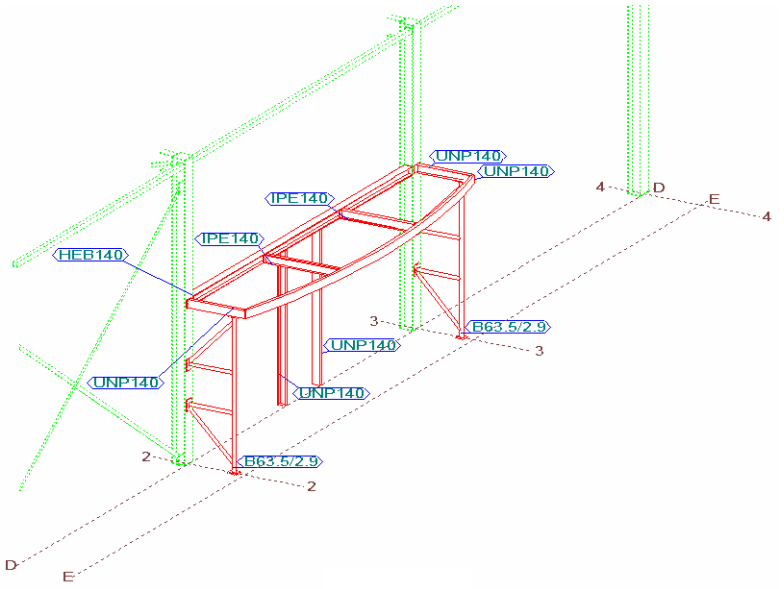


4. Click the icon ***Fit work area by parts in selected views*** or in the pull-down menu go to ***View*** > ***Fit work area,*** The work area is now sized to the awning:



5. In the GA drawing properties, adjust the setting "*phase*"

6. Now create a GA drawing by making use of the setting “phase”



If necessary, you can resize the view in the drawing.

Classifier for assembly- and cast unit drawings

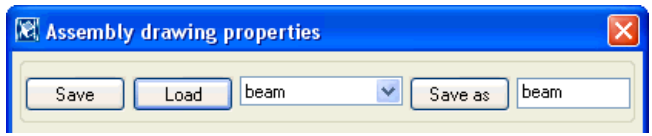
You can use the Drawing Classifier not only for general arrangement drawings, but also for assembly- and cast unit drawings.

You now learn how to create an assembly drawing in which several welded parts appear in different colors. These parts also appear with different part marks.

On the basis of an examples we will explain how the classifier works

Define drawing properties

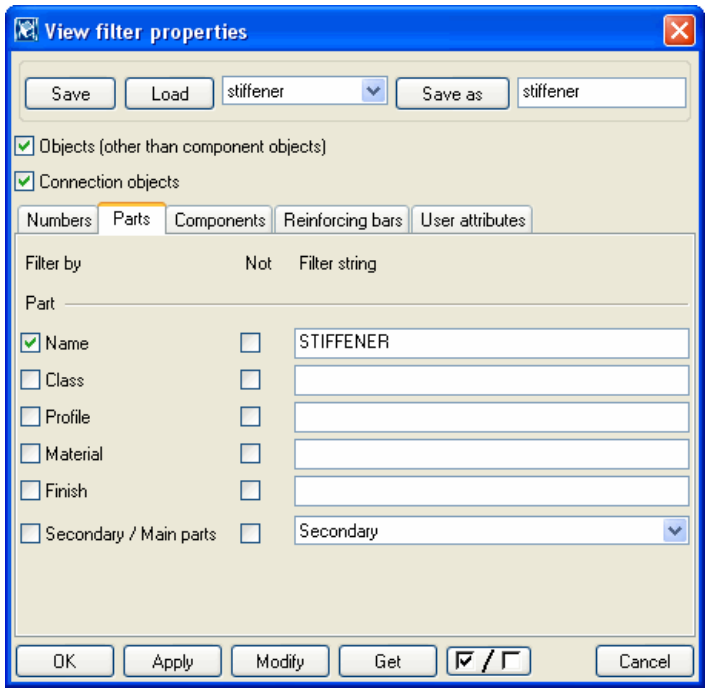
1. Go to **> Properties > Assembly drawing.** Create an assembly drawing property, enter the name “beam” in the text box and click <**Save as**>.



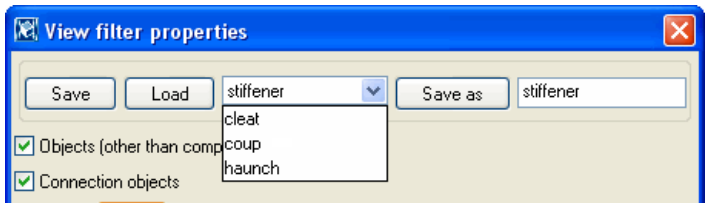
Creating view filters

2. In this example all different welded parts appear in a corresponding color on the assembly drawing. So for each welded part we create a view filter. You must create the filter for assembly drawings in the general arrangement drawing filter!

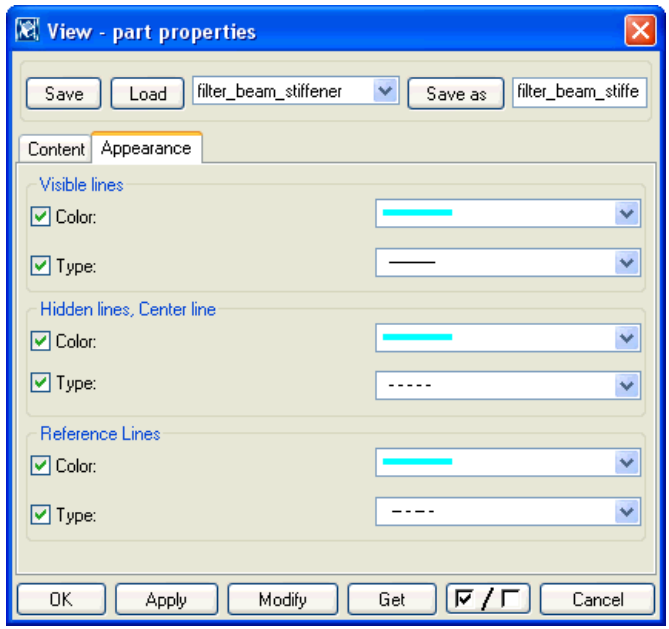
3. Open any drawing and double-click on a view. Click on the button **Filter.** In the dialog box **View filter properties** go to the tab **Parts** and define the setting. In the text box next to the <**Save as**> button, enter ***stiffener*** and click <**Save as**>, see picture below:



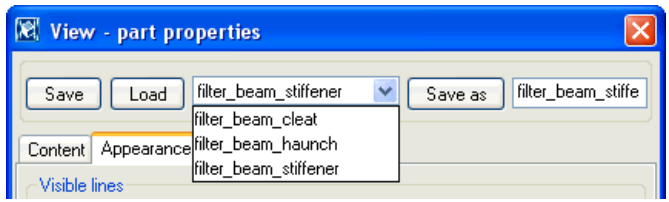
4. Repeat the steps for all welded parts you want to define. In this example: ***cleat*** and ***haunch***. Click <**Cancel**> to close the dialog box!



5. For each welded part you can define the corresponding color on the assembly drawing. In this example stiffener, cleat and haunch. Go to **> Properties > Assembly drawing... > Part.** Define the setting for stiffener. Enter the name ***filter_beam_stiffener*** in the text box and click <**Save as**>. (see picture below).

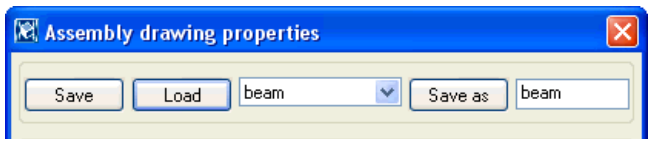


6. Repeat the steps for all parts you want to define. In this example: ***filter_beam_cleat*** and ***filter_beam_haunch***. Click <**Cancel**> to close the dialog box!

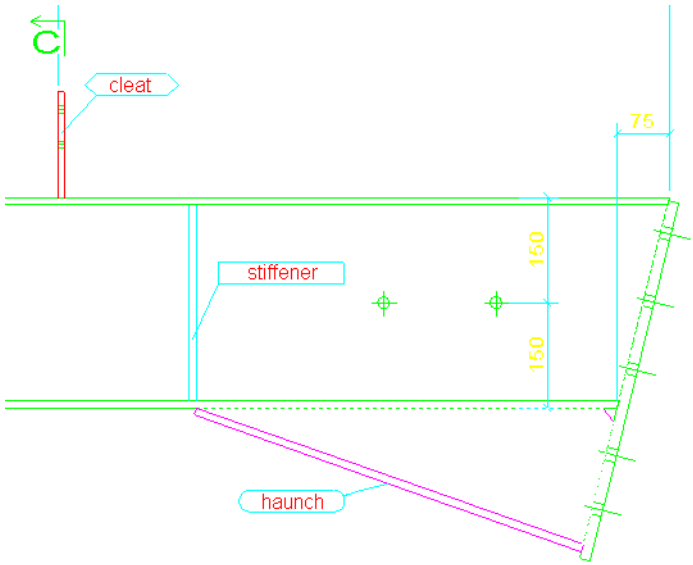


Creating drawings

7. Go to **> Properties > Assembly drawing...** Click the down arrow in the Load text box and select the ***beam*** drawing properties you created. Click <**Load**> (see picture below). Now create the general arrangement drawing.



You will see that a drawing is created in which different welded parts appear in different colors.



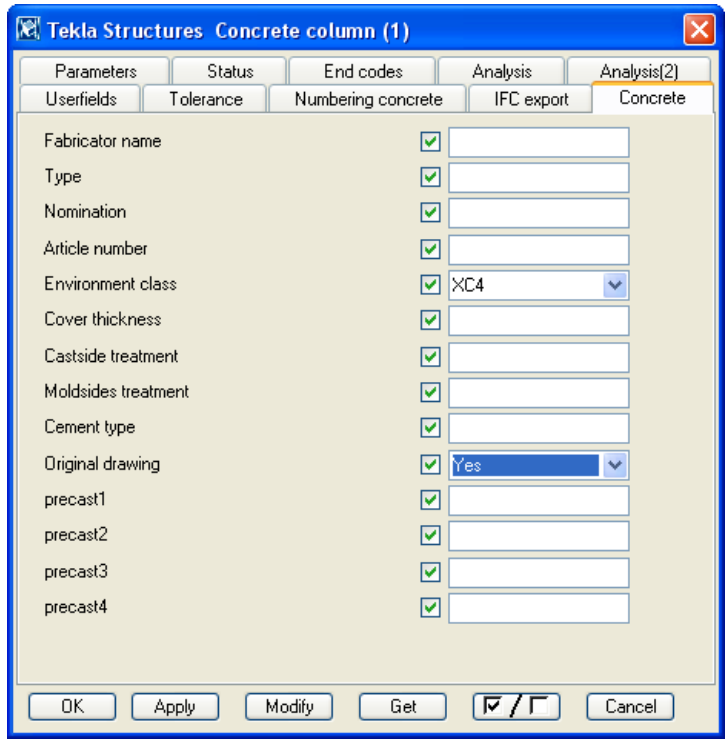


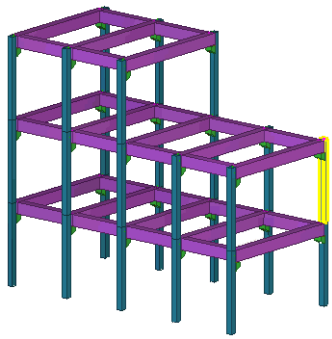
The Drawing Classifier can only be used for objects you can filter!

2.29 Original drawing

When you have several identical elements, only one element is used to create the drawing. In some cases it is handy to know from which original element the drawing has been created, i.e. to modify the user defined attributes or to reinforce that element.

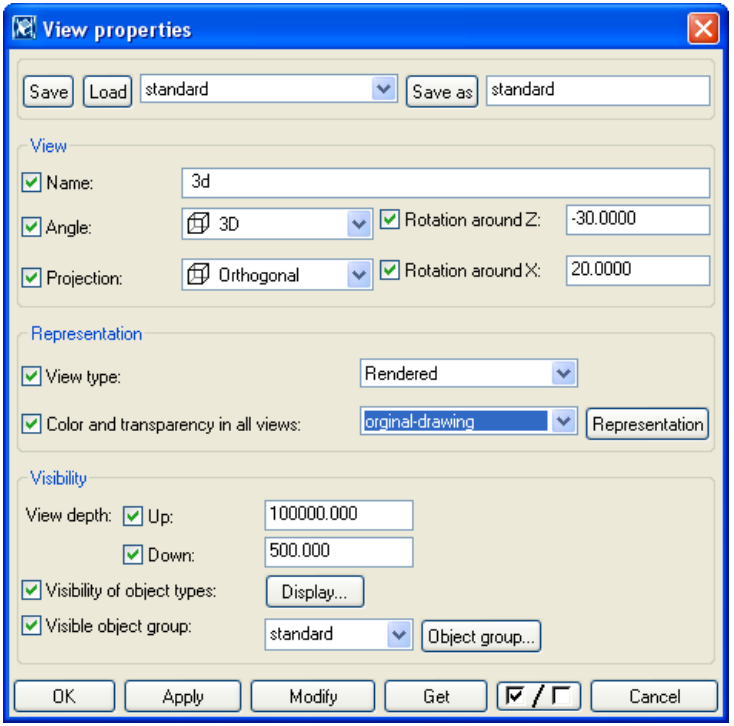
In the user defined attributes at the tab **Concrete** you can define an element to be the original for the drawing:

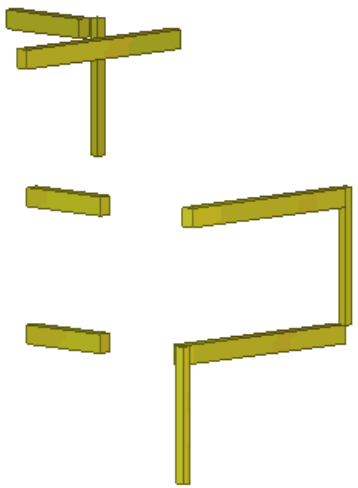




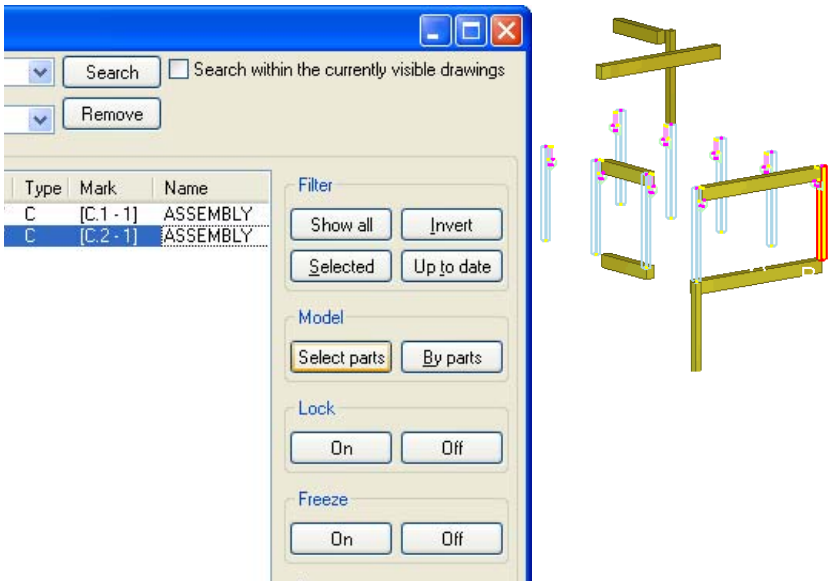
Inhere you set this element to be the original and by using a filter you can find out which elements are originals for the drawings.

Load the filter "original-drawing", all elements set to **Yes** in the user defined attributes will be displayed in the model view:





Then open the drawing list, select the drawing from which you want to know what the original element is and click **Select parts**. The original part will highlight.



You can also find out the original element in the model by using the list "id_original_drawing". Select the drawing in the drawing list, create a list from the selected drawing, select the ID-number in the list and the highlighted element in the model is the original for the drawing.

2.30 Explanation of file extensions

Description	File extension
Beam setting	*.prt
Column setting	*.clm
Concrete column setting	*.ccl
Concrete panel setting	*.cpn
Single part drawing setting	*.wd
Assembly drawing setting	*.ad
GA drawing setting	*.gd
Multi-drawing setting (in model)	*.md
View setting in the Model Editor	*.mvi
View setting in the Drawing Editor	*.vi
Reports	*.xsr
Report file (in Template/Reports editor)	*.rpt
Template file (in Template/Reports editor)	*.tpl
Wizard settings	*.dproc
Select Filter settings in the Model Editor	*.SObjGrp
Select Filter settings in the Drawing Editor	*.dsf
Weld setting (in model)	*.wld
Dimension setting	*.dim
Text file	*.txt
Table layout setting	*.lay
Symbol library file (in Symbol Editor)	*.sym

Databases	Filename	Location
Profile catalog	Profdb.bin	Profil
Material catalog	Matdb.bin	Profil
Bolt catalog	Screwdb.bin	Profil
Bolt assembly catalog	Assdb.bin	Profil
Plotter catalog	Plotdev.bin	System

2.31 Appendix

Concrete related reports

Lot-list-concrete

```
-------------------------------------------------------------------------------
Tekla Structures LOT LIST CONCRETE                          Page:     1
Model:templates and reports                                 Project:  Project number
                                                            Date:     04.01.2008
                                                            Time:     14:12:03

-------------------------------------------------------------------------------
    Unit       Qty.     Profile          Area. (m2)    Weight (kg)      Lot
-------------------------------------------------------------------------------
     W1          2     200*2600             34.6         7488.0         1
     C1          2     250*250               3.7          540.0         2
     B1          1     350*500              10.5         2520.0         3
-------------------------------------------------------------------------------
   Total for     5     units                48.9        10548.0
-------------------------------------------------------------------------------
```

Castunit-phase-location

```
-------------------------------------------------------------------------------
TEKLA STRUCTURES CASTUNIT PHASE LOCATION                    Page:   1
Model:templates and reports                                 Project:Project number
                                                            Date:   04.01.2008
                                                            Time:   14:19:23

-------------------------------------------------------------------------------
Unit        Name                Phase          Location     Top level  Bottom level
-------------------------------------------------------------------------------
B1          BEAM              1:Fase 1         C/4-5          +0.000     -0.500
C1          COLUMN            1:Fase 1         B/3            +3.600     +0.000
C1          COLUMN            1:Fase 1         B/4            +3.600     +0.000
W1          WALL              1:Fase 1         D/1-2          +2.600     +0.000
W1          WALL              1:Fase 1         E/1-2          +2.600     +0.000

-------------------------------------------------------------------------------
```

Castunit-embeds-list

```
-------------------------------------------------------------------------------
TEKLA STRUCTURES CASTUNIT EMBEDS LIST                       Page:   1
Model: templates and reports                                Project:Project number
                                                            Date:   11.01.2008
                                                            Time:   12:13:24

-------------------------------------------------------------------------------
Unit      Qty.   Name/Part    Size/Embed                  Weight(kg)   Vol.(m³)
===============================================================================
W2        1     WALL           200 *2600 *6000               7492      3.12
              -----------------------------------------------------------------
                4   D1         dowel_anchor DEMU2010-Ø16/M20x550
===============================================================================
W3        1     WALL           200 *2600 *6000               7488      3.12
              -----------------------------------------------------------------
                4   G1         GAIN 45-51x674
===============================================================================
          Total for         2    unit(s):                   14980      6.24
===============================================================================
```

Castunit-list

TEKLA STRUCTURES CASTUNIT LIST
Model:templates and reports

Page: 1
Project:Project number
Date: 04.01.2008
Time: 14:30:34

Unit	Quantity	Profile	Length	Weight (kg)	Volume (m³)	Grade
C1	2	250*250	3600	540	0.22	C45/55
W1	1	200*2600	6000	7488	3.12	C45/55
W2	1	200*2600	6000	7492	3.12	C45/55
W3	1	200*2600	6000	7488	3.12	C45/55
Total Cast Units: 5				23007 kg	9.6 m³	

Castunit-pos-list

TEKLA STRUCTURES CASTUNIT POSITION LIST
Model:templates and reports

Page: 1
Project:Project number
Date: 11.01.2008
Time: 12:16:26

Unit	Pos	Quantity	Profile	Length(mm)	Grade	Volume(m³/dm³)	Weight(kg)
W2	1				C45/55	3.12	7492
Betonl	1	200*2600	*6000		C45/55	3120.00	7488.00
W3	1				C45/55	3.12	7488
Betonl	1	200*2600	*6000		C45/55	3120.00	7488.00
					Total:	6.24	14980

This list only displays the cast unit shapes.

Combi_bnw_filler pl_treadedend_ list

```
--------------------------------------------------------------------------------
TEKLA STRUCTURES BOLTLIST COMPLETE                          Page:     1
Model: templates and reports                                Project:  Project number
                                                            Date:     04.01.2008
                                                            Time:     14:43:13

================================================================================
 Bolts, Nuts, Washers
--------------------------------------------------------------------------------
             Diam  Length(mm) Type                      Number   Type
      ------------------------------------------------------------------------
      Bolt    16      35      7990/125A/934               12   7990
      Bolt    20      70      7990/125A/934                6   7990
      Nut     16              934                          4
      Nut     20              934                          6
      Washer  16              125A                         4
      Washer  20              125A                         6
================================================================================
 Fillerplates
--------------------------------------------------------------------------------
      Partno.       Profile        Material       Length   Number    Weight
      ------------------------------------------------------------------------
      FLLRPLT1      FLAT10*160     S235JR          290.0   2         3.0
      ------------------------------------------------------------------------
                                              Totaal:       2         6.0
================================================================================
 Threaded ends
--------------------------------------------------------------------------------
      Partno.       Profile        Material       Length  Number
      ------------------------------------------------------------------------
```

This list shows bolts, nuts, washers, filler plates and threaded ends.

Cs_calc

```
I
I
I 04.01.2008


M D1        REBAR         4      1                       550                ANKER
                Phase 1
P REBAR      D1           1      1   DEMU2010-Ø16/M20    550        S500    ANKER
                Phase 1

M D2                      2      1                       128                ANKER
                Phase 1
P            D2           1      1   _M16                128        8.8     ANKER
                Phase 1

M G1                      4      1                       626                GAIN
                Phase 1
P            G1           1      1   GAIN 45-51          674        TIN     GAIN
                Phase 1

M S1        REBAR         4      1                       520                DOWEL-IN
                Phase 1
P REBAR      S1           1      1   DEMU2020-Ø16/M20    520        S500    DOWEL-IN
                Phase 1

M B1         CAST_UNIT    1      1                       6000
.050            Phase 1
P CONCRETE   Beton3       1      1   350*500             6000       C45/55  BEAM
.050            Phase 1

M C1         CAST_UNIT    1      1                       3600
.225            Phase 1
P CONCRETE   C1           1      1   250*250             3600       C45/55  COLUMN
.225            Phase 1
```

Report required for export to Calc.

CS_Calc_BD

Report required for the link to ConstruSteel.

Phase-castunit-list

```
------------------------------------------------------------------------------------
TEKLA STRUCTURES PHASE CASTUNIT LIST                          Page:     1
Model: templates and reports                                  Project:  Project number
                                                              Date:     04.01.2008
                                                              Time:     14:51:21

------------------------------------------------------------------------------------
Phase  Name        Cast unit   Number     Size h*w*l      Vol.(m³)     Weight (kg)
------------------------------------------------------------------------------------
 1     Phase 1
                   W1            1    2600* 200*  6000        3.12          7488
                   W2            1    2600* 200*  6000        3.12          7492
                   W3            1    2600* 200*  6000        3.12          7488
------------------------------------------------------------------------------------
           3  cast units in phase    1                        9.36         22468
====================================================================================

 2     Phase 2
                   C1            2     250* 250*  3600        0.22           540
------------------------------------------------------------------------------------
           2  cast units in phase    2                        0.45           540
====================================================================================

 3     Phase 3
                   B1            1     500* 350*  6000        1.05          2520
------------------------------------------------------------------------------------
           1  cast units in phase    3                        1.05          2520
====================================================================================

                       6  cast units in all phases          10.86         25528
------------------------------------------------------------------------------------
```

Phase_manager

```
TEKLA STRUCTURES PHASEMANAGER                                 Page:     1
Model:templates and reports                                   Project:  Project number
                                                              Date:     04.01.2008
                                                              Time:     14:53:28

------------------------------------------------------------------------------------
 Phase      Phase name          Phase comment 1       Phase comment 2
------------------------------------------------------------------------------------
     1      Phase 1
     2      Phase 2
     3      Phase 3
------------------------------------------------------------------------------------
```

Id-castunit-check-large-assembly

```
------------------------------------------------------------------------------------
 TEKLA STRUCTURES ID CASTUNIT CHECK FOR LARGE ASSEMBLIES LIST Page:     1
 Model:templates and reports                                  Project:  Project number
                                                              Date:     04.01.2008
                                                              Time:     15:00:53

------------------------------------------------------------------------------------
 ID           Cast unit  Name                          Size                 L*H*W
------------------------------------------------------------------------------------
 Id:   8895    W5        WALL                      12812      2600      3002    99.99
 Id:    744    W1        WALL                       6000      2600       200     3.12
 Id:    766    W2        WALL                       6000      2600       200     3.12
 Id:    912    W3        WALL                       6000      2600       200     3.12
 Id:    795    C1        COLUMN                     3600       250       250     0.23
 Id:    807    C1        COLUMN                     3600       250       250     0.23
------------------------------------------------------------------------------------
                                     END
------------------------------------------------------------------------------------
```

ld_castunit_ embedded_coor

```
TEKLA STRUCTURES ID_CASTUNIT + EMBEDS COÖRDINATE LIST          Page:    1
Model:templates and reports                                    Project:Project number
                                                               Date:    04.01.2008
                                                               Time:    15:03:20

ID              Assembly      Name                GRID POSITION     LEVEL
Id:766          W2            WALL                  D/1-2             +2.600
   Id:2418        D1             ANKER               D/1               +2.600
   Id:2677        D1             ANKER               D/1-2             +2.600
   Id:2931        D1             ANKER               D/1-2             +2.600
   Id:3184        D1             ANKER               D/2               +2.600
Id:912          W3            WALL                  D/1-2             +5.200
   Id:1403        G1             GAIN                D/1               +3.226
   Id:1656        G1             GAIN                D/1-2             +3.226
   Id:1907        G1             GAIN                D/1-2             +3.226
   Id:2157        G1             GAIN                D/2               +3.226

                                    END
```

ld-castunit-coordinates

```
TEKLA STRUCTURES ID_CASTUNIT COORDINATES LIST          Page:    1
Model:templates and reports                            Project: Project number
                                                       Date:    04.01.2008
                                                       Time:    15:06:50

ID            Unit          Name                Gridlocation    Level
Id:     795   C1            COLUMN              B/3               +3.600
Id:     807   C1            COLUMN              B/4               +3.600
Id:     744   W1            WALL                E/1-2             +2.600
Id:     766   W2            WALL                D/1-2             +2.600
Id:     912   W3            WALL                D/1-2             +5.200
Id:    8895   W5            WALL                B-C/4-5           +2.600
```

ld_castunit_sub assembly_coor

```
TEKLA STRUCTURES ID_CASTUNIT COÖRDINATE LIST           Page:    1
Model:templates and reports                            Project:Project number
                                                       Date:    04.01.2008
                                                       Time:    15:10:21

ID              Assembly      Name                GRID POSITION     LEVEL
Id:8944         W6            WALL                  B/1-2             +2.600
   Id:8956                    BEAM                  B/1-2             +0.000

                                    END
```

ld_bolt_list

```
-------------------------------------------------------------------------------------
 TEKLA STRUCTURES ID BOLTLIST                                    Page:     1
 Model:templates and reports                                     Project:  Project number
                                                                 Date:     04.01.2008
                                                                 Time:     15:12:08

-------------------------------------------------------------------------------------
 Number      Bolt                                                ID
-------------------------------------------------------------------------------------
   4         BOLT       7990          16 x 35                  Id:       6078
   4         BOLT       7990          16 x 35                  Id:       6913
   4         BOLT       7990          16 x 35                  Id:       7734
-------------------------------------------------------------------------------------
                                    END
-------------------------------------------------------------------------------------
```

Pressing a ID-line in the list will highlight the bolts in the model.

ld_bolt_list_100

```
-------------------------------------------------------------------------------------
 TEKLA STRUCTURES ID BOLT_LIST_100                               Page:     1
 Model:templates_ENU                                             Project:  project number
                                                                 Date:     16.11.2007
                                                                 Time:     17:02:10

-------------------------------------------------------------------------------------
 Number      Bolt                                    ID
-------------------------------------------------------------------------------------
   4         BOLT    931        16 x 100             Id:       25184
-------------------------------------------------------------------------------------
                                    END
-------------------------------------------------------------------------------------
```

This report only shows bolts with a length of 100 mm. Mostly these are bolts which are modelled incorrect.

Press the button **Fit work area by parts in selected views** if you cannot find the bolts.

ld_componentlist

```
-------------------------------------------------------------------------------------
TEKLA STRUCTURES COMPONENTLIST                                   Page:     1
Model:templates and reports                                      Project:  Project number
                                                                 Date:     04.01.2008
                                                                 Time:     15:26:32

-------------------------------------------------------------------------------------
                                                                                1=green
 Component                                          Part    Part    Part        2=yellow
 number              Number   Name                  MP      1st SP  2nd SP      3=red
-------------------------------------------------------------------------------------
 Id:10884                 1    Inbedded              Beton5                        1
 Id:10540      110        1    Concrete console      Beton2  Concre                1
-------------------------------------------------------------------------------------
                        2   components
-------------------------------------------------------------------------------------
```

Pressing a ID-line in the list will highlight the object in the model.

ld_length_list

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES ID LENGTH LIST                                  Page:     1
Model:templates_ENU                                              Project:  project number
                                                                 Date:     23.11.2007
                                                                 Time:     10:10:03

------------------------------------------------------------------------------------------
ID                        Part                  Length
------------------------------------------------------------------------------------------
Id:        664            Pr2                     3000
Id:        637            Pr1                     1500
Id:        974            P1                       275
Id:       1010            P1                       275
Id:       1224            P1                       275
Id:       1249            P1                       275
------------------------------------------------------------------------------------------
                                    END
------------------------------------------------------------------------------------------
```

ld_loose_parts

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES ID LOOSE PARTS LIST                             Page:    1
Model:templates and reports                                      Project: Project number
                                                                 Date:    04.01.2008
                                                                 Time:    16:06:31

------------------------------------------------------------------------------------------
ID              Part       Assembly  Profile              Grade        Length (mm)
------------------------------------------------------------------------------------------
Id: 731         W1         W1        200*2600             C45/55           6000
Id: 11160       C5         C5        250*250              C45/55           3600
Id: 11172       C5         C5        250*250              C45/55           3600
------------------------------------------------------------------------------------------
                                    END
------------------------------------------------------------------------------------------
```

This report shows assemblies consisting of only one part. For example plates you forgot to weld. Pressing a ID-line in the list will highlight the profile in the model.

ld_original_drawing

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES ID_ORIGINAL_DRAWING_LIST                        Page:     1
Model: templates and reports                                     Project:  Project number
                                                                 Date:     04.01.2008
                                                                 Time:     16:09:33

------------------------------------------------------------------------------------------
     Original part for drawing
------------------------------------------------------------------------------------------
     Id:731
------------------------------------------------------------------------------------------
                                    END
------------------------------------------------------------------------------------------
```

In the drawing list, select a drawing and create the report; if a line is selected, the corresponding part from which the drawing was created ,will highlight in the model.

Id_part_list

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES ID PART LIST                                    Page:    1
Model: templates and reports                                     Project: Project number
                                                                 Date:    04.01.2008
                                                                 Time:    16:27:24

------------------------------------------------------------------------------------------
 ID             Part     Profile              Grade       Length (mm)   Shorten
------------------------------------------------------------------------------------------
 Id: 760        Betonl   200*2600             C45/55          6000
 Id: 907        Betonl   200*2600             C45/55          6000
 Id: 2413       D1       DEMU2010-Ø16/M20     S500             550
 Id: 2674       D1       DEMU2010-Ø16/M20     S500             550
 Id: 2928       D1       DEMU2010-Ø16/M20     S500             550
 Id: 3181       D1       DEMU2010-Ø16/M20     S500             550
 Id: 1396       G1       GAIN 45-51           TIN              674
 Id: 1653       G1       GAIN 45-51           TIN              674
 Id: 1904       G1       GAIN 45-51           TIN              674
 Id: 2154       G1       GAIN 45-51           TIN              674
------------------------------------------------------------------------------------------
                                     END
------------------------------------------------------------------------------------------
```

Pressing a ID-line in the list will highlight the profile in the model.

Embedded-parts

```
------------------------------------------------------------------------------------------
 TEKLA STRUCTURES INBEDDED PARTS                                  Page:   1
 Model:templates and reports                                      Project:Project number
                                                                  Date:   04.01.2008
                                                                  Time:   16:32:12

------------------------------------------------------------------------------------------
 Part no.  Qty.     Profile          Length     Grade         Comment
------------------------------------------------------------------------------------------
 1002         1     350*350          250        C45/55
 1003         1     TRZ200-300-150   3000       C45/55
 1004         2     GAIN 42-48       500        GAIN
 2001         1     FLAT10*160       290        S235JR
 2002         1     FLAT15*350       210        S235JR
 Beton5       1     200*2600         3000       C45/55
------------------------------------------------------------------------------------------
 Number of embeds:   7
------------------------------------------------------------------------------------------
```

Locked_objects_ report

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES LOCKED OBJECTS                                  Page:    1
Model: templates_ENU                                             Project: project number
                                                                 Date:    23.11.2007
                                                                 Time:    14:21:19

------------------------------------------------------------------------------------------
Description (Name, Profile, ID)                  Owner (Domain, user)        Locked
------------------------------------------------------------------------------------------
PART IPE300 Id: 664                              PCLEON/leon                  Yes
PART IPE240 Id: 1436                             PCLEON/leon                  Yes
PART HEA240 Id: 1284                             PCLEON/leon                  Yes
PART HEA240 Id: 1274                             PCLEON/leon                  Yes
PART HEA240 Id: 675                              PCLEON/leon                  Yes
------------------------------------------------------------------------------------------
                                     END
------------------------------------------------------------------------------------------
```

Neoprene

TEKLA STRUCTURES NEOPRENE LIST
Model:templates and reports

Page: 1
Project:Project number
Date: 07.01.2007
Time: 12:05:06

Part	Unit	Quantity	Profile	Length (mm)	Area (m2)
NE01	NE01	4	FLAT5*100	100	0.040
	Total:	4 units			0.04 m2

Notification_ report

PROJECT NEWS!!!!

This is an automatic newsflash for project participants

This message is shown when anyone opens the model or runs "notification_report" report template

Customize the content by editing notification_report.rpt

Deactivate be setting XS_SHOW_NOTIFICATION_REPORT to FALSE.

Tips how to utilize this to improve project communication and workflow

can be found from the Release Notes and What's New material

NOTIFICATION OF ASSIGNMENTS CONTRACT NO: 2007-12345 Page: 1
Objects assigned to Beren Date: 13.11.2007

Object	Name	Assigned to	Assigned by	Comment	
Assembly	KOLOM	Berend	Berend		Assembly Id:9997
Part	KOLOM	Berend	Berend		Part Id:9992

1 assemblies, 1 parts and 0 drawings have been assigned to Berend

Area

TEKLA STRUCTURES AREA LIST
Model:templates and reports

Page: 1
Project:Project number
Date: 04.01.2008
Time: 16:34:51

Unit	Quantity	Volume (m³) Gross	Volume (m³) Net	Area (1 plane) Gross	Area (1 plane) Net	Area around(m²)
B6	1	1.02	1.02	2.92	2.92	10.57
C5	2	0.45	0.45	1.80	1.80	7.45
W6	1	1.70	1.70	8.51	8.51	22.14
Total Cast Units:	4			0.0 m²	13.2 m²	40.2 m²

Area-walls

TEKLA STRUCTURES AREALIST FOR WALLS
Model:templates and reports

Page: 1
Project:Project number
Date: 04.01.2008
Time: 16:36:12

Unit	Quantity	Volume (m³) Gross	Net	Area (1 plane) Gross	Net	Area around(m²)
W1	1	3.12	3.12	15.60	15.60	34.64
W2	1	3.12	3.12	15.60	15.60	34.78
W3	1	3.12	3.12	15.61	15.61	35.05
W4	1	2.60	2.60	13.00	13.00	29.04
W5	1	3.48	3.48	17.41	17.41	39.58
W6	1	1.70	1.70	8.51	8.51	22.14
Total walls:	6			85.7 m²	85.7 m²	195.2 m²

Pile-reference

TEKLA STRUCTURES PILE REFERENCE
Model: templates and reports

Page: 1
Project:Project number
Date: 07.01.2008
Time: 12:15:32

Assembly	Size	Number	Top of pile	Order length	Length (starter bars excl.)
P1	250*250	12	-4000	4250	3820
P2	250*250	5	-3500	3750	3320

Pile-reference-location

TEKLA STRUCTURES PILE REFERENCE LOCATION
Model: templates and reports

Page: 1
Project:Project number
Date 07.01.2008
Time: 12:19:59

Assembly	Number	Size	Top of Pile	Length	Location X,Y
P2		250*250	-3500	3320	18000 , 54000

Drawing_revision_list

TEKLA STRUCTURES DRAWING REVISION LIST
Model:templates and reports

Page: 1
Project: Project number
Date: 11.01.2008
Time: 13:18:33

Drawing	Date	Modification date
C [C.5 - 1]	04.01.2008	

Rev. no.	Rev. mark	Rev. date	Description
2	B	14-1-2008	

Rev. no.	Rev. mark	Rev. date	Description
1	A	7-1-2008	

Drawing_last_revision_list

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES DRAWING LAST REVISION LIST                          Page:     1
Model:templates and reports                                          Project:  Project number
                                                                     Date:     07.01.2008
                                                                     Time:     09:04:44

------------------------------------------------------------------------------------------
Drawing       Revision     Revision date     Description
------------------------------------------------------------------------------------------
C   [C.5 - 1] 2    B        14-1-2008
------------------------------------------------------------------------------------------
C   [W.1 - 1] 2    B        14-1-2008
------------------------------------------------------------------------------------------
G   [1]       2    B        14-1-2008
------------------------------------------------------------------------------------------
```

Starter-bars

```
------------------------------------------------------------------------------------------
 TEKLA STRUCTURES STARTER BAR LIST                                   Page:   1
 Model:templates and reports                                         Project:Project number
                                                                     Date:   07.01.20
                                                                     Time:   09:09:18

------------------------------------------------------------------------------------------
   Assembly Number    Starter bar type                                        Partno.
------------------------------------------------------------------------------------------
   SB1          14    DEMU2020-Ø16/M20x600                                    SB1
   SB2          21    DEMU2020-Ø16/M20x750                                    SB2

------------------------------------------------------------------------------------------
      Total:    35 starter bars
------------------------------------------------------------------------------------------
```

Sub-assembly-part-list

```
------------------------------------------------------------------------------------------
 TEKLA STRUCTURES SUB ASSEMBLY PART LIST                             Page:   1
 Model: templates and reports                                        Project:Project number
                                                                     Date:   07.01.2008
                                                                     Time:   09:14:40

------------------------------------------------------------------------------------------
 Assembly    Part      Number Profile          Length(mm)  Area(m2)     Weight(kg)
==========================================================================================
 B5                    1                                   1.59         58.9
------------------------------------------------------------------------------------------
             1005      2   FLAT10*70            275        0.04          1.5
             Pr5       1   IPE300              1296        1.50         55.9
==========================================================================================
                                                    Total (kg):          58.9
------------------------------------------------------------------------------------------
```

Drawing_list_file

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES DRAWING LIST FILE                                   Page:     1
Model:templates and reports                                          Project:  Project number
                                                                     Date:     07.01.2008
                                                                     Time:     09:15:57

------------------------------------------------------------------------------------------
A-=ASSEMBLY DR'G   W-=WORKSHOP(SINGLE PART)DR'G   G-=GENERAL DR'G   M-=MULTI DR'G
------------------------------------------------------------------------------------------
Name/no.    Subject          Size(h*w)      Created      Plotted      Plotfile
------------------------------------------------------------------------------------------
C   [C.5 -  ASSEMBLY          420x297    04.01.2008                D0000011218.dg
------------------------------------------------------------------------------------------
C   [W.1 -  ASSEMBLY          420x297    04.01.2008                D0000011198.dg
------------------------------------------------------------------------------------------
G   [1]     GENERAL DRAWING   1189x841    04.01.2008                D0000000152.dg
------------------------------------------------------------------------------------------
```

Drawing_list

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES DRAWING LIST                                    Page:     1
Model:templates and reports                                      Project:  Project number
                                                                 Date:     07.01.2008
                                                                 Time:     09:17:15

------------------------------------------------------------------------------------------
Drawing          Description      Date          Modify date   Rev.   Size
------------------------------------------------------------------------------------------
C   [C.5 - 1]    ASSEMBLY         04.01.2008                    2     420x297
C   [W.1 - 1]    ASSEMBLY         04.01.2008                    2     420x297
G   [1]          GENERAL DRAWING  04.01.2008                    2    1189x841
------------------------------------------------------------------------------------------
```

Multi-drawing-contentlist

```
------------------------------------------------------------------------------------------
 MULTIDRAWING LIST                                               Page:     1
 Model: templates and reports                                    Project:  Project number
                                                                 Date:     07.01.2008
                                                                 Time:     09:42:28

------------------------------------------------------------------------------------------
 Name        Drawing size   Created    Revision          Multidrawing content
------------------------------------------------------------------------------------------
 Multi-draw  1189*841     07.01.2008     0     POS.      NAME          PROFILE
------------------------------------------------------------------------------------------
                                          C Betonl    WALL          200*2600
                                          C C1        COLUMN        250*250
                                          C Concrete  BEAM          350*500
------------------------------------------------------------------------------------------
```

Multi_drawing_list

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES MULTI DRAWING LIST                              Page:     1
Model:templates and reports                                      Project:  Project number
                                                                 Date:     07.01.2008
                                                                 Time:     09:43:21

------------------------------------------------------------------------------------------
Type Drawing              Create date        Drawing name
------------------------------------------------------------------------------------------
 M   [1]                  07.01.2008         Multi-drawing
------------------------------------------------------------------------------------------
```

Xyz-coordinates-Total Station.txt

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES Totalstationlist                                Project: Project number
Model: templates and reports                                     Date:    07.01.2008
                                                                 Time:    09:45:47


------------------------------------------------------------------------------------------
 Number                  Start                                 End
                X          Y          Z          X          Y          Z
------------------------------------------------------------------------------------------
          ,   3000   ,  11999   ,    0     ,   3000   ,  11999   ,   3600
          ,   3000   ,  18000   ,    0     ,   3000   ,  18000   ,   3600
          ,   3000   ,   7071   ,    0     ,   3000   ,   7071   ,   3600
------------------------------------------------------------------------------------------
                                    Total parts: 3
------------------------------------------------------------------------------------------
                                    END
------------------------------------------------------------------------------------------
```

Report for Total Station. (tool for positioning columns on site)

Concrete related templates

Castunit

CASTUNIT: C5	
w*h*l 250*250 *3600	
Quantity:2	Volume (m³): 0.22
Grade: C45/55	Weight (kg): 540

Castunit2

Cover thickness:
Environmentclass: XC4
Castside:
Moldside:

Castunit_phase_number

CASTUNIT	SIZE w*h*l	WEIGHT	PHASE	NUMBER	TOTAL
W5	200*2600*12812	8356	1	1	1
W4	200*2600* 5000	6240	1	1	1
C4	250* 250* 3600	609	2	1	1
C1	250* 250* 3600	540	2	1	1
B6	350* 500* 5845	2452	1	1	1
B1	350* 500* 6000	2520	3	1	1

Castunit-revision

CASTUNIT LIST				
Unit	Profile	Qty.	Volume (m³)	Revision
C5	250*250	4	0.22	B 14-1-2008
W1	200*2600	1	3.12	B 14-1-2008
Total 5 unit(s)			13.38	

Company_name

Telephone +31 (0)316 340192 Fax +31 (0)316 342454

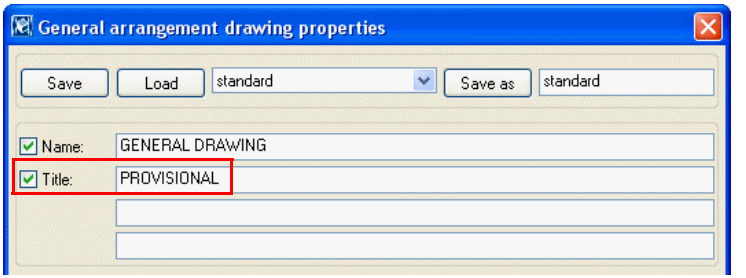
CONSTRUCTION INDUSTRIES SOFTWARE SUPPLIER

E-mail info@construsoft.com Internet: www.construsoft.com

Final_provisional

PROVISIONAL

Enter here in the drawing:



Cast-unit-list

CAST UNIT LIST			
Assembly	Size (W*H*L)	Number	Weight (kg)
B1	350 * 500 * 6000	1	2520
B6	350 * 500 * 5845	1	2452
C1	250 * 250 * 3600	1	540
C4	250 * 250 * 3600	1	609
W4	200 * 2600 * 5000	1	6240
W5	200 * 2600 * 3696	1	8356

phase-number-castunit

PHASE	NUMBER
1	2
2	1

Phasemanager

Phasemanager			
Phase	Phase naam	Comment 1	Comment 2
1	Phase 1		
3	Phase 3		

Driven-pile-list

DRIVEN PILE LIST

Fixed point: Soil drlling test report no.:
Level taken at: Level to be checked by builder
Type drive block: Pile type:

Pile no.	Pile size [mm]	Quantity	Pile length [mm]	Driving depth rel. to LEVEL [m]	Chop height rel. to LEVEL [m]	Fs (max) [kN]
P1	250*250	12	4250	-4.000	-0.180	
P2	250*250	5	3750	-3.500	-0.180	
Total quantity of piles:		17	(to be checked by builder)			

Comments:



Starter bar piles: > 300 mm
chop height piles: 20+ bottom level beam

Embeds

EMBEDS: B6		1 pcs.
Partnr.	Embeds (profile x length)	Quantity
1004	GAIN 42-48x500	2
	Total:	2

Embeds2

Embeds assembly: B7					
Partnr.	Profile	Length	Quantity	Grade	Commentar
D4	DEMU2010-Ø12/M16	400	1	S500	
D3	_M16	128	2	8.8	

Kistzijde

	kistzijde vuil werk
	kistzijde ruw aanstortvlak
	kistzijde schoon werk
	kistzijde schoon werk, inpoetsen
	kistzijde schoon werk, volgens monsterno. ....

BS_baseplate

Grid	Profile	Bottom side base plate
E/5	HEA240	-0.050
E/5-6	HEA240	+0.000

Pile-list

PILE LIST

Pile number	Pile size	Quantity	Length (mm) excl. starter bar	Top pile (m rel. to Level)	Driving depth (m rel. to Level)
P3	D300	10	3800	-0.200	-4.000

Pathname

D:\TeklaStructuresModels\13.1\NED\Construsoft\templates_trair

Revision

3	C	Notes carried out	21-1-2008
2	B	Modifications due to changed calculations	14-1-2008
1	A	Recess and various	7-1-2008
Rev. no.	Revision	Revision description	Revision date

Revision_last

3	C	Notes carried out	21-1-2008
Rev. no.	Revision	Revision description	Revision date

Wall-tie

W/1	WALL-TIE Ø6	20

Dowel-in-list

DI/1	Ø16	30	S500	800
Part	Profile	Quantity	Material	Length
DOWEL-IN LIST		Total: 30	dowel-in on this drawing	

Dowel-in-list2

DI/1	Ø 16	30	S500	800	+0.30	-0.50
Part	Profile	Qty	Grade	Length	T.S.	B.S.
DOWEL-IN LIST		Total: 30	dowel-in on this drawing			

Template_A

	Project:
	Drawing:
	Status:

Template_B

PROJECT:	Designer:
SUBJECT:	Date:
	Scale:
BUILDER:	

Template_B2

PROJECT:	Designer:
SUBJECT:	Date:
	Scale: See view
BUILDER:	

Stortzijde

▷	stortzijde vuil werk
▷	stortzijde ruw aanstortvlak
▶	stortzijde schoon werk gespaand
◆	stortzijde schoon werk, gerold
▶	stortzijde schoon werk, volgens monsterno. ....

Grid

ASSEMBLY	GRID
C2	E/5-6
C6	E/5

Tekla_structures

TEKLA®
Structures
www.construsoft.com

Tolerances

TOLERANCES	Length:	Embeds:
Width:	Thickness:	Height:
Diagonal:	Straightness:	Curve:
Flatness:	Endplane:	Perp. plane:

Steel related reports

Anchorlist

```
-------------------------------------------------------------------------------
TEKLA STRUCTURES ANCHOR LIST                                  Page:     1
Model:templates_ENU                                           Project:  project number
                                                              Date:     14.11.2007
                                                              Time:     16:09:59

-------------------------------------------------------------------------------
Part      Profile                    Number
-------------------------------------------------------------------------------
P1        ANCHOR_M16                   40
P6        ANCHOR_M12                   60
===============================================================================
                            Total    100   parts
-------------------------------------------------------------------------------
```

Use the "anchor filter" to select!

Assembly_bolt_list2

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES ASSEMBLY BOLT LIST                                    Page:     1
Model:templates_ENU                                                    Project:  project number
                                                                       Date:     14.11.2007
                                                                       Time:     16:32:43

------------------------------------------------------------------------------------------
AssemblyDiam.  Length    Type        Number          Connected assemblies
------------------------------------------------------------------------------------------
B1                                     1
------------------------------------------------------------------------------------------
        16     35        931-8.8         4
        16     45        931-8.8         4           B2
------------------------------------------------------------------------------------------
B2                                     2
------------------------------------------------------------------------------------------
        16     45        931-8.8         4           B1
------------------------------------------------------------------------------------------
```

Assembly_bolt_list3

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES ASSEMBLY BOLT LIST:                                   Page:     1
Model: templates_ENU                                                   Project:  project number
                                                                       Date:     14.11.2007
                                                                       Time:     16:34:00

------------------------------------------------------------------------------------------
Diam.  Length  Grade       Number Hole diam.Connected parts        for assemblies
------------------------------------------------------------------------------------------
 16    45        931-8.8       4     18     P9->P9                B2
 16    45        931-8.8       4     18     P9->P9                B1
 16    35        931-8.8       4     18     Pr8->
------------------------------------------------------------------------------------------
```

Assembly_bolt_list_s_w

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES ASSEMBLY BOLT LIST                                    Page:     1
Model:templates_ENU                                                    Project:  project number
                                                                       Date:     14.11.2007
                                                                       Time:     16:34:51

------------------------------------------------------------------------------------------
Assembly  Bolt diam.   Length  Number     Connected assemblies  Site/ Workshop
------------------------------------------------------------------------------------------
B1          16            35       8                           Site
B2          16            45       4          B1               Site
------------------------------------------------------------------------------------------
```

Assembly_COG

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES ASSEMBLY COG LIST                                     Page:     1
Model:templates_ENU                                                    Project:  project number
                                                                       Date:     14.11.2007
                                                                       Time:     16:36:17

------------------------------------------------------------------------------------------
Assembly Profile        Number   C e n t r e  o f  G r a v i t y       Weight(kg)
                                   X           Y           Z
------------------------------------------------------------------------------------------
B1       IPE300            1      12000.00    6002.16     -149.41          260.0
B2       IPE300            1      12000.00    1535.67     -150.00          129.6
------------------------------------------------------------------------------------------
                 Total     2 assemblies                                    389.7
------------------------------------------------------------------------------------------
```

Assembly_COG_grid

```
TEKLA STRUCTURES ASSEMBLY_COG_GRID MODEL:                              Page:1

                                                                 Date: 14.11.2007
                                                                 Time: 16:36:59

 Assembly Grid        Number  Name          C e n t r e  o f  g r a v i t y
          Point(s)                        X COORDIN.   Y COORDIN. Z COORDIN.

 B1       E/1-3          1    BEAM            12000.0      6002.2     -149.4
 B2       E/1-2          1    BEAM            12000.0      1535.7     -150.0
```

Assembly_coordinates_list

```
TEKLA STRUCTURES ASSEMBLY COORDINATES LIST                  Page:     1
Model:templates_ENU                                         Project:  project number
                                                            Date:     14.11.2007
                                                            Time:     16:49:50

Assembly Number  Phase   Name          Grid point          Level

B1           1   1       BEAM          E/1-3                +0.095
B2           1   1       BEAM          E/1-2                +0.000
```

Assembly_finish_list

```
TEKLA STRUCTURES ASSEMBLY FINISH LIST                       Page:     1
Model:templates_ENU                                         Project:  project number
                                                            Date:     14.11.2007
                                                            Time:     16:51:05

Assembly    Profile       Number     Area (m2)        Finish

B1          IPE300             1          7.14        Painted
B3          IPE300             1          3.56        Galvanised
                  Totals:    2         10.70 (m2)
```

Assembly_list

```
TEKLA STRUCTURES MERKENLIJST                                Page:    1
Model: templates_ENU                                        Project: project num
                                                            Date:    16.11.2007
                                                            Time:    16:19:17

 Assembly    Number      Profile              Length    Area (m2)     Weight (kg)

 B1              1       IPE300                 5999     260.0    7.14
 C6              1       IPE300                 2997     127.8    3.56
      Total      2  Assemblies                          10.70           387.6
```

Assembly_lot_list

```
TEKLA STRUCTURES ASSEMBLY LOT LIST                              Page:     1
Model:templates_ENU                                             Project:  project number
                                                                Date:     16.11.2007
                                                                Time:     16:21:18

  Assembly    Number    Profile            Length               Weight(kg)    Lot
  C6              1     IPE300             2996. 300.0 150.0        127.8      1
  C7              1     IPE300             3132. 300.0 150.0        133.5      1
  B1              1     IPE300             5999. 395.0 150.0        260.0      2
  Totals for      3  assemblies                                     521.4
```

Assembly_nut _list

```
TEKLA STRUCTURES ASSEMBLY NUT LIST                              Page:     1
Model:templates_ENU                                             Project:  project number
                                                                Date:     16.11.2007
                                                                Time:     16:23:06

    Bolt    DIN     Grade      NUT          NUMBER      Connected assemblies
     12     934       931      NUT-M12          4          B1
                                    Total:     4 * NUT-M12
     16     934       931      NUT-M16          4
     16     934       931      NUT-M16          4          B3
                                    Total:     8 * NUT-M16
                                    Total:    12  nuts
```

Assembly_part _hole_list

```
TEKLA STRUCTURES ASSEMBLY PART HOLE LIST                        Page:     1
Model:templates_ENU                                             Project:  project number
                                                                Date:     16.11.2007
                                                                Time:     16:24:39

Assembly    Part     Number  Profile/ hole  Grade   Length(mm)  Weight(kg)
B3                   1                                          129.6
            P9       1       FLAT10*150     S235JR       290       3.4
                     4       D18            8             45
            Pr10     1       IPE300         S235JR      2990       126.3
C6                   1                                          127.8
            P3       1       FLAT10*150     S235JR       290       3.4
                     6       D16            8             10
            Pr6      1       IPE300         S235JR      2987       126.1
                                                        Total:     259.2
```

Assembly_part_list

TEKLA STRUCTURES ASSEMBLY PART LIST
Model: templates_ENU

Page: 1
Project: project number
Date: 16.11.2007
Time: 16:25:34

Assembly	Part	Number	Profile	Length(mm)	Area(m2)	Weight(kg)
B3	Pr10	1	IPE300		3.56	129.6
	P9	1	FLAT10*150	290	0.10	3.4
	Pr10	1	IPE300	2990	3.46	126.2
C6	Pr6	1	IPE300		3.56	127.8
	P3	1	FLAT10*150	290	0.10	3.4
	Pr6	1	IPE300	2987	3.46	124.3
				Total (kg):		257.4

Assembly_part_per_phase_list

TEKLA STRUCTURES ASSEMBLY PART PER PHASE LIST
Model:templates_ENU

Page: 1
Project: project number
Date: 16.11.2007
Time: 16:27:13

Assembly	Part	Number	Profile	Grade	Name	Length(mm)	Phase
C6		1	IPE300				1
	P3	1	FLAT10*150	S235JR	PLATE	290	1
	Pr6	1	IPE300	S235JR	COLUMN	2987	1
B1		1	IPE300				2
	P9	1	FLAT10*150	S235JR	PLATE	290	2
	P10	1	FLAT10*100	S235JR	PLATE	95	2
	P11	1	FLAT10*130	S235JR	PLATE	290	2
	Pr8	1	IPE300	S235JR	BEAM	5979	2

Assembly_phase_list

TEKLA STRUCTURES ASSEMBLY PHASE LIST
Model:templates_ENU

Page: 1
Project: project number:
Date: 16.11.2007
Time: 16:28:02

Phase	Assembly Number	Profile	Area(m2)	Weight(kg)
1	C6 1	IPE300	3.56	127.8
	1	assemblies in phase 1	3.56	127.8
2	B1 1	IPE300	7.13	259.6
	1	assemblies in phase 2	7.13	259.6
	2	assemblies in all phases	10.69	387.4
			Total (kg):	257.4

Assembly_phase_list2

TEKLA STRUCTURES ASSEMBLY PHASE LIST
Model:templates_ENU

Page: 1
Project: project number
Date: 16.11.2007
Time: 16:30:25

Assembly	Phase	Number	Profile	Area(m2)	Weight(kg)
B1	2	1	IPE300	7.13	259.4
C6	1	1	IPE300	3.56	127.7
	All phases	2		10.69	387.2

Assembly_stud_list

TEKLA STRUCTURES ASSEMBLY STUD LIST
Model:templates_ENU

Page: 1
Project: project number
Date: 16.11.2007
Time: 16:39:30

Assembly	Diam.	Length	Type	Number
B1				1
	12	50	STUD	8
	16	100	STUD	4

Bolt_list

TEKLA STRUCTURES BOLT LIST
Model: templates_ENU

Page: 1
Project:project number
Date: 16.11.2007
Time: 16:40:33

Name	Type	Length (mm)		Number
BOUT-M12*40	931-8.8	4	Site	
BOUT-M16*35	931-8.8	4	Site	
BOUT-M16*45	931-8.8	4	Site	
	Total:	12		

Bolt_list_2

TEKLA STRUCTURES BOLT LIST
Model:templates_ENU

Page: 1
Project: project number
Date: 16.11.2007
Time: 16:41:25

Diam	Type	Type	Length (mm)	Number
12	931/125A/934	00110	40	4
16	931/125A/934	00110	35	4
16	931/125A/934	00110	45	4
				12

combi_bnw_ fillerpl_ treadedend_list

```
--------------------------------------------------------------------------------
TEKLA STRUCTURES BOLTLIST COMPLETE                          Page:     1
Model: templates_ENU                                        Project:  project number
                                                            Date:     26.11.2007
                                                            Time:     09:26:54

================================================================================
 Bolts, Nuts, Washers
--------------------------------------------------------------------------------
             Diam  Length(mm) Type                    Number   Type
      ------------------------------------------------------------------------
      Bolt    16       55     931/125A/934               4   931
      Nut     16              934                        4
      Washer  16              125A                       4
================================================================================
 Fillerplates
--------------------------------------------------------------------------------
      Partno.       Profile        Material       Length   Number    Weight
      ------------------------------------------------------------------------
      H1            PL2*150        S235JR           290.0   2       0.7
      ------------------------------------------------------------------------
                                              Totaal:         2       1.4
================================================================================
 Threaded ends
--------------------------------------------------------------------------------
      Partno.       Profile        Material        Length  Number
      ------------------------------------------------------------------------
      Pr15          R20            S235JR           200.0   4       0.5
      ------------------------------------------------------------------------
                                                   Total:     4       1.9
```

This list shows bolts, nuts, washers, filler plates and threaded ends.

Cs_calc

```
I
I
I 01.12.2006


M B4       STEEL        1      1                  2910
               Phase 1
P STEEL     P5          1      1    FLAT10*160    290      S235JR   ENDPLATE
               Phase 1
P STEEL     Pr5         1      1    IPE300        2874     S235JR   BEAM
               Phase 1

M C1       STEEL        1      1                  3490
               Phase 1
P STEEL     Pr6         1      1    HEA240        3490     S235JR   COLUMN
               Phase 1
```

Report required for export to Calc.

Cs_calc

Report required for the link to Construsteel.

Drawing_last _revision_list

```
--------------------------------------------------------------------------------
TEKLA STRUCTURES DRAWING LAST REVISION LIST                 Page:     1
Model:templates_ENU                                         Project:  project number
                                                            Date:     16.11.2007
                                                            Time:     16:53:34

--------------------------------------------------------------------------------
Drawing       Revision     Revision date     Description
--------------------------------------------------------------------------------
A   [B.1]     2    B        17-11-2007       Coating added
--------------------------------------------------------------------------------
W   [Pr.1]    1    A        01-12-2006
--------------------------------------------------------------------------------
```

Drawing_list

```
-------------------------------------------------------------------------------
TEKLA STRUCTURES DRAWING LIST                                  Page:     1
Model:Model_12_UK1                                             Project:
                                                               Date:     01.12.2006
                                                               Time:     15:34:05

-------------------------------------------------------------------------------
Drawing       Description      Date            Modify date   Rev.   Size
-------------------------------------------------------------------------------
A   [B.2]     ASSEMBLY         23.11.2006      01.12.2006      2     420x297
G   [1]       GA               15.11.2006                      0    1189x841
G   [2]       GA               16.11.2006                      0    1189x841
G   [3]       GENERAL DRAWING  23.11.2006                      0    1189x841
W   [Pr.1]    PART             06.09.2006      01.12.2006      1     297x210
-------------------------------------------------------------------------------
```

Drawing_list_file

```
-------------------------------------------------------------------------------
TEKLA STRUCTURES DRAWING FILENAME                              Page:     1
Model: templates_ENU                                           Project:  project number
                                                               Date:     16.11.2007
                                                               Time:     16:47:54

-------------------------------------------------------------------------------
A =ASSEMBLY DRW.   W =WORKSHOP DRW.   G =GENERAL ARR. DRW.   M =MULTI DRW.
-------------------------------------------------------------------------------
Drawing     Title            Size (h*l)    Created      Plotted       Plotfile
-------------------------------------------------------------------------------
A   [B.1]   BEAM               420x297   16.11.2007               D0000025142.dg
-------------------------------------------------------------------------------
A   [B.3]   BEAM               420x297   16.11.2007               D0000025112.dg
-------------------------------------------------------------------------------
A   [C.5]   COLUMN             420x297   16.11.2007               D0000025049.dg
-------------------------------------------------------------------------------
```

Drawing_revision_list

```
-------------------------------------------------------------------------------
TEKLA STRUCTURES DRAWING REVISION LIST                         Page:     1
Model:templates_ENU                                            Project:  project number
                                                               Date:     16.11.2007
                                                               Time:     16:52:01
-------------------------------------------------------------------------------
    Drawing                 Date                    Modification date
    A   [B.1]               16.11.2007
-------------------------------------------------------------------------------
            Rev. no.  Rev. mark       Rev. date     Description
               2          B           17-11-2007    Coating added
-------------------------------------------------------------------------------
            Rev. no.  Rev. mark       Rev. date     Description
               1          A           17-11-2007    Profile changed
-------------------------------------------------------------------------------
```

Flat_bars_list

```
TEKLA STRUCTURES FLAT BARS LIST                                  Page:     1
Model:templates_ENU                                              Project:  project number
                                                                 Date:     16.11.2007
                                                                 Time:     16:57:11

Profile         Part      Grade      Number   Length (mm)    Area(m2)   Weight (kg)
FLAT10*100      P10       S235JR          1          95        0.02          0.7
                                          1          95        0.02          0.7
FLAT10*130      P11       S235JR          2         290        0.08          3.0
                                          2         580        0.17          5.9
FLAT10*150      P9        S235JR          2         290        0.10          3.4
                                          2         580        0.19          6.8
                                Total:    5                    0.38         13.5
```

This list only displays flat bars, even if other profile types are selected.

Holes_list

```
TEKLA STRUCTURES HOLES LIST                                      Page:     1
Model:templates_ENU                                              Project:  project number
                                                                 Date:     16.11.2007
                                                                 Time:     16:58:07

Diameter        Hole diameter     Number
  12              D14                   4
  16              D18                   8
                    Total:             12
```

Id_assembly_list

```
TEKLA STRUCTURES ID ASSEMBLY LIST                                Page:    1
Model: templates_ENU                                             Project: project number
                                                                 Date:    23.11.2007
                                                                 Time:    14:41:25

ID              Assembly Profile              Length  Vol   Grid            Level
Id:       1277 C2       HEA240                 3500     0.2    B/2       +3.500  +3.500
Id:       1287 C3       HEA240                 3300     0.2   C/1-2      +3.500  +3.500
                                      END
```

Pressing a ID-line in the list will highlight the profile in the model.

ld_bolt_list

```
-------------------------------------------------------------------------------------
 TEKLA STRUCTURES ID BOLTLIST                                    Page:     1
 Model:templates and reports                                     Project:  Project number
                                                                 Date:     04.01.2008
                                                                 Time:     15:12:08

-------------------------------------------------------------------------------------
 Number      Bolt                                                ID
-------------------------------------------------------------------------------------
   4         BOLT       7990          16 x 35                  Id:       6078
   4         BOLT       7990          16 x 35                  Id:       6913
   4         BOLT       7990          16 x 35                  Id:       7734
-------------------------------------------------------------------------------------
                                      END
```

Pressing a ID-line in the list will highlight the profile in the model.

ld_bolt_list_100

```
-------------------------------------------------------------------------------------
 TEKLA STRUCTURES ID BOLT_LIST_100                               Page:     1
 Model:templates_ENU                                             Project:  project number
                                                                 Date:     16.11.2007
                                                                 Time:     17:02:10

-------------------------------------------------------------------------------------
 Number      Bolt                                    ID
-------------------------------------------------------------------------------------
   4         BOLT    931        16 x 100             Id:       25184
-------------------------------------------------------------------------------------
                                      END
-------------------------------------------------------------------------------------
```

This report only shows bolts with a length of 100 mm. Mostly these are bolts which are modelled incorrect. Press the button **Fit work area by parts** in selected views if you cannot find the bolts.

ld_componentlist

```
-------------------------------------------------------------------------------------
 TEKLA STRUCTURES COMPONENTLIST                                  Page:     1
 Model:templates_ENU                                             Project:  project number
                                                                 Date:     16.11.2007
                                                                 Time:     17:02:52

-------------------------------------------------------------------------------------
                                                                           1=green
 Component                                          Part    Part    Part    2=yellow
 number              Number   Name                  MP      1st SP  2nd SP  3=red
-------------------------------------------------------------------------------------
 Id:24314      14       1    Joining plates        Pr8     Pr9             1
 Id:23811      14       1    Joining plates        Pr10    Pr8             1
 Id:24757      1011     1    U stub                Pr8                     1
-------------------------------------------------------------------------------------
                       3   components
-------------------------------------------------------------------------------------
```

Pressing a ID-line in the list will highlight the profile in the model.

ld_length_gross

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES ID LENGTH GROSS LIST                          Page:    1
Model:templates_ENU                                            Project: project number
                                                               Date:    16.11.2007
                                                               Time:    17:04:59

------------------------------------------------------------------------------------------
 ID             Part       Assembly   Profile       Grade      Length    Length gross
------------------------------------------------------------------------------------------
 Id: 637        Pr11       C8         HEA240        S235JR      2549       3490
------------------------------------------------------------------------------------------
                                      END
------------------------------------------------------------------------------------------
```

This report only shows the profiles which differ in LENGTH and LENGTH_GROSS.

ld_length_list

```
------------------------------------------------------------------------------------------
 TEKLA STRUCTURES ID LENGTH LIST                               Page:     1
 Model:templates_ENU                                           Project:  project number
                                                               Date:     23.11.2007
                                                               Time:     10:10:03

------------------------------------------------------------------------------------------
 ID                        Part                  Length
------------------------------------------------------------------------------------------
 Id:       664             Pr2                     3000
 Id:       637             Pr1                     1500
 Id:       974             P1                       275
 Id:      1010             P1                       275
 Id:      1224             P1                       275
 Id:      1249             P1                       275
------------------------------------------------------------------------------------------
                                      END
------------------------------------------------------------------------------------------
```

Pressing a ID-line in the list will highlight the profile in the model.

ld_loose_parts

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES ID LOOSE PARTS LIST                           Page:    1
Model:templates_ENU                                            Project: project number
                                                               Date:    23.11.2007
                                                               Time:    10:12:01

------------------------------------------------------------------------------------------
 ID              Part       Assembly  Profile               Grade        Length (mm)
------------------------------------------------------------------------------------------
 Id: 675         Pr3        C1        HEA240                S235JR           3500
------------------------------------------------------------------------------------------
                                      END
------------------------------------------------------------------------------------------
```

This report shows assemblies consisting of only one part. For example plates you forgot to weld. Pressing a ID-line in the list will highlight the profile in the model.

Id_loose_parts_NOT

```
-------------------------------------------------------------------------------------------
TEKLA STRUCTURES ID LOOSE PARTS NOT                              Page:    1
Model:templates_ENU                                              Project: project number
                                                                 Date:    23.11.2007
                                                                 Time     10:13:32

-------------------------------------------------------------------------------------------
 ID              Part       Assembly  Profile              Grade         Length (mm)
-------------------------------------------------------------------------------------------
 Id: 664         Pr2        B2        IPE300               S235JR             3000
 Id: 1224        P1         B2        FLAT10*70            S235JR              275
 Id: 1249        P1         B2        FLAT10*70            S235JR              275
-------------------------------------------------------------------------------------------
                                    END
-------------------------------------------------------------------------------------------
```

This report is the opposite of the report id_loose_parts. So this report does not select loose plates, filler plates and wind braces consisting of one part. Pressing a ID-line in the list will highlight the profile in the model.

Id_original_drawing

```
-------------------------------------------------------------------------------------------
TEKLA STRUCTURES ID_ORIGINAL_DRAWING_LIST                        Page:    1
Model: templates_ENU                                             Project: project number
                                                                 Date:    23.11.2007
                                                                 Time:    10:31:11

-------------------------------------------------------------------------------------------
     Original part for drawing
-------------------------------------------------------------------------------------------
     Id:1284
-------------------------------------------------------------------------------------------
                                    END
-------------------------------------------------------------------------------------------
```

To use the report you need to select a drawing in the drawing list and create the report Id_original_drawing; if a line in the report is selected, the corresponding part will be highlighted in the model.

Id_part_list

```
-------------------------------------------------------------------------------------------
TEKLA STRUCTURES ID PART LIST                                    Page:    1
Model: templates_ENU                                             Project: project number
                                                                 Date:    23.11.2007
                                                                 Time:    13:52:55

-------------------------------------------------------------------------------------------
 ID             Part     Profile              Grade       Length (mm)   Shorten
-------------------------------------------------------------------------------------------
 Id: 1224       P1       FLAT10*70            S235JR           275
 Id: 1249       P1       FLAT10*70            S235JR           275
 Id: 637        Pr1      IPE300               S235JR          1500
 Id: 664        Pr2      IPE300               S235JR          3000
-------------------------------------------------------------------------------------------
                                    END
-------------------------------------------------------------------------------------------
```

Pressing a ID-line in the list will highlight the profile in the model.

ld_saw_list

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES ID SAWLIST                                    Page:    1
Model:templates_ENU                                            Project: project number
                                                               Date:    23.11.2007
                                                               Time:    13:55:47

------------------------------------------------------------------------------------------
 ID              Profile      Part      Number    Length(mm)     Weight
------------------------------------------------------------------------------------------
 Id: 1436        IPE240       Pr4          1         1222          31.7
 Id: 637         IPE300       Pr1          1         1500          59.3
------------------------------------------------------------------------------------------
                                      END
------------------------------------------------------------------------------------------
```

Pressing a ID-line in the list will highlight the profile in the model.

ld_weld_list

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES ID WELD LIST                                  Page:    1
Model: templates_ENU                                           Project: project number
                                                               Date:    23.11.2007
                                                               Time:    13:57:28

------------------------------------------------------------------------------------------
  Weld            Type         Size        Length      Weld number  Weld number CS
------------------------------------------------------------------------------------------
 Id:  1232        W10           3.0            55            8
 Id:  1242        W10           3.0           245           10
 Id:  1257        W10           3.0            55           11
 Id:  1266        W10           3.0           245           13
------------------------------------------------------------------------------------------
                                      END
------------------------------------------------------------------------------------------
```

Pressing a ID-line in the list will highlight the weld in the model.

Locked_objects_report

```
------------------------------------------------------------------------------------------
TEKLA STRUCTURES LOCKED OBJECTS                                Page:    1
Model: templates_ENU                                           Project: project number
                                                               Date:    23.11.2007
                                                               Time:    14:21:19

------------------------------------------------------------------------------------------
Description (Name, Profile, ID)                  Owner (Domain, user)        Locked
------------------------------------------------------------------------------------------
PART IPE300 Id: 664                              PCLEON/leon                   Yes
PART IPE240 Id: 1436                             PCLEON/leon                   Yes
PART HEA240 Id: 1284                             PCLEON/leon                   Yes
PART HEA240 Id: 1274                             PCLEON/leon                   Yes
PART HEA240 Id: 675                              PCLEON/leon                   Yes
------------------------------------------------------------------------------------------
                                      END
------------------------------------------------------------------------------------------
```

Material_list.csv

For more information about this list, see document **Tekla Structures.pdf** in the folder **disc:\TeklaStructures\version\environments\europe\pdf**.

Material_list

```
-------------------------------------------------------------------------------------
TEKLA STRUCTURES MATERIAL LIST                                  Page:     1
Model:templates_ENU                                             Project:  project number
                                                                Date:     23.11.2007
                                                                Time:     14:10:09

-------------------------------------------------------------------------------------
Profile         Part      Grade     Number  Length(mm)      Area(m²)    Weight(kg)
-------------------------------------------------------------------------------------
FLAT10*70       P1        S235JR       2           275          0.04           1.5
=====================================================================================
                                                    549          0.09           3.0
-------------------------------------------------------------------------------------
HEA240          Pr3       S235JR       1          3500          4.80         211.0
=====================================================================================
                                                   3500          4.80         211.0
-------------------------------------------------------------------------------------
IPE300          Pr1       S235JR       1          1500          1.74          59.3
=====================================================================================
                                                   1500          1.74          59.3
-------------------------------------------------------------------------------------
-------------------------------------------------------------------------------------
                                                   Totaal:       6.62         273.3
-------------------------------------------------------------------------------------
```

Material_report _list

```
-------------------------------------------------------------------------------------
TEKLA STRUCTURES MATERIAL REPORT LIST                           Page:     1
Model:templates_ENU                                             Project:  project number
                                                                Date:     23.11.2007
                                                                Time:     14:11:26

-------------------------------------------------------------------------------------
Name          Grade        Assembly Number Profile        Length(mm)   Part   Phase
-------------------------------------------------------------------------------------
BEAM          S235JR       B1         1    IPE300              1500   Pr1    1
-------------------------------------------------------------------------------------
                      Total number: 1
-------------------------------------------------------------------------------------
COLUMN        S235JR       C1         1    HEA240              3500   Pr3    1
-------------------------------------------------------------------------------------
                      Total number: 1
-------------------------------------------------------------------------------------
PLATE         S235JR       B1         2    FLAT10*70            275   P1     1
-------------------------------------------------------------------------------------
                      Total number: 2
-------------------------------------------------------------------------------------
```

Matrix

```
1
Construsoft bv                templates_ENU            23.11.2007 14:12:51
project number
designer
1,12
13,7
20,14
36,10
44,4
48,3
51,3
54,7
61,12
76,20
0,0
P1          2      FLAT10*70           275   1  1        1    1  S235JR   PLATE
Pr1         1      IPE300             1500   1  1        1    1  S235JR   BEAM
Pr3         1      HEA240             3500   1  1        1    1  S235JR   COLUMN
TITLES
```

Report required for exporting to Matrix.

Matrix2

```
------------------------------------------------------------------------
  TEKLA STRUCTURES BOLTS IN PARTS LIST            Date: 23.11.2007 Page: 1
  templates_                                      Time: 14:17:52
------------------------------------------------------------------------
  Diam  Grade  Hole       Number Length Connected parts
------------------------------------------------------------------------
   16           18            4  45     P2-> P2
   12           14            4  40     P4-> P3
```

Report can be used in DSTV converter.

Multi_drawing contentlist

```
------------------------------------------------------------------------------------
MULTIDRAWING LIST                                          Page:     1
Model: templates_ENU                                       Project:  project number
                                                           Date:     23.11.2007
                                                           Time:     14:56:01

------------------------------------------------------------------------------------
Name       Drawing size   Created    Revision          Multidrawing content
------------------------------------------------------------------------------------
COLUMNS     594 *420     23.11.2007     0     POS.      NAME         PROFILE
------------------------------------------------------------------------------------
                                          A C1        COLUMN       HEA240
                                          A C2        COLUMN       HEA240
                                          A C3        COLUMN       HEA240
------------------------------------------------------------------------------------
```

Multi_drawing _list

```
------------------------------------------------------------------------------------
TEKLA STRUCTURES MULTI DRAWING LIST                        Page:     1
Model:templates_ENU                                        Project:  project number
                                                           Date:     23.11.2007
                                                           Time:     14:52:23

------------------------------------------------------------------------------------
Type Drawing              Create date        Drawing name
------------------------------------------------------------------------------------
 M   [1]                  23.11.2007         COLUMNS
 M   [2]                  23.11.2007         PLATES
------------------------------------------------------------------------------------
```

Notification report

```
- - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
PROJECT NEWS!!!!

This is an automatic newsflash for project participants

This message is shown when anyone opens the model or runs "notification_report" report template

Customize the content by editing notification_report.rpt

Deactivate be setting XS_SHOW_NOTIFICATION_REPORT to FALSE.

Tips how to utilize this to improve project communication and workflow

can be found from the Release Notes and What's New material
- - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
------------------------------------------------------------------------
NOTIFICATION OF ASSIGNMENTS CONTRACT NO:   2007-12345   Page: 1
Objects assigned to Beren                               Date: 13.11.2007
------------------------------------------------------------------------
Object         Name        Assigned to    Assigned by       Comment
------------------------------------------------------------------------
Assembly      KOLOM          Berend         Berend                          Assembly Id:9997
Part          KOLOM          Berend         Berend                          Part Id:9992
------------------------------------------------------------------------
. assemblies, 1 parts and 0 drawings have been assigned to Berend
------------------------------------------------------------------------
```

Nut_list

```
------------------------------------------------------------------------
  TEKLA STRUCTURES NUT LIST FOR MODEL: templates_ENU              Page:1

                                                    Date: 23.11.2007
                                                    Time: 14:45:19
------------------------------------------------------------------------
  Diam  Name                    Type                    Finish    Number
------------------------------------------------------------------------
    12  NUT-M12                  934                                    4
------------------------------------------------------------------------
                                       Total nuts: 12    nuts:          4
------------------------------------------------------------------------
    16  NUT-M16                  934                                    4
------------------------------------------------------------------------
                                       Total nuts: 16    nuts:          4
------------------------------------------------------------------------
------------------------------------------------------------------------
                                                  Total nuts:         8
------------------------------------------------------------------------
```

Part_assembly_list

```
------------------------------------------------------------------------------------
TEKLA STRUCTURES PART ASSEMBLY LIST                          Page:     1
Model:templates_ENU                                          Project:  project number
                                                             Date:     23.11.2007
                                                             Time:     15:22:32

------------------------------------------------------------------------------------
  Part          Assembly     Number      Profile              Length (mm)
------------------------------------------------------------------------------------
  P2            B4               1       FLAT10*150               230
  P2            B5               1       FLAT10*150               230
  P3            B4               1       FLAT10*130               230
  P4            B6               1       FLAT10*110               130
  Pr5           B4               1       IPE240                   590
  Pr6           B5               1       IPE240                   677
------------------------------------------------------------------------------------
                      Total     6  parts
------------------------------------------------------------------------------------
```

Part_hole_list

```
------------------------------------------------------------------------------------
TEKLA STRUCTURES PART HOLE LIST                              Page:     1
Model:templates_ENU                                          Project:  project number
                                                             Date:     23.11.2007
                                                             Time:     15:23:54

------------------------------------------------------------------------------------
                       Number        Hole     Number
 Part      Profile     of parts      diam.    of holes    Bolt diam. and grade
------------------------------------------------------------------------------------
 P2        FLAT10*150    2
                                       18     4          16     931
------------------------------------------------------------------------------------
 P3        FLAT10*130    1
                                       14     4          12     931
------------------------------------------------------------------------------------
 P4        FLAT10*110    1
                                       14     4          12     931
------------------------------------------------------------------------------------
```

Number of holes and diameters per part number.

Part_list

```
TEKLA STRUCTURES PART LIST                                          Page:     1
Model:templates_ENU                                                 Project:  project number
                                                                    Date:     23.11.2007
                                                                    Time:     15:25:24

Part     Profile     Number  Grade     Length(mm) Weight/part   Comment
P2       FLAT10*150       2  S235JR          230        2.7
P3       FLAT10*130       1  S235JR          230        2.3
P4       FLAT10*110       1  S235JR          130        1.1
Pr5      IPE240           1  S235JR          591       18.1
              Total     5 parts                      27.0  kg
```

Part_list2

```
TEKLA STRUCTURES PART LIST                                          Page:     1
Model:templates_ENU                                                 Project:  project number
                                                                    Date:     23.11.2007
                                                                    Time:     15:26:45

Part Number Profile     Weight  Grade     Length   Area  Weight  Comment
                        (kg/m)             (mm)    (m2)   (kg)
P3     1    FLAT10*130    0.00  S235JR       230   0.07     2.3
                    Total for    1 part(s)   230   0.07     2.3
P4     1    FLAT10*110    0.00  S235JR       130   0.03     1.1
                    Total for    1 part(s)   130   0.03     1.1
Pr5    1    IPE240       30.70  S235JR       591   0.54    18.1
                    Total for    1 part(s)   591   0.54    18.1
Pr7    1    IPE120       10.40  S235JR       503   0.24     5.2
                    Total for    1 part(s)   503   0.24     5.2
                                                   0.88    26.8
```

Part_list_prelim_mark

```
TEKLA STRUCTURES PART LIST PRELIMARY MARK                           Page:     1
Model:templates_ENU                                                 Project:  project number
                                                                    Date:     23.11.2007
                                                                    Time:     15:27:45

Part     Profile      Number     Grade     Length(mm) Weight   Prelimary mark
P3       FLAT10*130        1     S235JR          230      2.3
P4       FLAT10*110        1     S235JR          130      1.1
Pr5      IPE240            1     S235JR          591     18.1
Pr7      IPE120            1     S235JR          503      5.2
              Total        4 parts                      26.8 Kg
```

Part_phase_list

```
TEKLA STRUCTURES PART PHASE LIST                                    Page:     1
Model:templates_ENU                                                 Project:  project number
                                                                    Date:     23.11.2007
                                                                    Time:     15:29:23

-------------------------------------------------------------------------------

  Phase   Part      Number     Profile             Area (m2) Weight (kg)
-------------------------------------------------------------------------------
      1   P2            2      FLAT10*150             0.08         2.7
      1   P3            1      FLAT10*130             0.07         2.3
      1   Pr5           1      IPE240                 0.54        18.1
      1   Pr6           1      IPE240                 0.63        20.8
-------------------------------------------------------------------------------
                        5 parts in phase  1           1.39        46.7
-------------------------------------------------------------------------------
      2   P1            2      FLAT10*70              0.04         1.5
      2   Pr2           1      IPE300                 3.48       126.6
-------------------------------------------------------------------------------
                        3 parts in phase  2           3.57       129.6
-------------------------------------------------------------------------------
                        8 parts in all phases         4.96       176.3
-------------------------------------------------------------------------------
```

Part_quantity_list

```
-------------------------------------------------------------------------------
 TEKLA STRUCTURES PART QUANTITY LIST FOR DSTVCONV            Page:     1
 Model:templates_ENU                                         Project:  project number
                                                             Date:     23.11.2007
                                                             Time:     15:32:53

-------------------------------------------------------------------------------
   Part       Number      Profile          Length     Grade
-------------------------------------------------------------------------------
   P1              2      FLAT10*70           275     S235JR
   Pr2             1      IPE300             3000     S235JR
-------------------------------------------------------------------------------
                   3
-------------------------------------------------------------------------------
```

Phase_manager

```
TEKLA STRUCTURES PHASEMANAGER                                       Page:     1
Model:templates_ENU                                                 Project:  project number
                                                                    Date:     23.11.2007
                                                                    Time:     15:34:16

-------------------------------------------------------------------------------
 Phase       Phase name           Phase comment 1       Phase comment 2
-------------------------------------------------------------------------------
      1      Phase 1
      2      Phase 2
-------------------------------------------------------------------------------
```

Pms1 Report which can be read into PMS (no example available)

Preliminary_mark_list

```
TEKLA STRUCTURES PRELIMARY MARK LIST                          Page:     1
Model:templates_ENU                                           Project:  project number
                                                              Date:     23.11.2007
                                                              Time:     15:35:47

Number   Prelim mark   Part    Profile         Grade      Length(mm)  Weight(kg)
  2                     P1     FLAT10*70       S235JR          275         1.5
                                                      Totals:   549         3.0
  1                    Pr2     IPE300          S235JR         3000       126.6
                                                      Totals:  3000       126.6
                                                         Weight total:   129.6 kg
```

Profiles_list

```
TEKLA STRUCTURES PROFILES LIST                                Page:     1
Model:templates_ENU                                           Project:  project number
                                                              Date:     23.11.2007
                                                              Time:     15:37:20

Profile          Part  Assembly  Number Length  Area    Weight  Matetial    Saw H/A
                                        (mm)     (m2)    (kg)
HEA240           Pr3     C2        1    3500    4.80   211.0    S235JR     0  0  1
                                   1    3500    4.80   211.0
IPE120           Pr7     B6        1     503    0.24     5.2    S235JR     0  0  1
                                   1     503    0.24     5.2
IPE240           Pr5     B4        1     591    0.54    18.1    S235JR     0  0  1
IPE240           Pr6     B5        1     678    0.63    20.8    S235JR     0  0  1
                                   2    1268    1.17    38.9
                                   4    part    6.20   255.2    kg
```

Displays all profiles, except plates.

Below “saw” are 2 columns showing the value “0”or “1”. (0=square, 1=bevelled).

M/S means if the part is a main part or secondary part. “1”=main part, “0”=secondary part.

Saw_list

```
------------------------------------------------------------------------------------
TEKLA STRUCTURES SAW LIST                                    Page:     1
Model:templates_ENU                                          Project:  project number
                                                             Date:     23.11.2007
                                                             Time:     15:38:42

------------------------------------------------------------------------------------
Profile          Part     Grade      Number    Length(mm)    Area(m2)  Weight
------------------------------------------------------------------------------------
IPE240           Pr4      S235JR          1          1222        1.13        31.7
IPE300           Pr1      S235JR          1          1500        1.74        59.3
------------------------------------------------------------------------------------
                                                                Total:         0.0
------------------------------------------------------------------------------------
```

This list only shows profiles which match the following requirements:

- profiles may not be plates

- at least one profile-end must be skew

Saw_list_part_assembly

```
------------------------------------------------------------------------------------
TEKLA STRUCTURES SAW LIST PART ASSEMBLY                      Page:     1
Model:templates_ENU                                          Project:  project number
                                                             Date:     23.11.2007
                                                             Time:     15:39:57

------------------------------------------------------------------------------------
Part   Profile    Number  Assembly Length     Saw angles     Saw angles    Weight
                                   (mm)      Start          End            (kg)
                                             Z    Y         Z     Y
------------------------------------------------------------------------------------
Pr4    IPE240          1    B3         1222    45.8  -0.0    -29.1   0.0        31.7
------------------------------------------------------------------------------------
                       1               1222                                    31.7
====================================================================================
Pr1    IPE300          1    B1         1500     0.0   0.0    -32.4   0.0        59.3
------------------------------------------------------------------------------------
                       1               1500                                    59.3
====================================================================================
------------------------------------------------------------------------------------
                                                                Total:      91.0
------------------------------------------------------------------------------------
```

Saw_list_W_A

```
------------------------------------------------------------------------------------
TEKLA STRUCTURES SAW LIST WITH ANGLES                        Page:    1
Model:templates_ENU                                          Project: project number
                                                             Date:    23.11.2007
                                                             Time:    15:40:55

------------------------------------------------------------------------------------
Profile      Length Number  Part   Weight   angle1-Z  angle1-Y  angle2-Z  angle2-Y
------------------------------------------------------------------------------------
IPE240         1222      1  Pr4      31.7      45.8      -0.0     -29.1       0.0
------------------------------------------------------------------------------------
               Total     1           31.7  kg
====================================================================================
IPE300         1500      1  Pr1      59.3       0.0       0.0     -32.4       0.0
------------------------------------------------------------------------------------
               Total     1           59.3  kg
====================================================================================
```

For each profile are the saw cut angles displayed (Y1, Y2, Z1, Z2)

Shorten_list

```
------------------------------------------------------------------------------------
TEKLA STRUCTURES SHORTEN LIST                                  Page:     1
Model:templates_ENU                                            Project:  project number
                                                               Date:     23.11.2007
                                                               Time:     15:42:15

------------------------------------------------------------------------------------
Profile         Part     Grade          Number  Length(mm)    Area(m2)  Weight (kg)
------------------------------------------------------------------------------------
IPE120          Pr7      S235JR              1         503        0.24         5.2
====================================================================================
                                                     503        0.24         5.2
------------------------------------------------------------------------------------
IPE240          Pr5      S235JR              1         591        0.54        18.1
IPE240          Pr6      S235JR              1         678        0.63        20.8
====================================================================================
                                                    1268        1.17        38.9
------------------------------------------------------------------------------------
IPE300          Pr2      S235JR              1        3000        3.48       126.6
====================================================================================
                                                    3000        3.48       126.6
------------------------------------------------------------------------------------
------------------------------------------------------------------------------------
                                                                Total:      170.8
------------------------------------------------------------------------------------
```

This report only shows profiles which meet the following conditions:

- profiles may not be plates
- both profile ends must be square

This report displays the opposite of the saw list.

Status_list

```
------------------------------------------------------------------------------
 Tekla Structures STATUSLIST for model:templates_ENU              Page: 1

                                                    Date:23.11.2007
                                                    Time:15:44:39
------------------------------------------------------------------------------
  Assembly   Number     Profile        Date           Status
------------------------------------------------------------------------------
  B1             1      IPE300
  B2             1      IPE300
------------------------------------------------------------------------------
     Total       2 assemblies
------------------------------------------------------------------------------
```

Steelproject Report which can be read into Steelproject (no example available)

Step_list

```
-------------------------------------------------------------------------------------
TEKLA STRUCTURES STEP LIST                                      Page:     1
Model:templates_ENU                                             Project:  project number
                                                                Date:     23.11.2007
                                                                Time:     15:46:17

-------------------------------------------------------------------------------------
Name                      Assembly  Number   Step: height  width  length    Phase
-------------------------------------------------------------------------------------
DEJO GRIP 180               STP1       9             43     180    900          1
=====================================================================================
                       Step total:      9
-------------------------------------------------------------------------------------
```

This report only shows profiles with material “STEP”.

The default material setting in the component is “STEP”.

In the component, pick a step profile.

Stud_list

```
-------------------------------------------------------------------------------------
TEKLA STRUCTURES STUD LIST                                      Page:     1
Model:templates_ENU                                             Project:  project number
                                                                Date:     23.11.2007
                                                                Time:     15:48:27

-------------------------------------------------------------------------------------
Diam.  Name                     Type          Length    Finish      Number
-------------------------------------------------------------------------------------
 16    M16*50                   STUD            50                      4
 20    M20*60                   STUD            60                      4
-------------------------------------------------------------------------------------
                                                   Studs total:         8
-------------------------------------------------------------------------------------
```

Super_assembly _list

```
TEKLA STRUCTURES SUPER ASSEMBLY LIST                            Page:     1
Model: templates_ENU                                            Project:  project number
                                                                Date:     23.11.2007
                                                                Time:     16:01:10

-------------------------------------------------------------------------------------
   Main assembly
        Sub assembly     Number              Level assembly  Weight (kg)
-------------------------------------------------------------------------------------
=====================================================================================
    B9                       1                    0              124.3
     ---------------------------------------------------------------------------------
         B10                 1                    1
```

Super_assembly_part_list

```
--------------------------------------------------------------------------------
TEKLA STRUCTURES SUPER ASSEMBLY PART LIST                    Page:     1
Model: templates_ENU                                         Project:  project number
                                                             Date:     23.11.2007
                                                             Time:     16:05:58

--------------------------------------------------------------------------------
Assembly     Number  Name         Profile                        Area (m2) Weight (kg)
================================================================================
B9             1     BEAM         IPE300                         3.4       124.3
               ----------------------------------------------------------------
               Part        Number   Profile          Length (mm)
               ----------------------------------------------------------------
               P13            2     FLAT10*150          290        0.10       3.4
               P14            1     FLAT10*150          290        0.10       3.4
               Pr12           1     IPE300             1480        1.71      62.4
    ============================================================================
     1x Submerk B10
               ----------------------------------------------------------------
               Part        Number   Profile          Length (mm)
               ----------------------------------------------------------------
               P13            1     FLAT10*150          290        0.10       3.4
               P14            1     FLAT10*150          290        0.10       3.4
               Pr13           1     IPE300             1033        1.20      43.6
================================================================================
               Total for      1     assembl(y)(ies):               3.39      124.3
================================================================================
```

Uda_list

```
--------------------------------------------------------------------------------
TEKLA STRUCTURES UDA LIST                                    Page:     1
Model:templates_ENU                                          Project:  project number
                                                             Date:     23.11.2007
                                                             Time:     16:08:14

--------------------------------------------------------------------------------
Part     Profile       Number Comment              Shorten  Cambering  Prel. mark
--------------------------------------------------------------------------------
P5       FLAT10*115       2
Pr3      HEA240           1
Pr14     L60*6            1                        4.0
--------------------------------------------------------------------------------
                  Total     4  parts
--------------------------------------------------------------------------------
```

This report displays information, filled in the “User Defined Attributes” (UDA) in the profile properties dialogs.

Washer_list

```
--------------------------------------------------------------------------------
  TEKLA STRUCTURES WASHER LIST FOR MODEL: templates_ENU            Page: 1

                                                        Date:  23.11.2007
                                                        Time:  16:09:16
--------------------------------------------------------------------------------
  Diam      Name                          Type       Number
--------------------------------------------------------------------------------
    12      WASHER-M12                    125A           40
    16      WASHER-M16                    125A            8
--------------------------------------------------------------------------------
                                             Total      48  washers
--------------------------------------------------------------------------------
```

xyz-coordinates-TotalStation

TEKLA STRUCTURES Totalstationlist
Model: templates_ENU

Project: project number
Date: 23.11.2007
Time: 16:13:38

Number	Start X	Start Y	Start Z	End X	End Y	End Z
,	3000	4363	-239	3000	4363	0
,	3000	4954	-239	3000	4954	0
,	3000	4975	-119	3000	4975	0
,	3000	4964	0	3000	4353	0
,	3000	4353	0	3000	3665	0
,	3000	4964	0	3000	5478	0

Total parts: 6

END

Report required for Total Station (as a tool on site for positioning columns).

Steel related templates

Anchorlist

CASTPLATES + ANCHORS				
A1	ANCHOR_M16	4	S235JR	588
P17	PL3*500	1	S235JR	500
Part	Profile	Number	Material	Length

This template only displays parts with the name ANCHOR and CASTPLATE.

Assembly_bolt_ list

BOLT LIST FOR ASSEMBLY B4					NUMBER: 1
Diameter	Grade	Type	Length	Number	Member(s)
16	8.8	Site	45	4	B5
BOLT LIST FOR ASSEMBLY B6					NUMBER: 1
Diameter	Grade	Type	Length	Number	Member(s)
12	8.8	Site	40	4	B4

Assembly_finish_ list

ASSEMBLY LIST						
Assembly	Phase	Profile	Number	Area (m2)	Weight (kg)	Finish
C2	1	HEA240	1	4.9	214.6	Painted
Total 1 assembl(y)(ies)				4.9	214.6	

Assembly_in_ assembly

Assembly	Number	In assembly
B10	1	B9

Assembly_list_1

ASSEMBLY LIST				
Assembly	Profile	Number	Area (m2)	Weight (kg)
B10	IPE300	1	1.4	50.4
Total 1 assembl(y)(ies)			1.4	50.4

Assembly_list_no_dummy

ASSEMBLY LIST WITHOUT DUMMY

Assembly	Profile	Number	Area (m2)	Weight (kg)
B10	IPE300	1	1.4	50.4
Total 1 assembl(y)(ies)			1.4	50.4

Assembly_lot_number

ASSEMBLY	LOT	NUMBER
B10	3	1

Assembly_part_list_super_assembly

MATERIAL LIST FOR SUPER ASSEMBLY: B9 Number:1

Assembly	Profile	Grade	Number	Length (mm)	Weight (kg)
P13	FLAT10*150	S235JR	2	290	6.8
P14	FLAT10*150	S235JR	1	290	3.4
Pr12	IPE300	S235JR	1	1480	62.4
B10	Subassembly		1		50.4
				Total	123.1

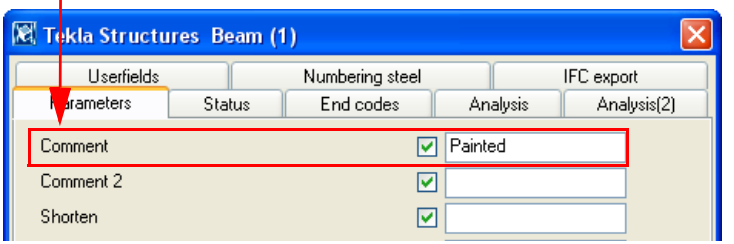
Assembly_partlist_comment

ASSEMBLY PART LIST

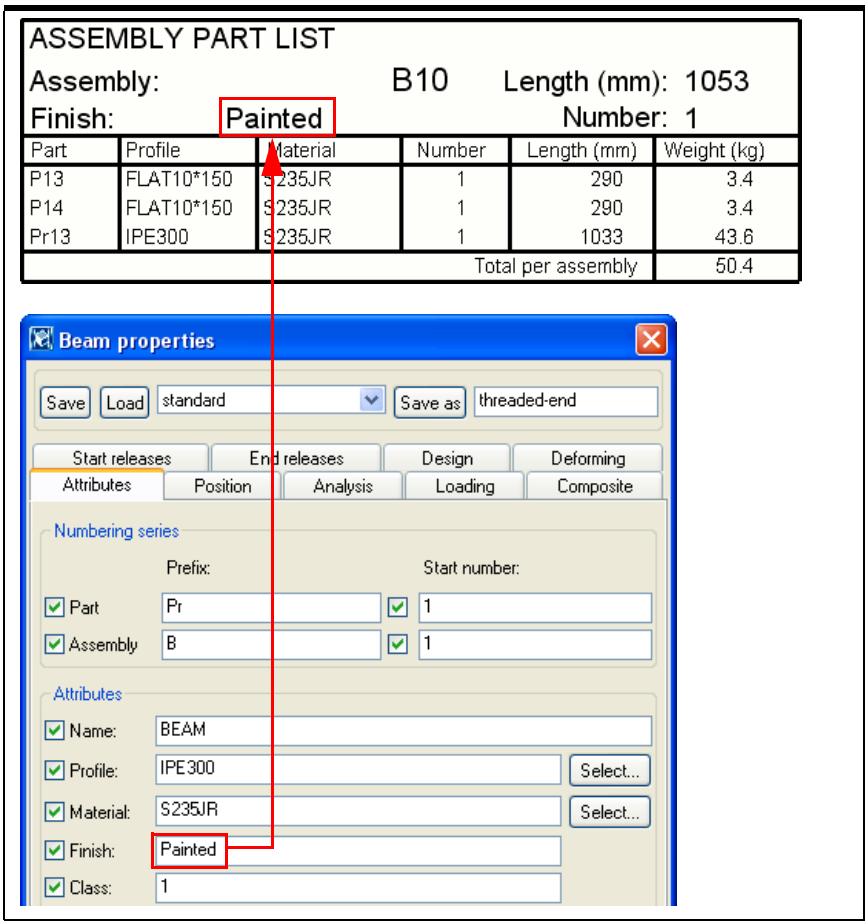
Assembly:0 Length (mm):1500

Finish: Number:1

Comment	Part	Profile	Material	Number	Length (mm)	Weight (kg)
	P13	FLAT10*150	S235JR	2	290	3.4
	P14	FLAT10*150	S235JR	1	290	3.4
Painted	Pr12	IPE300	S235JR	1	1480	62.4
					Total per assembly	69.3



Assembly_ partlist_finish



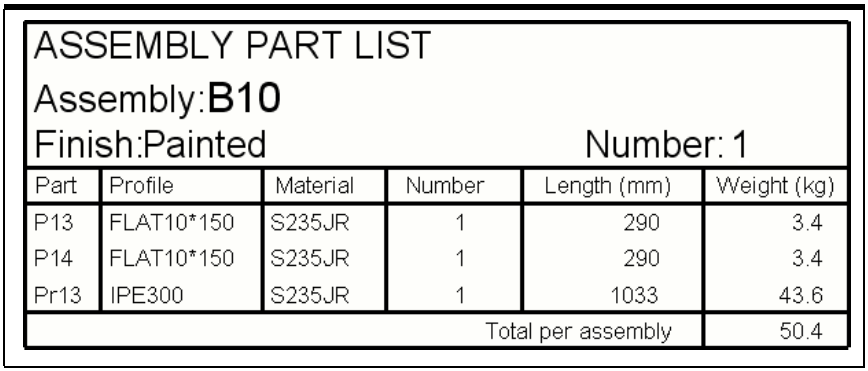
ASSEMBLY PART LIST

Assembly: B10 Length (mm): 1053

Finish: Painted Number: 1

Part	Profile	Material	Number	Length (mm)	Weight (kg)
P13	FLAT10*150	S235JR	1	290	3.4
P14	FLAT10*150	S235JR	1	290	3.4
Pr13	IPE300	S235JR	1	1033	43.6
				Total per assembly	50.4

Assembly_ partlist_finish_ chamber



ASSEMBLY PART LIST

Assembly:**B10**

Finish:Painted Number: 1

Part	Profile	Material	Number	Length (mm)	Weight (kg)
P13	FLAT10*150	S235JR	1	290	3.4
P14	FLAT10*150	S235JR	1	290	3.4
Pr13	IPE300	S235JR	1	1033	43.6
				Total per assembly	50.4

Assembly_ partlist_finish_ no_dummy

ASSEMBLY PART LIST

Assembly: **B10** Length (mm): 1053

Finish: Painted Number: 1

Part	Profile	Material	Number	Length (mm)	Weight (kg)
P13	FLAT10*150	S235JR	1	290	3.4
P14	FLAT10*150	S235JR	1	290	3.4
Pr13	IPE300	S235JR	1	1033	43.6
				Total per assembly	50.4

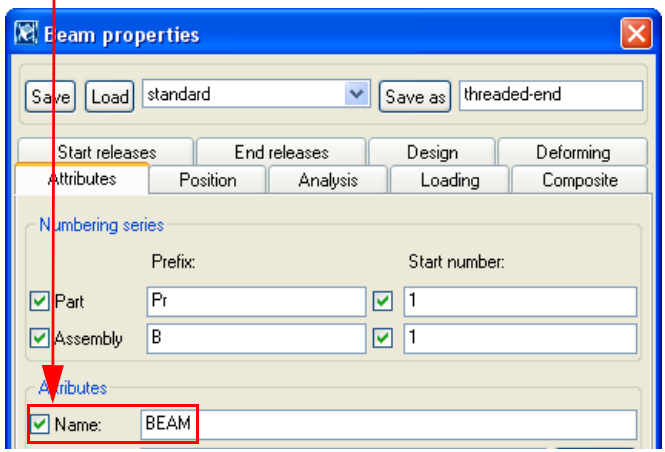
Assembly_ partlist_name

ASSEMBLY PART LIST

Assembly: **B10** Length (mm): 1053

Finish: Painted Number: 1

Name	Part	Profile	Material	Number	Length (mm)	Weight (kg)
PLATE	P13	FLAT10*150	S235JR	1	290	3.4
PLATE	P14	FLAT10*150	S235JR	1	290	3.4
BEAM	Pr13	IPE300	S235JR	1	1033	43.6
					Total per assembly	50.4



Assembly_phase _number

ASSEMBLY	PHASE	NUMBER
B10	1	1
B10	3	1
B10	4	1

Assembly_super_assembly

Assembly, Super assembly

Assembly: B14			1	assemblies			Level in assembly (0 -> 9): 0		
	Part	P13		Profile	FLAT10*150	Number 2	Weight	6.8	
	Part	P14		Profile	FLAT10*150	Number 1	Weight	3.4	
	Part	Pr12		Profile	IPE300	Number 1	Weight	62.4	
	Sub assembly	B10			Number 1		Weight	50.4	1
		Part	P13	Profile	FLAT10*150	Number 1	Weight	3.4	
		Part	P14	Profile	FLAT10*150	Number 1	Weight	3.4	
		Part	Pr13	Profile	IPE300	Number 1	Weight	43.6	
Total weight assembly:					123.1				

Castplate_toplevel

Grid	Profile	Top side castplate
D/1	PL3*500	-0.047

Company_name

Telephone +31 (0)316 340192 Fax +31 (0)316 342454

constrUsoft

CONSTRUCTION INDUSTRIES SOFTWARE SUPPLIER

E-mail info@construsoft.com Internet: www.construsoft.com

Drawing_part_assembly

Drawing part assembly

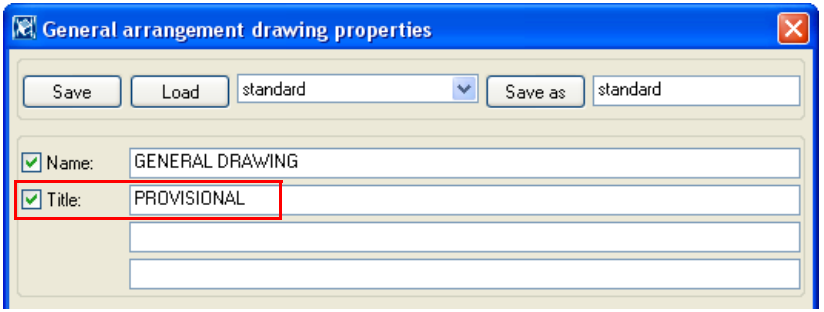
Drawing materiallist: A [B.10]

Profile	Material	Length	Part	Number	Assembly
FLAT10*150	S235JR	290	P13	6	B9
FLAT10*150	S235JR	290	P13	3	B10
			Total	9	pieces
FLAT10*150	S235JR	290	P14	3	B9
FLAT10*150	S235JR	290	P14	3	B10
			Total	6	pieces
IPE300	S235JR	1033	Pr13	3	B10
			Total	3	pieces

Final provisional

PROVISIONAL

Enter here in the drawing:



Grid

ASSEMBLY	GRID
B10	C/1-2

Grid_main_ assembly

ASSEMBLY	GRID
B15	B-D/2

Lot_assembly_ number

LOT	ASSEMBLY	NUMBER
3	B10	1

Part_assembly

Part	Profile	Length	Number	Assembly
P13	FLAT10*150	290	3	B10
P13	FLAT10*150	290	6	B15
P14	FLAT10*150	290	2	B10
P14	FLAT10*150	290	2	B15
Pr12	IPE300	1480	1	B15
Pr13	IPE300	1033	1	B10
		Total number:	15	

Part_phase

PART	PHASE	NUMBER
P13	4	4
P13	5	2

Partlist

P13	FLAT10*150	6	S235JR	290
Part	Profile	Number	Material	Length

Phase_assembly _number

PHASE	ASSEMBLY	NUMBER
4	B10	1
6	B10	1

Phase_number_ main_assembly

PHASE	NUMBER
4	1
6	1

Phasemanager

PHASEMANAGER			
Phase	Phase name	Comment 1	Comment 2
1	Phase 1	Hall	
2	Phase 2	Awning	
3	Phase 3	Front covering	

Plotdate

Tekla Structures model templates_ENU plotted on 26.11.2007

Revision

2	B	Galvanized	26-11-2007
1	A	Profile type changed	26-11-2007
Rev. no.	Revision	Revision description	Revision date

Revision last

2	B	Galvanized	26-11-2007
Rev. no.	Revision	Revision description	Revision date

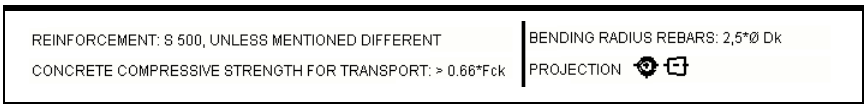
Tekla_Structures

TEKLA®
Structures
www.construsoft.com

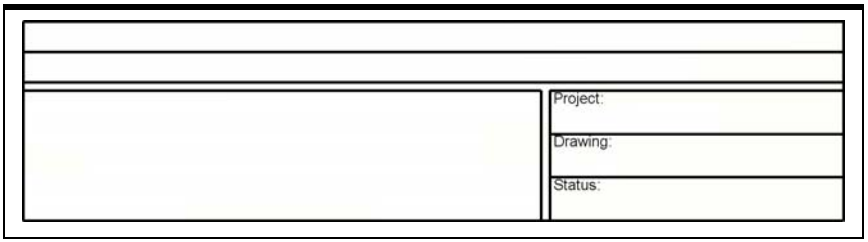
Template_A1

MATERIALGRADE: S235JR UNLESS MENTIONED DIFFERENT
BOLTGRADE: 8.8 UNLESS MENTIONED DIFFERENT
WELDINGS 4 ,UNLESS MENTIONED DIFFERENT
PROJECTION

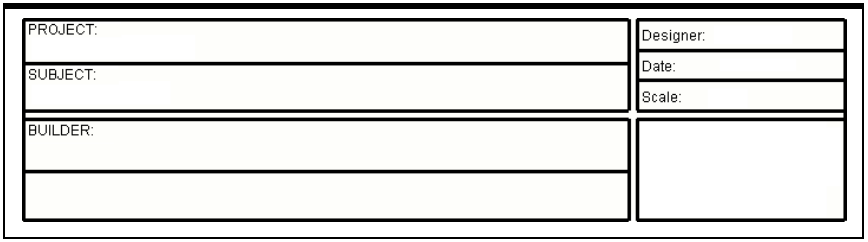
Template_A2



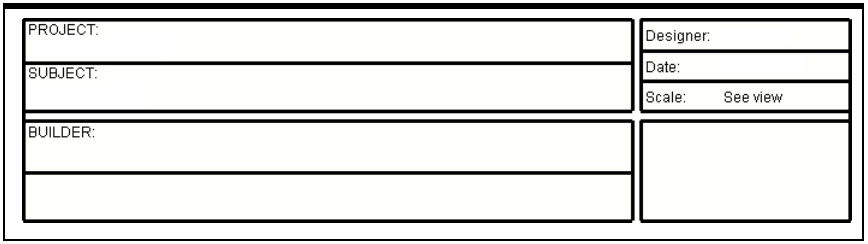
Template_A



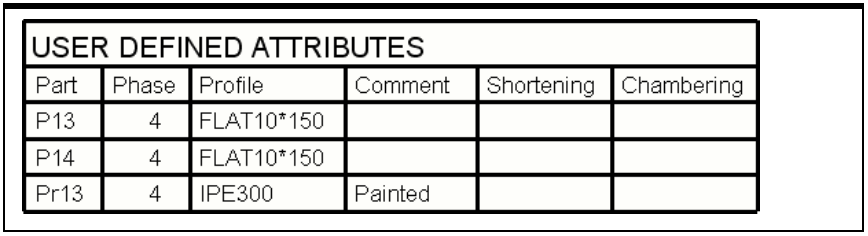
Template_B



Template_B2



UDA



USER DEFINED ATTRIBUTES

Part	Phase	Profile	Comment	Shortening	Chambering
P13	4	FLAT10*150			
P14	4	FLAT10*150			
Pr13	4	IPE300	Painted		

2.32 Notes:

3 Tekla Structures 13.1 Standard documents

Subjects

This document contains the following subjects:

- **General** (p.3)
- **AutoConnection and AutoDefaults** (p.4)
- **Ini file variables** (p.16)
- **Using characters in Tekla Structures** (p.61)
- **Classifier** (p.62)
- **Component database** (p.91)
- **Control numbers** (p.100)
- **Converting plates to flat bars** (p.103)
- **Create adjusted symbols** (p.105)
- **Creating toolbars, own pull-down menu's and shortkeys** (p.111)
- **Dimensioning Reference side** (p.128)
- **DSTV2DXF** (p.134)
- **DWG/DXF** (p.142)
- **Filter by user attributes** (p.166)
- **Hatching** (p.169)
- **ID reports** (p.178)
- **Key plan** (p.184)
- **Alternative numbering for Multi drawings** (p.194)
- **Tekla Stuctures models** (p.197)
- **Modify user defined attributes in the file objects.inp** (p.216)
- **NC files and pop marks** (p.231)
- **Super- and sub assemblies** (p.254)
- **Numbering UDA** (p.276)
- **Opening reports in Microsoft Excel and Word** (p.294)
- **Extend the contextmenu "Copy to"** (p.304)
- **Options dialog box** (p.307)
- **Ini files** (p.311)
- **Personal and other user settings** (p.323)

- **Parametric modeling** (p.327)
- **Preliminary marks** (p.332)
- **Profile catalog** (p.334)
- **Profile length calculation** (p.356)
- **Save and load defaults** (p.358)
- **Save Select switches and Snap settings** (p.364)
- **Sketching parametric profiles** (p.369)
- **Tekla Structures Import and Export** (p.392)
- **ArchiCAD 9 - TS 11.2 im- and export** (p.396)
- **ArchiCAD 11 - TS 12.0 im- and export** (p.402)
- **Autodesk Inventor im- and export** (p.406)
- **Create a 3D PDF from a Tekla Structures Model** (p.410)
- **IFC im- and export** (p.413)
- **Microstation im- and export** (p.418)
- **Pro/Engineer import** (p.423)
- **SolidWorks im- and export** (p.426)
- **Tekla Structures STEP IGES Converter** (p.432)
- **Stairs and railings** (p.437)
- **Standard parts** (p.473)
- **Surface treatment** (p.475)
- **Tips Windows Explorer** (p.497)
- **Unfolding profiles and plates** (p.504)
- **Web models** (p.512)
- **Weight, area and volume calculation** (p.524)
- **Xstwerkv** (p.536)

3.1 General

Besides Tekla Structures you can use the following programs:

DSTV2DXF	Program to convert NC files to *.dxf-files to drive plasmacutters and other cutting devices, see **DSTV2DXF** (p.134).
DWG Viewer	Program to view DWG and DXF files, the shortcut to *disc:\TeklaStructures\version\nt\bin\dwgviewer.exe* is located on the Desktop.
DSTV Convertor	Conversion program for NC files, see Xtranet.
Xstwerkv	Program to export to SCIA Cimsteel, Concreet and Matrix, see **Xstwerkv** (p.536).

3.2 AutoConnection and AutoDefaults

Topics

This chapter is divided into following sections:

AutoDefaults (p.6)

AutoConnection: manually (p.10)

AutoConnection: automatically (p.11)

Merging defaults (p.15)

AutoConnection and AutoDefaults will create the right connection or detail in a specific situation.

What is a ***connection*** and what is a ***detail***?

Detail 1 profile and one point

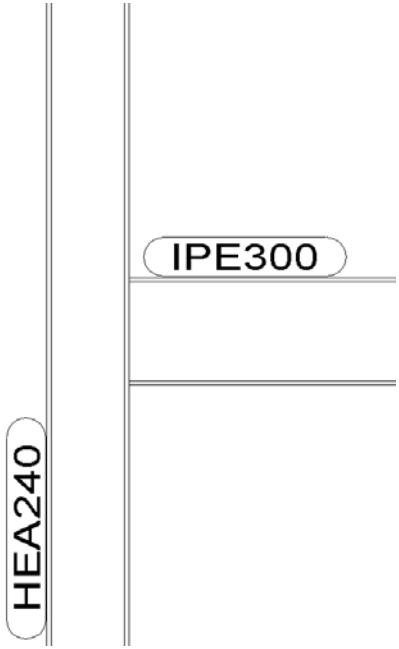
Connection 2 of more profiles

This means Tekla Structures recognizes which profile(s) are used and therefore Tekla Structures utilize a certain component setting. Because of this, the user does not have to select the component and the component setting.

However, two conditions have to be fulfilled:

- the AutoConnection must be defined
- the AutoDefaults must be defined

In the example below there is a connection between a column and a beam. If the conditions are fulfilled, Tekla Structures creates the correct connection. If one of the profile changes, the connection will be modified automatically according to the AutoDefaults settings.

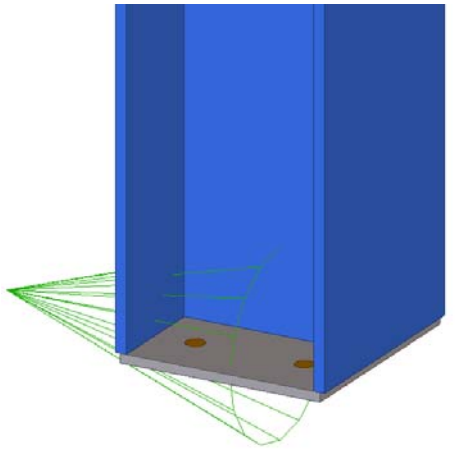

IPE300
HEA240

AutoDefaults

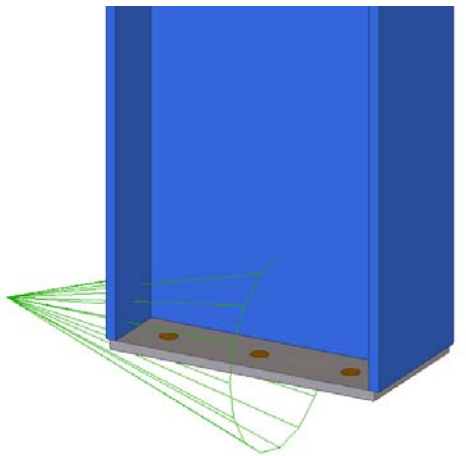
AutoDefaults determines which component setting belongs to which profile.

This means that per profile type a setting can be defined, but it can also mean that a component setting be linked to a certain size range. For example; all rectangular hollow sections with height 80mm, or all rectangular hollow sections with size 80 - 100mm will use the same component setting.

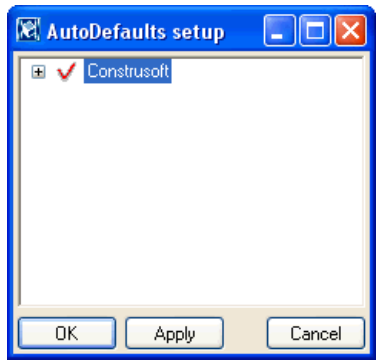
How is it done? Open a model. Position a column HEA240. Open component 1004 (Base plate) and load setting AD. Now create the component in the model. A neat base plate will be created.



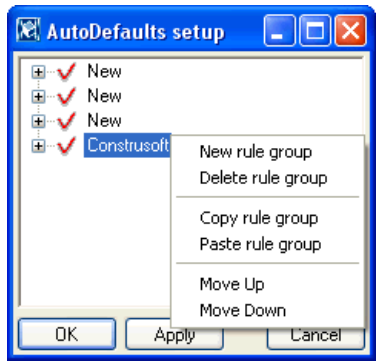
Now change the column into a IPE300. Both the base plate as well as the bolt pattern will change.



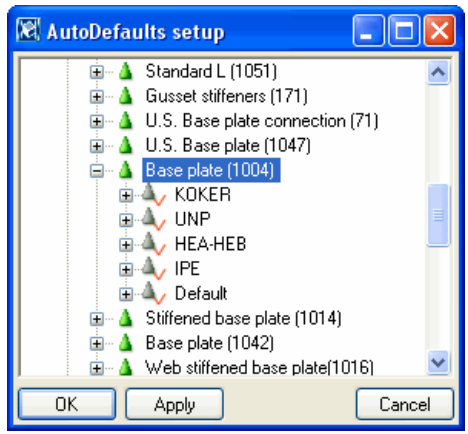
How to set this? Go to **Set up** > **AutoDefaults...**



The AutoDefaults group called **Construsoft** will be shown. Multiple groups can be defined. The number of groups is unlimited. Usually, 5 groups will be sufficient. Creating, copying, moving and deleting groups can be done with the right mouse button. Renaming a group can be done with F2.

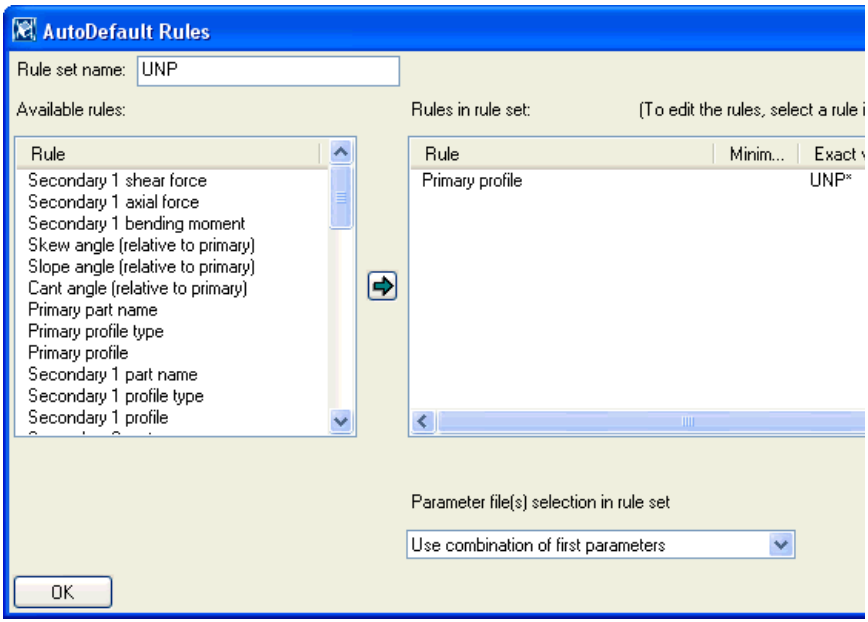


Double-click the Construsoft group. All component pages will appear. Double-click page 5, that is the page which contains the base plate component, than double-click component 1004.



Several rule sets are defined. The name of a rule is meaningless, it's only for more clearness for the user.

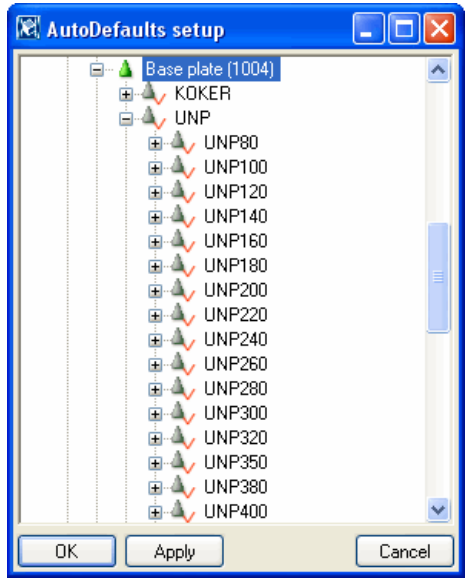
Right mouse click on 'UNP', now select "Edit rule set" from the contextmenu.



The rule is displayed in the right side of the dialog box. In this example, the main (primary) part must be an UNP profile. The size of the profile is irrelevant. So, whether it is an UNP140 or UNP300, that does not matter. If the profile matches the rule, Tekla Structures can enter the loop.

If you close the dialog box by pressing the cross-button, the changes made in the dialog box will not be stored. After applying changes, always close the dialog box with the <**OK**> button.

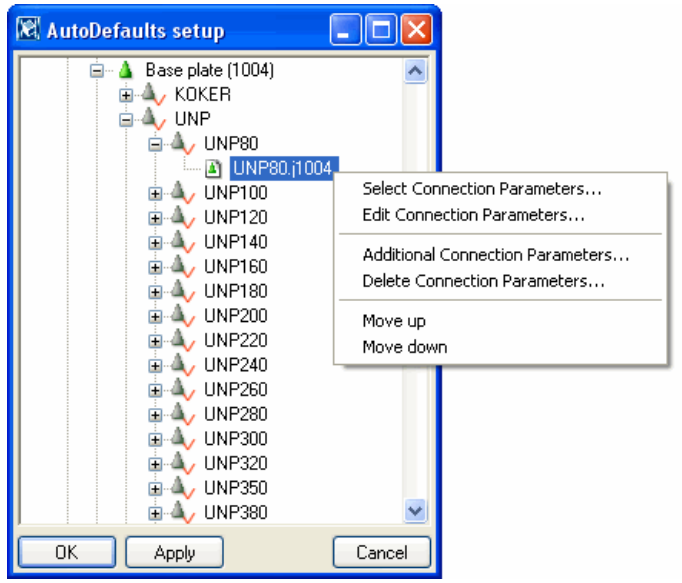
Click <**OK**> to close the dialog box. You will return to the previous dialog box, AutoDefaults Setup.



Right-mouse-click on rule UNP80, select Edit Rule set from the contextmenu.

One rule is set: the primary profile must be an UNP80 profile. All UNP profile sizes have this same rule, but for their accompanying sizes of course.

Now double-click UNP80. A new icon appears in the tree. This is the component setting that is used for this rule.



Click right mouse on the component setting, a contextmenu appears. The option **Select connection parameters** will open another dialog box where a component setting can be selected. If **Edit Connection parameters** is selected, all possible component settings are displayed. These settings can be edited also.

In this way you can attach an other component setting to this UNP80 rule.

A component setting is made for each UNP size. For example; if a UNP200 is present in the model, Tekla Structures will operate as follows:

A UNP200profile matches the UNP-rule (Primary part=UNP*). So the loop will be entered. Next, Tekla Structures comes to UNP80. A UNP200 profile does not match the rule for the UNP80 profile (rule= prim.prof UNP80). Tekla Structures will continue downwards to UNP100, UNP120, etc. until it comes to the UNP200 profile, which rule matches the profile. Now Tekla Structures will enter the loop and check which component setting is attached.

Exercise

Make that a tube column D<100 mm gets 2 bolts, tube column D100 - 200 needs 4 bolts and tube column D>200 needs 6 bolts.

First create 3 settings for base plate component1004. After that, define the Auto-Defaults.

What if it also counts for seamless tubes? What to edit?

AutoConnection: manually

Position a HEA240 column and a IPE300 beam in the model, like described on page 1. For component #144, load setting "AD" and apply the component to the column-beam connection.

A decent connection will be created. When modifying the beam profile to a IPE160 profile, the connection will change.

Also this feature is defined in AutoDefaults. Take a look at this setting.

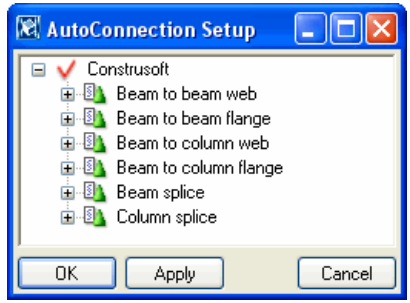
In this way you can generate all connections in the model. When a modification is needed. all connection will change automatically (if needed).

For more automate, another function is added: automatic AutoConnection

AutoConnection: automatically

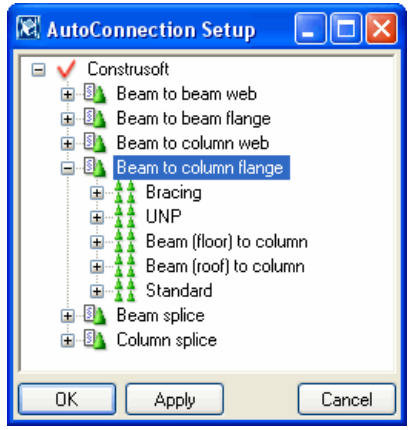
For faster modelling also AutoConnection has to be defined. In AutoConnection is defined which component has to be applied in which situation.

Six situations are defined (**Tools > AutoConnection**):

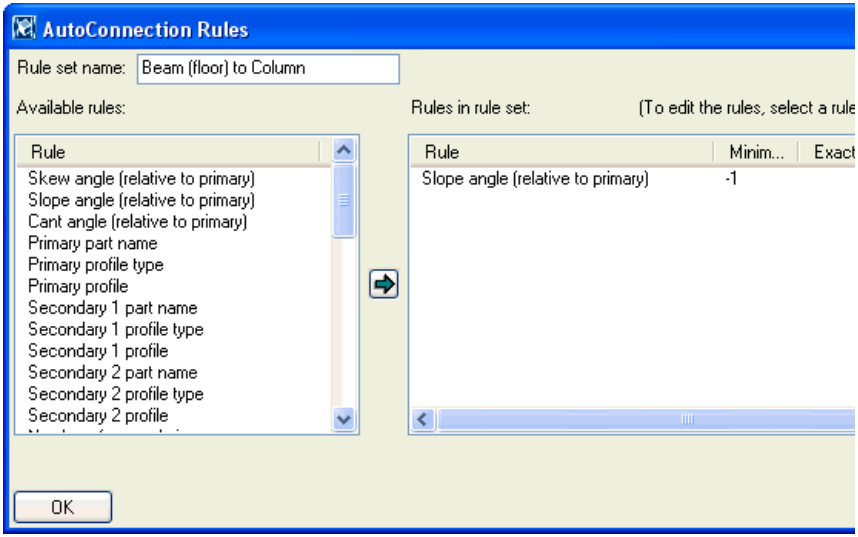


Just like in AutoDefaults, several groups can be created. Each group has 6 options.

The connection mentioned in the previous paragraph is of type **Beam to Column (flange)**. The setting looks like:

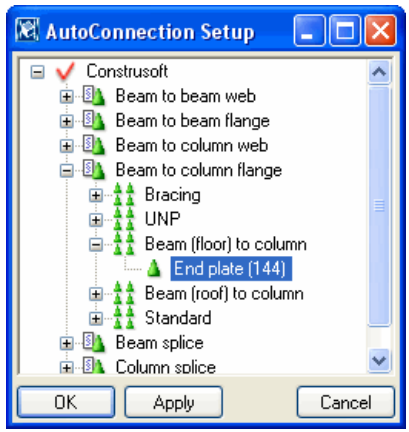


By pressing the right mouse button on a rule (*Edit Rule set*), you can see the condition for that rule.



In the rule mentioned above, the slope angle of the connecting beam may not vary more than -1 and +1 degree compared to the primary part (in this case the column).

If the situation meets this rule, than component 144 will be used. See picture.

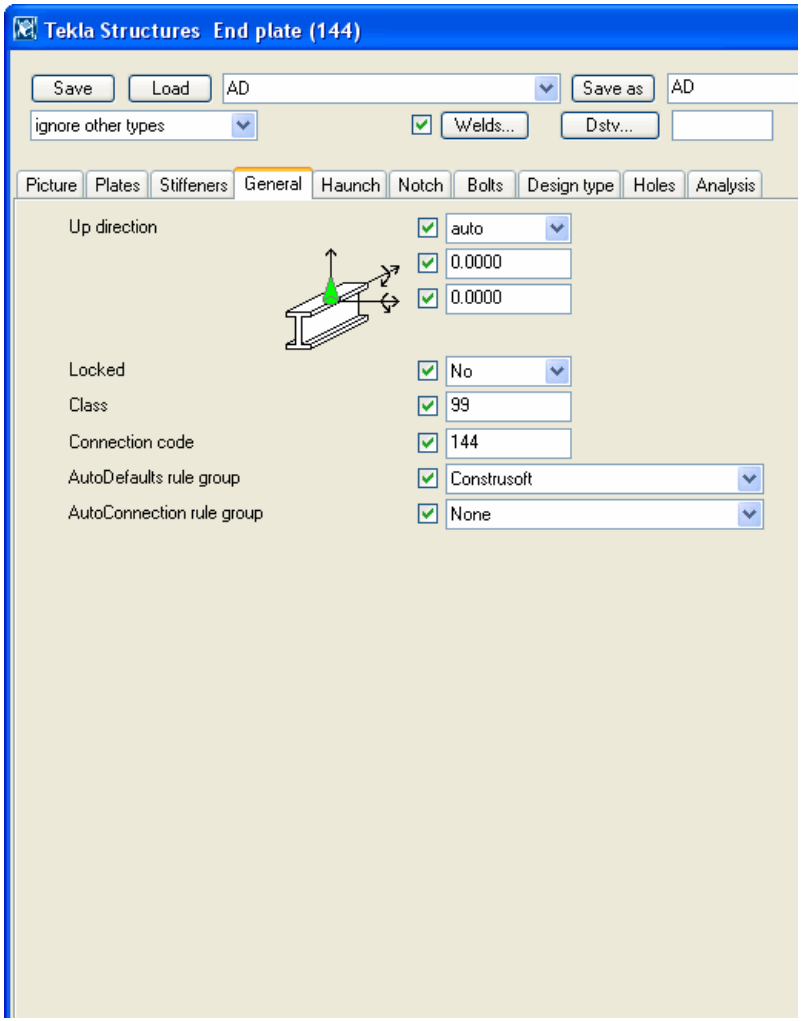


Method

1. First create the settings for the component to be used
2. Test this setting as described in *AutoConnection: manually op pagina 10*
3. Finally, define the AutoConnection

Extra information

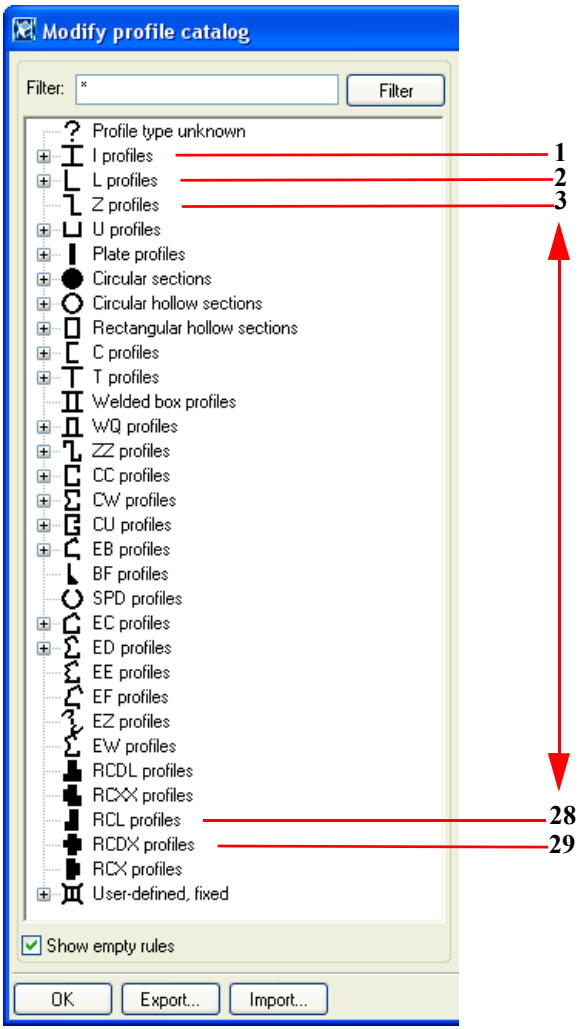
When you load component setting **AD**, all input fields will be empty. But the option AutoDefaults Group is defined (tab **General**).



If this setting is applied, Tekla Structures will consult the definitions in AutoDefaults. But if you enter a single value in de component dialog box, that value will taken. For example: if a plate thickness is set to 20mm, but in AutoDefaults that thickness is set to 10mm, than the value of 20mm will be taken. Even if the connected profiles are modified, the plate will retain the 20mm value. So, it is recommended that the **AD**-setting only is defined in the AutoDefaults group.

For correct functioning of the AC/AD, it is recommended to set the direction of the component to **auto.**

	Description
Profile name	Name of the profile, like BEAM, COLUMN, BRACING.
Profile type	Based on the number of the predefined cross section.



	Description
Profile	Names like IPE300, HE?140 en UNP*
Height	The ‘h’ value in the profile catalog
Web thickness	The ‘s’ value in the profile catalog
Flange thickness	The ‘t’ in the profile catalog
Material	Material of the profiles, set in the profiles’ properties
Flange width	The ‘b’ value in the in the profile catalog
Several Force rules	

Rule sets can be copied. Remind that AutoDefaults will adapt the component number.

Merging defaults

AutoDefaults are saved in file ***defaults.zxt***. Several defaults.zxt files can be merged. To do this:

- Create a folder named def1 and place the file defaults.zxt in this folder.
- Rename the file to ‘defaults.txt.gz’. The can now be extracted with Winzip.
- Extract the file to folder def1.
- The file defaults.zxt will appear in the folder.
- Do the same for the second defaults.zxt and create a folder named def2.
- Open the defaults.txt in Wordpad. Merge the two defaults.txt in one file. Remind that the names of the root are unique. See picture. The text next to ROOT must be different!
- After merging, save the file and rename it to defaults.zxt

```
ROOT   "Construsoft"
```

- Copy the file to the ‘attributes’ folder of your model, or to the folder ts. Saving the file in the attributes folder first is the best option, in this way you can test the file!

3.3 Ini file variables

Topics

This chapter is divided into following sections:

XS_ALWAYS_CONFIRM_SAVE_WHEN_CLOSING_DRAWING (p.18)

XS_BACKGROUND_COLOR (p.19)

XS_BASE_LINE_WIDTH (p.19)

XS_BASICVIEW (p.20)

XS_BLACK_DRAWING_BACKGROUND (p.21)

XS_CHAMFER_DISPLAY_LENGTH_FACTOR (p.21)

XS_CHAMFER_ACCURACY_FACTOR (p.21)

XS_CHANGE_WORKAREA_WHEN_MODIFYING_VIEW_DEPTH (p.21)

XS_CS_CHAMFER_DIVIDE_ANGLE (p.22)

XS_CONSIDER_NEIGHBOUR_PARTS_IN_HIDDEN (p.22)

XS_CREATE_ROUND_HOLE_DIMENSIONS (p.22)

XS_DIMENSION_DIGIT_GROUPING_CHARACTER (p.24)

XS_DIMENSION_DIGIT_GROUPING_COUNT (p.25)

XS_DIMENSION_EXTENSION_LINE_AWAY_FACTOR (p.25)

XS_DIMENSION_EXTENSION_LINE_TOWARD_FACTOR (p.25)

XS_DONT_SHOW_POLYBEAM_MID_EDGES (p.26)

XS_DO_NOT_COPY_GA_DRAWING_VIEW_FILTER_AUTOMATICALLY_FROM_MODEL_VIEW (p.27)

XS_DO_NOT_CREATE_PROFILE_DIMENSIONS_FOR_CONCRETE (p.27)

XS_DO_NOT_DISPLAY_CHAMFERS (p.27)

XS_DO_NOT_REMOVE_END_ABSOLUTE_DIMENSIONS (p.27)

XS_DRAW_CAST_PHASE_INTERNAL_LINES (p.28)

XS_DRAW_CHAMFERS_HANDLES (p.29)

XS_DRAW_LONG_HOLE_DIMENSIONS (p.29)

XS_DRAWING_GRID_LABEL_FRAME_FIXED_WIDTH (p.31)

XS_DRAWING_SHEET (p.31)

XS_DSTV_PRINT_NET_AND_GROSS_LENGTH (p.32)

XS_FLAT_THICKNESS_TOLERANCE (p.32)

XS_GRID_COLOR (p.33)

XS_HIDDEN_LINES_CHECK_TOLERANCE (p.33)

XS_MARK_ALL_BOLT_GROUPS_SEPARATELY (p.34)

XS_MARK_LEADER_LINE_POSITION_TYPE_FOR_NO_FRAME (p.35)

XS_MARK_LEADER_LINE_POSITION_TYPE_FOR_RECTANGULAR_FRAME (p.35)

XS_MAX_DECIMALS_IN_PROFILE_NAME (p.36)

XS_MIS_FILE_DIRECTORY (p.36)

XS_NO_END_VIEWS_TO_INCLUDED_SINGLE_DRAWINGS (p.37)

XS_NO_RELATIVE_SHAPE_DIMENSIONS (p.38)

XS_NO_SINGLE_PART_DRAWINGS_FOR (p.38)

XS_OMIT_MARKS_OF_HIDDEN_PARTS_IN_GA_DRAWINGS (p.38)

XS_OMIT_MARKS_OF_PARTS_OUT_OF_VIEW_PLANE_LIMIT_ANGLE (p.39)

XS_PART_MARKS_ALWAYS_VISIBLE_IN_DRAWING (p.39)

XS_PART_POSITION_TO_LEADING_EDGE (p.40)

XS_PART_POSITION_TO_LEADING_EDGE_IN_COLUMNS_ALSO (p.41)

XS_PIXEL_TOLERANCE (p.42)

XS_REFERENCE_ROUND_SEGMENTS (p.42)

XS_RENDERED_CURSOR_LINE_WIDTH (p.42)

XS_RENDERED_FOG_START_VALUE (p.42)

XS_RENDERED_PIXEL_TOLERANCE_SCALE (p.43)

XS_ROTATE_CUT_VIEWS (p.43)

XS_ROUND_SEGMENTS (p.45)

XS_SMALL_TUBE_ROUND_SEGMENTS (p.45)

XS_LARGE_TUBE_ROUND_SEGMENTS (p.45)

XS_SHOW_REVISION_MARK_ON_DRAWING_LIST (p.45)

XS_SINGLE_EXCLUDE (p.46)

XS_SINGLE_NO_RELATIVE_SHAPE_DIMENSIONS (p.48)

XS_SOLID_BUFFER_SIZE (p.48)

XS_SOLID_USE_HIGHER_ACCURACY (p.48)

XS_STANDARD_GUSSET_WIDTH_TOLERANCE (p.49)

XS_STANDARD_STIFFENER_WIDTH_TOLERANCE (p.50)

XS_TUBE_UNWRAP_WITH_CUT_HOLES (p.50)

XS_UPDATE_MARK_PLACING_IN_DRAWING (p.52)

XS_UPSIDE_DOWN_TEXT_ALLOWED (p.53)

XS_USE_ASSEMBLY_NUMBER_FOR (p.55)

XS_USE_SMALLER_GUSSET_PLATE (p.57)

XS_USE_TUBE_INNER_LENGTH_IN_DIMENSIONING (p.58)

XS_VIEW (p.58)

XS_VIEW_DIM... (p.58)

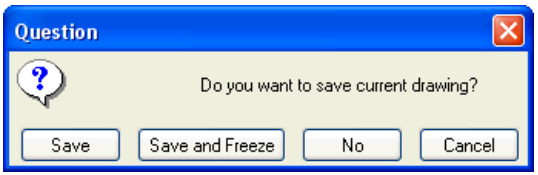
XS_VIEW_PART_LABEL_COLOR (p.60)

XS_ALWAYS_CONFIRM_SAVE_WHEN_CLOSING_DRAWING

Category **Drawing properties**

If you close a drawing without making any changes, Tekla Structures does not prompt you to save the drawing.

Enter TRUE in the value field to always display the confirmation dialog box:



XS_BACKGROUND_COLOR

Category **Model view**

Use this variable and also the variables XS_BACKGROUND_COLOR2 up to and XS_BACKGROUND_COLOR4 to define the background color in rendered views. Define the color using the RGB (Red Green Blue) values. The scale is from 0 to 1.

Enter the same value for all the background color variables to make use of an even color.

Examples

Color	RGB	Color	RGB
Black	0 0 0	Red	1 0 0
White	1 1 1	Yellow	1 1 0
Blue	0.21 0.46 0.88	Green	0 1 0
Dark blue	0 0 1	Grey	0.75 0.75 0.75
	0.9 0.9 0.9 0.9 0.9 0.9 0.2 0.3 0.5 0.2 0.3 0.5		



An even background color is more calm for your eyes and uses less memory compared to a gradient background color.

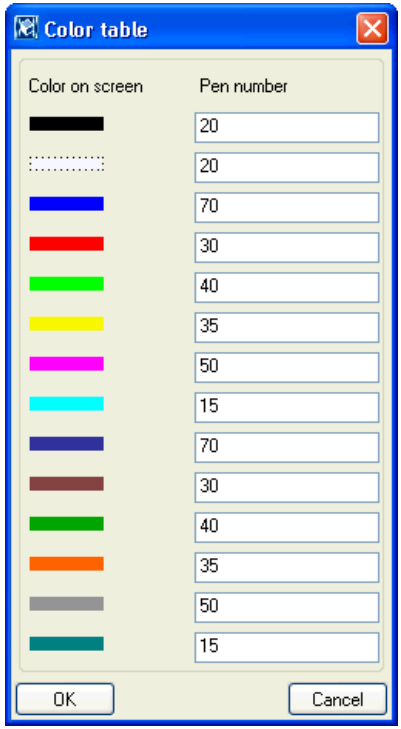
The changes are stored in the file `options_Username.ini`.

Re-open the view to activate the changes.

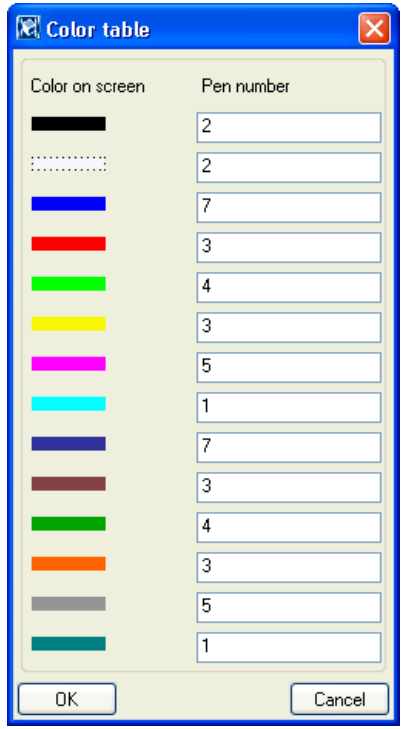
XS_BASE_LINE_WIDTH

Category **Plotting**

Use to specify the base line width for printed drawings. The default value is 0.1.



XS_BASE_LINE_WIDTH=0.01



XS_BASE_LINE_WIDTH=0.1

In case you want to make use of line thicknesses such as 0,35 mm, enter the value 35 in the dialog box **Color table** and enter the value 0.01 for the variable.

The final line thicknesses in a printed drawing is base line width multiplied by pen numbers from the **Color table** dialog box.

XS_BASICVIEW

Category **Model view**

Use to control the size and position of basic views (Part basic views and Connection basic views).

Example

XS_BASICVIEW_POSITION_X=100

XS_BASICVIEW_POSITION_Y=20

XS_BASICVIEW_WIDTH=650

XS_BASICVIEW_HEIGHT=570

XS_BLACK_DRAWING_BACKGROUND

Category **Drawing view**

Set to TRUE to have a black backgound and colored lines in drawings. To enable white background and black lines, clear the Value field.

XS_CHAMFER_DISPLAY_LENGTH_FACTOR

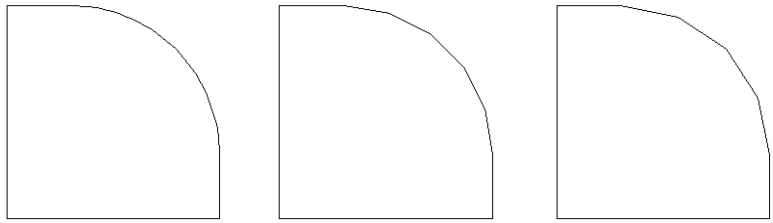
Category **Model view**

Use to adjust the chamfer length. The higher the value, the longer the chamfer-symbol.

XS_CHAMFER_ACCURACY_FACTOR

Category **Speed and accuracy**

Use to define the number of points used for chamfers. The default value is 1.0. Use values smaller than 4.0 to increase the number of points. Use values larger than 1.0 to decrease the number of points (smallest value is 0.1)



=0.1 =3.0 =8.0

XS_CHANGE_WORKAREA_WHEN_MODIFYING_VIEW_DEPTH

Category **Model view**

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

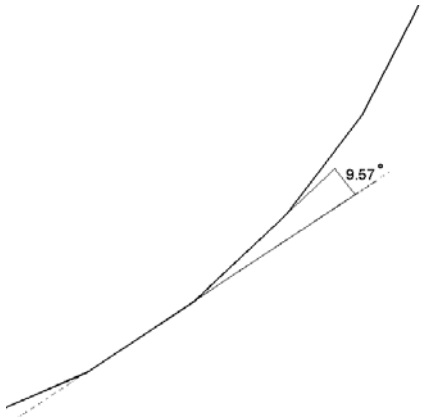
Set to TRUE to automatically adjust the work area when you change the display depth.

XS_CS_CHAMFER_DIVIDE_ANGLE

Category **Speed and accuracy**

Use to change the angles dividing a cross section chamfer (default 10.0 degrees).

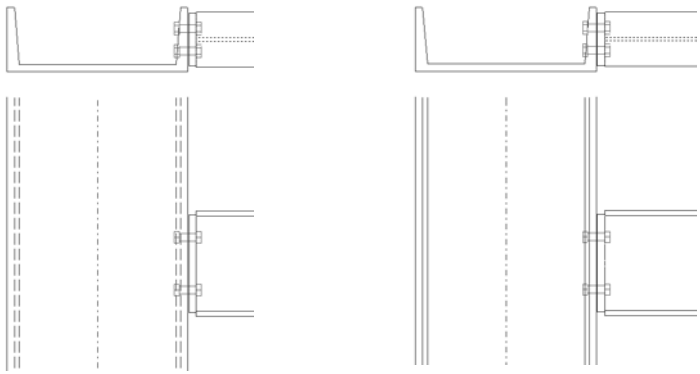
Example XS_CS_CHAMFER_DIVIDE_ANGLE=10.0



XS_CONSIDER_NEIGHBOUR_PARTS_IN_HIDDEN

Category **Drawing properties**

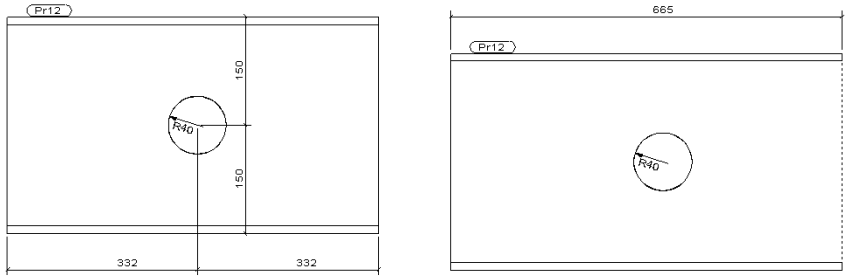
Set to draw parts behind neighboring parts using hidden lines.



XS_CREATE_ROUND_HOLE_DIMENSIONS

Category **Dimensioning: parts**

Enter TRUE in the Value field to have Tekla Structures dimension the round hole center point in drawings.

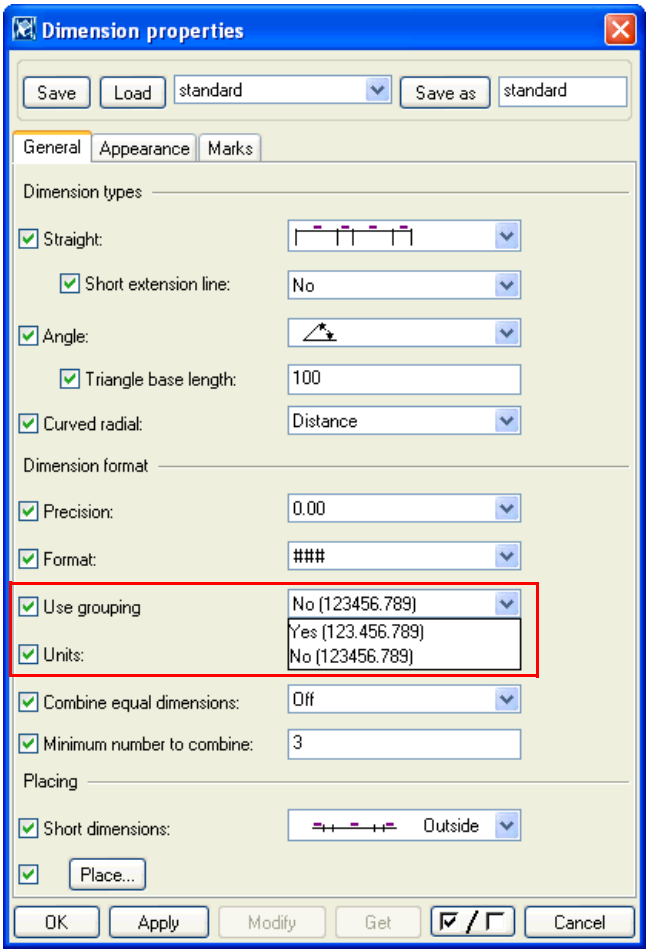


XS_DIMENSION_DIGIT_GROUPING_CHARACTER

Category **Dimensioning: general**

Use to specify which separator character to use in large dimension values: for example a comma (,), where 154321 would become 154,321.

If no character is specified, Tekla Structures uses a space in dimensions when you select **Yes** in **Dimension properties** > **General** > **Use grouping**.



See also **XS_DIMENSION_DIGIT_GROUPING_COUNT** (p.25)

XS_DIMENSION_DIGIT_GROUPING_COUNT

Category **Dimensioning: general**

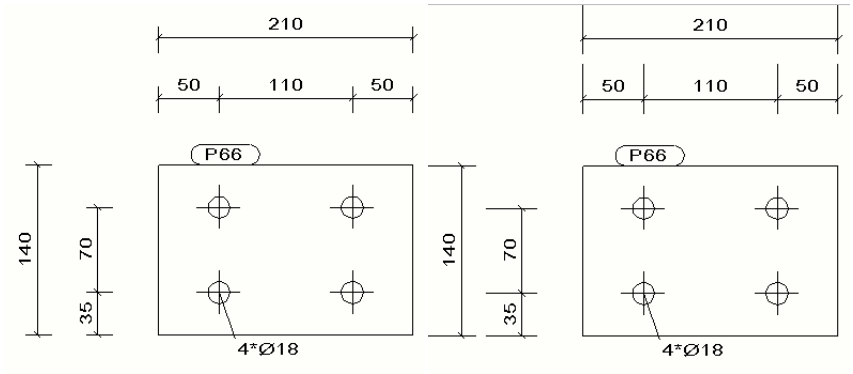
Use to specify the number of digits after which a separator character will be inserted in dimension values: for example 3, where 154321 would become 154 321.

See also **XS_DIMENSION_DIGIT_GROUPING_CHARACTER** (p.24)

XS_DIMENSION_EXTENSION_LINE_AWAY_FACTOR

Category **Dimensioning: general**

Use the **Yes** option for **Short extension line** on the tab **General** in the Dimension properties to create extension lines all the same length

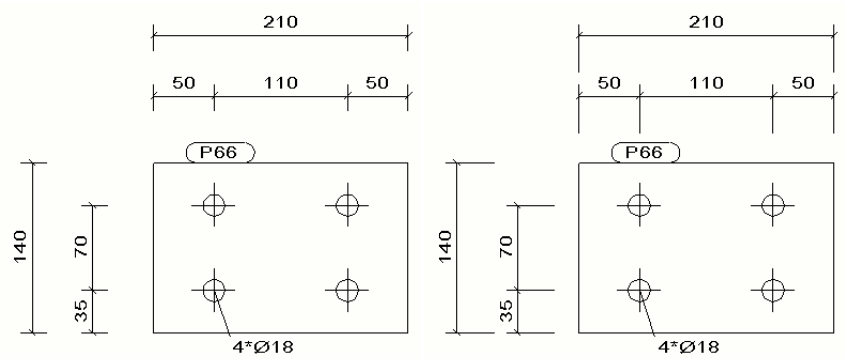


1.0 3.0

XS_DIMENSION_EXTENSION_LINE_TOWARD_FACTOR

Category **Dimensioning: general**

Use the **Yes** option for **Short extension line** on the tab **General** in the Dimension properties to create extension lines all the same length



1.2 3.0

XS_DONT_SHOW_POLYBEAM_MID_EDGES

Category **Drawing properties**

Set to TRUE to hide polybeam fold and bend lines in drawings.



XS_DO_NOT_COPY_GA_DRAWING_VIEW_FILTER_AUTOMATICALLY_FROM_MODEL_VIEW

While creating general arrangement drawings this variable takes or doesn't take over the filter settings from the model view. This variable is standard set to "rem". Set this variable to make it work like in earlier versions.

For example in a model view only phase 1 is visible. Phase 2 and 3 are invisible. Creating a general arrangement drawing results in a drawing containing the filter settings from the model, so only phase 1 is visible. Phase 2 and 3 are invisible.

The General arrangement drawing properties don not show what filter is in use. However, Tekla Structures knows what filter is in use and will take that into account when, for example part marks change. Also the view properties know what filter is in use.

XS_DO_NOT_CREATE_PROFILE_DIMENSIONS_FOR_CONCRETE

Set to TRUE to prevent Tekla Structures from automatically displaying the profile dimensions of concrete parts in cast-unit drawings.

XS_DO_NOT_DISPLAY_CHAMFERS

Category **Model view**

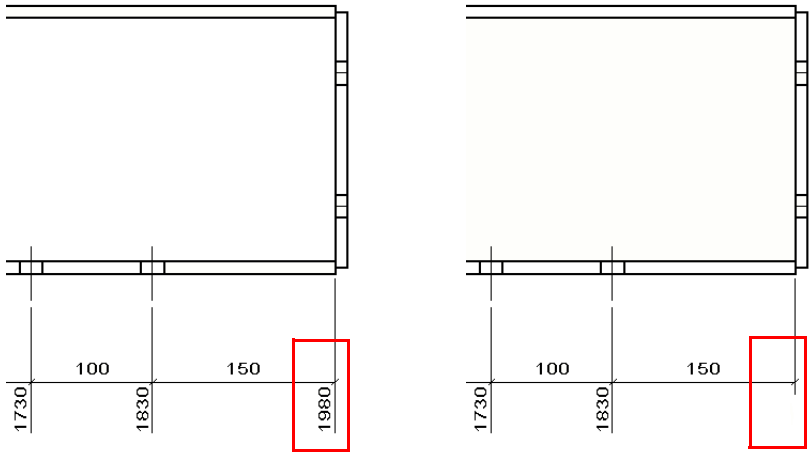
Set to TRUE to draw polygon plates without chamfers. This variable only takes effect in rendered views. Useful when a picture is needed.

The disadvantage is that you cannot select the chamfers in the model any more.

XS_DO_NOT_REMOVE_END_ABSOLUTE_DIMENSIONS

Category **Dimensioning: parts**

When you use absolute dimensions, by default, Tekla Structures removes the last absolute vertical dimension. To prevent this, set to TRUE.

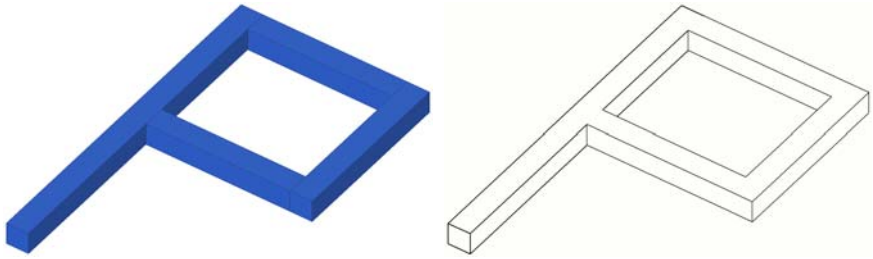


XS_DRAW_CAST_PHASE_INTERNAL_LINES

Category **Concrete detailing**

Use to show or hide edge lines of concrete cast units in drawings. Set to TRUE to show overlapping edge lines between adjacent cast units.

If cast unit edges touch each other, the coinciding lines will not be visible on cast unit drawings. However, these lines remain visible in the modeling environment:



Model: contact planes visible **Drawing**: no lines

For part's contact planes for parts which are not added using the Part Add command, a variable is available to define whether the lines are visible or not.

If you want the contact lines being visible on cast unit drawings, set the variable `XS_DRAW_CAST_PHASE_INTERNAL_LINES` in **Tools** > **Advanced options** > **Concrete detailing** to TRUE.

If you want to hide the contact plane lines, set the variable to FALSE.

Default settings for this variable are depending which module is in use. Go to **Advanced options** to find out which setting is being used.

If the planes don't (just) coincide, the part edges will be visible on drawings. Copy variable

set XS_HIDDEN_LINES_CHECK_TOLERANCE=0.1 from file `env_global_default.ini` (located in folder ..*environments**country-independent*) to file `user.ini` (located in folder ..*nt**bin*) to enter a margin for contact lines being visible of being hidden on drawings.

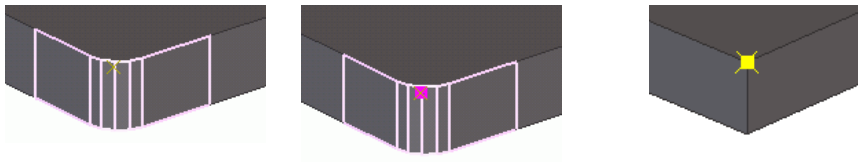
If parts are not added to each other (using the Part Add command), then the contact planes will always be visible on drawings.

XS_DRAW_CHAMFERS_HANDLES

Category **Model view**

Hide or show the handles or chamfers of contour plates and concrete slabs. The options are:

- `HANDLES`: Only shows handles.
- `CHAMFERS`: Only shows chamfers.
- `CHAMFERS_AND_HANDLES`: Shows both handles and chamfers.

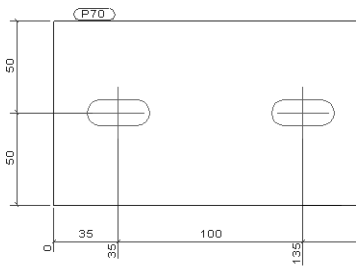


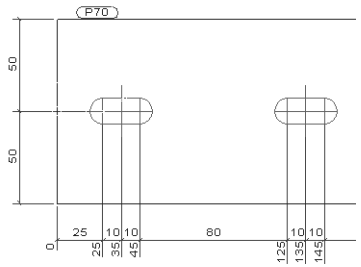
CHAMFER CHAMFER_AND_HANDLES HANDLES

XS_DRAW_LONG_HOLE_DIMENSIONS

Category **Dimensioning: bolts**

Set to 1 to dimension slotted holes to the center points of the curve. By default slotted holes are dimensioned to the center points of the holes.


P70
50
50
35
100
0
35
135

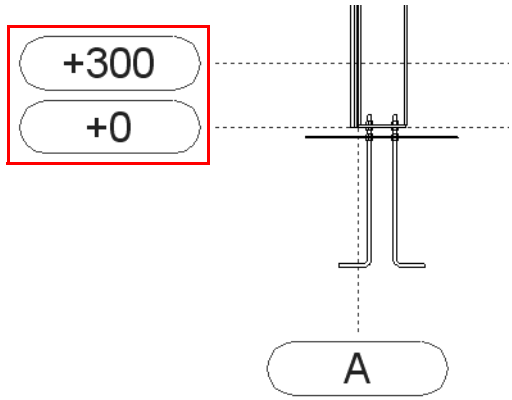

P70
50
50
25
10
10
80
10
10
0
25
35
45
125
135
145

XS_DRAWING_GRID_LABEL_FRAME_FIXED_WIDTH

Category **Drawing properties**

Use this variable to define the grid label frame size. This is useful for example when you want all the grid label frames to be the same size no matter whether there are one or two digits in the frame. This gives the grid a consistent appearance.

The default value is 0, which indicates that the width of the grid label frame depends on the width of the grid label. To increase or decrease the width of the frame, enter the desired value in millimeters, in this example, value 12 has been used:



XS_DRAWING_SHEET

Category **Drawing view**

Use to define the initial position of the drawing sheet. This is useful when using a dual display. If the Tekla Structures model view is maximized, the first opened drawing will be maximized too. If another drawing is opened, that drawing will not be maximized by default. To have the drawing opened in a large window, you can set this in the following ways:

XS_DRAWING_SHEET_POSITION_X=0

XS_DRAWING_SHEET_POSITION_Y=0

XS_DRAWING_SHEET_HEIGHT=810

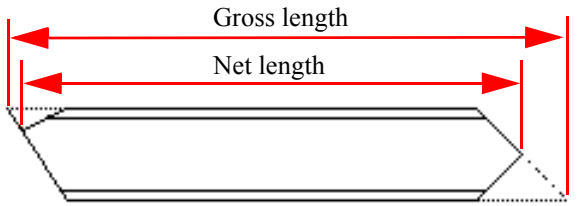
XS_DRAWING_SHEET_WIDTH=2000

The values 810 and 2000 determine the size of the drawing window. The exact values depend of the used toolbars and the location of those toolbars.

XS_DSTV_PRINT_NET_AND_GROSS_LENGTH

Category **CNC**

Use this variable to insert two length values in NC DSTV files. This variable is set to TRUE by default.



Because of this, both net and gross length are included in the NC files' header, according to the DSTV standard:

In the NC file

```
ST
** B1.nc1
  project number
  1
  B1
  B1
  S235JR
  1
  IPE300
  I
    2704.89,2710.64
     300.00
     150.00
      10.70
       7.10
      15.00
     42.200
      1.159
      0.000
     15.745
      0.000
      0.000
```

XS_FLAT_THICKNESS_TOLERANCE

Category **Plate work**

Use to set the checking tolerance for flat bars. Tekla Structures uses this value to check plate thickness to determine whether to convert it to a flat bar. The default value is 0.4 mm.

XS_GRID_COLOR

Category **Model view**

Use this variable to change the color of the grid in the Model Editor.



Define the color of the grid using RGB values:

```
XS_GRID_COLOR = <Red> <Green> <Blue>
```

Define the values on a scale of `0` to `1`. Reopen the view for the change to take effect.

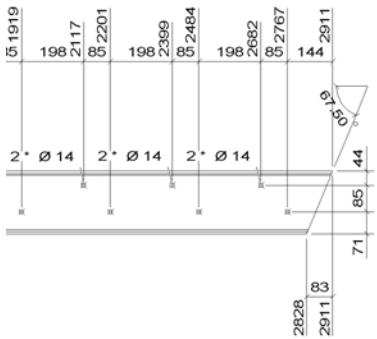
The changes are stored in the file `options_Username.ini` in the folder `disc:\TeklaStructures\version\nt\bin` and are used for all models.

XS_HIDDEN_LINES_CHECK_TOLERANCE

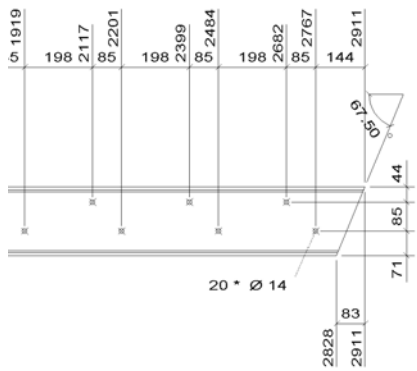
Defines the distance within which Tekla Structures treats part lines inside cast units as overlapping lines. The default value is 0.01.

XS_MARK_ALL_BOLT_GROUPS_SEPARATELY

Use this variable for combining hole-/bolt labels. By default, the labels are not combined.



hole-/bolt labels are displayed per bolt group, by default



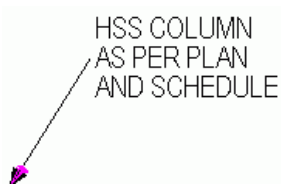
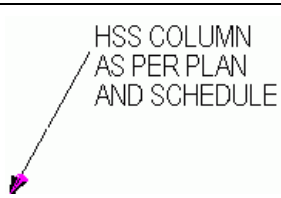
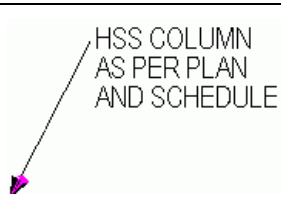
hole-/bolt labels are combined

XS_MARK_LEADER_LINE_POSITION_TYPE_FOR_NO_FRAME

Category **Marking: general**

To use different options for a leader line with **no** frame, use this variable. The default is `0`.

The following table includes information on the new positioning options.

Position	Appearance	Set variable to
Nearest corner		0
In the middle of the text area		1
1/3 from the top of the text area		2
In the middle of the first row of text		3

Leader line extension length

To define the length of the leader line extension, use the variable `XS_MARK_LEADER_LINE_EXTENSION_LENGTH` in **Tools > Advanced options > Marking: general**. The default length is 0. Define the length in millimeters.

XS_MARK_LEADER_LINE_POSITION_TYPE_FOR_RECTANGULAR_FRAME

Category **Marking: general**

To use the different options for a leader line **with** a rectangular frame, use this variable. The default is 0.

See also XS_MARK_LEADER_LINE_POSITION_TYPE_FOR_NO_FRAME (p.35).

XS_MAX_DECIMALS_IN_PROFILE_NAME

Category **Profiles**

Use to control the number of decimals in profile names. The default value is 1.

Plates are being showed on drawings and reports as: e.g. PL137.8/10.

Set this variable to 0 and the plate will be changed into PL138/10.



Also the thickness of the plate will also be changed:

PL1.5/100 will be PL2/100!

XS_MIS_FILE_DIRECTORY

Category **CNC**

Points to the folder where NC and MIS files are created. The default is the current model folder.

You can also enter folder as a relative path to the current model folder using the ".\ " characters in the **Value** field. If you set `XS_MIS_FILE_DIRECTORY=.\NC`, Tekla Structures creates NC and MIS files in NC folder under the current model folder.

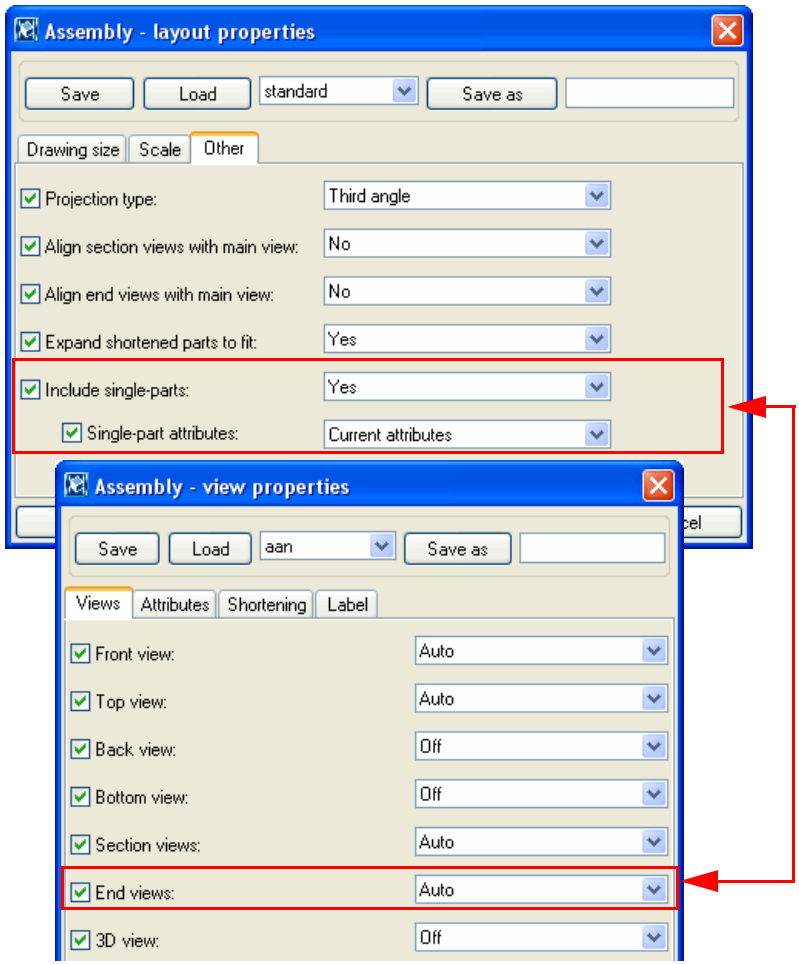
Example

```
set XS_MIS_FILE_DIRECTORY=c:\TeklaStructures\mis-files
```

XS_NO_END_VIEWS_TO_INCLUDED_SINGLE_DRAWINGS

Category **Single part view in assembly drawing**

Set to TRUE to prevent Tekla Structures from creating end views when you create an assembly drawing and select to include single-part drawings. If this variable is disabled, Tekla Structures creates end views based on the properties of the single-part drawings.

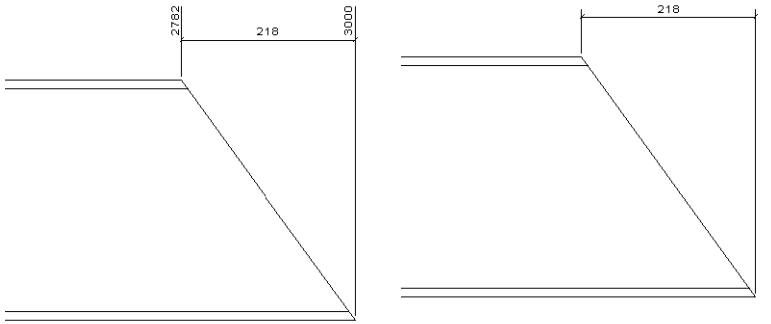


This variable is only effective when the option ‘Single Part Attributes’ is set to *Current attributes*. If another option is set, then the chosen setting will determine if end views are added to the single part views.

XS_NO_RELATIVE_SHAPE_DIMENSIONS

Category **Dimensioning: parts**

Set to force shape dimensions to be the same as the dimension type you select on assembly drawings.



Example XS_NO_RELATIVE_SHAPE_DIMENSIONS=TRUE

By default, shape dimensions are always relative, no matter which dimension type you select.

XS_NO_SINGLE_PART_DRAWINGS_FOR

Category **Single part view in assembly drawing**

Set as follows to prevent Tekla Structures producing single part drawings for assemblies containing one part:

XS_NO_SINGLE_PART_DRAWINGS_FOR=LOOSE_PARTS

XS_OMIT_MARKS_OF_HIDDEN_PARTS_IN_GA_DRAWINGS

Category **Marking: parts**

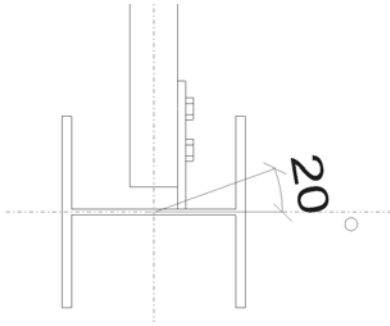
Set to TRUE to hide part marks of parts hidden by other parts from views in general arrangement drawings.

Example If you have stiffeners either side of a beam, one will be hidden by the beam in front of it. Set this variable to TRUE to prevent Tekla Structures displaying the part mark of the hidden part.



XS_OMIT_MARKS_OF_PARTS_OUT_OF_VIEW_PLANE_LIMIT_ANGLE

Use to hide the part marks for parts outside the current view plane by defining the limit for inclusion as an angle.



You also need to set **Parts out of view plane** field in the **View part mark properties** to **Not visible** to hide parts from the view that are outside the angle you specify here. The default value is 20.0.

XS_PART_MARKS_ALWAYS_VISIBLE_IN_DRAWING

Category **Marking: parts**

Forces Tekla Structures to show marks for hidden parts in drawings. To define the drawing type, use one or more of the following letters:

Letter	Drawing type
W	Single part drawings
A	Assembly drawings

Letter	Drawing type
M	Multi drawings
G	GA drawings
C	Cast-unit drawings

Mark frames and leader lines appear as unbroken lines.

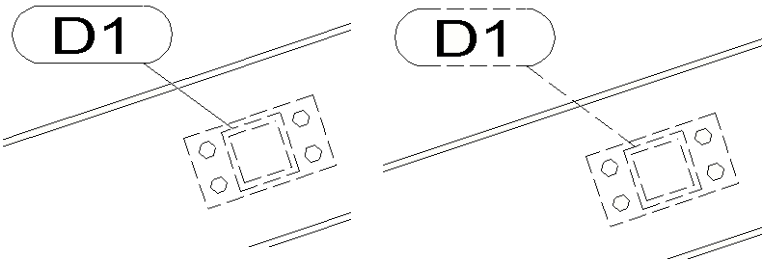
Examples

To show marks for hidden parts in assembly drawings:

set XS_PART_MARKS_ALWAYS_VISIBLE_IN_DRAWING=A

To show part marks in cast-unit and general arrangement drawings:

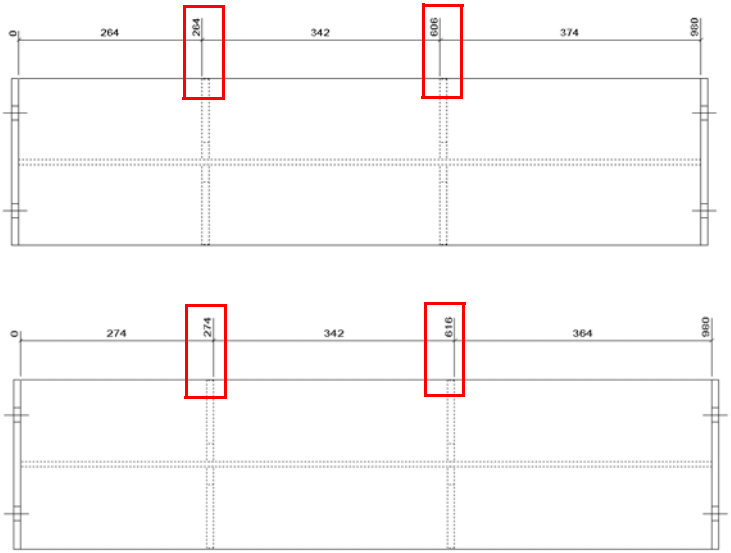
set XS_PART_MARKS_ALWAYS_VISIBLE_IN_DRAWING=CG



XS_PART_POSITION_TO_LEADING_EDGE

Category **Dimensioning: parts**

Forces Tekla Structures to dimension the position of beams to the leading edge, except columns.



XS_PART_POSITION_TO_LEADING_EDGE_IN_COLUMNS_ALSO

Category **Dimensioning: parts**

Forces Tekla Structures to dimension the position of columns to the leading edge.



XS_PIXEL_TOLERANCE

Category **Model view**

Variable to define the search area of the snap function. The default value is 25. If you are accidentally creating tiny zoom windows, set this variable to a higher value. Tekla Structures uses pixel tolerance to differentiate between a click and a drag when zooming. If the mouse moves less than the value while the mouse button is depressed, it is considered a mouse click.

XS_REFERENCE_ROUND_SEGMENTS

Category **Import**

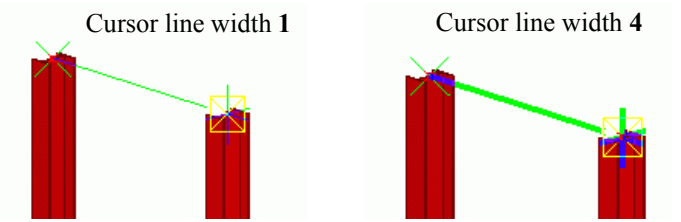
Use to define the number of round segments to use in reference models. The default is 16. Enter a bigger value to display the arcs less sharp-edged.

XS_RENDERED_CURSOR_LINE_WIDTH

Category **Model view**

Use to set cursor line width in rendered views. Enter the cursor line width in the **Value field**.

- Possible values are 1, 2, or 4. Any other value is handled as 1.
- The default is 2.



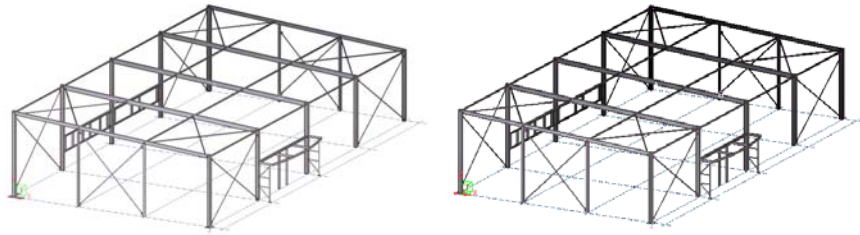
XS_RENDERED_FOG_START_VALUE

In rendered views distant objects appear progressively darker than close ones. Use the following variables to control the shade of objects:

set XS_RENDERED_FOG_START_VALUE=0.25

set XS_RENDERED_FOG_END_VALUE=0.50

Use values from 0 to 1 to control the shade of objects. The higher the value, the darker the distant objects.



XS_RENDERED_PIXEL_TOLERANCE_SCALE

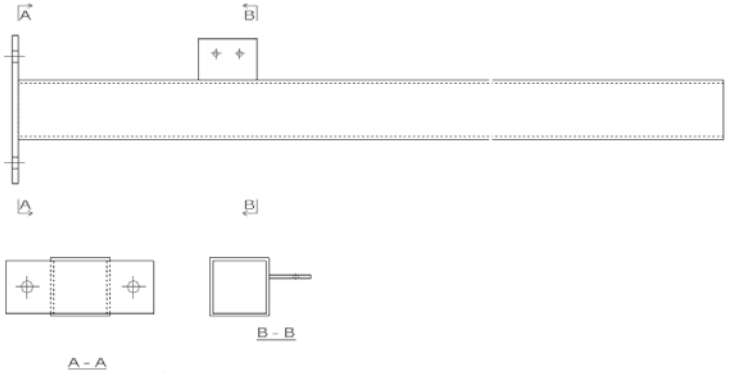
Tekla Structures uses pixel tolerance to differentiate between clicking and dragging the mouse when zooming. Use this variable to define the pixel tolerance.

The default value is 0.7. If the mouse moves less than the value you specify with the mouse button depressed, it is treated as a click.

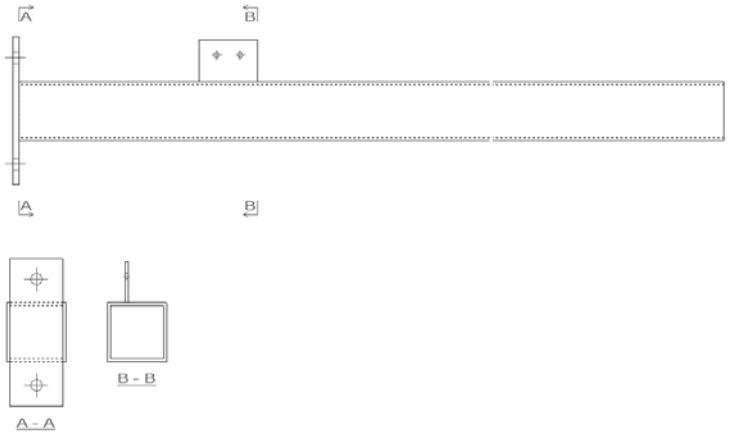
XS_ROTATE_CUT_VIEWS

Category **Drawing View**

Use to specify the rotation of section views.



BY_MAIN_VIEW (default) uses the same rotation as the orientation as the main view.



BY_SYMBOL_MAIN_VIEW, uses the orientation of the view that contains the section symbol.

This only applies to section views that Tekla Structures creates automatically. Manually-created views have the same rotation as the view they are created from.

XS_ROUND_SEGMENTS

Category **Speed and accuracy**

This variable defines the number of segments for representing tubes with diameters between 45 - 1000 mm.

For profiles between 100 -1000 mm, the default value is used.

For profiles between 45 -100 mm, 80% of this value is used, a tube Ø 60 shall be represented by 32 segments, if the variable value is set to 40 (0,8 * 40 = 32).

The default is 40.

XS_SMALL_TUBE_ROUND_SEGMENTS

Category **Speed and accuracy**

Use to define the number of segments in round tubes with a diameter smaller than 45 mm. The default value is 12 segments. Using a smaller value increases Tekla Structures's performance.

XS_LARGE_TUBE_ROUND_SEGMENTS

Category **Speed and accuracy**

This variable defines the number of segments for large tubes (> 1000 mm).

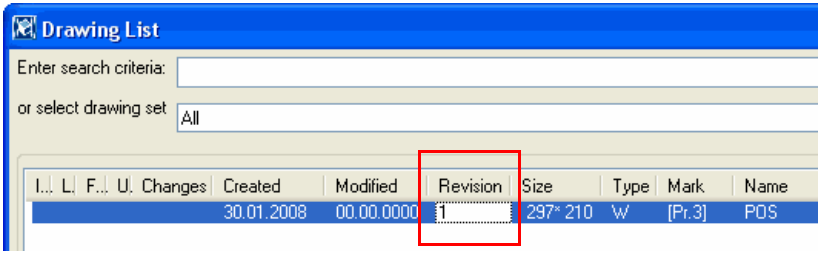
The default is 100.

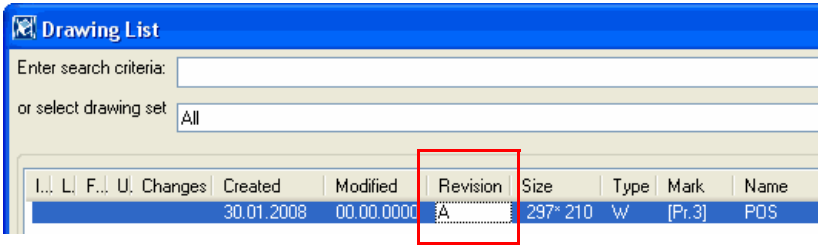
XS_SHOW_REVISION_MARK_ON_DRAWING_LIST

Category **Speed and accuracy**

Include the Revision mark field in the drawing list in the modeling environment.

To include the Revision mark field, enter 1. To include the Revision number field, enter 0. By default, the Revision number field is shown.



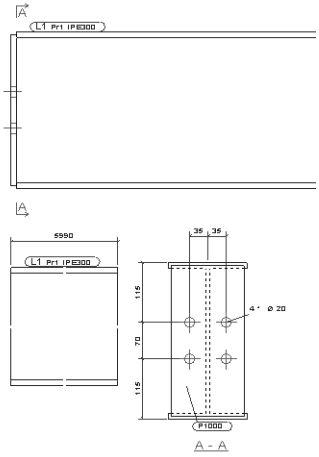


XS_SINGLE_EXCLUDE

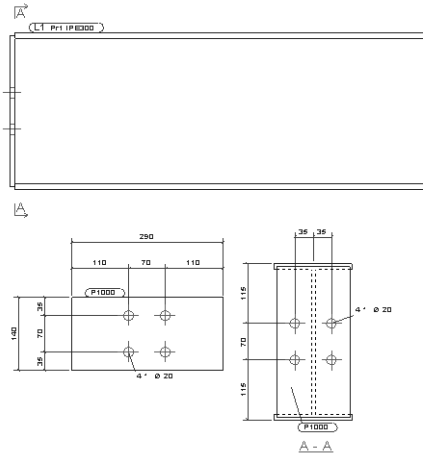
Category **Single part view in assembly drawing**

Use to exclude single-part views from assembly drawings. The options are:

- NONE includes all single part views.
- MAIN_SHAFT includes single part views of all parts, except assembly main parts.
- ALL_SHAFTS includes single part views of all parts, except those with another object welded to them (i.e. primary parts).
- AUTOMATIC includes single part views of all parts, except the longest main part in the assembly.
- ALL_BUT_MAIN_PART includes single part views of assembly main parts only.



ALL_BUT_MAIN_PART

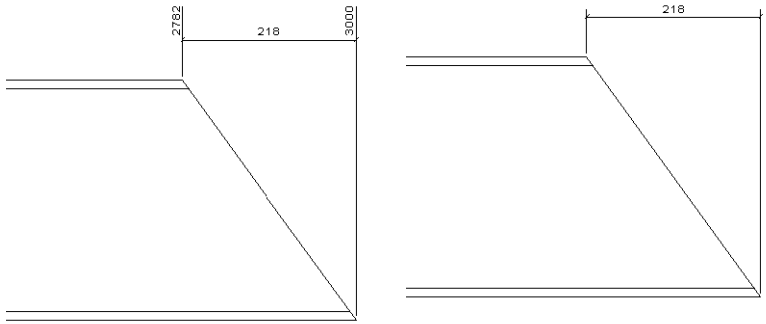


MAIN_SHAFT

XS_SINGLE_NO_RELATIVE_SHAPE_DIMENSIONS

Category **Dimensioning: Parts**

Forces shape dimensions to be the same as the dimension type you select on single part drawings.



XS_SOLID_BUFFER_SIZE

Category **Speed and accuracy**

Increasing this value also increases the memory used by Tekla Structures processes.

Here are approximate recommendations for the solid buffer size, based on the amount of RAM your system has:

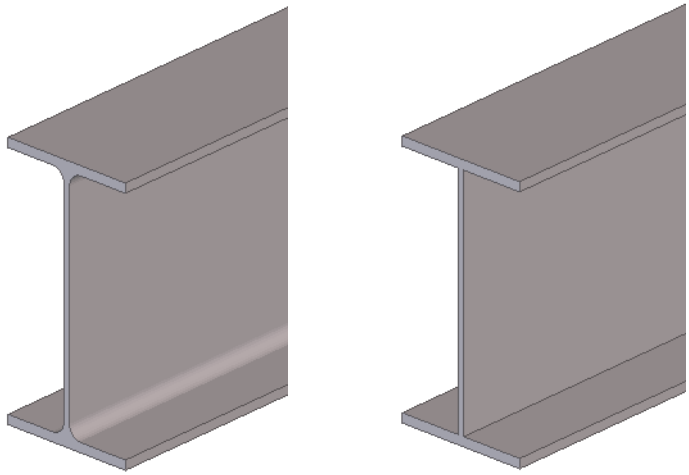
Physical memory:	Solid buffer size value:
1024 MB	1500000
2048 MB	3000000

or as guideline: 1,5 * physical memory (RAM): 3Gb ==> ± 4500000

XS_SOLID_USE_HIGHER_ACCURACY

Use this variable to model profiles more accurately in model views, drawings, in the Web Viewer and in XML export files using profile roundings and sloping flanges. Using this variable also makes the weight value more accurate.

By default this variable is not used.



This variable increases the number of faces in solids, which slows down Tekla Structures. We recommend using this variable only when needed.

Limitations

- In components, some stiffeners might bite into flanges of columns or beams.
- Some bolt edge distances are not correctly calculated.

XS_STANDARD_GUSSET_WIDTH_TOLERANCE

Category **Components**

Use to define the tolerance between the actual and standard widths of gusset plates created using modeling tools or connections. Tekla Structures uses a standard plate file to define the default plate width including the tolerance value. Enter the value in millimeters.

Components that use this variable are:

- Welded gusset (10)
- Bolted gusset (11)
- Tube gusset (20)
- Corner tube gusset (56)
- Corner bolted gusset (57)
- Wraparound gusset (58),
- Hollow brace wraparound gusset (59)
- Wraparound gusset cross (60)
- Gusseted cross (62)
- Corner wrapped gusset (63)

XS_STANDARD_STIFFENER_WIDTH_TOLERANCE

Category **Components**

Use to define the tolerance between the actual and standard widths of stiffener plates created using modeling tools or connections. Enter the value in millimeters.

Tekla Structures reads the standard widths from the fltprops.inp file.

Components that use this variable are:

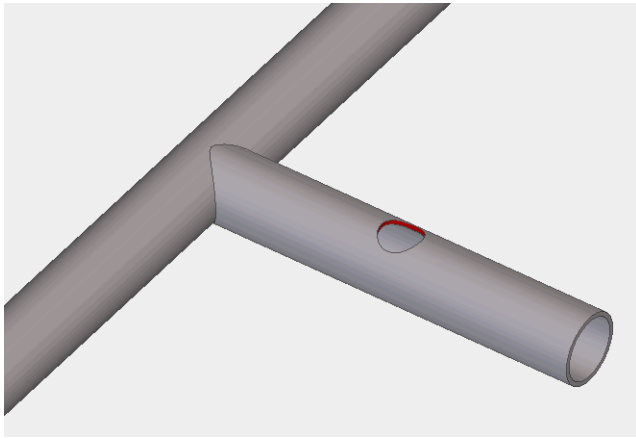
- Stiffeners (1003)
- Multiple stiffeners (1064)
- Haunch (40)

XS_TUBE_UNWRAP_WITH_CUT_HOLES

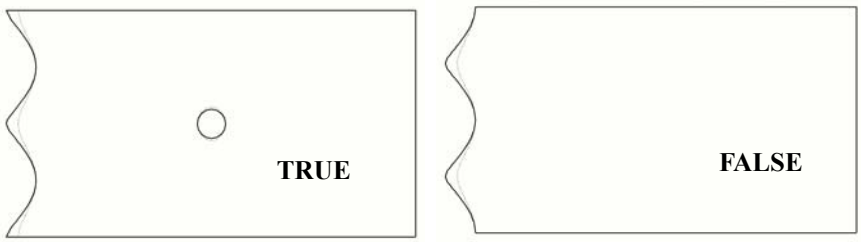
Category **Drawing Properties**

This variable is set to TRUE by default to include cut holes in workshop drawings or NC files of unwrapped CHS (circular hollow section) profiles.

Model



Drawing



However, it turns out that this variable disturbs the creation of unfolded views of tubular profiles which do not contain any holes.

Because of this, first create the unfolded views of tubular profiles which contain holes, after that change the variable and then create the unfolded views of tubular profiles which do not contain holes.

If you set this variable to TRUE, the variables XS_TUBE_UNWRAP_LIMIT_THICKNESS and XS_TUBE_UNWRAP_PAPER_THICKNESS have no effect.

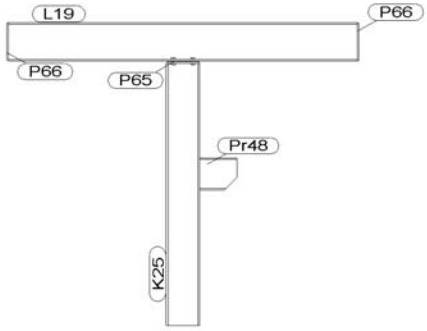
Note also that using this variable does not affect conical tube profiles.

XS_UPDATE_MARK_PLACING_IN_DRAWING

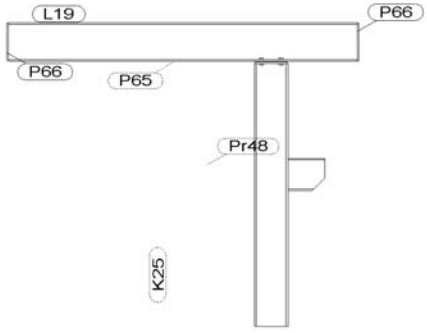
Category **Marking: Parts**

Updates the position of updated marks in specific drawing types. Use the letters in the following table to specify the drawing types. By default, this variable applies to all drawing types.

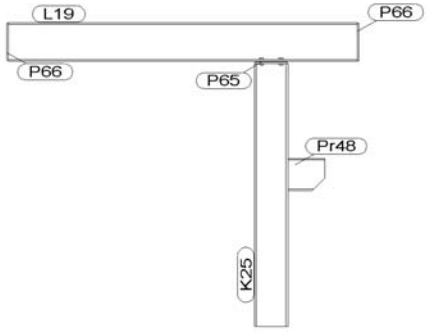
Letter	Drawing type
W	Single part drawings
A	Assembly drawings
M	Multi drawings
G	GA drawings
C	Cast-unit drawings



Situation: column is moved 500 in +x direction.



Drawing type not specified: Marks stay at original position after updating the drawing

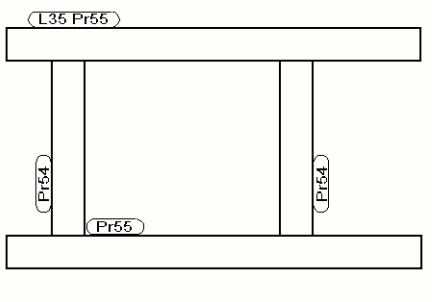


Drawing type specified: Marks are moved when updating the drawing.

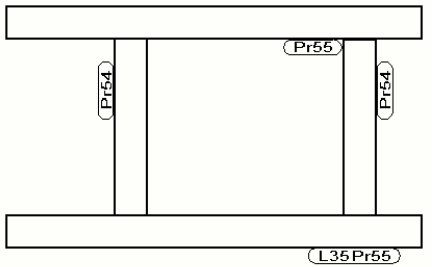
XS_UPSIDE_DOWN_TEXT_ALLOWED

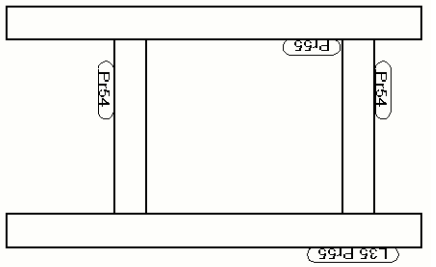
Category **Concrete detailing**

Set to TRUE to place part marks or other text upside down to illustrate that identical parts have a different orientation. If you do not set this variable, Tekla Structures automatically changes all marks or text that are rotated more than 90 degrees.



View will be orientated like this. Select view and rotate the view 180 degrees.





FALSE

TRUE

XS_USE_ASSEMBLY_NUMBER_FOR

Category **Numbering**

This variable defines wether the part number of the assembly number will be replaced in the output (drawings, reports, NC files) by the assembly number.

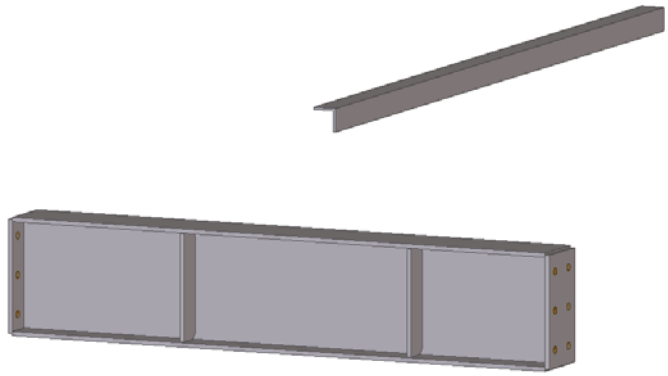


In Tekla Structures the part number will be used while numbering, however this is not visible in the output.

By default, the variable is empty, so nothing his entered.

Example STEEL

The following example shows a beam with some welded stiffeners and a single L profile by which the assembly consists of only one (main) part:



EMPTY

TEKLA STRUCTURES ASSEMBLY PART LIST
Model:

Page: 1
Project: project nummer
Date: 16.10.2007
Time: 14:32:21

Assembly	Part	Number	Profile	Length(mm)	Area(m2)	Weight(kg)
L1		1	IPE300		2.63	96.8
	Pr1	1	IPE300	1944	2.26	83.8
	P1	2	STRIP10*150	290	0.10	3.4
	P2	4	STRIP10*70	275	0.04	1.5
W1		1	L60/6		0.46	10.4
	Pr2	1	L60/6	1900	0.46	10.4
				Total (kg):		107.2

LOOSE_PART

```
-------------------------------------------------------------------------------
TEKLA STRUCTURES ASSEMBLY PART LIST                          Page:    1
Model:                                                       Project: project nummer
                                                             Date:    16.10.2007
                                                             Time:    14:32:21

-------------------------------------------------------------------------------
Assembly     Part      Number Profile          Length(mm)  Area(m2)   Weight(kg)
===============================================================================
L1                     1      IPE300                       2.63         96.8
-------------------------------------------------------------------------------
             Pr1       1      IPE300             1944      2.26         83.8
             P1        2      STRIP10*150         290      0.10          3.4
             P2        4      STRIP10*70          275      0.04          1.5
===============================================================================
W1                     1      L60/6                        0.46         10.4
-------------------------------------------------------------------------------
             W1        1      L60/6              1900      0.46         10.4
===============================================================================
                                                  Total (kg):          107.2
-------------------------------------------------------------------------------
```

Conclusion:

For L1 this doesn't matter regarding to the option EMPTY.

For W3 the part number will be left out in the output.

MAIN_PART

```
-------------------------------------------------------------------------------
TEKLA STRUCTURES ASSEMBLY PART LIST                          Page:    1
Model:                                                       Project: project nummer
                                                             Date:    16.10.2007
                                                             Time:    14:32:21

-------------------------------------------------------------------------------
Assembly     Part      Number Profile          Length(mm)  Area(m2)   Weight(kg)
===============================================================================
L1                     1      IPE300                       2.63         96.8
-------------------------------------------------------------------------------
             L1        1      IPE300             1944      2.26         83.8
             P1        2      STRIP10*150         290      0.10          3.4
             P2        4      STRIP10*70          275      0.04          1.5
===============================================================================
W1                     1      L60/6                        0.46         10.4
-------------------------------------------------------------------------------
             W1        1      L60/6              1900      0.46         10.4
===============================================================================
                                                  Total (kg):          107.2
-------------------------------------------------------------------------------
```

Conclusion:

For L1 the part number will be left out in the output.

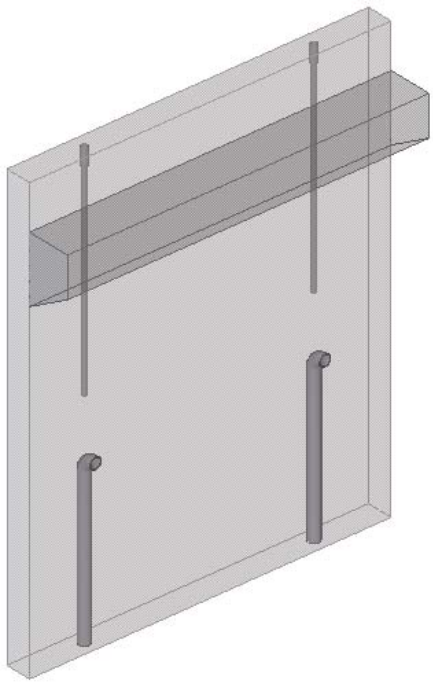
For W3 nothing changes regarding to the option LOOSE_PART in the output.



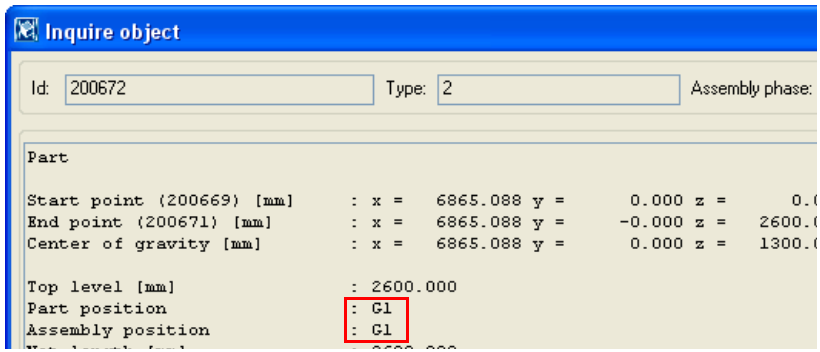
Do not use the same prefix for parts and assemblies when set the variabele to XS_USE_ASSEMBLY_NUMBER_FOR=MAIN_PART.

Example CONCRETE

For concrete counts the same as for steel, so check the examples above. Concrete uses LOOSE_PART by default for numbering.



This means that the gain only gets an assembly number:



XS_USE_SMALLER_GUSSET_PLATE

Category **Components**

Set to TRUE to minimize the size of rectangular gusset plates created by gusset connections.

You can create smaller gusset plates by using a single bracing and secondary bolts dimensioned to the middle of the secondary part.

Tekla Strutures generates a triangular gusset plate when the primary part is located between diagonals.

XS_USE_TUBE_INNER_LENGTH_IN_DIMENSIONING

Category **Dimensioning: Parts**

Set to TRUE to have Tekla Structures dimension the overall length of tube profiles along the inner surface instead of outer surface.

XS_VIEW

Category **Model View**

Use to define the default window size of views and their initial position on the screen.

XS_VIEW_POSITION_X=0

XS_VIEW_POSITION_Y=0

XS_VIEW_WIDTH=650

XS_VIEW_HEIGHT=500

Similar to XS_BASICVIEW... variables.

XS_VIEW_DIM...

Category **Model View**

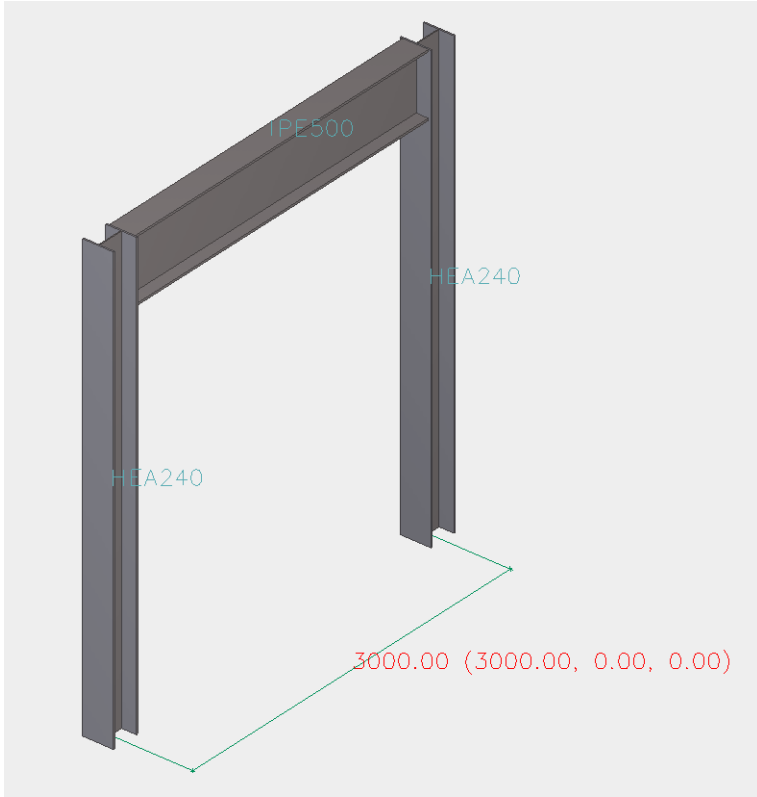
You can change the color of part marks, dimensions en dimensions texts in rendered model views.

How to To change a color:

1. Go to **Tools > Advanced options > Model view**.
2. Change the variable as below. Enter a RGB (Red Green Blue) value. The scale is from 0 to 1. For example, to change the color to red, enter "1 0 0".

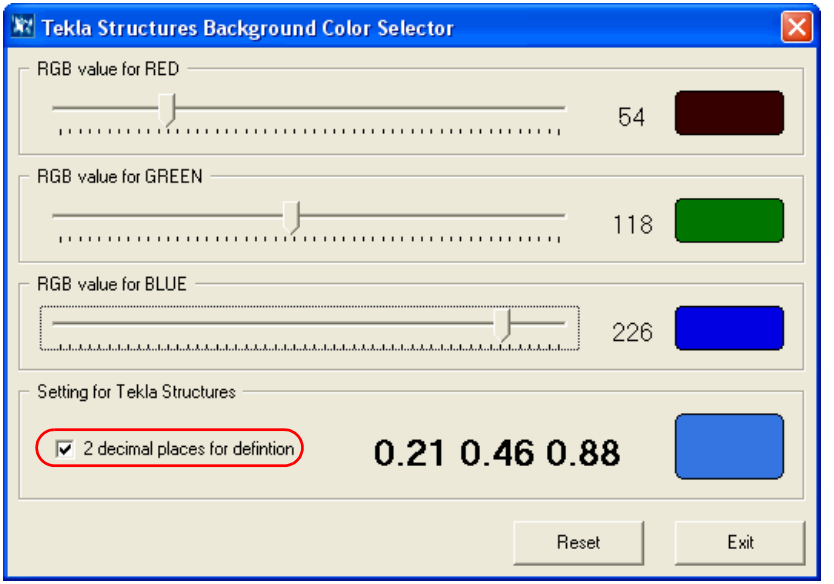
Variable	Use for
XS_VIEW_DIM_LINE_COLOR	Changing the dimension color
XS_VIEW_DIM_TEXT_COLOR	Changing the dimension text color
XS_VIEW_PART_LABEL_COLOR	Changing the part mark color

Example



Finding the right RGB values

Each color has an accompanying RGB value. Tekla has developed a tool to find the RGB values. The tool is named **TS_Background_Color.exe** and can be downloaded from the Construsoft Xtranet website.



RGB values can be entered with 2 decimals.

XS_VIEW_PART_LABEL_COLOR

Category **Model View**

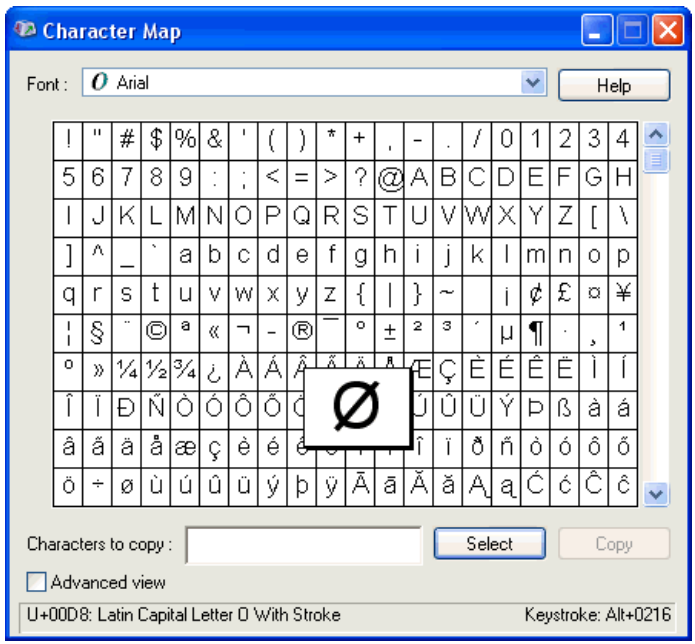
See also `XS_VIEW_DIM_...`

3.4 Using characters in Tekla Structures

Text can be added to Tekla Structures drawings. This text can possibly contain characters like Ø and €. How to add these type of characters?

Go to **Start** > **Programs** > **Accessories** > **System tools** > **Character Map**.

The following dialog box shall appear:



In the ‘Font’-list box the font can be selected which you also use in Tekla Structures. Next, all available characters for this font are displayed. When selecting a character, a code shows up in the right bottom corner of the dialog box: ‘Keystroke: <Alt + code>’

Character	Keystroke	Character	Keystroke
ø	Alt + 155	€	Alt + 0128
Ø	Alt + 157	°	Alt + 248
Ø	Alt + 0216		

3.5 Classifier

Topics

This chapter is divided into following sections:

Classifier principle (p.62)

“Phase” properties (p.65)

“Profile” properties (p.68)

“Material” properties (p.72)

"Supplier" properties (p.76)

Exercise (p.84)

Classifier for assembly- and cast unit drawings (p.87)

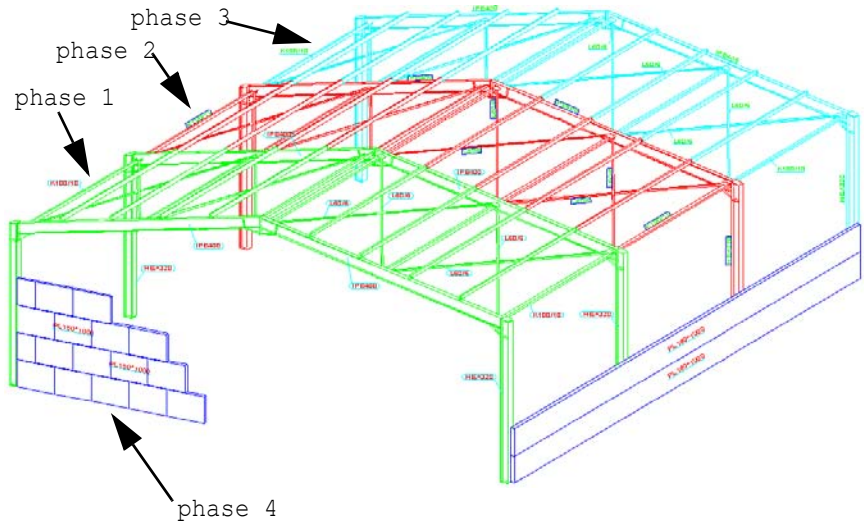
Classifier principle

The classifier enables you to create different drawing styles.You can use the classifier to control the drawing properties for different types of model objects, for example:

- Different phases in separate colors
- Different parts in separate colors
- Different material grades in separate colors

Except colors, you can also apply this to the position of part marks for each separate profile, phase and material grade.

The picture below shows an example how to use the classifier. The phases appear in separate colors. Also the position of the part marks differs for each phase.

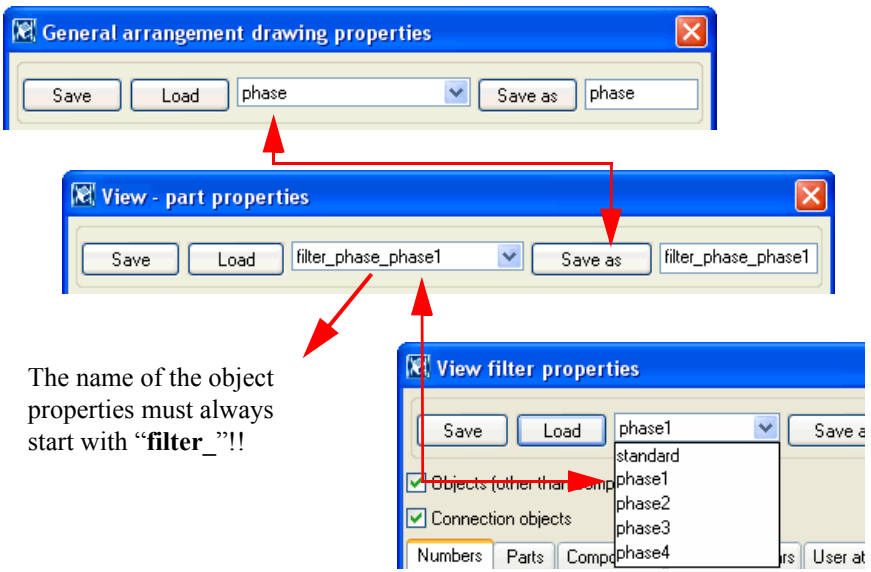


The classifier works by using a specific naming convention for saved properties. We create them in steps. Once created, you can use them, of course, again and again. It enables you to create drawings containing a specific style every time!

The classifier uses the following properties:

- general arrangement drawing properties
- view filter properties
- object properties, for example; part, part mark, bolt and bolt mark

The classifier works by using a specific naming convention for saved properties (see picture below).



The name of the object properties must always start with "**filter_**"!!

Make sure that the names that you use for the properties match to each other!

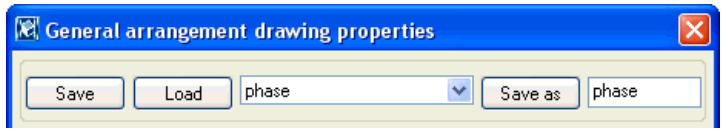
On the basis of several examples we will explain how the classifier works.

“Phase” properties

You now learn how to create a drawing in which different phases appear in different colors.

Define drawing properties

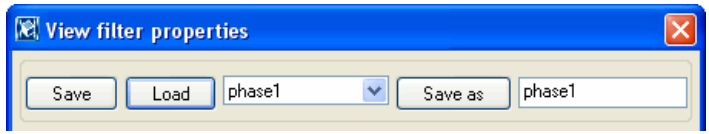
1. Go to **> Properties > General arrangement drawing.** Create a general arrangement drawing property, enter the name *“phase”* in the text box and click <**Save as**>.



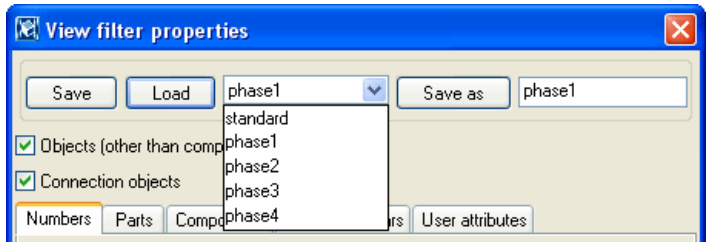
Creating view filters

2. Create view filters to define the specific group of model objects, whose appearance you want to define. In this example every phase appears in a corresponding color on a drawing. So for each phase we create a view filter, one view filter for phase 1, one view filter for phase 2, etc.

3. Open any drawing and double-click on a view. Click on the button **Filter.** In the dialog box **View filter properties** go to the tab **Numbers** and define the setting so that phase 1 is filtered (see picture below). Enter the name ***phase 1*** in the text box and click <**Save as**>.



4. Repeat the steps for all phases you want to define. In this example: ***phase 2***, ***phase 3*** and ***phase 4***. Click <**Cancel**> to close the dialog box!



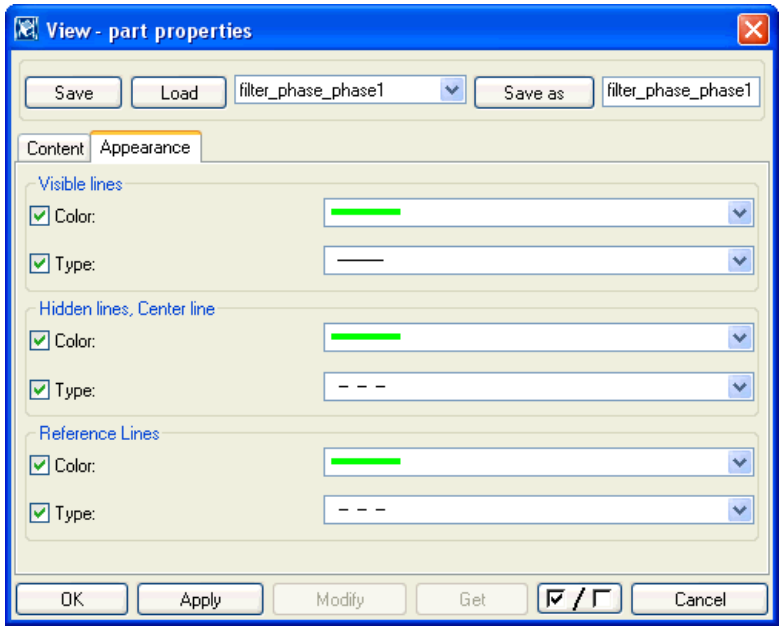


Do not click <**Apply**>, as this will apply the current filter to the entire drawing and the Classifier will not work! Just click <**Cancel>.**

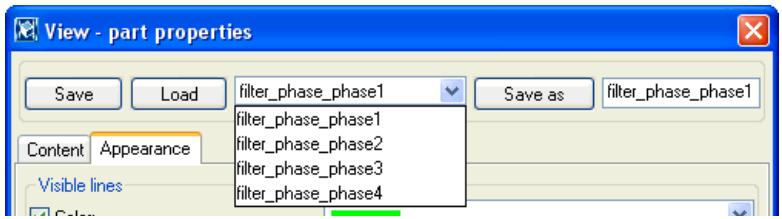
Always use view filters when you use the Drawing Classifier. The Drawing Classifier does not recognize select filters!

Creating drawing object property files

5. For each phase you can define the corresponding color who appears on a drawing. In this example phase 1, 2, 3 and 4. Double-click on a view and click on the button **Part.** Define the setting for phase 1. Enter the name ***filter_phase_phase1*** in the text box and click <**Save as**>. (see picture below).

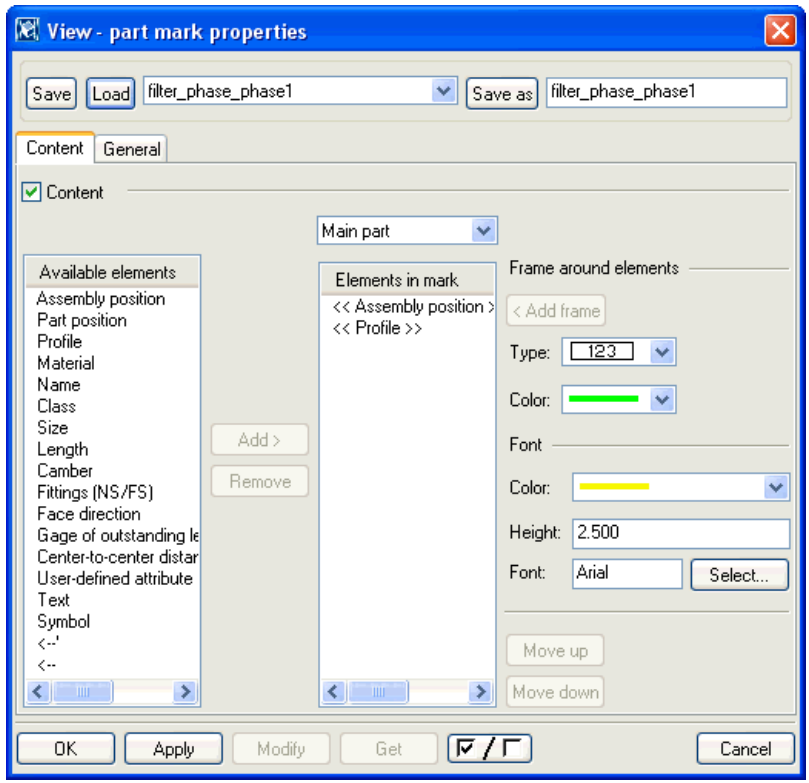


6. Repeat these steps for all phases you want to define. In this example: ***filter_phase_phase2***, ***filter_phase_phase3*** and ***filter_phase_phase4***. Click <**Cancel**> to close the dialog box!

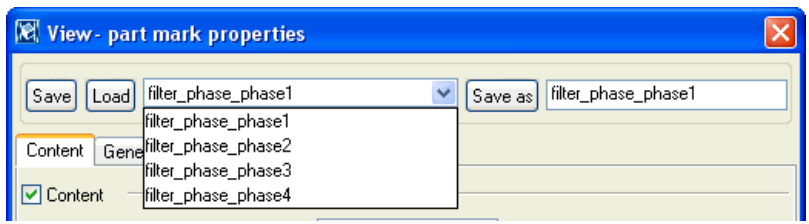


7. For all phases a specific color is defined. Of course it is possible to define part mark property files for each phase, or to change the appearance. Follow the steps we used for defining the part colors for each phase.

8. So now you can define the appearance and position of part mark property files for each phase. In this example phase 1, 2, 3 and 4. Double-click on a view and click on the button **Part mark.** Define the setting for phase 1. Enter the name ***filter_phase_phase1*** in the text box and click <**Save as**>. (see picture below).

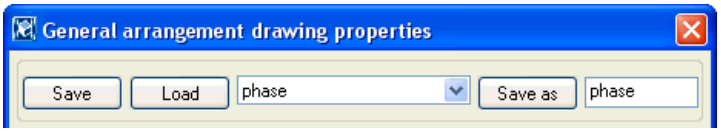


9. Repeat the steps for all phases you want to define. In this example: ***filter_phase_phase2***, ***filter_phase_phase3*** and ***filter_phase_phase4***. Click <**Cancel**> to close the dialog box!

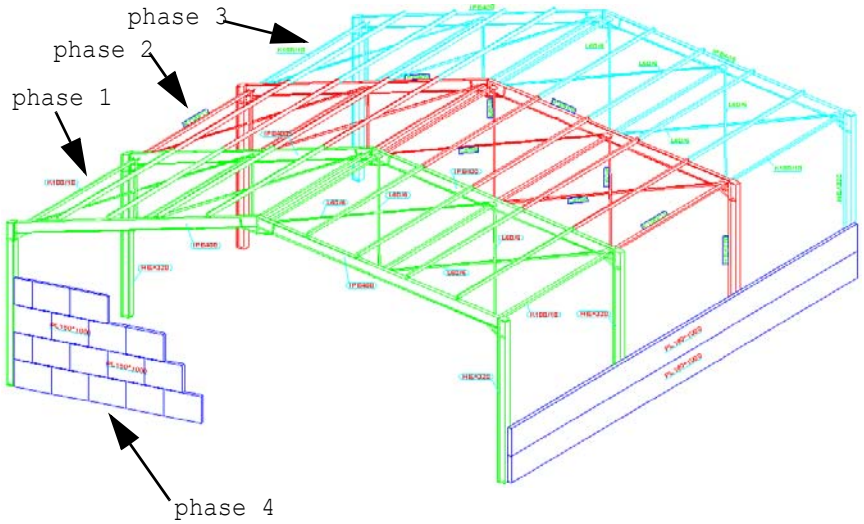


Creating drawings

10. Go to **> Properties > General arrangement drawing...** Click the down arrow in the Load text box and select the ***phase*** drawing properties you created. Click <**Load**> (see picture below). Now create the general arrangement drawing.



You will see that a drawing is created in which different phases appear in different colors.

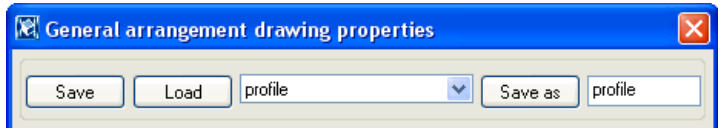


“Profile” properties

You now learn how to create a drawing in which different parts for each part name appear in different colors. So all columns in a specific color, all beams in a specific color.

Define drawing properties

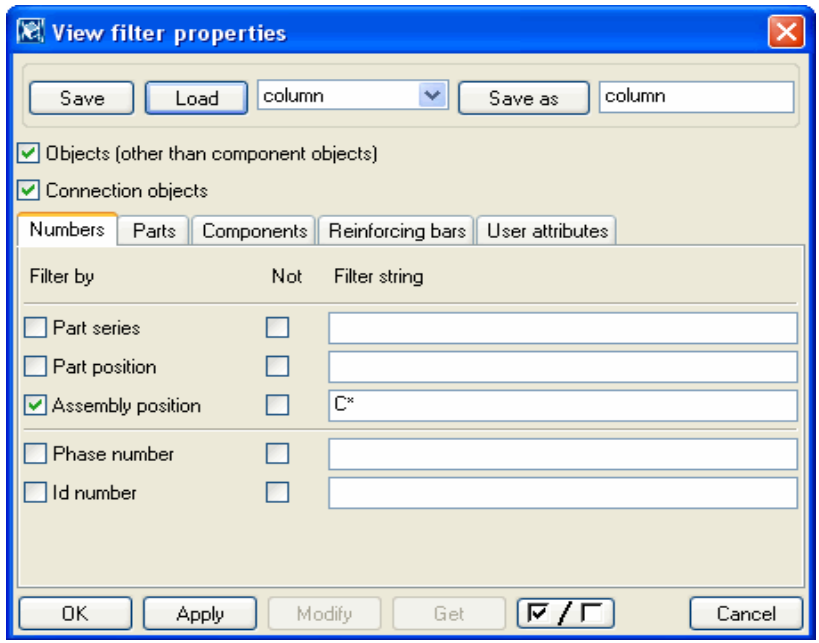
1. Go to **> Properties > General arrangement drawing.** Create a general arrangement drawing property, enter the name “profile” in the text box and click <**Save as**>.



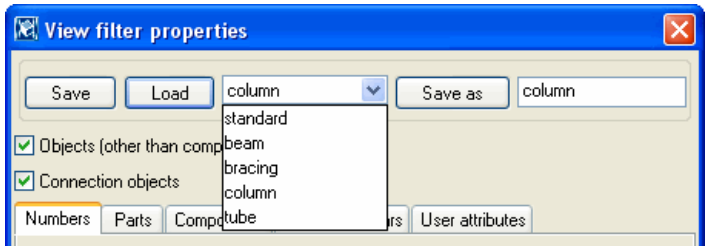
Creating view filters

2. Create view filters to define the specific group of model objects, whose appearance you want to define. In this example every type of profile appears in a corresponding color on a drawing. So for each profile we create a view filter, one view filter for columns, one view filter for beams, etc.

3. Open any drawing and double-click on a view. Click on the button **Filter.** In the dialog box **View filter properties** go to the tab **Numbers,** and in the filter string text box next to Filter by Assembly position enter “**C***”. (for beams B*). In the text box next to the <**Save as**> button, enter “column” and click <**Save as**>. (see picture below)

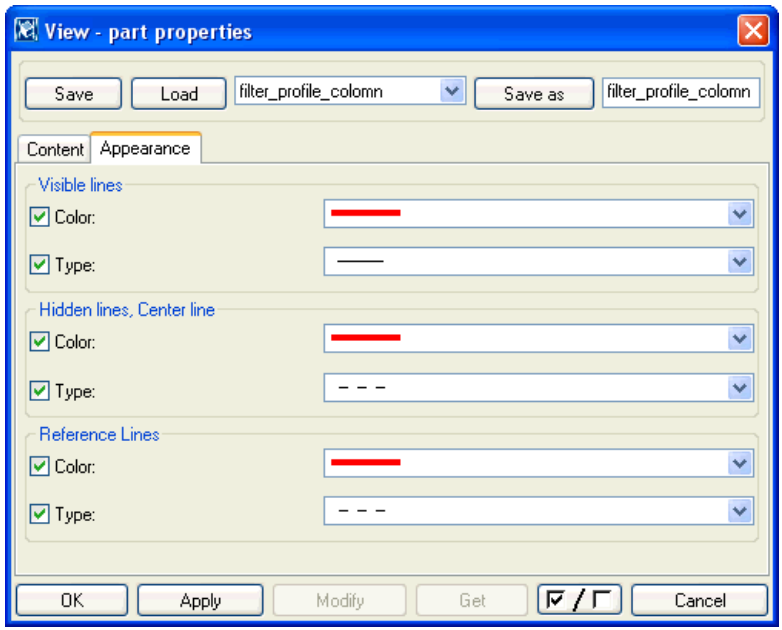


4. Repeat the steps for all profiles you want to define. In this example: ***beam***, ***bracing*** and ***tube***. Click <**Cancel**> to close the dialog box!

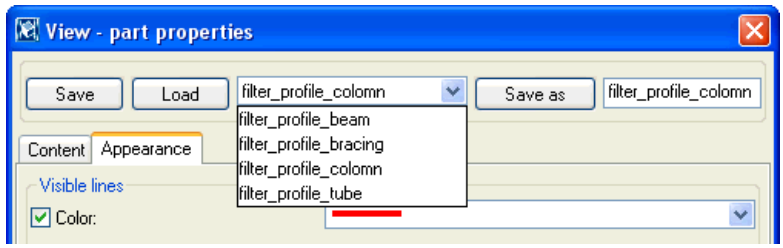


Creating drawing object property files

5. For each profile you can define the corresponding color who appears on a drawing. In this example column, beam, tube and bracing. Double-click on a view and click on the button **Part.** Define the setting for columns. Enter the name ***filter_profile_column*** in the text box and click <**Save as**>. (see picture below).

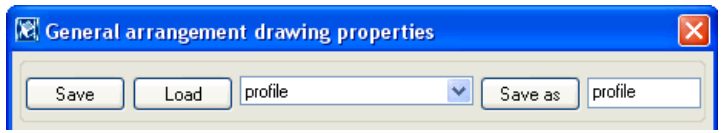


6. Repeat the steps for all profiles you want to define. In this example: ***filter_profile_tube***, ***filter_profile_bracing*** and ***filter_profile_beam***. Click <**Cancel**> to close the dialog box!

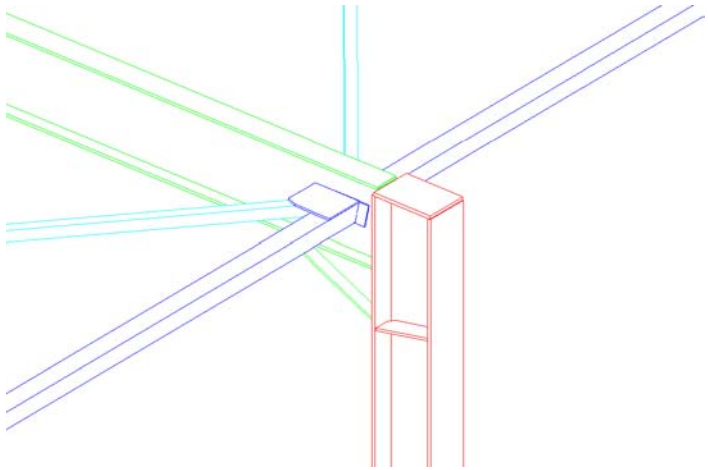


Creating drawings

7. Go to **> Properties > General arrangement drawing...** Click the down arrow in the <**Load>** text box and select the ***profile*** drawing properties you created. Click <**Load>** (see picture below). Now create the general arrangement drawing.



You will see that a drawing is created in which different types of profiles appear in different colors.

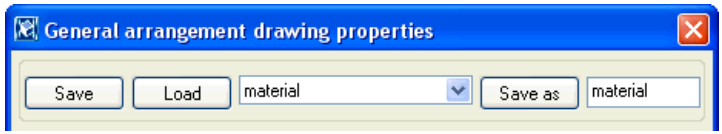


“Material” properties

You now learn how to create a drawing in which different types of material grades appear in different colors.

Define drawing properties

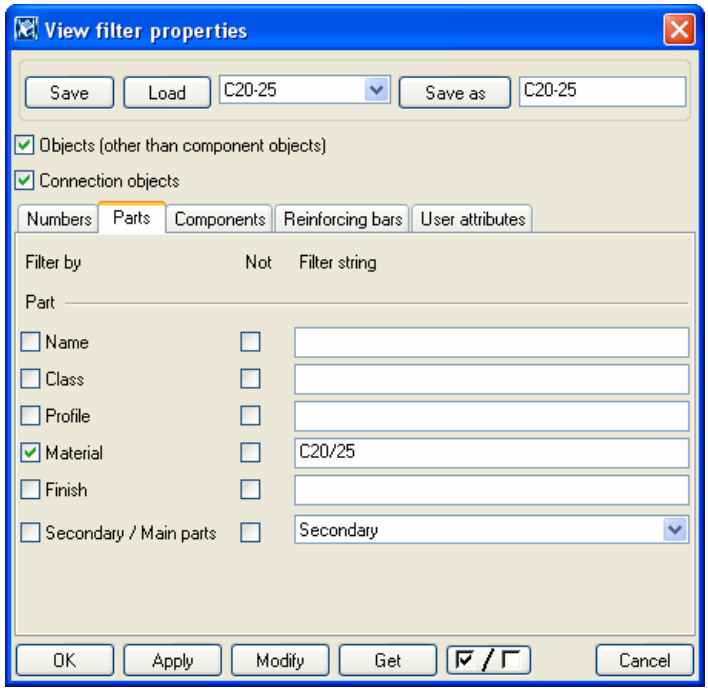
1. Go to **> Properties > General arrangement drawing.** Create a general arrangement drawing property, enter the name “material” in the text box and click <**Save as**>.



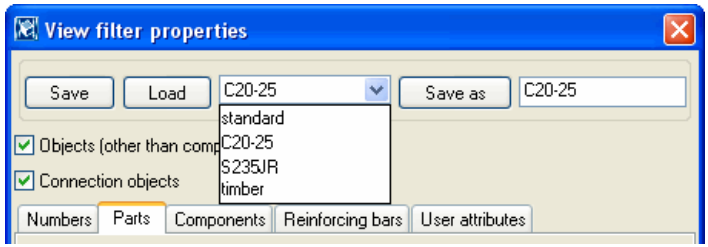
Creating view filters

2. Create view filters to define the specific group of model objects, whose appearance you want to define. In this example every material grade appears in a corresponding color on a drawing. So for each grade we create a view filter.

3. Open any drawing and double-click on a view. Click on the button **Filter.** In the dialog box **View filter properties** go to the tab **Parts,** and in the filter string text box next to Part Material enter "**C20-25**". In the text box next to the <**Save as**> button, enter "C20-25" and click <**Save as**>. (see picture below):

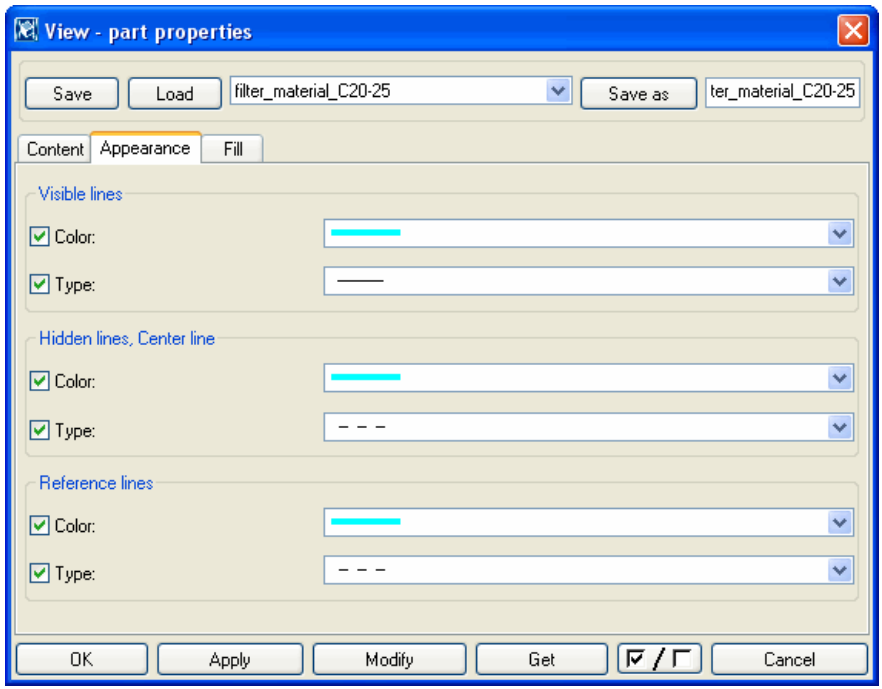


4. Repeat the steps for all material grades you want to define. In this example: ***timber***, ***S235JR***. Click <**Cancel**> to close the dialog box!

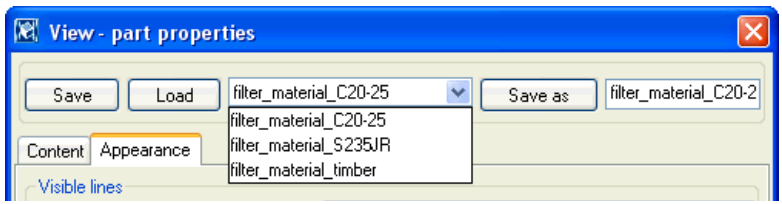


Creating drawing object property files

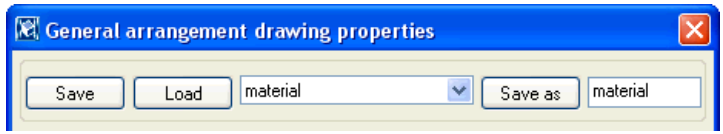
5. For each material grade you can define the corresponding color who appears on a drawing. In this example C20-25, timber and S235JR. Double-click on a view and click on the button **Part.** Define the setting for material grade C20-25. Enter the name ***filter_material_C20-25*** in the text box and click <**Save as**>. (see picture below).



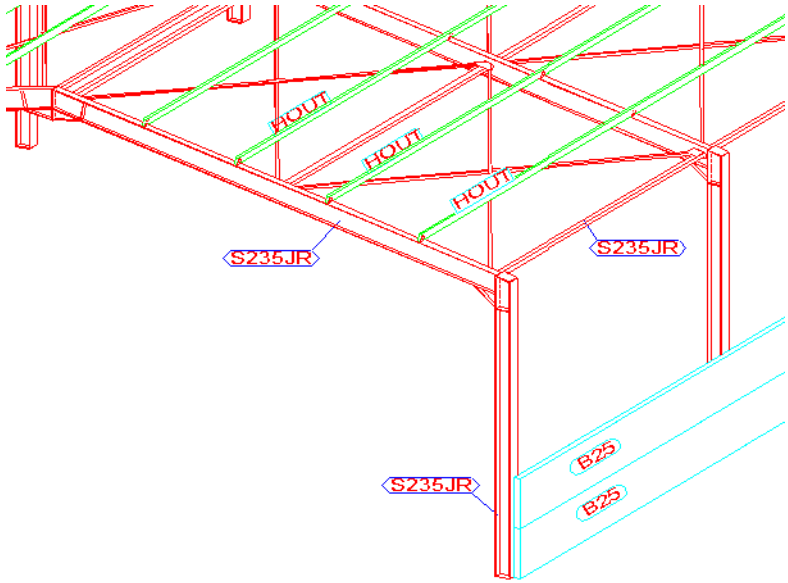
6. Repeat the steps for all material grades you want to define. In this example: ***filter_material_timber*** and ***filter_material_S235JR***. Click <**Cancel**> to close the dialog box!



7. Go to **> Properties > General arrangement drawing...** Click the down arrow in the <**Load>** text box and select the ***material*** drawing properties you created. Click <**Load>** (see picture below). Now create the general arrangement drawing.

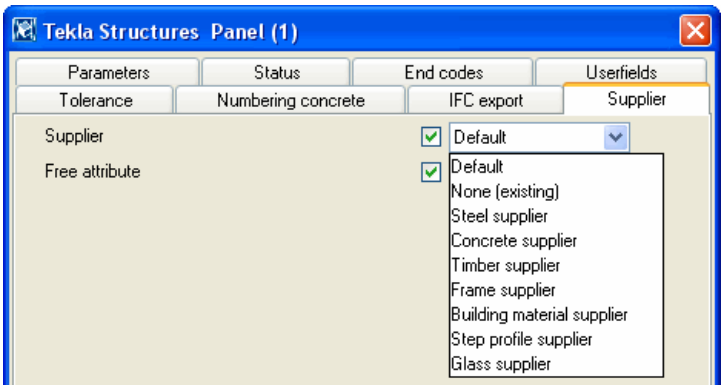


You will see that a drawing is created in which different types of material grades appear in different colors.



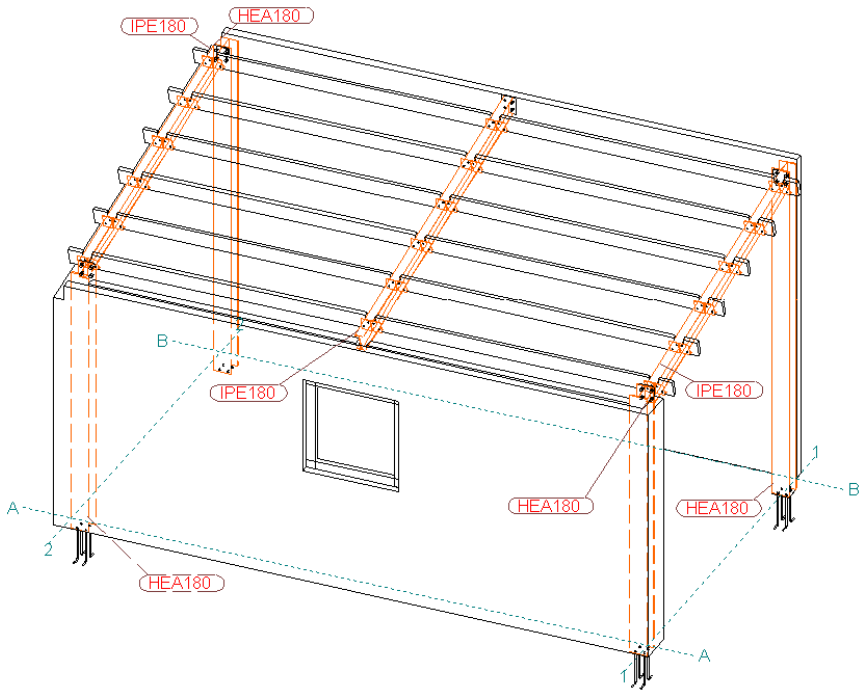
"Supplier" properties

You now learn how to create a GA drawing in which user-defined attributes are used. In this case the user-defined attributes *Steel supplier*.



All parts with the user-defined attribute *Steel supplier* appear on the drawing in Orange, also the **Hidden lines** and the **Own hidden lines** will be displayed. Besides, all marks are displayed.

The other parts will appear on the drawing in White. The Hidden lines, the Own hidden lines and the marks will not be displayed:

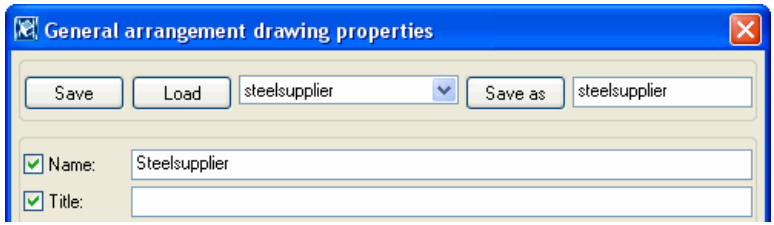


Open the model `Supplier`, you can download the model from the Construsoft Xtranet.

This model contains parts in which the user-defined attributes on the tab **Supplier** are set to, for example, *Steel supplier* or *Glass supplier*.

Define drawing properties

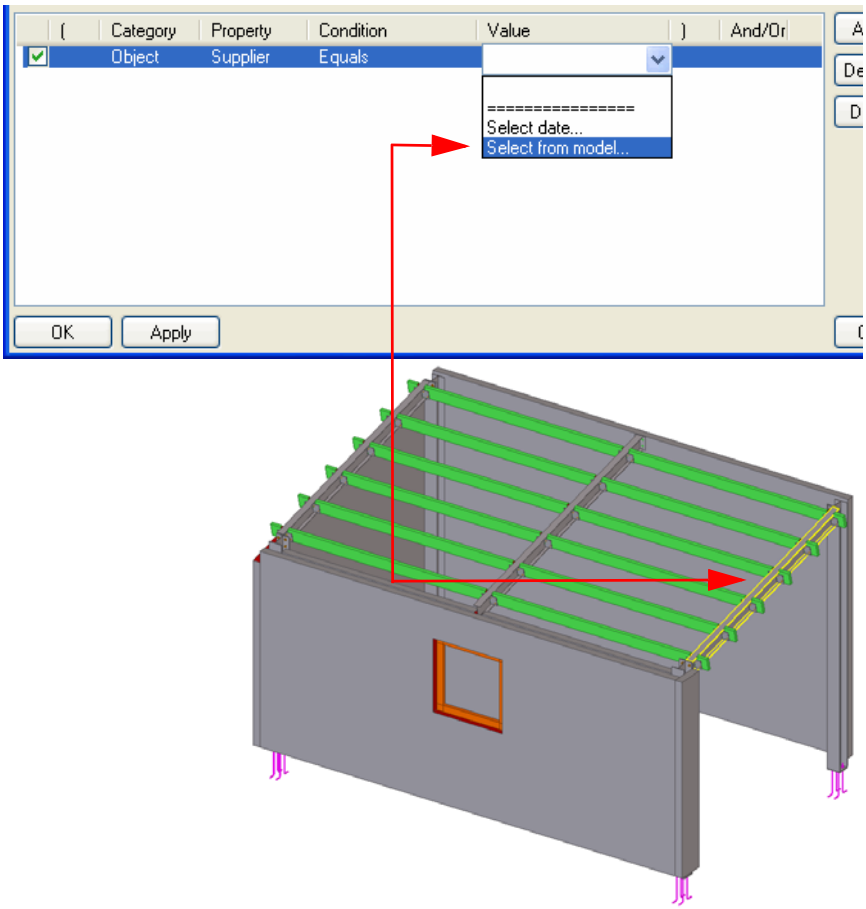
1. Go to **> Properties > General arrangement drawing.** Create a general arrangement drawing property, enter the name “steelsupplier” in the text box and click <**Save as**>.



Creating view filters

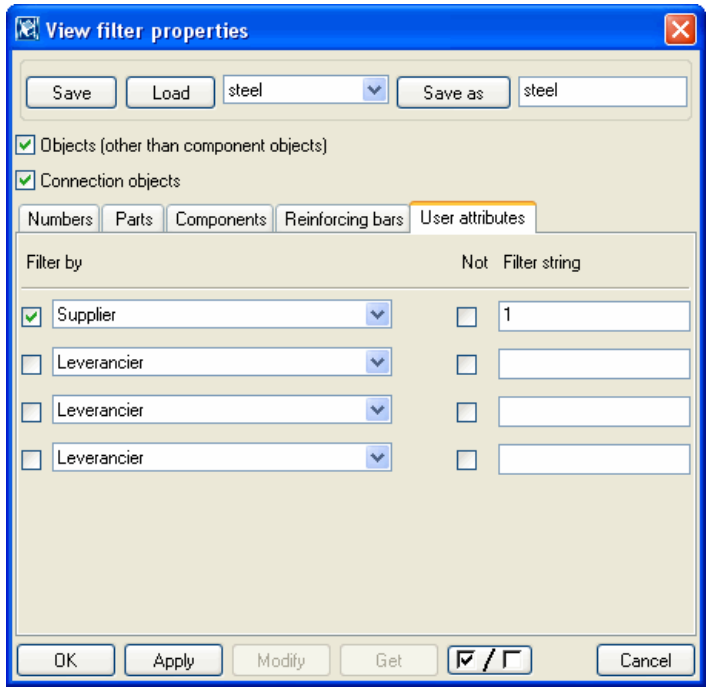
First we have to find out in the model what we need to enter in the view filter in the drawing:

2. The view filters define the specific group of model objects, whose appearance you want to define. In this example all parts containing the user-defined attribute *Steel supplier* appear on a drawing in a specific color.
3. Click **Setup** > **Select filter...** to open the dialog box **Object group- select filter**
4. To find out easily what to enter for the filter properties, go to the column **Category** and select **Object**
5. In the column **Property** select the option **Supplier** in the list box
6. In the column **Condition** select the option **Equals**
7. In the column **Value** select the option **Select from model...** to select the concerning part in the model.

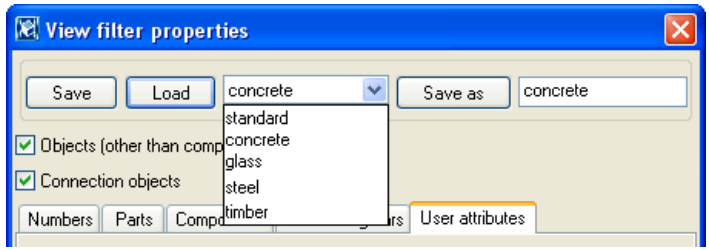


8. Click on a steel part, in the column **Value** the digit **1** appears. You can filter on this.

9. Open any drawing and double-click on a view. Click on the button **Filter.** In the dialog box **View filter properties** go to the tab **User attributes** and define the setting. Save this setting ***steel*** in <**Save as**> as follows:



10. Repeat the steps if necessary for the other suppliers you want to define. Click <**Cancel**> to close the dialog box!





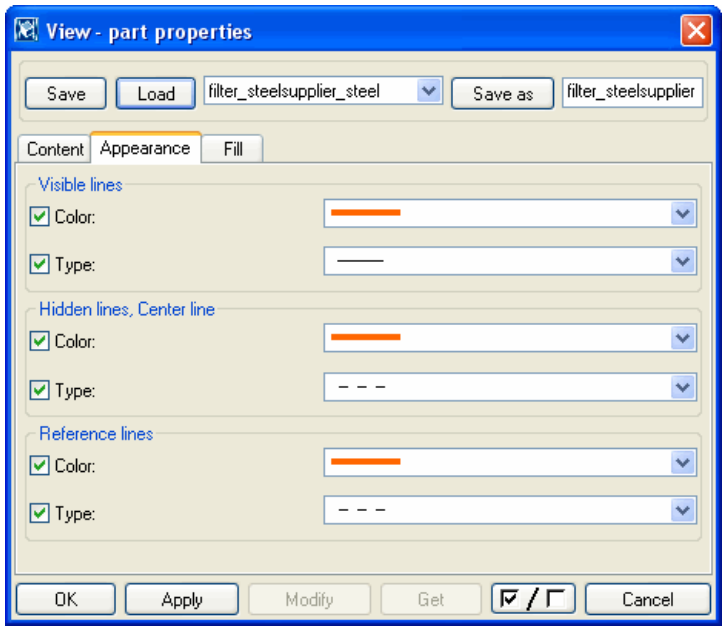
Do not click <**Apply**>, as this will apply the current filter to the entire drawing and the Classifier will not work!



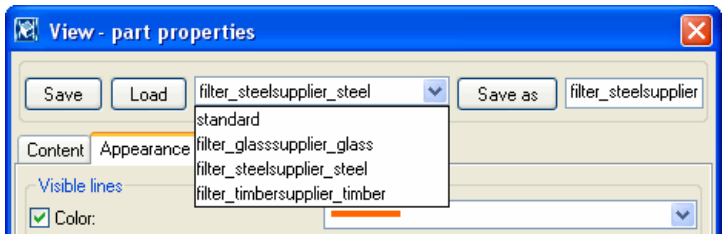
Always use view filters when you use the Drawing Classifier. The Drawing Classifier does not recognize select filters!

Creating drawing object property files

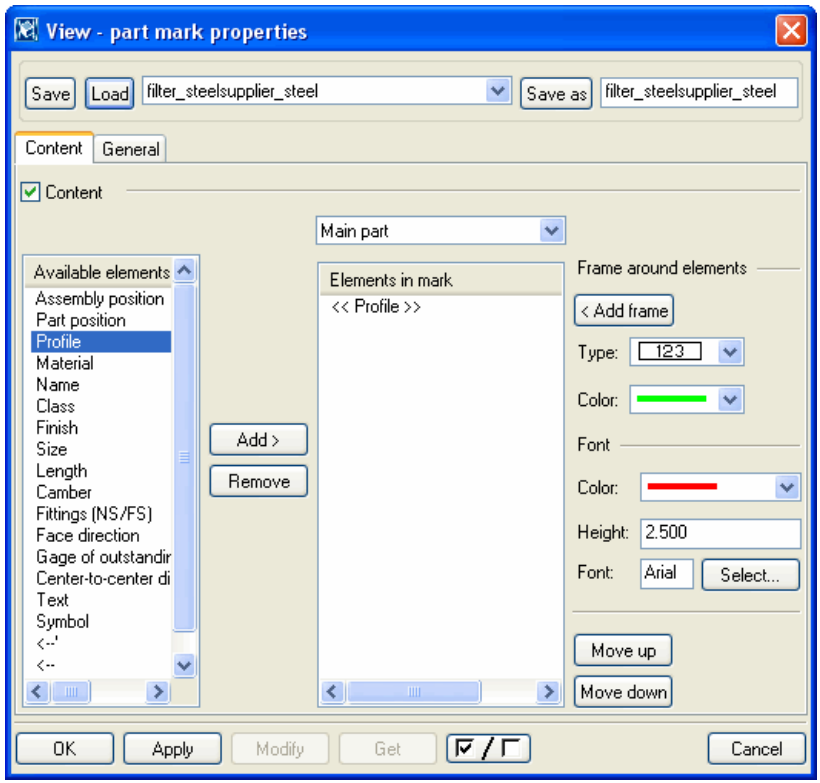
11. For user-defined attribute *Steel supplier* you can define the corresponding color who appears on a drawing. Double-click on a view and click on the button **Part.** Define the setting for steel. Enter the name ***filter_steelsupplier_steel*** in the text box and click <**Save as**>. (see picture below).



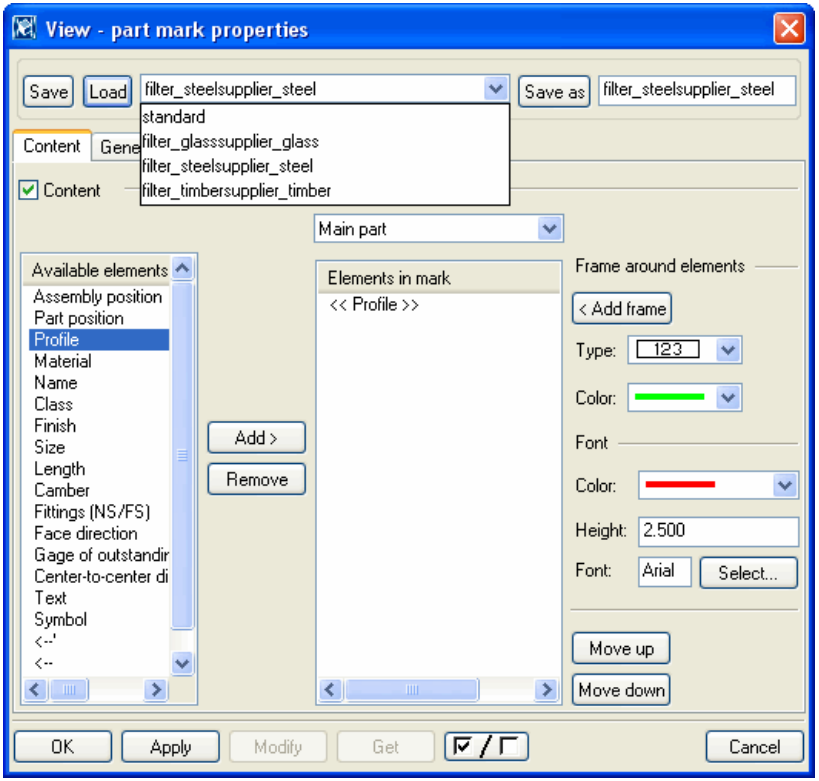
12. Repeat the steps for the user-defined attribute values you want to define. In this example: ***filter_timbersupplier_timber*** and ***filter_glasssupplier_glass***. Click <**Cancel**> to close the dialog box!



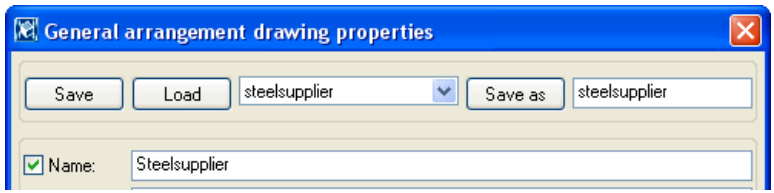
13. You can also define the part marks. Double-click on a view and click on the button **Part marks.** Define the setting for steel. Enter the name ***filter_steelsupplier_steel*** in the text box and click <**Save as**>. (see picture below).



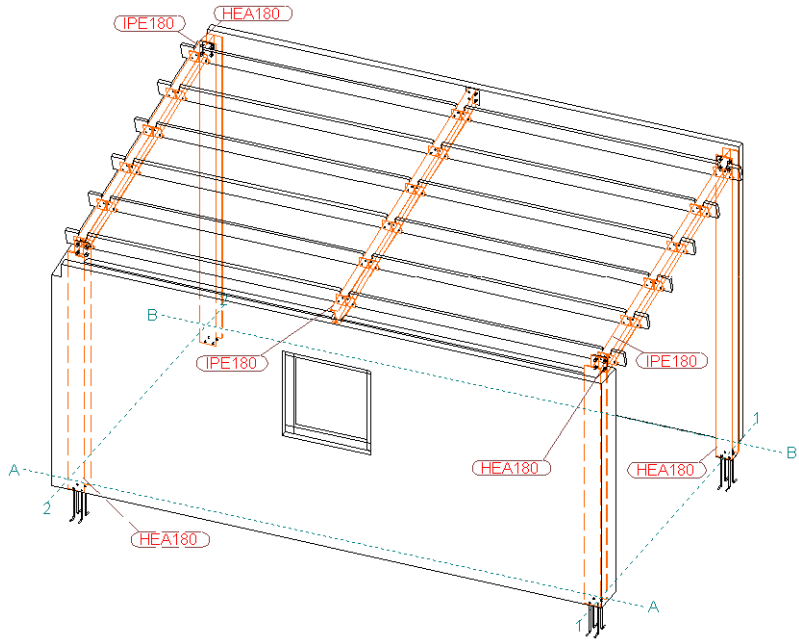
14. Repeat the steps for other user-defined attribute values you want to define. Click <**Cancel**> to close the dialog box!


View - part mark properties
Save
Load
filter_steelsupplier_steel
Save as
filter_steelsupplier_steel
standard
filter_glasssupplier_glass
filter_steelsupplier_steel
filter_timbersupplier_timber
Content
Gene
Content
Main part
Available elements
Assembly position
Part position
Profile
Material
Name
Class
Finish
Size
Length
Camber
Fittings (NS/FS)
Face direction
Gage of outstandir
Center-to-center di
Text
Symbol
<--'
<--
Add >
Remove
Elements in mark
<< Profile >>
Frame around elements
< Add frame
Type:
123
Color:
Font
Color:
Height:
2.500
Font:
Arial
Select...
Move up
Move down
OK
Apply
Modify
Get
Cancel

15. Go to **> Properties > General arrangement drawing...** Click the down arrow in the <**Load>** text box and select the ***Steelsupplier*** drawing properties you created. Click <**Load>** (see picture below). Now create the general arrangement drawing.



You will see that all parts with the user-defined attribute *Steel supplier* appear on the drawing in Orange, also the Hidden lines, the Own hidden lines and the marks are displayed.

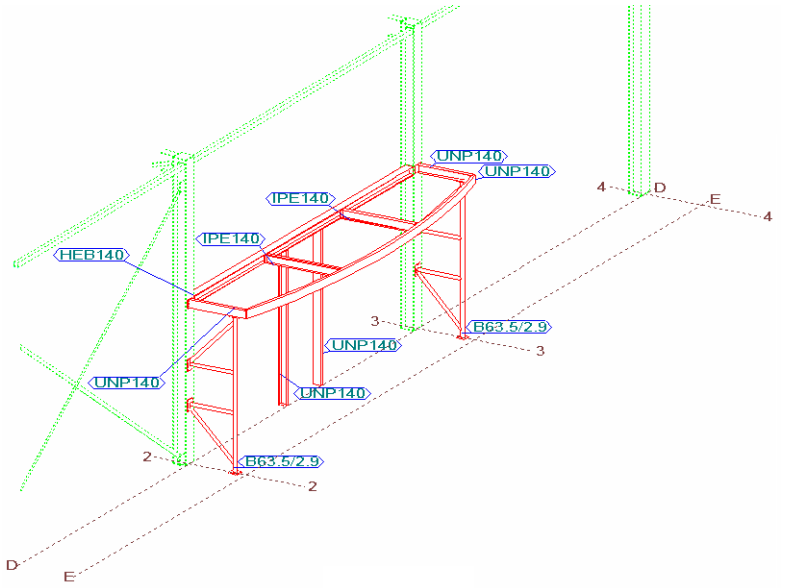


Exercise

It may occur that an existing project needs some additions, for example an awning. On a GA-drawing the following has to be set:

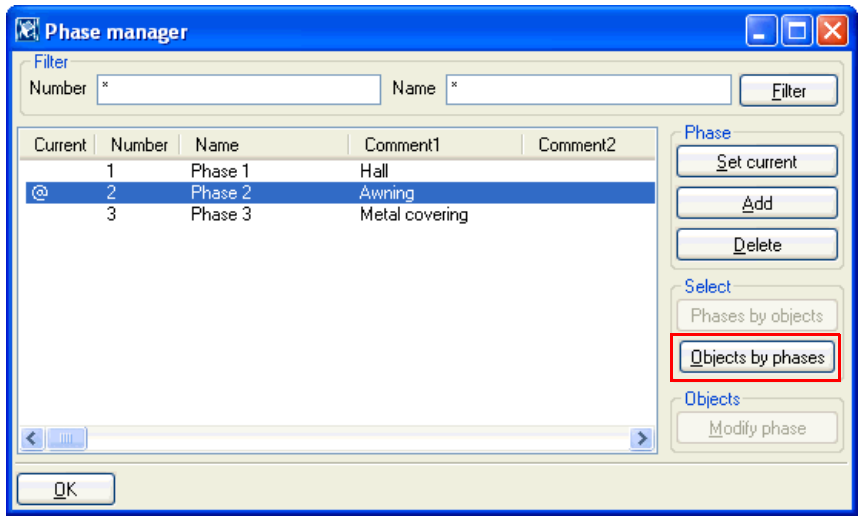
- Only for the awning, part marks have to be added, not for the existing structure
- The parts of the existing structure are displayed as dotted lines, the part lines of the awning are displayed as full lines.

For this exercise you can make use of model *Hal_1*. Create a GA drawing and make use of a classifier setting:



Step plan

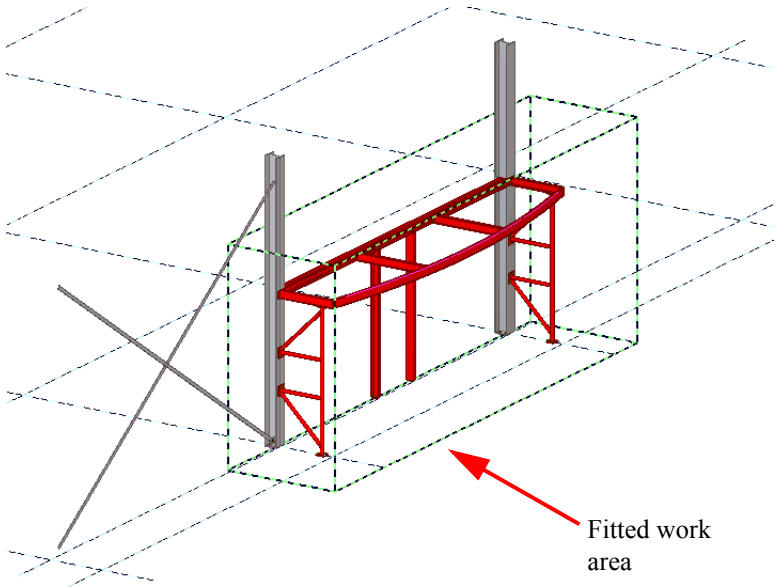
1. Make sure the awning is in a unique phase in the model (e.g. phase2)
2. Open the phase manager



3. Select phase 2 and click <*Objects by phase*>, all parts who belong to the awning will highlight in the model view

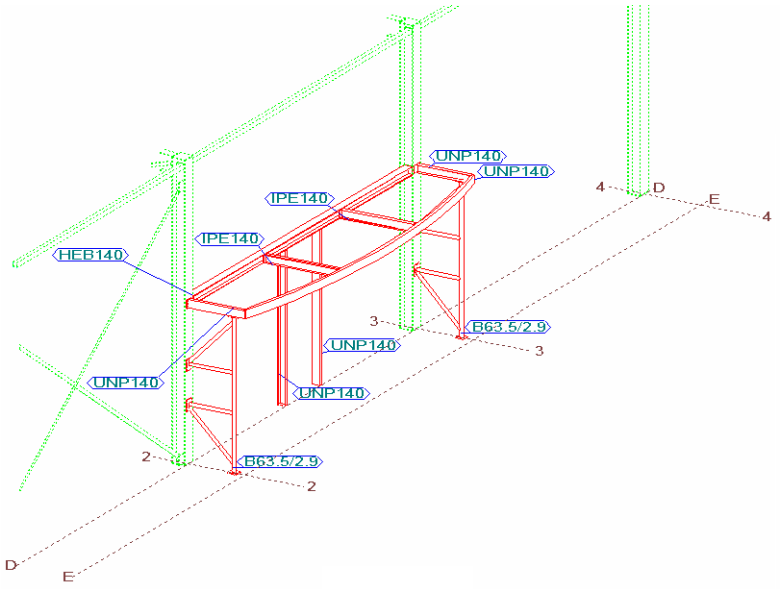


4. Click the icon ***Fit work area by parts in selected views*** or in the pull-down menu go to ***View*** > ***Fit work area,*** The work area is now sized to the awning:



5. In the GA drawing properties, adjust the setting "*phase*"

6. Now create a GA drawing by making use of the setting “phase”



If necessary, you can resize the view in the drawing.

Classifier for assembly- and cast unit drawings

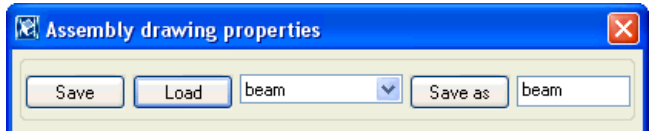
You can use the Drawing Classifier not only for general arrangement drawings, but also for assembly- and cast unit drawings.

You now learn how to create an assembly drawing in which several welded parts appear in different colors. These parts also appear with different part marks.

On the basis of an examples we will explain how the classifier works

Define drawing properties

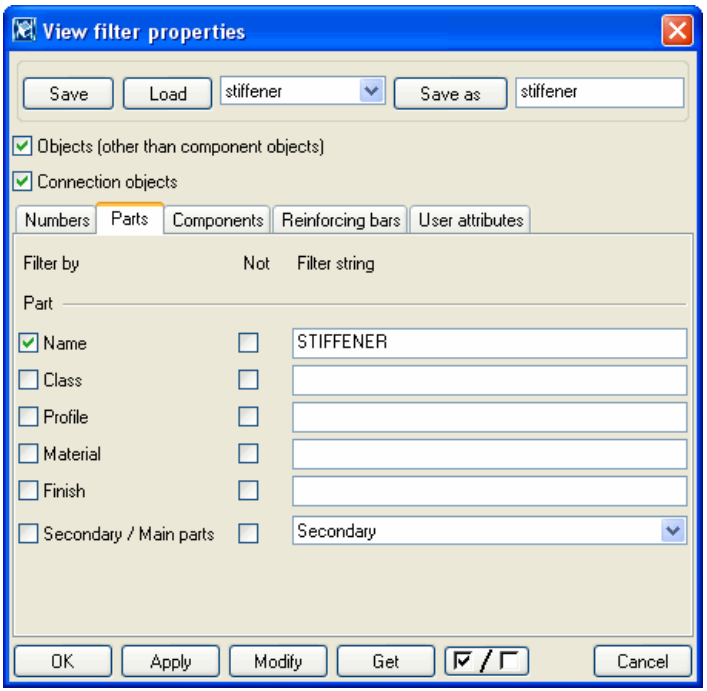
1. Go to **> Properties > Assembly drawing.** Create an assembly drawing property, enter the name “beam” in the text box and click <**Save as**>.



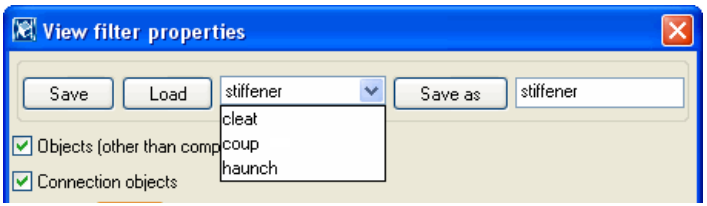
Creating view filters

2. In this example all different welded parts appear in a corresponding color on the assembly drawing. So for each welded part we create a view filter. You must create the filter for assembly drawings in the general arrangement drawing filter!

3. Open any drawing and double-click on a view. Click on the button **Filter.** In the dialog box **View filter properties** go to the tab **Parts** and define the setting. In the text box next to the <**Save as**> button, enter ***stiffener*** and click <**Save as**>, see picture below:

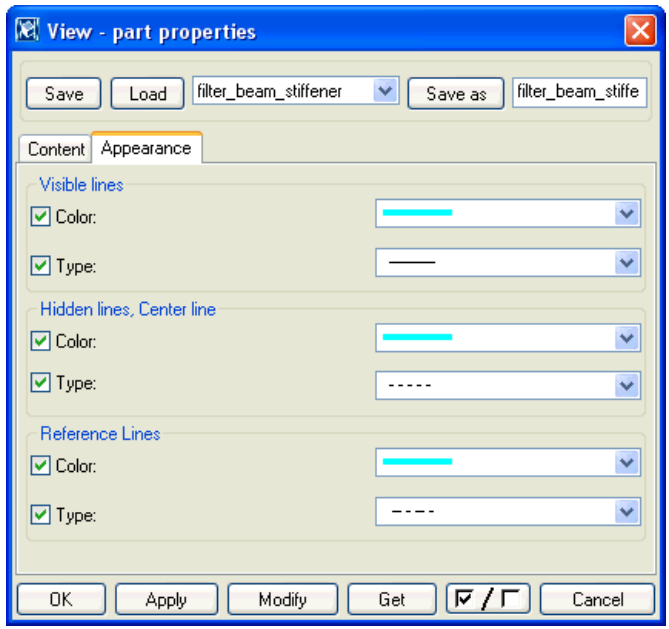


4. Repeat the steps for all welded parts you want to define. In this example: ***cleat*** and ***haunch***. Click <**Cancel**> to close the dialog box!

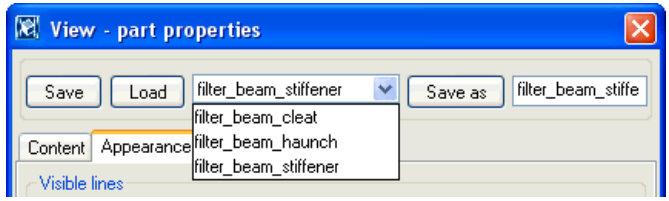


Creating drawing object property files

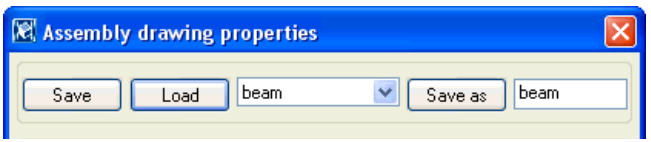
5. For each welded part you can define the corresponding color on the assembly drawing. In this example stiffener, cleat and haunch. Go to **> Properties > Assembly drawing... > Part.** Define the setting for stiffener. Enter the name ***filter_beam_stiffener*** in the text box and click <**Save as**>. (see picture below).



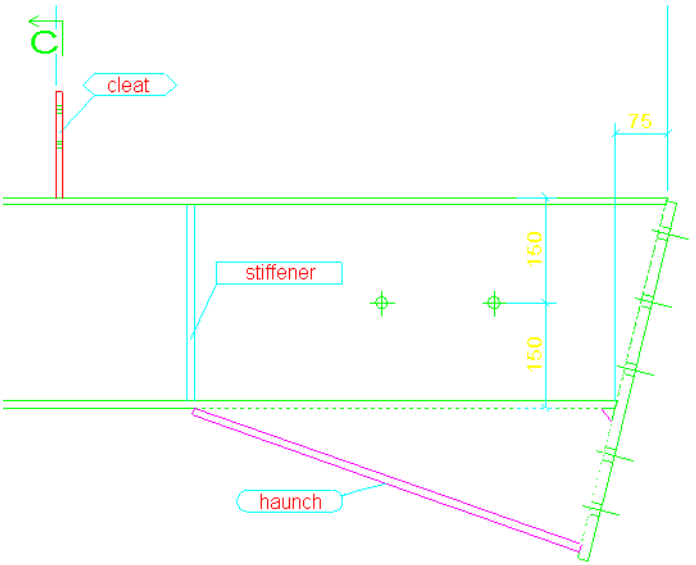
6. Repeat the steps for all parts you want to define. In this example: ***filter_beam_cleat*** and ***filter_beam_haunch***. Click <**Cancel**> to close the dialog box!



7. Go to **> Properties > Assembly drawing...** Click the down arrow in the Load text box and select the ***beam*** drawing properties you created. Click <**Load**> (see picture below). Now create the general arrangement drawing.



You will see that a drawing is created in which different welded parts appear in different colors.





The Drawing Classifier can only be used for objects you can filter!

3.6 Component database

Topics

This chapter is divided into following sections:

Add components to favorites (p.91)

Store (p.92)

Adding keywords (p.94)

Adding descriptions (p.98)



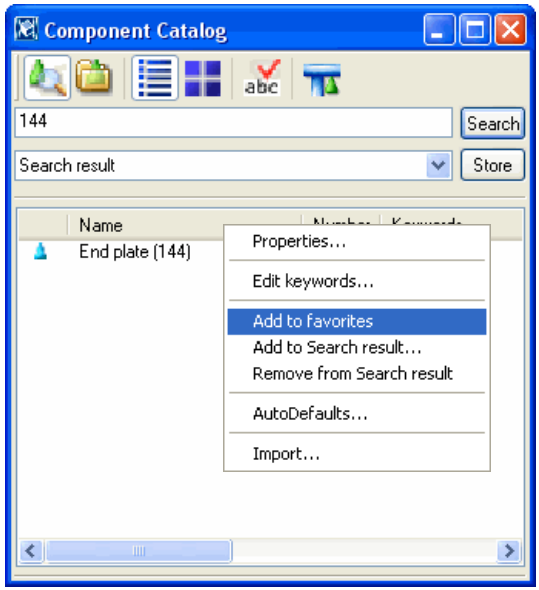
The Component catalog contains several useful functions to order components so that they can be easier found in the database. Click the icon or use shortcut <**Ctrl**> + **F** to open the dialog box.

Add components to favorites

The command **Add to favorites....** adds components to your personal favorites.

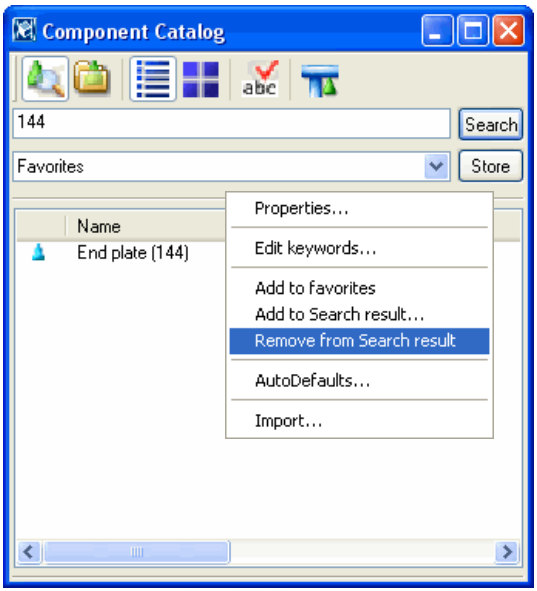
Step plan

1. Look up the component you want to add
2. Click right mouse on the line and select **Add to favorites**



3. The component is now added to the list Favorites

Use **Remove from Search result** to remove the component from the list.

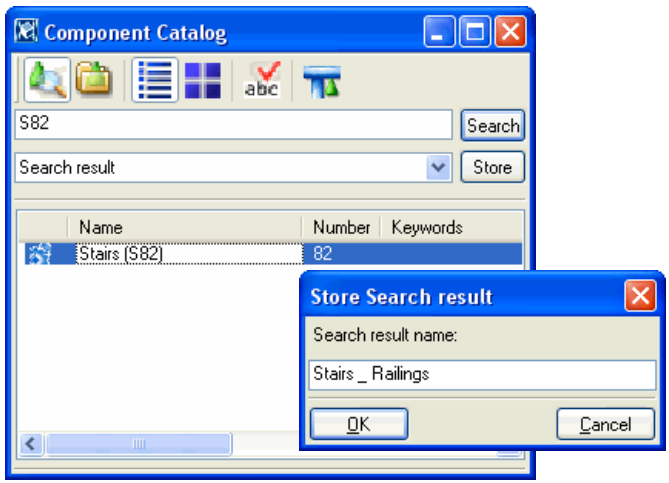


Store

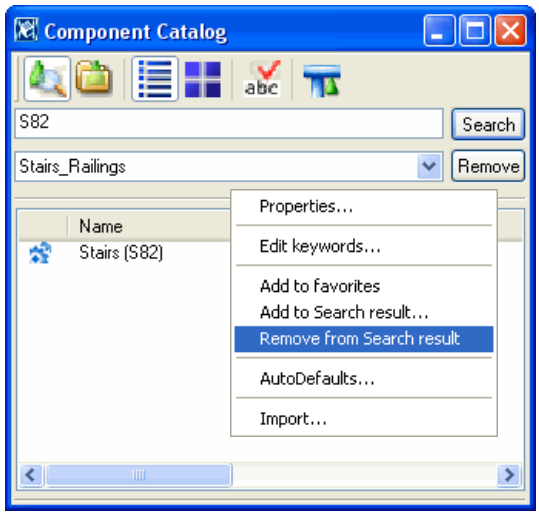
The command **Store** adds components to a collection you can name by yourself, for example *Stairs_Railings*.

Step plan

1. Look up the component you want to add
2. Select the line and click right mouse, now select **Store**
3. Enter a name for the group the component will belong to



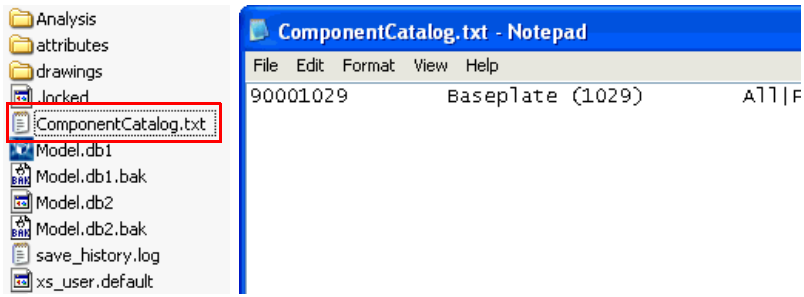
Use **Remove from Search result** to remove the component from the list.



Components may be grouped in different collections, so e.g. in *Favorites* but also in *Strairs_Railings* and *Steel construction*!

When you modify the component catalog, so when you add components to a collection, change or delete, these modifications are stored in the model folder. To apply these modifications for all models, do the following:

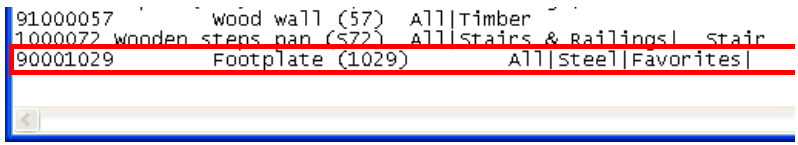
1. You have added component *Foot plate (1029)* to your favorites
2. Open the model folder and open the file *ComponentCatalog.txt* (using e.g. Notepad)



3. Select the line in the file
4. Click right mouse and select **Copy** or press <**Ctrl**> + **C**
5. Close the file *ComponentCatalog.txt* and go to folder

   disk:\TeklaStructures\version\environments\europe\system
6. Open file *ComponentCatalog.txt (using e.g. Notepad)*

7. Place the cursor at the last line, click right mouse and select **Paste** (or press <**Ctrl**> + **V**)



8. Save the file
9. Restart Tekla Structures and open an existing model or create a new model. You will see that the component is available in the *Favorites* group.

When components are added to a new collection, the specific line which includes the name of that collection, also has to be copied to the *system* folder, e.g. the line:

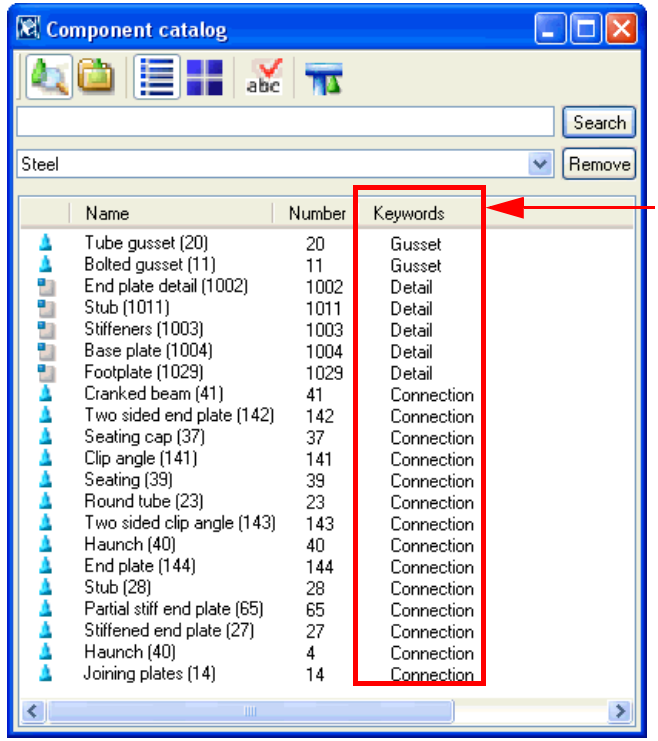
COLLECTION Base plates

Copy this line to the top of the file close to the other collection names.

Delete the file *ComponentCatalog.txt* in the model folder or clear up the file to make sure that collections will not appear twice in the dialog box Component database.

Adding keywords

Keywords can be used to add an extra name to the list. Like other columns, you can sort this name column too.

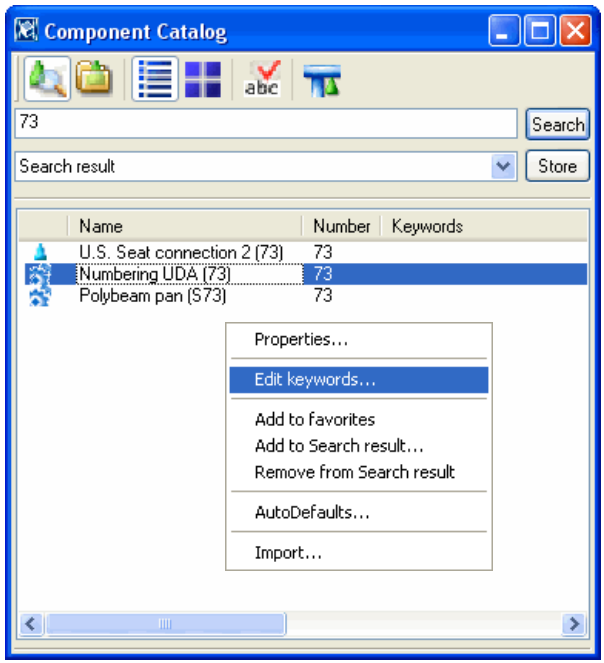


Clicking the *Keywords* column will change the sorting order.

You want to add keyword “numbering” to component *Numbering UDA (73)*.

Step plan

1. Search for the component in the component catalog
2. Click right mouse and select **Edit keywords...**



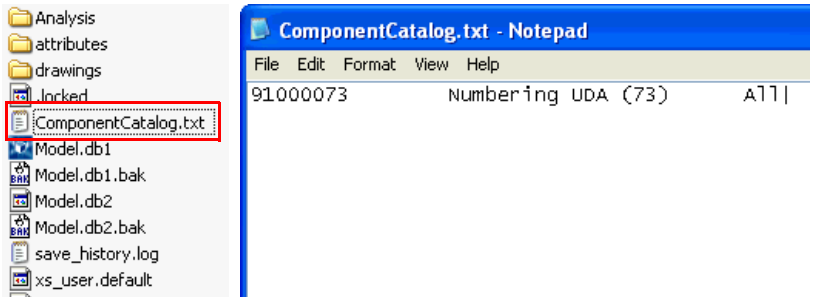
3. The dialog box **Keywords** will be opened. Enter the keyword **numbering** and click **Add**, the keyword will be added.



When you modify the component catalog, so when you add keywords to components, change or delete, these modifications are stored in the model folder.

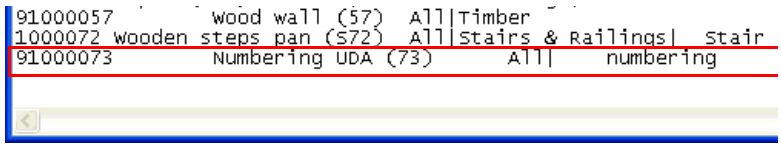
To apply these modifications for all models, do the following:

1. You have e.g., added keyword “numbering” to *Component 73 (Numbering UDA)*
2. Open the model folder
3. Open file *ComponentCatalog.txt* (using e.g. Notepad)



4. Select the line
5. Click right mouse at the line and select **Copy** (or press <**Ctrl**> + **C**)
6. Close the file *ComponentCatalog.txt* and go to folder

   disk:\TeklaStructures\version\environments\europe\system
7. Look up file *ComponentCatalog.txt*
8. Open this file in e.g. Notepad
9. Go to the last line, click right mouse at the line and select **Paste** (or press <**Ctrl**> + **V**)





If there is an existing line in the file **ComponentCatalog.txt** for Component Numbering UDA (73), you have to replace this line!

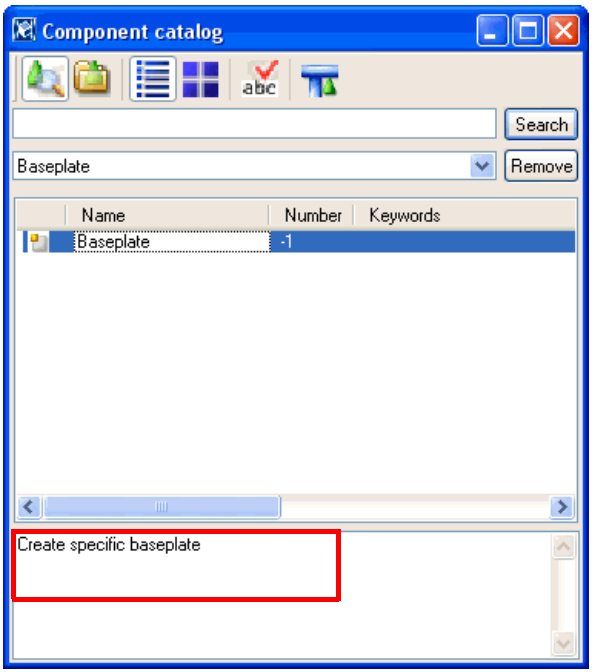
10. Save the file
11. Restart Tekla Structures and open an existing model or create a new model. You will see the keyword has been added.



Do not add keywords in file **ComponentCatalog.txt** but only in the component catalog in Tekla Structures!

Adding descriptions

You have the option to add descriptions to components or to modify them. This feature is useful for users who created Custom Components.



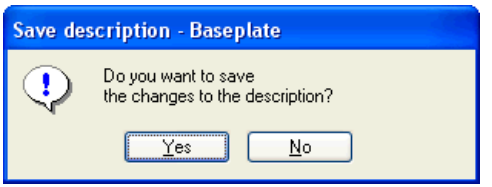
The description shows the purpose of the component.



A description may contain **80** characters maximum!

Step plan

1. Select a component or a Custom Component
2. Click button **Description** and add or edit a description
3. Click button **Store**, the following dialog box appears:



When you modify the component catalog, so when you add descriptions to components, change or delete, these modifications are stored in the model folder.

To apply these modifications for all models, do the following:

1. You have e.g, added description “component to add prefixes and start numbers” to *Component 73 (Numbering UDA)*
2. Open the model folder and go to the folder **attributes**
3. Select the contents of the file *ComponentsDescriptions.csv*
4. Copy this contents to the file *ComponentsDescriptions.csv* in the folder *drive:\TeklaStructures\version\environments\europe\system*
5. Restart Tekla Structures and open an existing model or create a new model. You will see the description has been added.

3.7 Control numbers

Topics This chapter is divided into following sections:

Assigning control numbers to parts (p.100)

Lock and unlock control numbers (p.102)

Assigning control numbers to parts

Control number is a consecutive numeric attribute which you can assign to parts in the model.

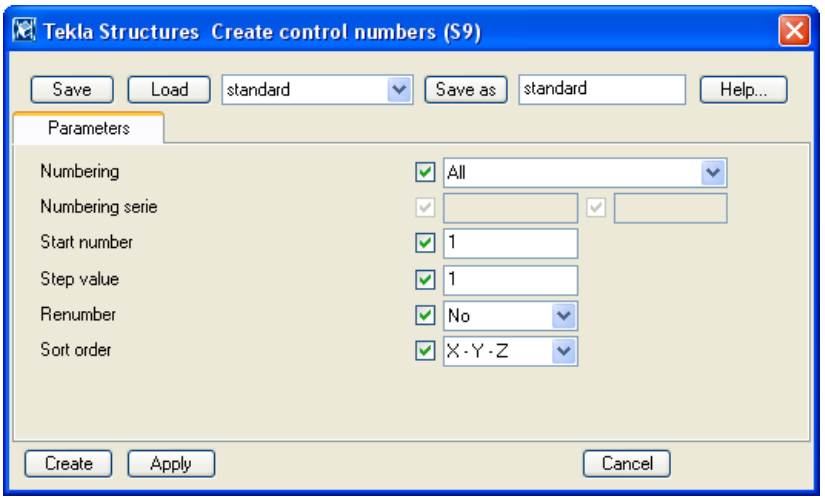
These numbers are used to identify the location of parts (for example, precast concrete parts) in the building or structure.

Control numbers are unique, so two parts cannot have the same number. Control numbers can be consecutive throughout the entire model, or you can define numbering series.

The numbers are shown as attribute ACN in reports, drawings, and when you use the **Inquire** > **Object...** command.

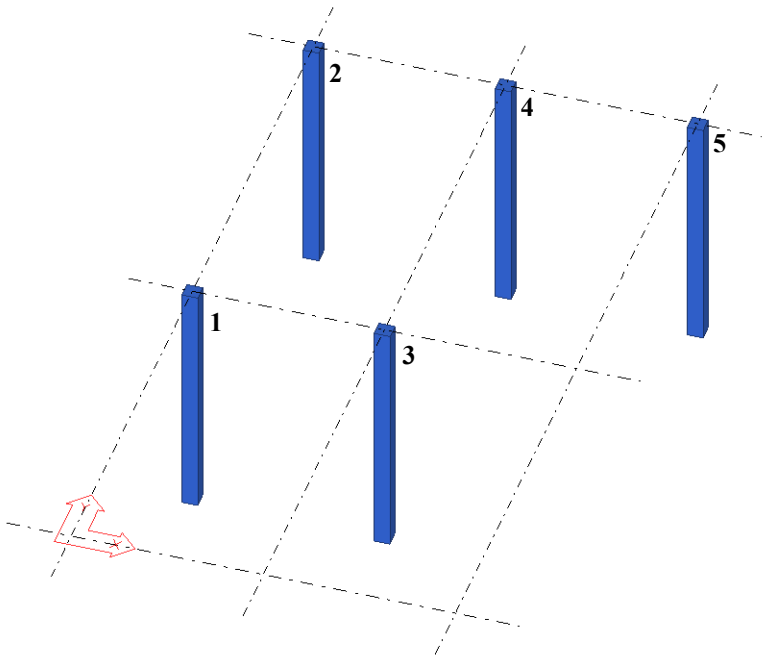
To assign control numbers:

1. Check that numbering is up-to-date (**Tools** < **Numbering** > **Modified**).
2. Click **Tools** < **Numbering** > **Create control numbers...** to open the **Create control numbers** dialog box:



3. Select the parts to which you want to assign control numbers. If you do not select anything, all parts will be numbered.

4. To assign consecutive numbers to all parts, select **All** in the **Numbering** list box. To create control numbers for one numbering series, select **By numbering serie** and enter the prefix and start number in the **Numbering serie** fields.
5. Enter the start number and step value. For example, if the start number is 2 and step value 3, the parts receive control numbers 2, 5, 8, 11, etc.
6. If you already have parts with control numbers in the model, you can define if the existing numbers will be renumbered. To renumber, select **Yes** in the **Renumber** list box.
7. In the **Sort order** list box, select the order in which the parts will be numbered. Tekla Structures numbers parts according to the global coordinate system. The options are **None**, **X-Y-Z**, **Y-X-Z**, **Z-X-Y**, and **Z-Y-X.** For example, in the following image the sort order is X-Y-Z.

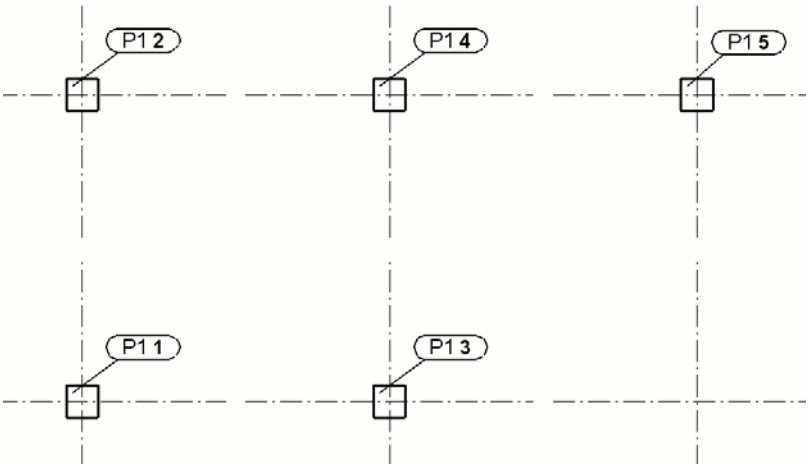


8. Once you have defined the properties, click <**Apply**> and <**Create**>. Tekla Structures applies the control numbers to the parts.

By default, a setting **pile-layout** is added.

Pile layout on the drawing

The control numbers are displayed in the marks:

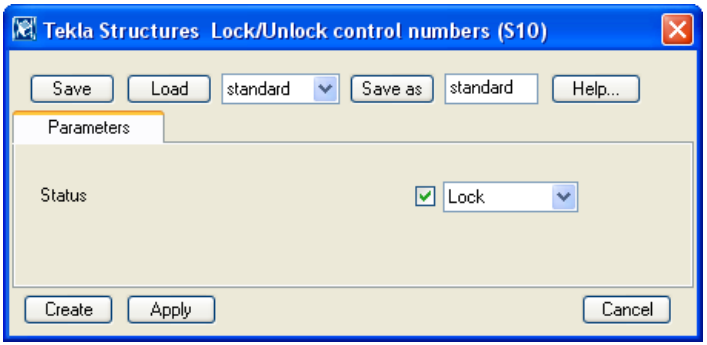


Lock and unlock control numbers

Now you can lock and unlock control numbers. Tekla Structures does not renumber locked control numbers. The locking status is shown with user-defined attribute `ACN_STATUS`.

To lock or unlock control numbers:

1. Click **Tools** > **Numbering** > **Lock/Unlock control numbers** to open the **Lock/Unlock control numbers** dialog box:



2. If you do not select any parts, this command will affect the control numbers of all parts.
   - To only unlock or lock the control numbers of specific parts, select the parts.
3. Select **Lock** or **Unlock** in the **Status** list box.
4. Click <**Apply**> and <**Create**>.

3.8 Converting plates to flat bars

Plates can be converted automatically to flat bars. You can set which flat bar sizes are available for converting. This can be decided during modelling or after modelling.

The profiles will retain their notation in the model (PL or PLT), but during output, such as reports, drawings and NC-files, the notation will converted to **FLAT** (for the plates that can be converted of course).

To enable converting plates to flat bars, two conditions must be fulfilled:

1. In the Tekla Structures batch file, the following variables must be set:

   set XS_FLAT_PREFIX=FLAT

   set XS_USE_FLAT_DESIGNATION=TRUE

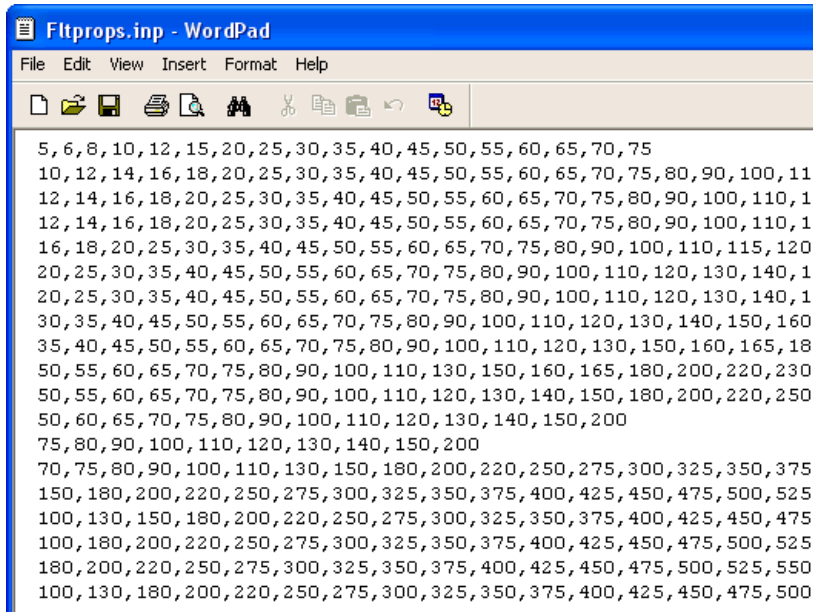
   set XS_CHECK_FLAT_LENGTH_ALSO=TRUE

   set XS_USE_NEW_PLATE_DESIGNATION=TRUE

   set XS_FLAT_TOLERANCE=1.0

   set XS_FLAT_THICKNESS_TOLERANCE=0.4

2. The file fltprops.inp must contain the flat bar sizes:



disk:\TeklaStructures\version\environments\europe\profil



In reports and templates which return the length, from version 7.x both variable ”LENGTH” as “LENGTH_GROSS” may be used.

The file fltprops.inp is located in the ‘profil’ folder, but it can also be stored in the model folder. This way you can have a unique flat bars table per project/customer.

3.9 Create adjusted symbols

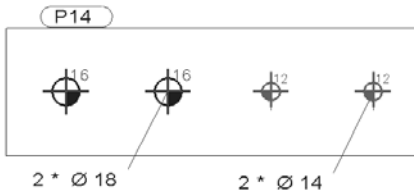
Topics

This chapter is divided into following sections:

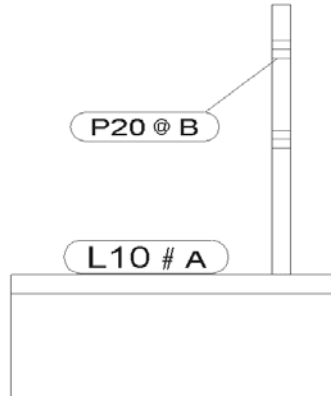
How to create bolt symbols? (p.105)

How to create material symbols? (p.107)

In Tekla Structures it is possible to create your own bolt- and material symbols.



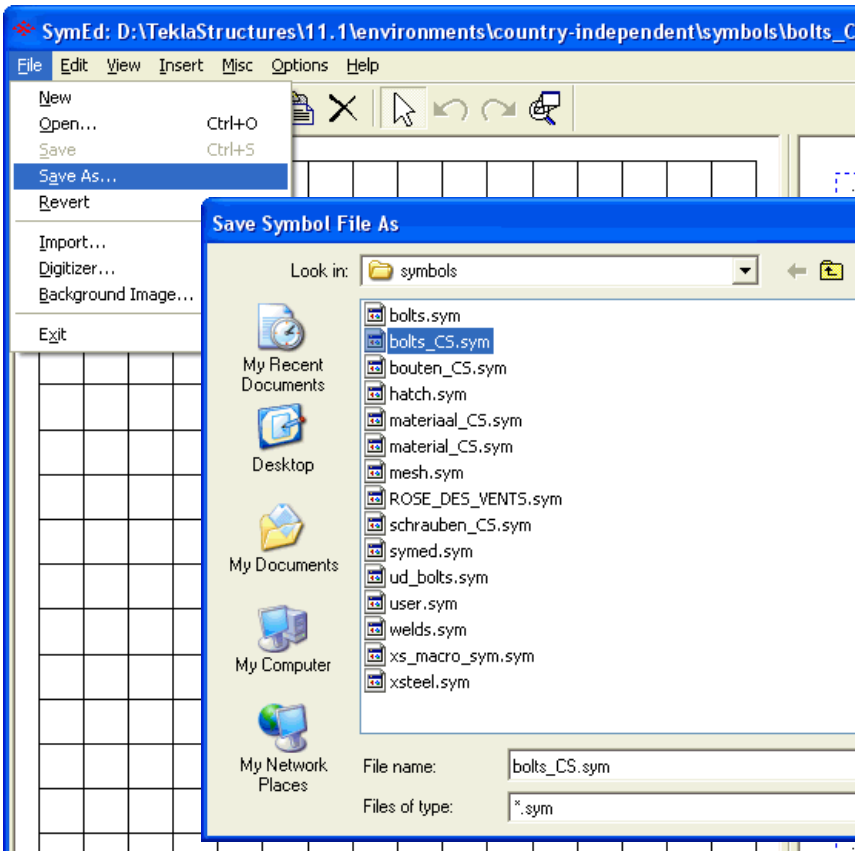
Various bolt symbols



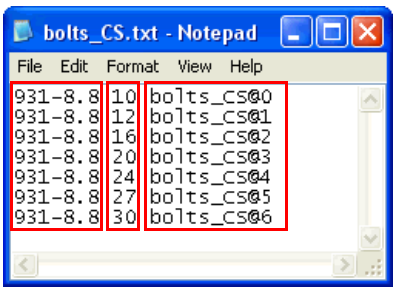
Various material symbols

How to create bolt symbols?

- Create the bolt symbols in the Symbol Editor. Save this symbol file by using the name “bolts_CS.sym” in the symbol-directory, *disk:\TeklaStructures\version\environments\country-independent\symbols*.



- Open a text processor like Notepad or Wordpad to create a text file. The text contains 3 rows:
  - The first row contains the bolt standard.
  - The second row contains the bolt size.
  - The third row contains the name of the symbol file and the symbol number, separated with an @- symbol. (Tekla Structures replaces the bolts containing the entered standard by the entered symbol in the drawing)

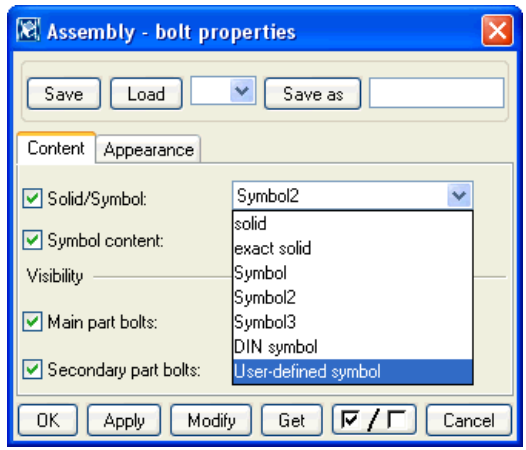


- Save the text file by using the name “bolts_CS.txt”
- Set the following variable in the batch file:

set XS_USER_DEFINED_BOLT_SYMBOL_TABLE=bolts_CS.txt

When behind the equality sign (=) not the full path name is entered, Tekla Structures will successively search in the model-, firm- en system directory

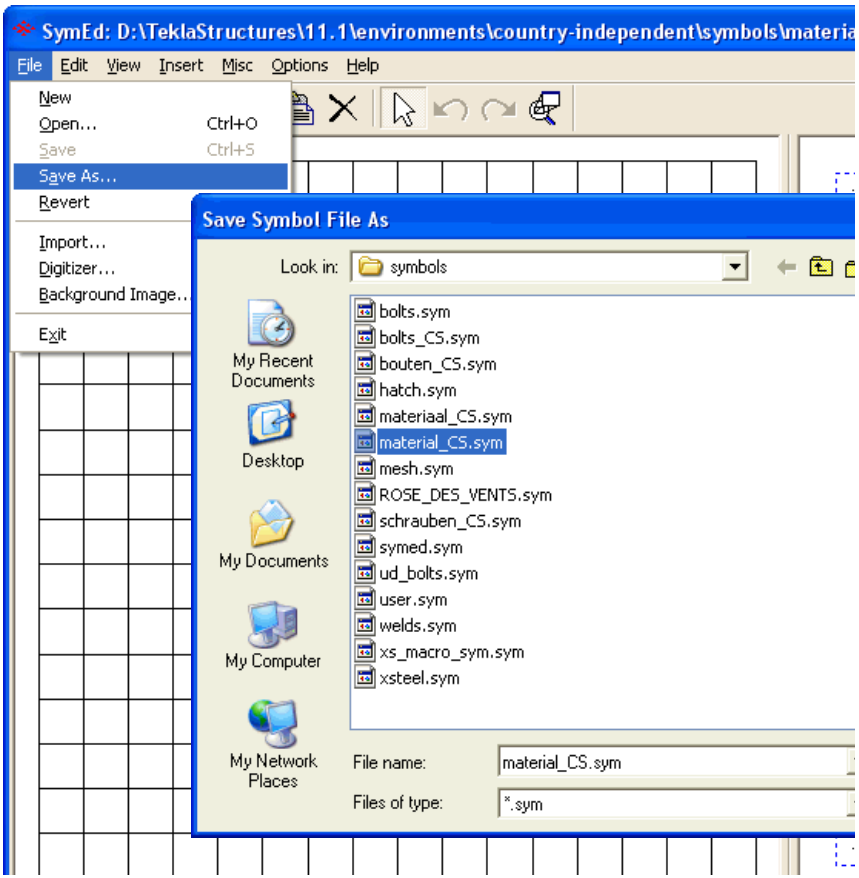
- Open a drawing to change the bolt symbol



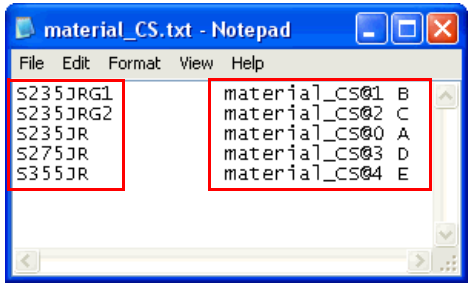
- Now double-click the bolt, the view or the drawing to change the bolt symbol. Select ***User defined symbol*** and click <**Modify>.**

How to create material symbols?

- Create the material symbols in the Symbol Editor. Save this symbol file by using the name “material_CS.sym” in the symbol directory



- Open a text processor like Notepad or Wordpad to create a text file. The text contains 2 rows:
  - The first row contains the material name used in Tekla Structures.
  - The second row contains the description of the symbol. This description replaces the material name in the drawings. The definition can also contain characters, figures and other special characters:



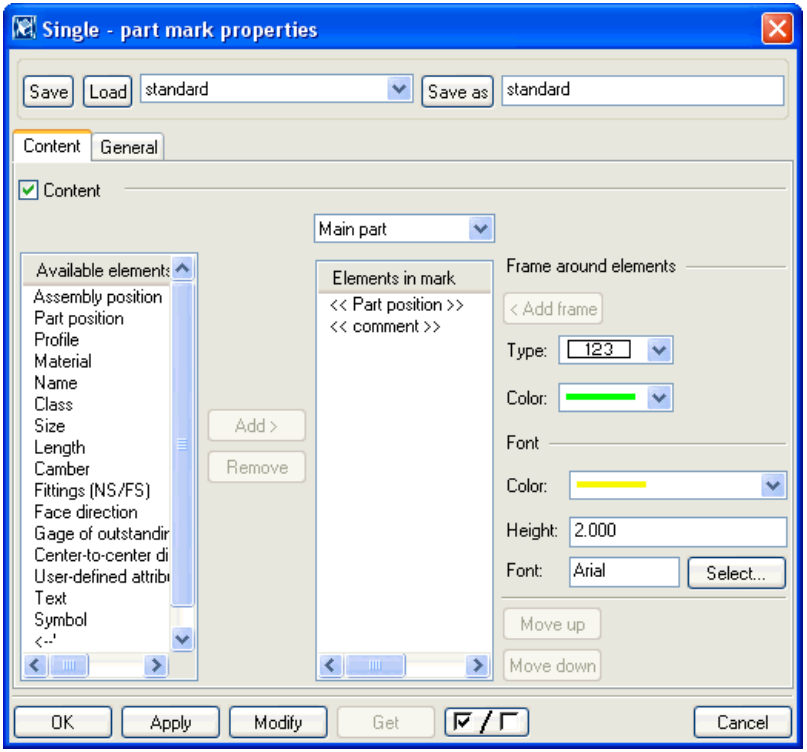
- Save the text file by using the name *material_CS.txt*
- Set the following variable in the batch file:

set XS_MATERIAL_SYMBOL_REPRESENTATION_FILE=material_CS.txt



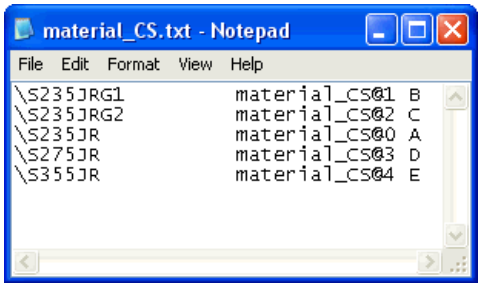
When behind the equality sign (=) not the full path name is entered, Tekla Structures will successively search in the model-, firm- en system directory

- Now open a drawing in which you want to change the material symbol.
- Double-click a mark, a view or the drawing you want to change the mark.





- If you want to use the original material name, add a back-slash symbol before the material type (\). See below.



3.10 Creating toolbars, own pull-down menu's and shortkeys

Topics

This chapter is divided into following sections:

Toolbars (p.111)

Creating 'own' pull-down menu's (p.118)

Using shortkeys in Tekla Structures (p.121)

Copying toolbars and shortkeys (p.126)

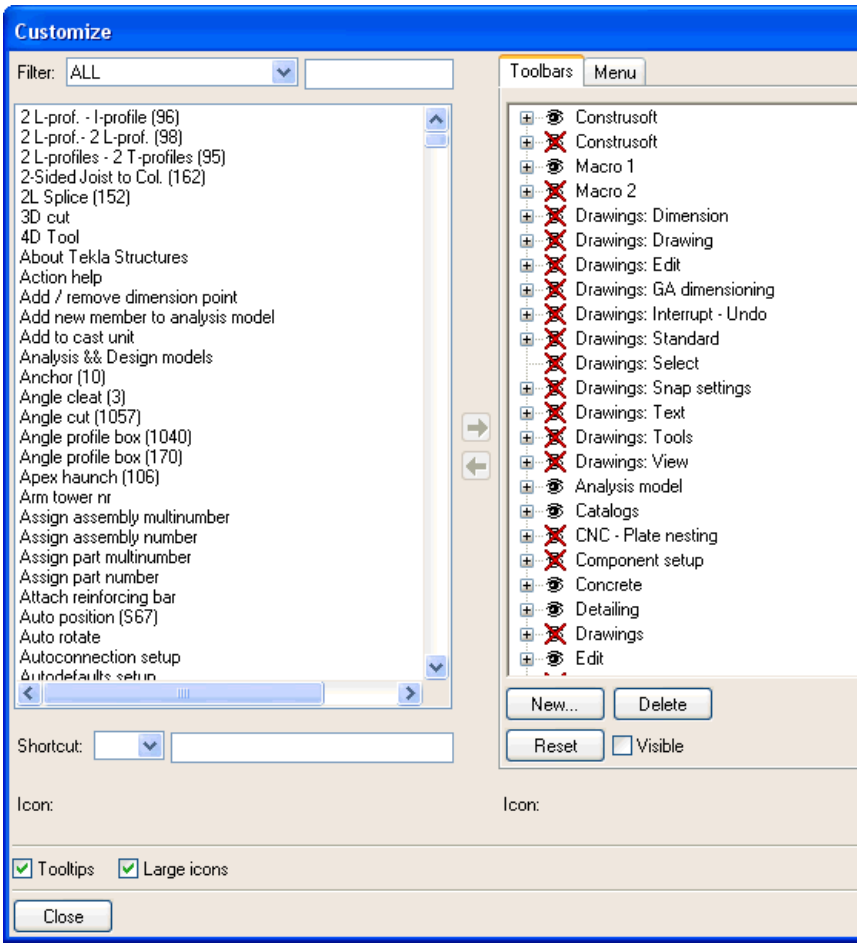
Toolbars

Several commands can be accessed through toolbar icons. This is the fastest way of activating commands.

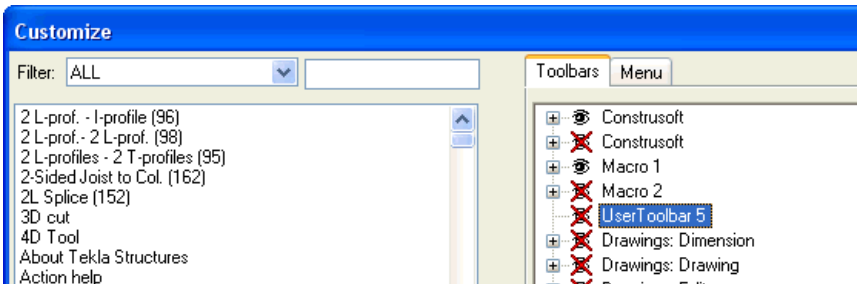
Some commands are not available in a default Tekla Structures toolbar. However, you can add commands to default toolbars, or you can create your own toolbars.

Creating new toolbars

Go to **Tools** > **Customize.** The dialog box below will appear.



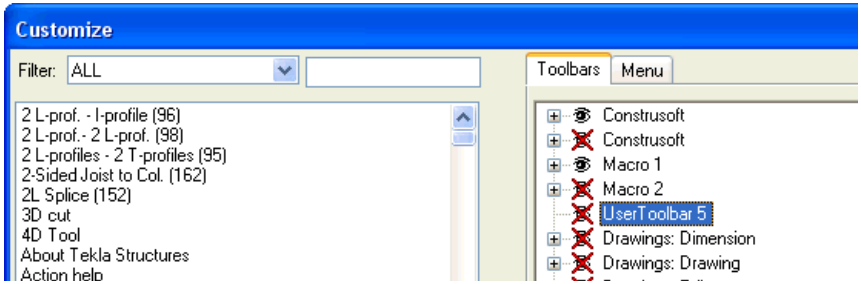
Click <**New**>



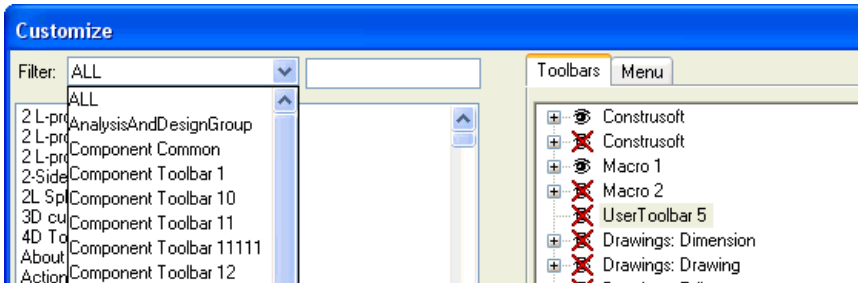
A new toolbar, named **User Toolbar 5** will be added to lthe list.

The name of the toolbar can be changed by clicking it once.

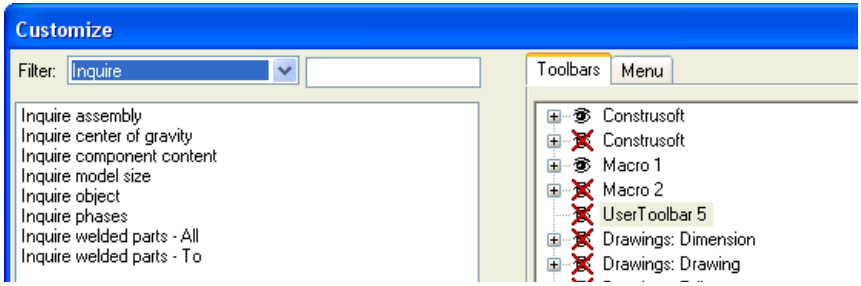
The left column contains a list with all available Tekla Structures commands.



With the Filer listbox, you can filter commands by groups.



Example: Adding command **Inquire assembly** to toolbar User toolbar 1. From the *Filter* listbox, select 'Inquire'.

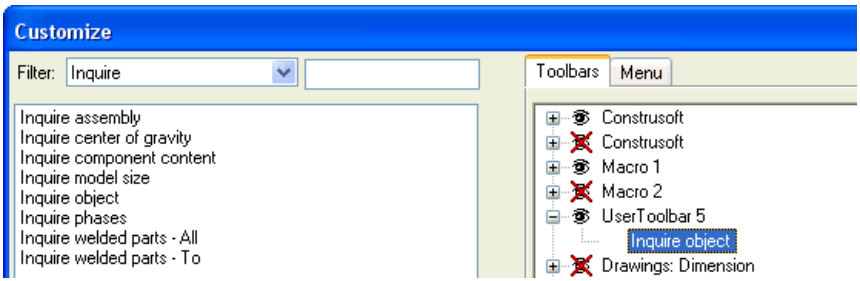


The list will now contain commands from the Inquire group only. Make sure a toolbar is selected where the new command should be added to. Select command Inquire assembly and double click it. (Or press the right-arrow button). The command will now be added to the selected toolbar.


Right-arrow button for adding a command to a toolbar.


Left-arrow button for removing a command from a toolbar.

In front of the toolbar, a '+'-character is displayed. This means that at least one command is put in that toolbar. By pressing the + character, you can see which command(s) are in the toolbar.





If multiple commands are added to a toolbar, you can reorder the commands with the buttons opposite.

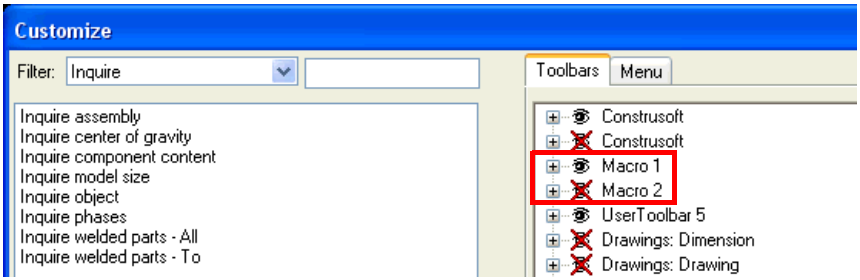
In front of each toolbar, an ‘eye’ is displayed. The appearance of this eye can be with or without a cross. If the eye is not ‘crossed’, it means that the toolbar is visible in Tekla Structures. A ‘crossed-eye’ toolbar is invisible in Tekla Structures. Toggle between visible and invisible state can be done by checking/ unchecking the ‘Visible’ option.

Individual ’leaf through’ toolbars

It is possible to create leaf-through toolbars.

In these toolbars, you can add frequently used components for instance. Creating these toolbars basically works the same as creating ‘normal’ toolbars, although there are some things to pay attention to.

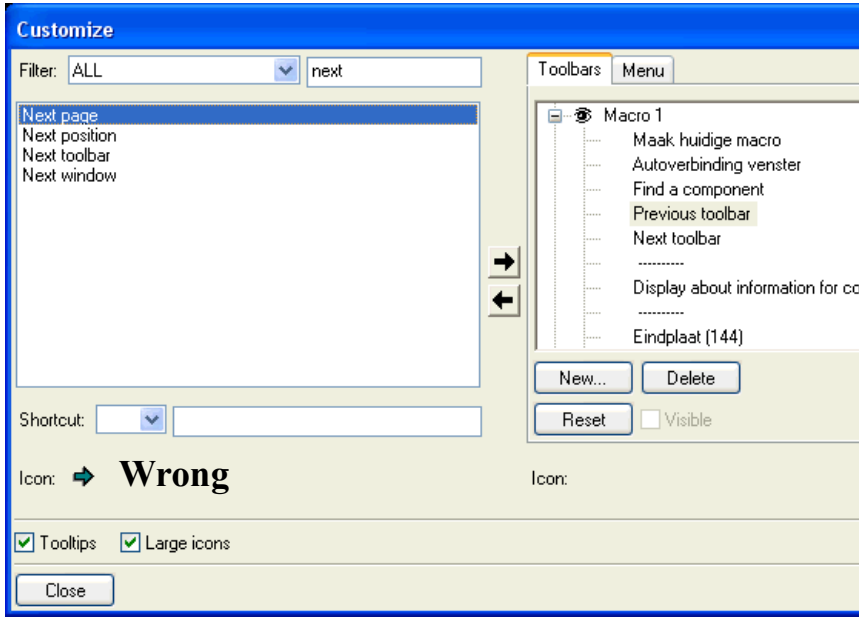
Go to **Tools > Customize** and create a new toolbar. Name this toolbar **Macro 1**.

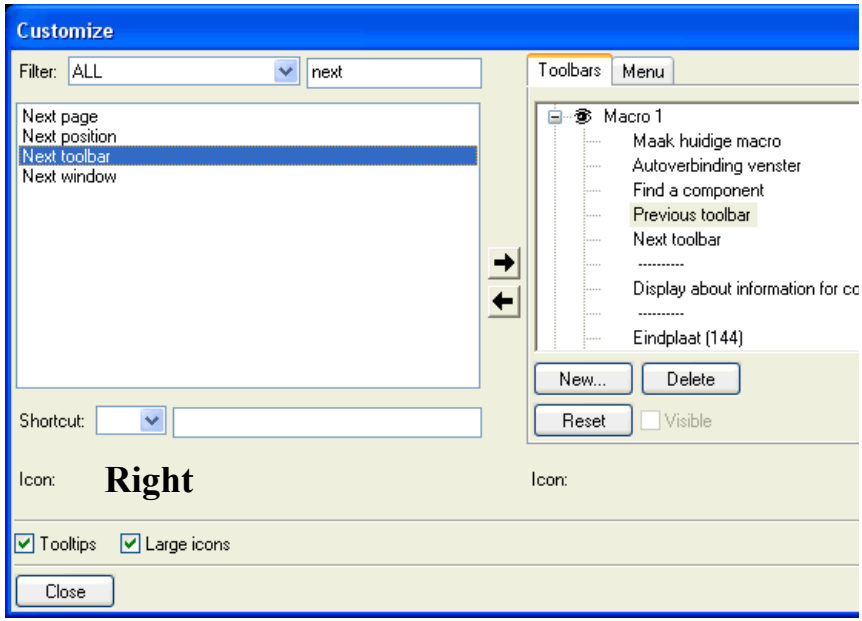


To create a ‘leaf through’ toolbar, two commands, **Previous page** and **Next page,** must be added to the toolbar.

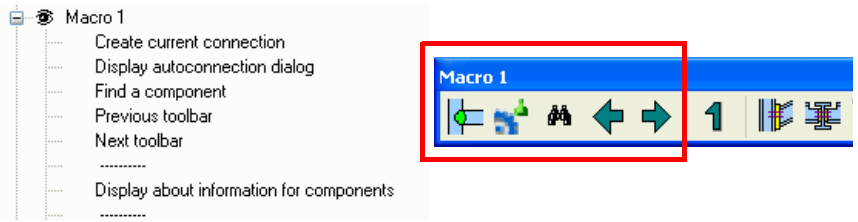
The easiest way of selecting these commands is to enter “prev” and/ or “next” in the filter string in the dialog box.

Note: these items are double listed. Make sure you add ‘Previous page’ and ‘Next page’ (and not ‘Previous toolbar’ or ‘Next toolbar’)!





Also to typical component toolbar buttons **Create current connection**, **Create User connestion,** or a pagenumber **Display information about connections** can be added.



Select the components which you want to add to the toolbar. The creation of the 2nd or 3rd toolbar is in the same way.



The toolbars' names must be equal, only their running numbers must vary.



The sytem folder:

(disc:\TeklaStructures\version\environments\europe\system)

contains the files:

TS_FUD_W2K_XP.reg (on behalf of the module *Full Detailing*)

TS_PCD_W2K_XP.reg (on behalf of the module *Precast Concrete Detailling***)**

TS_PRM_W2K_XP.reg (on behalf of the module *Project Management***)**

TS_RCD_W2K_XP.reg (on behalf of the module *Reinforced Concrete Detailing***)**

TS_SDE_W2K_XP.reg (on behalf of the module *Standard Design***)**

TS_STD_W2K_XP.reg (on behalf of the module *Steel Detailling***)**

TS_TID_W2K_XP.reg (on behalf of the module *Timber Detailling***)**

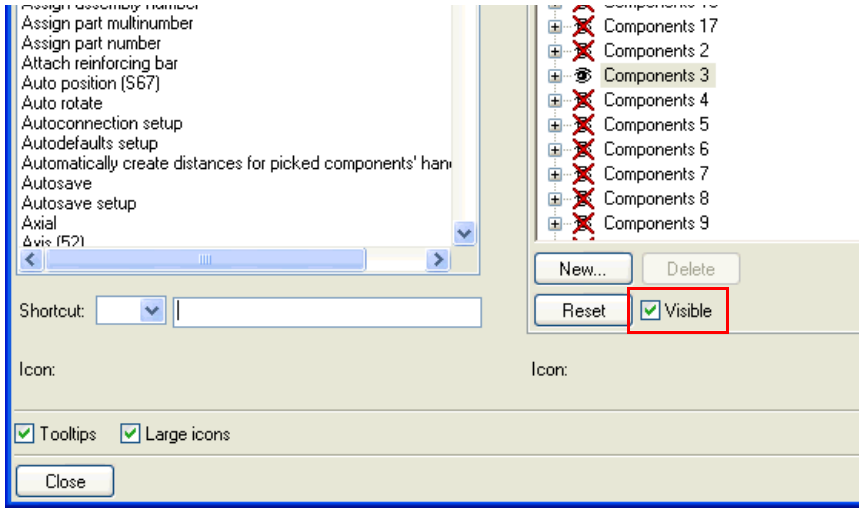
TS_VIE_W2K_XP.reg (on behalf of the module *Viewer***)**

(only for Windows 2000 and XP).

By doubleclicking one of these files, depends on the module you have, a number of schortcuts and toolbars will be added to the Windows Registry. Also, 2 'leaf through' toolbars are added (***Macro 1 and 2***), containing a number of frequently used components. The content of these toolbars may be edited to your own opinion.

If personal 'leaf through' toolbars are used, you can opt to switch off the general component toolbars to save screen space.

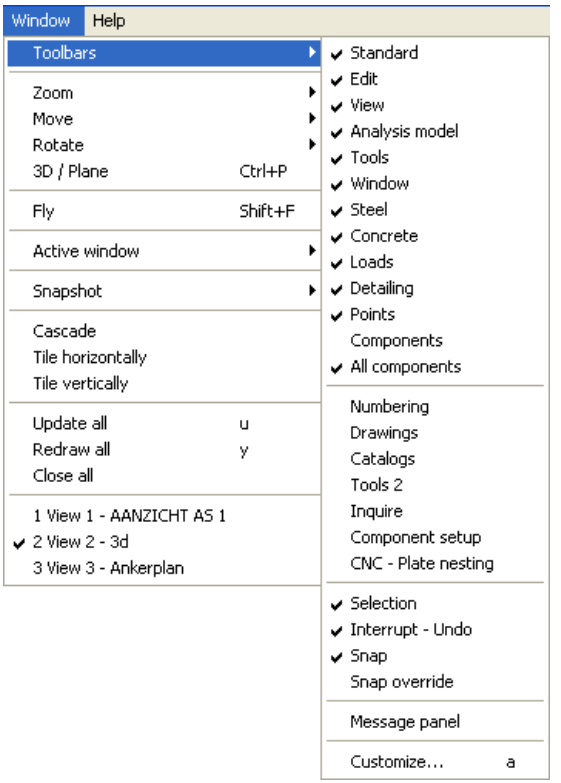
The best way to switch off toolbars is by unchecking the 'Visible'-option for the selected toolbar in the Customize dialog box (**Window** > **Toolbars** > **Customize**).



Now the standard componentpages are disabled. By checking the *Visible* check-box, the pages will be visible again. To make the default component toolbars invisible permanently, the Tekla Structures batch file contains a line to do this:

rem set XS_CREATE_STANDARD_JOINT_TOOLBARS=FALSE

The default toolbars can be made visible or hidden by checking or unchecking the toolbars in pulldown menu **Window > Toolbars.** If checked, the toolbar is visible.



The user defined toolbars are not listed in the pulldown-menu. To hide or display those toolbars, you always have to do that in the **Customize** dialog box by checking/ unchecking the ***Visible***-option.

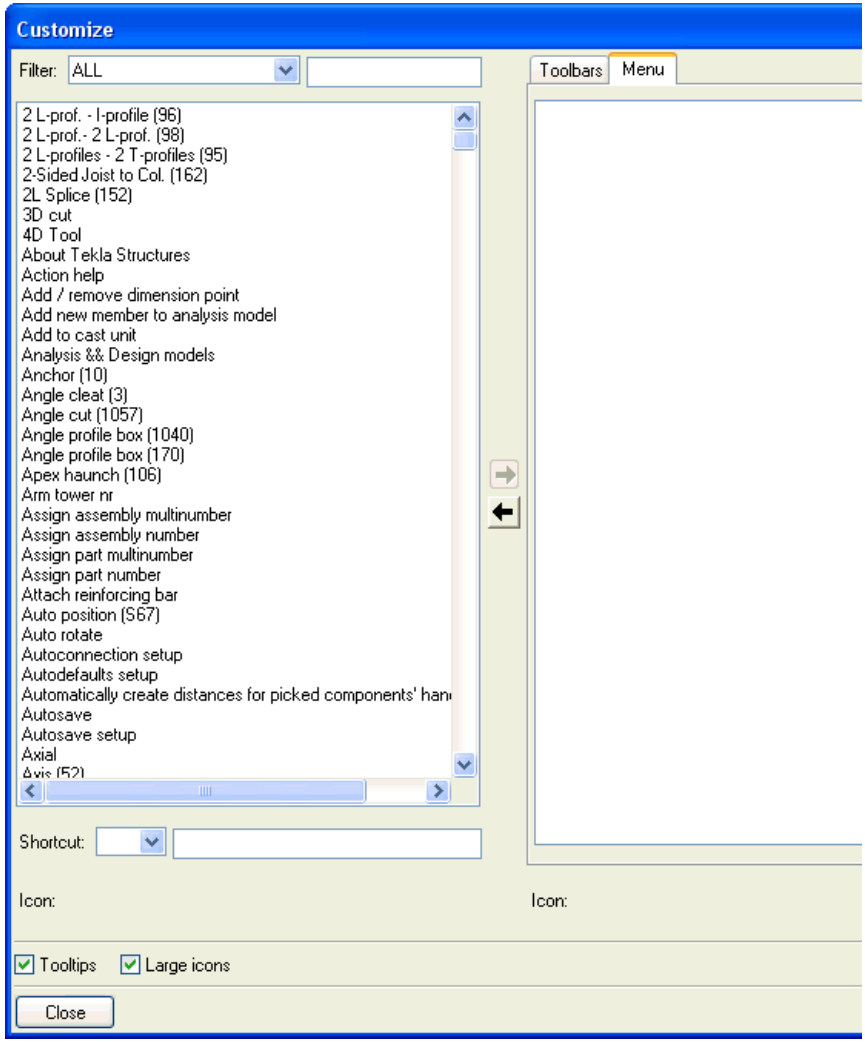
Creating 'own' pull-down menu's

Tekla Structures offers the possiblilty to create your own pull-down menus. For both the Model environment as the Drawing environment, one pull-down menu can de added.

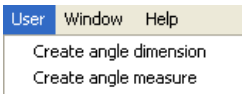
Pull-down menu in the Model Editor

Go to **Window > Toolbars > Customize** or **Tools > Customize.**

- Select the tab **Menu**



- On the left side, select a command which you want to add to the new pull-down menu
- Click the arrow button to add the command to the menu
- Repeat these steps untill all desired commands are added to the menu
- Close the dialog box
- Open and close a drawing, or restart Tekla Structures to have the modifications carried through



Now, you will see a new menu is added to the environment. The menu is called '**User**'. Note: the name of this menu can not be changed.



It can be convenient to put a 'deep' command in an own pull-down menu. (For example: **Tools** > **Numbering** > **Full** or **File** > **Catalog** > **Profile** > **Modify**.

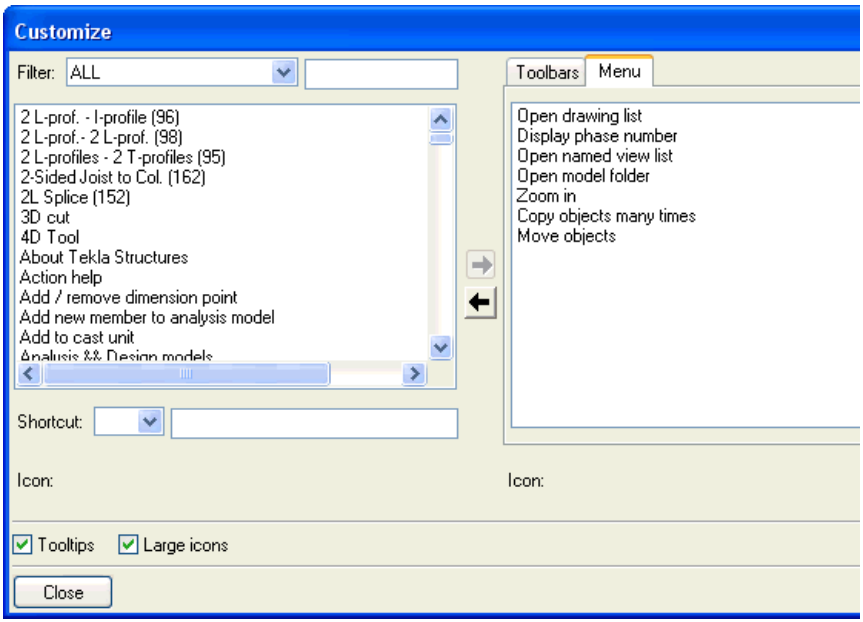
You can even add a command for opening the Customize-dialog box.

Pull-down menu in the Drawing Editor

To add a user pull-down menu in the drawing environment, a drawing has to be opened.

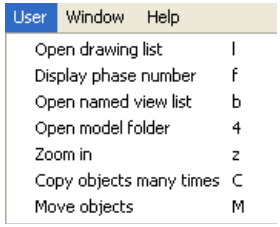
Go to **Window > Toolbars > Customize** or **Tools > Customize.**

- Select the tab **Menu**



- In the left column, select a command which has to be added to the user pull-down menu
- Click the right-arrow button to add the command to the menu

- Repeat these steps until all desired commands are added
- Close the dialog box
- Note: the user pull-down menu is only visible after re-openeing a drawing or resarting Tekla Structures.



The name of the pull-down menu is ***User***. The name of this menu can not be changed.



Our advise is to add these commands!

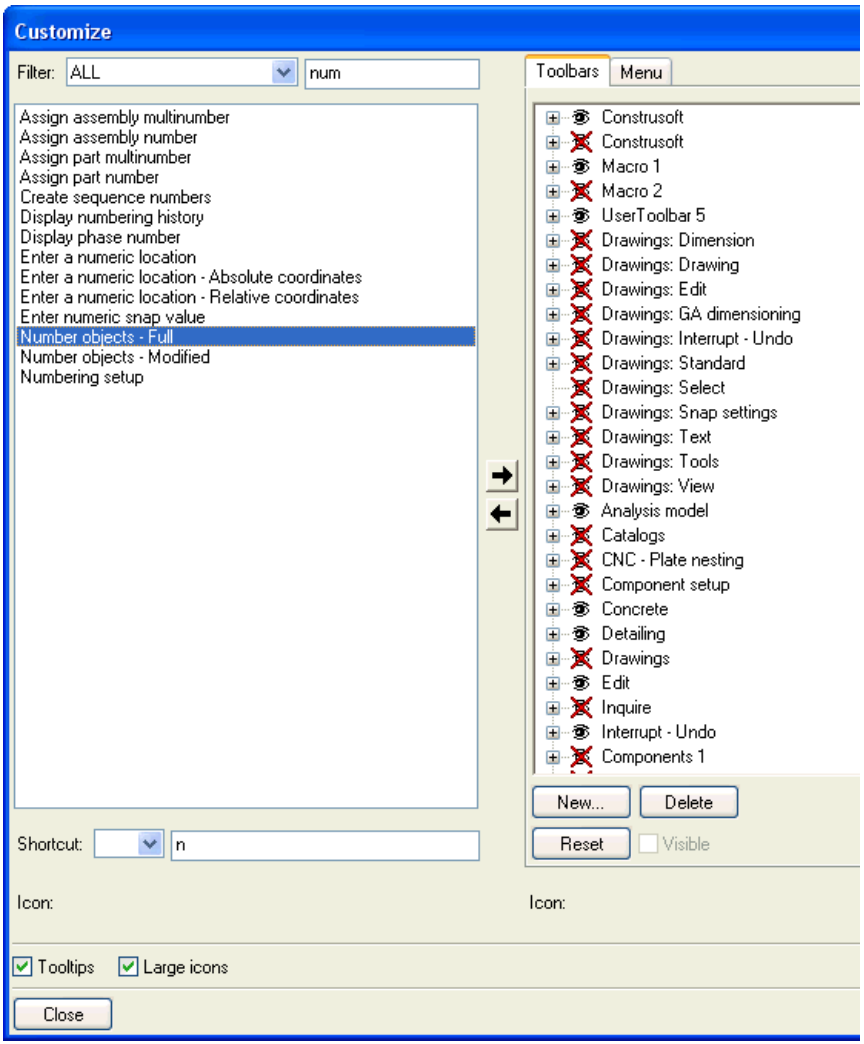
Using shortkeys in Tekla Structures

To access Tekla Structures commands more quickly, you can make use of short-keys.

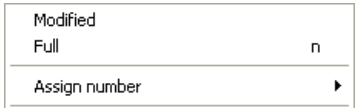
For example, for the **Number objects > Full** command, you can set the "**n**" character as shortkey.

Step plan:

- Go to **Window > Toolbars > Customize** (or **Tools > Customize...**_



- In the left column of the dialog box, look up the "**Number objects > Full** " command. (Use a filter for quick look up; in this case select 'Tools' from the Filter-listbox and enter the characters "**num**". Only commands which match these criteria will be shown in the list)
- Click the command "**Number objects > Full**" and press the "**n**" key.
- Reopen a model or restart Tekla Structures
- Go to **Tools > Number objects > Full**



- You will see that an "***n***" character is added to the Numbering command.
- When pressing the "***n***"-key, the Full numbering command is carried out.

Overview shortkeys in Tekla Structures:

	Command		Command
a	Customizing toolbars	**m**	Move > Translate
c	Copy > Translate	**n**	Numbering > Full
e	Open model folder	**o**	Ortho
f	Phasemanager	**q**	Create clip plane
g	Redraw all	**s**	Smart Select
h	Rollover highlight	**t**	Xsnap
i	Inquire assembly	**u**	Update all
k	Inquire object	**v**	Pick rotation point
l	Drawing list	**w**	Wizard

	Command		Command
1	Create single-part drawing	**3**	Create GA drawing
2	Create assembly drawing	**4**	Create cast unit drawing

	Command		Command
Enter	Repeat last command	**Alt + p**	Print
Alt + Enter	Properties	**Ctrl + c**	Undo last polygon pick
Home	Zoom original	**Space bar**	Finish polygon input

	Command		Command
Ctrl + b	Named views	**Ctrl + m**	Move
Ctrl + c	Copy multi	**Ctrl + p**	2D / 3D
Ctrl + f	Find a component	**Ctrl + w**	Weld

	Command		Command
Ctrl + a	Select all objects	**Shift + k**	Column properties
Shift + b	Bolt	**Shift + l**	Beam properties
Shift + c	Copy > With 3 points	**Shift + m**	Move > With 3 points
Shift + f	Fitting	**Shift + z**	Line cut
Shift + h	Construction line		

Drawing Editor

	Command		Command
a	Customizing toolbars	**l**	Drawing list
b	Black and white drawing	**m**	Move objects
c	Copy objects many times	**o**	Ortho
e	Open model folder	**s**	Smart Select
f	Phasemanager	**t**	Xsnap
j	Close drawing	**u**	Update all
o	Inquire > Object	**z**	Zoom in

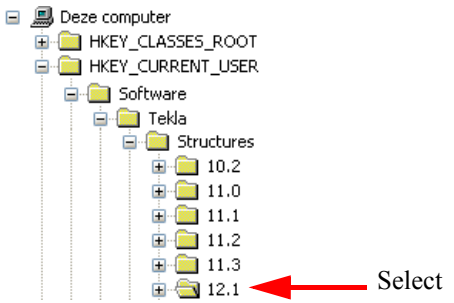
	Command
Ctrl + b	Open named view list
Alt + Enter	Properties
Home	Zoom original
Enter	Repeat last command
Ctrl + Page Up	Open previous drawing
Ctrl + Page Down	Open next drawing
Shift + A	Associative symbol

Copying toolbars and shortkeys

It is possible to copy the produced toolbars and shortkeys to another PC or from Tekla Structures 12.1 to Tekla Structures 13.1 These setting are stored in the Windows Registry.

To copy the settings, an export file of the registry has to be made:

1. Close Tekla Structures
2. In Windows, go to **Start > Run**
3. Enter **regedit** and click <**OK**>
4. Select the Tekla Structures version for which you want to export the toolbars and shortkeys. For example, copying the toolbars and shortkeys from version 12.1 to 13.1, select HKEY_CURRENT_USER\Software\Tekla\Structures\12.1, see picture below:

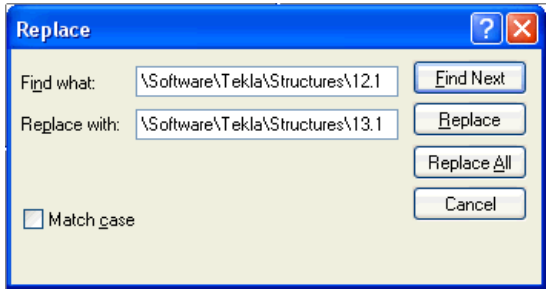


5. In the menu, go to **File > Export**
6. Enter a filename and a location for the registry file.
7. Close the Registry.
8. The file (extension *.reg) is stored on the specified location.

If you want to use the toolbars and shortkeys on another PC, then copy that reg-file to that PC and double-click the file. Answer the question “Are you sure you want to add the information....” with *Yes*.

If you want to export the toolbars and shortkeys to another Tekla Structures version, e.g. from version 12.1 towards version 13.1, the following actions need to be done:

1. Right click the exported registry file and select “Open with” from the contextmenu. Select Notepad.
2. Replace the line (using **Ctrl+H** on the keyboard, or go to **Edit > Replace)**: **HKEY_CURRENT_USER\Software\Tekla\Structures\12.1** through **HKEY_CURRENT_USER\Software\Tekla\Structures\13.1** and click the <**Replace all**>-button



3. Close the file, with saving of course.
4. Double-click the file, answer the question (Are you sure you want to add this information to the Registry) with *Yes*. The toolbars and shortkeys are now imported in the new version.

3.11 Dimensioning Reference side

You can define a reference side to create dimensions (for example, you might want Tekla Structures to dimension round bars to the middle of the profile rather than the reference line).

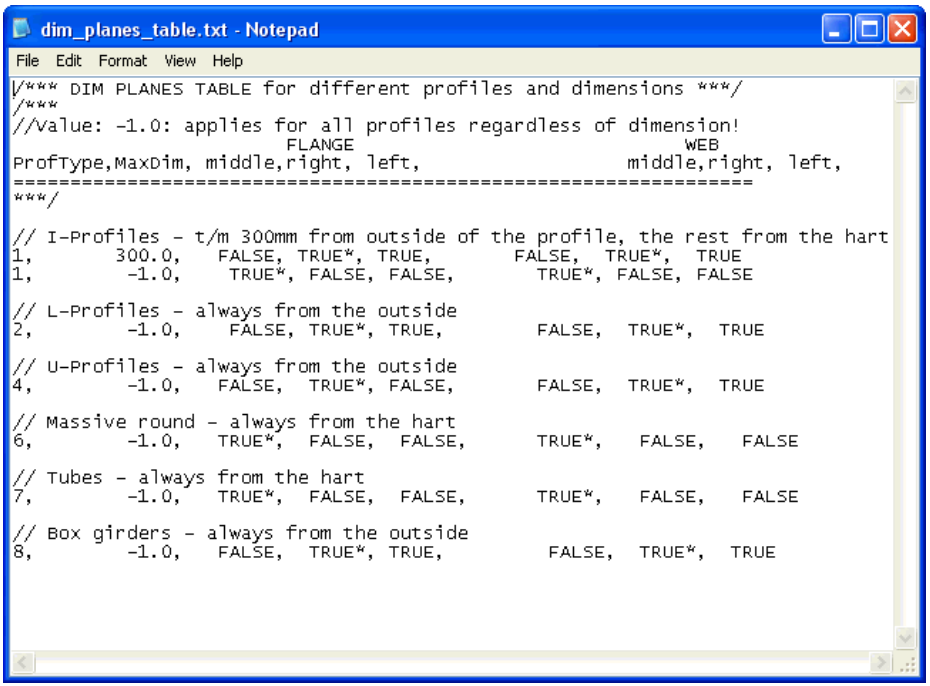
The reference side of dimensions is controlled by the file *dim_planes_table.txt*.

This file is located in folder profil (..*environments**europe**profil*).

In the file **env_europe.ini** in the folder (..*environments**europe*) it is refered to the variable

set XS_PART_DIMENSION_PLANES_TABLE=%XS_DIR%\environments\europe\profil\dim_planes_table.txt

The file looks as follows:

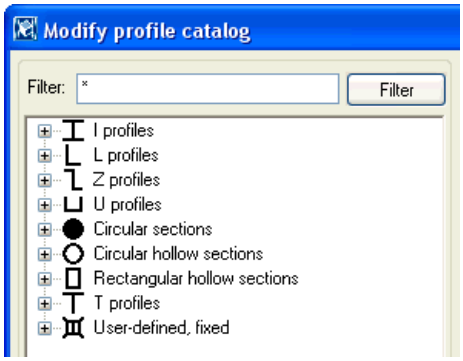


A line starts with the profile type:

1: I sections (HEA, HEB, IPE etc.),

2: L sections: angle steel,

etc.....



The second option is the maximum size (the **h** in the profile catalog).

If this value is set to -1.0, than all profiles are taken.

In the example above:

All I-sections upto and including a height of 300:

FALSE, TRUE*, TRUE, FALSE, TRUE*, TRUE

and the bigger ones:

TRUE*, FALSE, FALSE, TRUE*, FALSE, FALSE

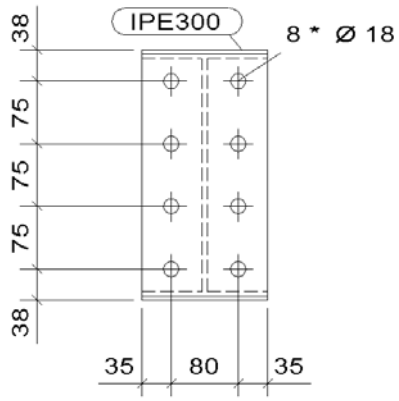
The value **TRUE** must occur at least once, as well in the FLANGE as in the WEB part.

If the value **TRUE*** (with an asterisk) is used, it means that is the preferred value (in case more than one TRUE-values are set).

Examples

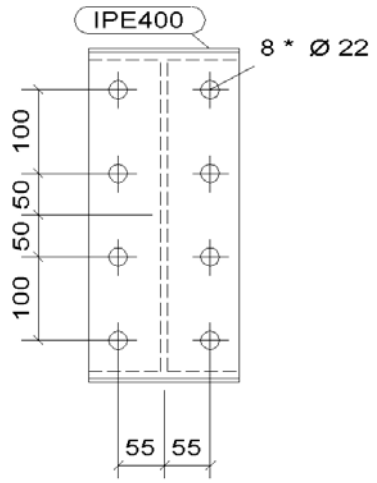
Below an example of an **IPE300**. This dimension is dimensioned according to line:

1, 300.0, FALSE, TRUE*, TRUE, FALSE, TRUE*, TRUE



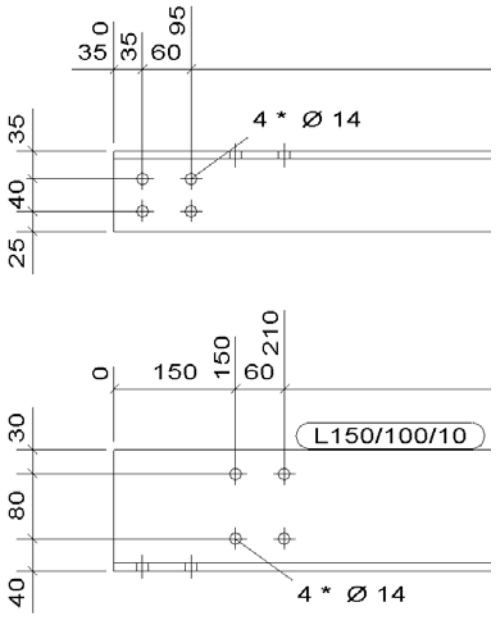
Below an example of an **IPE400**. This dimension is dimensioned according to line:

1, -1.0, TRUE*, FALSE, FALSE, TRUE*, FALSE, FALSE

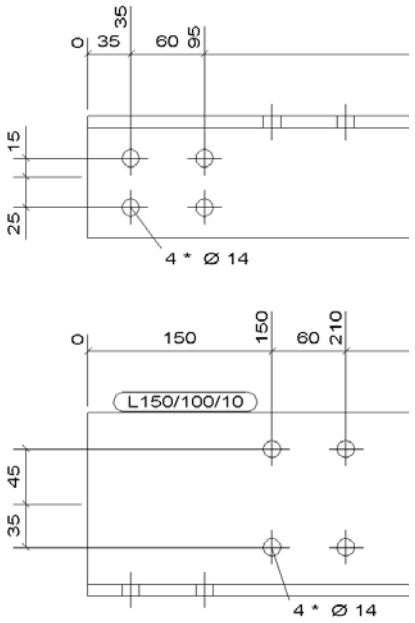


Below an example of a **L100/10**. This dimension is dimensioned according to line:

2, -1.0, FALSE, TRUE*, FALSE, FALSE, TRUE*, FALSE

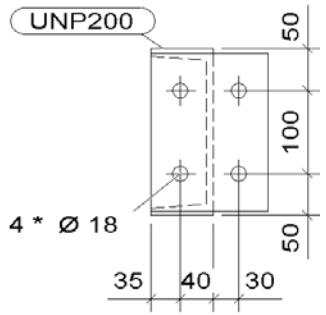


2, -1.0, TRUE*, FALSE, FALSE, TRUE*, FALSE, FALSE



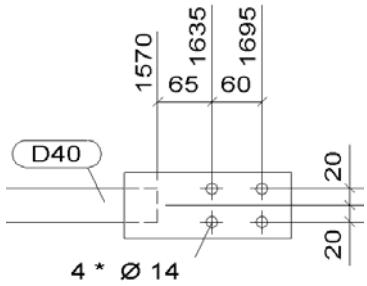
Below an example of an **UNP200**. This dimension is dimensioned according to line:

4, -1.0, FALSE, TRUE*, FALSE, FALSE, TRUE*, TRUE



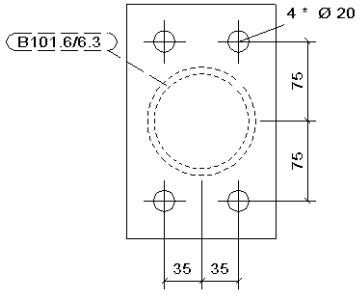
Below an example of a **D40**. This profile is dimensioned according to line:

6, -1.0, TRUE*, FALSE, FALSE, TRUE*, FALSE, FALSE



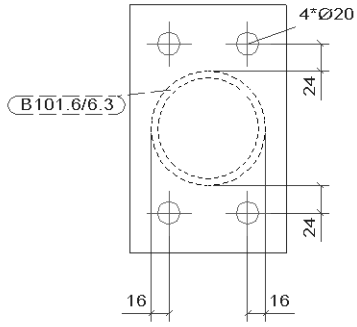
Below an example of a **B101.6/6.3**. This profile is dimensioned according to line:

7, -1.0, TRUE*, FALSE, FALSE, TRUE*, FALSE, FALSE



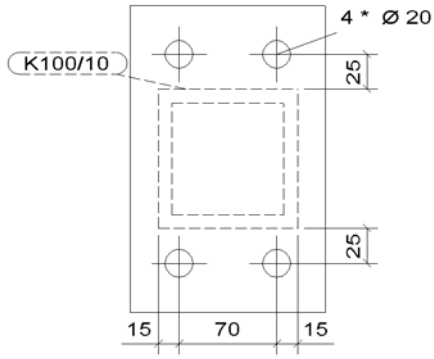
or according to line:

7, -1.0, FALSE, TRUE*, TRUE, FALSE, TRUE*, TRUE



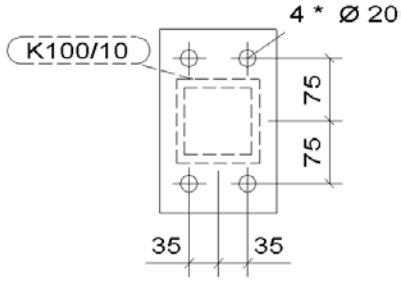
Below an example of a **K100/10**. This profile is dimensioned according to line:

8, -1.0, FALSE, TRUE*, TRUE, FALSE, TRUE*, TRUE



or according to line:

8, -1.0, TRUE*, FALSE, FALSE, TRUE*, FALSE, FALSE



3.12 DSTV2DXF

Topics

This chapter is divided into following sections:

Copying toolbars and shortkeys (p.126)

Settings in construsoft.def (p.135)

Environment variables (p.135)

Text settings (p.138)

Colors and layers (p.140)

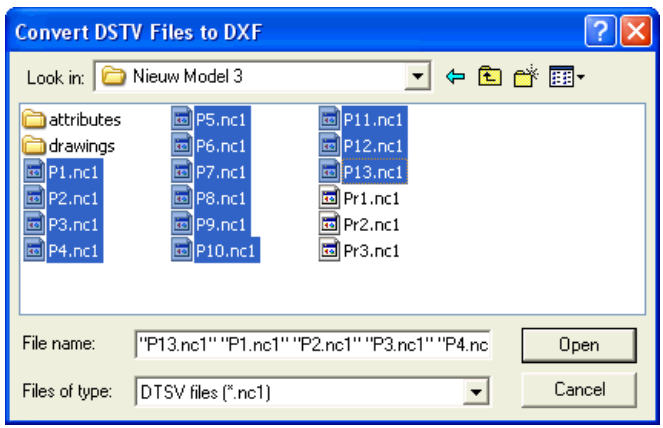
This programme converts dstv files to dxf files, so it creates DXF-files from Tekla Structures NC data, to drive plasma cutters and other cutting devices.

In the dxf files all superfluous information, like dimension lines, is removed. Only the profile (cutting) contour, the position of the holes and text which can be adjusted is in the file.

The dxf file can contain only one side of the profile. That's also why this programme is best suited for plates.

Step plan

1. In Tekla Structures, create NC files, ***File > CNC > NC files***. These NC files have the extension **.nc1* and are stored in the current model folder.
2. Go to the folder *disk:\TeklaStructures\version\nt\dstv2dxf* and double-click on the file *dstv.bat*. A browser will open. Leaf through the Tekla Structures models in which the nc files are stored. Select the nc files and click **Open.**



3. The nc files from the Tekla Structures model are now converted to dxf files in the folder *C:\TeklaStructures\10.1\nt\dstv2dxf\dxf*.

   If Tekla Structures is not installed on drive-C, adjust this folder at INPUT_FILE_DIR en OUTPUT_FILE_DIR in the file *construsoft.def*.

Settings in construsoft.def

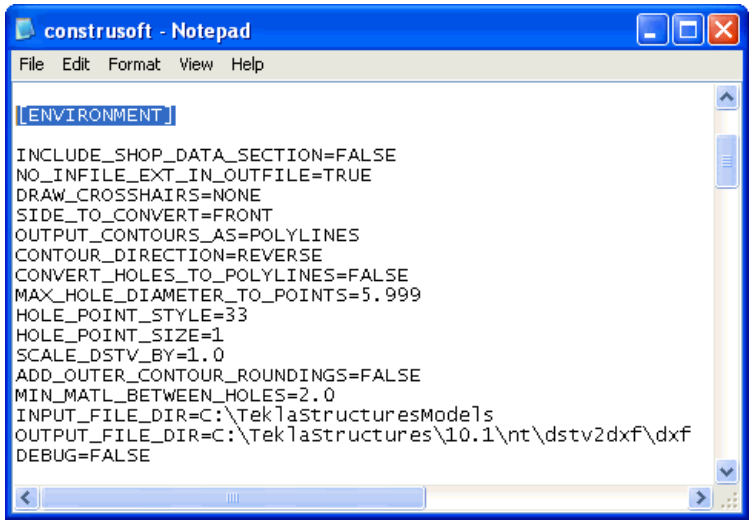
Many settings can be defined by a user in the file *construsoft.def*, such as: the colors and layers of plates and holes of the dxf file and the height and the color of the desired printed text.

These settings are defined in the file *construsoft.def* in the folder:

disk:\TeklaStructures\version\nt\dstv2dxf.

Use a text editor to open this file (e.g. Notepad, Wordpad)

Environment variables



Include shop data section

Include SDS Data in the DXF file. Use to specify if you want to include a special data section in the DXF file to allow the DXF file to be more usefully imported into CNC software written by Shop Data Systems. Please note that including this special data section in the DXF file makes the DXF file unreadable by AutoCAD.

Options: TRUE or FALSE - *Standard*: FALSE

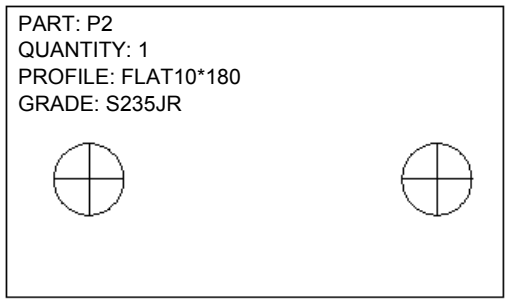
No infile ext in outfile

Use to leave input file extension ”dxf” to the converted output file. E.g. file *P2.nc1* results in *P2.nc1*.dxf if this value is set to TRUE.

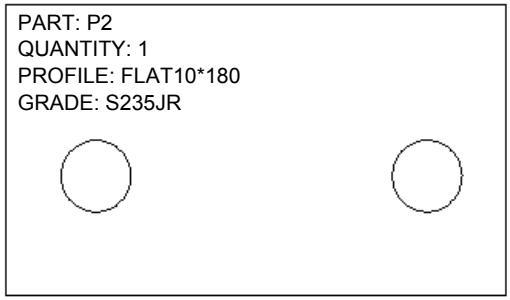
Options: TRUE or FALSE - *Standard*: TRUE

Draw crosshairs

Draw crosshairs for holes and slotted holes.



Crosshairs



No crosshairs

Options: HOLES, LONG_HOLES, BOTH, NONE - *Standard*: NONE

Side to convert

Define which side of the member to convert. Only one side can be created. That's why this programme is best suited for plates, but you can also create a dxf file of one side of a beam.

Options: FRONT, TOP, BACK or BELOW - *Standard:* FRONT

Output contour as

Convert contours as polylines or lines and arcs.

Options: POLYLINES or LINES_ARCS - *Standard*: POLYLINES

Contour direction

Define the contour direction.

Options: REVERSE or FORWARD - *Standard*: REVERSE

Convert holes to polylines

Convert holes to polylines.

Options: TRUE or FALSE - *Standard:* FALSE

Max hole diameter to points

Convert small holes to points in DXF file.

Options: <value> - *Standard*: 4.99

Hole point style & Hole point size

Point style and size for holes.

Standard: respective 33 and 1 (do not change these values!)

Scale dstv by

Use 0.03937 for scale to imperial units. Use 1.0 for scale to metric units

Standard: 1

Add outer contour rounding

Add holes to roundings.

Options: TRUE or FALSE - *Standard:* FALSE

Min matl between holes

Define how close holes can be in slotted hole conversion.

Options: <value> - *Standard*: 2.0

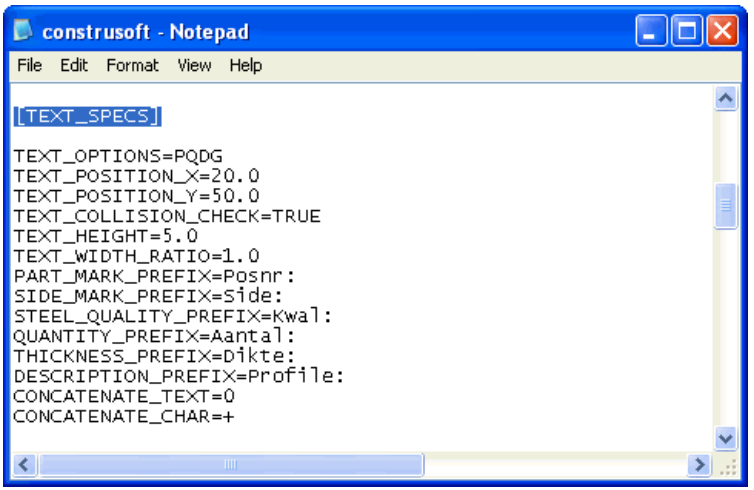
Input file dir & Output file dir

Directories for input and output files.

Debug

Show processing data on DOS window.

Options: TRUE or FALSE - *Standard:* FALSE



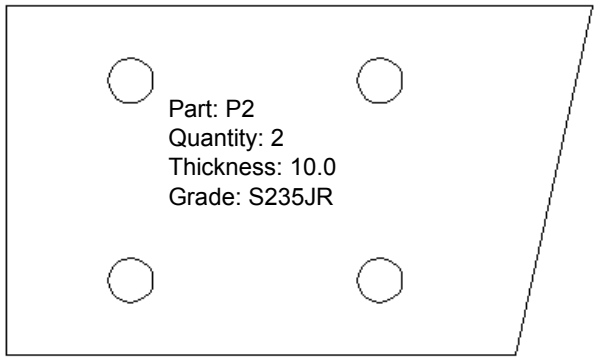
The chapter [TEXT_SPECS] in the file *construsoft.def* contains many lines which affect the text on dxf files.

Text options

Define the text options you want in the DXF file after TEXT_OPTIONS. You have the following options:

- S adds a side mark
- P adds the part mark
- B adds the part mark and side mark
- Q adds the quantity
- G adds the steel grade
- T adds the thickness
- D adds the profile description

PQTG is set by default, which results in Part mark, Quantity, Thickness and Grade:

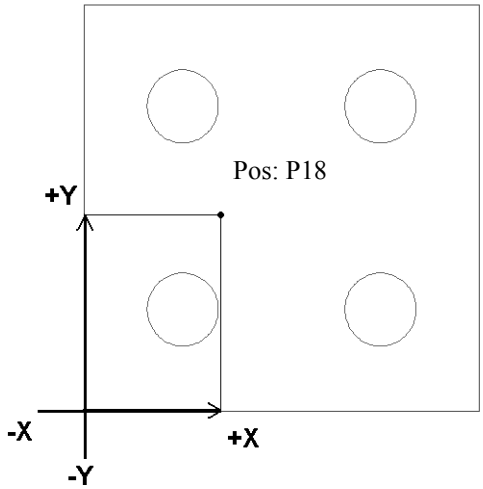


Text position X, Y

In case plates are nested, the text must be added to the plates. You can affect the location of the text:

- TEXT_POSITION_X=20.0
- TEXT_POSITION_Y=50.0

(X/Y location of lower left corner of first line of text, from the origin point <0,0> of the DXF file).



Text height

Define the height of the text.

DSTV2DXF_TEXT_SIZE=5

Text width ratio

Text height units are millimeters. Width ratio is a percentage of text height.

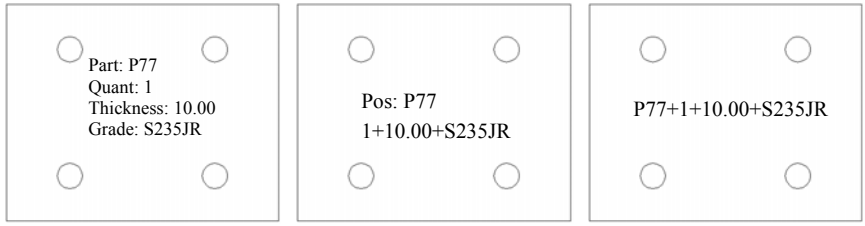
Prefix

Several prefixes for text items, such as *Part mark* and *Thickness*.

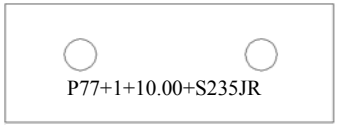
Concatenate Text & Char

The line DSTV2DXF_CONCATENATE_TEXT combines text items (part mark, quantity, thickness, grade) into one or two lines. Values:

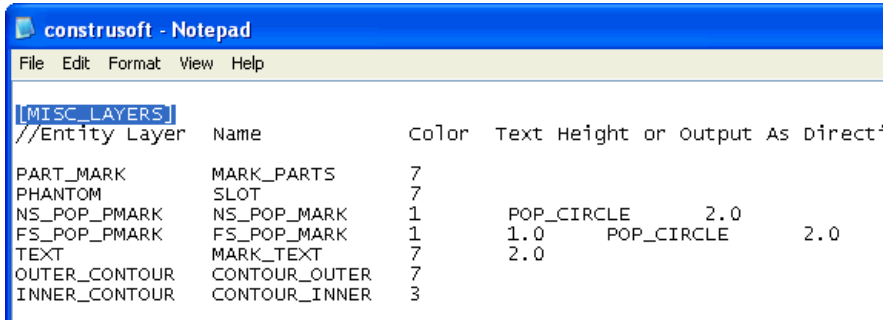
- **=0**: Text lines not combined.
- **=1**: Part mark text on one line, other texts combined on another line.
- **=2**: All text on one line.



The line CONCATENATE_CHAR=+ defines a separator (max 19 characters) for the text items.



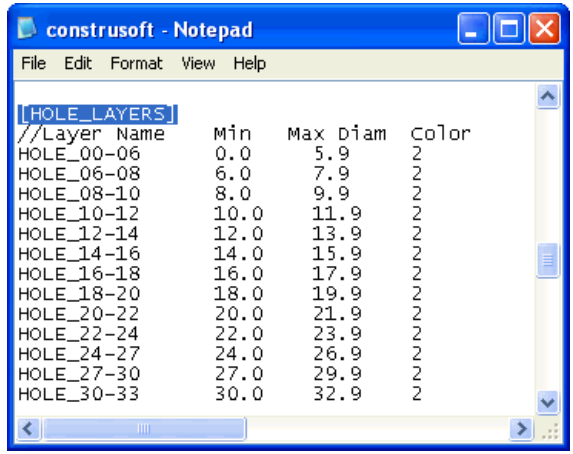
Colors and layers

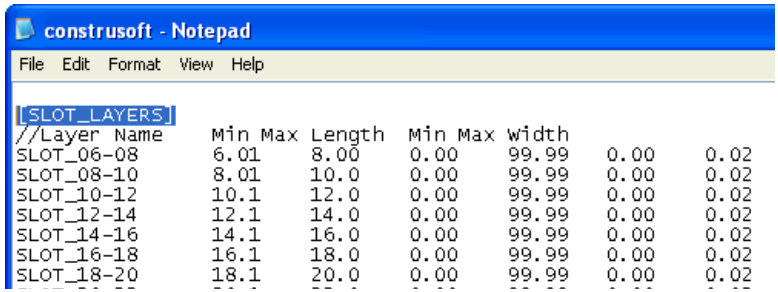


Define in chapter [MISC_LAYERS] the names and the colors of the parts in the dxf file, e.g. the outer contour of the plate, the inner contour and the text.

The adjustable colors correspond to the standard Autocad colors. (1=red, 2=yellow, 3=green, 4=blue, 5=cyan, 6=magenta, 7=white)

Define in chapter [HOLE_LAYERS] and [SLOT_LAYERS] per diameter the layers and the colors for holes and slotted holes in the same way.





3.13 DWG/DXF

Topics

This chapter is divided into following sections:

Reference model in the Model editor (p.142)

Reference model in the Drawing editor (p.146)

Importing 2D and 3D dwg/dxf files (p.147)

Exporting Tekla Structures models (p.148)

Dwg/dxf import into drawings (p.150)

Export of drawings (p.151)

DXF import in Template Editor 2.x (p.163)

DXF import in Template Editor 3.x (p.164)

Dwg/dxf logo's in the drawing layout (p.164)

Tekla Structures offers several possibilities regarding importing and exporting dwg/dxf files, as well in the model environment as in the drawing environment.

Dxf is an universal accepted file type, which is supported by most 2D and 3D application. The most well known application is AutoCAD. The extension of Acad drawing files is dwg. This document speaks about dwg/dxf files, however these are two different file formats.

This document discusses all dwg/dxf exchange possibilities - also the ones mentioned in previous versions:

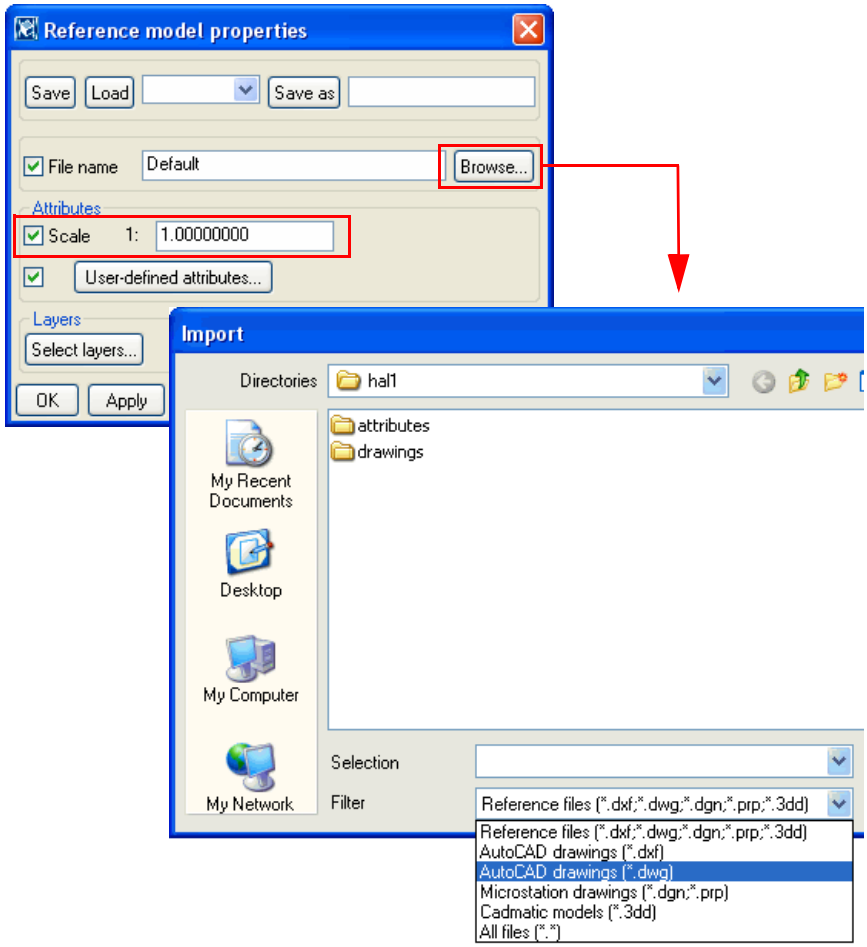
Reference model in the Model editor

In case there are digital drawings available from third parties (e.g. architect or builder) in dwg or dxf format, it can be useful to insert these drawings as a 'place-mat' into the Tekla Structures model. It is possible to snap to the reference models' lines, which allows a fast generation of the structure.

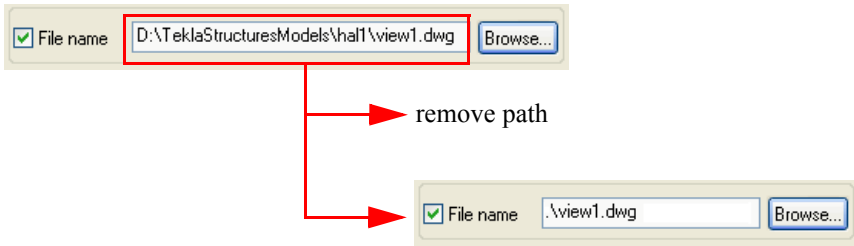


In toolbar *Profiles*, click the button <**Create reference model**> to import a dwg or dxf file, or go to **Parts > Reference model**.

With the <**Browse**> button, the file can be searched. The Filter list box enables to filter for a specific file type; *dwg*, *dxf* or *dgn*. Also a scaling factor can be set for the file.



As the reference model creates a link to the dwg/dxf file, it is advisable to store the dwg/dxf file(s) in the Tekla Structures model folder before the importing process taking place. This provides removing the path towards the file, which facilitates viewing the reference object in case the model is send to a third party.



Another thing to take care of is the files origin (the point with coordinates 0,0,0). This point coincides with the picked point in Tekla Structures when inserting the reference model. Moving the reference model is rather easy in Tekla Structures.

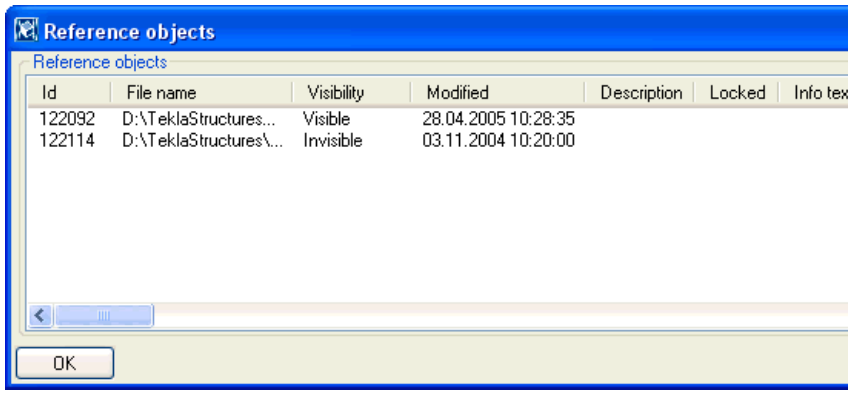
The easiest way to insert a reference model, is inserting it in a plane. Preferably a top view or foundation plan view. It may happen that the reference model is not visible straight away. If this is the case, than check:

- if the work plane is equal to the view plane
- if the scale of the dwg/dxf is set correctly
- if the intended origin van de dwg/dxf is not out of bound
- if 'Reference models' is checked in dialog box **View Properties** > **Display**
- if the work area is updated

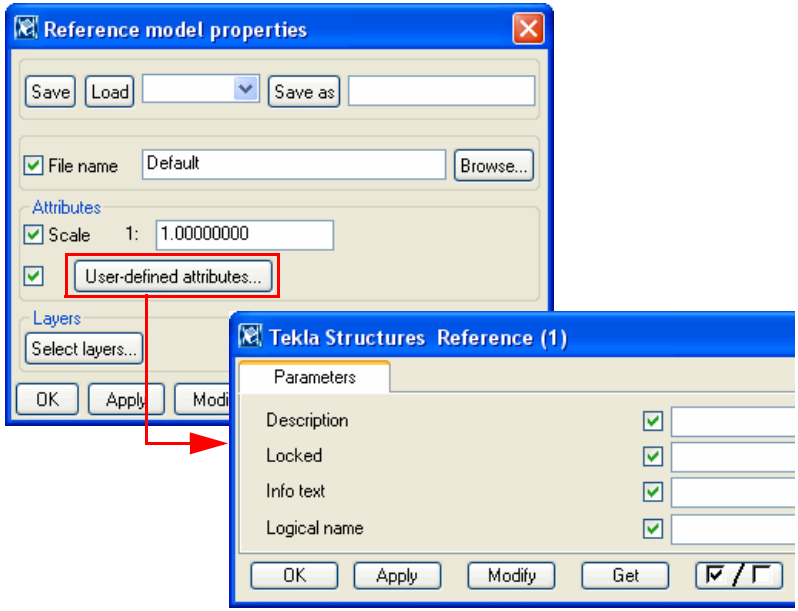
In wire frame views, the reference model is not represented. In rendered view, the lines have the same color as in the original dwg/dxf.

In case there are multiple reference models in a model, each reference model can be hidden/ made visible individually. Also User Defined Attributes can be assigned to reference models, which facilitates recognition of reference models in lists, etc.

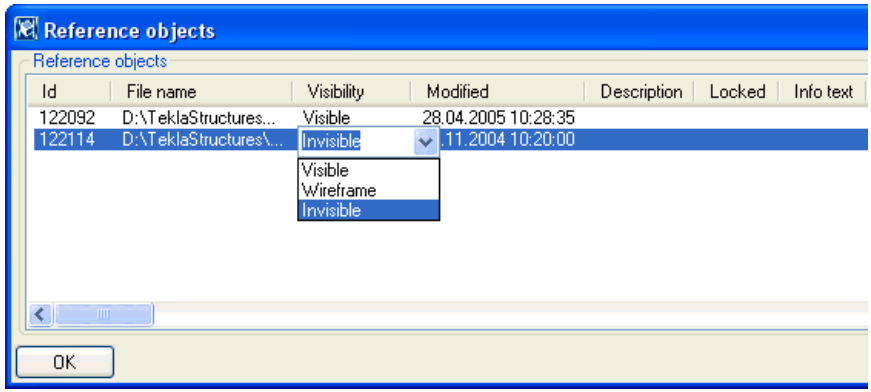
To open de dialog box, go to **File > Reference model**. The following dialog box will appear:



By selecting a line in this dialog box, the corresponding object will highlight in the model. The other way around, selecting a reference model in the model view will highlight a line in the dialog box. The possible entered text for User defined attributes are filled in columns *Description*, *Name* and *Info text*.



The visibility of the several reference models can be set in column *Visibility*. By clicking the column *Visibility*, a list box will be displayed.



Reference model in the Drawing editor

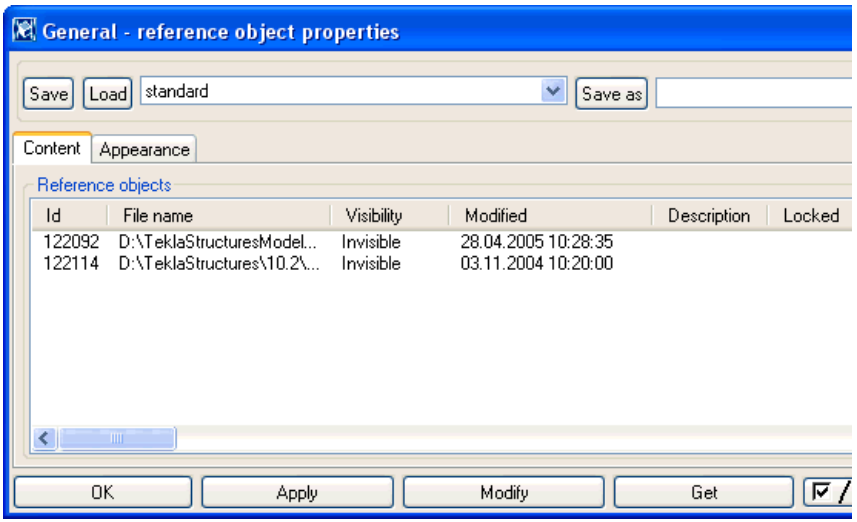
Reference models can be displayed in general arrangement drawings. U can use third parties drawings (e.g. architect or builder) as a reference model. You can set this on drawing- and view level.

Drawing level

To display reference models and to change the visibility:

- Open the dialog box **General arrangement drawing properties**
- Click the button **Reference objects..**.

Tekla Structures opens the dialog box **General - reference object properties**. The tab **Content** shows a list with all reference models in the model:



- Select **Visible** in the row **Visibility** to display the selected reference model in the drawing.
- Go to the tab **Appearance** to set the color and the line type for all reference models.

View level

Use the dialog boxes **View properties** and **View reference objects properties** to display reference models in drawing views, independent from the drawing properties.

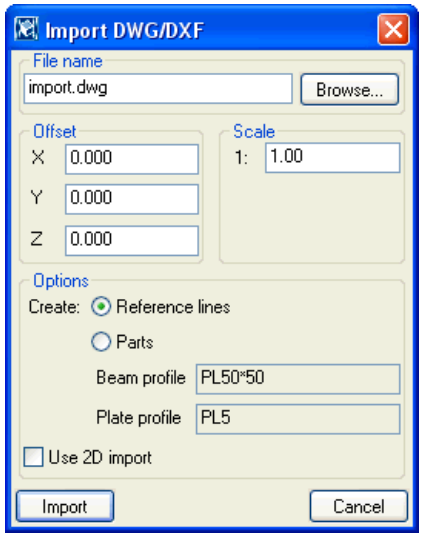
Importing 2D and 3D dwg/dxf files

Dwg/dxf files can be imported in a Tekla Structures model. The lines are displayed as construction lines.

In relation to the reference model, the disadvantage of dwg/dxf files is that the Tekla Structures model can become very heavy and slow.

The main advantage of importing is when profiles are almost equal. In that case all lines in the dwg/dxf can be converted to a certain profile. You could also select this option when a dwg/dxf is three-dimensional (3D).

Go to **File > Import > DWG/DXF**. The dialog box below appears.



By pressing the <**Browse>** button, a search dialog box will open to loop up the file.

The X-, Y-, and Z-values which can be entered for *Origin*, determine the translation from the dwg/dxf origin to the Tekla Structures models' origin.

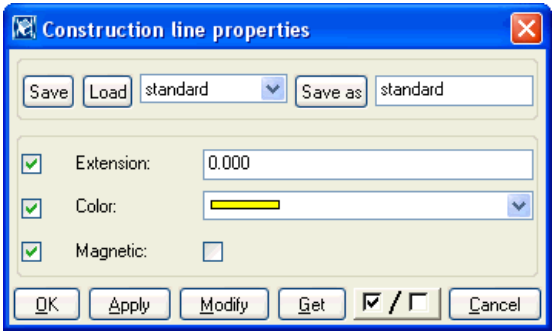
Also a scaling factor can be set. In the *Options* frame can determined if lines have to be converted to reference lines (the yellow construction lines) or to a profile.

The option *Use 2D import* can be checked in case of 3d dwg/dxf files which needs to be flattened imported into the model.

The layer structure doesn't have a negative effect on the Tekla Structures model. All lines will be displayed, irrespective of color and layer, and whether they are locked, disabled and or frozen.

When the dxf/dwg file is imported, you can change the line color (in fact these are construction lines).

Select the lines, hold down the <**Ctrl**> key and double-click one of the lines, the following dialog box appears:



Select a color and click <**Modify**>.

However, some objects will not be displayed; this depends if the object is loaded in a 2d view or in a 3d view.

See the table below for details:

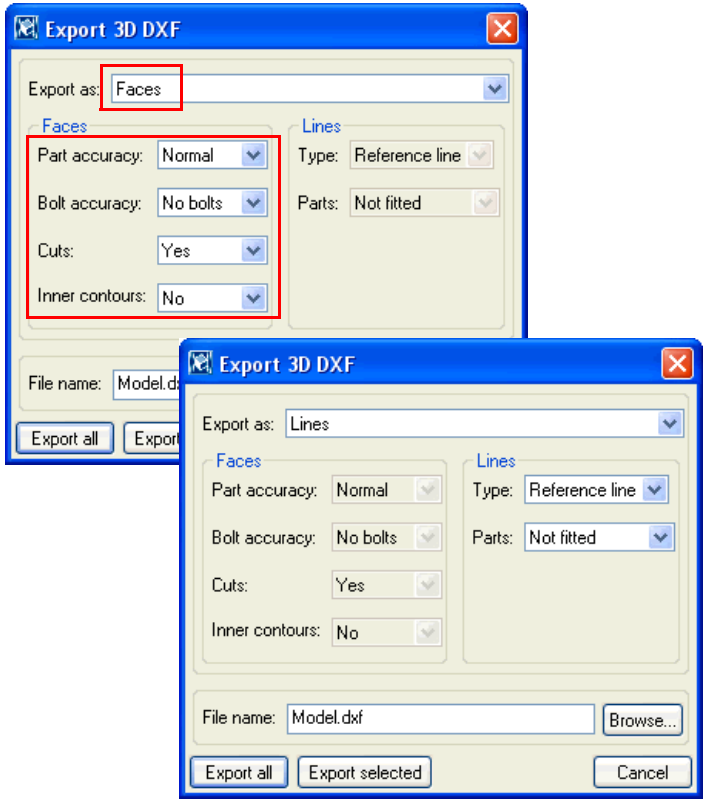
	2d	3d
(p)line	+	+
cirkel	+	+
arc	+	+/-
elips	+	-
tekst	+	-
arcering	+	-
maatvoering	+	-

Note: In 3d views, arcs are displayed as bend lines.

Exporting Tekla Structures models

Exporting Tekla Structures models can be done in various ways. In this document, we limit our selves to Dxf export. With this feature, Tekla Structures profiles can be exported to the dxf format. The file can be read in AutoCAD (among other programs). The 3D properties are retained during export.

Go to **File > Export > 3D DXF**. The following dialog box will appear:



Tekla Structures parts can be exported in 2 formats: *faces* and *lines*.

- If *faces*-export is used, the profiles are exported in it's entirely, in other words, all lines that are visible in the Tekla Structures model, will be visible in the dxf. However, profiles are exported as 'blocks'.
- If *Lines*-export is chosen, the profile are exported as a single line. The remainder options, like accuracy, cuts, inner contours, etc., speak for themselves. Take into account that every option affect the size of the export file.

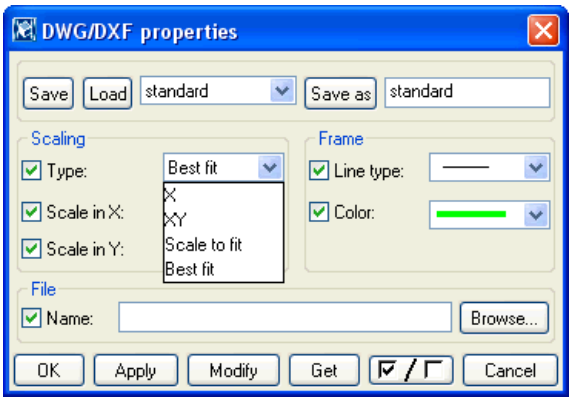
In the *Filename* input field, the name of the export file can be entered. Note that the file extension has to be added to the filename. By default, the file is stored in the model folder. By pressing the <**Browse**> button, an alternative location can be selected.

The last option is determining what has to be exported, the entire model or just a selection of it. This is controlled with the <**Export all**> and <**Export selected**> buttons.

Dwg/dxf import into drawings

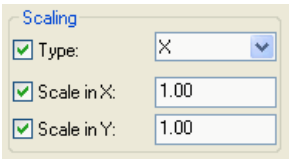
Dwg/dxf graphical objects can be added to Tekla Structures drawings. Dwg/dxf files are most used as details from third parties (architects, builders, etc.).

In the drawing environment, go to **> Properties > Dwg/dxf**.



By pressing the <**Browse**> button, the desired dwg/dxf file can be selected. A link to the file is being created. Therefore, the best way is to store the dwg/dxf file in the model folder, and remove the file path afterwards. In the 'Frame'-frame, the line type and line color of the frame can be set. If no frame is needed, set the color to black. At 'Scaling', 4 scaling types are present.: <X>, <XY>, <Scale to fit> and <Best Fit>.

- **X**: The value set for X is determinative. A possible value for Y will be ignored



- **XY**: The width and height (resp. X and Y) of the dwg/dxf are determined by the values for input fields 'Scale in X' and 'Scale in Y'.
- **Scale to Fit**: The scale of the dwg/dxf depends on the size of the frame around the dwg/dxf. However, the initial width/height ratio stays intact.
- **Best Fit**: The height and width of the dwg/dxf depends on the size of the frame surrounding the dwg/dxf. The initial width/height ratio can be distorted.

After filling in the dialog box *DWG/DXF properties*, click <**Modify**> and <**OK>**. Next, go to **Create > Dwg/dxf**. Depending on the scaling settings, the statusbar indicates what to do.

If *Scale to Fit* or *Best Fi*t is chosen, than a frame has to be created on the drawing by picking two points.

Note: the *Snap to any position* in the *Snap settings*-toolbar has to be enabled! If *Scaling* is set to 'X' or 'XY', only one point has to be picked on the drawing. The insertion point is the top-left corner of the dwg/dxf file.

All common drawing objects like line, p-line, circle, ellipse etc. displayed on a drawing. Also hatches and blocks are displayed. Layer properties (on/off, frozen, locked) are taken into account, i.e. a layer that is disabled will not be displayed.

Bear in mind the following deviations:

Objects drawn in AutoCAD in layer *defpoints* are displayed in the Tekla Structures plot, unlike the Acad plots. Specific line types (like zigzag and dashed) can nor be converted and therefore not be displayed in Tekla Structures drawings. The named Acad colors (red, yellow, green, etc.) are 'translated' to homonymous colors in Tekla Structures. Numbered AutoCAD colors (8 through 250) are converted to one of the 14 available Tekla Structures colors. The grey colors 251 through 256 are plotted grey on Tekla Structures drawings. However, the line thickness of these colors can not be defined, so the lines are plotted very thick. So it is recommended to avoid these colors in dwg/dxf files.

Fonts in dwg/dxf files are converted to a font specified in the Tekla Structures batch file. The default font is Arial. If the dwg/dxf file contains other fonts that have to be displayed on the Tekla Structures drawing, than the so called shape file (=Acad font) of this font needs to be copied to the Tekla Structures model folder (shape files have the extension *.shx and can be copied from Acads sub folder 'Fonts'). For example, the round character 'Ø'; for a correct representation on a Tekla Structures drawing, add the symbol in Acad with shortcut "%%c", instead of keyboard shortcut like <Alt> + 0216.

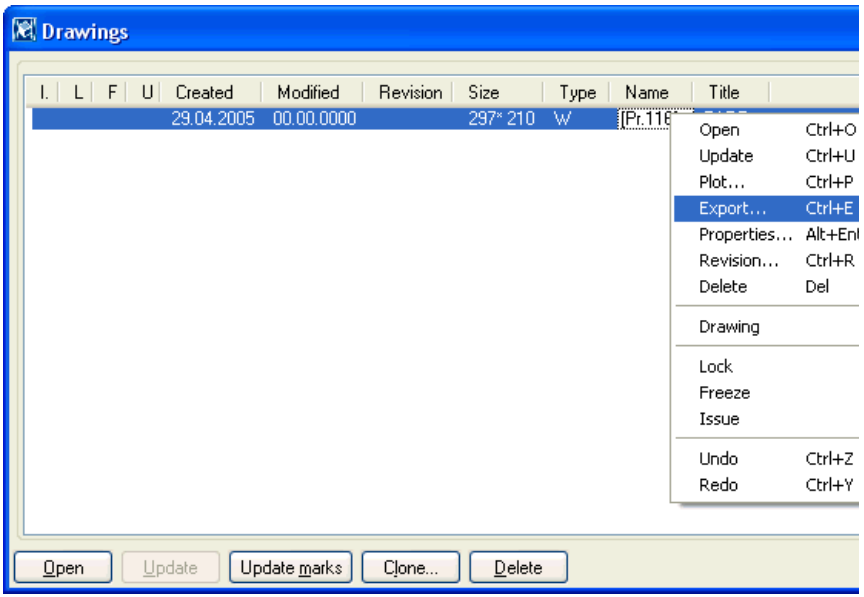
Export of drawings

Tekla Structures drawings can be exported towards dwg/dxf files. These files can be used to send to third parties which do not have Tekla Structures, but have e.g. AutoCAD instead.

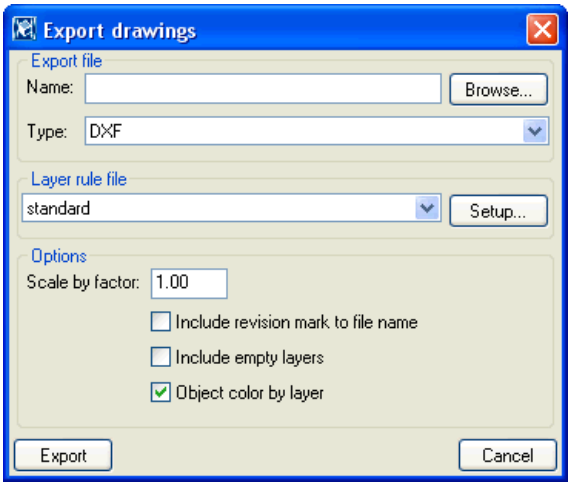
InTekla Structures layers can be defined beforehand for the several objects (profiles, bolts, marks, etc.).

With this the original layer structure can be maintained. Also for possible finishing of drawings the filter and layer options can be used (freezing, layer on/off, etc.).

The <**Export**> command can be activated from the drawing list: right mouse click on the drawing(s), then select 'Export' from the contextmenu.



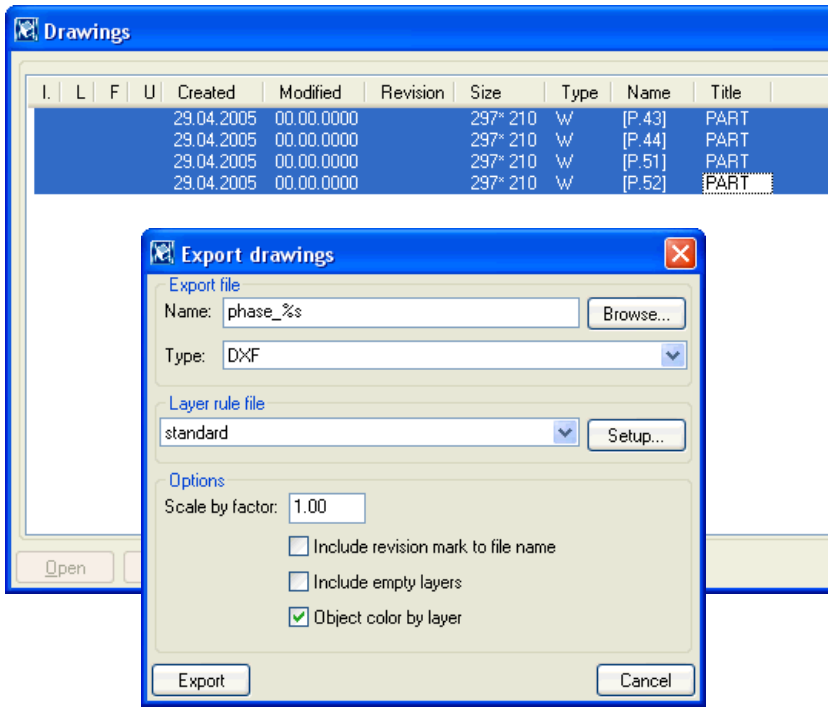
The *Export drawings* dialog box will appear:



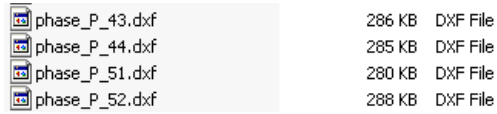
In input field *Name*, a file name may be specified. If this field is left blank, the export file shall get the same name as the Tekla Structures drawing name.

Example: assembly drawing *A.2* will be named *A_2.dxf*. If no path is specified, the file will be stored in the model folder by default.

If multiple drawings have to be exported, and you want to add the phase in the filename, you can use a '%s' in the filename. This string will return the drawing name in the filename. For more clarification, see the pictures below.

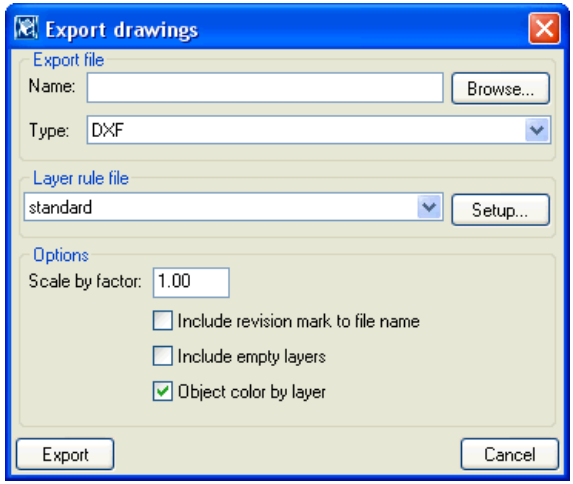


By entering the name as in above picture, in the Windows Explorer the files will look like this.



In the *Export drawings* dialog box, for list box *Type* the file type can be set.

If the exported drawings will be opened in AutoCAD, the file type DWG could be chosen. For further editing in another programme, 'DXF' would be the better option.



At *Layer rule file*, a beforehand saved layer rule file can be loaded and/or the setup of a rule file can be edited/created. This is described in another paragraph.

At *Options*, a scaling factor can be set. The export output will be increased or decreased.

A check can be placed in front of *Include revision mark in file name*. In case a drawing gets a revision, the check makes that the previous drawing won't be overwritten.

Example: drawing *A.5* has received revision number 2; the file name will be *A_5_2.dxf*.

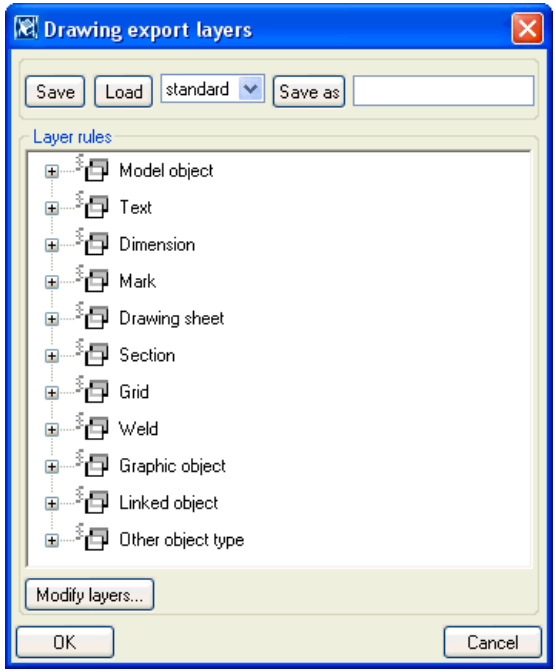
Include empty layers can be checked if you want to export all layers, even if no objects are in a certain layer.

Object color by layer can be checked if you want to define the layer color beforehand. How this works, is described further in this document.

By pressing the <**Export**> button, the export operation will start.

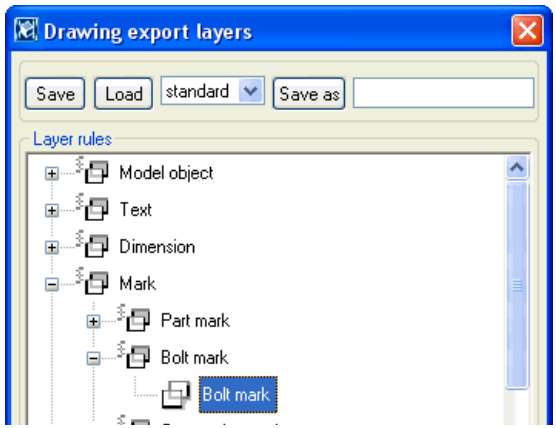
Setup Export Layers

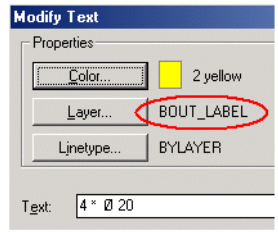
In dialog box *Export drawings*, click <**Setup**>. The dialog box *Drawing Export layers* shall appear.



In this dialog box, every single Tekla Structures drawing object can be allocated to layers. By pressing the '+' in front of an object group, the group will be opened and the layer can be viewed.

In the example below is determined that the bolt marks turn up in the layer *Bolt_mark*. In the opened dwg/dxf beside it is shown.

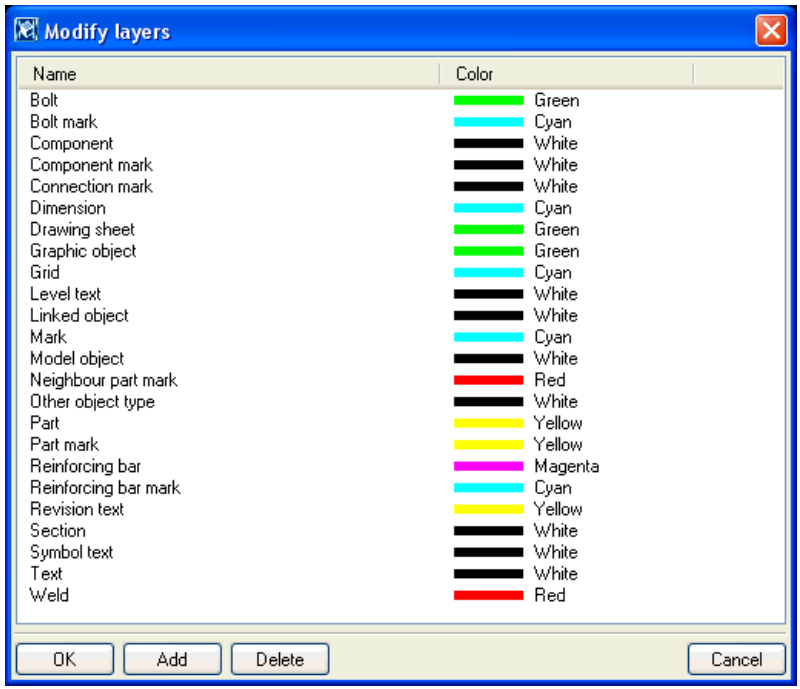




Changing layer names

It may happen that layer names, as set in the default *layer rules*, do not correspond with the layers you or your builder use.

To edit the layer names in dialog box *Drawing export layers*, click <**Modify layers**>. The dialog box below will appear:



Select the layer for which you want to change the name. Remove the text using the backspace key (or mouse) and enter another name. Next, click <**OK**>

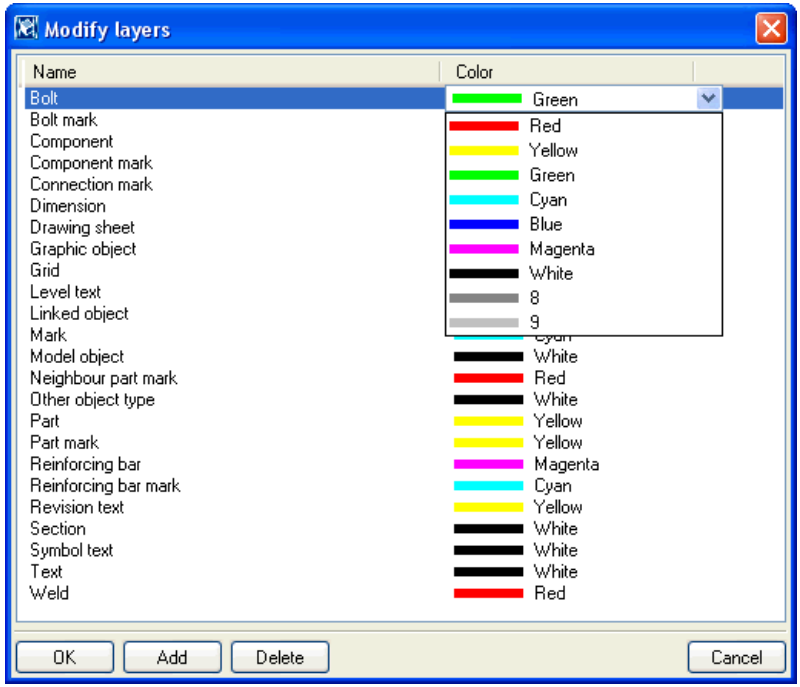
Deleting layer names

It is impossible to delete default layers. If the output needs to be done to a single layer, or only a couple of layers, than you have to change the name of the layers and refer to them, or else it is not possible to check *Include empty layers* in dialog box *Export drawings*.

However, it is possible to delete user defined layers.

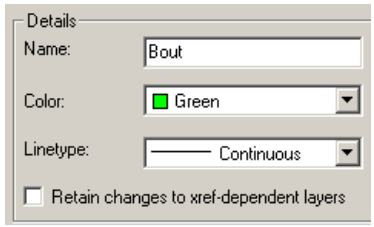
Defining layer colors

In column *Color*, the layer color can be defined. This is the color of the exported file. By default, all layers are set to white. If this setting is left unchanged, than all objects in the dwg/dxf will be whits. By double clicking a color, an alternative color can be selected.

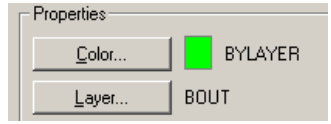


Example: the bolts should be green in the dwg/dxf.

In the dialog box, click *Modify layers*. Than, select Bolt and double-click the color. Change this color to green (see picture above). Finally, click <**OK**>, <**Apply**>, <**OK**> and <**Export**>. In AutoCAD, the layer structure of the dwg/dxf-export looks as follows.



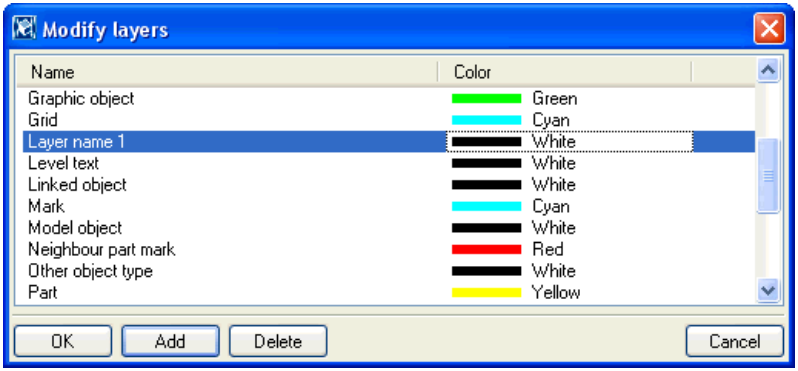
layer properties



object properties

Adding layer names

To add a layer name, in the dialog box *Modify layers*, click <**Add**>. A new layer is added; **Layer name1**.

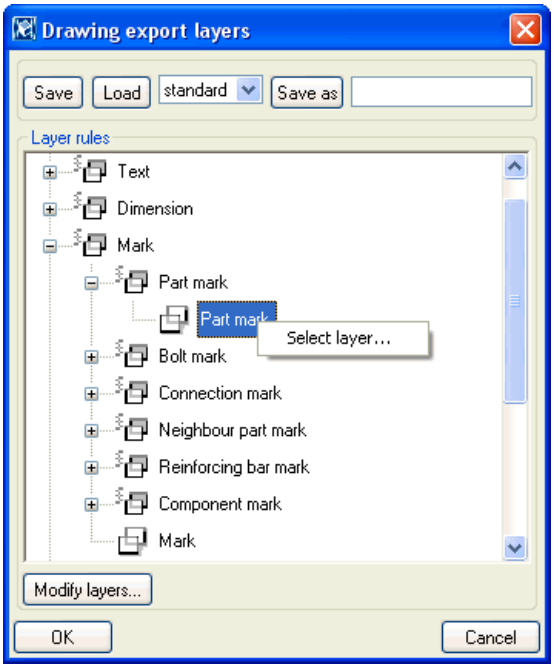


By clicking the name, the name can be changed and a desired color can be chosen.

Assigning a layer name to an object

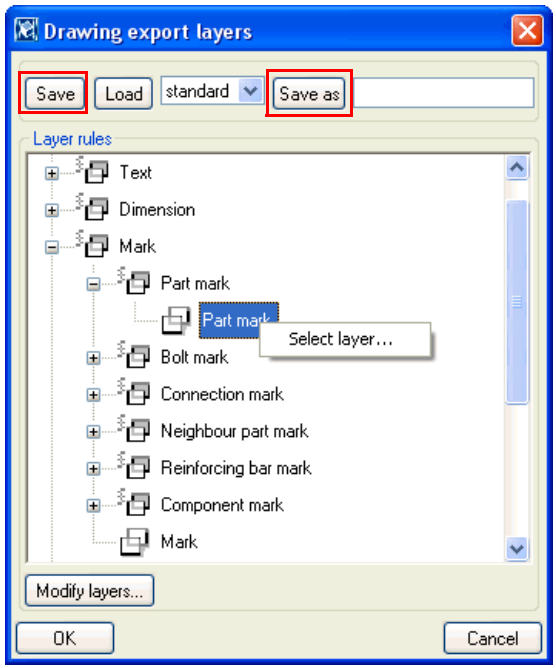
To assign a layer name to an object (e.g. a part mark), do the following:

In the tree structure, browse to the part mark. Here is defined that the *part_mark* layer is applied. By right clicking the *part_mark*, a contextmenu will appear, with option *Select layer*.



By clicking this option, a layer list is displayed. Select the export layer. Next, click <**Apply**> or <**OK**>.

To actually carry through this modification, the standard setting needs to be saved, or a new setting has to be created using the <**Save as**> button.

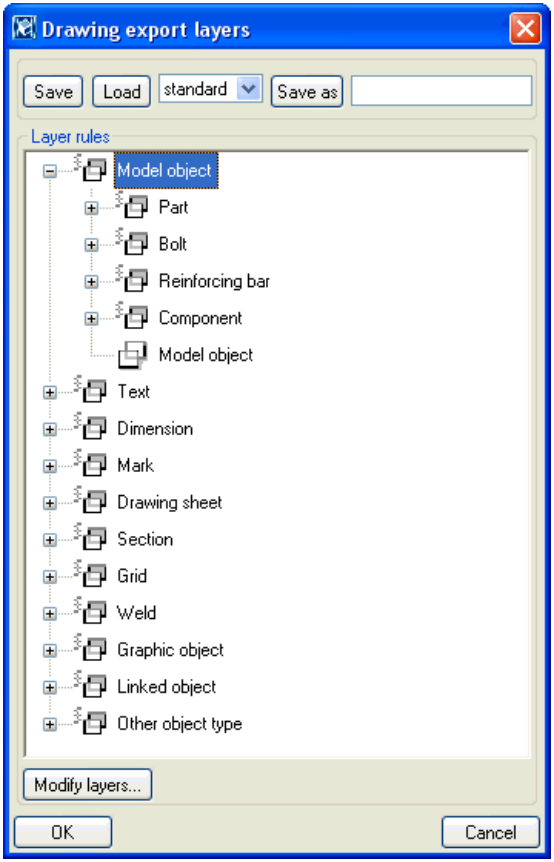




Don’t forget to click <**Save**> or <**Save as**> before closing the dialog box 'Drawing export layers'.

Adding rules using Layer manager rules

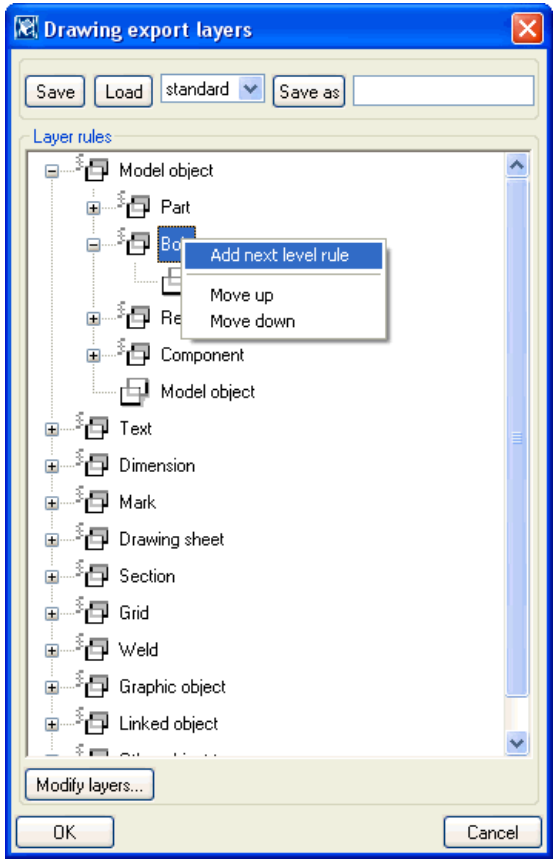
To the main rules *Text through Other object* type, no (sub) rules can be added. This is only possible in top level rule *Model object*.



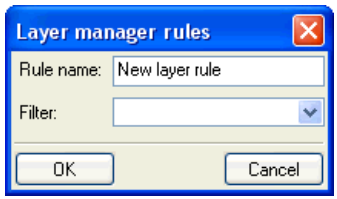
The 'branch' *Model object* is subdivided into groups *part* through *component*. In these rules, several subdivisions can be added.

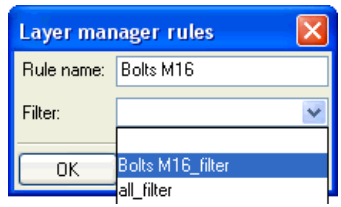
For example, all bolts are put in layer *bolts* by default.

Suppose you want to subdivide the bolts per diameter. First, create a model filter for the various diameters. Next, right mouse click on rule name *bolts*. Select option *Add next level rule*.

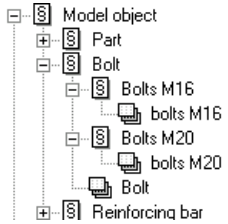


The dialog box *Layer manager rules* appears:

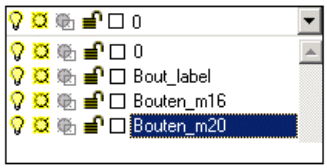


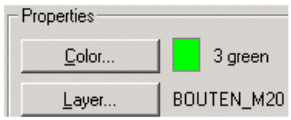


For *Rule name*, type the desired rule name. Next, for list box *Filter*, select the appropriate model filer. Than select the layer name (create layer at first!; see Adding layer names). In this way, a complete tree structure can be made for bolt diameters.

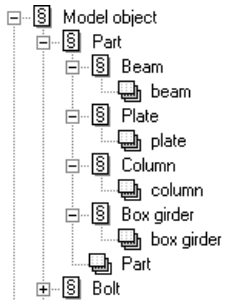


The output (e.g. AutoCAD) looks as follows:



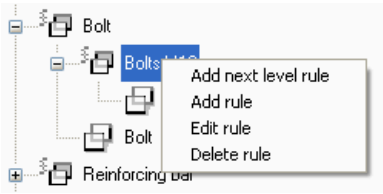


The same can be done for parts or other objects. Everything according your own liking/requirements, or your customers' requirements of course.



In the tree structure above, the named parts (beam, compression bar, plate, column) are placed in their assigned layer name. The remaining parts, which are not defined in the rule, are placed in the layer name 'part'. (de latter layer name in the example above)

Editing, deleting rules can be done by clicking right mouse on a rule, and select the desired command from the contextmenu.



Font type conversion

In the batch file is a font type defined for dxf/dwg export.

set XS_DXF_FONT_NAME=standard

If one uses more than one font type on a Tekla Structures drawing and these font type have to be exported as well, then this can be defined in file dxf_fonts.cnv in folder :\TeklaStructures\versie\environments\country-independent\fonts. In this file (to open with a text editor) one can write the font type in name in Tekla Structures at the left and the font type to use in i.e. Acad at the right.

DXF import in Template Editor 2.x

If you want to put a company logo into your template, a possible way to do this is by importing this logo as dxf file.

In the template editor, go to **File** > **Import DXF**. Browse to the dxf file and select the file.

There are some things that have to be taken into account before importing the dxf:

- The dxf-file must be saved as a Acad R12 file.
- The logo must be drawn on the correct scale, and the left bottom corner must be positioned on the origin.
- Take in mind that hatches and blocks will not be imported in the Template Editor. These objects have to be exploded beforehand.
- Bend lines (circles, arcs, etc.) in the dxf-file are not imported. These objects have to be replaced by lines beforehand.
- Reckon with colors that may be different from the original dxf after importing the dxf file. See the table below.

DXF	Template
white	magenta
red	white
yellow	red
green	green
cyan	blue
blue	cyan
magenta	yellow
grey	black

DXF import in Template Editor 3.x

All conditions regarding dxf import in template editor 2.x are not needed anymore when importing a dxf/dwg in template editor 3.x.

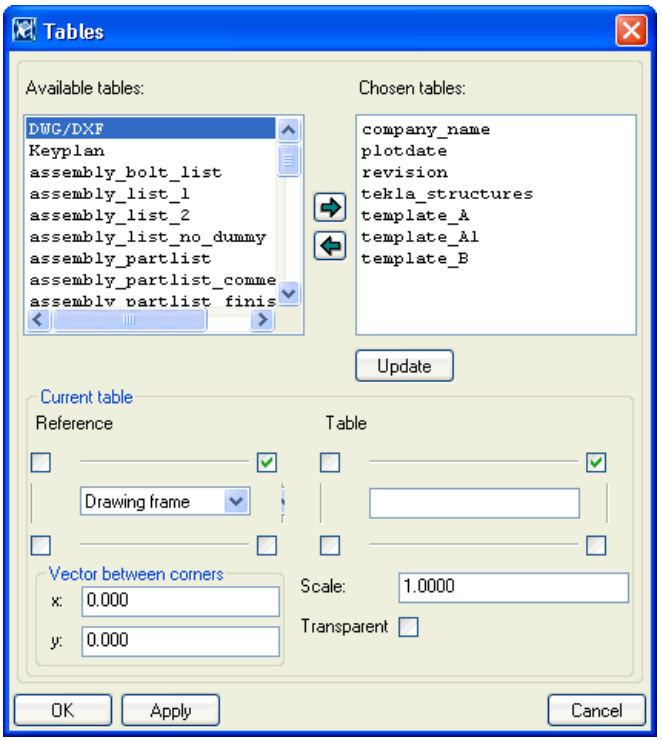
For more information about how to import a file we refer to document Template Editor, see *Converteer een AutoCAD Bestand naar een Template op pagina 39*

Dwg/dxf logo's in the drawing layout

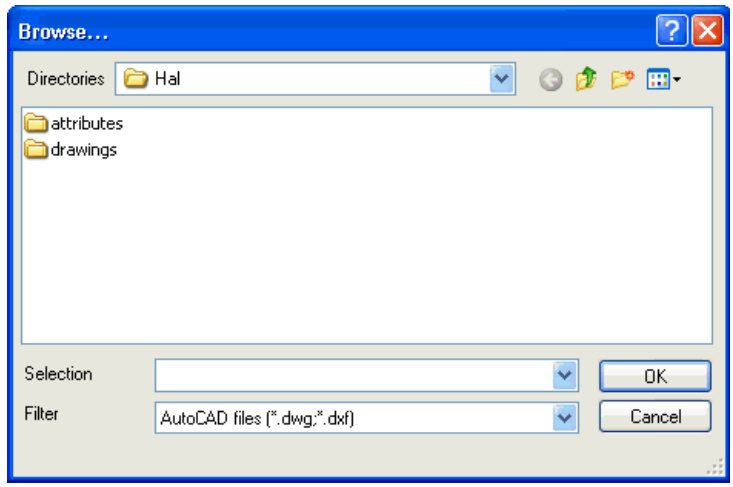
When you have a company logo created as a dwg or dxf file, you can add this file directly to a template.

Also third party drawings (principals/architects) can be provided with the right template quick and easily.

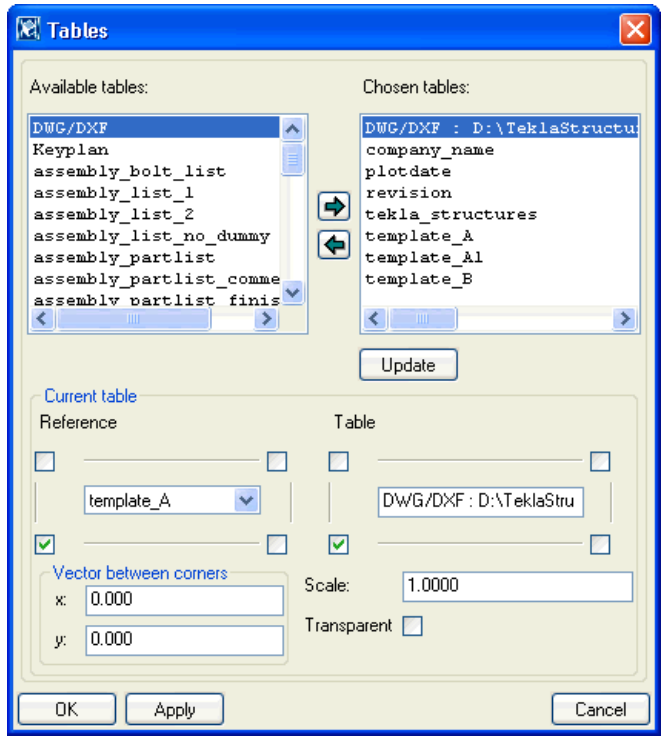
Go to **Properties** > **Layout** and select the template you want to add the dwg/dxf file to.



In *Available tables* select the option <DWG/DXF> and add this template to the *Chosen tables* using the green arrow in the middle. A browse dialog box appears, where the dxf/dwg file can be located:



The dwg/dxf file is now added to the *Chosen tables*. Use the reference check boxes - and possibly a scale factor and/or an offset distances- to position the dwg/dxf on the drawing



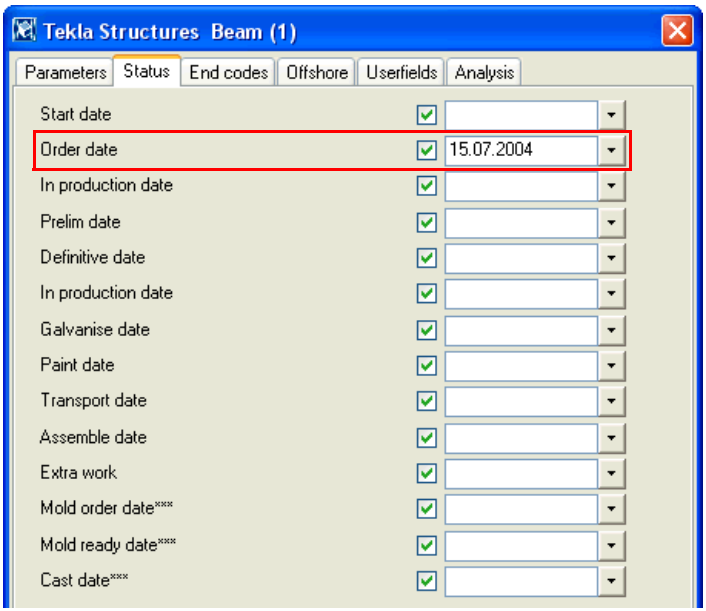
3.14 Filter by user attributes

In Tekla Structures it is possible to filter by user attributes.

You can for instance add a date to a beam in the user attributes. Go to **> *Properties beam* > *User defined attributes* >** tab ***Status.*** You enter several dates such as Start date, Order date, etc.

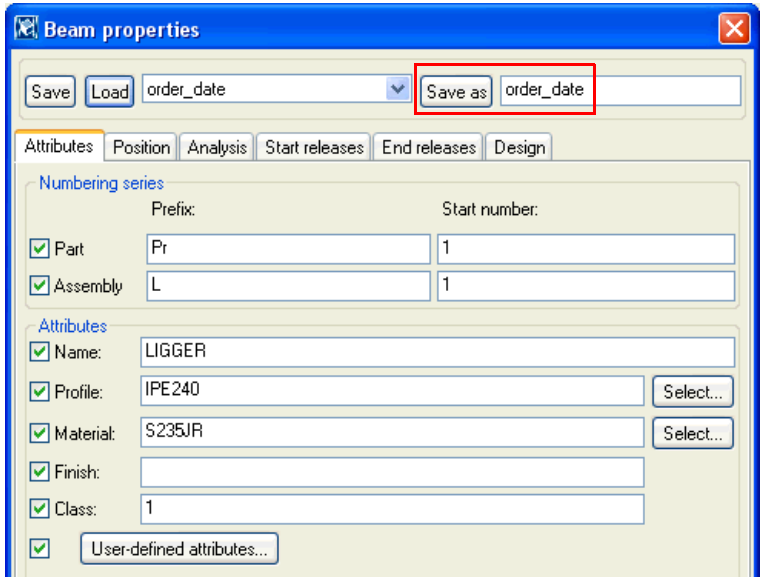
Later on it could be handy to filter by one or more of these dates. On the basis of an example this document explains how to do that.

E.g. for a beam an *Order date* is added in the user attributes. See following dialog box:



This beam now contains the order date 15.07.2004. If you want to filter all parts containing that order date, do the following:

Save this setting (containing order date 15.07.2004) in the beam properties. For instance the name ***order_date.*** See following picture:

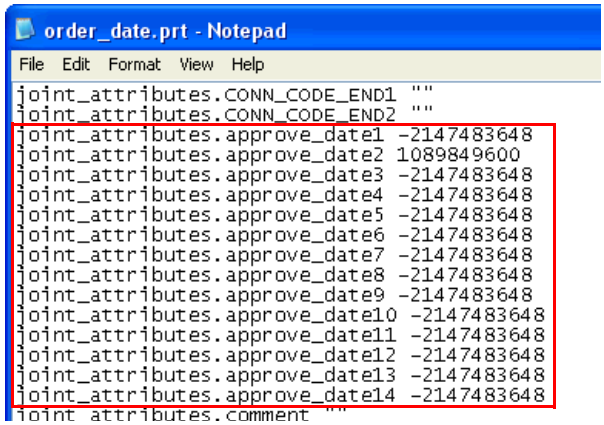


This setting *order_date* is stored in the model folder.

Open the model folder. Go to **> *Tools* > *Open model folder*** or press the icon *Open model folder*. Go to the folder *attributes* in which the settings are stored. You will see that there are two files added namely:

- order_date.prt
- order_date.prt.more

Open the file ***order_date.prt.more***. *(*Use a text editor, e.g. Notepad or Wordpad to open it). This file contains several lines which are very important. See picture below:



This file contains the lines *date 1* up to and including *date 14*. So these 14 lines belong to the several dates you can fill in by ***User defined attributes > tab Status.***. Namely:

line *date 1* belongs to *Start date*

line *date 2* belongs to *Order date*

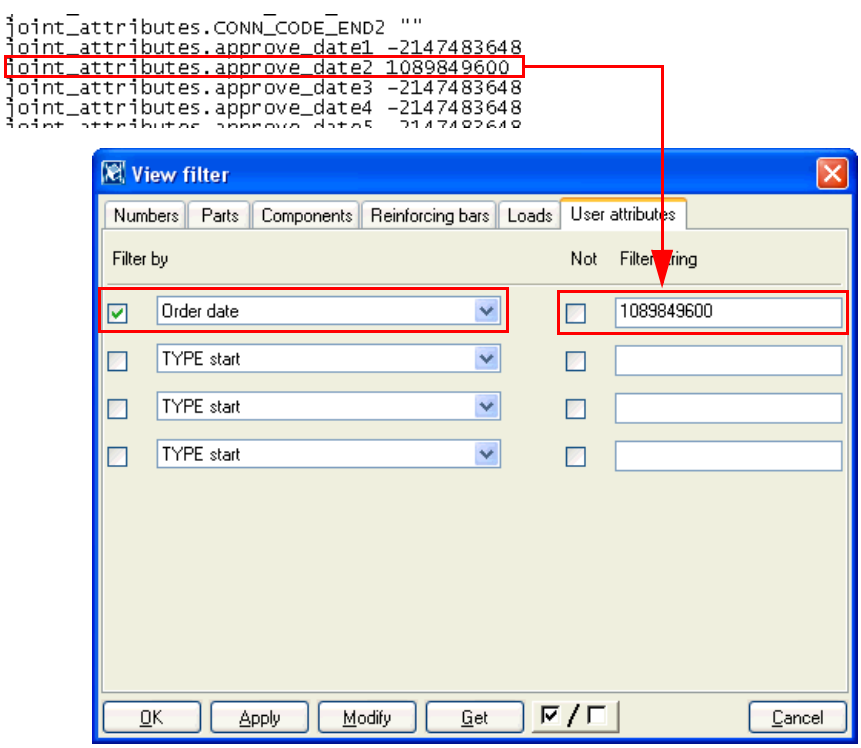
line *date 3* belongs to *In production date*

line *date 4* belongs to *Prelim date*

etc.

All lines end up with a number. This is the number you have to use while filtering on a specific date.

In this example we used *Order date.* So the number which now is important, is in line 2. See the picture below:



The number in that line must be filled in (see picture above). In this way all parts containing *order date* 15.07.2004 will be filtered.

This is the only way to filter by dates filled in by ***User defined attributes > tab Status***.

3.15 Hatching

Topics

This chapter is divided into following sections:

Hatching automatically (p.169)

Setting up hatching (p.169)

Configuration file hatches.htc (p.170)

Variable HATCH_SEGMENT_BUFFER (p.172)

Variable SECTION_LINE_COLOR (p.173)

Manual hatching (p.173)

Hatching cast units (p.175)

Hatching automatically

In Tekla Structures, profile cross sections can be hatched automatically on drawings. This is done using materials which are connected to a hatching type (hatch pattern). By default, this functionality is disabled.

Setting up hatching

You can set the following variables in **Tools > Advanced options > Hatching** to determine which drawing types must be hatched (GA-, assembly, single part- of concrete- element drawing).

XS_DRAWING_GA_HATCH_SCHEMA=hatches.htc

XS_DRAWING_ASSEMBLY_HATCH_SCHEMA=hatches.htc

XS_DRAWING_SINGLE_PART_HATCH_SCHEMA=hatches.htc

XS_DRAWING_CAST_UNIT_HATCH_SCHEMA=cast_unit.htc

Furthermore you can set which hatching configuration file has to be used.

The lines refer to the file *hatches.htc* in the folder:

schijf:\TeklaStructures\versie\environments\europe\system

The file *hatches.htc* is installed by default and will be used for all drawing types. Creating an own configuration file is an option too, of course.

Configuration file hatches.htc

```
/*****************************/
/*   TS hatch schema file  */
/*****************************/

/* Material Type, Material name, Hatch name, Scale, Colour, Auto rota
/* 1:white 2:red 3:green 4:blue 5:cyan 6:yellow 7:magenta 120:greysch
   1       2       3       4    5
STEEL,S235JR,ANSI31,0.5,5 1

CONCRETE,B25,hardware_SOLID,1,120,1
CONCRETE,B35,hardware_SOLID,1,120,1
CONCRETE,B45,hardware_SOLID,1,120,1
CONCRETE,C20/25,hardware_SOLID,1,120,1
CONCRETE,C28/35,hardware_SOLID,1,120,1
CONCRETE,C35/45,hardware_SOLID,1,120,1
CONCRETE,C45/55,hardware_SOLID,1,120,1
CONCRETE,C53/65,hardware_SOLID,1,120,1
CONCRETE,C60/75,hardware_SOLID,1,120,1
CONCRETE,BRIK,ANSI32,0.5,3,1

TIMBER,PINE,WOOD1,1.0,5,1
TIMBER,OSB,ANSI37,0.1,5,1
TIMBER,FERMACELL,ANSI37,0.1,5,1

MISCELLANEOUS,ALUMINIUM,ALU,0.2,1,1
MISCELLANEOUS,RUBBER,ANSI34,0.1,1,1
MISCELLANEOUS,ISOL_HARD,DIAMONDS,1.0,2,1
MISCELLANEOUS,ISOL_ZACHT,UNIPARQ,0.25,2,1
MISCELLANEOUS,GLASS,SQDOTS,0.1,5,1
```

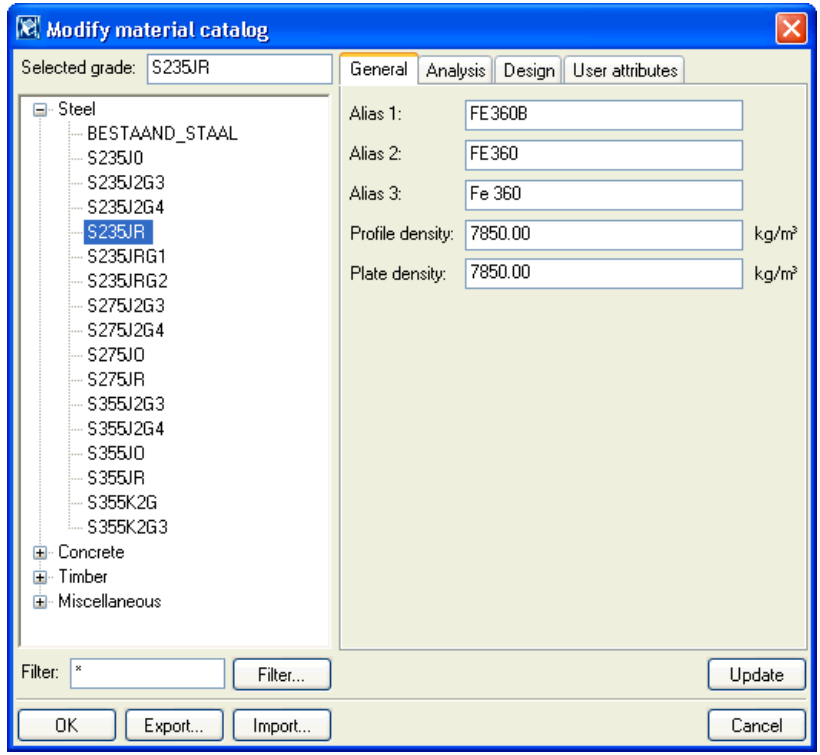
1. material group as defined in the material catalog
2. material grade
3. hatching type
4. hatching scale
5. line color

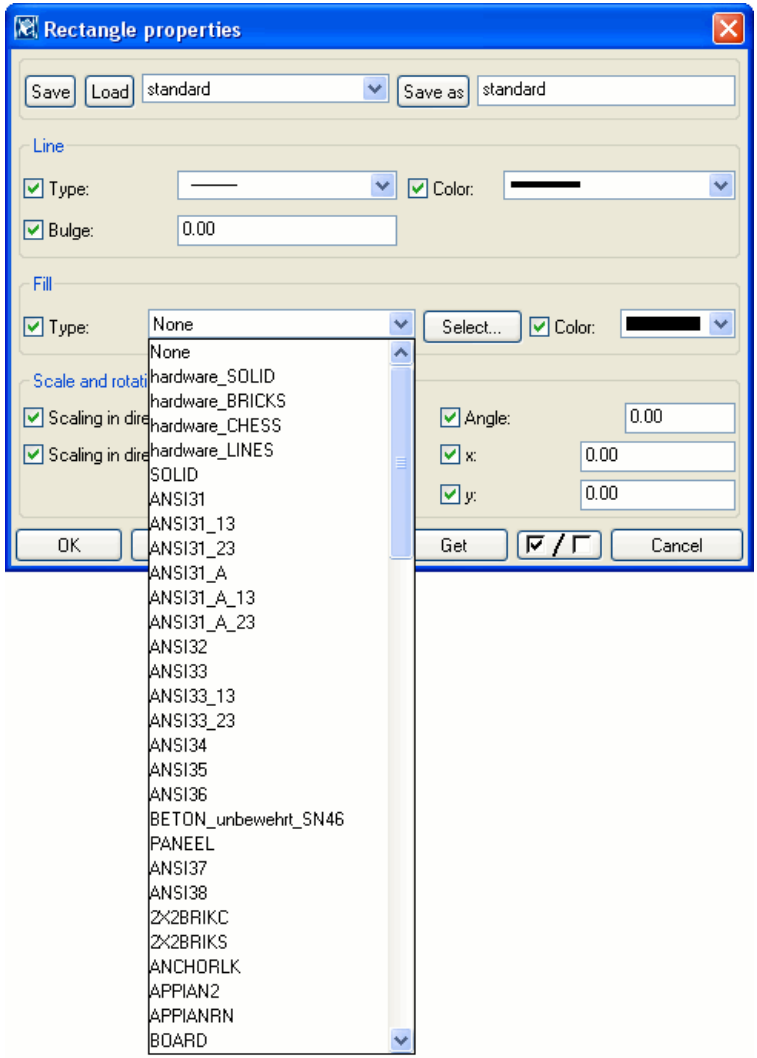


Never change the name of the material group.

Description of line: **STEEL,S235JR,ANSI31,0.5,5**

In the material catalog will be searched for grade S235JR in group STEEL.


Modify material catalog
Selected grade: S235JR
General
Analysis
Design
User attributes
Steel
BESTAAND_STAAL
S235J0
S235J2G3
S235J2G4
S235JR
S235JRG1
S235JRG2
S275J2G3
S275J2G4
S275J0
S275JR
S355J2G3
S355J2G4
S355J0
S355JR
S355K2G
S355K2G3
Concrete
Timber
Miscellaneous
Alias 1: FE360B
Alias 2: FE360
Alias 3: Fe 360
Profile density: 7850.00 kg/m³
Plate density: 7850.00 kg/m³
Filter: *
Filter...
Update
OK
Export...
Import...
Cancel



Next, there will be searched for hatching type ANSI31. Next, the scale factor is set. Finally, the hatch color is chosen. In this case color #5.

Variable HATCH_SEGMENT_BUFFER

To speed up opening drawings that contain hatches, Tekla Structures now buffers hatches. Use the following variable to adjust the size of the buffer:

XS_HATCH_SEGMENT_BUFFER_SIZE

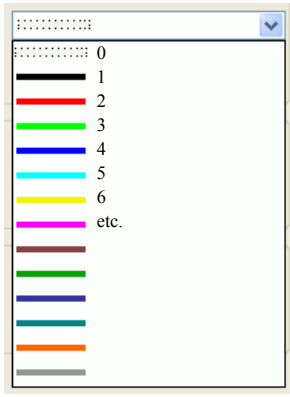
The default value is 150000. If the drawing contains very complicated hatches, try using a larger value. For small hatches, use a smaller value. You do not need to restart Tekla Structures to activate changes to this variable.

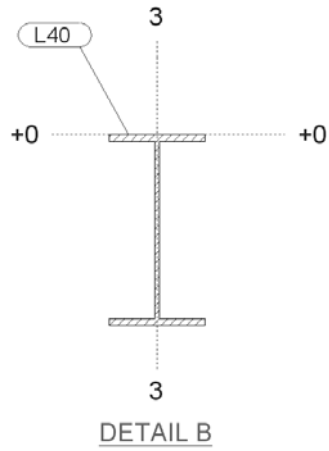
Variable SECTION_LINE_COLOR

To automatically show lines in a default color for automatic hatching in section views. Set the next variable:

XS_SECTION_LINE_COLOR=2

Enter a value for the color (from 1 to 14). The number corresponds to the color sequence in the drawing object properties:

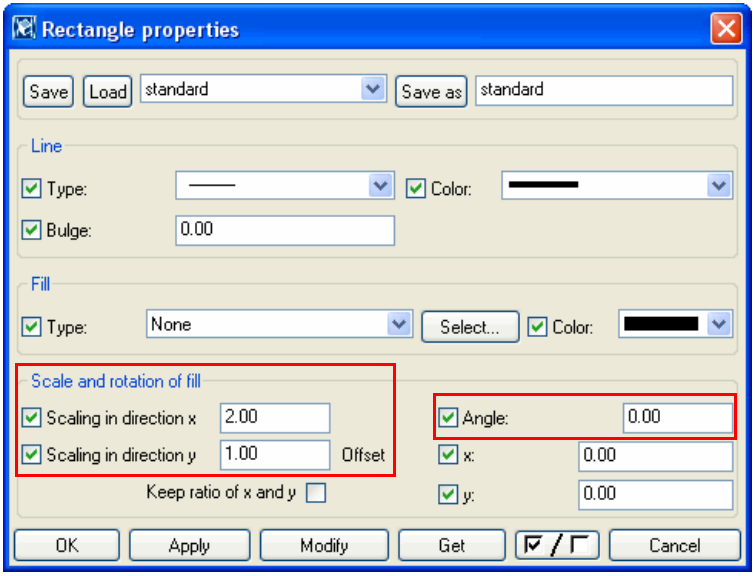




This line is shown in the red color in the drawing.

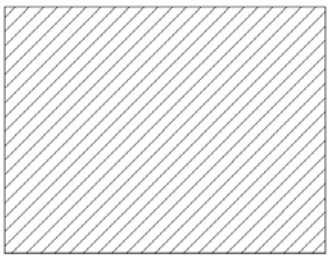
Manual hatching

In Tekla Structures manual hatching is increased.



All hatchings can be scaled individual.

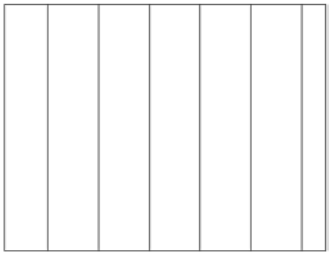
You can also use two scales and an angle for each type of hatching, see pictures below.



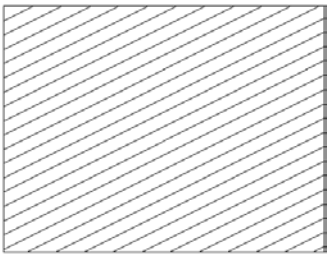
ANSI31, scale 1, angle 0



ANSI31, scale 5, angle 0



ANSI31, scale 5, angle 45



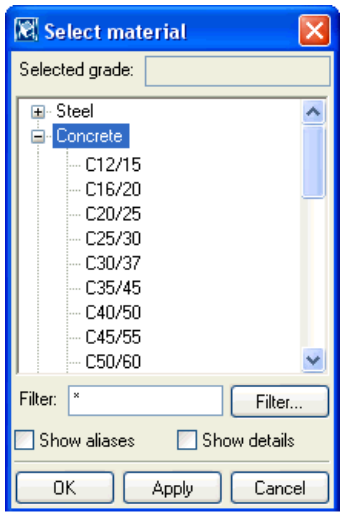
ANSI31, scale 1 and 2, angle 0

Hatching cast units

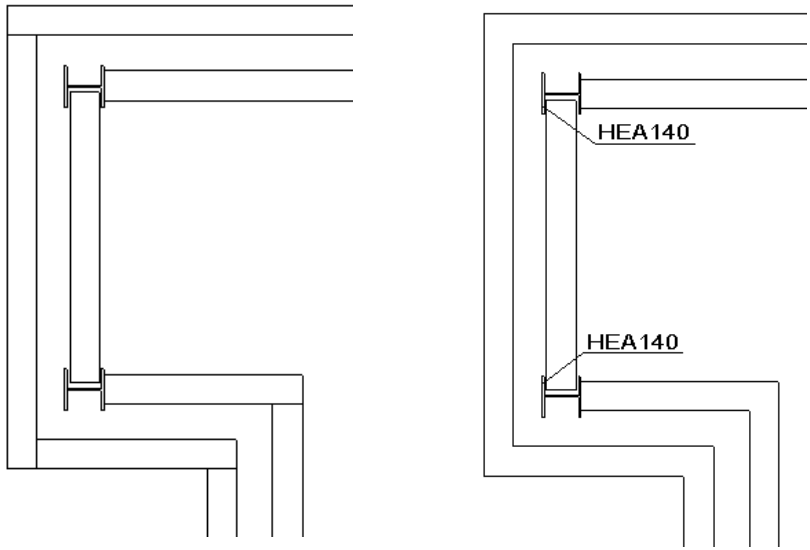
Hatching cast units automatically on general arrangement drawings and/or cast unit drawings can be done on the same way. For cast unit drawings the file *cast_unit.htc* is installed. For general arrangement drawings the file *general.htc* (or any other custom made file) can be used.

When there is a tangent plane between two cast unit parts, modelled by the use of parts in the *concrete* group, the tangent line is not visible on the drawing.

This goes for all materials in the *concrete* group, so except several grades of concrete also for e.g. brickwork and sand-lime bricks. (or any other grade created in the concrete group).

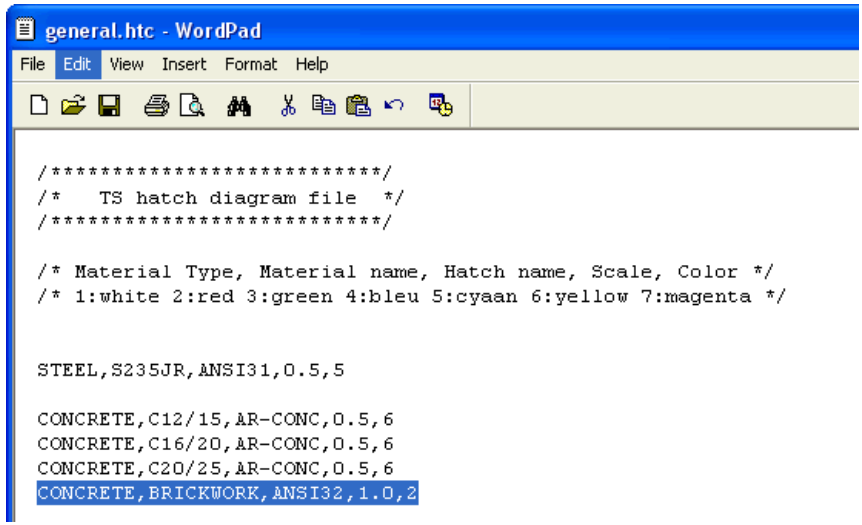


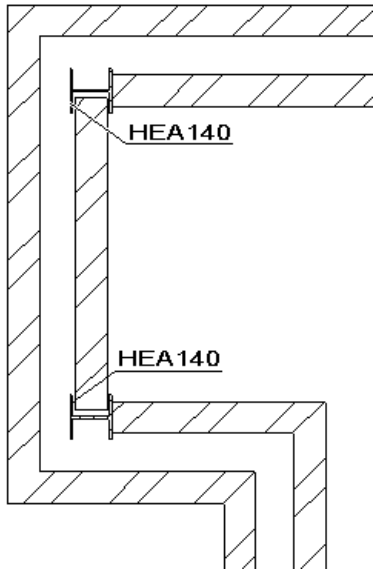
The following example shows several walls which contain tangent planes. The tangent lines are not visible on the drawing.



model drawing

In the file *general.htc* a line is added for hatching brickwork automatically.





To show the tangent lines of cast units on drawings, use variable ***XS_DRAW_CAST_UNIT_INTERNAL_LINES*** or make use of another phase while modelling.

3.16 ID reports

Topics

This chapter is divided into following sections:

Id_boltlist (p.178)

Id_boltlist_100 (p.178)

Id_weldlist (p.179)

Id_lengthlist (p.179)

Id_length_gross (p.180)

Id_loose_parts (p.180)

Id_loose_parts _NOT (p.181)

Id_original_drawing (p.181)

Id_partlist/id_assemblylist (p.181)

Id_sawlist (p.182)

The main feature of these lists is that the lines in the lists which contain profiles can be selected. Subsequently, the accompanying profile will highlight in the model. This can be useful in case a profile is hard to locate.

Several ID lists are available inTekla Structures:

Id_boltlist

This list can be very handy when you search for bolts with a specific length in a model. While selecting the line in the list, the accompanying bolt(s) will highlight in the model.

This 'highlight-functionality' only works when the list is opened in the standard Tekla Structures list viewer (on dialog).

Id_boltlist_100

This list only classifies bolts with a length of 100 mm. Mostly, these are bolts which are not modelled correctly. While selecting the line in the list, the accompanying bolt(s) will highlight in the model



Select the button ***Fit work area by parts in selected views*** if you cannot find or see the bolts in the model view.

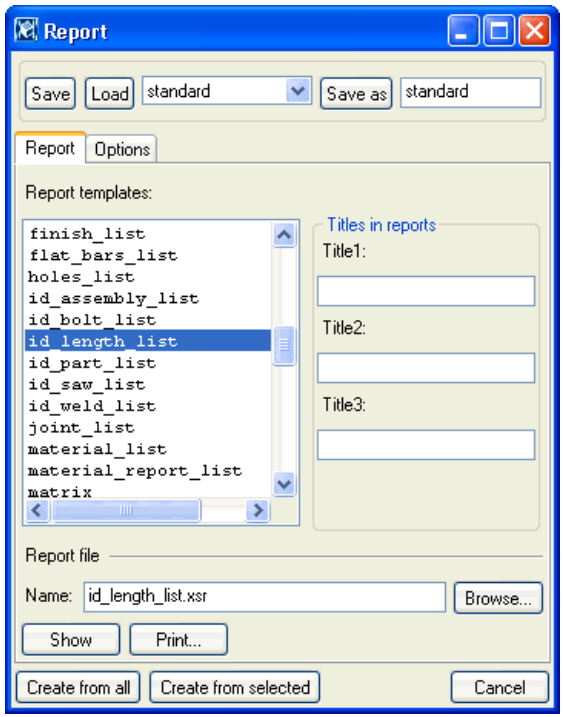
This 'highlight-functionality' only works when the list is opened in the standard Tekla Structures list viewer (on dialog).

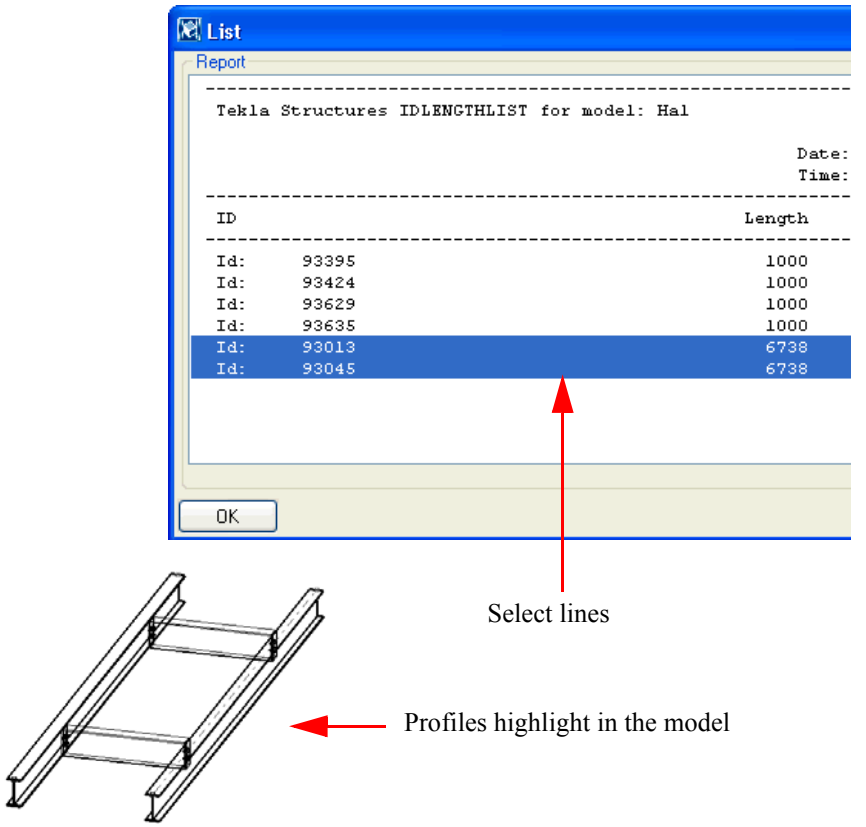
ld_weldlist

This list can be handy when you search for welds in a model. While selecting the line in the list, the accompanying weld(s) will highlight in the model. This 'high-light-functionality' only works when the list is opened in the standard Tekla Structures list viewer (on dialog).

ld_lengthlist

This list can be handy to classify profiles with a certain length and to easily select them (or all) in the model.





Now that the profiles in the model highlight, the accompanying drawings in the drawing list can be filtered.

ld_length_gross

This list only classifies profiles which differ in LENGTH and LENGTH_GROSS.

ld_loose_parts

This list only classifies one part assemblies. For example loose plates you forgot to weld.

Id_loose_parts _NOT

This list is the opposite of the list *id_loose_parts*. So this list classifies no one part assemblies, such as loose plates, filling plates or bracings consisting only one part.

Id_original_drawing

Use this report **Id_original_drawing** to find out to which part the drawing is linked:

To use the report: In the drawing list, select a drawing and create the report **Id_original_drawing:**

```
-------------------------------------------------------------------------------
TEKLA STRUCTURES ID_ORIGINAL_DRAWING_LIST          Page:     1
Model:                                             Project:  project number
                                                   Date:     13.11.2007
                                                   Time:     13:30:23

-------------------------------------------------------------------------------
     Original part for drawing
-------------------------------------------------------------------------------
     Id:858
-------------------------------------------------------------------------------
                                    END
-------------------------------------------------------------------------------
```

While selecting the line in the list, the accompanying part will highlight in the model.

Id_partlist/id_assemblylist

Example: occasionally it can happen that a folded plate will not be fully displayed in the model after creating the part. The only edges displayed are the chamfer symbols. To solve this, change the rotation of this contour plate.



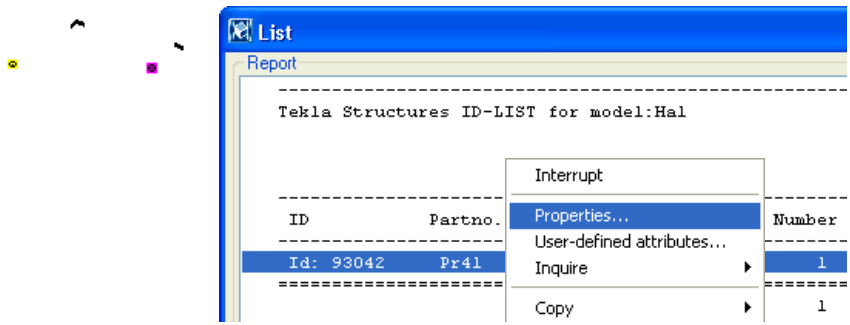
Example of wrong represented folded plate

The possible drawing of the contour plate shall only contain an empty frame. However, the profile will be listed correctly in reports.

To create an id-list of a folded plate profile:

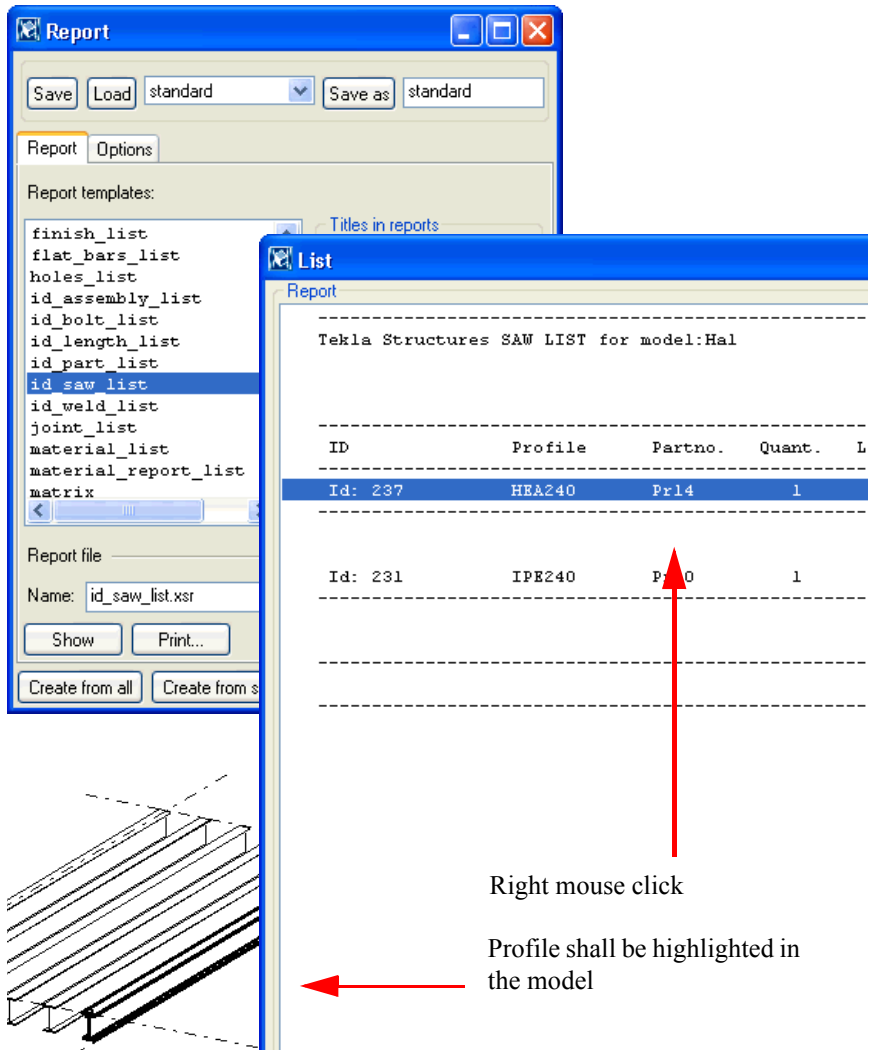
- Select a chamfer of the profile
- Create a list of selected parts
- In the report, the Id-number line can be selected. From the contextmenu (right mouse click), the profiles' properties dialog box can be accessed. NOTE: report must be opened using "In dialog".

To repair the plates' representation, it is sufficient to set another parts' rotation.



Id_sawlist

The report ***ID_sawlist*** only shows profiles which have skew ends (i.e. angle not equal to 90°).



When a ID_sawlist is created from all parts, and all lines in the report are selected, all profiles with skew profile ends (i.e. not equal to 90 degrees.) will be highlighted in the model. Next, the Wizard can be used to create single part drawings of the selected parts.

3.17 Key plan

Topics

This chapter is divided into following sections:

Conditions (p.185)

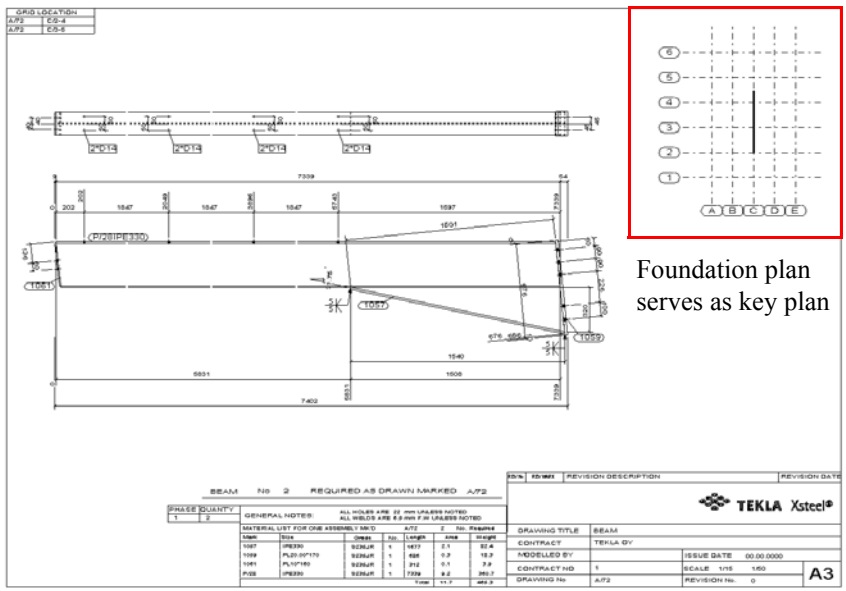
Create a key plan-drawing for a GA drawing (p.185)

Adding a key plan to a drawing lay out for a GA drawing (p.186)

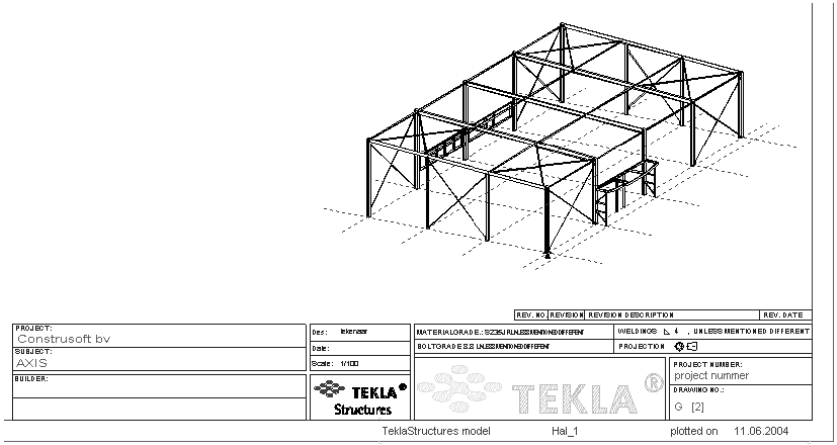
Create a key plan-drawing for a single part- or an assembly drawing (p.190)

Adding a key plan to a drawing lay out for an assembly drawing (p.191)

A ***key plan*** is a small “map” in a drawing that locates an assembly, a cast unit, or a part in the model. The key plan includes the model grid and the assembly, cast unit, or part shown in the drawing.



Also general arrangement drawings or details can be used. Key plans are just drawing views appearing on other drawings. In stead of drawing views also ***.dwg** or ***.dxf** files could be used.



Using key plans reduces editing all drawings manually. It is possible to add a ***key plan*** and/or a ***dwg/dxf*** for all types of drawings.

Conditions

To create a key plan, you need to use a drawing that contains only **one view** (of the right **scale** and **size**). Tekla Structures only uses the view from the drawing. The view position, drawing size, and templates are not relevant to a key plan drawing.

Set the view properties as follows:

- Tekla Structures uses the scale of the key plan view in every drawing using it.
- In the drawing view properties, set the drawing view boundaries **(By extreme)** so that all parts are visible.
- Other drawing view properties also affect the key plan. If you set part and bolt marks to visible in the key plan view, they will also be visible in the small key plan included in the drawing.
- Tekla Structures automatically includes the right part when you insert the small key plan into the drawing.

Create a key plan-drawing for a GA drawing

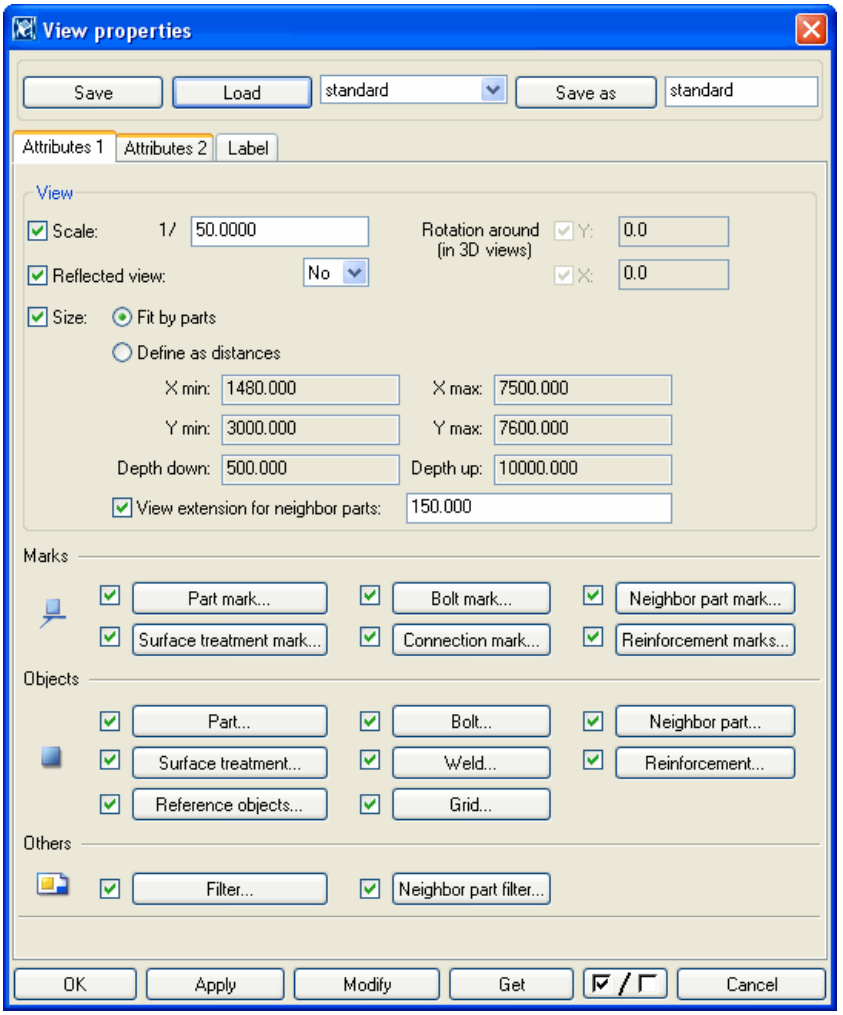
Example

An overview of ***hall 1*** on all GA drawings (see picture previous page).

Step plan

1. In the Model editor, click **Drawing** > **General arrangement...**
2. In the **Options** list box, select **Empty drawing.** Select the **Open drawing** check box**.** Click **Create.** Tekla Structures opens the Drawing editor.

3. Create one view on the GA drawing. Mind the scale of the key plan!
4. Make sure that in **Filter** nothing is filled in.



5. Close the drawing.

Adding a key plan to a drawing lay out for a GA drawing

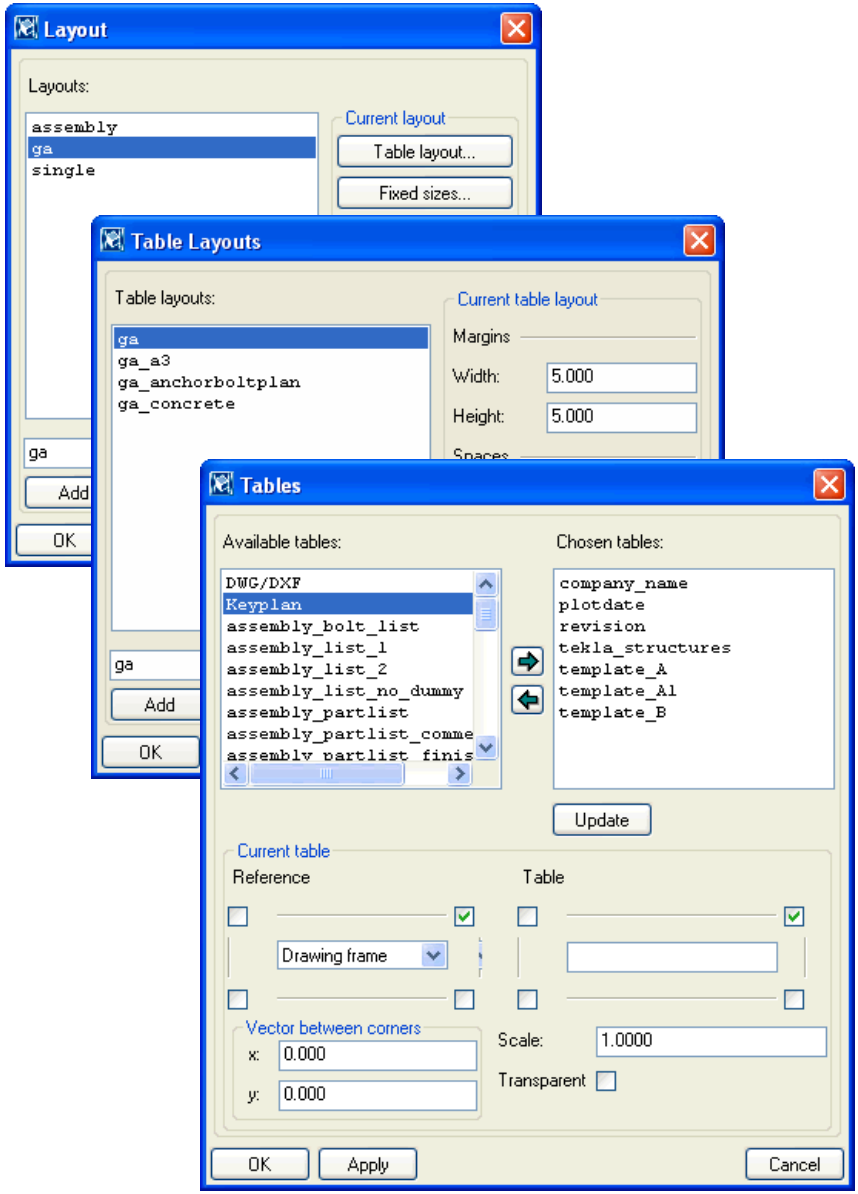
Step plan

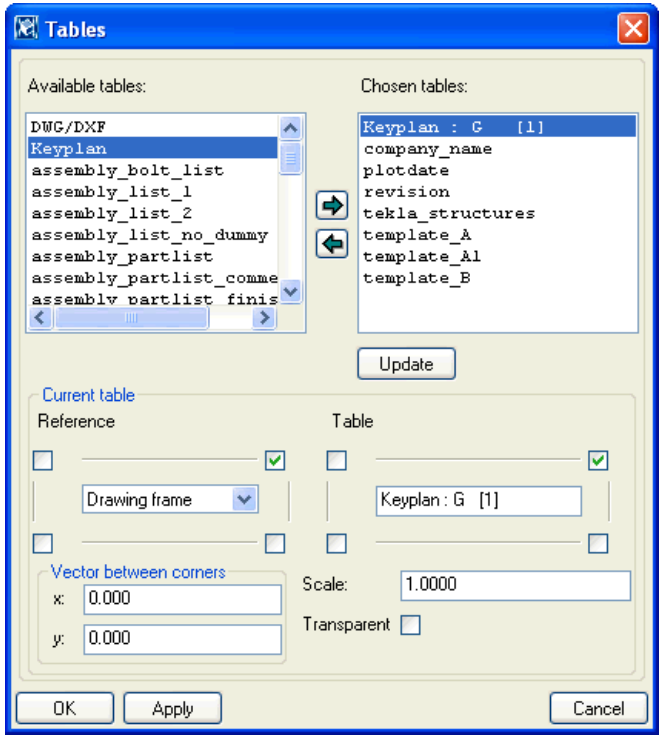
1. In the Model editor, click **Properties** > **Layout...**
2. Select the layout you want to modify. In this case the layout **ga** and the table **ga**.
3. Select the layout **ga**, then click **Table layout**. In **Table layouts** select **ga** and click **Tables.**

4. From the **Available tables** list, double-click **Keyplan.** The drawing list appears.
5. Select the key plan drawing and click **OK.**
6. From the **Chosen tables** list, select **Key plan.**



You cannot scale the key plan in the **Tables** dialog box. The properties of the key plan drawing view define the key plan scale and size.


Layout
Layouts:
assembly
ga
single
Current layout
Table layout...
Fixed sizes...
ga
Add
OK
Table Layouts
Table layouts:
ga
ga_a3
ga_anchorboltplan
ga_concrete
Current table layout
Margins
Width:
5.000
Height:
5.000
ga
Add
OK
Tables
Available tables:
DWG/DXF
Keyplan
assembly_bolt_list
assembly_list_1
assembly_list_2
assembly_list_no_dummy
assembly_partlist
assembly_partlist_comme
assembly_partlist_finis
Chosen tables:
company_name
plotdate
revision
tekla_structures
template_A
template_A1
template_B
Update
Current table
Reference
Table
Drawing frame
Vector between corners
x:
0.000
y:
0.000
Scale:
1.0000
Transparent
OK
Apply
Cancel

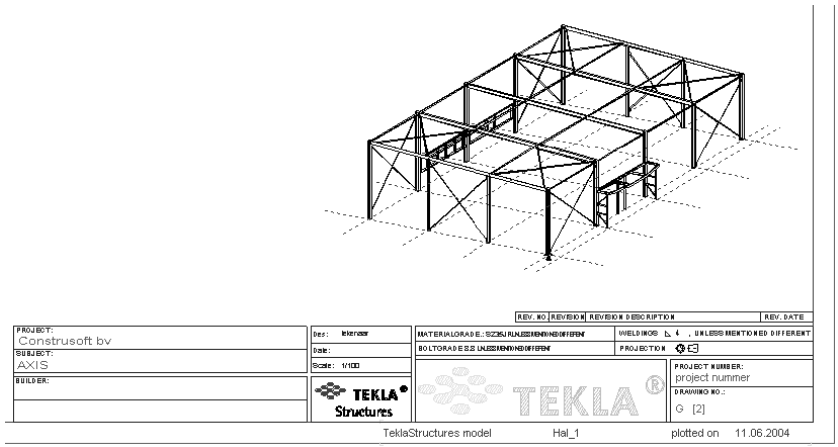


The key plan is now added to the chosen tables. The key plan contain the name of the connected drawing. (in this case G [1]).

By toggling the check boxes you can determine the position

It is also possible to add several key plans.

7. Open or create a GA drawing on which the layout ***ga*** is used.



The lay out **ga.lay** is stored in the folder attributes in the model directory and will be used for all layouts named ***ga*** in the actual model.

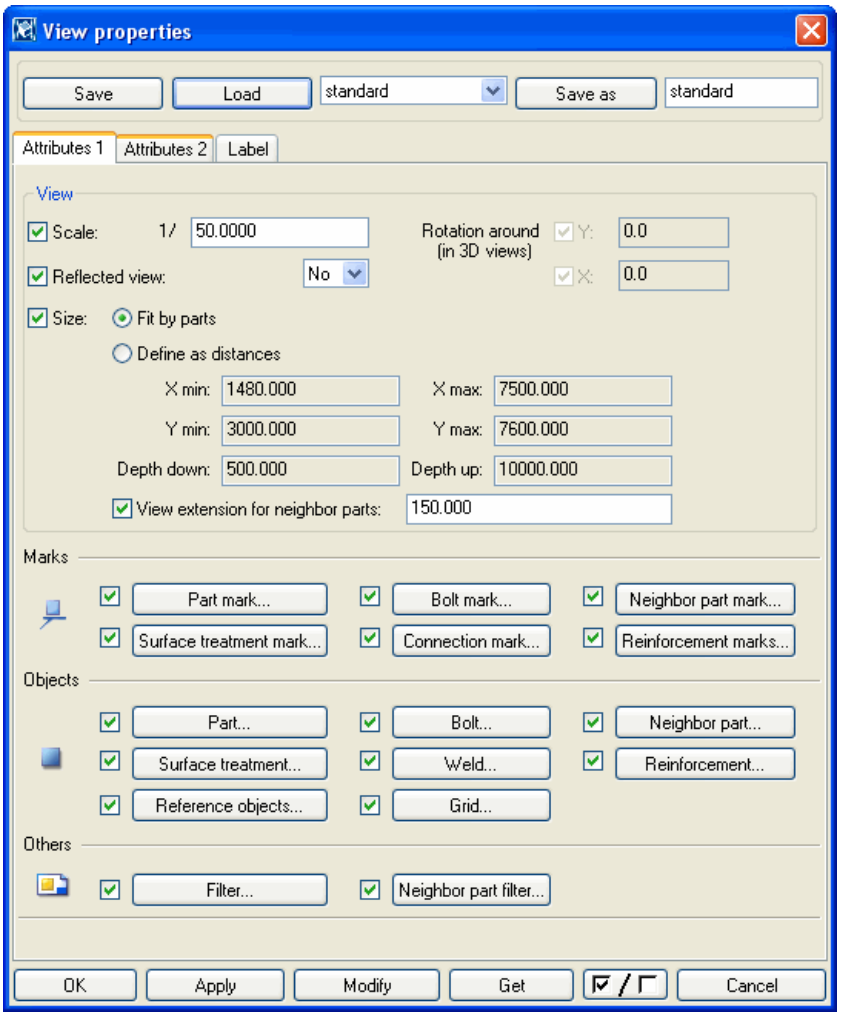
Create a key plan-drawing for a single part- or an assembly drawing

Example

A foundation plan of ***hall 1*** on all assembly drawings (see picture previous page).

Step plan

1. In the Model editor, click **Drawing** > **General arrangement...**
2. In the **Options** list box, select **Empty drawing.** Select the **Open drawing** check box**.** Click **Create.** Tekla Structures opens the Drawing editor.
3. Create one view on the GA drawing, in this case a foundation plan. Mind the scale of the key plan!
4. Make sure that in **Filter** nothing is entered.

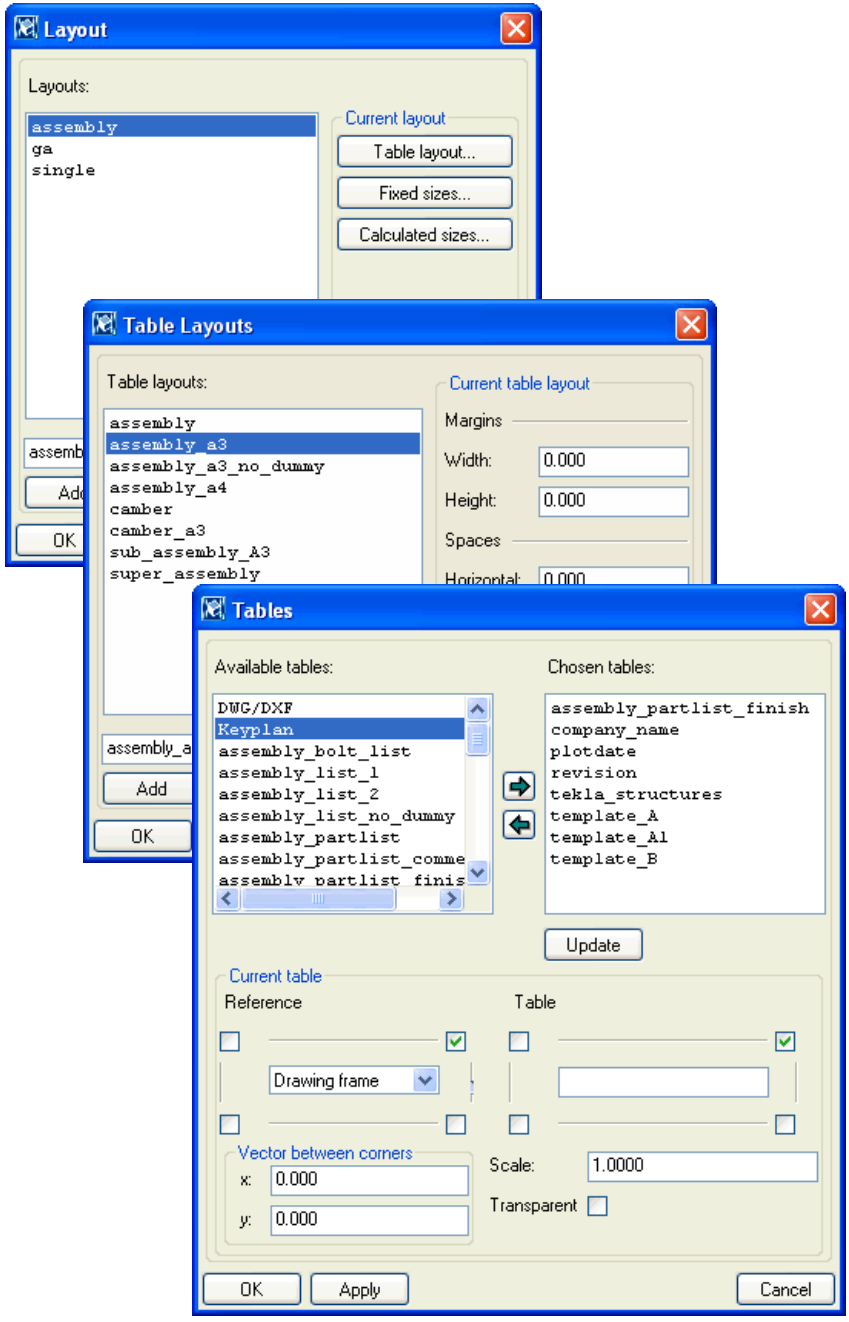


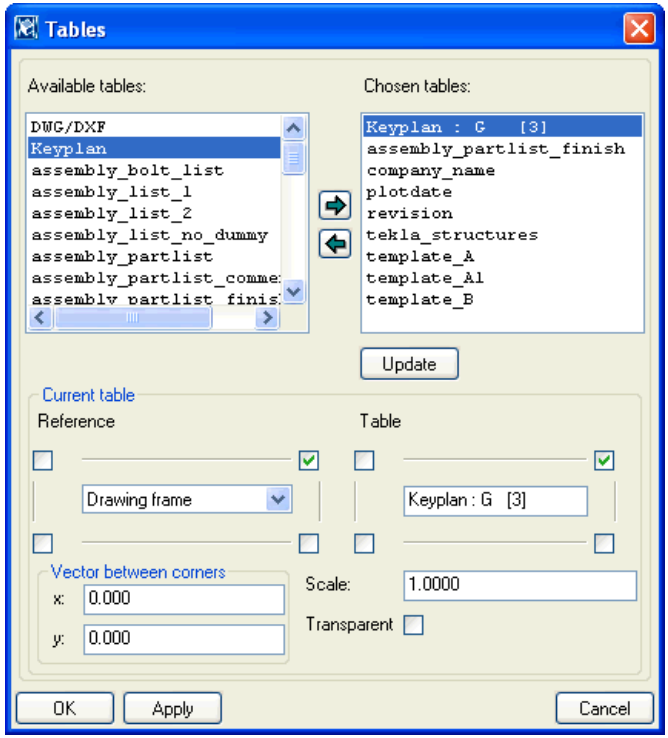
5. Close the drawing.

Adding a key plan to a drawing lay out for an assembly drawing

Step plan

1. In the Model editor, click **Properties** > **Layout...**
2. Select the layout you want to modify. In this case the layout **assembly** and the table **assembly_A3**.
3. Select the layout **assembly**, then click **Table layout**. In **Table layouts** select **assembly_A3** and click **Tables.**
4. From the **Available tables** list, double-click **Keyplan.** The drawing list appears.
5. Select the key plan drawing and click **OK.**
6. From the **Chosen tables** list, select **Key plan.**


Layout
Layouts:
assembly
ga
single
Current layout
Table layout...
Fixed sizes...
Calculated sizes...
OK
Table Layouts
Table layouts:
assembly
assembly_a3
assembly_a3_no_dummy
assembly_a4
camber
camber_a3
sub_assembly_A3
super_assembly
Current table layout
Margins
Width:
0.000
Height:
0.000
Spaces
Add
OK
Tables
Available tables:
Chosen tables:
DWG/DXF
Keyplan
assembly_bolt_list
assembly_list_1
assembly_list_2
assembly_list_no_dummy
assembly_partlist
assembly_partlist_comme
assembly_partlist_finis
assembly_partlist_finish
company_name
plotdate
revision
tekla_structures
template_A
template_A1
template_B
Add
Update
Current table
Reference
Table
Drawing frame
Vector between corners
x:
0.000
y:
0.000
Scale:
1.0000
Transparent
OK
Apply
Cancel

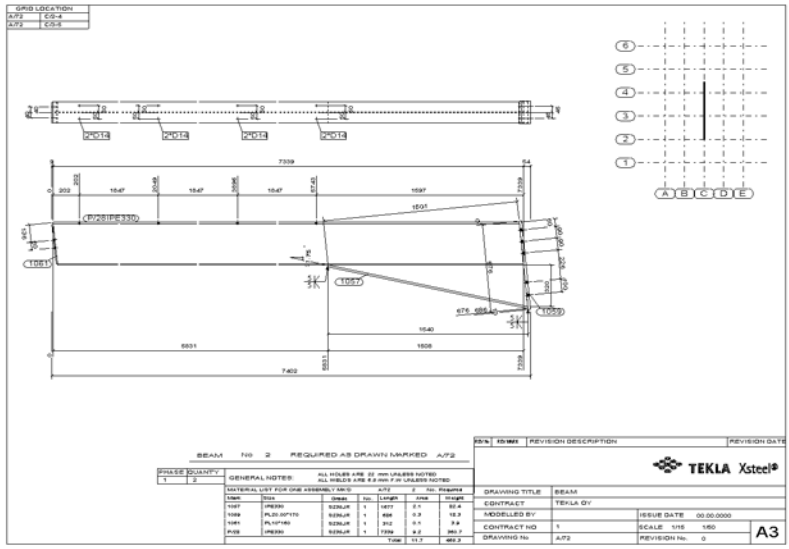


The key plan is now added to the chosen tables. The key plan contain the name of the connected drawing (in this case G [3]).

By toggling the check boxes you can determine the position

It is also possible to add several key plans.

7. Open or create an assembly drawing on which the layout ***assembly*** is used



The lay out **assembly.lay** is stored in the folder attributes in the model directory and will be used for all layouts named ***assembly*** in the actual model.

3.18 Alternative numbering for Multi drawings

Topics

This chapter is divided into following sections:

Set up Multi-numbering (p.194)

Method (p.195)

Note with regard to Multi numbering (p.196)

Assembly drawings placed on multi drawings normally keep their assembly number on the multi drawing, e.g. assembly B1 retains assembly number B1.

An alternative way of assigning numbers to assemblies is to assign the assembly number on the basis of the name of the multi drawing, followed by a unique number. So when the multi drawings' name is '4625', the assemblies on that multi drawing get the assembly numbers 4625/01, 4625/02 etc.

A	[C.1]	COLUMN
A	[C.2]	COLUMN
A	[C.3]	COLUMN
A	[C.4]	COLUMN
A	[C.5]	COLUMN
A	[C.6]	COLUMN
A	[C.7]	COLUMN
A	[C.8]	COLUMN
A	[C.9]	COLUMN
A	[C.10]	COLUMN
M	[1]	1
M	[2]	2

Standard numbering for multi drawings

A	[4625-01]	COLUMN
A	[4625-02]	COLUMN
A	[4625-03]	COLUMN
A	[4625-04]	COLUMN
A	[4625-05]	COLUMN
A	[4625-06]	COLUMN
A	[4626-01]	COLUMN
A	[4626-02]	COLUMN
A	[4626-03]	COLUMN
A	[4626-04]	COLUMN
M	[1]	4625-
M	[2]	4626-

Alternative way of numbering for multi drawings

Set up Multi-numbering

To use this way of numbering, some variables need to be set as follows in **Tools** > **Advanced options** > **Numbering**:

XS_USE_ASSEMBLY_NUMBER_FOR=MAIN_PART

XS_USE_MULTI_NUMBERING_FOR=ASSEMBLIES

XS_USE_NUMERIC_MULTI_NUMBERS_FOR=ASSEMBLIES

XS_MIN_NUMBER_OF_ASSEMBLY_MULTI_CHARACTERS=2

XS_ALLOW_DRAWING_TO_MANY_MULTI_DRAWINGS=1

XS_SWITCH_MULTI_NUMBERS_FOR=ASSEMBLIES

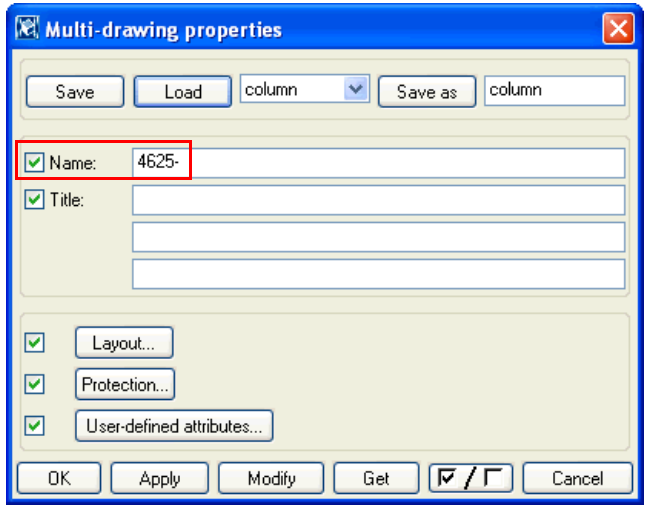
XS_ASSEMBLY_MULTI_NUMBER_FORMAT_STRING=%%ASSEMBLY_MULTI_DRAWING_NUMBER%%%%ASSEMBLY_MULTI_DRAWING_POS%%

Method

1. When the model is numbered, all assemblies get numbers assigned, according to the entered assembly prefixes and start numbers (e.g. C/1, B/1 etc.) The assembly number changes only just when the assembly drawing is linked to the multi drawing.
2. Next create assembly drawings. Also now the assemblies in the drawing list have their original number.

13.05.2005	00.00.0000	420* 297	A	[C.1]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.2]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.3]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.4]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.5]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.6]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.7]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.8]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.9]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.10]	COLUMN

3. Go to **Properties** > **Multi-drawing...** Enter in the field '*Name*' a new name or number. In the following example, the number '4625'



4. Create one or more multi drawings: **Drawing** > **Multi-drawing** > **Multi-drawing**

13.05.2005	00.00.0000	420* 297	A	[C.1]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.2]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.3]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.4]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.5]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.6]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.7]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.8]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.9]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.10]	COLUMN
13.05.2005	00.00.0000	1189* 841	M	[1]	4625-
13.05.2005	13.05.2005	1189* 841	M	[2]	4626-

5. Now open a multi drawing and select one or more assembly drawings you want to link to the multi drawing.
6. Go to **Create** > **Link drawing.** The assembly drawings now get linked to the multi drawing.
7. Go to **Edit** > **Place views** to reorder the views om the drawing. The assembly drawing' numbers in the drawing list get updated automatically.

13.05.2005	13.05.2005	420* 297	A	[4626-01]	COLUMN
13.05.2005	13.05.2005	420* 297	A	[4626-02]	COLUMN
13.05.2005	13.05.2005	420* 297	A	[4626-03]	COLUMN
13.05.2005	13.05.2005	420* 297	A	[4626-04]	COLUMN
13.05.2005	13.05.2005	420* 297	A	[4626-05]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.6]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.7]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.8]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.9]	COLUMN
13.05.2005	00.00.0000	420* 297	A	[C.10]	COLUMN
13.05.2005	00.00.0000	1189* 841	M	[1]	4625-
13.05.2005	13.05.2005	1189* 841	M	[2]	4626-

8. Do the same for the other multi drawings.

13.05.2005	13.05.2005	420* 297	A	[4626-01]	COLUMN
13.05.2005	13.05.2005	420* 297	A	[4626-02]	COLUMN
13.05.2005	13.05.2005	420* 297	A	[4626-03]	COLUMN
13.05.2005	13.05.2005	420* 297	A	[4626-04]	COLUMN
13.05.2005	13.05.2005	420* 297	A	[4626-05]	COLUMN
13.05.2005	13.05.2005	420* 297	A	[4626-06]	COLUMN
13.05.2005	13.05.2005	420* 297	A	[4626-07]	COLUMN
13.05.2005	13.05.2005	420* 297	A	[4626-08]	COLUMN
13.05.2005	13.05.2005	420* 297	A	[4626-09]	COLUMN
13.05.2005	13.05.2005	420* 297	A	[4626-10]	COLUMN
13.05.2005	00.00.0000	1189* 841	M	[1]	4625-
13.05.2005	13.05.2005	1189* 841	M	[2]	4626-

Note with regard to Multi numbering

When an assembly drawing is removed from a multi drawing, the assembly number is NOT automatically changed into its original assembly number. To pull this off, select the profile(s) in the model and clear the selected profile(s) by means of **Tools** > **Numbering** > **Clear selected**.

3.19 Tekla Stuctures models

Topics

This chapter is divided into following sections:

Tekla Structures model structure (p.197)

Exchanging Tekla Structures models (p.199)

Zip a Tekla Structures model (p.199)

Tekla Structures models on a CD-rom (p.204)

Check database (p.206)

Correct database (p.207)

DWG/DXF, text files and reference-objects (p.208)

Layout files and templates (p.209)

Catalogs (p.210)

Ini files (p.211)

Overlap in the numbering (p.211)

Sending models to the Construsoft Help desk (p.212)

Creating a Back Up (p.214)

Tekla Structures model structure

It is important to store the Tekla Structures models in a well-organized way.

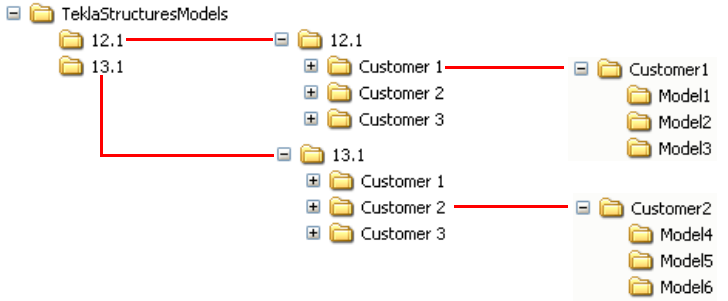
Especially when you have models from different Tekla Structures versions and when you work for several customers.



Very often a model created in a particular version is opened in another version, with all serious consequences.

Several versions

If you are working with several versions (e.g. Tekla Structures 12.1 or Tekla Structures 13.1), you have to use orderly structure. A possible folder structure can be:

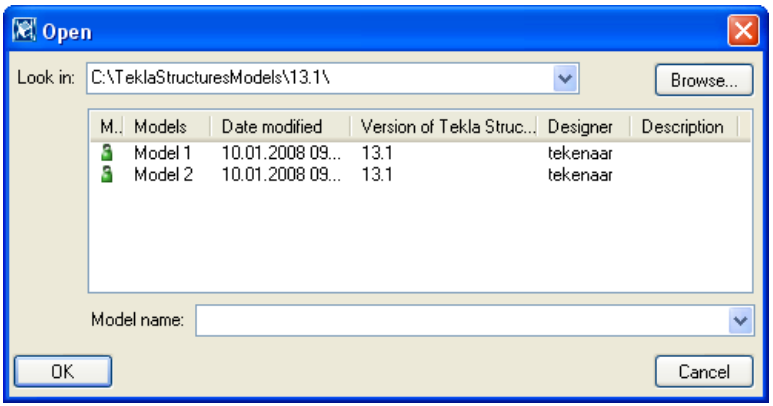


This will prevent that models are opened in versions which they are not meant for.

If the variable XS_RUNPATH is set in a way that the correct path name per version is used in Tekla Structures, these kind of problems are eliminated, i.e.:

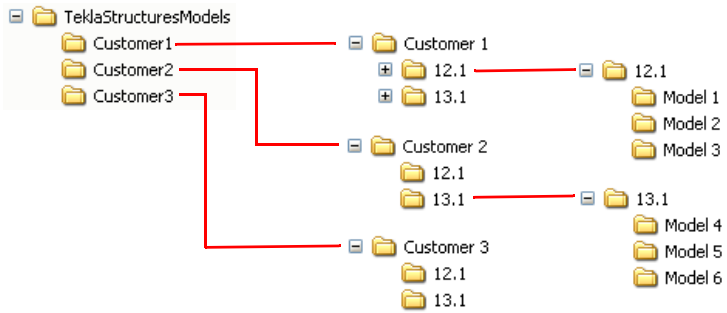
set XS_RUNPATH=C:\TeklaStructuresModels\13.1

When you open a model in Tekla Structures now, always the correct path name is entered:



Different versions en several customer

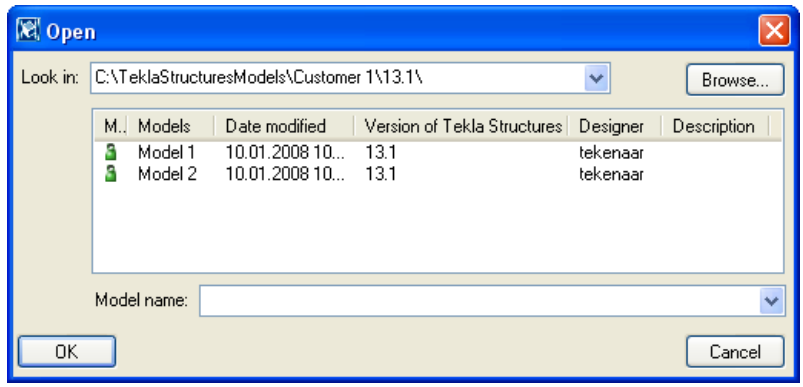
When you work with different Tekla Structures versions en several customers, a division like the picture below could be used:



A unique variable per customer can be used where is referred to a line, i.e.:

set XS_RUNPATH=C:\TeklaStructuresModels\Customer 1\13.1

When a model is opened, the correct customer and version is selected.



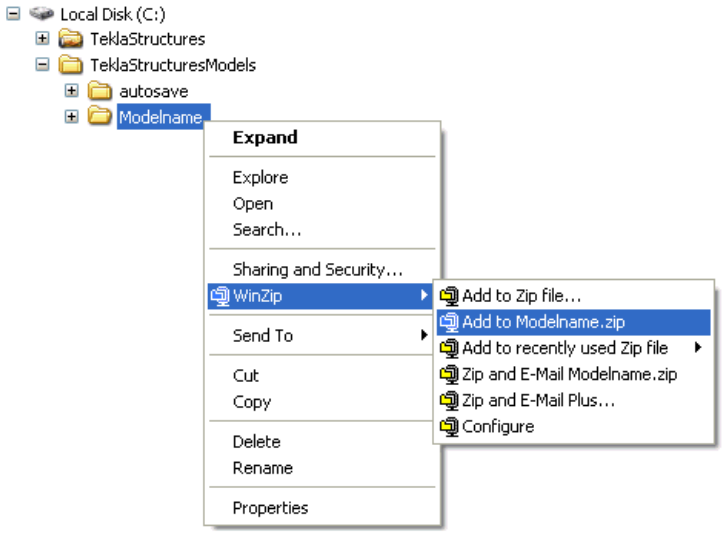
Exchanging Tekla Structures models

When exchanging Tekla Structures models between customers, some problems may arise. This because apart from the Tekla Structures model, also additional files have to send too. Which files are necessary, is explained in the paragraph below.

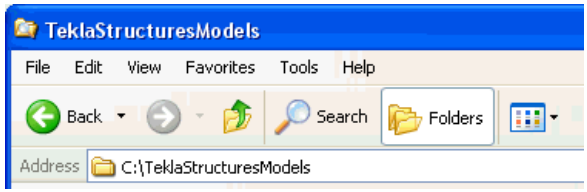
Zip a Tekla Structures model

Because the size of a model can be quite big, it can be useful to zip the model before sending. This has two big advantages: model will be compressed and you can add the path in the zip-operation. The receiver just has to extract the file with the option ‘Use Folder names’ enabled, and all files are stored on their exact location.

The best way of zipping a model is to zip a complete folder. Right mouse click on the model folder.

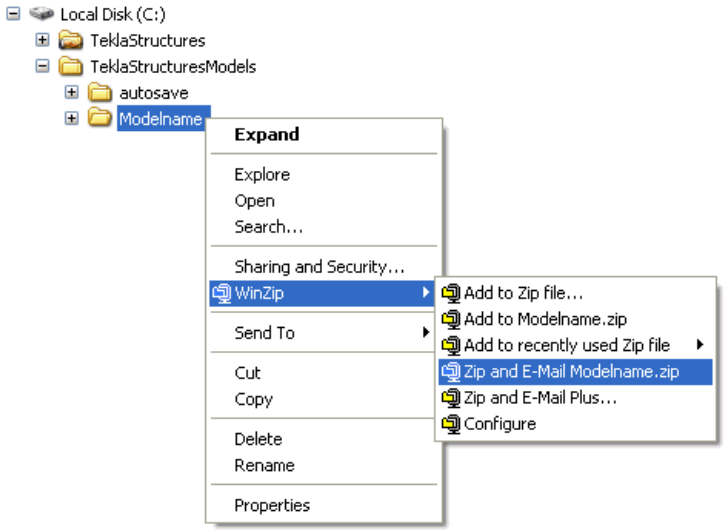


Model **Modelname** is located in *C:\Tekla StructuresModels*. The zipped file **Modelname.zip** will also be stored in the folder *C:\Tekla StructuresModels*.



A more quick way is to use the ‘Zip and E-mail’ option.

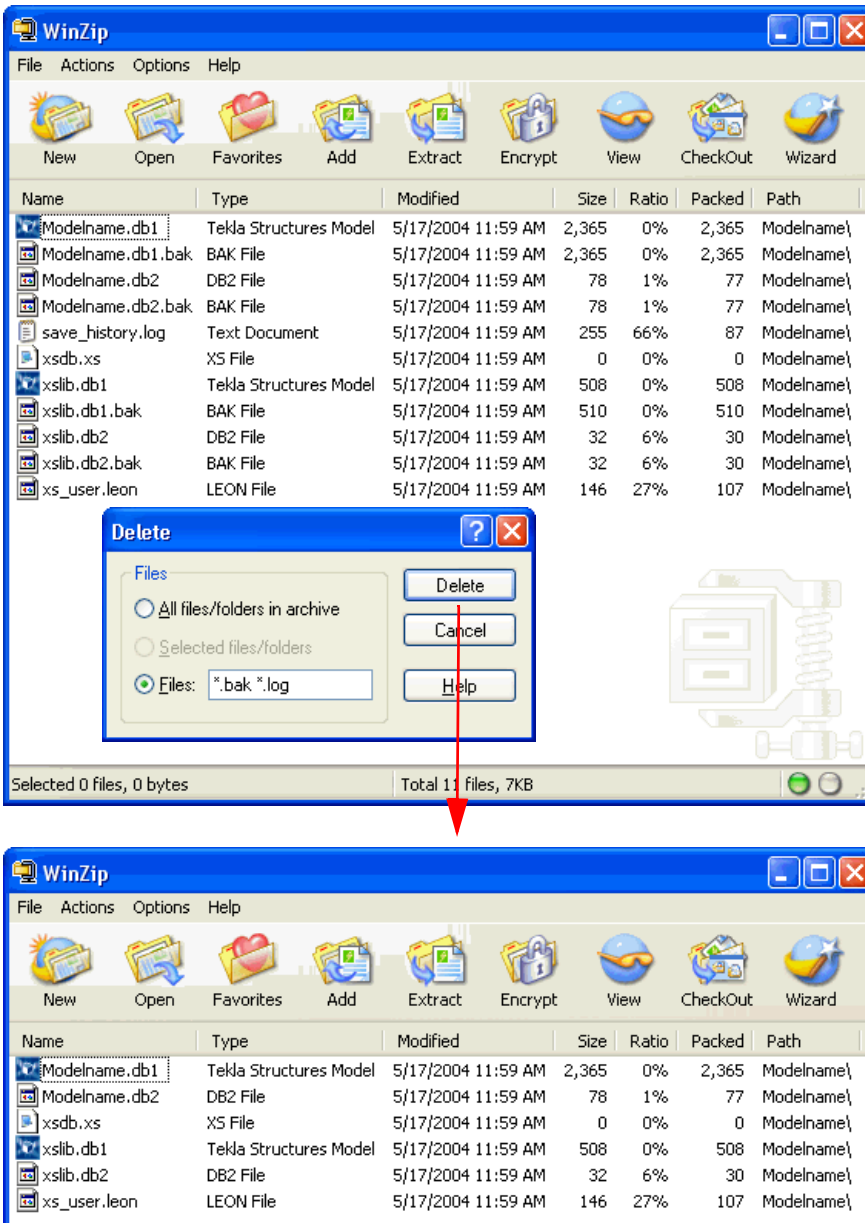
Right mouse click on the model folder, then select ‘Zip and E-mail Modelname.zip’:



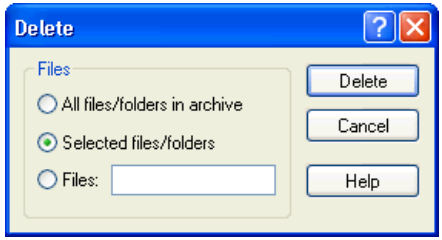
Before you send the model, you can decrease the file size by deleting some redundant files.

Step plan

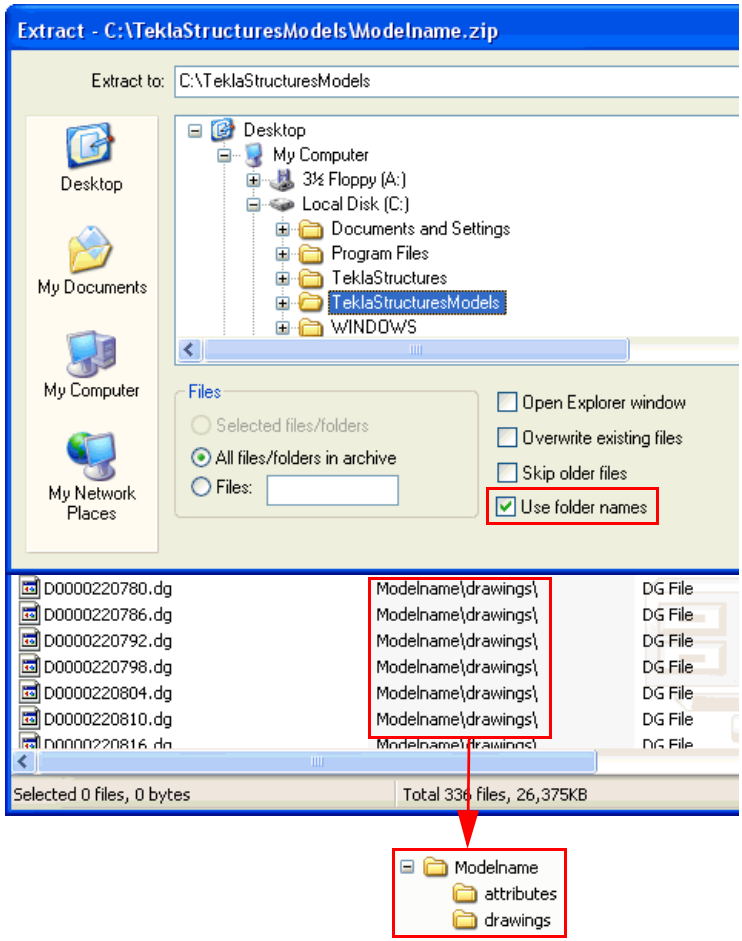
- Open the zipped file **Modelname.zip**.
- Select the files to be removed, go to **Actions** > **Delete** or press the <**Delete>** button.
- When using the **Actions** > **Delete** option, enter the file extensions to be removed, separated by a space character, possible redundant files are: ***.log**, ***.pdf**, .***dwg**, .***bak**, ***.history**.



Furthermore, files can be removed from the zip file manually (use ***Shift/Ctrl*** for selecting multiple files).

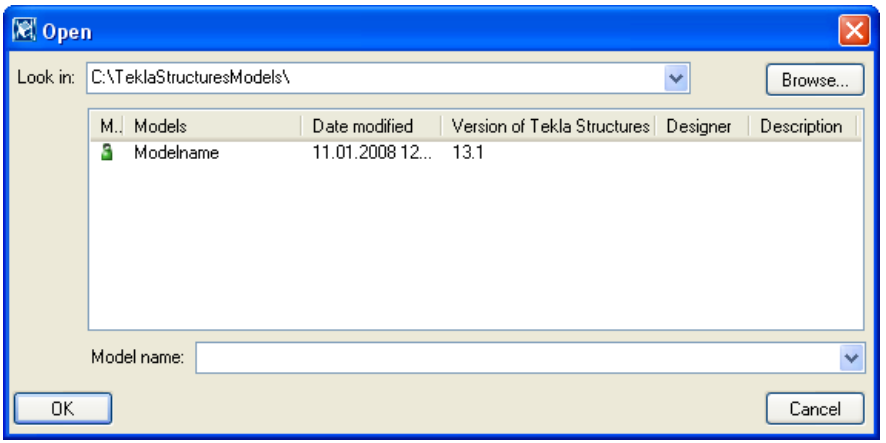


When extracting the zip file, all paths are taken along, so that the file structure stays intact. Make sure that the option **Use folder names** is checked.

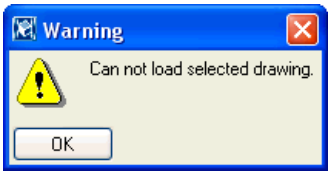


The “db1” file only

Sometimes, only the ***.db1** file is send. To open this Tekla Structures model, you first have to create a folder which name is identical to the name of the ***.db1** file.



However, opening a drawing will be impossible, as the drawing files (*.*dg*) are missing! When trying to open a drawing from the drawing list, an error will appear:



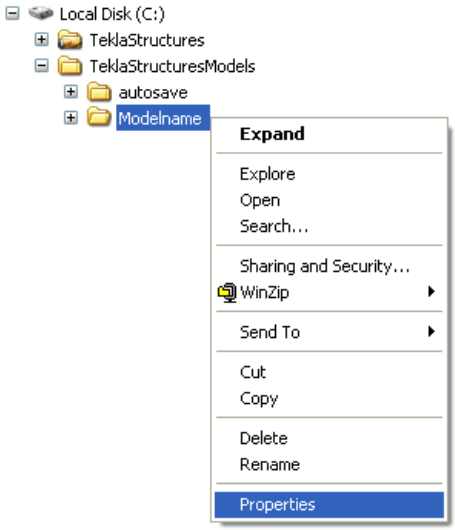
Because none of the drawings in the drawing list can be opened, first you have to delete the drawings in the list. After that, new drawings can be created. Note that the new created drawing will be stored in the model folder, instead of the usual ‘drawing’-folder. This also counts for e.g. saved component- or view settings. You have two create the sub folders **attributes** and **drawings** manually, if you want to keep the initial structure intact!

Tekla Structures models on a CD-rom

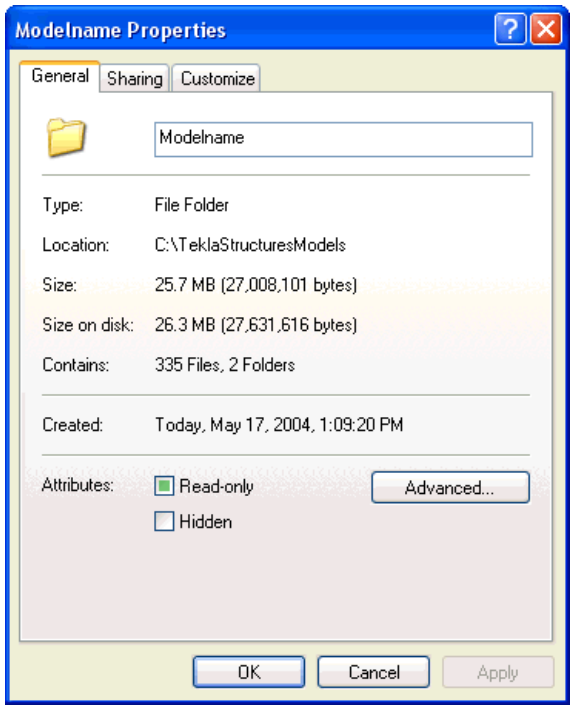
A model that has been received on CD-rom and copied to the hard disk, always has the ‘read only’ feature enabled. To be able to save the model, the read only feature has to be unchecked first.

Step plan

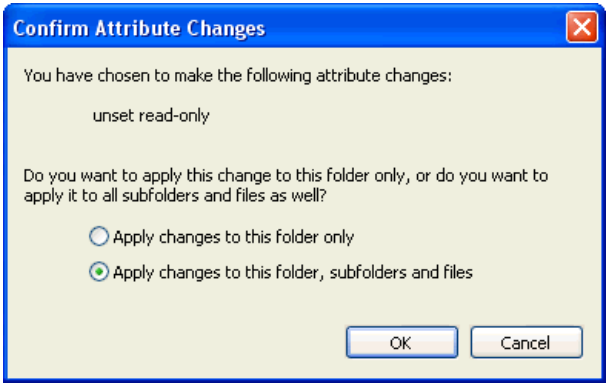
In the Windows Explorer, go to the model folder. Right mouse click the model-folder.



A dialog box will appear:



Uncheck the **Read only** option. Then click <**Apply**> and **<OK>**. The following dialog box appears:

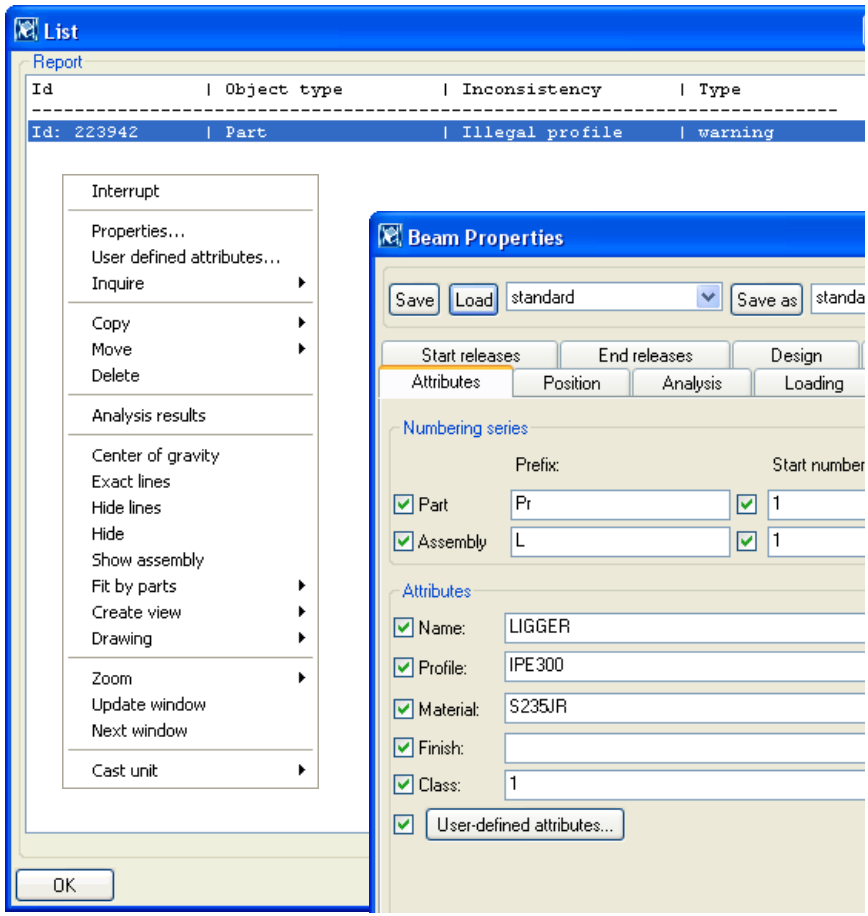


Select the option **Apply changes to this folder, subfolders and files**, than click **<OK>**. The Read Only attribute is now removed from all files and the Tekla Structures model can now be saved.

Check database

A third-party model has always to be checked for errors and if all profiles are present in the model. This command is located in menu **Tools > Check Database**.

The possible errors will be stored in the log file *check_database.log* in the model folder and displayed on your screen, in the list, right click a line for the properties:



From this list, you can recognize which profiles are missing. These are indicated as **Illegal profiles** in the list. You can add these profiles to the profile catalog. If there are no errors in the model, the following message will be displayed in the status bar:

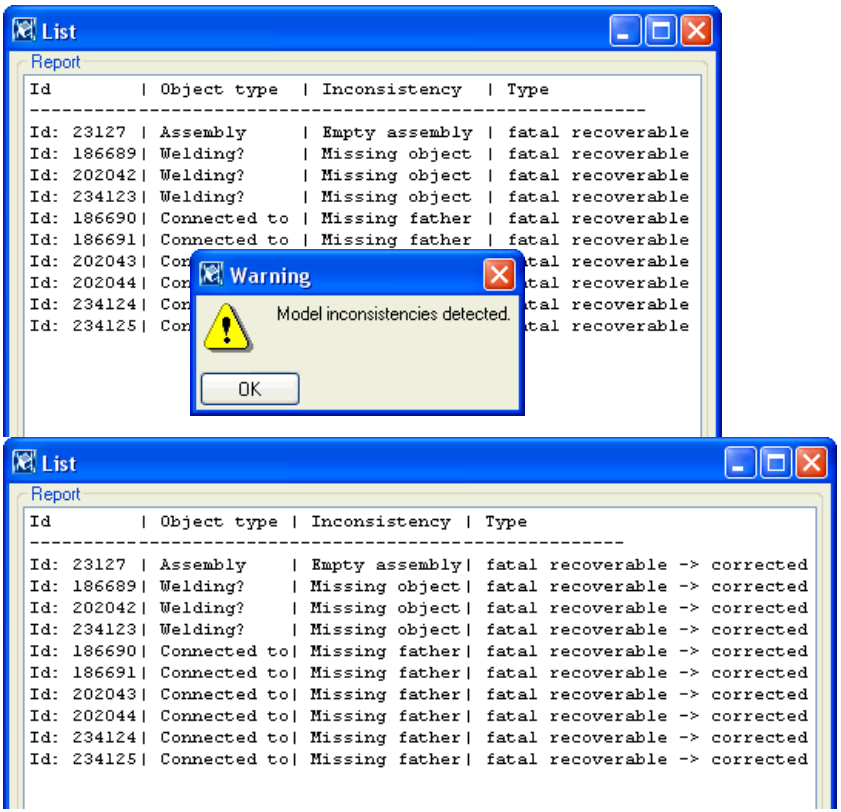
"Database checked and OK"

Correct database

This command is equal to the **Check database** command, but has possibility to recover errors. It can be necessary to repeat the Correct database command until all errors are gone. Check the result with the **Check database** command.



This command does not add profiles to the catalog, this **always** has to be done manually.



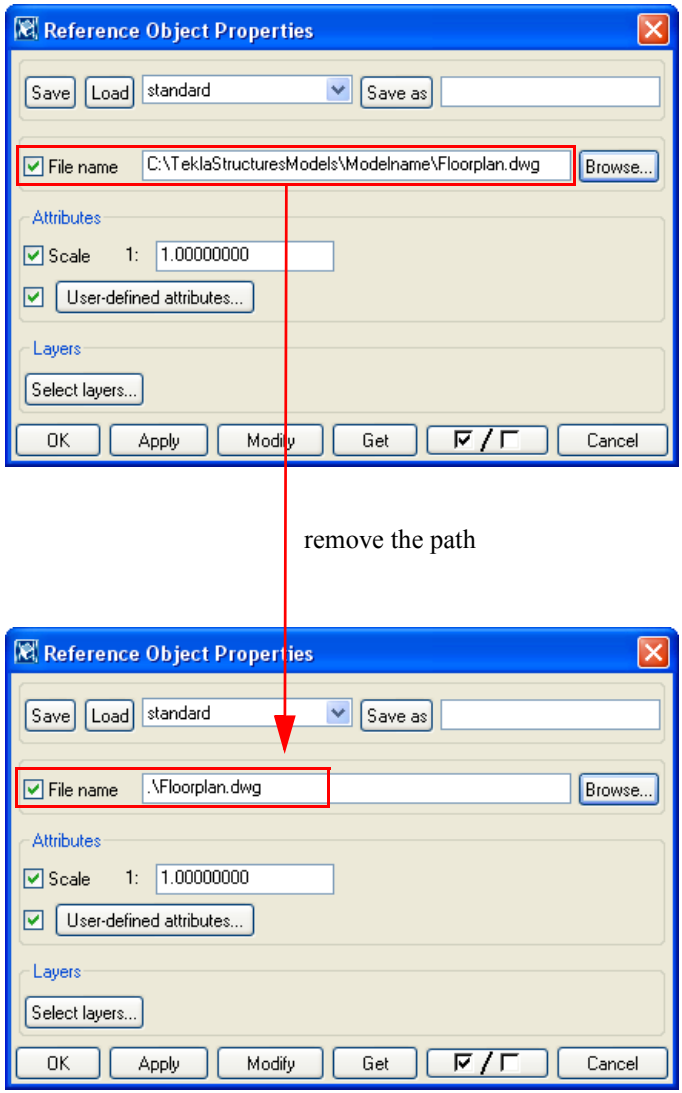
DWG/DXF, text files and reference-objects

Do not enter the path so that Tekla Structures can find the file at all times.

There is a maximum number of characters for a path name!

Step plan

Store the file in the Tekla Structures model folder. As soon as this file is added to the model (in the example: ***Reference model)*** remove the path:



Layout files and templates

*.*lay* (layout) and *.*tpl* (templates) files belong together. If the *.*lay* files are send with the model, make sure you also send the template files (stored in the **attributes** folder). By default the *.*lay* files are stored in the Tekla Structures system-folder.

Catalogs

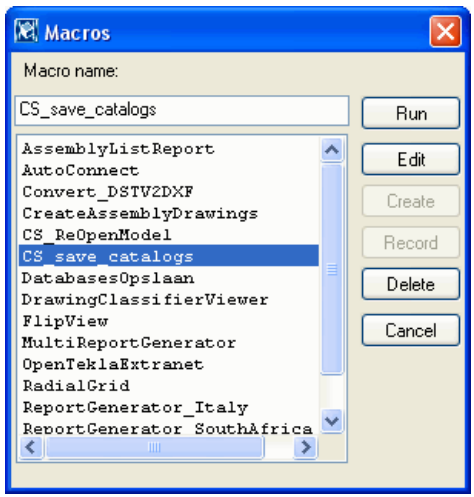
Catalog	Name
profdb.bin	profile catalog
matdb.bin	material catalog
screwdb.db	bolts catalog
assdb.db	bolt assembly catalog

These files are located in the model folder.

You can use the macro **CS_save_catalogs** to store these databases in the model folder quickly. Run this macro once.

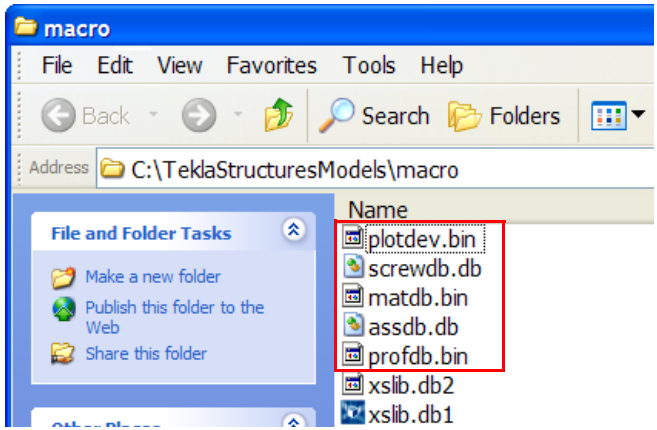
Step plan

- Go to **Tools** > **Macros**, the following dialog box appears:



- Select the routine **CS_save_databases** and click **<Run>**

The macro starts and saves the plotter- (plotdev.bin), the material- (matdb.bin), the bolts (screwbd.bin), the bolt assembly- (assdb.bin) and the profile database (profdb.bin) in the model folder.



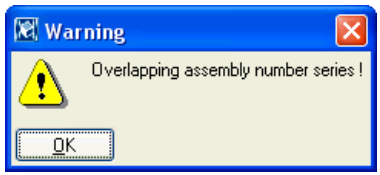
When the model is being zipped, the just saved databases are also being zipped.

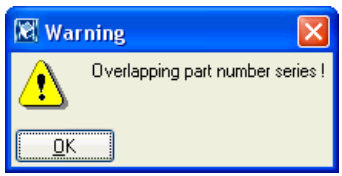
Ini files

If settings are used which differ from the default ones, also include these file settings in the zipped model file. You can also hand the changes that have been made to the default ini file.

Overlap in the numbering

Sometimes, when numbering the model, the following message may appear:





This warning appears when there is a matter of overlapping numbering series: 2 different assemblies/ parts haven identical part- or assembly numbers. The user is responsible that there are enough free numbers in the numbering series.

Hand made changes in start numbers result in overlaps in your model. Whether it is about single parts or assemblies, an error message reports.

Example

The model contains 6 different parts in a series with prefix **P** and start number **1** (**P1** -> **P1**, **P2**, **P3**, **P4**, **P5**, **P6**) and there are 3 different parts in another series with prefix **P** and start number **5** (**P5** -> **P5**, **P6**, **P7**). The numbers **P5** and **P6** will cause a conflict.

A warning will appear in Tekla Structures when an overlap in numbering is found.

The overlapping series are stored in the numbering history (log file).

When a report like that appears, go to **Tools** > **Display log file** > **Numbering history log**. The following dialog box appears:

```
*** Numbering (maic): Fri Jan 09 14:57:57 2004
  Full numbering
  Compare modified to old parts
  Compare new to old parts
  Tolerance: 1.010000
                L/1 Max Part     0 Max Rebar     0 Max Assembly     1
                K/1 Max Part     0 Max Rebar     0 Max Assembly     2
                P/1 Max Part     2 Max Rebar     1 Max Assembly     0
               Pr/1 Max Part     3 Max Rebar     1 Max Assembly     0
                L/1 Max Part     0 Max Rebar     0 Max Assembly     1
                K/1 Max Part     0 Max Rebar     0 Max Assembly     2
                P/1 Max Part     2 Max Rebar     1 Max Assembly     0
               Pr/1 Max Part     3 Max Rebar     1 Max Assembly     0
                L/1 Max Part     0 Max Rebar     0 Max Assembly     1
                K/1 Max Part     0 Max Rebar     0 Max Assembly     2
                P/1 Max Part     2 Max Rebar     1 Max Assembly     0
               Pr/1 Max Part     3 Max Rebar     1 Max Assembly     0
Assembly id:     925  series:K/2  K/0 -> K/2
Overlap: K 1+2 -> K 2+1
************************************************
```

Click an ID line in the log file, the corresponding profile in the model will highlight.



Solve these problems immediately, this may save a lot of time later!

Sending models to the Construsoft Help desk

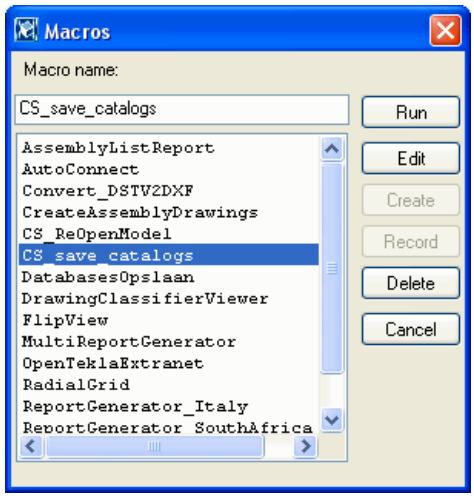
When you send a model (or just a problem) to the Construsoft help desk, you have to comply for some rules. Because of that, the help desk can analyze your problem much faster and offer you a solution.

- In the e-mail, give a clear explanation about the problem and in which version the problem occurs. Also possibly note who the one is at the help desk you discussed with.
- Only send what is important in a model, so no huge models. Make us of the command **Copy from model**

- Divide the “problem” in a phase so that there is no need for searching by the help desk, use the **Phasemanager**
- Fit, if necessary, the work area to the intended parts
- Send the correct catalogs, when you have added some profiles, for example, also send the file **profdb.bin**. You can make use of the macro is macro **CS_save_catalogs**. This macro saves all databases in the modelfolder. This can useful when you want to email a model to Construsoft Support and to be sure that all (modified) databases are present in de model:

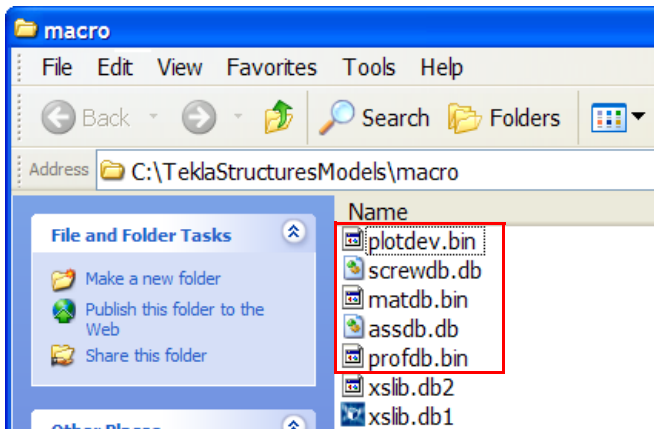
Step plan

1. Go to **Tools** > **Macros**, the dialog appears:



2. Select the routine **CS_save_databases** and click **<Run>**

   The macro starts and saves the plotter- (plotdev.bin), the material- (matdb.bin), the bolts (screwbd.bin), the bolt assembly- (assdb.bin) and the profile database (profdb.bin) in the model folder.



   When the model is being zipped, the just saved databases are also being zipped.



These e-mails can be treated much faster!

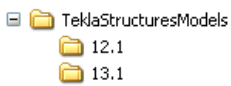
Always use the correct e-mail address:

Questions about	E-mail address
Support	ts-support@construsoft.com
Licences	licenses@construsoft.com

Creating a Back Up

Not only storing Tekla Structures model folders is important, also creating back-ups of model folders and Tekla Structures settings is very important, such as:

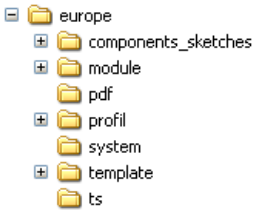
- the folder in which the models are stored, i.e. **TeklaStructuresModels.** Inhere are all models are stored or a sub folder, e.g. 13.1. Models created in an older version are possible backed up already.



- the **Europe** folder in

  disk:\TeklaStructures\version\environments

  This folder includes the sub folders **profil** (contains among other things the profile catalog *profdb.bin*) and **ts** (contains custom-made settings).





If multiple **Europe** folders are used for various customers, you need to create back ups of these folders too.

- the (modified) **ini** files**,** in case you work for several customers

3.20 Modify user defined attributes in the file objects.inp

Topics

This chapter is divided into following sections:

Adding a new tab (p.217)

Change the name of a user-defined attribute field (p.220)

Adding an empty attribute (p.222)

Adding a text line (p.223)

Part representation in model views (p.225)

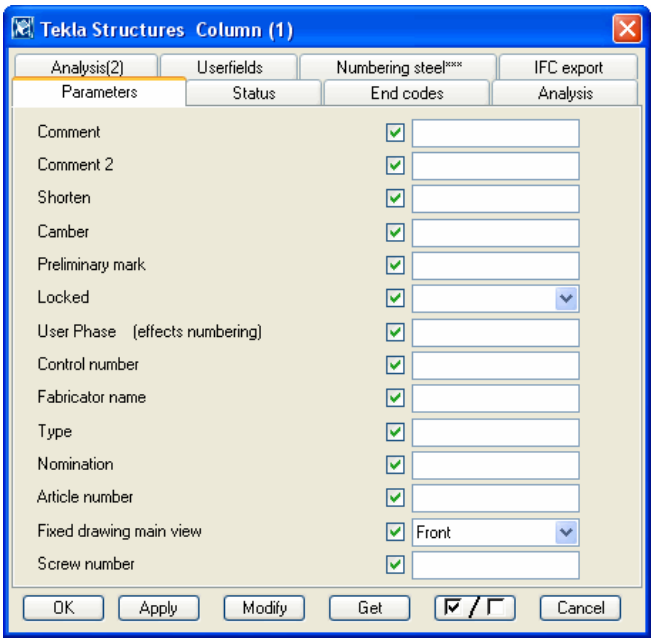
Filtering parts in model views (p.227)

Displaying user defined attributes in templates and reports (p.228)

Usage

Adding user-defined attributes fields in the file `objects.inp` in the model folder.

Many dialog boxes in Tekla Structures contain user-defined attributes for various objects, such as beams, columns, bolts and drawings.



You can modify the existing user-defined attributes fields, which are displayed in the dialog boxes, in the file `objects.inp`. You can also add one or more user-defined attributes fields or empty it, you can also add a new tab containing user-defined attributes fields.



Modifying the original file `objects.inp` in the folder `..version\environments\country-independent\inp`, must be carried out for every new installation.

You **cannot** copy the original file `objects.inp` from one to another version.

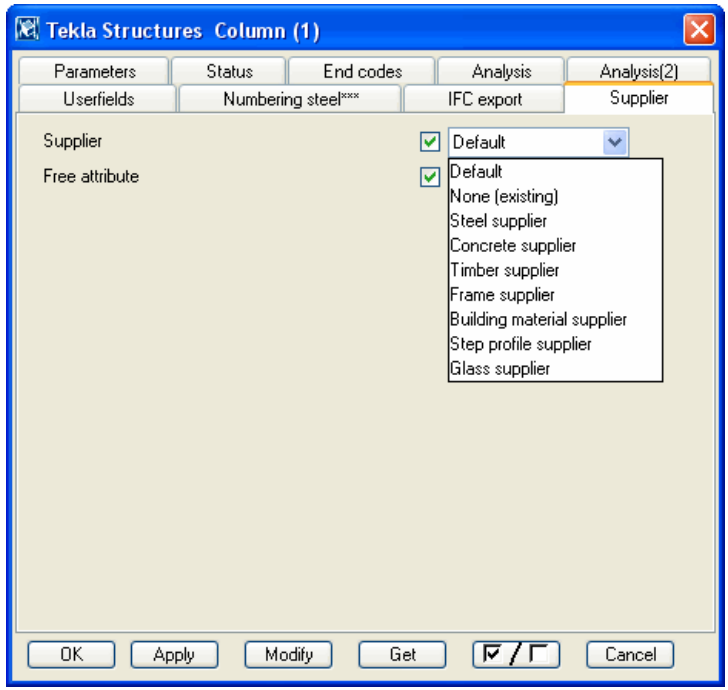
Adding a new tab is the best option.

Next you will find some examples to clarify this.

Adding a new tab

You want to add the tab `Supplier` for all available profiles:

In here you want to have a list box to select the supplier from that specific part. This, to easily select all parts which should be supplied by a steel supplier, for example.



You also want to make use of a free attribute.

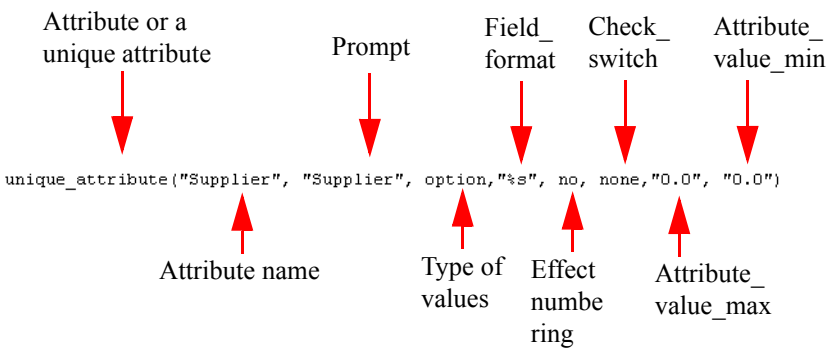
Step plan

1. Download the model **Example objects.inp** from the Construsoft Xtranet.
2. Open the file `objects.inp` in the model "example objects.inp", the following appears:

```
part(0,"Part")
{
 tab_page("Supplier")
 {
   unique_attribute("Supplier", "Supplier", option,"%s", no, none,"0.0", "0.0")
   {
     value("j_default", 2)
     value("None (existing)", 0)
     value("Steel supplier", 0)
     value("Concrete supplier", 0)
     value("Timber supplier", 0)
     value("Frame supplier", 0)
     value("Building material supplier", 0
     value("Step profile supplier", 0)
     value("Glass supplier", 0)
   }
   unique_attribute("Free attribute", "Free attribute", string,"%s", yes, none,"0.0",
   {
       value("0.0", 0)
   }
 }
 tab_page("Supplier","Supplier",1)
 modify(1)
}
beam(0,"j_beam")
{
tab_page("Supplier","Supplier",22)
```

1 2 3

1 The fields in the file:



Field	Description
Attribute	Regular attribute, which is copied with other part properties.
Unique attribute	Non-copyable attribute. The value of the attribute is never copied to another part. For example part checking status attributes are usually non-copyable.
Attribute name	Attribute name, used to find the attribute value.
Prompt	The text Tekla Structures displays in the dialog box.
Type of value	`integer`, `float` for numbers `string` for text `option` for lists `date` for date with small calendar `date_time_min` for date and time [12:00] with small calendar `date_time_sec` for date and time [12:00:00] with small calendar
Field_format	Definition of the field format in the dialog box. Uses C program terminology: • "%s" for strings • "%d" for numbers
Effect numbering	See **Effect on numbering** (p.221)
Check_switch	none, check_max, check_min, check_maxmin
Attribute_value_max	The maximum possible value for the attribute
Attribute_value_min	The minimum possible value for the attribute

2 The values which appears in the list box.

3 Defines in which parts this list box appears.

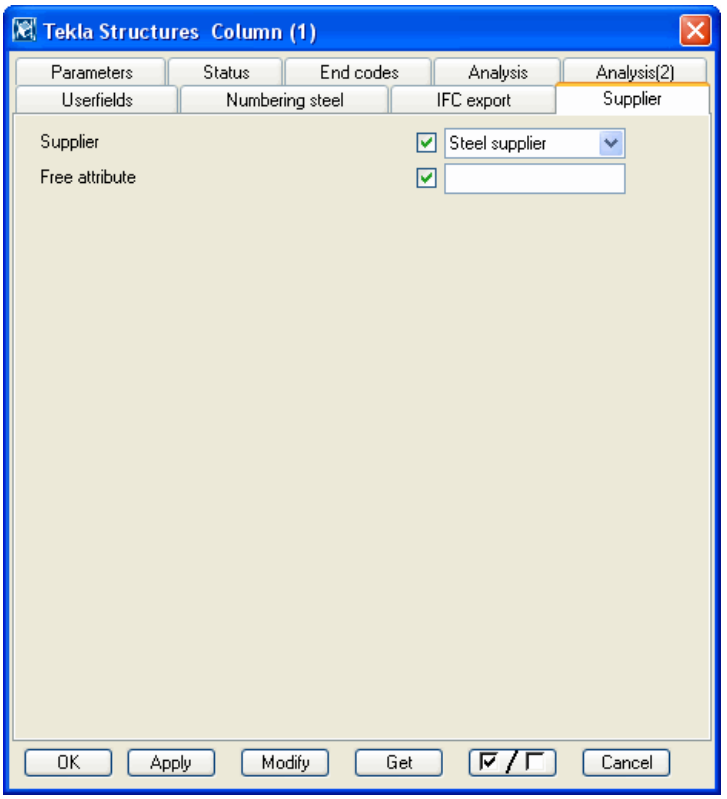
In this model you see that the modified file `objects.inp` is stored in the model folder. Because of this you can easily work per model or for a specific customer and use modified user defined-attributes.

The file in the model folder and the original file are merged so that if there are user-defined attributes in any of the files, they are displayed in Tekla Structures. 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 last read `objects.inp` file will be used. If Tekla Structures encounters attributes with the same name but from different types, an error message will appear.

Change the name of a user-defined attribute field

You want to change the name of the user-defined attribute field **Free attribute** in the **Column** dialog box, which by default looks like this:



Step plan

1. Open the file objects.inp in the model folder using a standard text editor (e.g. Notepad).
2. Find the line:

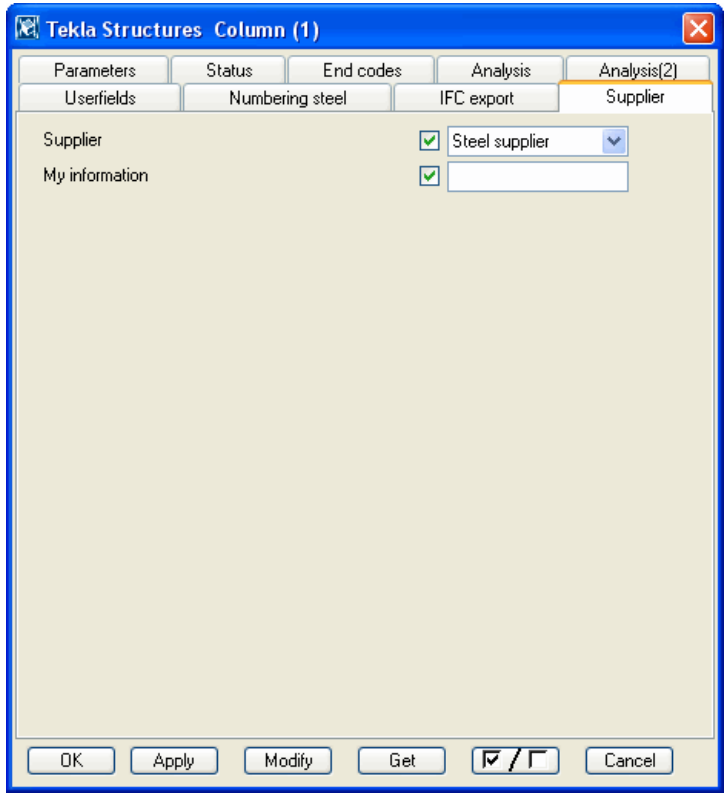
   ```
   unique_attribute("Free attribute", "Free
   attribute", string,"%s", no, none,"0.0", "0.0")
   ```

3. Change the line:

   ```
   unique_attribute("Free attribute", "My
   information", string,"%s", no, none,"0.0", "0.0")
   ```

4. Save the file objects.inp.

5. Close Tekla Structures and restart the programme for the changes to take effect. The **Column** properties dialog box will now look like this:



Effect on numbering

You can set whether the user-defined attribute affects numbering or not. This is useful when you have members that are identical in all respects, apart from their user-defined attributes.

If you want Tekla Structures to consider the user-defined attribute when numbering, use the option `yes` in `objects.inp`:

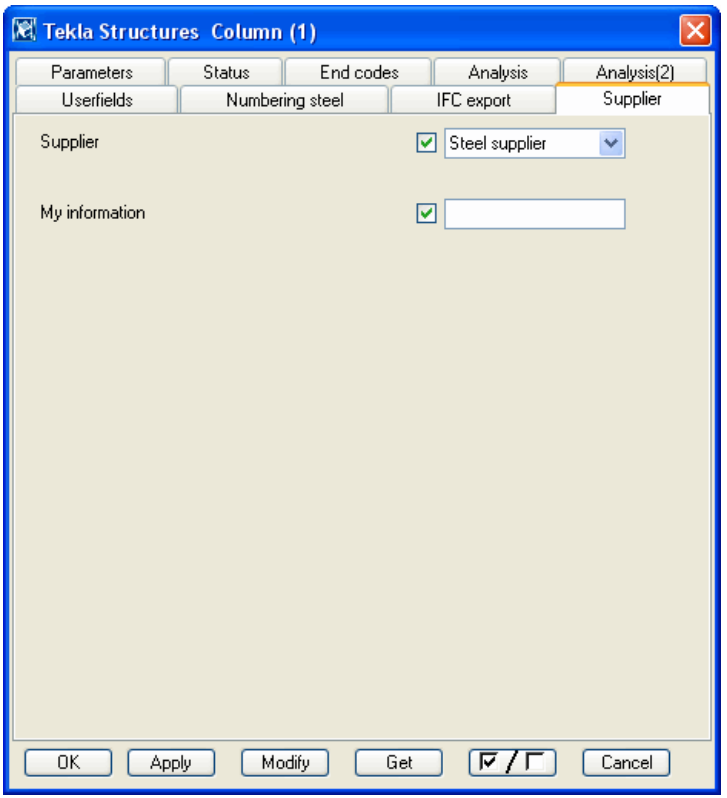
```
unique_attribute("Free attribute", "My
information", string,"%s", yes, none,"0.0", "0.0")
```

In this case, Tekla Structures assigns different marks to parts that are identical, but have different user-defined attributes.

When you insert **`no`** instead of **`yes`**, then this line will not affect numbering.

Adding an empty attribute

You want to add an empty user-defined attribute field in the **Column** dialog box to make it look like this:



Step plan

1. Open again the file objects.inp in the model folder using a standard text editor (e.g. Notepad).
2. Find the next line:

   ```
   unique_attribute("Free attribute", "My
   information", string,"%s", no, none,"0.0", "0.0")
   ```

3. Add the following line:

   ```
   attribute("LABEL1", " ", label, "%s", no, none,
   "0.0", "0.0")
   ```

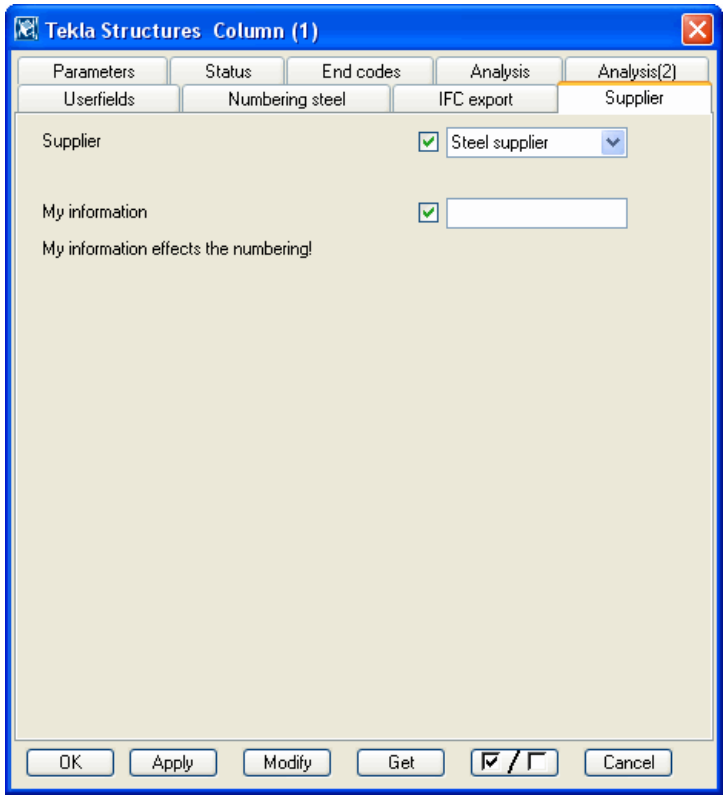
In the file objects.inp

```
}
attribute("LABEL1", " ", label, "%s", no, none, "0.0", "0.0")
unique_attribute("Free attribute", "My information", string,"%s", no, none,"0.0", "0.0")
{
```

4. Save the file objects.inp.
5. Close Tekla Structures and restart the program for the changes to take effect.

Adding a text line

You want to add a text line to the **Column** dialog box to make it look like this:



Step plan

1. Open again the file objects.inp in the model folder using a standard text editor (e.g. Notepad).
2. Add the following line:

   ```
   attribute("LABEL1", "My information effects the
   numbering!", label, "%s", no, none, "0.0", "0.0")
   ```

In the file objects.inp

```
}
attribute("LABEL1", " ", label, "%s", no, none, "0.0", "0.0")
unique_attribute("Free attribute", "My information", string,"%s", no, none,"0.0", "0.0")
{
    value("0.0", 0)
}
attribute("LABEL1", "My information effects the numbering!", label, "%s", no, none, "0.0", "0.0")
```

3. Save the file objects.inp.
4. Close Tekla Structures and restart the program for the changes to take effect.

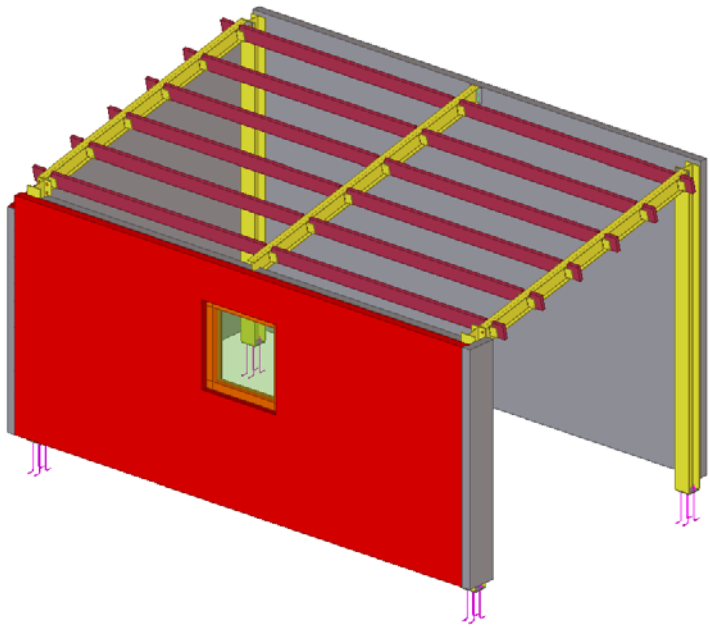
The modified file objects.inp now looks like this:

```
part(0,"Part")
{
 tab_page("Supplier")
  {
    unique_attribute("Supplier", "Supplier", option,"%s", no, none,"0.0", "0.0")
    {
        value("j_default", 2)
        value("None (existing)", 0)
        value("Steel supplier", 0)
        value("Concrete supplier", 0)
        value("Timber supplier", 0)
        value("Frame supplier", 0)
        value("Building material supplier", 0)
        value("Step profile supplier", 0)
        value("Glass supplier", 0)
    }
    attribute("LABEL1", " ", label, "%s", no, none, "0.0", "0.0")
    unique_attribute("Free attribute", "My information", string,"%s", no, none,"0.0", "0.0")
    {
        value("0.0", 0)
    }
    attribute("LABEL1", "My information effects the numbering!", label, "%s", no, none, "0.0", "0.0")
  }
}
beam(0,"j_beam")
{
tab_page("Supplier","Supplier",22)
modify(1)
}
contourplate(0,"j_irregular_plate")
{
tab_page("Supplier","Supplier",22)
modify(1)
}
beamortho(0,"j_ortho_beam")
{
tab_page("Supplier","Supplier",22)
modify(1)
}
twinprofile(0,"j_twin_profiles")
{
tab_page("Supplier","Supplier",22)
modify(1)
}
column(0,"j_column")
```

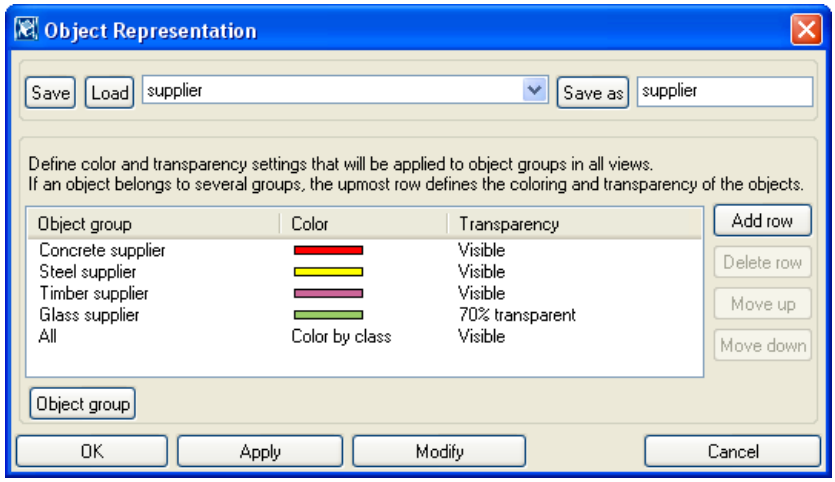
Part representation in model views

You can now define the colors and transparency of parts in model views by which the user defined attribute `Supplier` in taken into account.

Model view



To open the dialog box **Object Representation**, click **Setup** > **Object representation...**. Load the setting **supplier**, the following dialog box appears:



This dialog box includes several default settings taking the supplier into account (in the Object group). Parts for which **concrete supplier** is selected in the list box in the user defined attribute `Supplier` are represented in the color **red**, parts for which **glass supplier** is selected in the user defined attribute `Supplier` are represented in the color **green** plus a transparency of 70%, etc.

In the dialog box, the object representation manager starts "assinging" colors at the top of the list. The last line in the dialog box always contains the line **All - Color by class.** This means that the remaining parts who do not belong to any object group, will be represented according to their class.

When you press <**Modify**>, this setting will be applied for **all** model views.

You can also create Object groups by yourself.

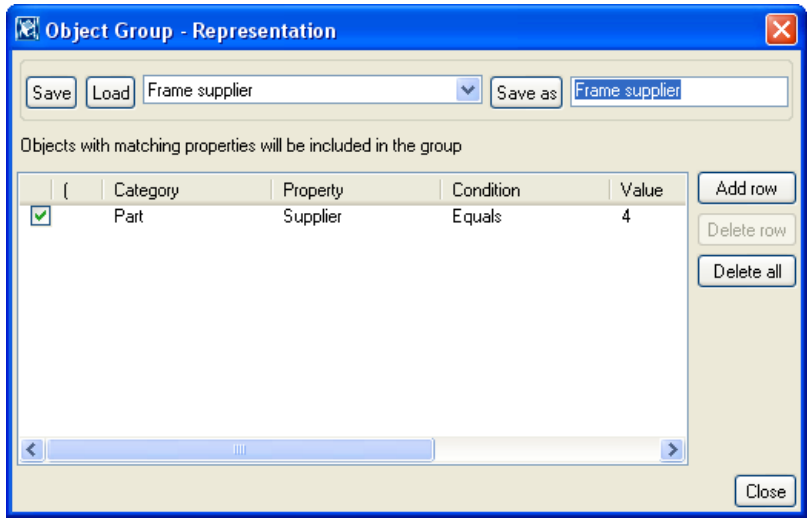
Example

We will create a new Object group for parts for which **frame supplier** is selected in the list box in the user defined attribute `Supplier.` These parts are represented in the color **light blue.**

Step plan

1. Load the setting **Supplier** in the dialog box **Object Representation** and click **Add row**. A Object group will be added.
2. Click **All**. A list box will pop-up that includes several option. These are the default options for Object groups. There is no option included directly to filter on **frame supplier**, therefore click **Create new group...**
3. The dialog box **Object group - Representation** appears.
4. Leave the **Category** option set to **Part**.
5. Select **Supplier** in the column **Property**.
6. Leave the **Condition** option set to **Equals**.

7. In the column **Value**, select the option **Select from model...** in the list box and select a part (frame) in the model, the value 4 will be entered. The dialog box now looks like this:



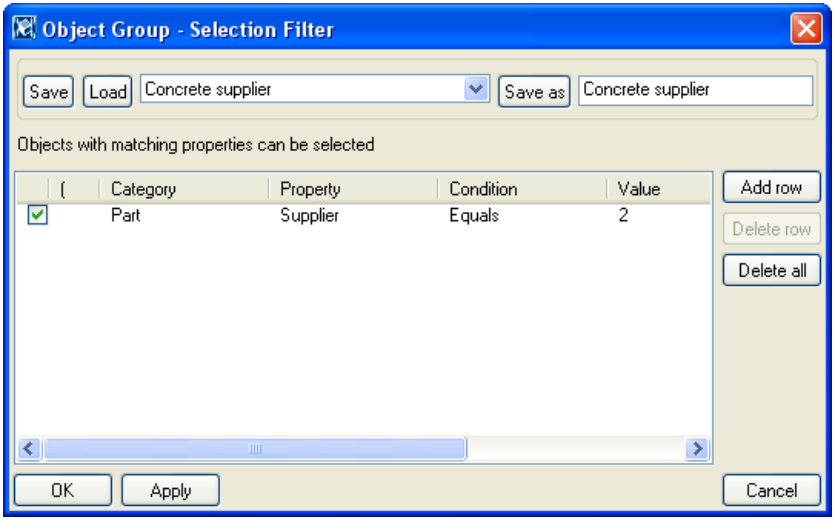
8. Store this setting as **Frame supplier**
9. Click <**Close**>.
10. Click **All** in the dialog box **Object Representation** and select **Frame supplier** in the list box
11. Select the color and possibly set (i.e. for glass) the transparency
12. Click <**Apply**> and <**Modify**>

The setting **supplier** will be applied for all model views.

Filtering parts in model views



You can use dialog box **Object group- selection filter** to define which parts you can select. Go to **Setup**> **Selection filter...** or click the icon, the following dialog box appears:



In this dialog box you can also create lots of settings, just like you did before for the representation of the model views.

Select filter dialog box

You can find the settings in the select filter dialog box:



Displaying user defined attributes in templates and reports

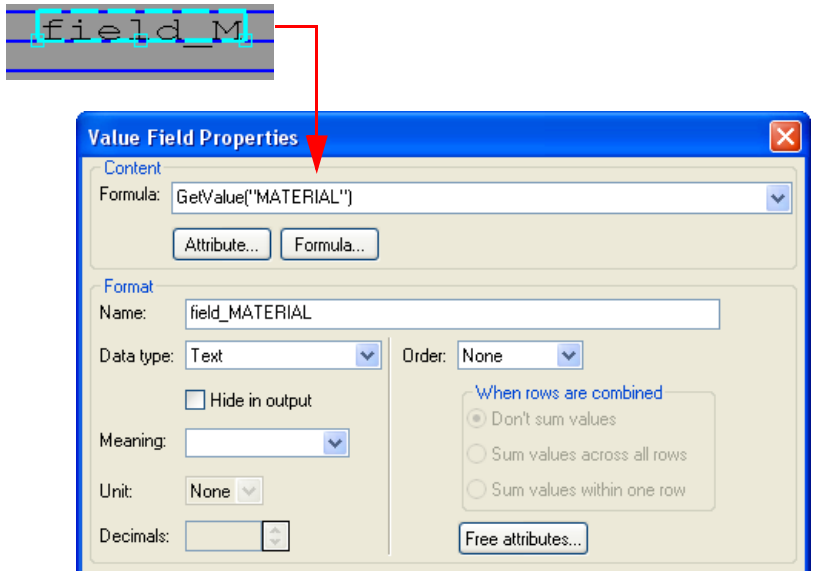
To display the added user defined attributes in templates and reports, you need to use the name of these user defined attributes as they are described in the file `objects.inp`.

Example

You want to display the supplier on a report for the selected parts:

Step plan

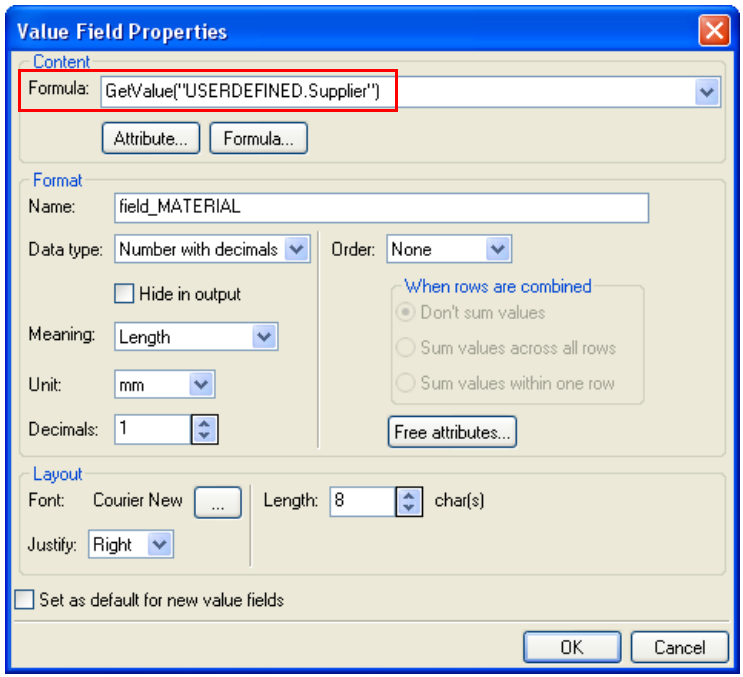
1. Open the report you want to modify in the Template Editor
2. Double-click on the **Value Field** you want to modify:



3. Now open the file `objects.inp` in the model folder of the model **Example objects.inp**
4. In the list box **Formula** in the dialog box **Value Field Properties,** modify the name of the user defined attribute as it is described in the file `objects.inp`:

```
part(0,"Part")
{
 tab_page("Supplier")
  {
    unique_attribute("Supplier", "Supplier", option,"%s",
    {
        value("j_default", 2)
        value("None (existing)", 0)
        value("Steel supplier", 0)
        value("Concrete supplier", 0)
        value("Timber supplier", 0)
        value("Frame supplier", 0)
        value("Building material supplier", 0)
        value("Step profile supplier", 0)
        value("Glass supplier", 0)
```

5. The contents of this line starts with text **USERDEFINED.** (it affects a user defined attribute at last):



6. If needed, you can also modify the other settings in the dialog box, next click <**OK**> and store the report in the Template Editor

7. Next generate a report and you will see that the suppliers are displayed on the report:

```
------------------------------------------------------------------------------
TEKLA STRUCTURES SUPPLIER LIST                        Page:     1
Model:example objects.inp                             Project:  project nummer
                                                      Date:     22.02.2008
                                                      Time:     13:24:18

------------------------------------------------------------------------------
Profile    Supplier           Number  Length(mm)      Area(m²) Weight(kg)
------------------------------------------------------------------------------
100*2650   Concrete supplier    1           6000         31.93      3576.0
==============================================================================
                                           6000         31.93      3576.0
------------------------------------------------------------------------------
112*88     Frame supplier       1           1000          0.42         5.2
==============================================================================
                                           1000          0.42         5.2
```

3.21 NC files and pop marks

Topics

This chapter is divided into following sections:

General (p.231)

Settings (p.232)

Files and part selection (p.233)

Holes and cuts (p.236)

Hard stamp (p.239)

Variable XS_NC_DISABLE_PIECE_IDENTIFICATION_ FIX (p.240)

Pop marks (p.242)

Parts to pop-mark (p.243)

Pop-marking options (p.252)

Extra information (p.253)



Use this command to create NC-files and MIS (*M*anufacturing *I*nformation *S*ystem)-lists of all parts or selected parts.



It is not possible to generate NC-files of curved beams. Use polybeams instead.

MIS-lists are generated according to the Dstv, EJE or KISS format and are stored in the model folder.

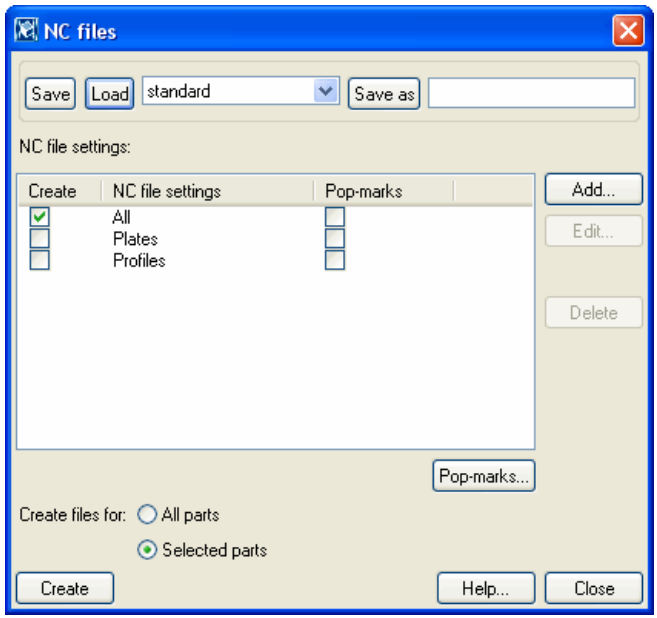
General

When you start creating NC-files, mind the following: a fitting operation fits a profile end to the selected fitting line. A fitting operation may occur only once to both ends of the profile. With fitting, the shortest part of the profile will be deleted automatically.



When shortening profiles, ALWAYS USE the **FIT**-command. When the Line-Cut command is used for shortening, the length values of profiles in NC files can be wrong!

To open the dialog box, go to **File** > **CNC** > **NC files..**. The following dialog box will appear.



Settings

By default, the following settings are included:

- All
- Plates
- Profiles

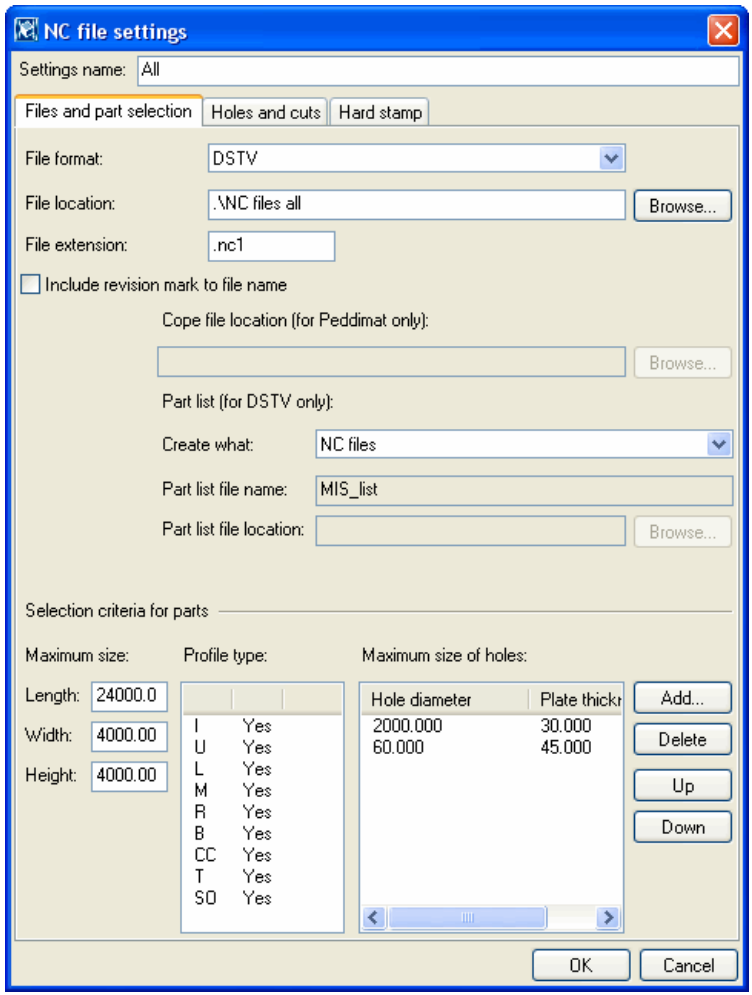
Because of that, only NC files for plates are created when the setting ***plates*** has been toggled on. You can also set for plates not to create slotted holes in the NC file according to DSTV, but to create a single hole in the centre of the slot.

When you press the button <**Create**>, Tekla Structures will check in the column **Create** which options are toggled on.

Select the setting **Plates** end press the button <**Edit>.** The following dialog box, which includes 3 tabs, will appear:

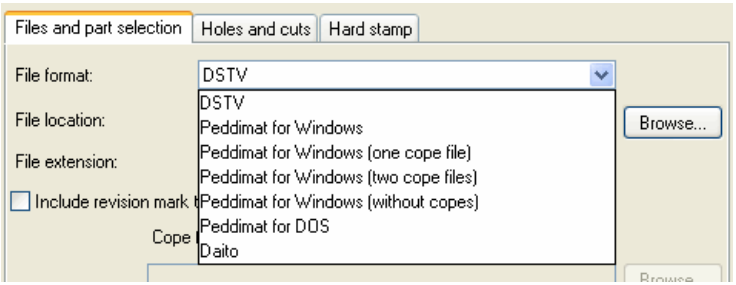
- **Files and part selection** (p.233)
- **Holes and cuts** (p.236)
- **Hard stamp** (p.239)

Files and part selection



File format

Use this option to set the output format for the NC files:

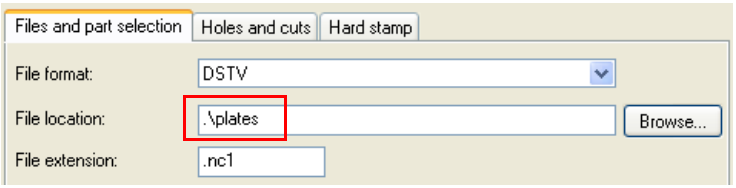


NC-files are generated according to the DSTV-standard (*D*eutscher *St*ahlbau *V*erband).

File location

The NC-files are stored in the model folder by default.

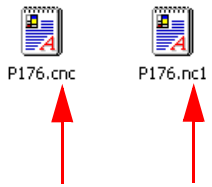
The variable **XS_MIS_FILE_DIRECTORY=.\NC** is set to store the NC-files in a specific folder in the current model folder.



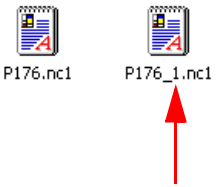
In case the variable above is not set and the text **.\plates** is entered in **File location**, a new folder named plates will be created in the model folder in which the NC-files are stored.

File extension

To define the file extension:



Include revision mark to file name



Create what

List box to define which files have to be generated. The options are:

`NC files`	NC-files only
`Part list`	MIS-list only (xsr-file)
`NC files and part list`	Both NC-files and a MIS-list
`Combined NC files and part list`	NC-data embedded in MIS-list (xsr-file)

- **Part list file name**

To define the name of the MIS-list.

- **Part list file location**

To define the location of the MIS-list.

Selection criteria for parts

- **Maximum size:**

The maximum size options define the maximum length, width, and height of the parts the machine can handle. Larger parts are sent to other machines.

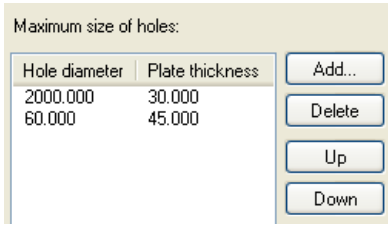
- **Profile type:**

The machine can handle all the profile types which have the option **Yes** in the Profile type list. Profile types are named according to the DSTV standard:

- I = I- H-profiles
- U = U-profiles
- L = L-profiles
- M = Rectangular tubes
- R = Round bars and tubes
- B = Plates
- CC = CC-profiles
- T = Tee shaped profiles
- SO = Z profiles and all the other types of profile

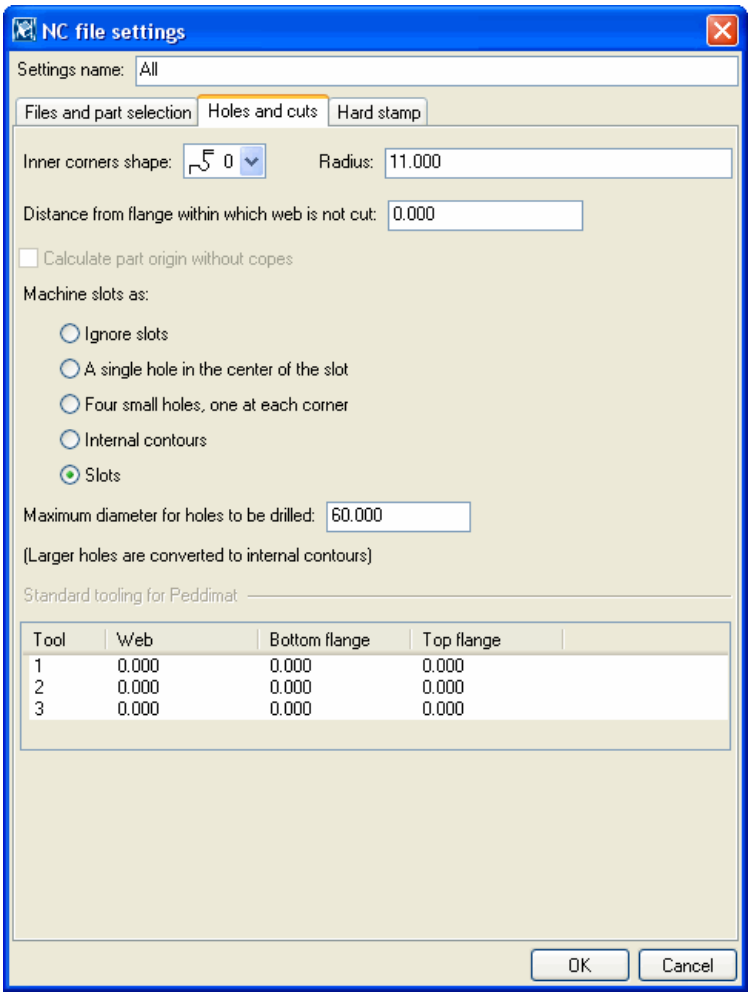
- **Maximum size of holes:**

The Maximum size of holes list defines how large holes the machine is able to drill. Parts containing larger holes are sent to other machines. The hole size is connected to material thickness (or plate thickness). Each row contains the maximum hole diameter and material thickness.



Row 60 45 means that when the material thickness is 45 mm or smaller, holes of a diameter of 60 mm and smaller will be drilled. The remaining rows serve the same purpose.

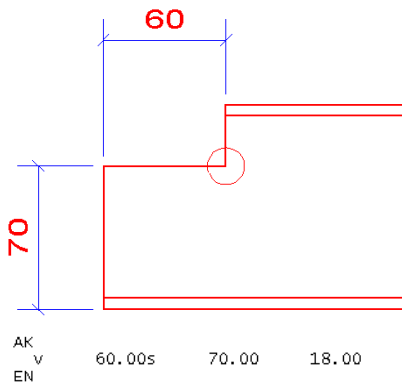
Holes and cuts



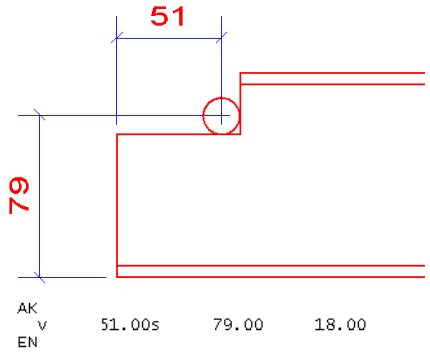
Inner corners shape

Use the options in the list box to define the shape of the inner contour, which can be radial, tangential or square. Inner contours are always rounded. Enter a radius for rounding in the Radius field:

- **Radial (0)**



- **Tangential (1)**



- **Square (2)**, no holes are created

Distance from flange within which web is not cut

Calculate part origin without copes

Machine slots as

- **Ignore slots**
  Deletes slotted holes from the NC-file.
- **A single hole in the center of the slot:**
  Drills a single hole in the center of the slotted hole.
- **Four small holes, one at each corner:**
  Drills 4 smaller holes, one at each corner.
- **Internal contours:**
  Flame-cuts the slots as internal contours
- **Slots:**
  Leaves slots as they are.

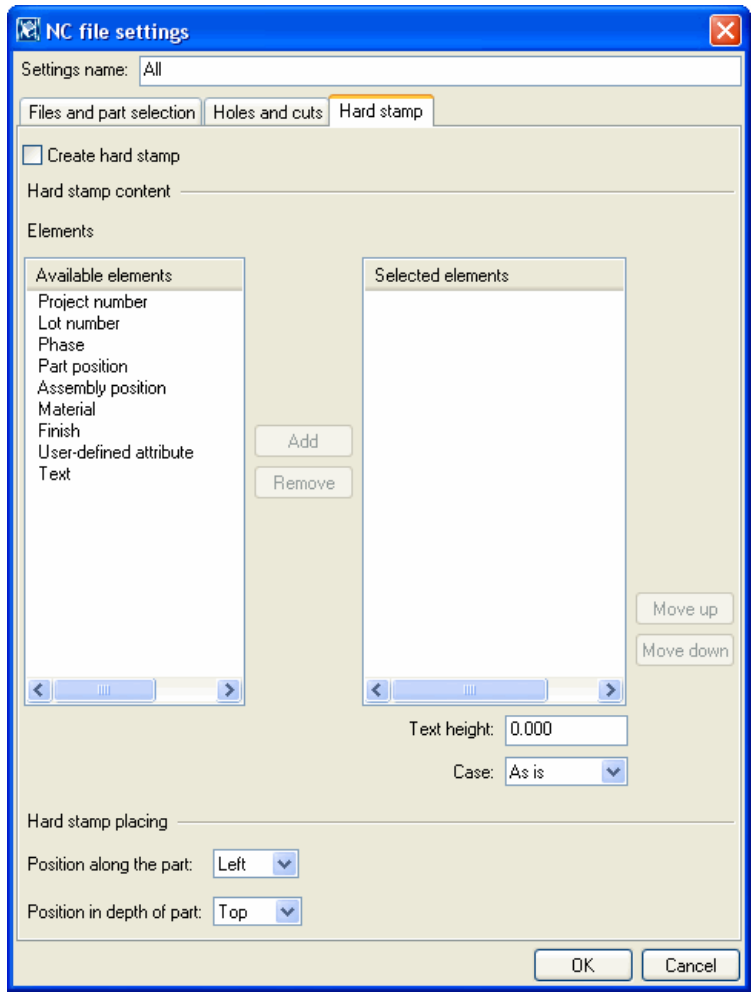
Maximum diameter for holes to be drilled

Holes and slotted holes that are larger than the maximum hole diameter are manufactured as internal contours.

Standard tooling for Peddimat

See for more information Tekla Structures Help.

Hard stamp



Create hard stamp

Use the options to define the properties of hard stamps on main and/or secondary parts.

In Tekla Structures you **can** create hard stamps for secondary parts, **not** for plates!

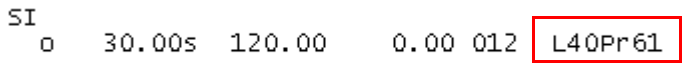
To include hard stamps for secondary parts in DSTV files, set variable XS_SECONDARY_PART_HARDSTAMP to TRUE in **Tools** > **Advanced options** > **CNC**. Set to FALSE to create hard stamps only for main parts.

In the column **Available elements** you can define which elements to include in hard stamps and the order in which the different elements appear (**Move up** / **Move down**). You can also define the text height and the case.

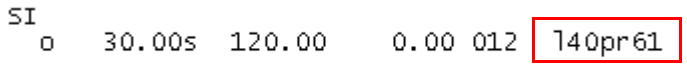
Case

Use the options in the list box **Case** to define whether the prefixes are taken as they are or taken as lower- or upper cases in the hard stamp. You have three options:

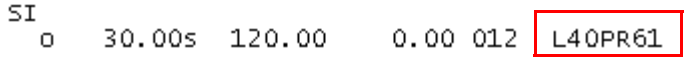
- **As is**



- **Lower case**



- **Upper case**



Hard stamp placing

Use the options in the Position along the part and Position in depth of part list boxes to define the position of hard stamps on parts.



When you have set the variable XS_HARD_STAMP_BY_ORIENTATION_MARK to TRUE you can effect the hard stamp location with the variable XS_ORIENTATION_MARK_MOVE_DIST_FOR_BEAMS that can be found in **Tools** > **Advanced options** > **Marking: parts**.

Variable XS_NC_DISABLE_PIECE_IDENTIFICATION_ FIX

In Tekla Structures the DSTV format was fixed to follow the DSTV standard even better than before, by switching the places of phase and drawing identification in the DSTV file header. Because of this change, the dstv2dxf-converter does not add correct texts to DXF files, and some CNC controller software interprets the DSTV files incorrectly.



To enable the fix and create similar DSTV files as before, set the variable XS_NC_DISABLE_PIECE_IDENTIFICATION_FIX to TRUE in **Tools** > **Advanced options** > **CNC**.

```
ST
** P26.nc1
  project nummer
  1
  P26
  P26
  S235JR
  1
  STRIP10*100
  B
```

TRUE (by default)

```
ST
** P26.nc1
  project nummer
  P26
  1
  P26
  S235JR
  1
  STRIP10*100
  B
```

FALSE

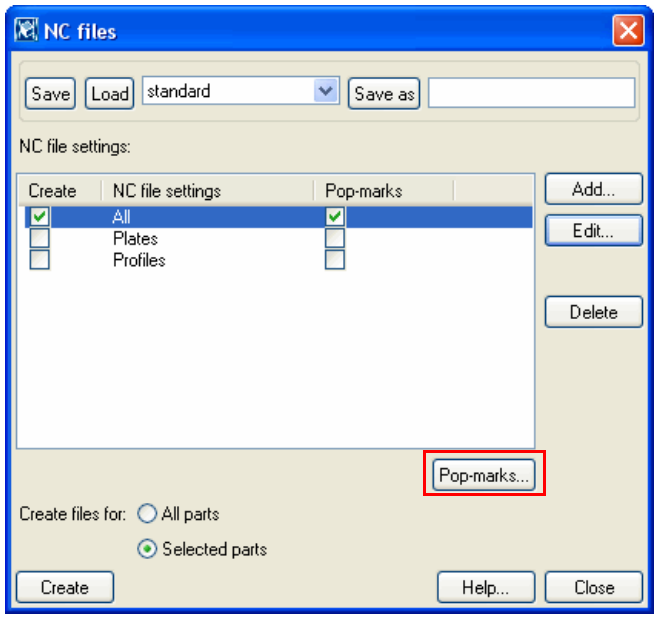
Pop marks

Automating the workshop is getting more and more important. Drill-saw units and punches are used more and more in the manufacturing process. NC-files can be generated by Tekla Structures and besides you can generate pop marks.

Pop marks are used as reference points for welded parts. These pop marks are created on main parts and reflect the position of secondary parts or holes in the main part.

In case you want to add pop marks to the NC-files, you need to define part names in the dialog box **Pop-mark settings....** Go to **File** > **CNC** > **NC files** > **Pop-marks.**

If the **Pop-marks** check box is selected, the pop-mark settings are used when you create NC files.





In case the generation of pop marks is switched on after all profiles already have been numbered, the model has to be numbered again, because pop marks can effect part numbering.

Tekla Structures gives **NO** warning when it necessary to number after the option to generate pop marks is switched on. Please be aware of this!

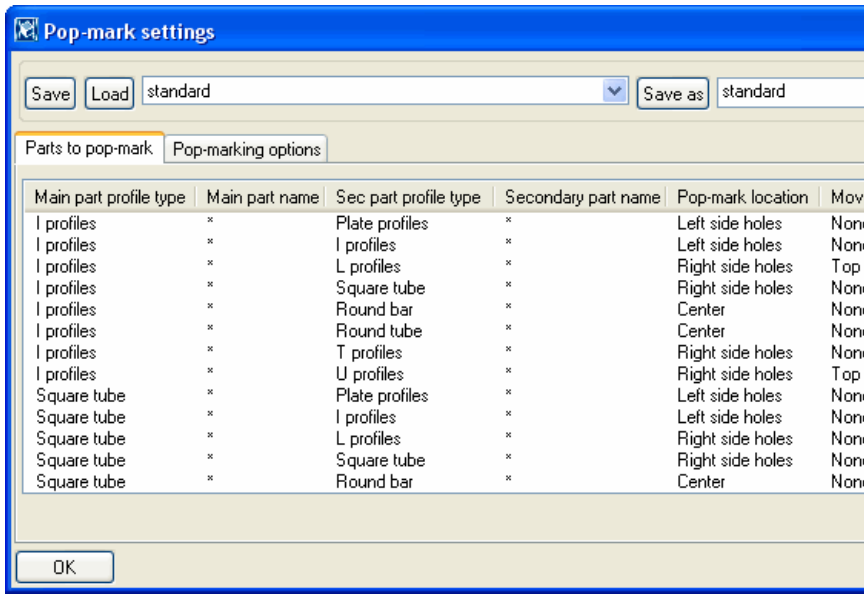
You can access the **Pop-mark settings** dialog box by clicking the **Pop-marks..** button in the **NC files** dialog box.

A dialog box containing 2 tabs appears:

- **Parts to pop-mark** (p.243)
- **Pop-marking options** (p.252)

When you store pop mark settings in the dialog box **Pop-mark settings** by using the button <**Save as**>, a file with the extension *.ncp will be created in the folder **attributes** in the current model folder.

Parts to pop-mark



You can define the following values for the pop-marks:

Option	Description
Main part profile type	The main part profile types to pop-mark. The list box contains profile types corresponding to the DSTV standard, e.g. U, I, M etc.
Main part name	The main part profile names to pop-mark. You can enter several part names separated by commas, e.g. COLUMN, BEAM etc.
Secondary part profile type	Secondary part profile type.
Secondary part name	Secondary part name. You can enter several part names separated by commas.

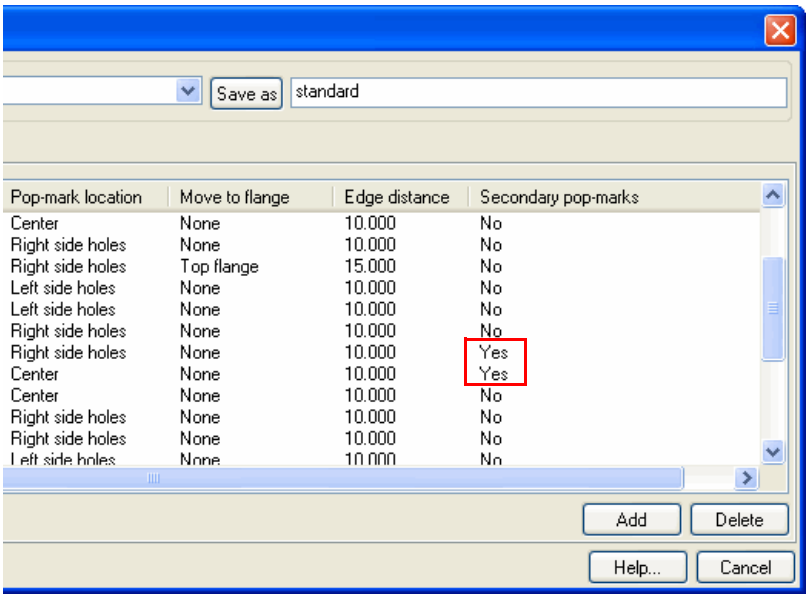
Option	Description
Pop-mark location, see also **Examples Pop-mark location** (p.249)	Defines how the secondary part is projected onto a main part.
	Both sides: Combines Left side and Right side.
	Left side: The left side of the secondary part is marked on the main part. The left side is the side of the secondary part that points in the negative direction of the main part local coordinate system.
	Right side: The right side of the secondary part is marked on the main part.
	Center: Center of the secondary part.
	Left side holes: Marks the main part with the hole positions on the left side of the secondary.
	Right side holes: Marks the main part with the hole positions on the right side of the secondary.
	Both side holes: Combines Left side holes and Right side holes.
	Middle line: Marks two points on the middle line of the secondary part x axis.
Move to flange, see also **Examples Move to flange** (p.251)	Moves pop-marks to the main part flange. The options are: None, Top flange, Bottom flange, Both flanges.
Edge distance, see also **Example Edge distance** (p.251)	Minimum distance from a pop-mark to the edge of the main part. Tekla Structures does not create pop-marks inside this distance. If a pop-mark is inside the defined edge distance, Tekla Structures moves it. (with the exception of **Center**).
Secondary pop-marks	To create pop-marks in secondary parts.



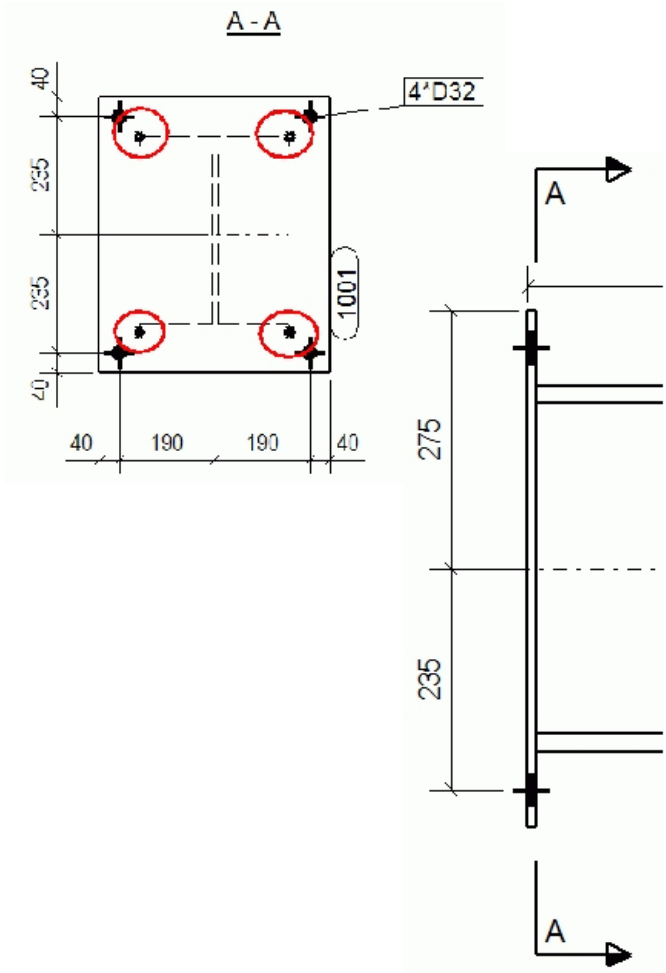
The order of the lines in the dialog box **Pop-mark settings** is important. Tekla Structures uses the match it finds, so you should put the most limiting definition first, and the most generic one last.

Pop-marks in secondary parts

You can create pop-marks into secondary parts. To do this, select **Yes** in the **Secondary pop-marks** column for the desired rows.



You can use this option for example when pop-marking base plates to help see how the base plate is located in relation to the column. In the example below, the option **Both sides** has been used.



You can create pop-marks even in situations where the primary and secondary part are not in full contact with each other.

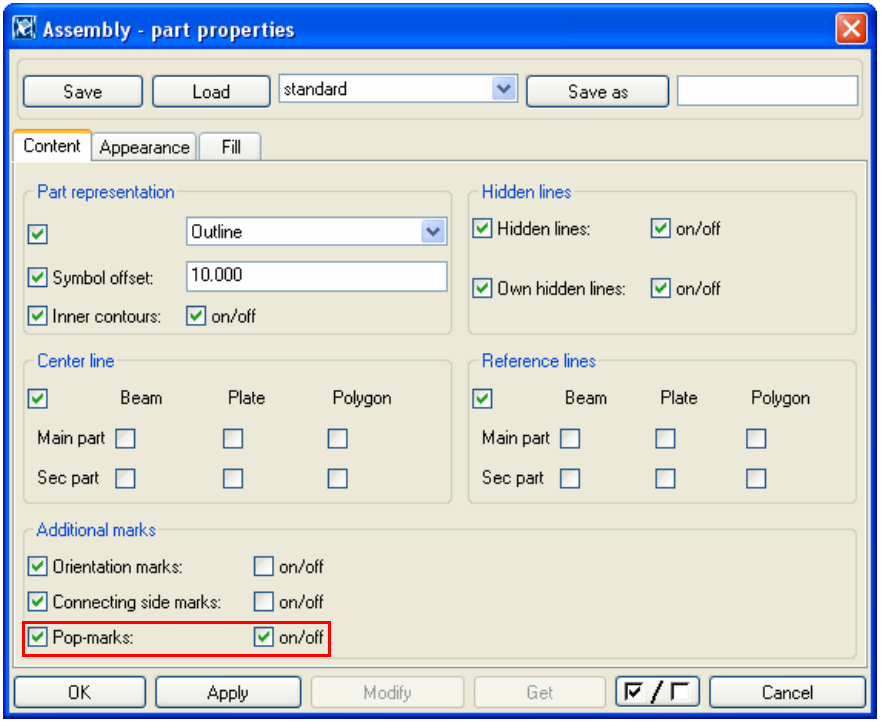
Show pop-marks in drawings

You can show pop-marks in assembly- and single part drawings.

Step plan

1. Double-click the background of the drawing to display the **Assembly drawing properties** dialog box.
2. Click **Part...**

3. In the **Assembly - part properties** dialog box, select the options **Pop-marks** and **on/off**.

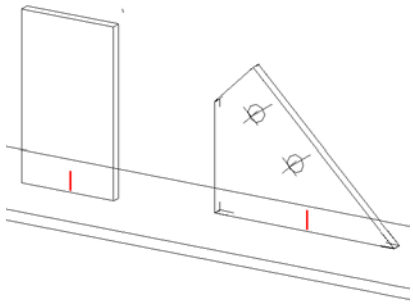
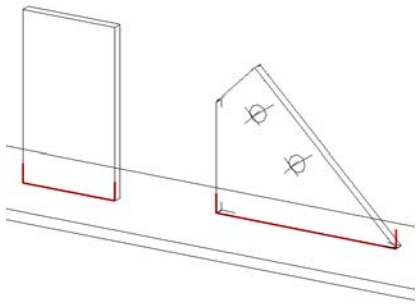
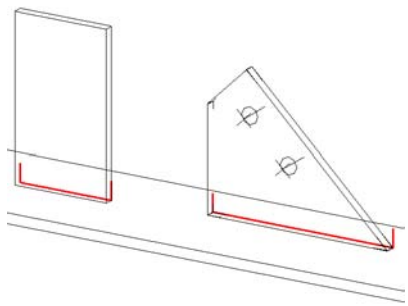
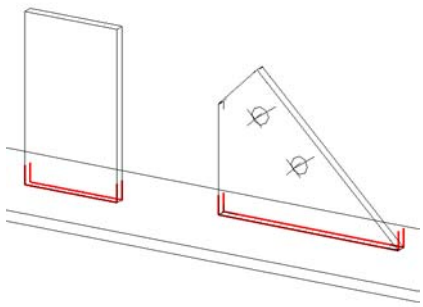


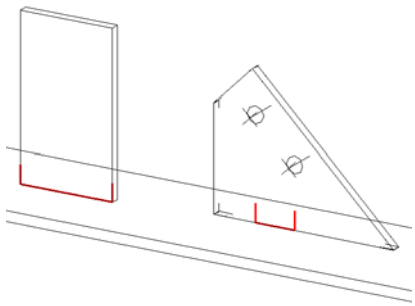
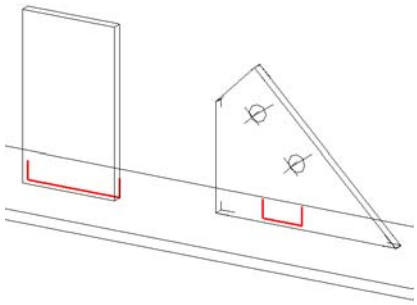
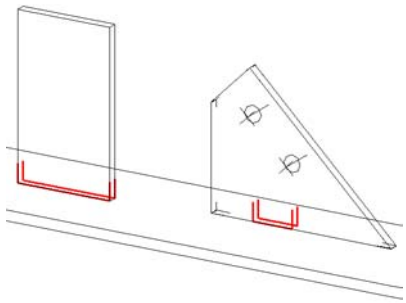
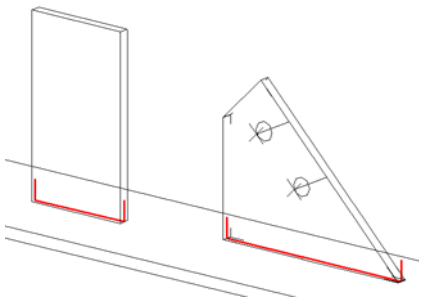
4. Click **Modify**.

You can customize the appearance of the pop-mark symbol that is displayed in the drawing by using the variables in **Tools** > **Advanced options** > **Drawing properties**:

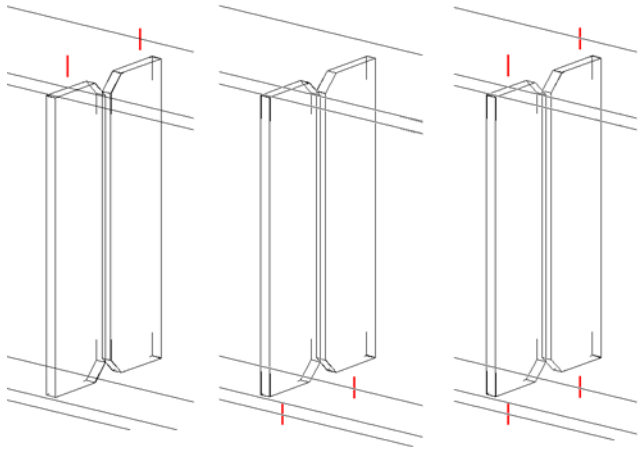
- `XS_POP_MARK_SYMBOL`: Enter the location of the pop-mark symbol. The default is `xsteel@0`.
- `XS_POP_MARK_HEIGHT`: Enter a decimal value for the height of the symbol in millimeters. The default is `2.0`.
- `XS_POP_MARK_COLOR`: Enter an integer value for the color of the symbol. The default is `1`.

Examples Pop-mark location

Example	Type
	**Center:** Creates a pop mark in the center of a part
	**Left side:** Creates 2 pop marks on the left side of the part edge
	**Right side:** Creates 2 pop marks on the right side of the part edge
	**Both sides:** Creates 4 pop marks, on all part edges

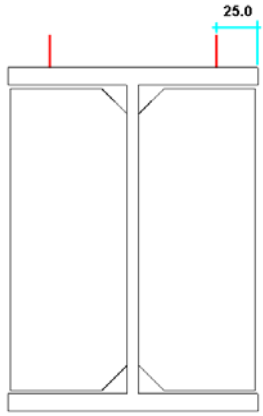
Example	Type
	**Left side holes:** Creates 2 pop marks on the location where the holes are projected on the profile (left side). If no holes are found,Tekla Structures takes the contour to place the pop marks
	**Right side holes:** Creates 2 pop marks on the location where the holes are projected on the profile (right side). If no holes are found, Tekla Structures takes the contour to place the pop marks
	**Both side holes:** Same as 2 previous options, but now for both sides
	**Middle line:** Creates 2 pop marks on both ends and in the middle of the secondary parts

Examples Move to flange



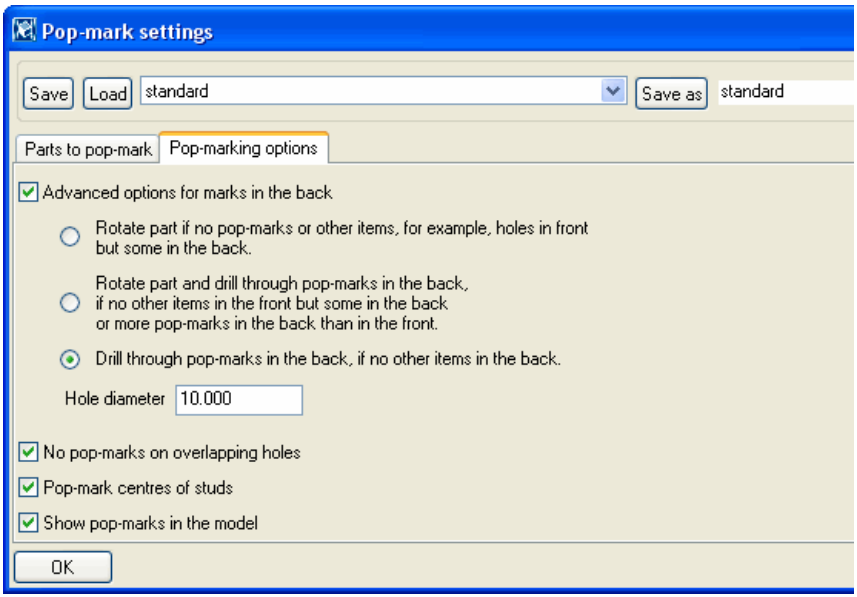
Top flange | Bottom flange | Both flanges

Example Edge distance



The value 25.0 reflects the edge distance compared to the profile edge

Pop-marking options



Advanced options for marks in the back

- **Rotate part if no pop-marks or other items...:**

Rotate the part if there are items on the back of it and none on the front

- **Rotate part and drill through pop-marks in the back...:**

Drill the pop-marks through to the back, if there are no other items there. The hole diameter must also be defined.

- **Drill through pop-marks in the back...:**

Rotate the part and drill pop-marks through to the back if there are no items on the front but some in the back or more pop-marks on the back than on the front. The hole diameter must also be defined.

No pop-marks on overlapping holes

This option deletes a pop-mark if a hole overlaps it.

Pop-mark centres of studs

This option adds pop-marks to the center point of studs. Uncheck this option to prevent studs from being pop-marked.

Show pop-marks in the model

This option displays pop-marks in model views.

Extra information



Tekla Structures can rename the part's part number into the parts assembly number. For example, a column has assembly number A8 and part number P26. It is possible to make the part number equal to the assembly number, so that the part number will be A8 also. This feature is disabled by default!

To modify this variable in a model, you do next:

Step plan

1. Go to **Tools** > **Advanced options > Numbering**
2. On the right pane, select the variable XS_USE_ASSEMBLY_NUMBER_FOR
3. Modify the setting in the field **Value** to MAIN_PART
4. Click <**Apply**> and <**OK**>

The modification is stored in the file `options.ini` in the model folder.

When you want to apply this setting for all models, you copy the file `options.ini` to the folder `ts`.



Pop-marking for NC files also affects numbering!!!

When pop-marking is active (pop-marks), Tekla Structures will assign different numbers to identical parts if they have different pop marks.

3.22 Super- and sub assemblies

Topics

This chapter is divided into following sections:

General (p.254)

Add as Sub-assembly (p.258)

Make into Assembly (p.266)

Set new main sub-assembly (p.270)

Context menu options (p.271)

Assembly number of the super- and sub assembly (p.272)

Add loose parts to an assembly (p.273)

Alternatives (p.274)

General

You can make use super- and sub-assemblies in Tekla Structures.

Super-assembly

A super-assembly is an assembly which consists of multiple assemblies.

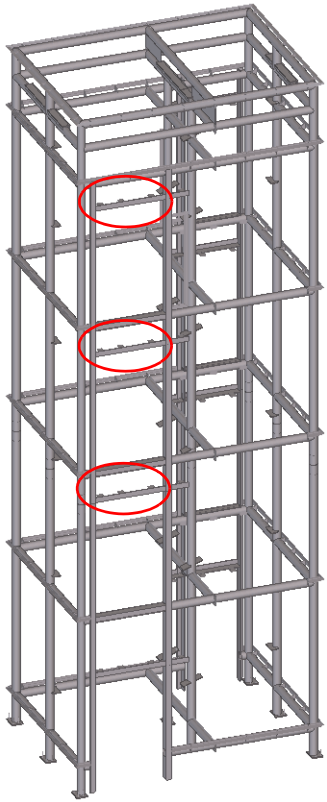
Sub-assembly

A sub-assembly (semi-manufactured product) is an assembly which is part of a larger assembly.

When do you use super-assemblies?

Using super-assemblies is useful when creating large assemblies or when assemblies contain identical sub-assemblies.

One large assembly:



If below assembly exists in multiple assemblies:

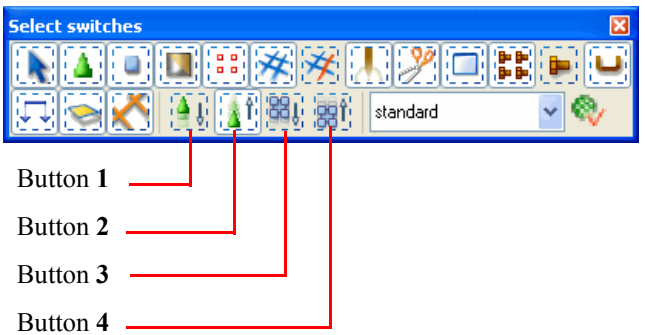


Why using super-/sub assemblies

In the workshop you can create several smaller identical assemblies which are later welded or bolted to a larger assembly or to multiple assemblies (semi-manufactured product). This means that you will have an assembly drawing of the sub-assembly and the weld dimensions of the sub-assembly are not included on the assembly drawing of the super-assembly anymore. The drawing of the super-assembly includes the dimension lines indicating how to assemble the sub-assemblies. This allows easier assembling and reduces errors because of clear drawings.

How to define super-/sub assemblies in Tekla Structures?

In Tekla Structures you must use button **3** when working with super- and sub-assemblies:





When activating button **3**, you can select any level in a super assembly, from the highest level (super-assembly), through the sub-assemblies, to the lowest level (loose parts).

With button **4** you can select the opposite direction, from loose parts up to super-assemblies.

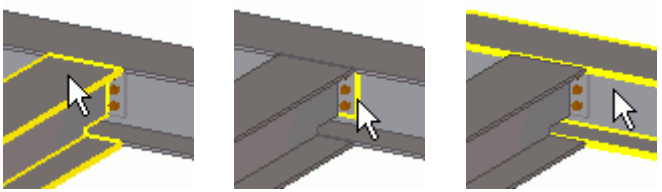
The level of a selected assembly is displayed in the **Status bar**.

In this bar the text ***Use shift + scroll wheel to view different assembly hierarchy levels*** is displayed, more on the right in this bar, a 0 (null) is displayed to indicate the level:



Rollover highlight

If these icons are activated, every assembly will be highlighted if you roll-over the mouse pointer in rendered model views. This is caused by the function **Roll-over highlight**. This way you can see more easily the objects which you can select.



Activate

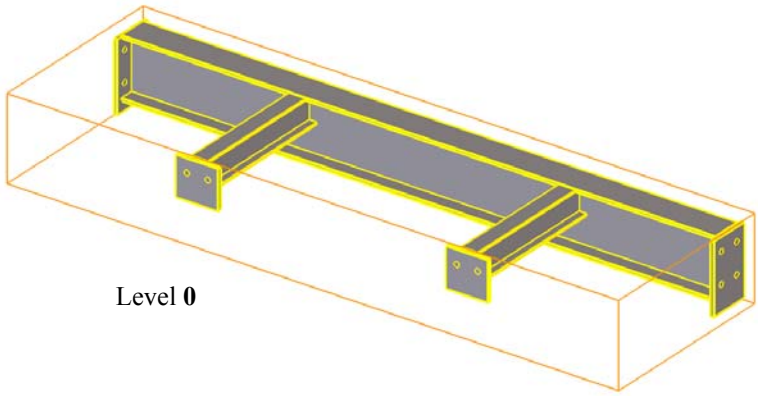
You can switch on or off **Rollover highlight** by pressing shortkey **H**, or by clicking **Setup** > **Rollover highlight.**

By default, this function is enabled. If this function is switched off, you have to select the profile.

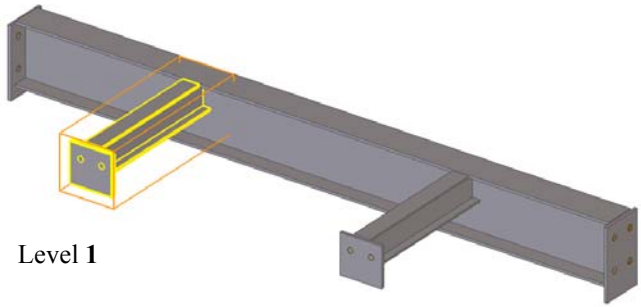
To switch to a specific level in the assembly, hold down the <**Shift**> key and scroll the mouse wheel. If button **3** is activated, the 0 (null) will change to 1 or another number.

Hierarchy in a super-assembly

So, the large assembly (super-assembly) is 0 (null), the smaller assembly (sub-assembly) will get 1 etcetera. You can continue to 9, this means you can create 10 assembly levels. See also following pictures



Level **0**



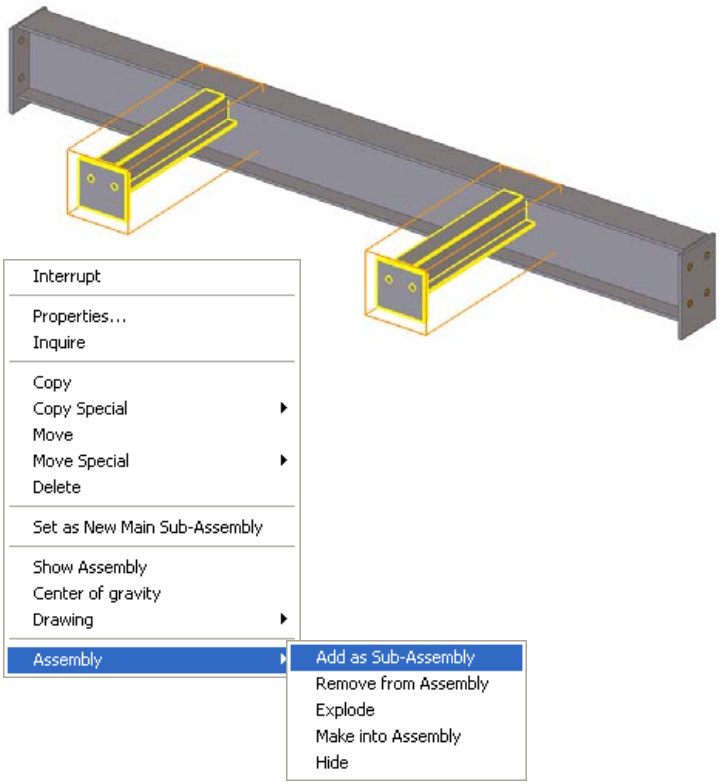
Level **1**

Tekla Structures includes the commands **Add as Sub-assembly** and **Make into Assembly** to create sub-/super-assemblies, these options will be treated in detail in the following chapters.

Add as Sub-assembly

Step plan

1. Create the profiles as shown in the previous example
2. Click on button **3**
3. Press the <**Shift**> button and select the two short beams which will be added as a sub-assembly
4. Click the right mouse button
5. In the pull menu go to **Assembly** > **Add as sub-assembly**

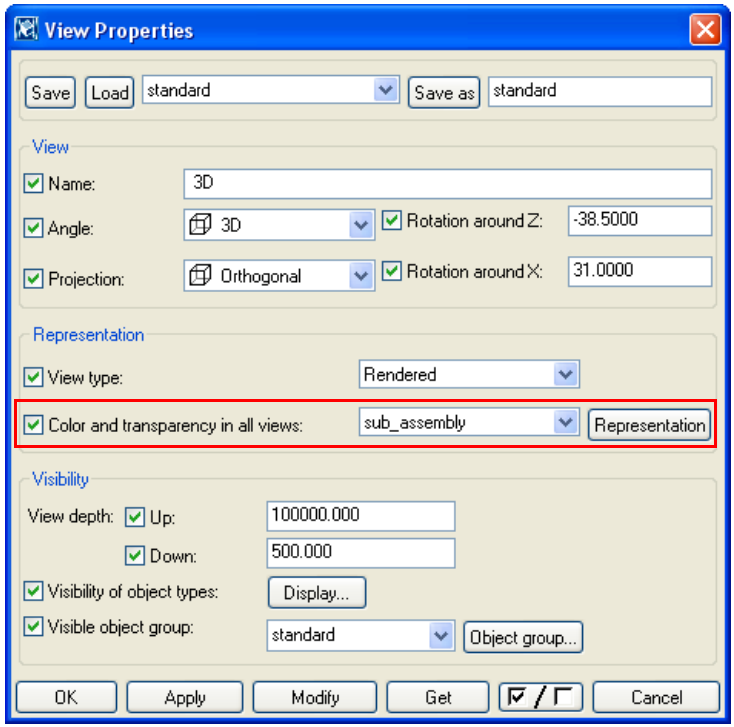


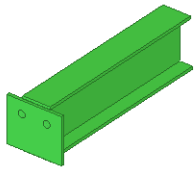
6. Pick the main part

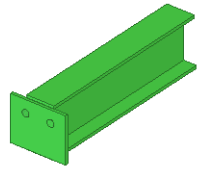
   The two short beams are now added as a sub-assembly.

View- and select filter

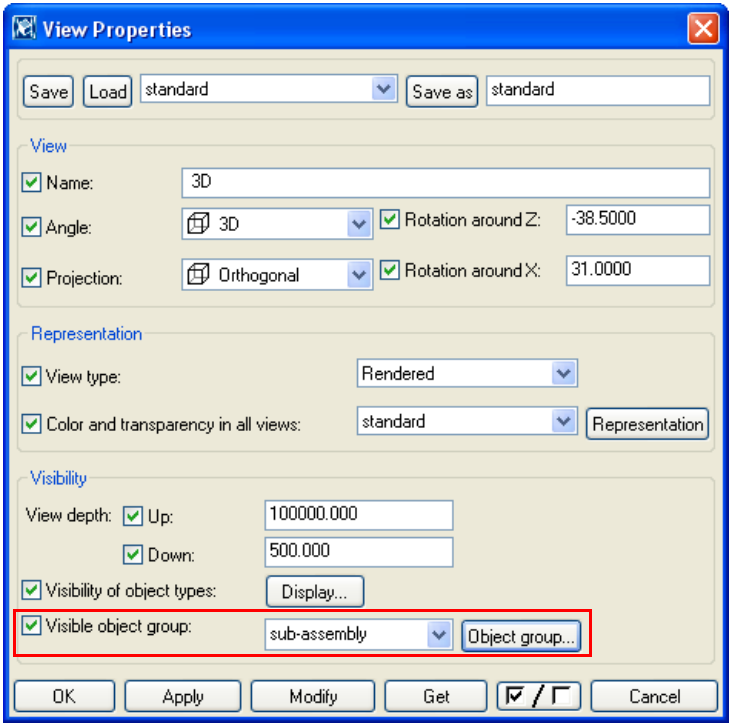
You can set how to display sub- and super-assemblies in the modelviews and which objects can be selected. Double-click in a modelview and select the desired option in the listbox, e.g. to display sub-assemblies:







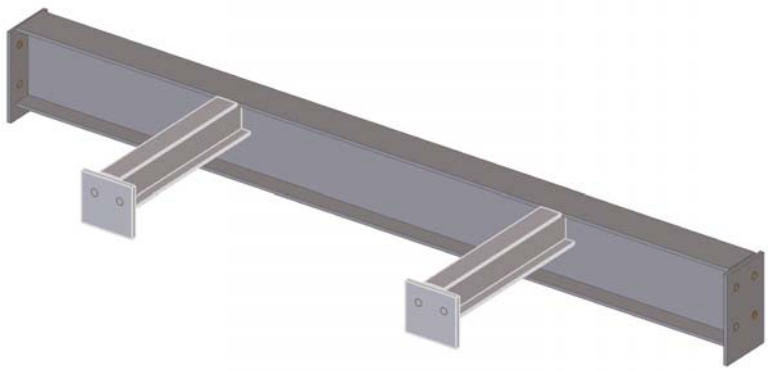
To select the sub-assemblies:


View Properties
Save
Load
standard
Save as
standard
View
Name:
3D
Angle:
3D
Rotation around Z:
-38.5000
Projection:
Orthogonal
Rotation around X:
31.0000
Representation
View type:
Rendered
Color and transparency in all views:
standard
Representation
Visibility
View depth:
Up:
100000.000
Down:
500.000
Visibility of object types:
Display...
Visible object group:
sub-assembly
Object group...
OK
Apply
Modify
Get
Cancel

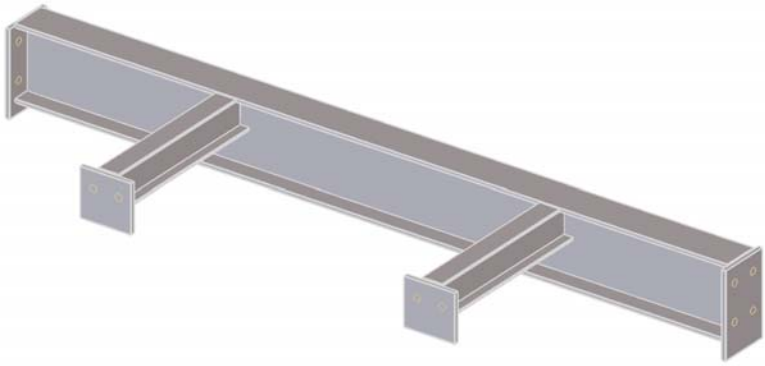
Generating output

To generate the correct output, it must be clear of which objects the "super-assembly" consists:

Sub-assemblies The two short beams:



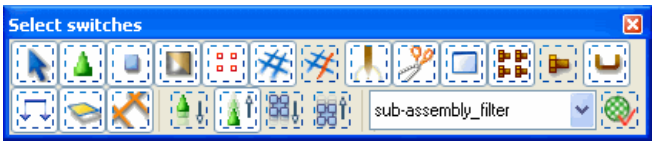
Super-assembly The entire "super-assembly":



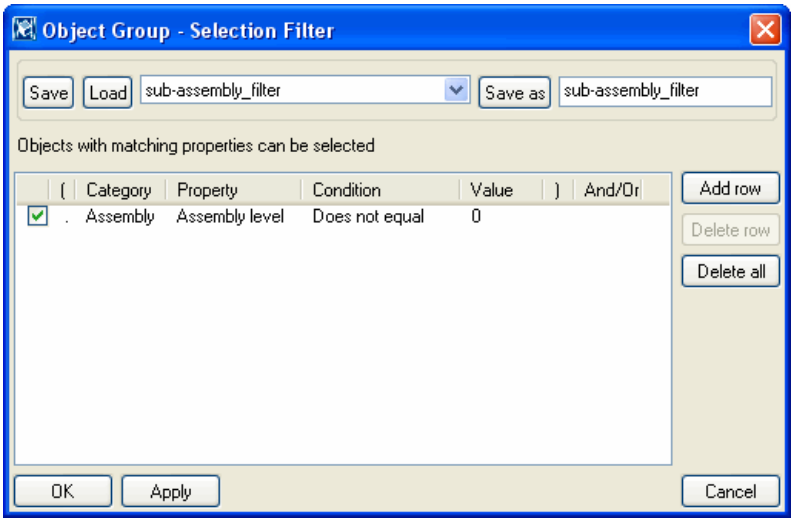
We will now start generating output for both sub- and super-assemblies

Sub-assemblies

To select sub-assemblies, to create workshop drawings for example, you make use of the **sub-assembly_filter**:



You will see in the **Object group - select filter** dialogbox that the **sub-assembly_filter** only selects assemblies which assembly level is not equal to "0":





While generating assembly drawings from the selected **sub-assemblies**, you **always** make use of button **2** to select sub-assemblies!



Button **1** selects at the highest level, because of that, stiffeners or end plates, created by a component, will highlight but will not be selected!!!

We will now create assembly drawings from the sub-assemblies:

Step plan

1. Number the model
2. In the pull-down menu go to **Properties** > **Assembly drawing**
3. Load the setting *sub-assembly*
4. Select the **sub-assembly_filter**
5. Activate button **2** and select the subassemblies in the modelview
6. Create the assembly drawings from the sub-assemblies

7. Open the drawing *SUB ASSEMBLY*

   This is a "normal" assembly drawing, only the name is not BEAM or COLUMN, but SUB ASSEMBLY. As you can see in the assembly part list, the sub-assembly occurs 2 times in the model.

ASSEMBLY PART LIST

Assembly: L17 Length (mm): 365

Finish: Number: 2

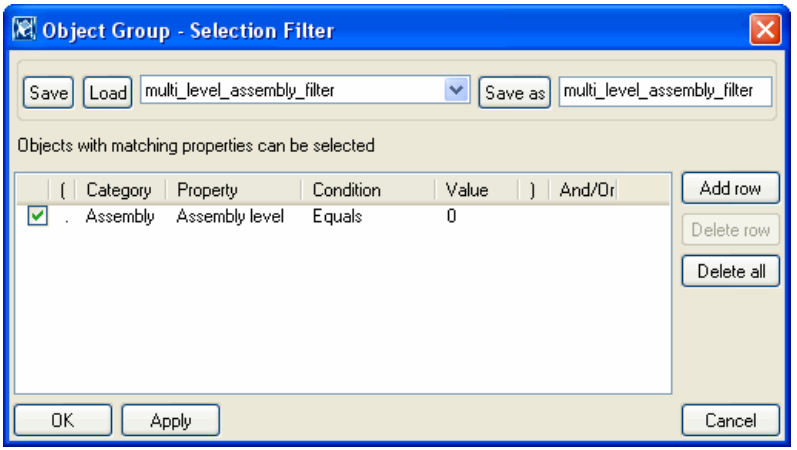
Part	Profile	Material	Number	Length (mm)	Weight (kg)
L17	IPE100	S235JR	1	359	2.9
P30	FLAT6*90	S235JR	1	110	0.5
				Total per assembly	3.4

Super-assembly

To select the super-assembly, to create workshop drawings for example, you make use of the **multi_level_assembly_filter**:



You will see in the **Object group - select filter** dialogbox that the **multi_level_assembly_filter** selects assemblies which assembly level is equal to "0":





While generating assembly drawings from the selected **super-assemblies**, you **always** make use of button **3** to select super-assemblies!

We will now create the assembly drawing from the super-assembly.

Step plan

1. In the pull-down menu go to **Properties** > **Assembly drawing**
2. Load the setting *super-assembly*
3. Select the **multi_level_assembly_filter**
4. Activate button **3** and select the super-assembly in the modelview
5. Create the assembly drawings from the super-assembly
6. Open the drawing *SUPER ASSEMBLY*

On this drawing, the template **assembly_part_list_super_assembly** is displayed. The information of the "main assembly" is listed as usual. For the sub-assemblies, only a reference is listed. This because these sub-assemblies are already assembled:

MATERIAL LIST FOR SUPER ASSEMBLY: L16				Number:1	
Assembly	Profile	Grade	Number	Length (mm)	Weight (kg)
L16	IPE200	S235JR	1	1969	44.1
P3	FLAT10*100	S235JR	2	190	3.0
L17	Subassembly		2		6.7
				Total	53.8

By using the command **Add as sub-assembly** we have created 2 assembly drawings:

- **1** assembly drawing for the 2 short beams (they have identical assembly numbers)
- **1** assembly drawing for the super-assembly



No drawing has been created from the longer beam!

Superassembly partlist

```
------------------------------------------------------------------------------
TEKLA STRUCTURES SUPER ASSEMBLY PART LIST            Page:     1
Model:                                               Project:  project nummer
                                                     Date:     02.05.2008
                                                     Time:     13:05:52

------------------------------------------------------------------------------
Assembly     Number  Name          Profile                 Area (m2) Weight (kg)
==============================================================================
L16            1     LIGGER        IPE200                  1.9       54.0
               ---------------------------------------------------------------
               Part        Number   Profile     Length (mm)
               ---------------------------------------------------------------
               L16            1     IPE200          1969    1.51      44.1
               P3             2     FLAT10*100       190    0.04       1.5
    ==========================================================================
     2x Sub-assembly L17
               ---------------------------------------------------------------
               Part        Number   Profile     Length (mm)
               ---------------------------------------------------------------
               L17            1     IPE100           359    0.14       2.9
               P30            1     FLAT6*90         110    0.02       0.5
==============================================================================
               Total for      1     assembl(y)(ies):        1.93      54.0
==============================================================================
```

Superassembly list

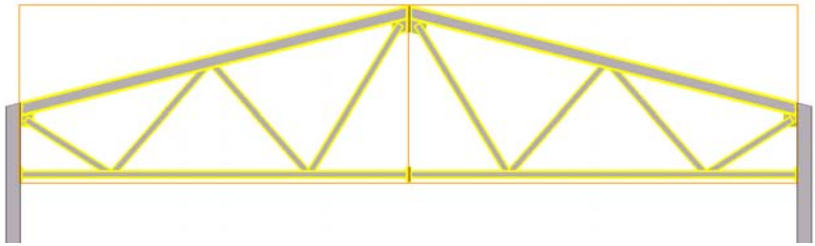
```
TEKLA STRUCTURES SUPER ASSEMBLY LIST                 Page:     1
Model: Opfris_2007_UK                                Project:  project nummer
                                                     Date:     02.05.2008
                                                     Time:     13:08:39

------------------------------------------------------------------------------
   Main assembly
        Sub assembly     Number        Level assembly  Weight (kg)
------------------------------------------------------------------------------
==============================================================================
    L16                    1              0               54.0
    ---------------------------------------------------------------------------
         L17               2              1
```

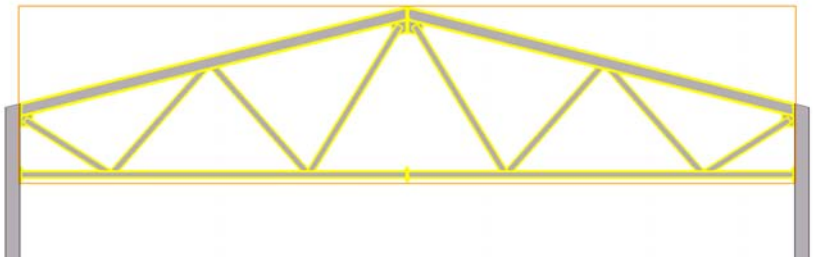
Make into Assembly

You make use of this option when you want to create a super-assembly which exists of several (sub-)assemblies, e.g. a truss:

Sub-assemblies

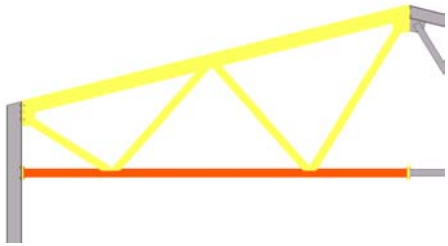


Super-assembly



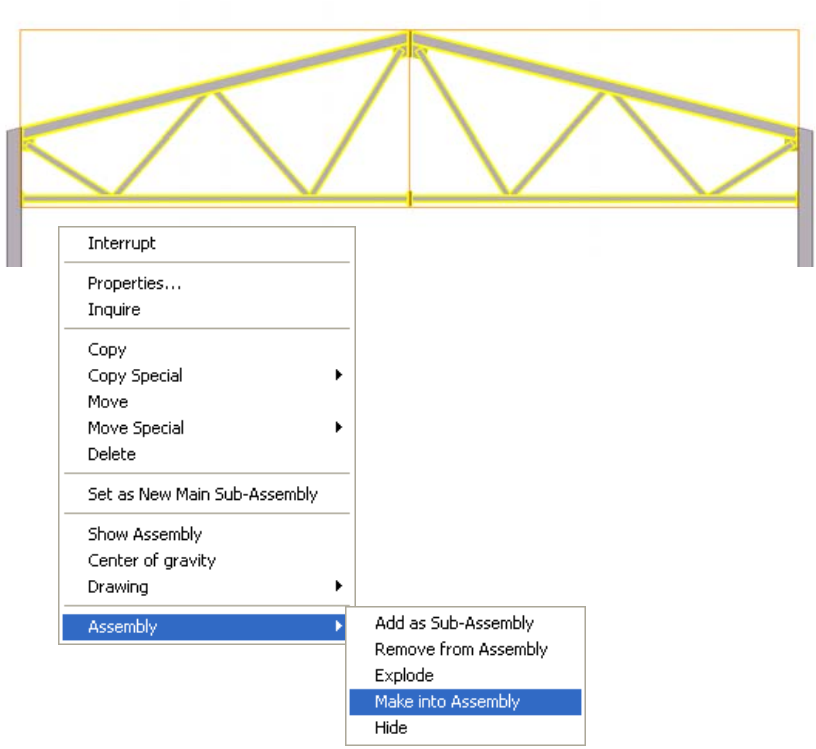
Step plan

1. Click on button **2**
2. Create the two sub-assemblies by welding the parts (belonging to a sub-assembly) manually to the main part, e.g. the bottom girder



3. Now click on button **3**
4. Select the two assemblies by keeping pressed down the <**Shift**> button

5. Click the right mouse button and select **Make into Assembly**



The two sub-assemblies (the left- and the right side of the truss) together form the super-assembly (the entire truss).

We will now generate output for both sub-assemblies and the super-assembly.

Generating output

Sub-assemblies

We will now generate the assembly drawings from the sub-assemblies: so the left- and the right side of the truss

Step plan

1. Number the model
2. Select the **sub-assembly_filter**



3. Activate button **2** and select the sub-assemblies in the modelview
4. In the pull-down menu go to **Properties** > **Assembly drawing**
5. Load the setting *sub-assembly*
6. Generate the drawings from the sub-assemblies:
   - **1** assembly drawing for the left side of the truss
   - **1** assembly drawing for the right side of the truss
7. Open one of the assembly drawings *SUB ASSEMBLY*

This drawing includes template **assembly_partlist_finish**:

ASSEMBLY PART LIST

Assembly: SUB_L3 Length (mm): 5884

Finish: Number: 1

Part	Profile	Material	Number	Length (mm)	Weight (kg)
P4	K100/6	S235JRG2	1	1517	24.8
P5	K100/6	S235JRG2	1	2142	34.5
P6	K100/6	S235JRG2	1	2145	35.6
P9	FLAT10*200	S235JRG2	1	230	3.6
P10	FLAT10*200	S235JRG2	1	291	4.6
P17	K100/6	S235JRG2	1	2621	44.2
P20	PL10*182.4	S235JRG2	1	246	3.0
P25	HEA200	S235JRG2	1	6091	255.7
P26	FLAT10*150	S235JRG2	1	391	4.6
PL1	PL10*175.3	S235JRG2	1	194	2.2
SUB_L3	K100/6	S235JRG2	1	5874	99.7
				Total per assembly	512.4

Super-assemblies

We will now generate the assembly drawing from the super-assembly:

Step plan

1. Select the **multi_level_assembly_filter**



2. Activate button **3** and select the super-assembly
3. In the pull-down menu go to **Properties** > **Assembly drawing**
4. Load the setting *super-assembly*
5. Generate the drawing from the super-assembly
6. Open the drawing *SUPER ASSEMBLY*

On this drawing, the template **assembly_part_list_super_assembly** is displayed. For the sub-assemblies, only a reference is listed. This because these sub-assemblies are already assembled:

MATERIAL LIST FOR SUPER ASSEMBLY: SUB_L10					Number:1
Assembly	Profile	Grade	Number	Length (mm)	Weight (kg)
SUB_L3	Subassembly		1		518.0
SUB_L8	Subassembly		1		518.0
				Total	1036.1

Superassemblypartlist

```
-------------------------------------------------------------------------------------
TEKLA STRUCTURES SUPER ASSEMBLY PART LIST                    Page:     1
Model:                                                       Project:  PROJECT NUMMER
                                                             Date:     02.05.2008
                                                             Time:     13:54:49
-------------------------------------------------------------------------------------
Assembly     Number  Name          Profile                        Area (m2) Weight (kg)
=====================================================================================
SUB_L10        1     LIGGER        K100/6                          25.8     1054.0
               -------------------------------------------------------------------
               Part        Number   Profile           Length (mm)
               -------------------------------------------------------------------
    ===========================================================================
     1x Sub-assembly SUB_L3
               -------------------------------------------------------------------
               Part        Number   Profile           Length (mm)
               -------------------------------------------------------------------
               P4              1    K100/6              1517          0.58      24.8
               P5              1    K100/6              2142          0.81      34.5
               P6              1    K100/6              2145          0.82      35.6
               P9              1    FLAT10*200           230          0.10       3.6
               P10             1    FLAT10*200           291          0.13       4.6
               P17             1    K100/6              2621          1.00      44.2
               P20             1    PL10*182.4           246          0.08       3.0
               P25             1    HEA200              6091          6.94     255.7
               P26             1    FLAT10*150           391          0.13       4.6
               PL1             1    PL10*175.3           194          0.06       2.2
               SUB_L3          1    K100/6              5874          2.23      99.7
    ===========================================================================
     1x Sub-assembly SUB_L8
               -------------------------------------------------------------------
               Part        Number   Profile           Length (mm)
               -------------------------------------------------------------------
               P4              1    K100/6              1517          0.58      24.8
               P5              1    K100/6              2142          0.81      34.5
               P6              1    K100/6              2145          0.82      35.6
               P9              1    FLAT10*200           230          0.10       3.6
               P10             1    FLAT10*200           291          0.13       4.6
               P19             1    K100/6              2621          1.00      44.2
               P20             1    PL10*182.4           246          0.08       3.0
               P25             1    HEA200              6091          6.94     255.7
               P26             1    FLAT10*150           391          0.13       4.6
               PL1             1    PL10*175.3           194          0.06       2.2
               SUB_L8          1    K100/6              5874          2.23      99.7
=====================================================================================
               Total for       1    assembl(y)(ies):                25.76     1054.0
=====================================================================================
```

Superassembly list

```
TEKLA STRUCTURES SUPER ASSEMBLY LIST                         Page:     1
Model:                                                       Project:  PROJECT NUMMER
                                                             Date:     02.05.2008
                                                             Time:     14:06:44
-------------------------------------------------------------------------------------
   Main assembly
        Sub assembly     Number              Level assembly  Weight (kg)
-------------------------------------------------------------------------------------
=====================================================================================
    SUB_L10                 1                    0              1054.0
    ---------------------------------------------------------------------------------
        SUB_L3              1                    1
        SUB_L8              1                    1
```

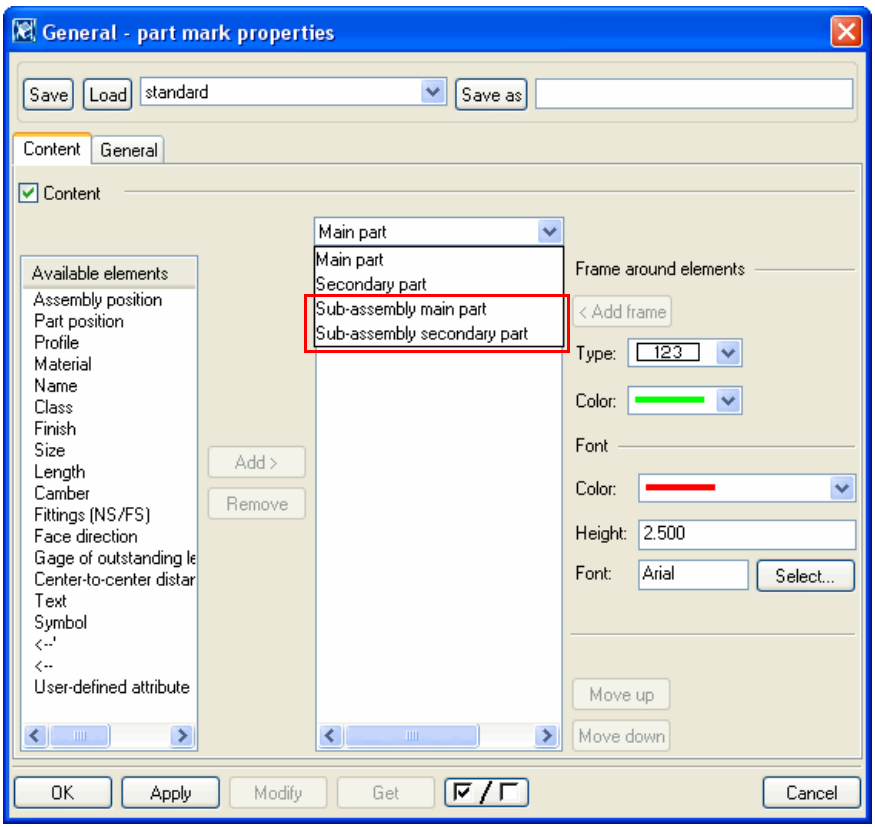
By default, Construsoft has created reports and drawing templates for 2 levels: 0 and 1. If you use more levels, you have to modify the templates and reports.

By using the command **Make into Assembly** we created **3** assembly drawings in total:

- **1** assembly drawing for the left side of the truss
- **1** assembly drawing for the right side of the truss
- **1** assembly drawing for the super-assembly

Drawings

If you look in the part marks, you can define marks for sub-assemblies. This also counts for General drawings:



Set new main sub-assembly

In some cases, when you have a complex truss or a lattice work, it may occur that you want to set another part as the assemblies' main part.

The assemblies' main part can be redefined very easily:

- Select the correct assemblies' main part

- Click the right mouse button and select **Set as new main sub-assembly** in the contextmenu
- The selected profile is the main part of the assembly.



You cannot redefine the assemblies' main part when you use the module SDE (Standard Detailing) or RCD (Reinforced Concrete Detailing).

Context menu options



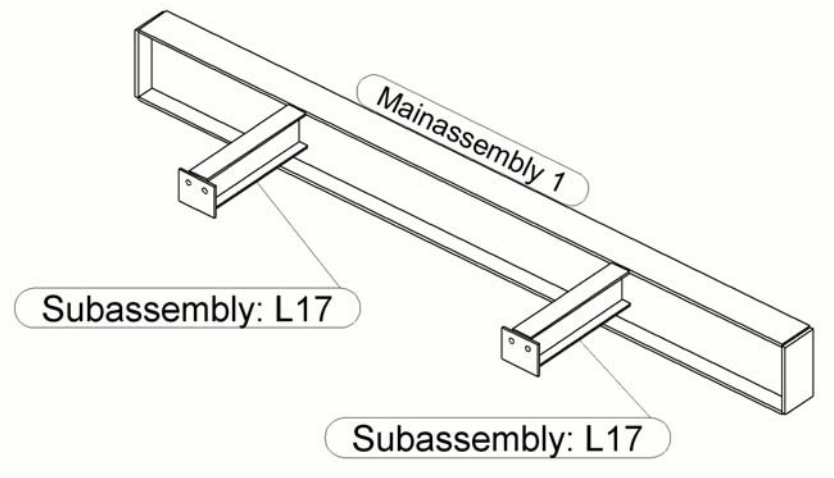
To be able to select any level of a super assembly, you make use of the buttons **3** and **4** to add, remove, explode or crate. Only by using these two icons, an orange colored box will appear around the parts and you will be able to select an option in the following contextmenu:

- Remove from Assembly
- Explode
- Make into Assembly

Add as Sub-assembly	See above.
Remove from Assembly	Use this option to remove a part from an assembly. This option is the opposite from option *Add as Sub-assembly.*
Explode	To let the super-assembly fall out into separate assemblies.
Make into Assembly	See above.
Hide	To hide an object(s) temporary in a modelview

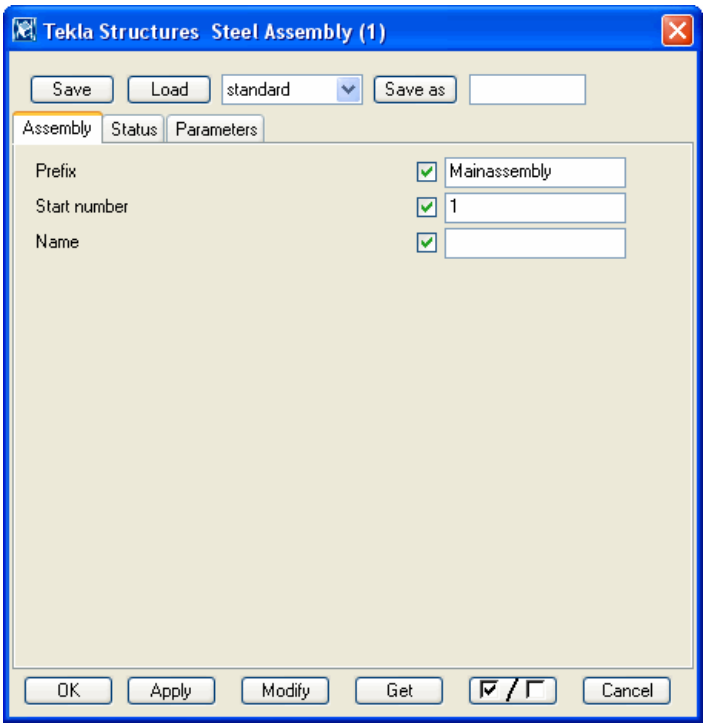
Assembly number of the super- and sub assembly

It is useful to adjust the prefix' of the super- and sub-assemblies that the mark content in drawings can be like this:





To do this, click button **3** and double-click the super-assembly, the next dialog box appears:



Add loose parts to an assembly

You have the possibility to add loose parts to an assembly.

Because of this, the part will **not** be welded to the assembly but added!

Step plan



1. Select the part you want to add
2. Make sure that button **1** or button **2** is selected
3. Right mouse click and select **Add to assembly**
4. Now select the assembly to add the loose part to



Do not use the commands for creating sub- and super assemblies to weld for example a stub to a beam.

In case you want to weld the loose part instead of adding it, use the command **Create weld**.

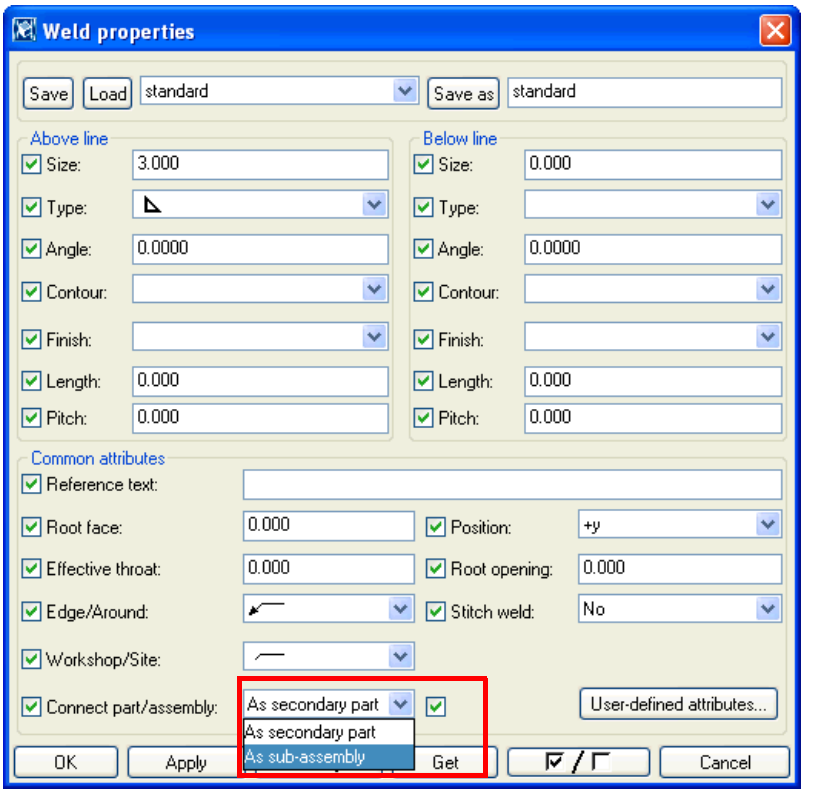
The part will now be welded to the assembly and not be added.



You **cannot** create pop-marks from parts to which other parts are added!

Alternatives

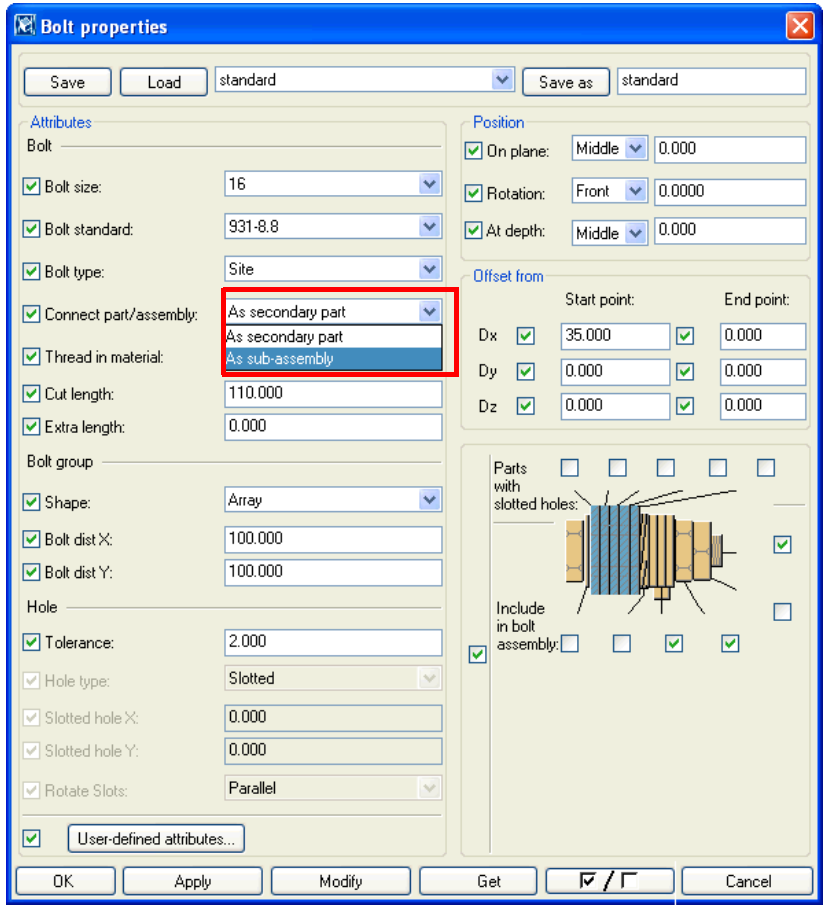
You can also create sub-assemblies using welds of bolts. To do this, select setting *As sub-assembly* in the dialog box *Weld properties* or *Bolt properties*, see picture below:





If you want to use pop marks in NC files, use welds.

If you want to define sub-assemblies using bolts, the method is identical:



However: the previous described method for creating super-assemblies is preferable above the welding/ bolting method!

3.23 Numbering UDA

Topics

This chapter is divided into following sections:

Modeling (p.276)

Changing the prefix (p.281)

Changing the start numbers (p.286)

Phase (p.289)

Welds (p.292)

Configuration (p.292)



Component ***Numbering UDA (73)*** can modify numbering settings. This means:

- Changing start numbers of profiles
- Changing prefixes of profiles
- Assigning unique numbers to identical profiles



When you enter a radius in the XZ plane in the profile properties of curved beams, the beam will become a straight profile.

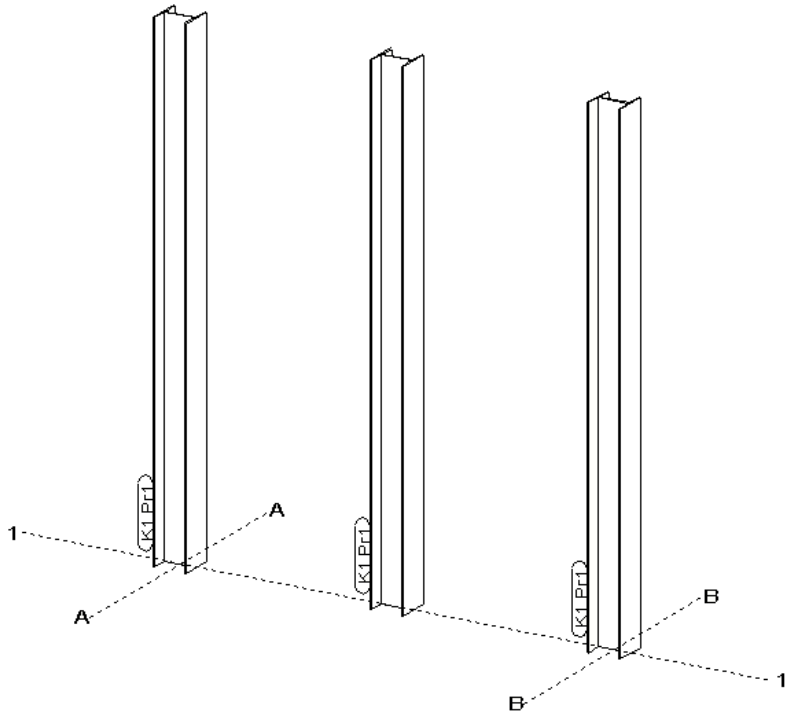
Everything you enter in the field **Finish** in the profile properties, will be deleted when you use this component on that specific profile.

Modeling

The model is created as usual. To assign unique numbers to parts, do the following:

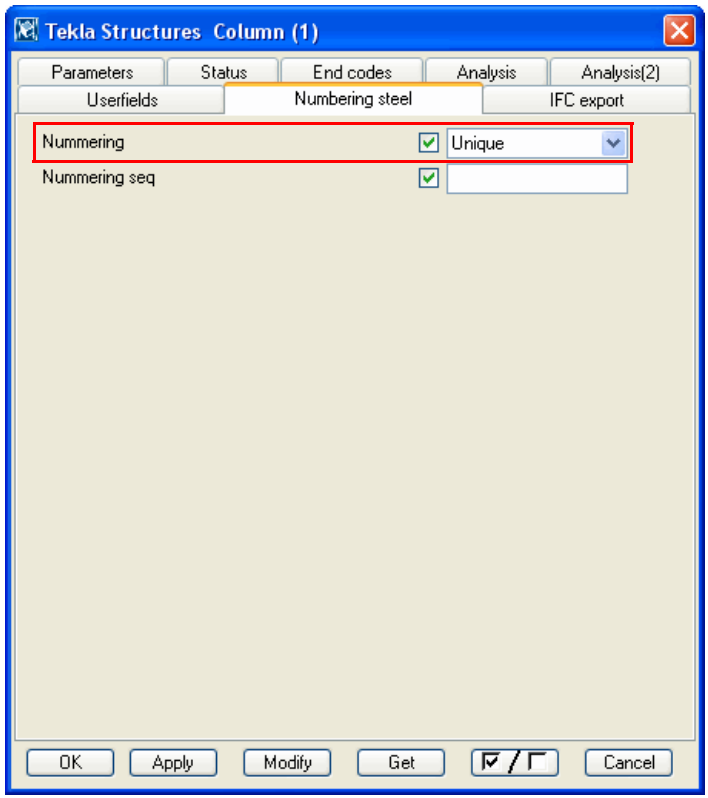
Example

The model includes 3 columns. Now full numbering as usual. All columns have the same part and assembly number:


K1 Pr1
K1 Pr1
K1 Pr1
1
A
A
B
B
1

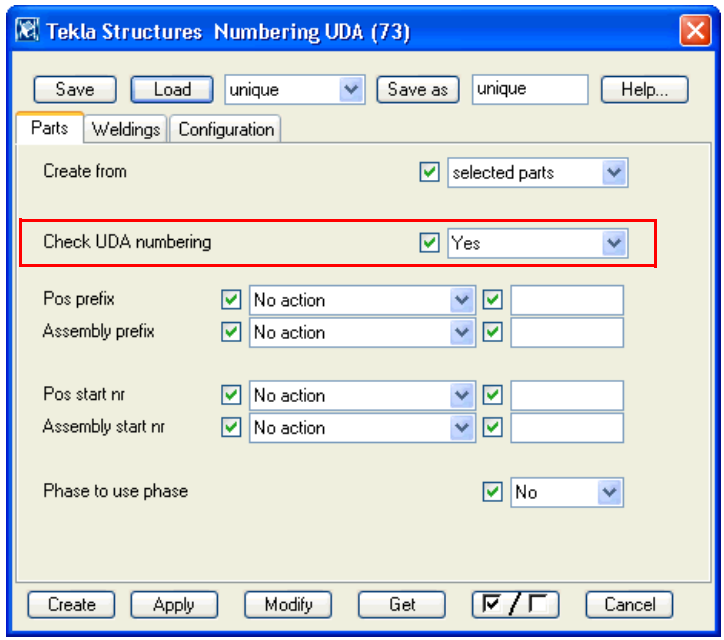
Step plan

1. The first step is to change the numbering for all columns in the user defined attributes to **Unique**.
2. In the user defined attributes of the parts, go to the tab **Numbering steel** and select **Unique** for **Numbering**:



3. Now select all parts you want to assign a unique number to. Select the setting **unique** in the component and click **Create**.

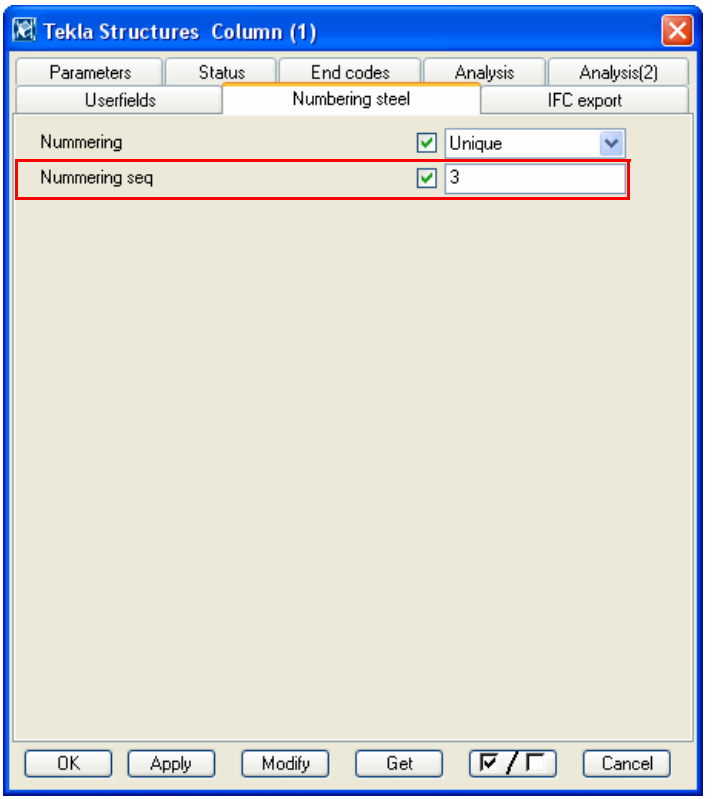
4. Check that **UDA numbering** is set to **Yes**:



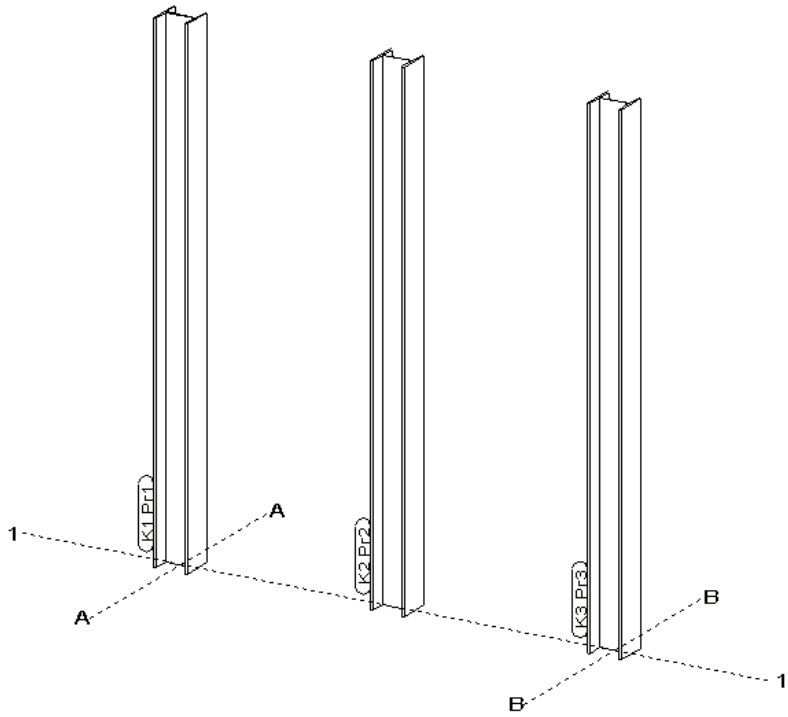


This component is only effective when clicking **Create**. Clicking **Apply** makes no sense!!!

If you now check the columns' user defined attributes you will see that a unique number is added:



5. Next full numbering as usual.



All columns became unique part- and assembly numbers.

Changing the prefix

Make use of this component to change parts' prefixes and start numbers. When you, for instance, only want to change the start number, Tekla Structures also checks the prefixes because you cannot set a checkbox per prefix/start number.

For both part and assembly numbers it is possible by means of this component, the options are:

No action
Clean up and add phase
Add phase
Add the text
Make equal to text
Clean up

No action

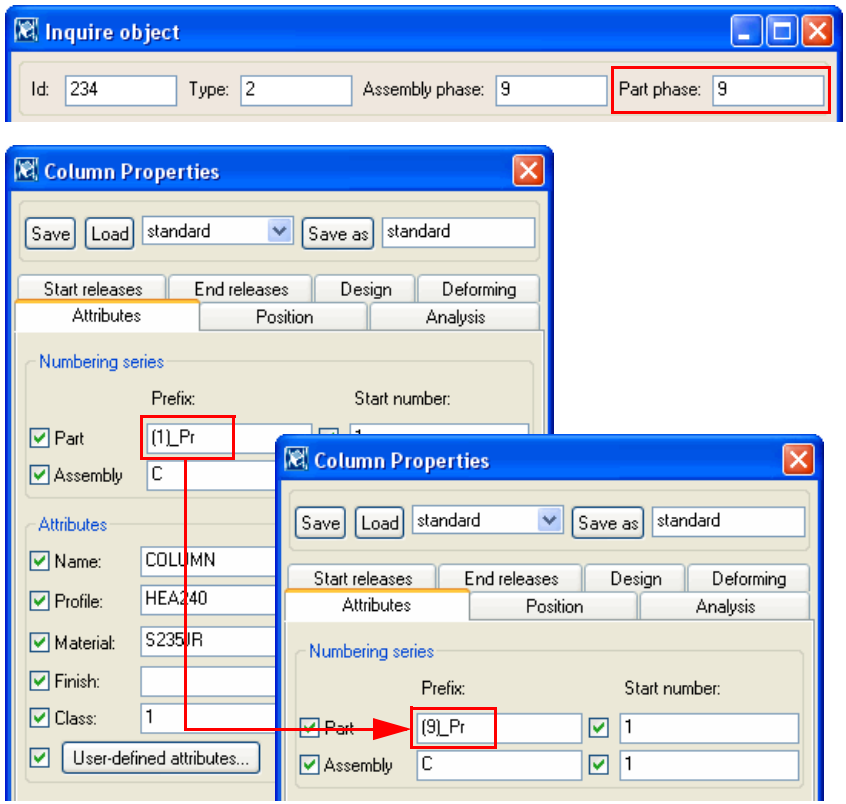
When you select this option, this line will not be executed.

Clean up and add phase

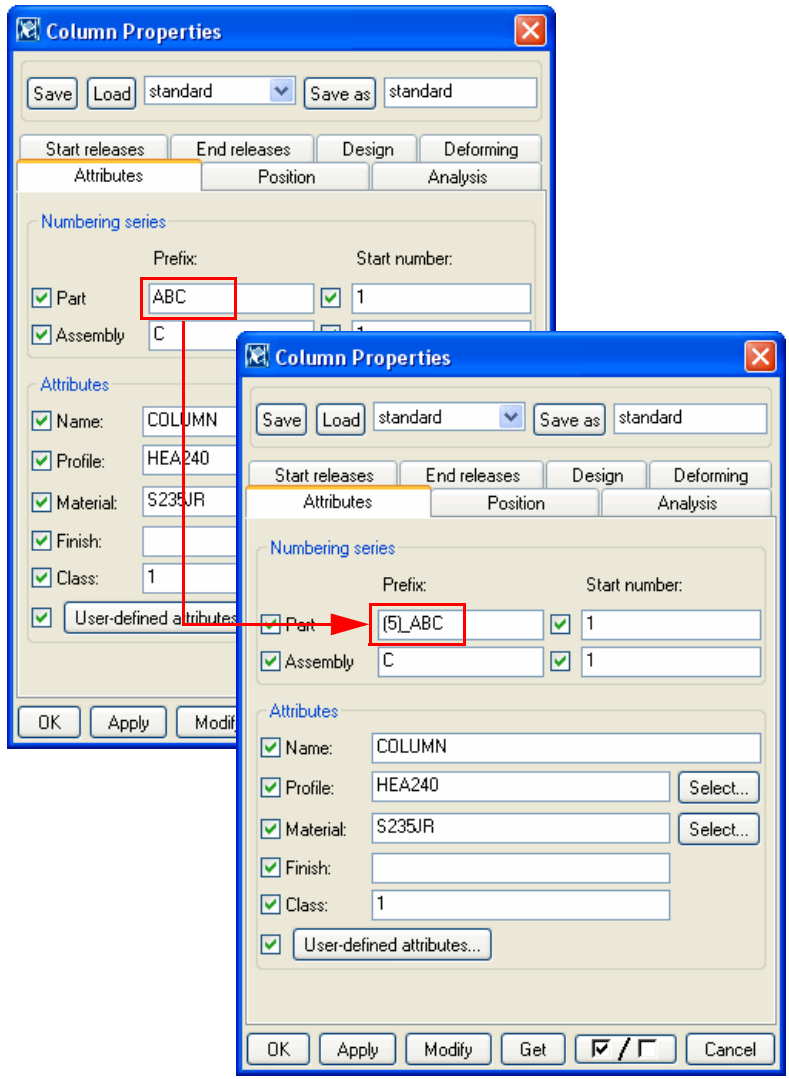
An already filled in phase number is getting deleted and the new phase number is getting added. This is required when parts' phase numbers change.



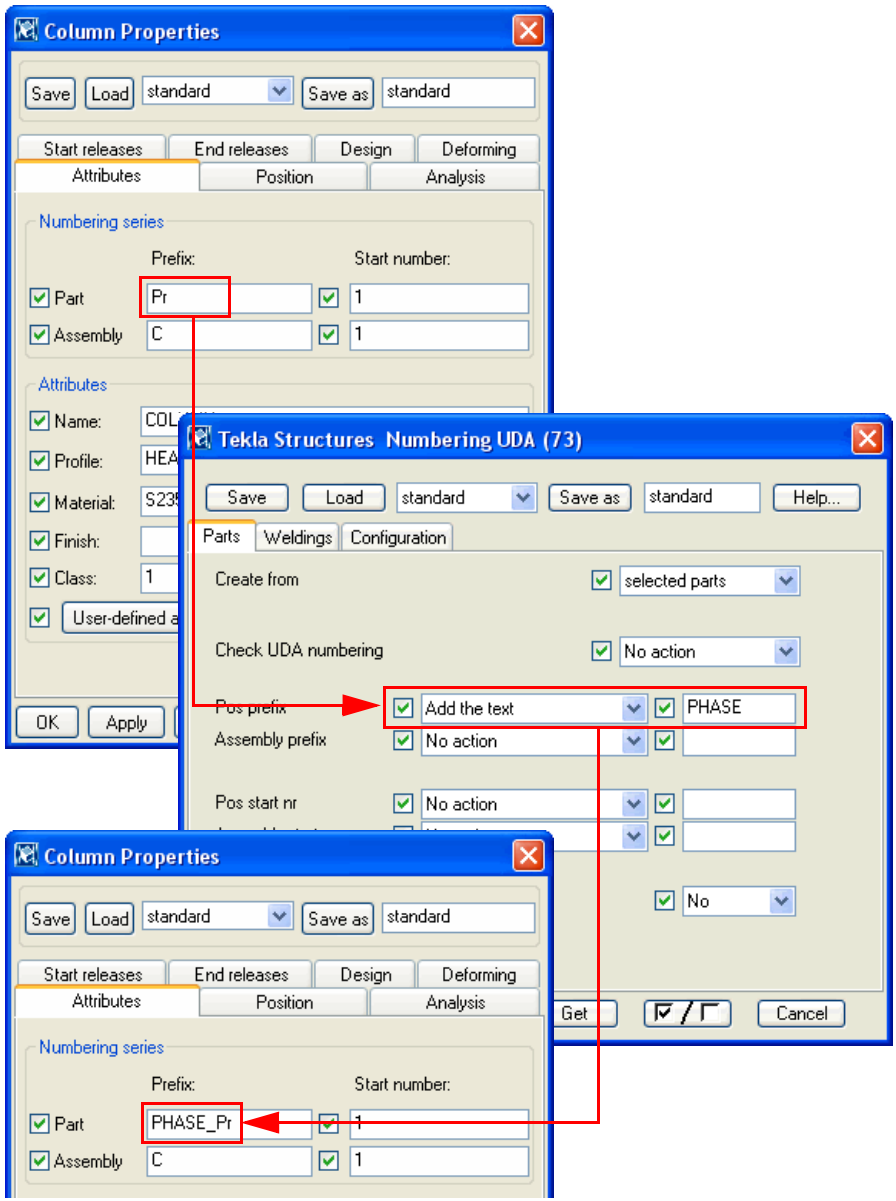
The phase number which is added to the prefix is always the number of the phase in which the part is located in!



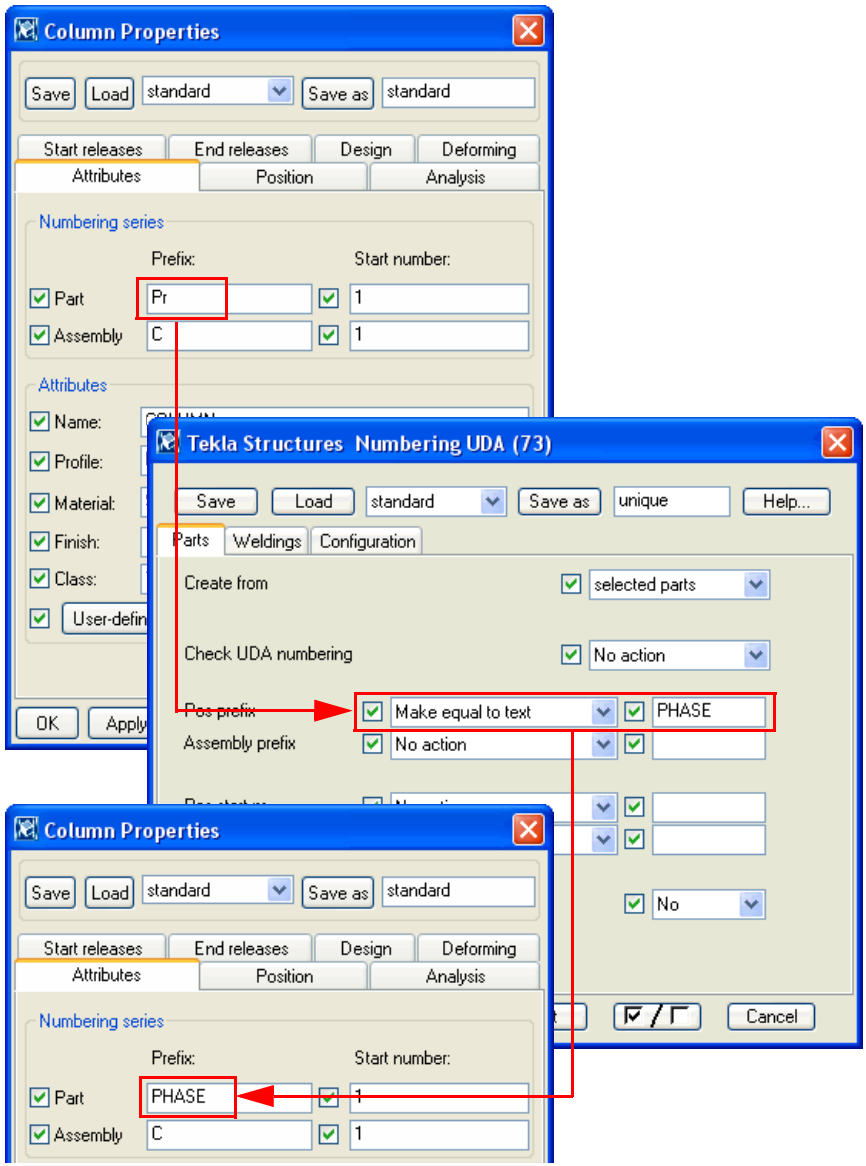
Add phase


Column Properties
Save
Load
standard
Save as
standard
Start releases
End releases
Design
Deforming
Attributes
Position
Analysis
Numbering series
Prefix:
Start number:
Part
ABC
1
Assembly
C
Attributes
Name:
COLUMN
Profile:
HEA240
Material:
S235JR
Finish:
Class:
1
User-defined attributes...
OK
Apply
Modify
Get
Cancel
(5)_ABC
Select...

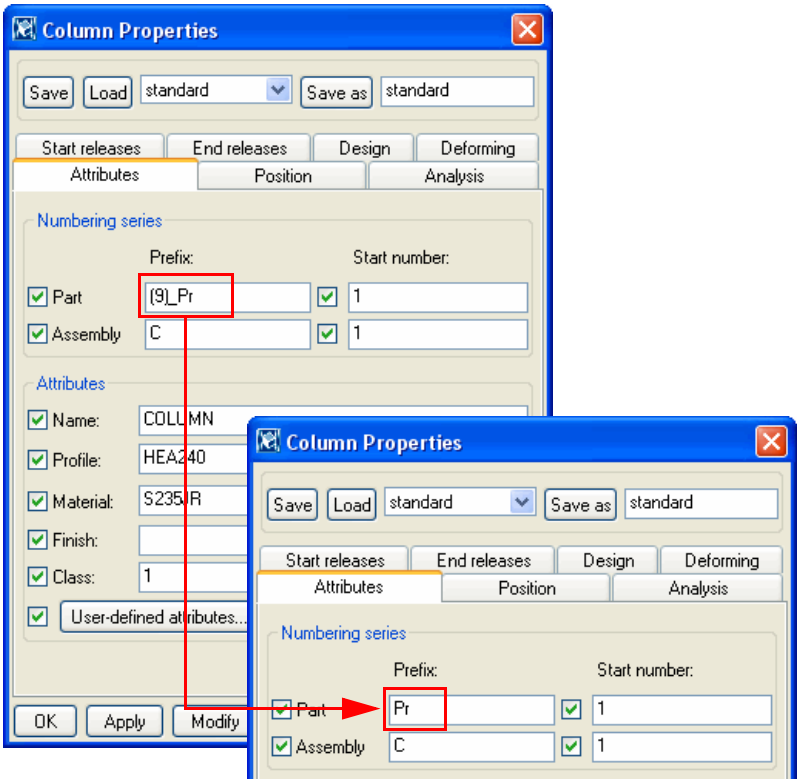
Add the text


Column Properties
Save
Load
standard
Save as
standard
Start releases
End releases
Design
Deforming
Attributes
Position
Analysis
Numbering series
Prefix:
Start number:
Part
Pr
1
Assembly
C
1
Attributes
Name:
Profile:
Material:
Finish:
Class:
User-defined a
OK
Apply
Tekla Structures Numbering UDA (73)
Save
Load
standard
Save as
standard
Help...
Parts
Weldings
Configuration
Create from
selected parts
Check UDA numbering
No action
Pos prefix
Add the text
PHASE
Assembly prefix
No action
Pos start nr
No action
No
Get
Cancel
Column Properties
Save
Load
standard
Save as
standard
Start releases
End releases
Design
Deforming
Attributes
Position
Analysis
Numbering series
Prefix:
Start number:
Part
PHASE_Pr
1
Assembly
C
1

Make equal to text



Clean up



Changing the start numbers

You can change start numbers for both part- and assembly numbers.

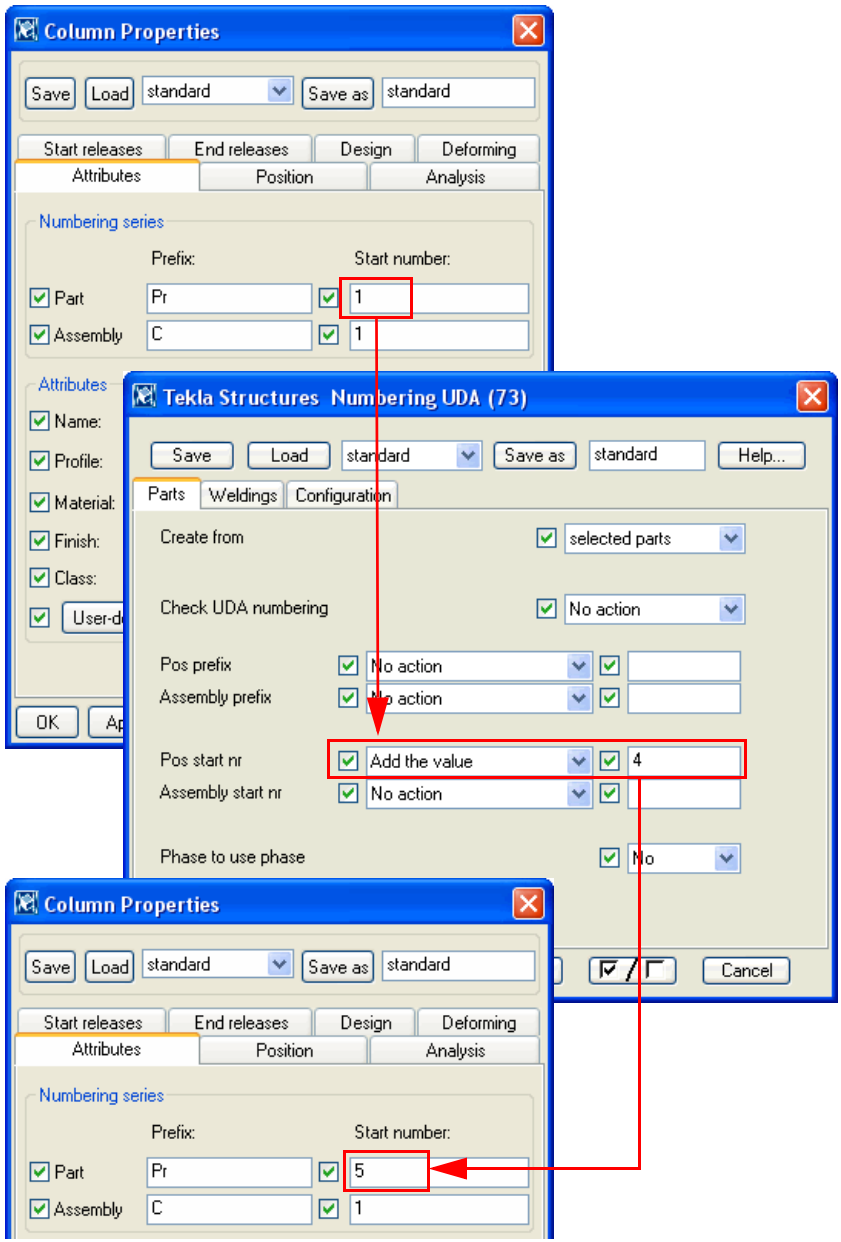
No action
Add the value
Make equal to

No action

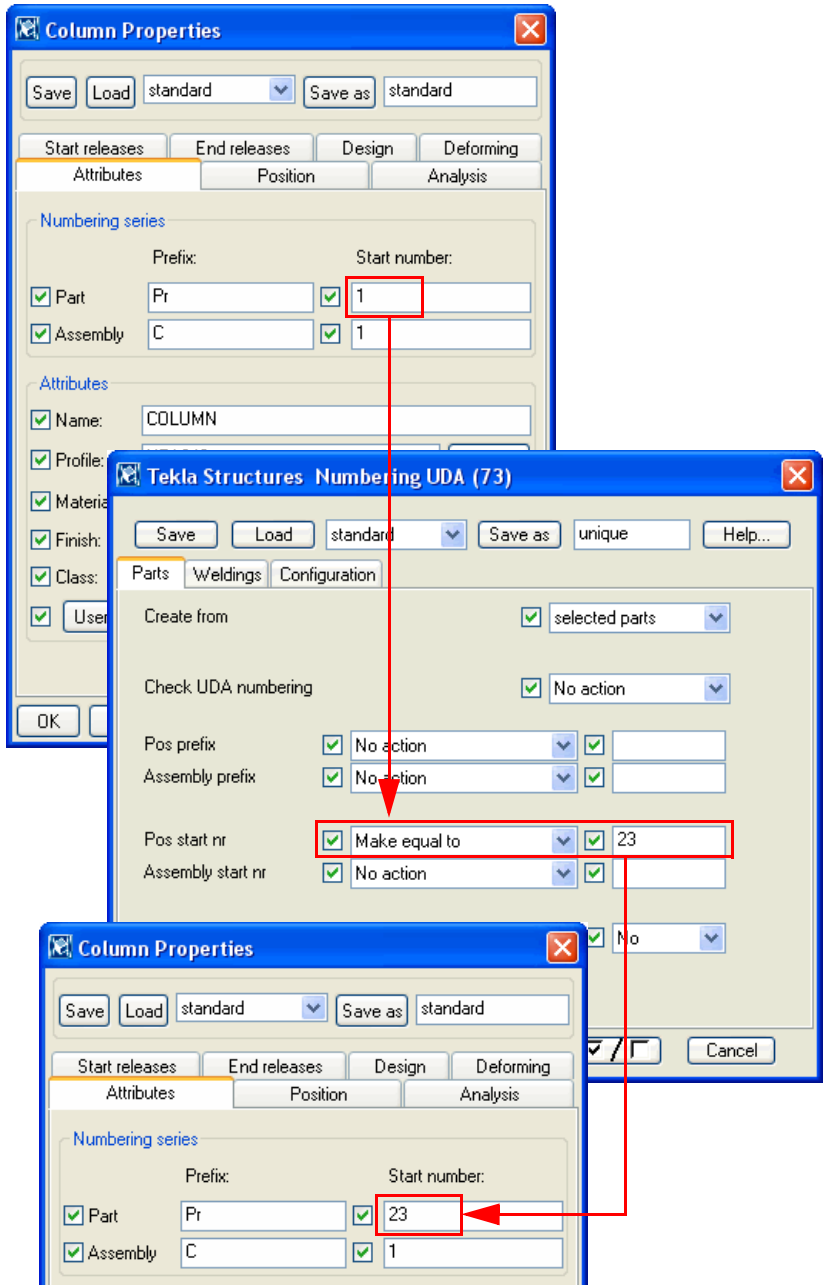
When you select this option, this line will not be executed.

Add the value

The value of the start number will be increased by the entered value.

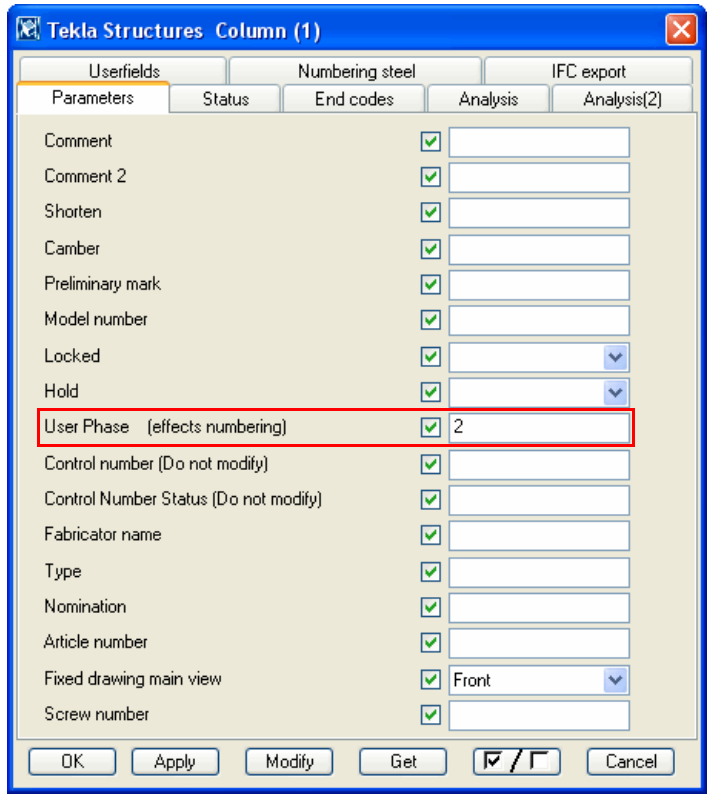

Column Properties
Save
Load
standard
Save as
standard
Start releases
End releases
Design
Deforming
Attributes
Position
Analysis
Numbering series
Prefix:
Start number:
Part
Pr
1
Assembly
C
1
Attributes
Name:
Profile:
Material:
Finish:
Class:
User-d
OK
Ap
Tekla Structures Numbering UDA (73)
Save
Load
standard
Save as
standard
Help...
Parts
Welding
Configuration
Create from
selected parts
Check UDA numbering
No action
Pos prefix
No action
Assembly prefix
No action
Pos start nr
Add the value
4
Assembly start nr
No action
Phase to use phase
No
Cancel
Column Properties
Save
Load
standard
Save as
standard
Start releases
End releases
Design
Deforming
Attributes
Position
Analysis
Numbering series
Prefix:
Start number:
Part
Pr
5
Assembly
C
1

Make equal to



Phase

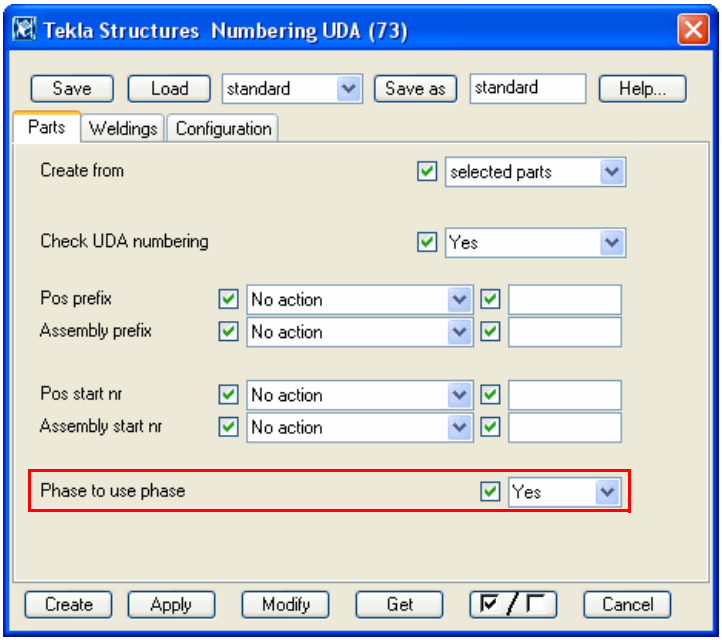
You can enter the phase number in the parts' user defined attributes. Two identical parts containing different numbers, get different part- and assembly numbers.



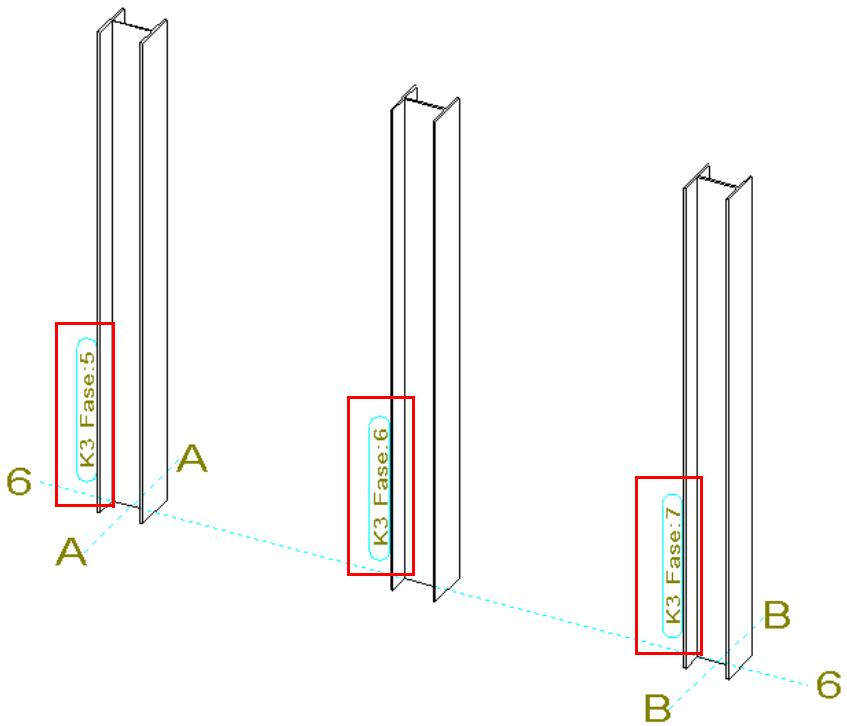
When you make use of this option to select parts and assign numbers:

- It takes a lot of time and
- there is no connection between the essential phase number and the manually added phase number.

You can overcome this by making use of this component.

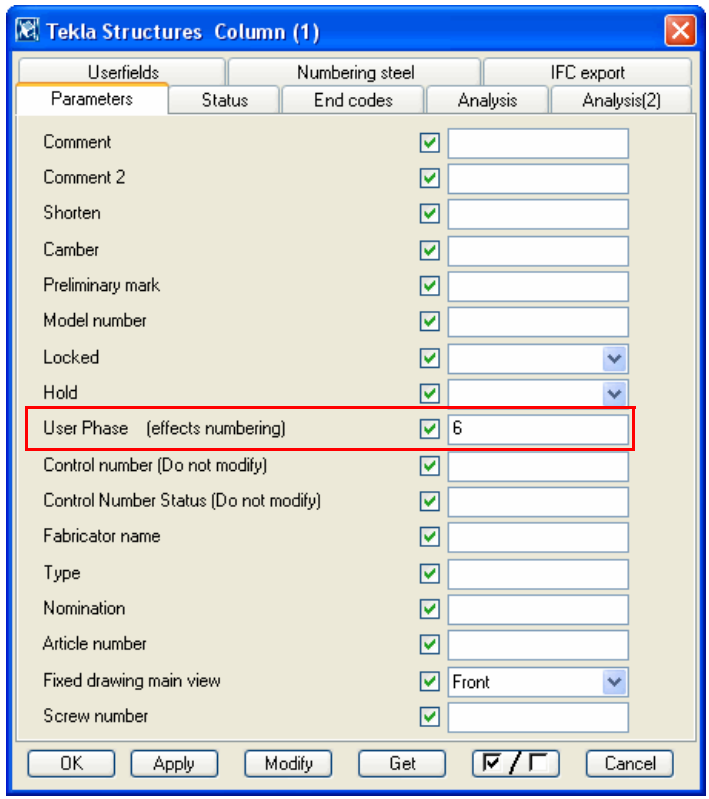


Example The 3 columns above are identical and each column is located in a different phase.



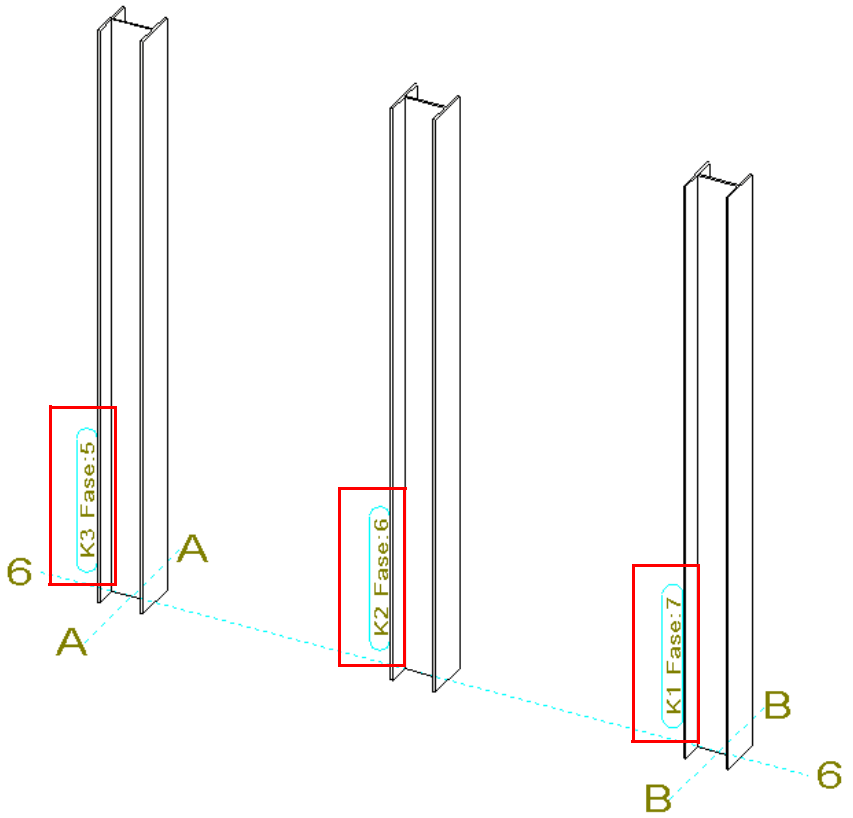
Step plan

1. Load the setting **phase_UDA** in the component
2. Select the columns and click **Create**.
3. Select the column in the middle, click right mouse and select **User-defined attributes..**



The component has now entered the selected parts' phase number.

After a full numbering you will see that the 3 columns are different because of the different values. You can check the other columns by yourself.

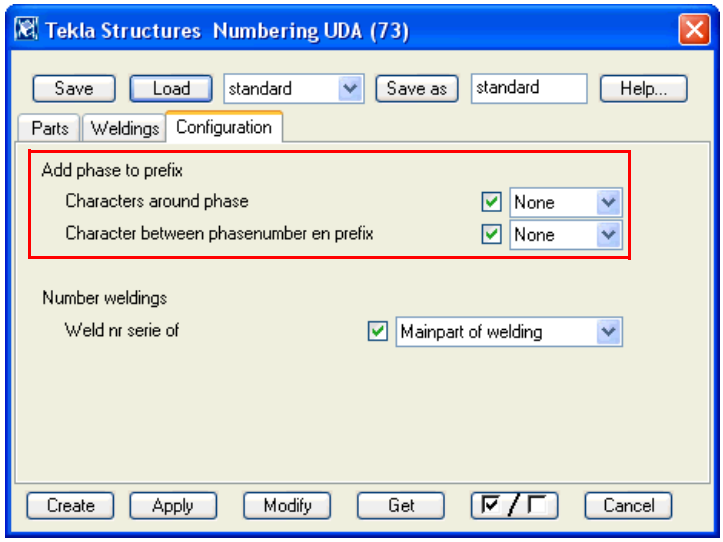


Welds

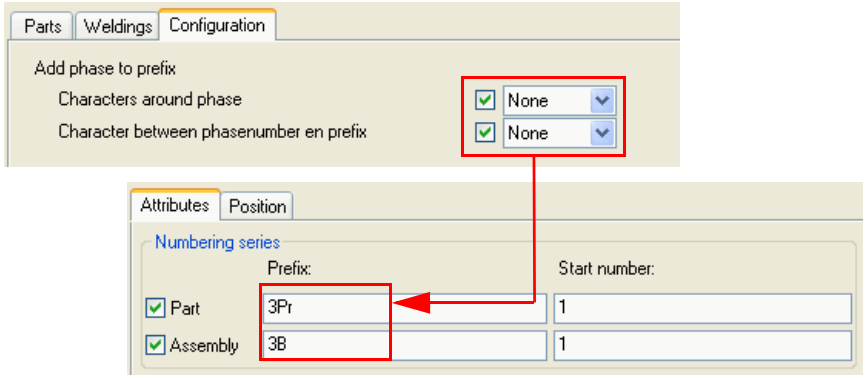
Make use of this component to assign numbers to the welds. Every weld becomes a unique weld number. This is very often used in the offshore industry.

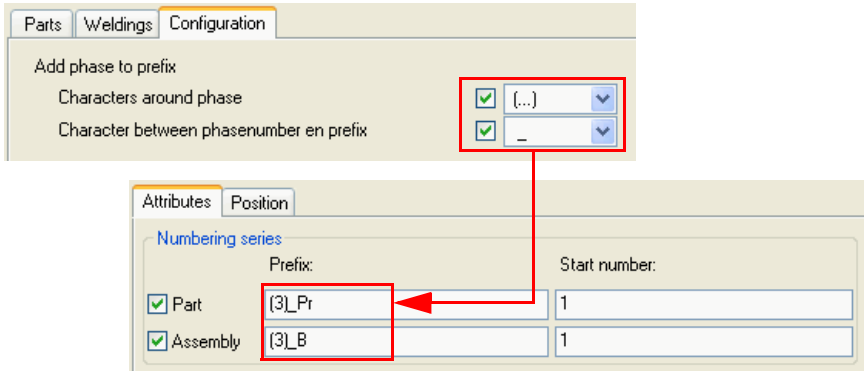
Configuration

In the tab **Configuration** you can add characters around the phase and characters between the phase number and the prefix.


Tekla Structures Numbering UDA (73)
Save
Load
standard
Save as
standard
Help...
Parts
Welding
Configuration
Add phase to prefix
Characters around phase
None
Character between phasenumber en prefix
None
Number weldings
Weld nr serie of
Mainpart of welding
Create
Apply
Modify
Get
Cancel

Examples


Parts
Welding
Configuration
Add phase to prefix
Characters around phase
None
Character between phasenumber en prefix
None
Attributes
Position
Numbering series
Prefix:
Start number:
Part
3Pr
1
Assembly
3B
1


Parts
Welding
Configuration
Add phase to prefix
Characters around phase
(...)
Character between phasenumber en prefix
_
Attributes
Position
Numbering series
Prefix:
Start number:
Part
(3)_Pr
1
Assembly
(3)_B
1

3.24 Opening reports in Microsoft Excel and Word

Topics

This chapter is divided into following sections:

Opening reports directly in Microsoft Excel (p.294)

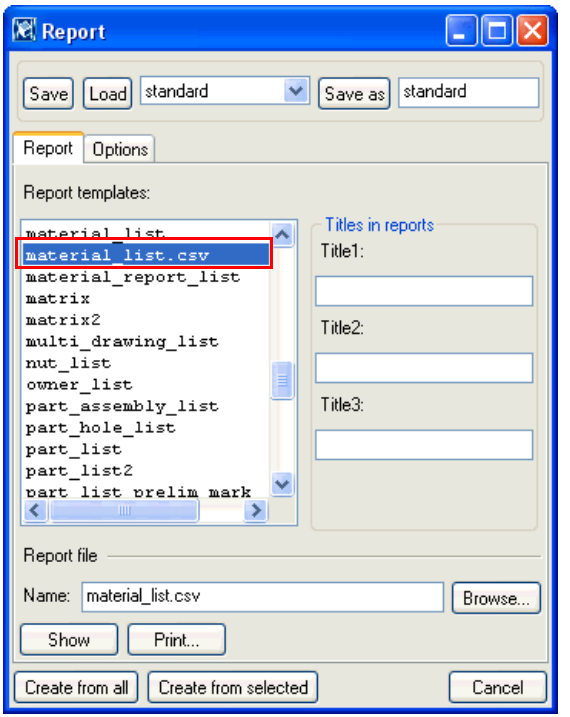
Importing reports in Excel (p.295)

Opening reports automatically in Word (p.299)

Opening reports directly in Microsoft Excel

Reports generated in Tekla Structures can be opened directly in Microsoft Excel©, e.g. the *materiallist.* (This report is a standard report.)

The filename is: *material_list.rpt.csv.*

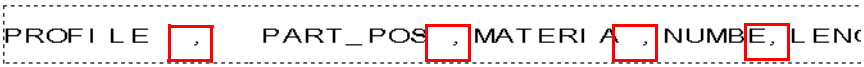


This report will be listed as *material_list.csv* in the dialog box.

Next, the content of the file has to be modified, because Excel needs to know where the separation(s) have to be.

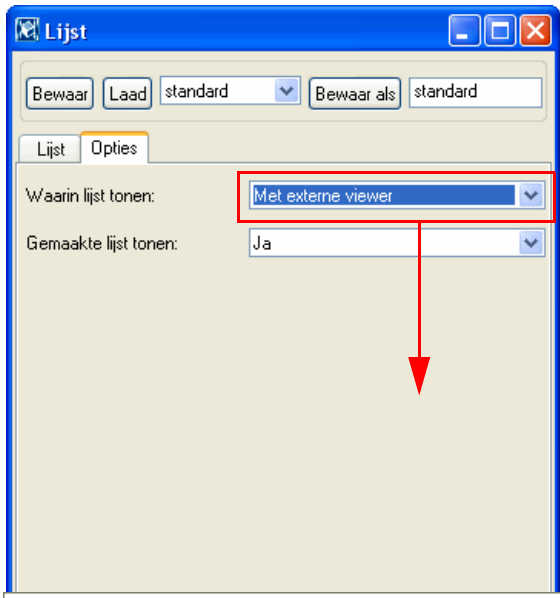
Place semicolons (;) between the variables

(In a non-english Excel version, use comma's instead of semicolons)



Now, when the report is generated, the file extension won't be xsr, but it will get extension **.csv.* The file name will be *material_list.csv.*This file type is automatically recognized by Excel.

If you generate a report, please take care of the settings in tab **Options:**



TEKLA STRUCTURES MATERIAL LIST Page: 1
Model:Modelname Project: project nummer
Date: 01.02.2008
Time: 13:48:05

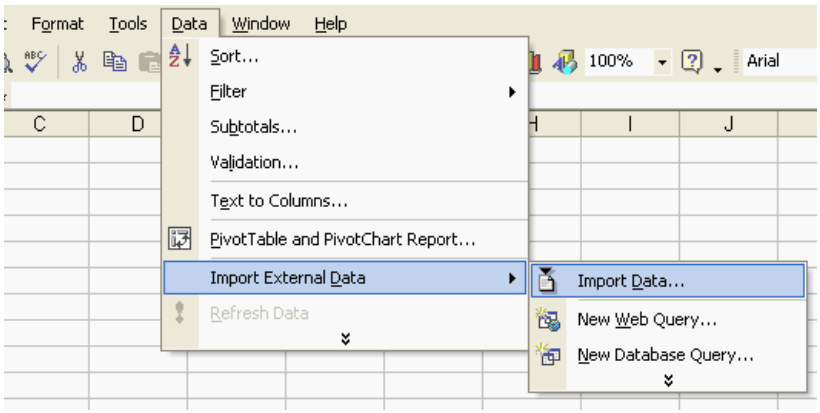
Profile	Part	Grade	Number	Length(mm	Area(m²)	Weight(kg)
FLAT10	P1	S235JR	1	290	0.10	3.4
290	0.10	3.4				
HEA24	Pr1	S235JR	1	3500	4.80	211.0
3500	4.80	211.0				
IPE300	Pr2	S235JR	1	2874	3.33	121.3
2874	3.33	121.3				
	335.7					

Importing reports in Excel

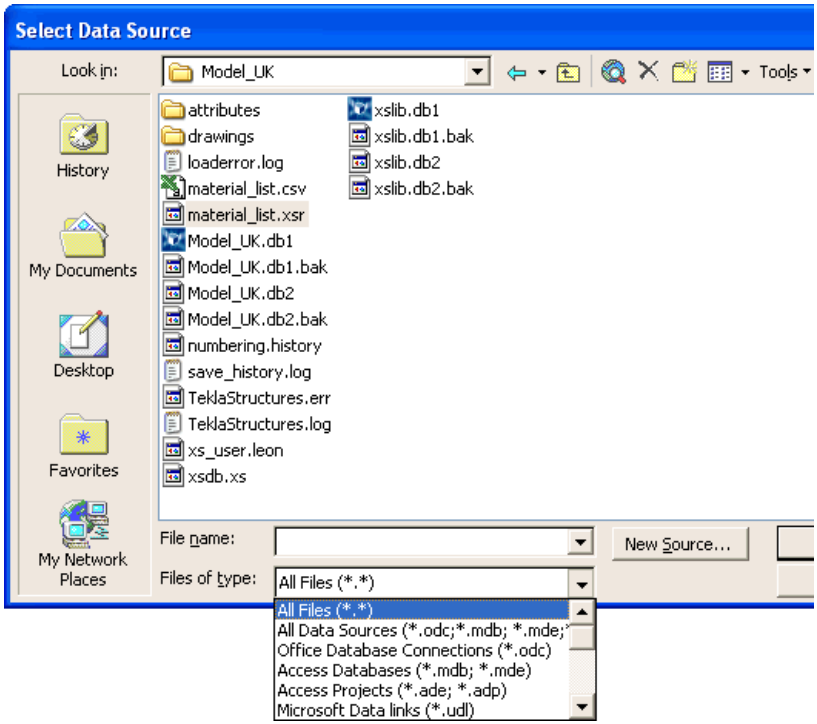
Again, for example we take the report *materiallist.*

Step plan

1. Create a materiallist in Tekla Structures
2. Now open Microsoft Excel and go to **Data** > **Import External Data** > **Import data...**

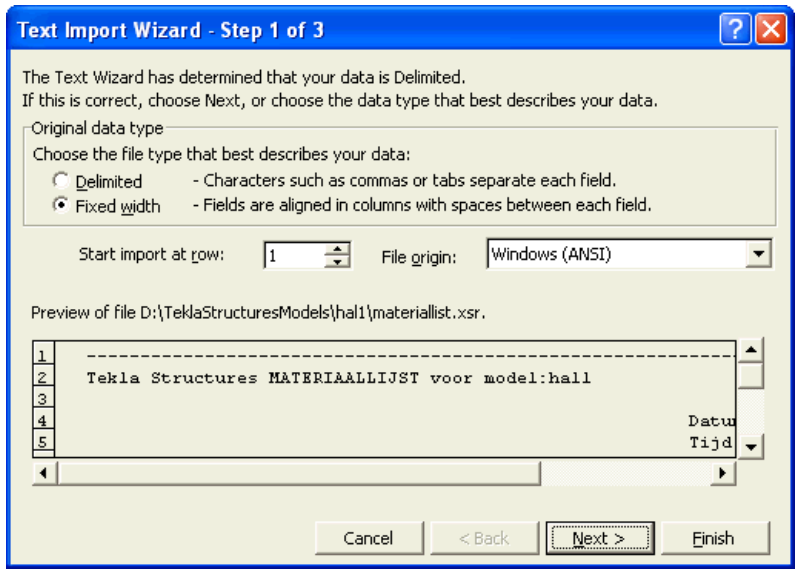


The following dialog box appears:

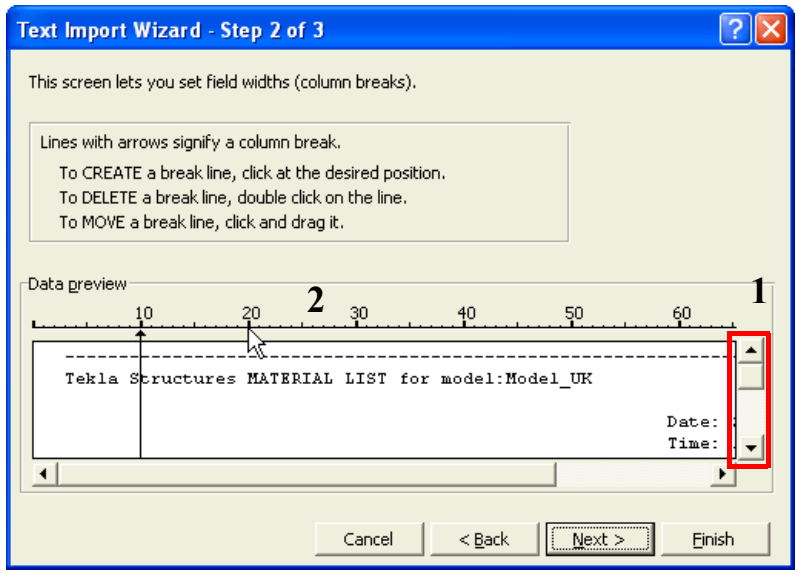


3. Set files of type to **All files (*.*)** so that reports in Tekla Structures with the extension **.xsr* appear in the box.
4. Click <**Open**>

5. The **Text Import Wizard** appears:

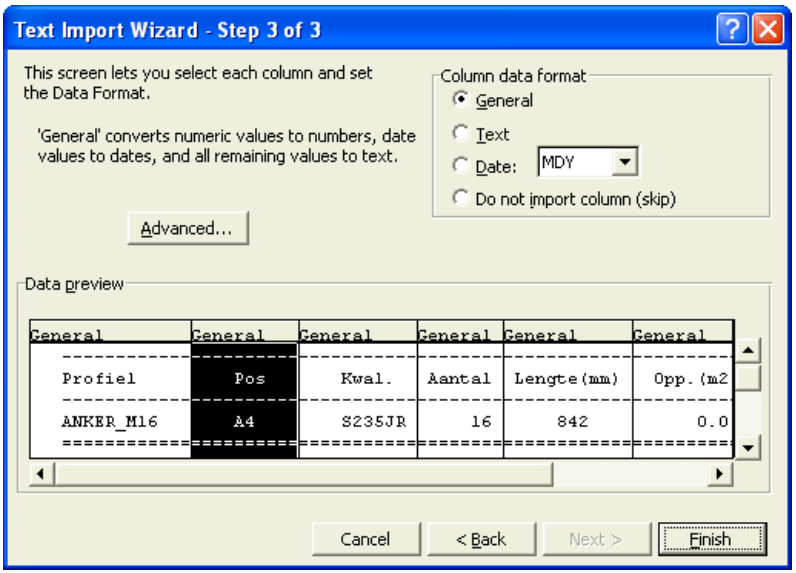


6. Set to **Fixed width** and click <**Next>**

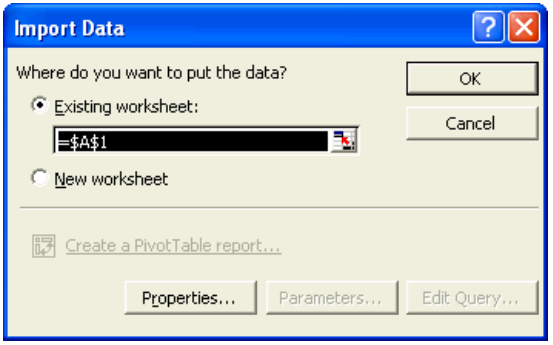


7. Make use of the slider bar to go to the position in the report to set the column breaks **(1)** and click the point to create the break **(2)**

After setting the breaks, you can always afterwards change the position of the breaks.



8. Click **<Finish>**, the following dialog box appears:



- Click <**OK**>, the materiallist in Excel looks like this:

Profile	Partno.	Grade	Number	Length(mm)	Area(m2)
HEA140	Pr85	S235JR	1	5076	
			1	5076	
K80/8	Pr102	S235JR	1	6911	
			1	6911	
L60/6	Pr108	S235JR	1	4038	
			1	4038	

Opening reports automatically in Word

Reports can be opened in Word automatically. There are two options.

Change the name of the reports

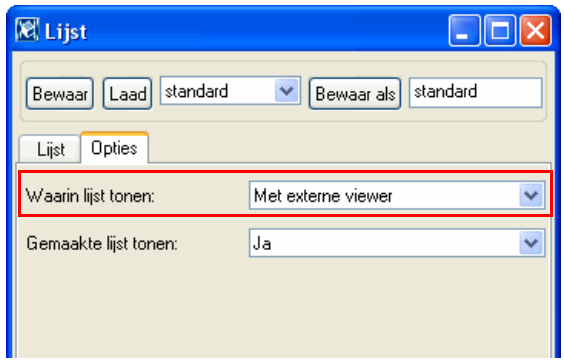
Step plan

1. Go to the folder *...environments\europe\module\std*



The folder depends on the module you use, **std** could aslo be **pcd** or **sde**.

2. Select a report, for example **phasemanager.rpt** and rename this file to **phasemanager.doc.rpt**
3. Set in Tekla Structures the dialog box **Report** as follows:



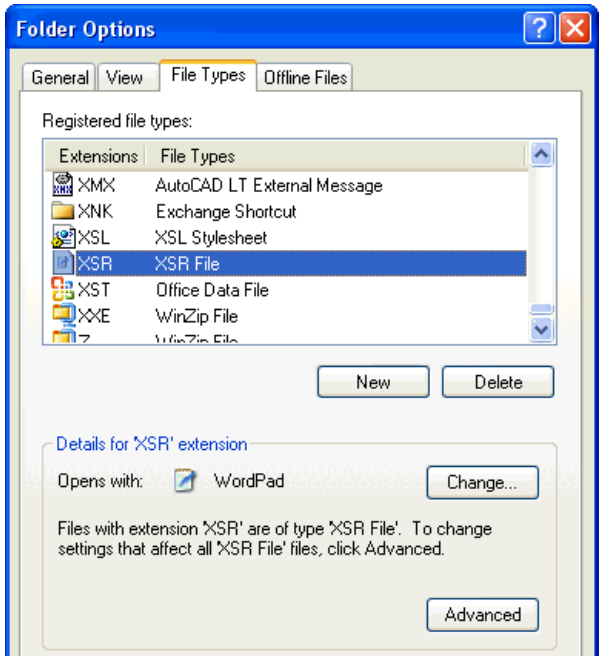
When you now create a report, it will be opened automatically in Word.

The file **phasemanager.doc** will be stored in the folder `Reports` in the model folder.

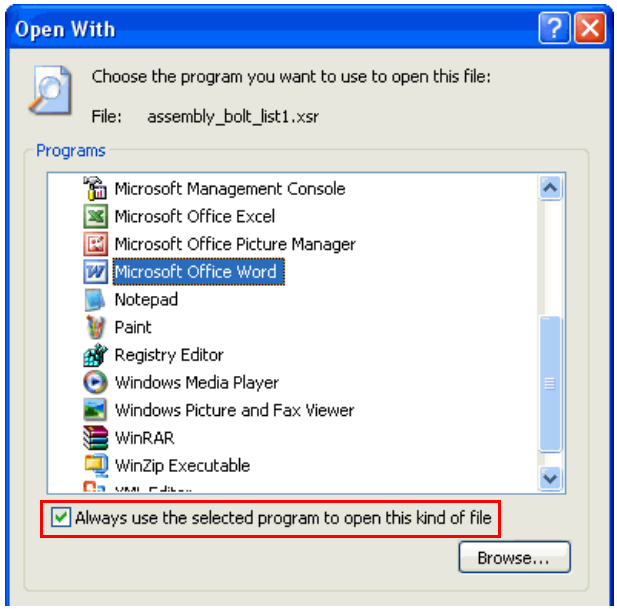
Link the file extension *.xsr

Step plan

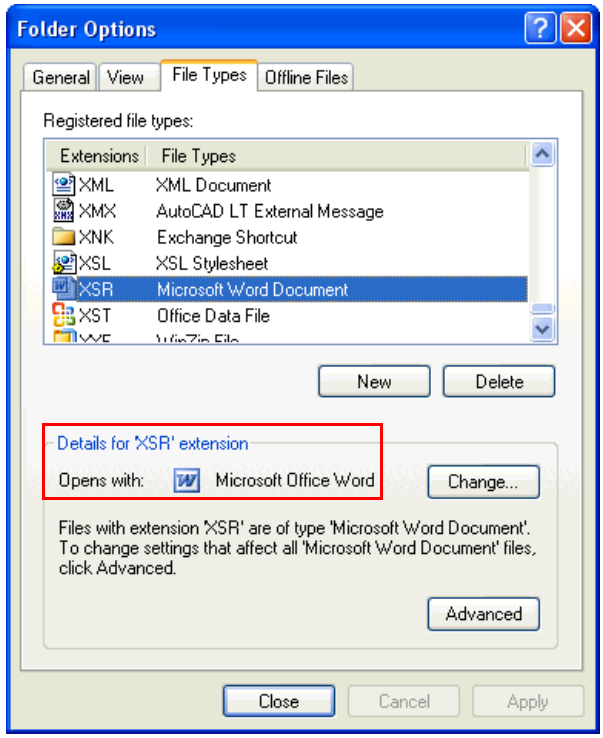
1. In the Explorer, go to **Tools** > **Folder options...**
2. Select the extension **XSR** on the tab **File types**.



3. Click on the button <**Apply...**>, the following dialog box appears:

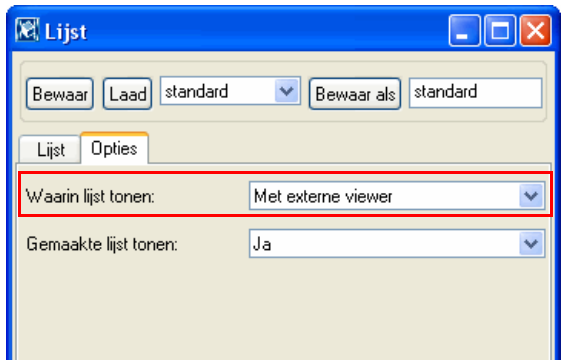


4. Now select **Microsoft Office Word** in the list of programs. Enter a check-box by **Always use the selected program to open this kind of file** and click <**OK**>.



The file extension XSR is now linked to Word.

5. Set in Tekla Structures the dialog box **Report** as follows:

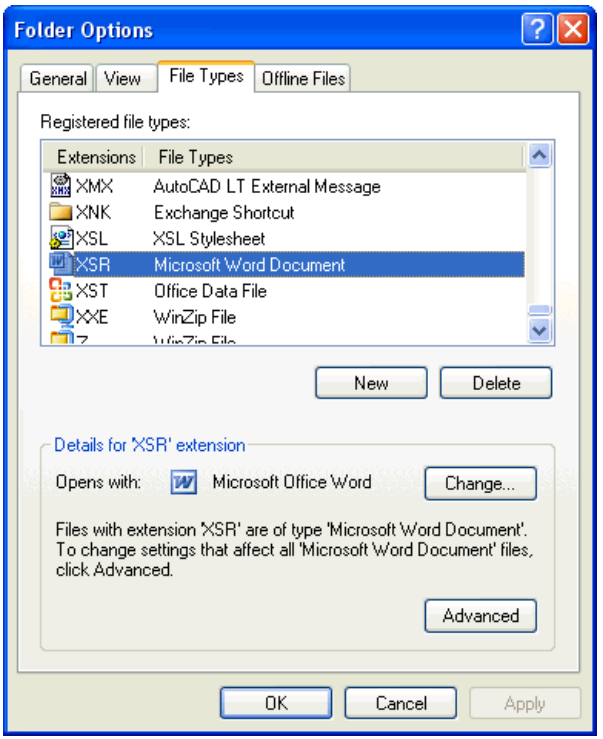


The report will be opened in Word.

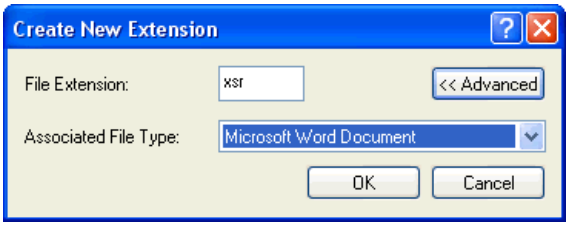
Sometimes, this will not work properly, in those cases you do next:

Step plan

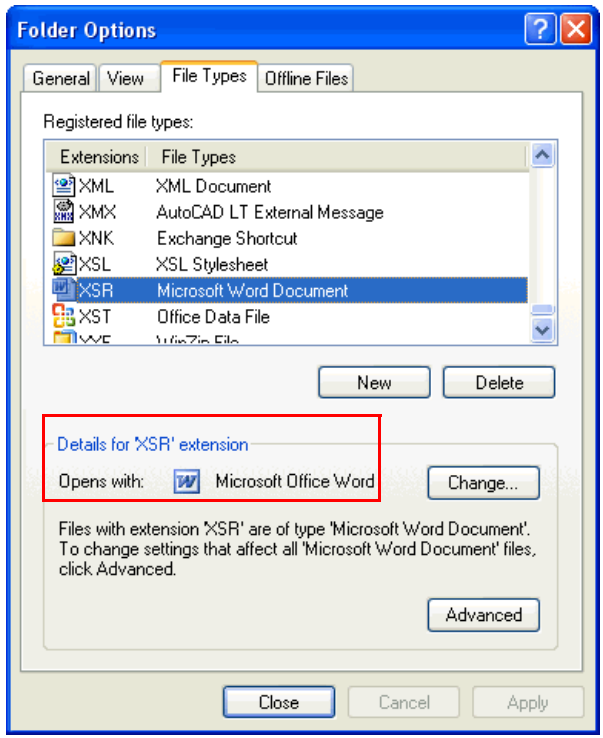
1. In the Explorer, go to **Tools** > **Folder options...**
2. Select the extension **XSR** on the tab **File types**.



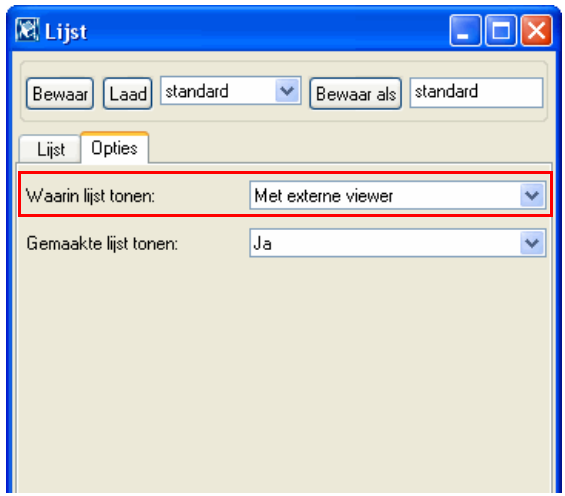
3. Click the button <**Delete**> and <**Yes**>, the file extension is now deleted.
4. Click the button <**New**>, the following dialog box appears:



5. Enter the extension **xsr** and select **Microsoft Word-document** in the list-box, click <**OK**>. File extension XSR is now linked to Word:



6. Set in Tekla Structures the dialog box **Report** as follows:



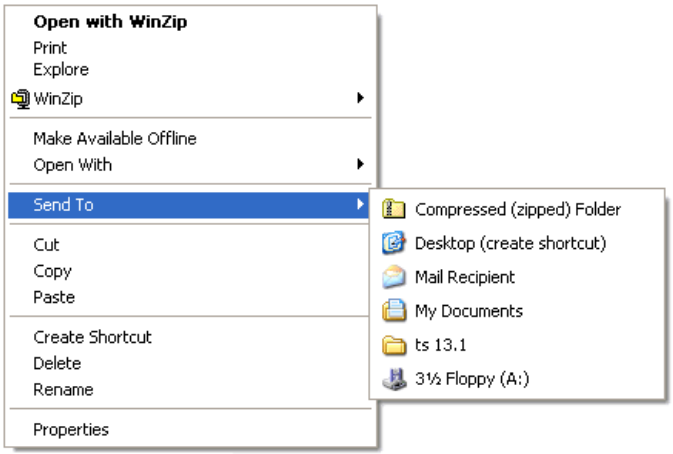
The report will be opened in Word.

3.25 Extend the contextmenu "Copy to"

It happens very often that you, as a Tekla Structures user, copy settings that you have made by yourself from the *model* folder to the folder *ts* to make use of these settings in all models. Copying these files can be done very easily.

When you select a file in the Windows explorer and you click the right mouse button, a contextmenu appears. This menu contains the option: “Send To”.

You have the possibility to extend the option “Send To” with several locations. It is very handy to add the location “ts” to the contextmenu “Send To”. In this way you copy settings to the folder “ts” quick and easily. See the following picture:



Step plan

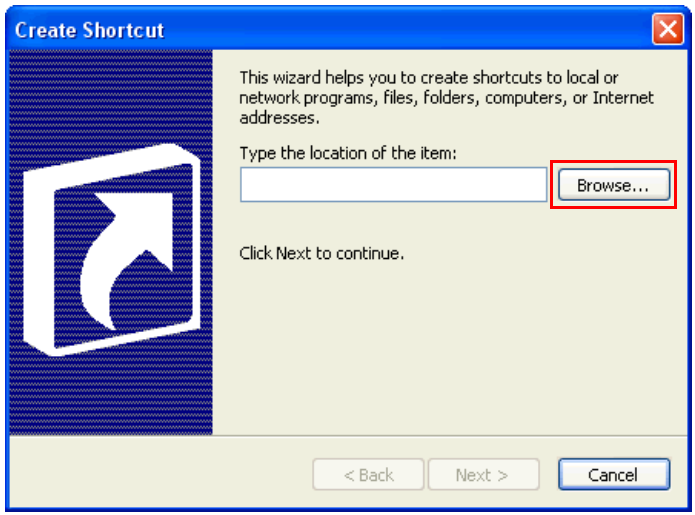
1. Start the Windows explorer and go to the file

   C:\Documents and Settings > **(your name)** > **Send To.**

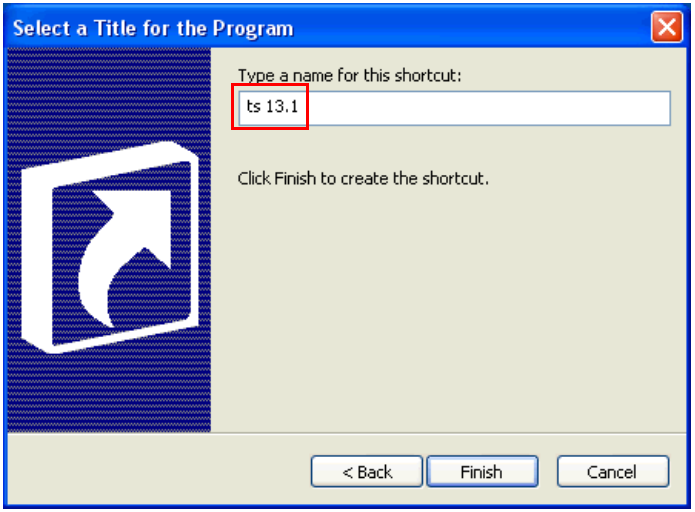
   When you have opened this folder, you will find the existing locations belonging to the contextmenu “Send To”.

   When the folder **SendTo** not appears, in the Explorer, go to **Tools** > **Folder options**... In the tab *View,* you toggle on a check box for "Show hidden files and folders" in the Advanced settings.

2. Open the folder “Send To” and go to **File** > **New** > **Shortcut** in the Windows explorer in the pull-down menu. The following dialog box appears:

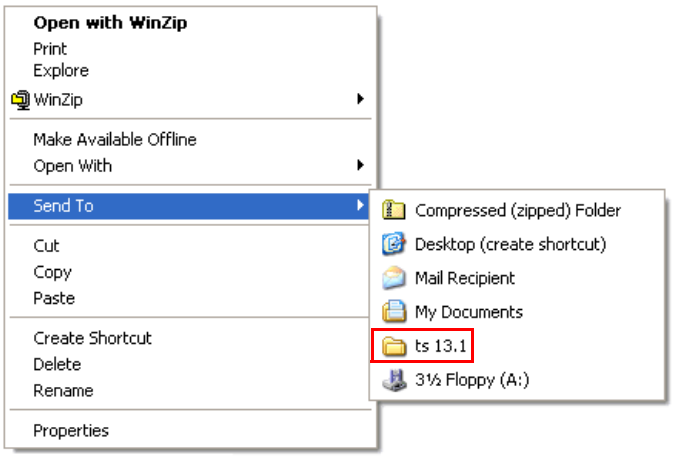


3. Click <**Browse**> and select the location that has to be added in the contextmenu “Send To” (*drive:\TeklaStructures\version\environments\europe\firm\ts*) and click <**Next>.** The following dialog box appears:



4. Type in a name and click <**Finish**>.

When you select a file in the Windows explorer and you click Right mouse button > “Send To”, the added location appears in the menu! So from now on you can use this option to copy files from the model folder to the folder “ts”!



3.26 Options dialog box

Topics

This chapter is divided into following sections:

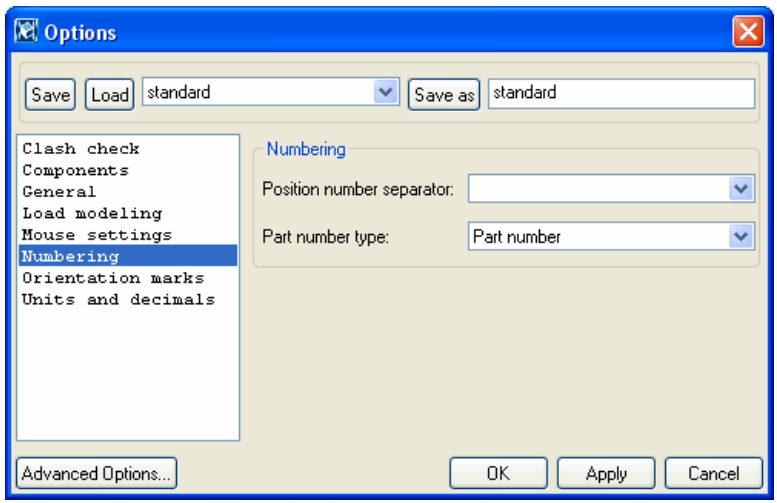
Numbering (p.307)

Components (p.308)

The categories **Components** and **Numbering** in the dialog box **Options** concern the general numbering and properties of **parts created by components.**

Go to **Tools** > **Options** to open the dialog box.

Numbering



Position number separator

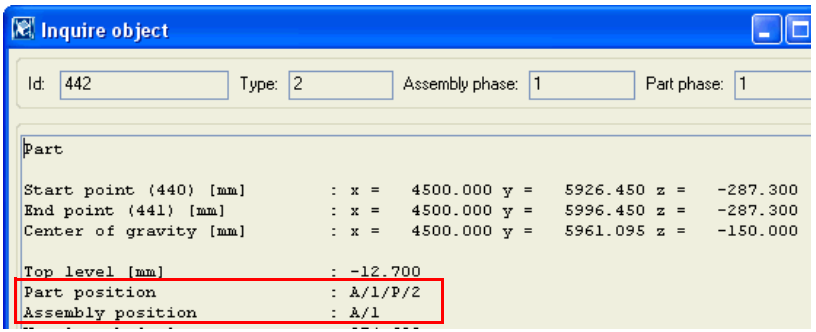
The separating character is a character used between the prefix and the running-number. The options are:

Separator	Example
None	P123
Point	P.123
Comma	P,123
Slash	P/123
Minus	P-123

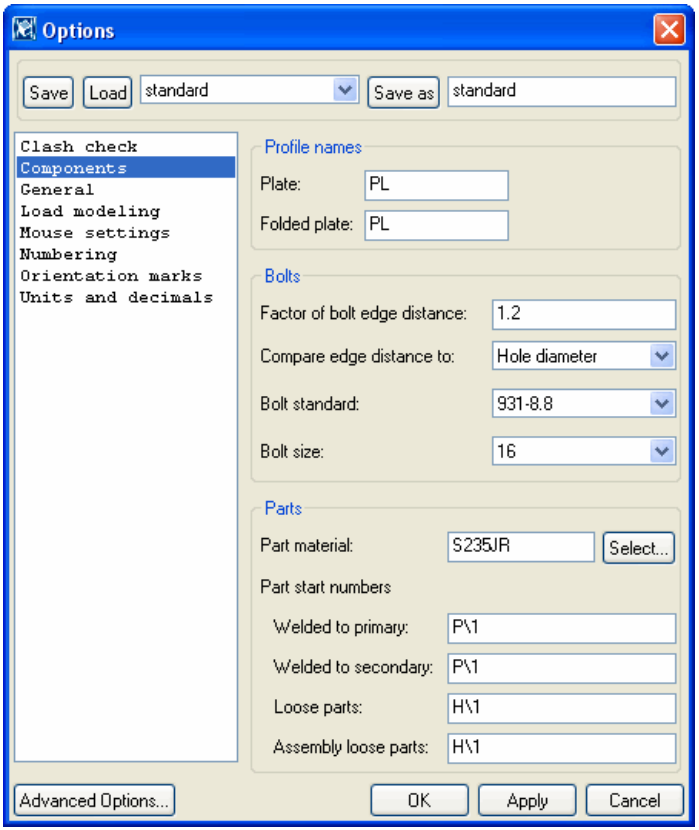
Part number type

Part number: only the part- or assembly number is returned.

Combined assembly-/ part number: the part number consists of the part number and the assembly number:



Components



Profile names

Plate

Indication for plates created by components. If **PL** is established, a plate created by a component will be indicated as **PL**10*140.

Folded plate

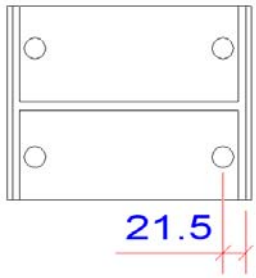
Set to **PL** by default. Do not change this.

Bolts

Factor bolt-edge distance

This factor checks the bolt edge distance validity for bolts/holes. If the distances are valid, the component symbol will be displayed in green color. If the distances are invalid, the symbol will be yellow colored.

Example



Set to **bolt diameter**:

Entered **Compare edge distance to** Factor: 1.2

Bolt size: M16

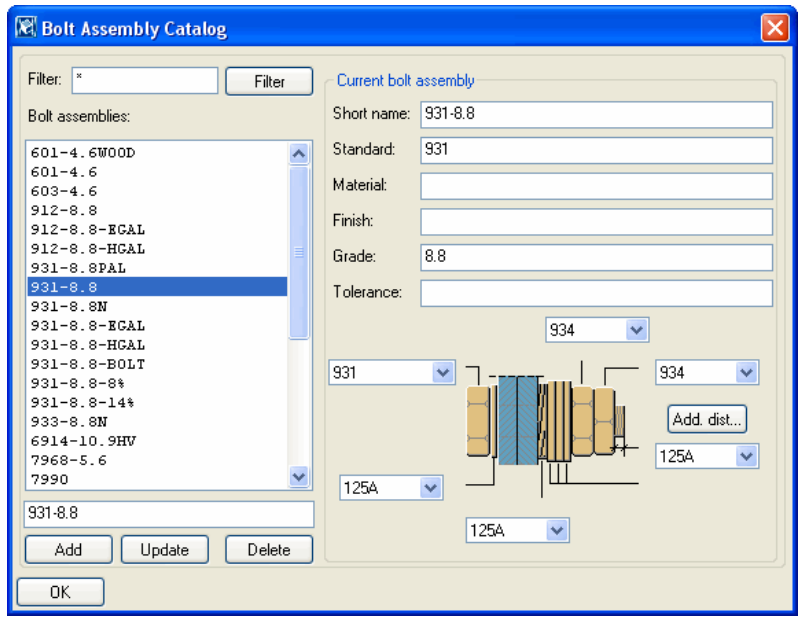
Minimal edge distance:16 * 1.2 = 19.2 mm.

Edge distance is bigger than minimal distance of 19.2 mm, so the component symbol will be green colored.

For manually modelled bolts/holes a ‘beep’ sounds.

Bolt standard

The default bolt standard for bolts created with components.The standard must exist in the bolt standard catalog (**File > Catalog > Bolts > Modify assemblies**):



Bolt diameter

The default bolt diameter on the basis of the bolt standard.

Parts

Part material

The default material for parts created with components.

Part start numbers

Welded to primary

To set the part numbers' prefix and accompanying start number for welded parts created by components. In the component dialog box (tab Parts), no prefix and start number needs to be entered. Prefix and start number need to be separated by a backslash-character (\). By default set to P/1.

Welded to secondary

Here the prefix and accompanying start number can be set for component parts which are welded to a secondary part. By default set to P/1.

Loose parts

Here the prefix and accompanying start number can be set for component parts which are not welded to another part (i.e. part number for loose parts). By default set to H/1.

Assembly loose part

Like Loose parts, but this input field defines the prefix and accompanying start number for the assembly number of component parts (i.e. assembly number for loose parts). By default set to H/1.

3.27 Ini files

Topics

This chapter is divided into following sections:

Batch files replaced by ini files (p.311)

Modifying variables (p.312)

Copying batch file settings to ini files (p.314)

Working for multiple contractors (p.315)

Reference to server (p.320)

Batch files replaced by ini files

Ini files

Batch files are no longer used in Tekla Structures; `ini` files are used instead (*.ini).

This change was necessary to certify Tekla Structures for Windows Vista and to make it Windows Vista compatible.

`Ini` files can be modified in the same way as `batch` files.

`Ini` files are text files, which can be opened in any text editor.

The modifications in relation to Tekla Structures 12.1 are:

TS 12.1	TS 13.1

`europe_env.bat`
in folder
`..\bat\environment`
	`env_europe.ini` in folder
	`..\environments\europe`

	`env_global_default.ini` in folder `..\environments\country-independent`
`nld.bat` in folder `..\bat\language`	`lang_nld.ini` in folder `..\13.1\nt\bin`
`user.bat` in folder `..\bat\user`	`user.ini` in folder `..\13.1\nt\bin`

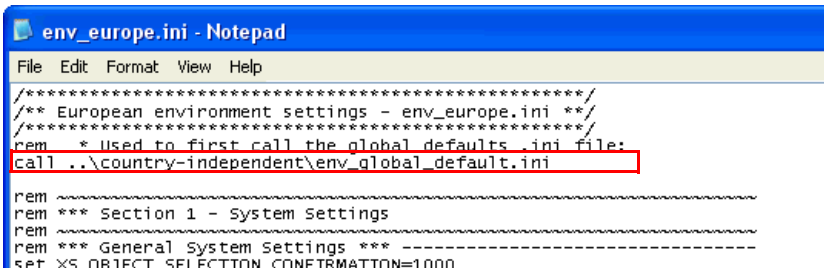


How are the new `ini` files being read? The Tekla Structures 13.1 shortcut contains the following lines:

C:\TeklaStructures\13.1\nt\bin\TeklaStructures.exe -i
C:\TeklaStructures\13.1\environments\europe\env_europe.ini -i
C:\TeklaStructures\13.1\environments\europe\std.ini -i
C:\TeklaStructures\13.1\nt\bin\user.ini

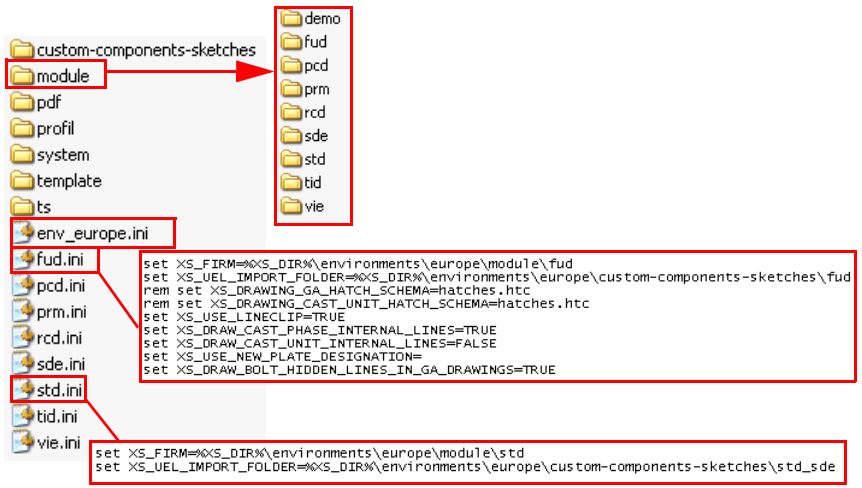
These `ini` files are being read as:

1. The file **TeklaStructures.exe** is being executed
2. The file **env_europe.ini** is being read



In this file the line started with `call ...` is being read, the file **env_global_default.ini** is being read.

3. The file **std.ini** is being read, dependable of the module being used:



This, to make difference between the various software configurations (e.g. STD (Steel Detailing) and SDE (Standard Design). Customer specific settings using the STD module are now stored in folder `...europe\module\std`.

4. The file **user.ini** is being read

The `ini` files are being read in this order. This means that lines in file **user.ini** have the highest priority and therefore will overrule identical lines in previous `ini` files.

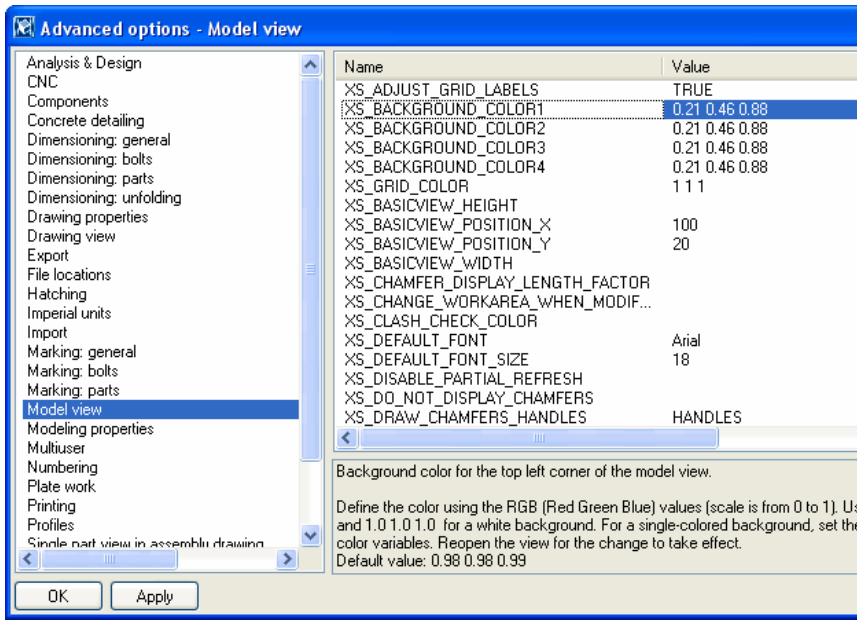
Modifying variables

You can modify variables on two levels:

- Model level
- User-specific level



Go to **Tools** > **Advanced options...** or press short key **<Ctrl> + E**. The following dialog will appear:

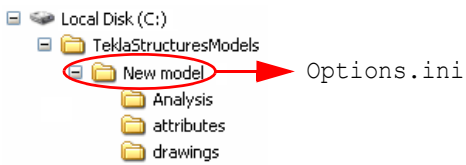


Variables are grouped in categories. Categories are listed on the left side. Click a category to view the accompanying variables.

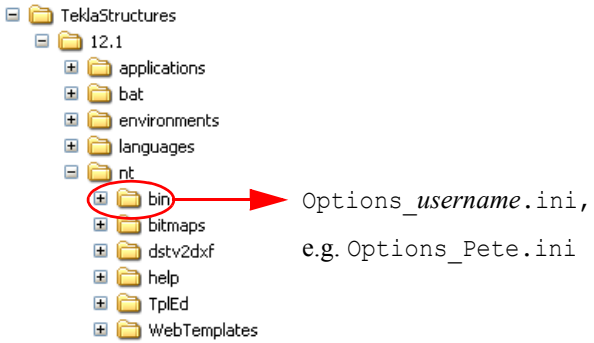
Several variables have a short description to explain the working of the variable. We plan to have descriptions for all variables in a future version.

Modifications are stored as follows:

1. This file contains modified variables on model level; these changes apply to the current model only.



2. This file contains all modified user-specific variables which apply to all models which are opened by that particular user.



If you want to make changes which affect all models, we recommend to apply modifications in file **user.ini**, this file is stored in folder: `..\TeklaStructures\13.1\nt\bin`.



Note: when copying lines to **user.ini**, be sure that the lines start with `set`!

In this way the changes are automatically copied when installing a next version of Tekla Structures.

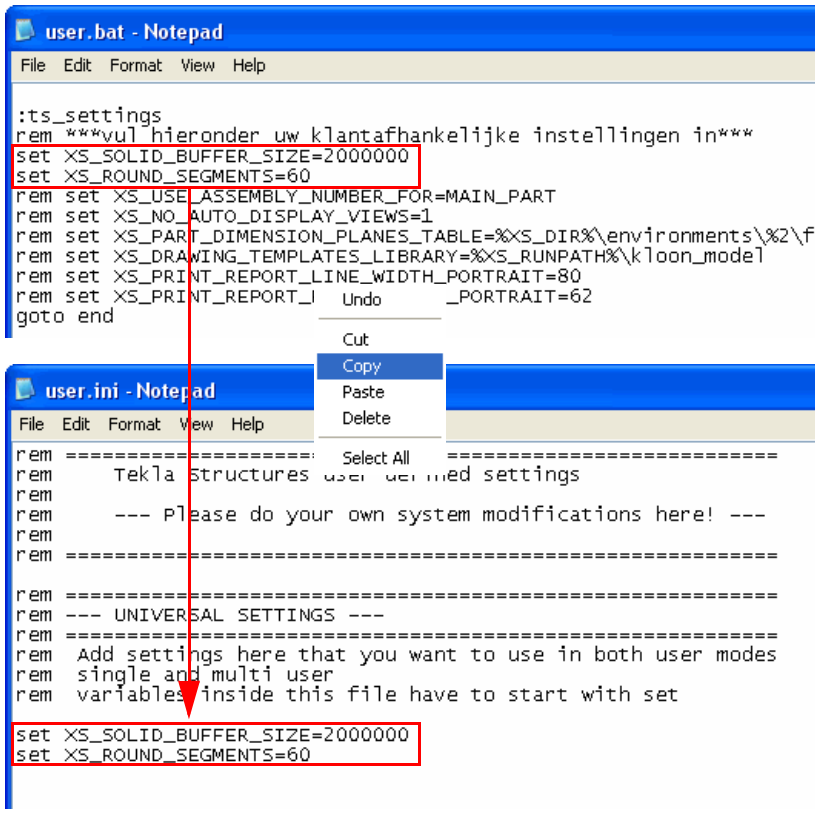
Copying batch file settings to ini files

When you already use Tekla Structures you want to copy your old 'batch' settings to Tekla Structures 13.1.

Step plan

1. Open file **user.ini**, located in folder: `..\TeklaStructures\13.1\nt\bin`.
2. Open file **user.bat** from the previous version, located in folder: `..\TeklaStructures\version\nt\bin`.

3. Copy de desired lines from file `user.bat`.



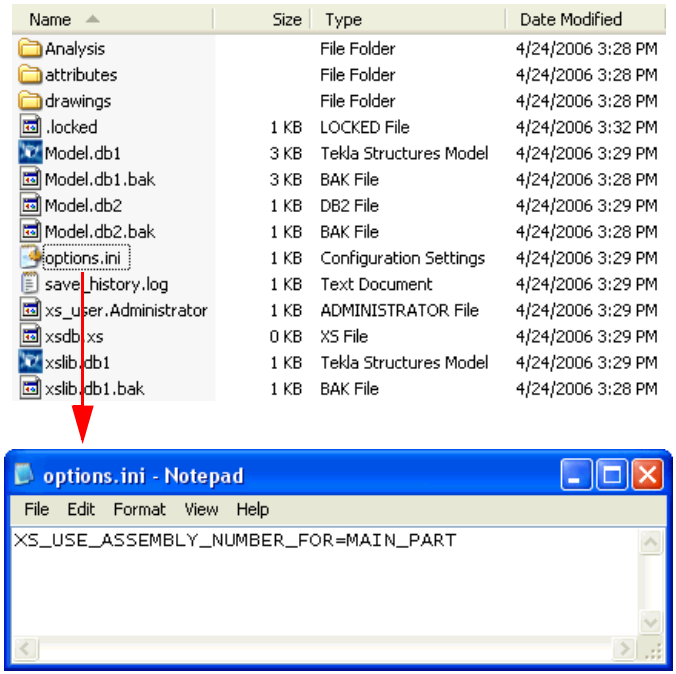
4. Paste the lines in file `user.ini`.
5. Save file `user.ini`.

Working for multiple contractors

Using a default model

If you work for multiple contractors, you can choose to use a standard model for each contractor.

You can do this by storing a **options.ini** file into a default model. This file contains the modified contractor-specific variables.



It is also possible to save the specific settings (e.g. drawing settings, templates and template-layouts) in the contractors default model. (`*.bin` files in the model folder, other files in `attributes` folder).

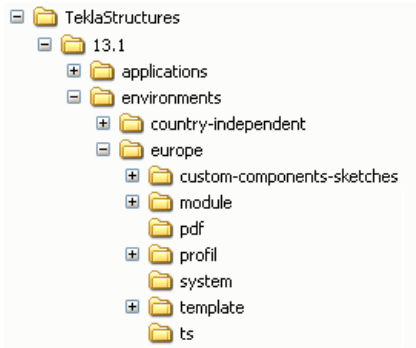
Each time you save the default model to a new name, all settings are automatically saved into the new model, so all model variables are used.

Notes

- If you want to modify the saved file `options.ini` (in the model folder), it is allowed to modify the file directly using a random text editor (e.g. Notepad). So it isn't necessary to use the **Advanced options.**
- Copying the file `options.ini` from one model to another model is allowed. If the target model is already opened, re-opening the model is required to apply the changes.

Using a contractor depended folder

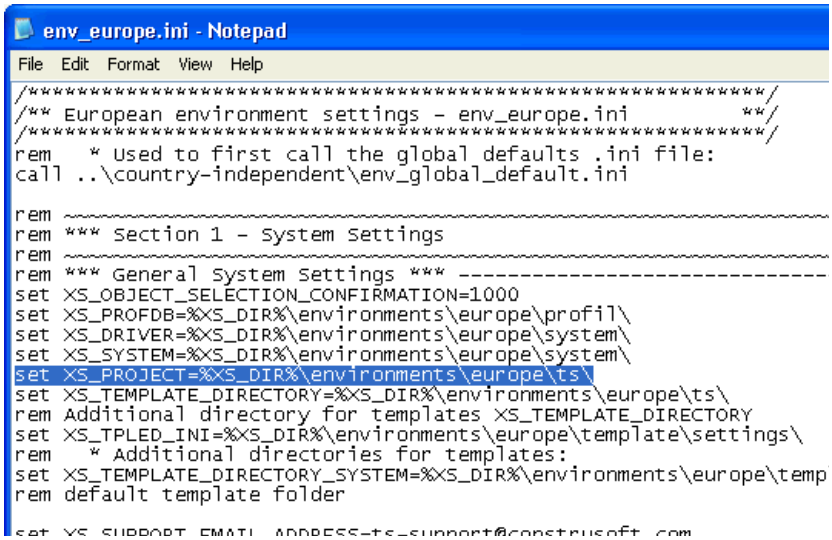
If you use a standard model, then the variables and setting files are stored in the model folder; the other method is to store these contractor-specific files in a special folder (like storing your personal files in the folder `ts`.



Create multiple shortcuts, to which the variables and settings are linked.

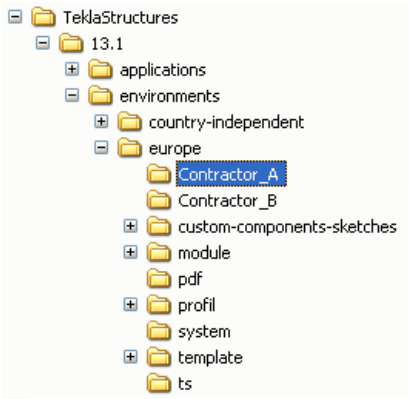
When starting Tekla Structures, several `ini` files are read, including `env_europe.ini` and `user.ini`.

By default the settings in the folder `ts` are used, because the file `env_europe.ini` contains a reference to that folder:



If you are going to use settings from a contractor (e.g. drawing settings, templates and layouts), then it is best to ask for the contractors entire `ts` folder; this folder contains all contractor specific settings.

Next, in the `europe` folder, create a new folder, name it after the contractors name and copy the content of the contractors `ts` folder to this folder:

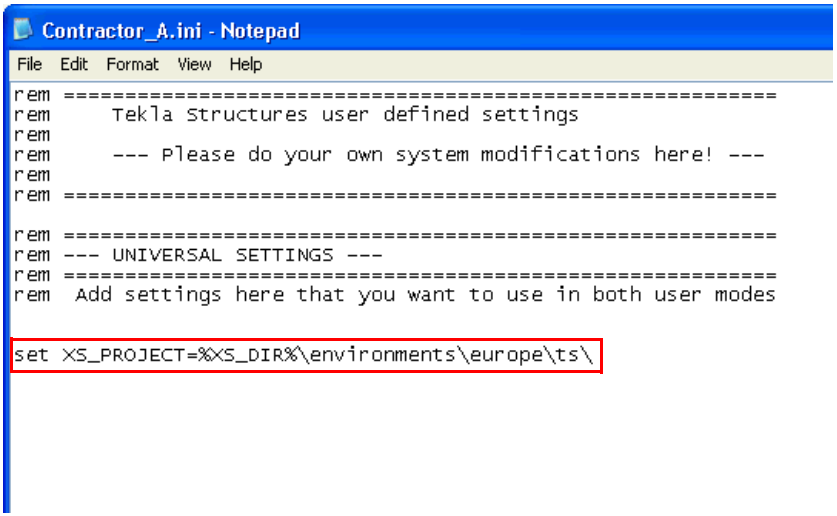


Now go to file `user.ini` (located in folder ..\TeklaStructures\13.1\nt\bin) and create a copy of this file. Rename the file to `Contractor_A.ini`.

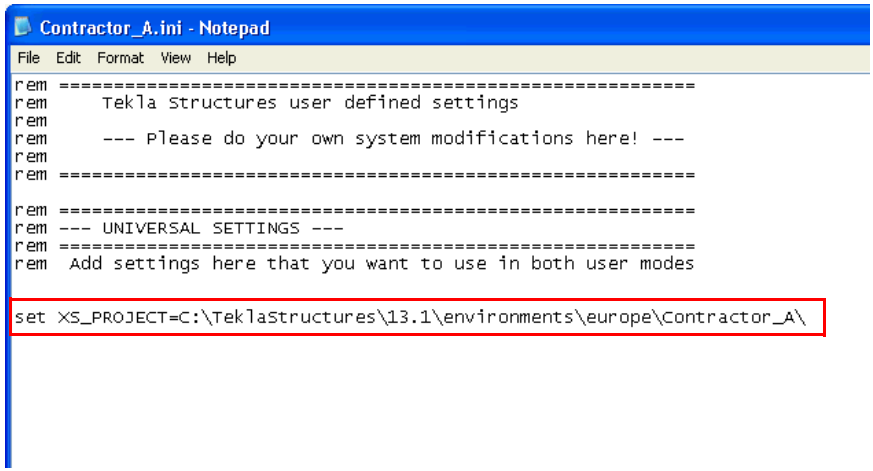
Next, copy the following line from file **env_europe.ini**

```
set XS_PROJECT=%XS_DIR%\environments\europe\ts\
```

and add this line to file `Contractor_A.ini`:



Change the path by replacing the link into the full path:

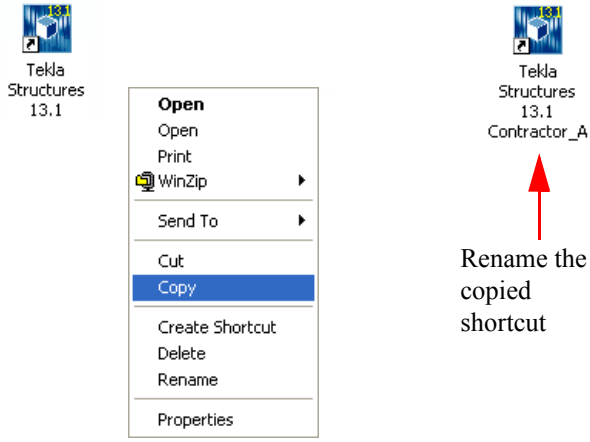




You **must** enter the full path.

Replacing `ts` into `Contractor_A` is not sufficient!

Copy the shortcut on the desktop and rename it:



Go to the shortcut's properties. For 'Target', replace the link to `user.ini` into `Contractor_A.ini`. This results in this file being read, including variable:

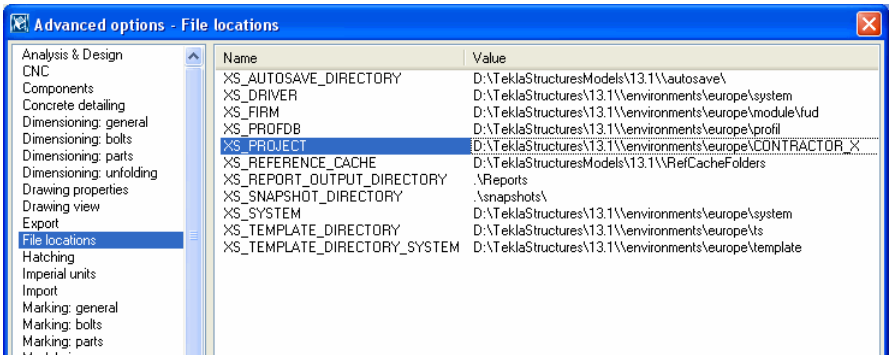
```
set
XS_PROJECT=C:\TeklaStructures\13.1\environments\europe\Cont
ractor_A\
```

When working for this contractor, always start Tekla Structures using this shortcut.

Using a standard model and Contractor specific settings

You can also use a standard model but have the settings in the contractor depended folder instead in the model folder.

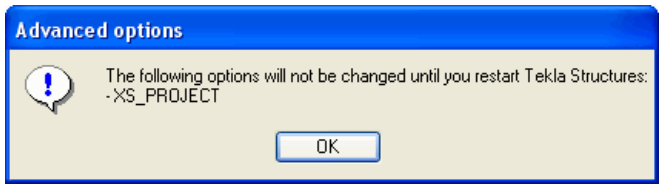
- Go to **Tools** > **Advanced options...**
- On the left side, select the **File locations** category



- On the right side, click line XS_PROJECT

Change the value to C:\TeklaStructures\13.1\environments\europe\Contractor_A

- Click <**Apply**>, the next dialog shall appear:



- In dialog Advanced Options, click <**OK**>

The file `options.ini` has been added to the standard models folder. This means that the folder Contractor_A will be read first when opening a model.

Reference to server

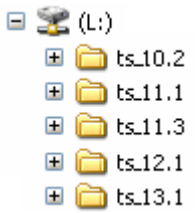
The `ts` folder is installed automatically. If multiple licenses are in use in one company, then it is useful that every user uses the same settings for profiles, drawings, templates, etc.

Therefore it is useful to place the `ts` folder on a server disc, so that every user uses the settings from that location.

This increases uniformity and cuts maintenance time because only one folder needs to be maintained. Also upgrading towards a new Tekla Structures version is much easier.

We recommend to create separate ts folders per Tekla Structures version, because some settings are different per version, e.g. when new fields of User Defined Attributes are added.

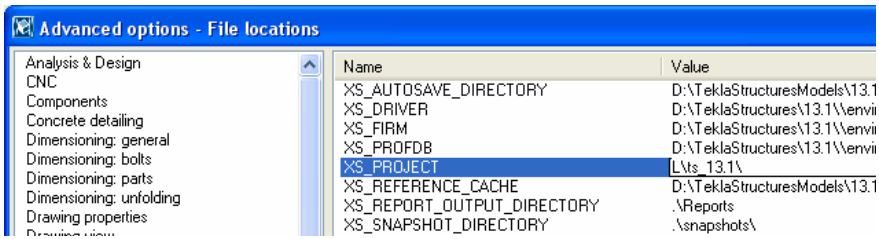
Summary, don't use 10.2 settings in the 13.1 version, but create new settings in 13.1 and put them in a new ts folder:



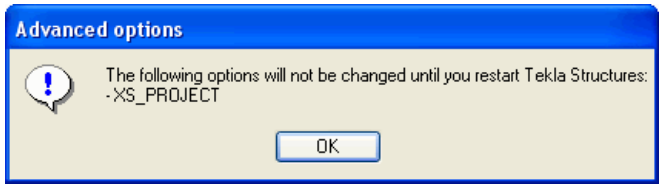
How to set-up a reference to the workstations?

Step plan

- Go to **Tools** > **Advanced options...** and select the **File locations** category
- On the right side, click XS_PROJECT
- Change the value to, e.g. `L:\ts_13.1\`



- Click <**Apply**>, the next dialog shall appear:



- In dialog Advanced Options, click <**OK**>
- Now, open the model folder en double click the file **options.ini**
- Copy the line `XS_PROJECT=L:\ts_13.1\`

- Open the file **user.ini** (located in folder **..\TeklaStructures\13.1\nt\bin)**
- Paste the line in file **user.ini**
- Save the file **user.ini** and restart Tekla Structures.

Autosave

We recommend to save the autosave files (set by variable XS_AUTOSAVE_DIRECTORY in **Tools** > **Advanced options > Files locations**) to a local disc, and **not** to a server disc. This decreases the network traffic.

Pro's and Con's:

Settings (files from the folders `system`, `ts` and `profil`) are stored in the cache memory. The advantage is that all settings have to be loaded just once. Especially if all data is stored on a server, or when running in multi-user mode.

There is also one disadvantage: speed.

Tekla Structures loads the settings, but this takes more time because reading from a network location is about 5 times slower then reading from a local disc.

3.28 Personal and other user settings

Topics

This chapter is divided into following sections:

Personal settings (p.324)

Exceptions (p.326)

Customer depending settings (p.326)

While installing Tekla Structures, general settings will be installed. Also specific settings will be installed, they are related to the module you use (e.g. steel or concrete).

General settings

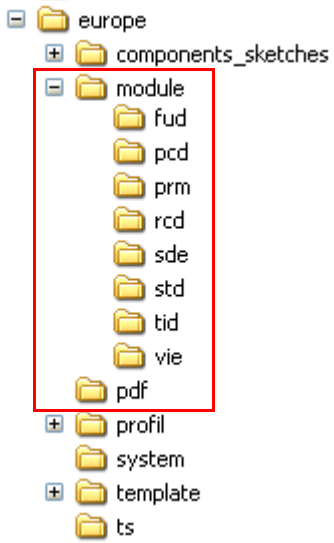
General settings are stored in the folder `...europe\system` and in the folder `...europe\profil`.

Specific settings

Specific settings for customers, working with the module STD for example, are stored in the folder `...europe\module\std`.

So this depends on the software configuration you use, the following modules can be used:

- Full Detailling (FUD)
- Precast Concrete Detailling (PCD)
- Project Management (PRM)
- Reinforced Concrete Detailing (RCD)
- Standard Design (SDE)
- Steel Detailing (STD)
- Timber Detailing (TID)
- Viewer (VIE)



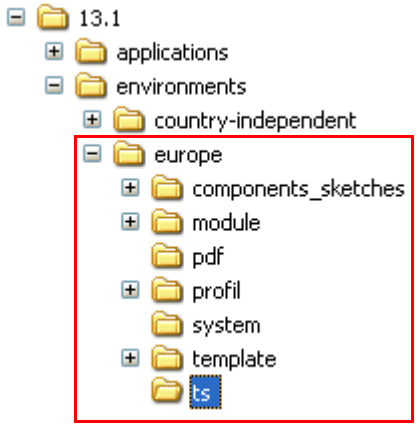
All folders, except the folder `ts` are updated by Construsoft per version.



Our advise is not to change the settings in these folders to make sure that you always use the most recent settings.

Personal settings

The folder `ts` is located in `...environments\europe`:



Of all folders indicated above, the folder `ts` is the one with the highest priority.

For instance, when there is a profile catalog (`profdb.bin`) stored in the folder `...europe\profil`, and there is stored a profile catalog in the folder `...europe\ts`, the second one will be used while modelling.

Files in the model folder or in the folder `attributes` in the model folder do have an even higher priority.

In the folder `ts` you store personal settings, for instance drawing settings (`*.wd`, `*.ad` and `*.gd` files), the profile-, material-, and plotter database (`profdb.bin`, `matdb.bin` and `plotdev.bin`), table layout files (`*.lay`) and templates and reports. (`*.tpl` and `*.rpt` files).

Also project- and numbering properties, “save default” files, component-settings (`*.j*`) and profile properties (`*.prt` and `*.clm`) can be stored in the folder `ts`.

While installing an new Tekla Structures version, the contents of the folder `ts` will be copied to the `ts` folder of the new version automatically.

There is one file that you cannot just store in the folder `ts`:

The file dim_planes_table .txt

To store the file `dim_planes_table.txt` in the folder `ts`, you need to modify the **Advanced options**.

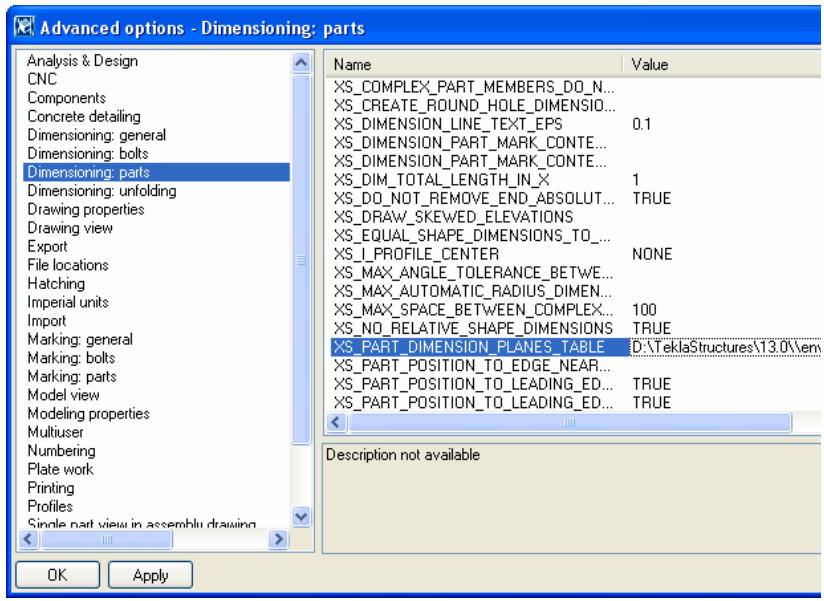
The reference side of dimensions for profiles is controlled by the file `dim_planes_table.txt`.

See also

For more information about the reference side of dimensions, see **Dimensioning Reference side** (p.128).

Step plan

1. Go to **Tools** > **Advanced options...**
2. Select the category **Dimensioning: part**s on the left pane



3. On the right pane, select the variable XS_PART_DIMENSION_PLANES_TABLE
4. Modify in the field **Value** the path to the folder `ts`, e.g. *disc:\TeklaStructures\version\\environments\europe\profil\\dim_planes_table.txt*
5. Click <**Apply**>, the following dialog box appears:



6. Click <**OK**> and click <**OK**> in the dialog box Advanced options
7. Copy the modified file `dim_planes_table` to the folder `ts`.

See also For more information, see **Ini files** (p.311).

Exceptions

Identical templates and reports

Templates and reports, stored in the folder `ts`, have precedence compared to templates with identical names, stored in the folder `..\europe\template` and report with identical names, stored in the folder `..europe\module\std` or for instance `..\module\sde`, depending on the software configuration that is used.



These templates must be created with the new Template editor.

Text files

Text files (**.txt*) stored in the folder `ts` have no precedence to text files in the folder `...europe\system` and the folder `...europe\profil`.

Symbols

Symbols are stored as `*.sym` files and located in the folder `..environments\country-independent\symbols`. Modified and home-made symbol files must be stored in this folder. If you want to use them in a new version, you have to copy them by yourself.

Customer depending settings

See also For information about using customer depending settings, see **Working for multiple contractors** (p.315).

3.29 Parametric modeling

Topics

This chapter is divided into following sections:

Making construction lines magnetic in the model (p.327)

Bind objects to planes (p.328)

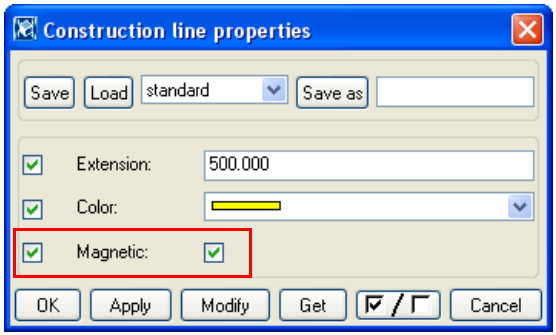
Binding multiple handles to a plane (p.328)

Construction planes (p.331)

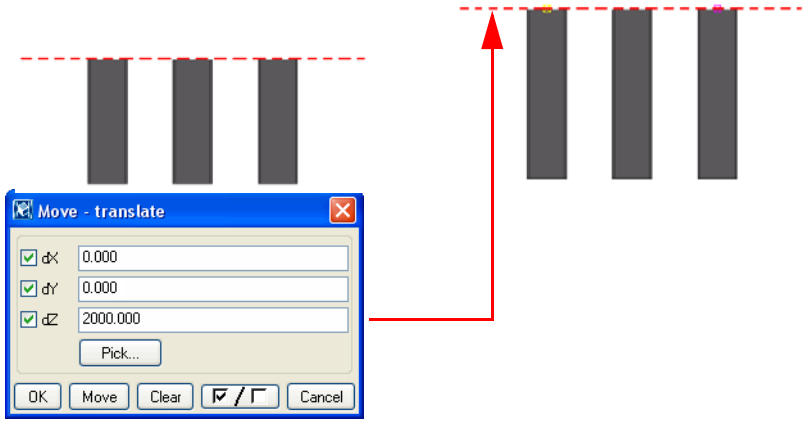
Making construction lines magnetic in the model



You can make construction lines magnetic. To make a construction line magnetic, open the **Construction line properties** dialog box and select the **Magnetic** checkbox:

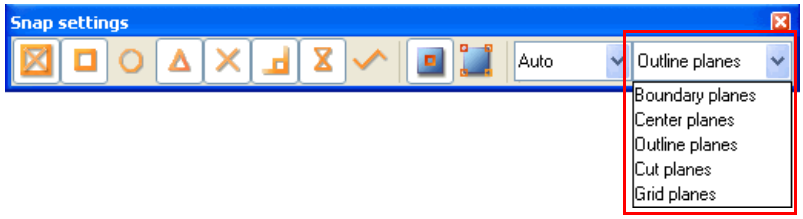


When the construction line is magnetic, the profiles' handles on the line moves with the construction line. In the following example, only the construction line moves, the columns' handles on the line automatically move with the construction line:



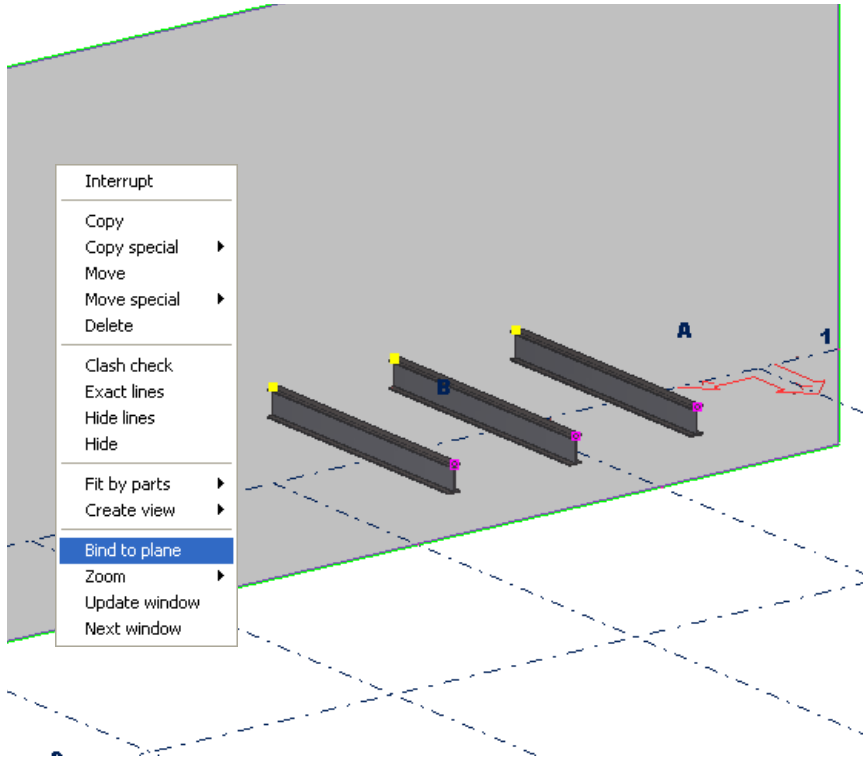
Bind objects to planes

When you bind model objects to planes by selecting the profiles' handle and using the command **Right mouse button** > **Bind to plane**, you can select in the second list box in the toolbar **Snap settings** to which type of plane you want to bind to.

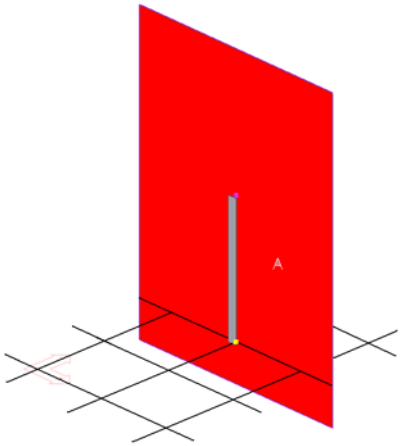


Binding multiple handles to a plane

A major improvement in creating and editing custom components (and parametric modeling in general), is the possibility to bind multiple handles to a plane at once. This makes it more easy to define planes.

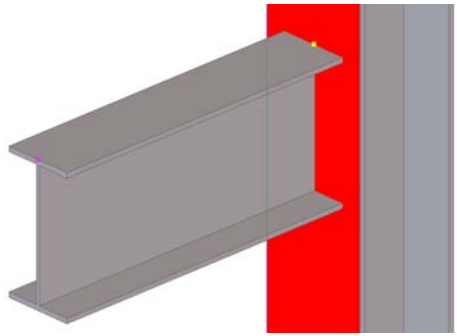


Examples of planes



Grid planes

Outline planes



When the grid changes, for example, the binded handle will move with the grid plane.

See also For more information about plane types, see manual *Custom components*.

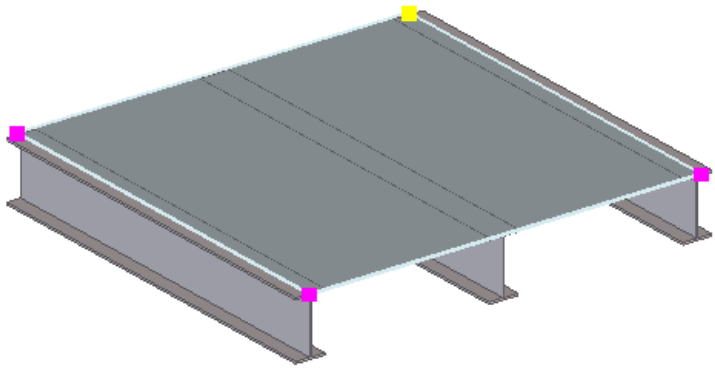
Construction planes



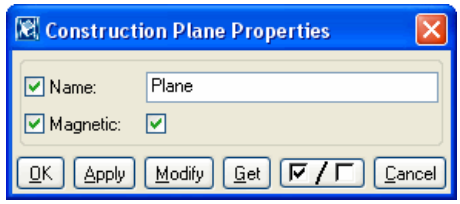
This command creates a construction plane. When the construction plane is magnetic, the profiles' handles on the plane move with the construction plane.

To add a construction plane:

1. Click the **Create construction plane** icon
2. Pick four points
3. Click the middle mouse button. The plane will be created by Tekla Structures:



4. Double-click the plane in the model. The dialog box **Construction Plane properties** appears:



5. Enter a name for the plane (not necessary).
6. Select **Magnetic**, if you want Tekla Structures to bind touching objects to the plane. When you move a magnetic plane, objects directly on that plane automatically move with it.

3.30 Preliminary marks

Topics

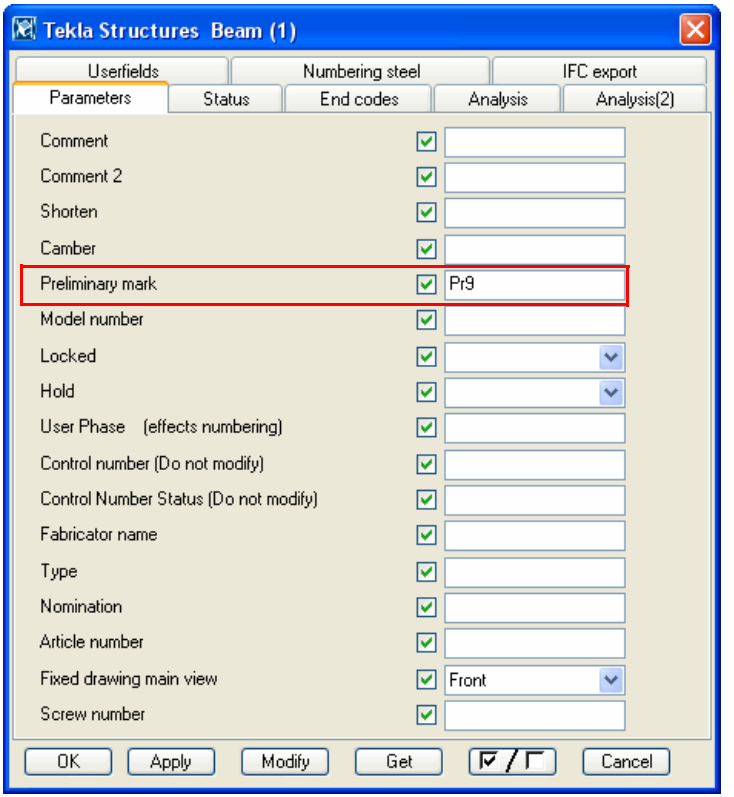
This chapter is divided into following sections:

Example (p.332)

Note (p.333)



To assign part position numbers to preliminary marks of selected parts, click **Tools** > **Numbering** > **Preliminary marks**. The preliminary marks are stored in the parts' user defined attributes.



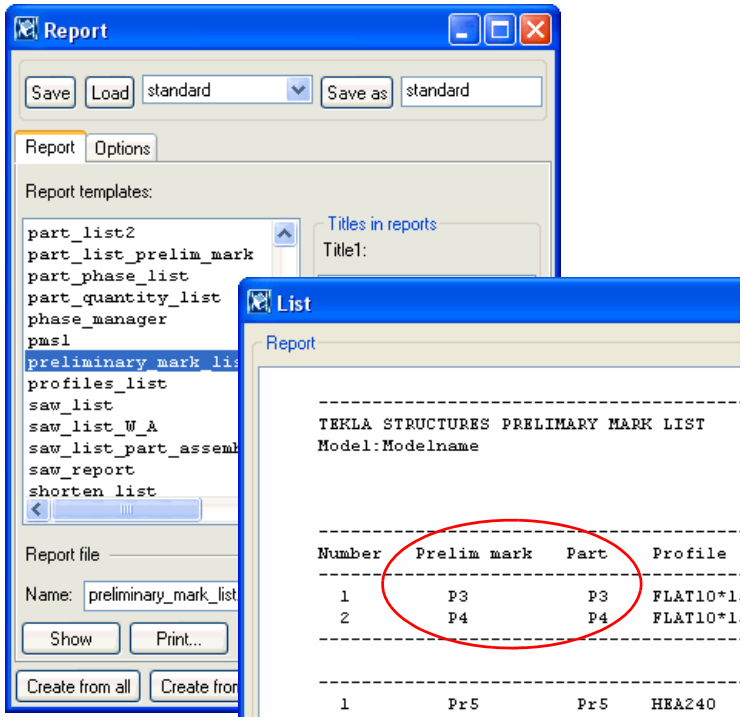
The main reason for the use of preliminary marks is to have a continuous reference to the numbers of the ordered materials.

Example

A user has a new order and all cutted steel profiles have been supplied.

The order will be carried out as follows:

1. A model containing all the main profiles has been created: beams, columns, windbracings, etc. with no connections (possibly).
2. Go to **Tools** > **Numbering** > **Full**
3. Select parts to assign a preliminary mark to.
4. To obtain preliminary marks, click **Tools** > **Numbering** > **Preliminary mark**. The part position numbers of the selected parts are stored in the parts user defined attributes.
5. Now generate a Bill of materials (report: ***preliminary mark list***) containing all main profiles which have to be ordered.



The order can be fully detailed en the drawings can be generated. At this point the drawing numbers can be connected to the numbers of the ordered profiles. Possibly the temporarily numbers on the drawing or reports can be printed.

Note

Some companies use unique numbers for all profiles, even if the profiles are equal. In that case, set variabele XS_UNIQUE_NUMMERS to TRUE in **Tools** > **Advanced options** > **Numbering.**

3.31 Profile catalog

Topics

This chapter is divided into following sections:

Adding profiles using defining cross sections (p.334)

Adding cross section using a DWG file (p.341)

Adding a cross section by using a profile from a plate (p.343)

Exporting and importing profiles (p.346)

In part 1 of the Base training manual (Modeling), something is explained about the profile catalog. In this chapter the profile catalog will be examined further.

Adding profiles using defining cross sections

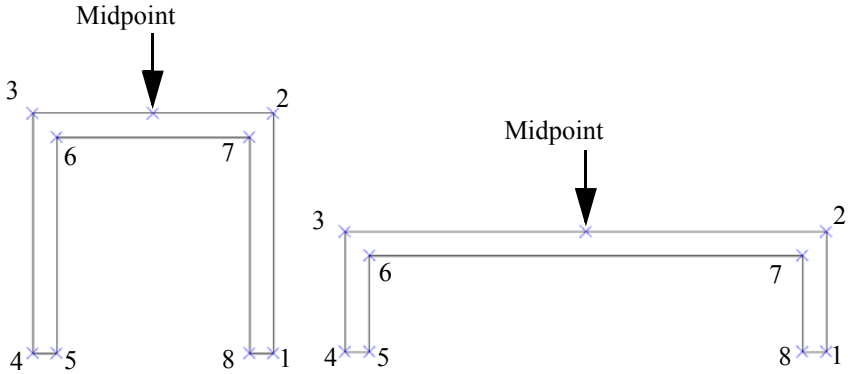
Profiles can be added to the catalog by defining cross sections. Example: a spiral stairs step represented below has to be added to the catalog.



Step plan

1. In the **Foundation plan view** or **Top view**, create a section for both profile ends using a folded plate profile
2. Place a point on all corners (see the picture below), place also point for the midpoint of the cross section
3. Go to **File > Catalogs > Profiles > Define Cross section**
4. Now pick the polygon points, use the order 1-2-3-4-5-6-7-8-1 (always start at the right-bottom corner and pick the points counterclockwise)
5. Next click the middle mouse button and pick a center point
6. Enter a name for the new created cross section, e.g. STEP1

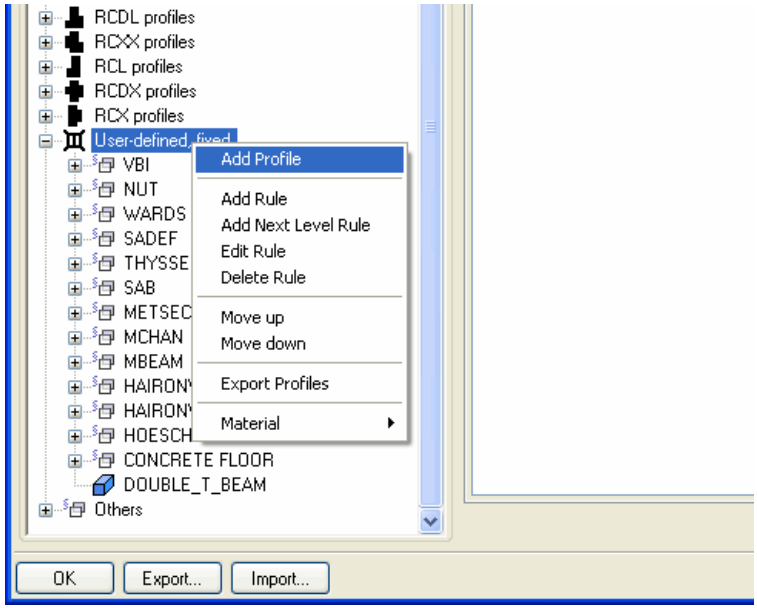
7. Do the same for the other profile (name the cross section STEP2).



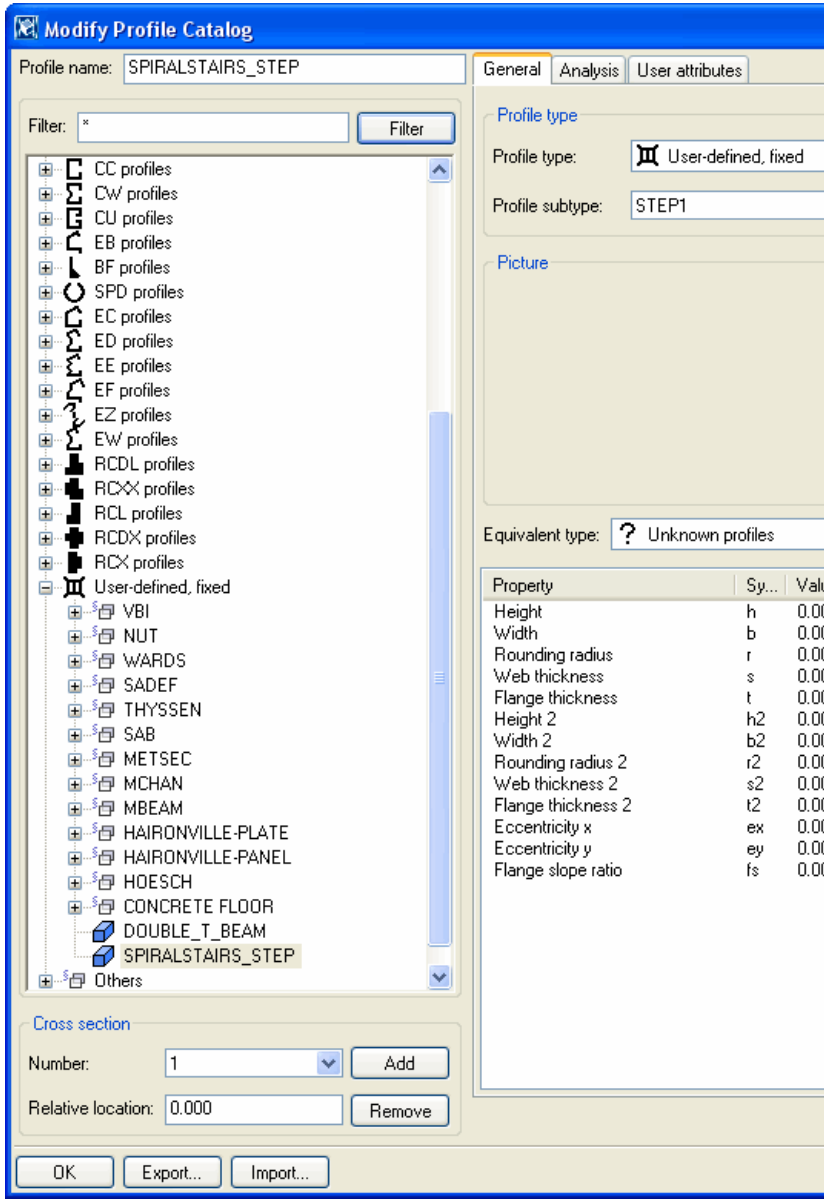
Both cross sections are created now. Now the step profile can be created.

8. Go to **File > Catalogs > Profiles > Modify**

9. Select the option **User defined, fixed**, click right mouse and select **Add profile**:

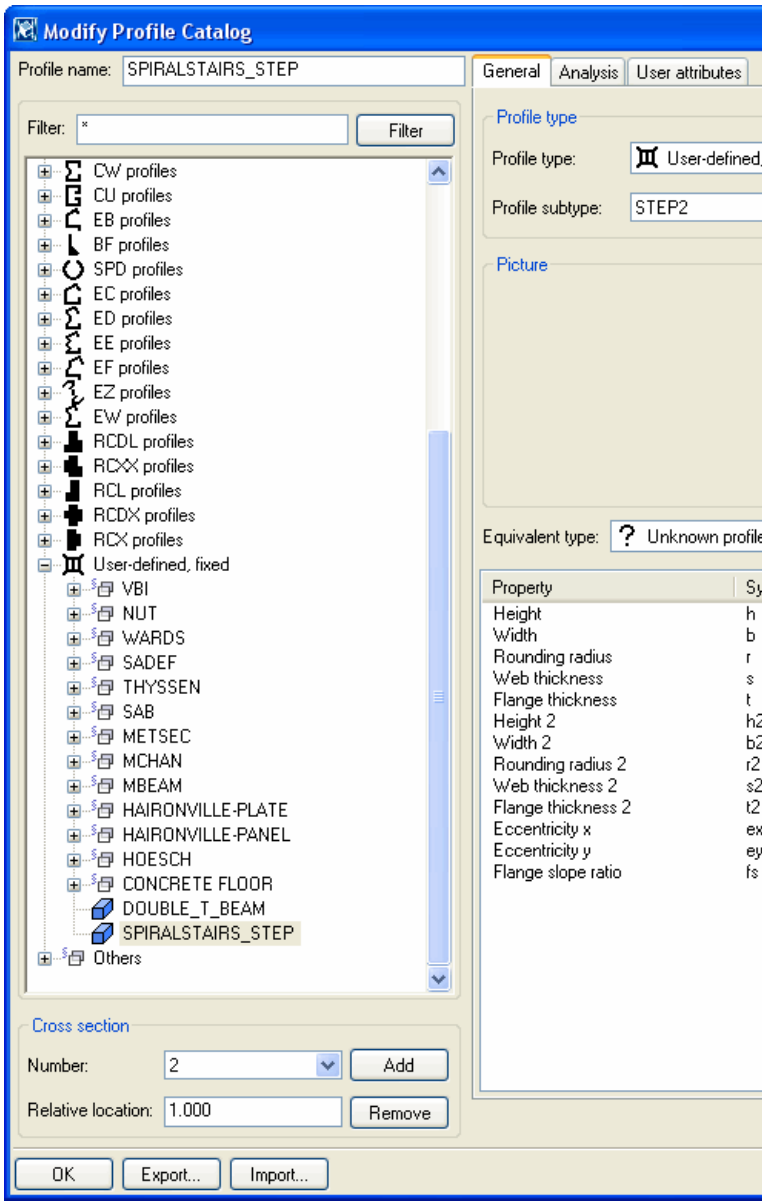


The new profile is added with the name PROFILE1 to the database. Select PROFILE1 to change the name into SPIRALSTAIRS_STEP. From the **Profile subtype** list box, select STEP1:

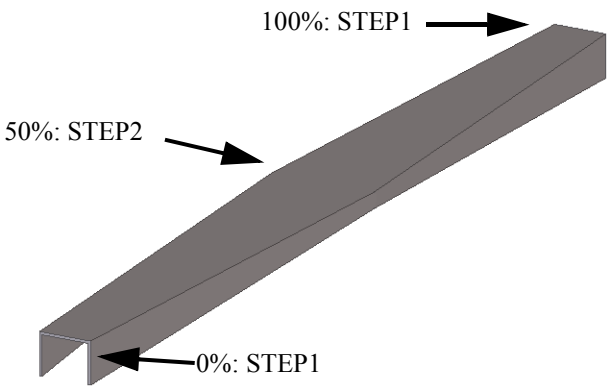


10. Click <**Update**>
11. Now add the second cross section to the profile, click <**Add**>
12. The list box **Cross section Number** now has two options. The Relative location is set to 1.00. This means that on 100% of the profile length, cross section no.2 is displayed

13. For list box **Profile subtype**, select STEP2. Next, click <**Update**> and close the dialog box by pressing <**OK**>



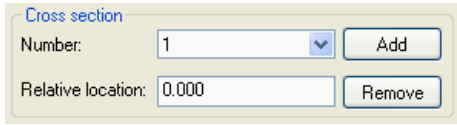
It is also possible to create profiles which are build from several (i.e. more than 2) cross sections. Suppose the profile in the previous example should start with STEP1(0%), halfway (50%) the profile cross section STEP2 should be displayed, and the end of the profile (100%) cross section STEP1 again:

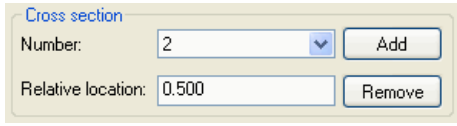


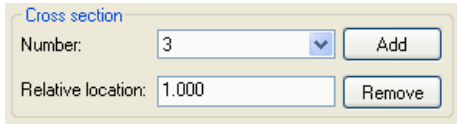
Step plan

1. Click the button <**Add**>, another cross section is added to the profile, the Relative location is set to 1.00 by default
2. At **Profile subtype**, select STEP1 and click <**Update**>

   The only thing left to do is changing cross section 2: the Relative location of this cross section is still set to 1.00!



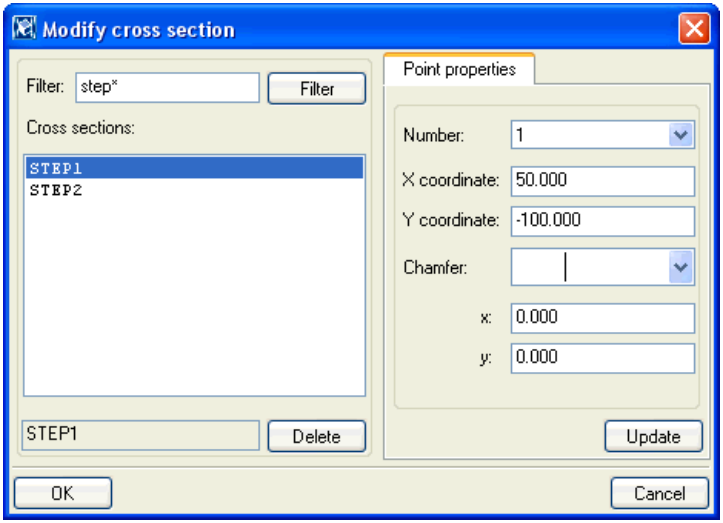




3. Select 2 from the Number list box, change the value for Relative location to 0.50
4. Click <**Update**> and <**OK**>.

When defining cross sections, initially it is only possible to have right angles.

However, afterwards the angles can be edited with command **Modify Cross section**, go to **File > Catalogs > Profiles > Modify Cross section:**



In the left side of the dialog box, the (to be) modified cross section can be selected. On the right side of the dialog box, the vertexes can be edited in tab **Point properties**. From the Chamfer-list box several chamfers can be selected for each vertex. After each modification, click <**Update**> to apply the changes.

For defining cross sections with multiple contours, follow the next procedure:

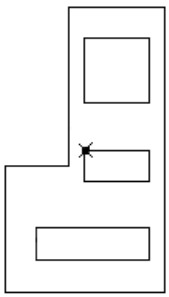
After defining the outer contour, just continue with defining with the inner contour(s), so do not press the middle button at this time. Only if the last contour is defined, press the middle mouse button to abort picking points. Finally, a name has to be entered for the cross section.

Multiple contours within one cross section are separated as follows:

Outer contour: start no. 1

1st Inner contour: start no. 1001

2nd Inner contour: start no. 2001, etc.

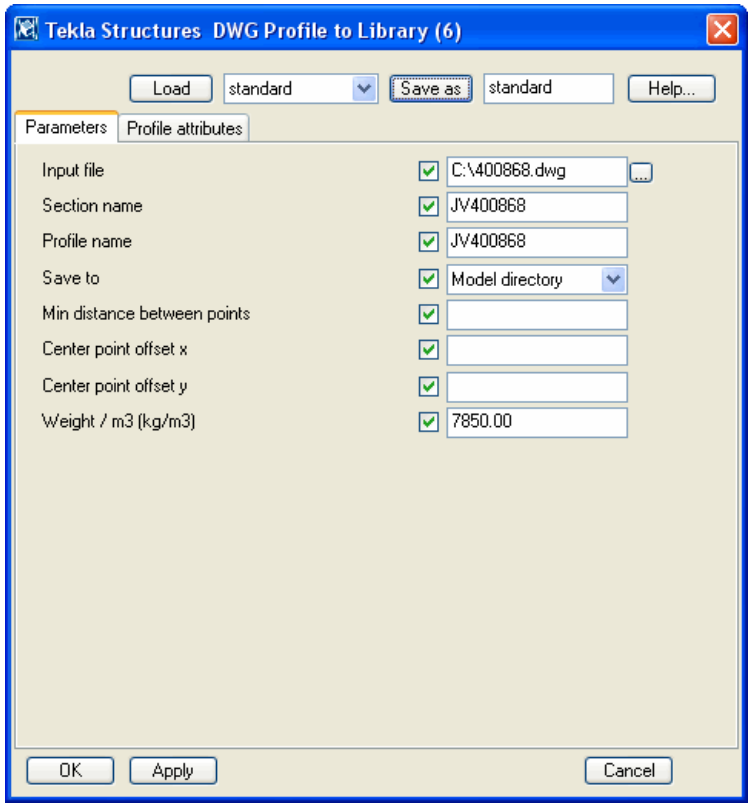


Adding cross section using a DWG file



If a cross section is available in DWG format, you can use this cross section to add it as a DWG-profile to the catalog by using component **DWG Profile to Library (6)**.

It is important that the cross section is created as a closed polyline (p-line). Double-click the component, the next dialog box appears:



Input file Enter the name of the DWG file (including extension)



The path name should not contain to many characters; it is preferred to place DWG file for example on your local hard disk.

Section name Enter a name for the cross section to be stored in the catalog.

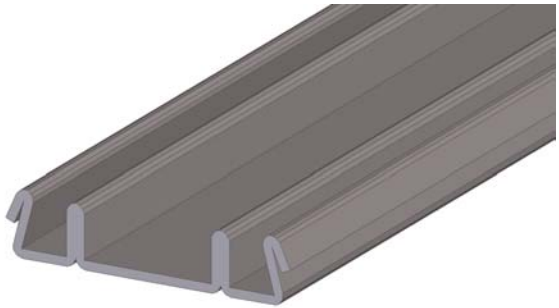
Profile name Enter a name for the profile to be stored in the catalog.

For **Section name** and **Profile name**, capitals have to be used (or a combination with numbers), e.g. **JV407301**!

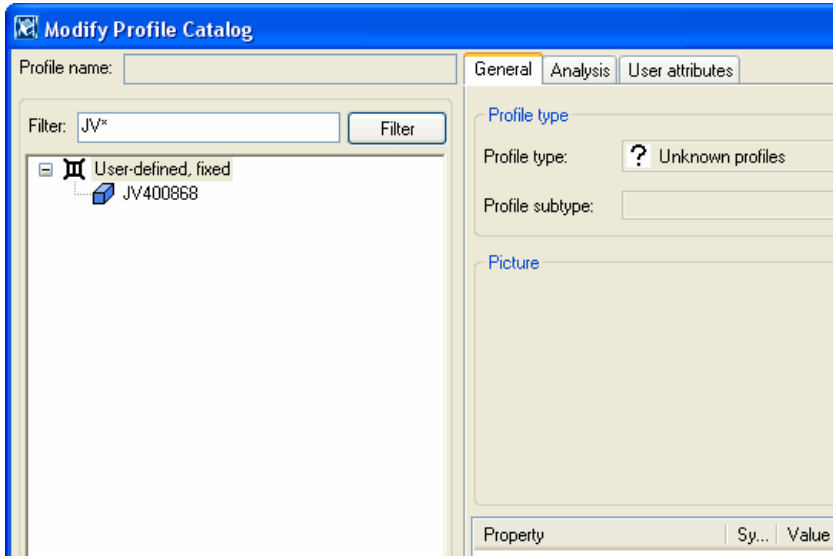
The folder DWG (..*environments**europe**profil**DWG*) contains two *.zip file which includes a huge number of DWG files.

If everything is entered in the dialog box, the component can be executed by using two points in the model

The profile is stored in the file **profdb.bin** in the current model folder.



Click the icon or go to **File > Catalogs > Profiles > Modify** to open the catalog dialog box. The new profile is added to the group **User defined, fixed**:



See also For more information, Zie “Adding a rule” on page 347.

You can also use the DWG file as a reference object in a model and create a contour plate (if this is useful concerning the shape/contour of the profile).

Now make use of component **Profile cross-section from plate (10)** to add this contour plate to the profile catalog, Zie “Adding a cross section by using a profile from a plate” on page 343.

If the shape/contour of the profile is too complicated, you can add the profile using defining cross section, Zie “Adding profiles using defining cross sections” on page 334.

Adding a cross section by using a profile from a plate

With this component, a contour plate shape can be used to add profiles and cross sections to the catalog.

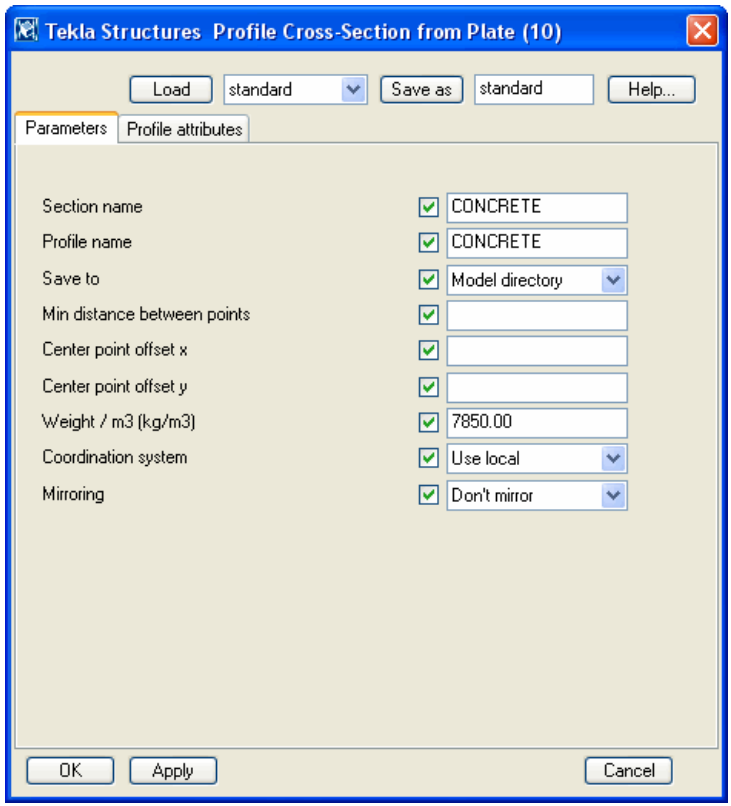
Example

The contour plate below should be added to the profile catalog:



Step plan

1. Double-click component **Profile cross-section from plate (10)** to open the dialog box:



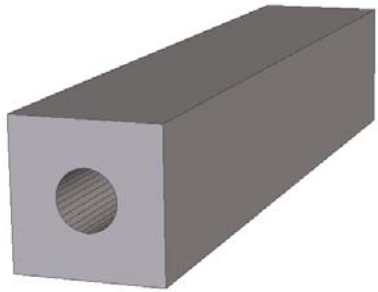
2. For **Section name** and **Profile name**, enter a name, use capitals

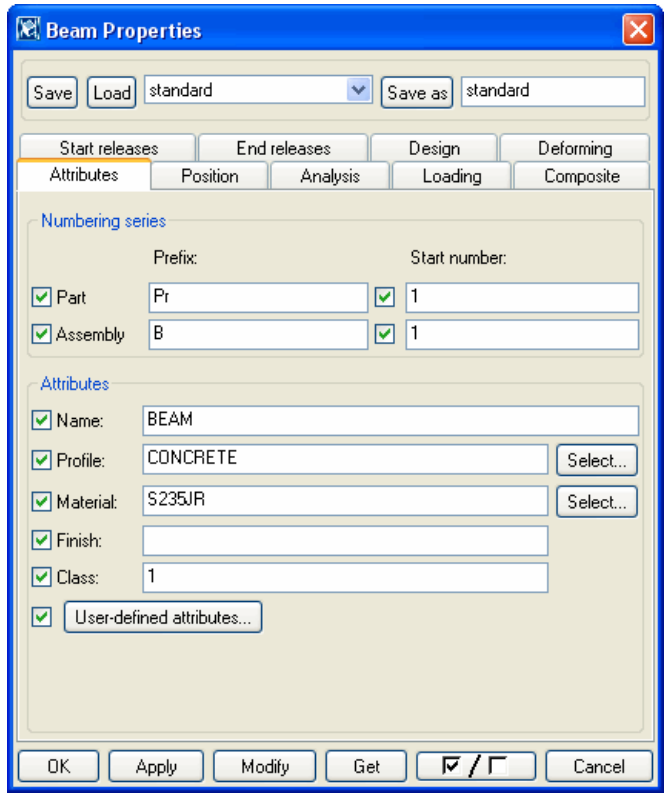
   Possibly, an center point offset can be set (for both x and y-direction) and some options regarding the coordinate system and Mirroring can be defined. In tab **Profile properties**, no changes have to be made.

3. Click **<Apply>** and **<OK>**

4. Pick an object (the contour plate)

   A profile will be created (length 1 meter) with the shape of the contour plate. As the profile is not placed at the correct position, the best thing is to delete the profile and replace it on the correct position.





Exporting and importing profiles

These options work together and are used to merge databases. These are useful commands for:

- Upgrading to a new Tekla Structures version, in case you want to use a modified database from a previous version.
- Combining databases which are stored on different locations.
- Sharing database information with other Tekla Structures users.

Exporting profiles

The <**Export**> button in the dialog box **Modify Profile Catalog** exports profiles in the catalog to an ASCII text file. The file extension is ***.lis** and the file is saved in the model folder.

Example

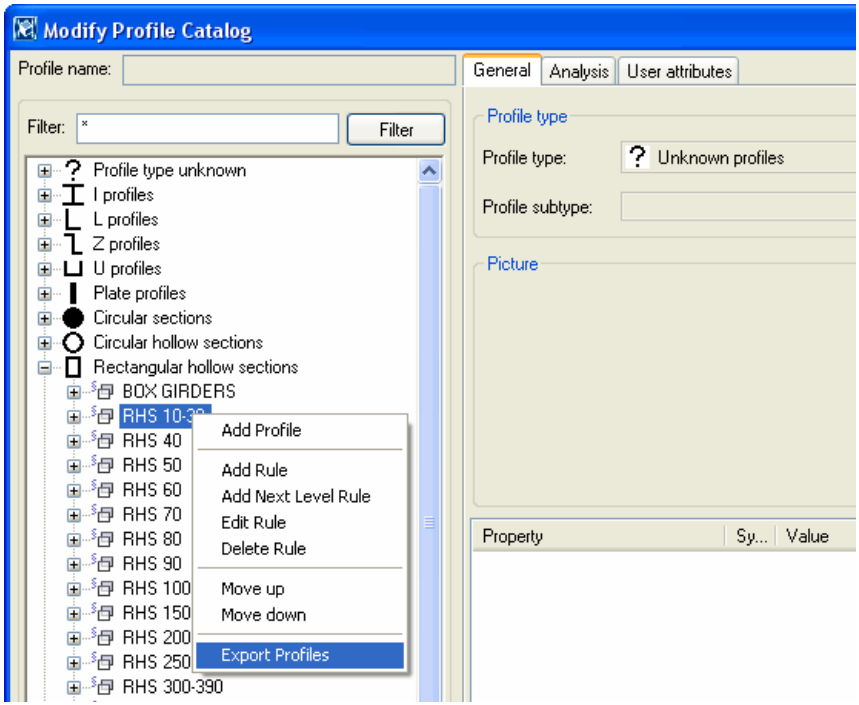
To export the entire contents of the profile catalog:

1. Open the Profile catalog dialog box, go to **File > Catalog > Profiles > Modify**
2. Click the <**Export**> button at the bottom of the dialog box
3. Select a location for the export file
4. Enter a filename, e.g. **exportprofdb.lis**. If you select an existing file, it will be overwritten.



The entire profile database, including user defined cross sections, will be exported with this operation.

Is is also possible to export a single profile, or one ‘tree branch’. To do this, right mouse click on the profile or the branch and select **Export profiles**:



Importing profiles

With this command, an exported profile catalog (***.lis**) can be imported.

Example

1. Open a new model
2. Open dialog box **File > Catalog > Profiles > Modify**
3. Click the <**Import**> button
4. Select the location of the to be imported *.lis file and press <**OK**>

   The profile (or branch) shall be imported.

Adding a rule

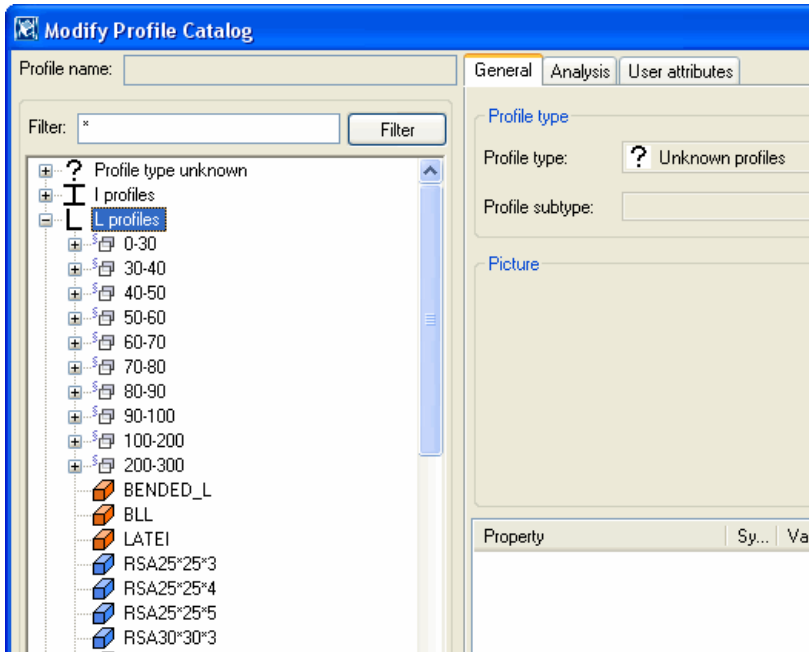
Example

Importing the RSA profiles:

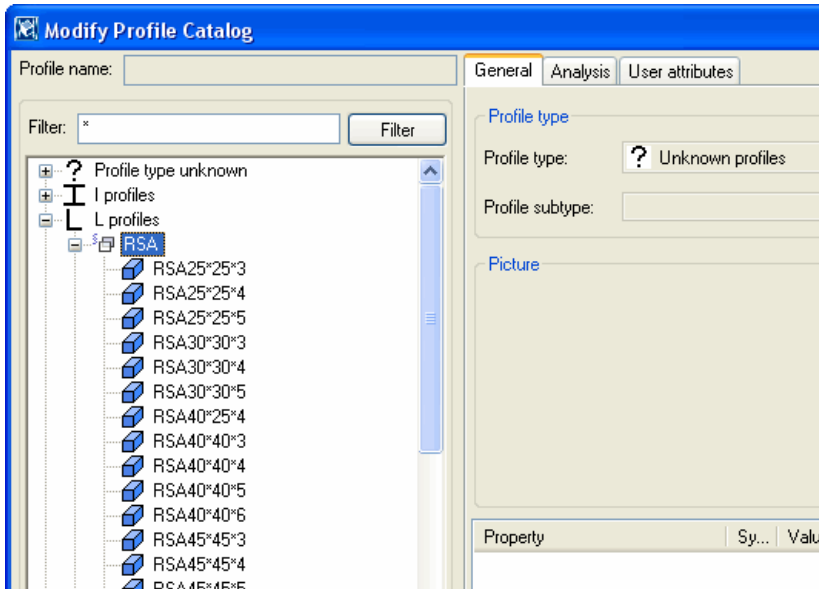
1. Extract the file **lis_UK.zip** (in the folder ..*\environments\europe\profil\lis\UK*)
2. In the model, go to **File > Catalog > Profiles > Modify** to open the dialog box **Modify Profile Catalog**
3. Click the button <**Import**> and browse to the folder where the file **rsa.lis** is stored, and select the file

4. Click <**OK**>

In the catalog tree, the profiles are stored in the folder **User defined, Fixed**:

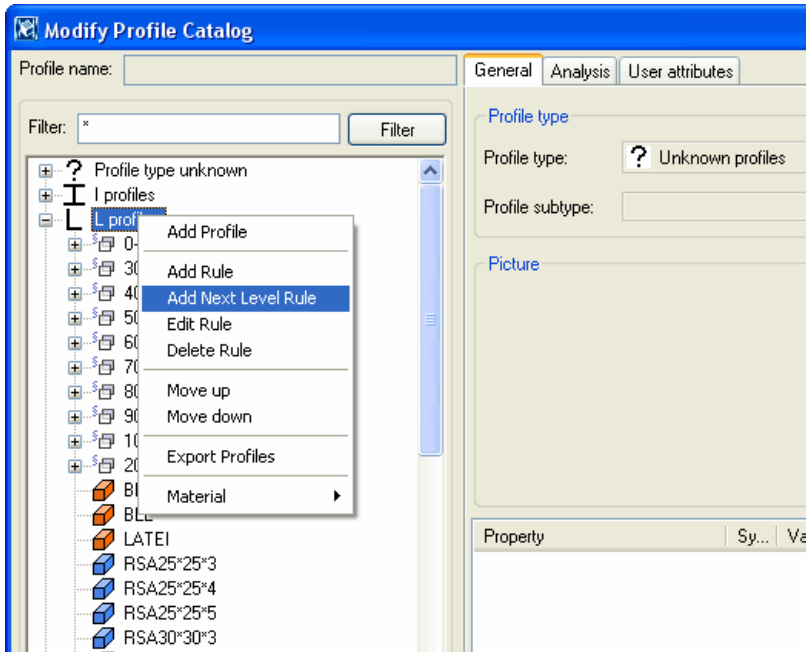


It is advisable to put these files in a separate branch (folder):

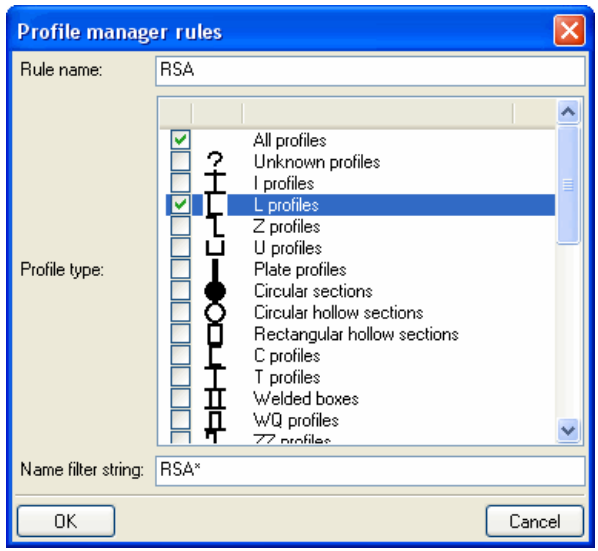


Step plan

1. Click right mouse on the group **L profiles** and select **Add Next level rule** from the pop-up menu (you can also select **Add rule** but then you have to select a top level rule first)



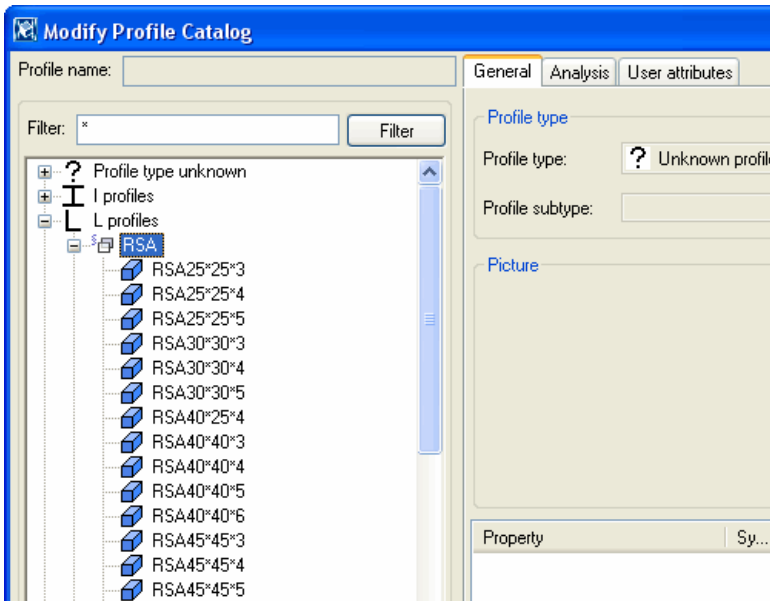
2. The dialog box **Profile manager rules** appears:



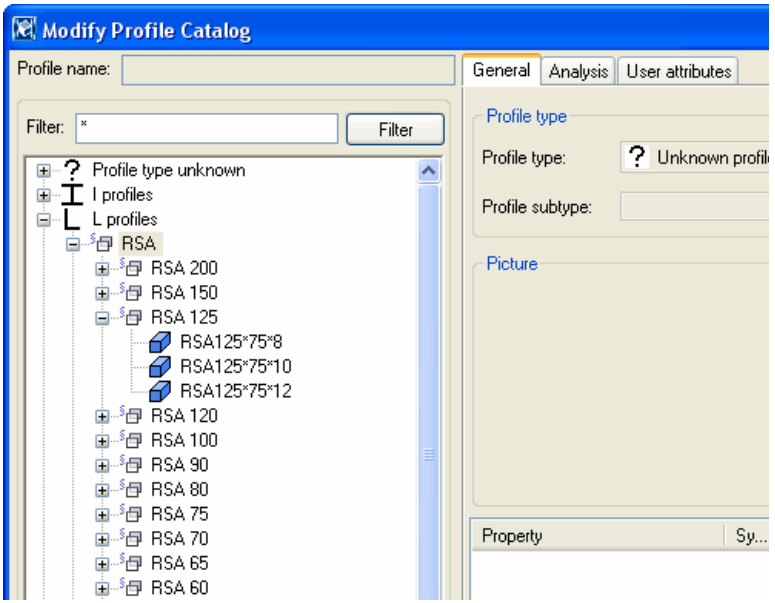
3. For **Rule name**, enter the name of the subdirectory

4. For **Profile type**, select **L profiles** from the list box, because these profiles are based on DWG cross sections
5. By entering **RSA*** for **Name filter string**, only profiles starting with “RSA” are listed in this directory, click <**OK**>

   The RSA profiles are stored in the subdirectory:

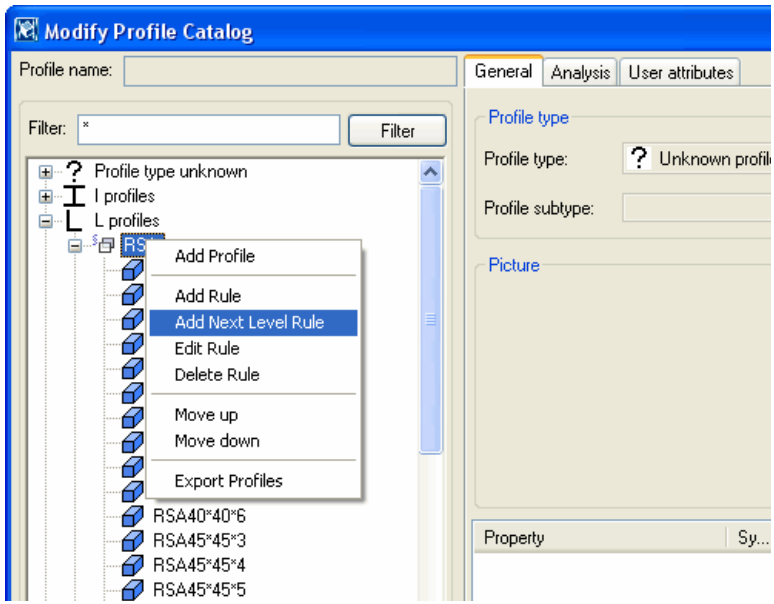


In the folder RSA, you can also create subdirectories:

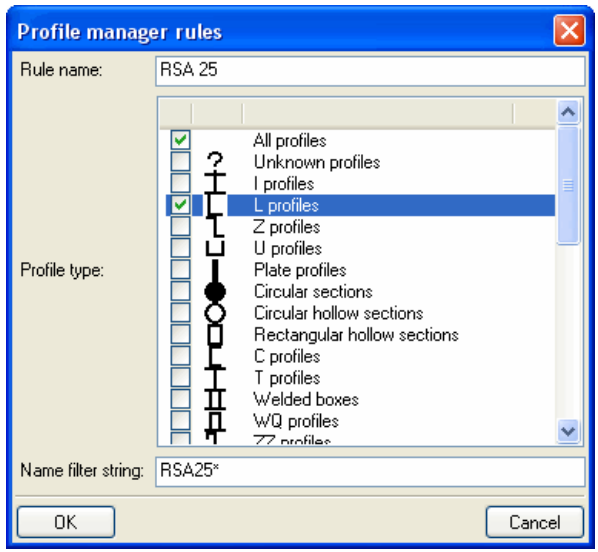


Step plan

1. Select the folder **RSA** and click the right mouse, select the option **Add Next Level Rule**



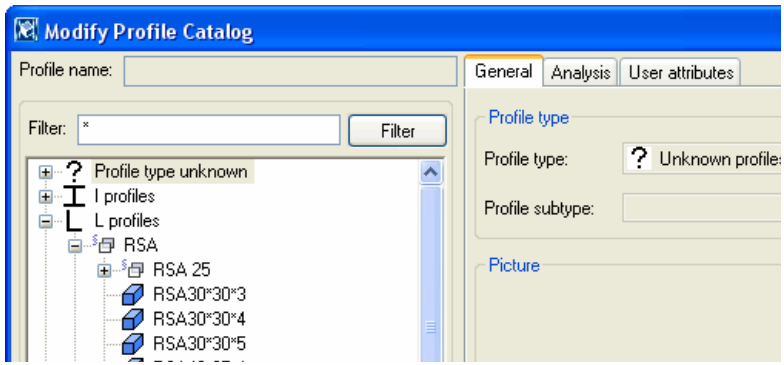
2. The dialog box **Profile manager rules** appears again:



3. For **Rule name**, enter the name of the subdirectory
4. For **Profile type**, select **L profiles** from the list box

5. By entering **RSA 25*** for **Name filter string**, only profiles starting with "RSA25" are listed in this new directory, click <**OK**>

   The RSA profiles are stored in the subdirectory:



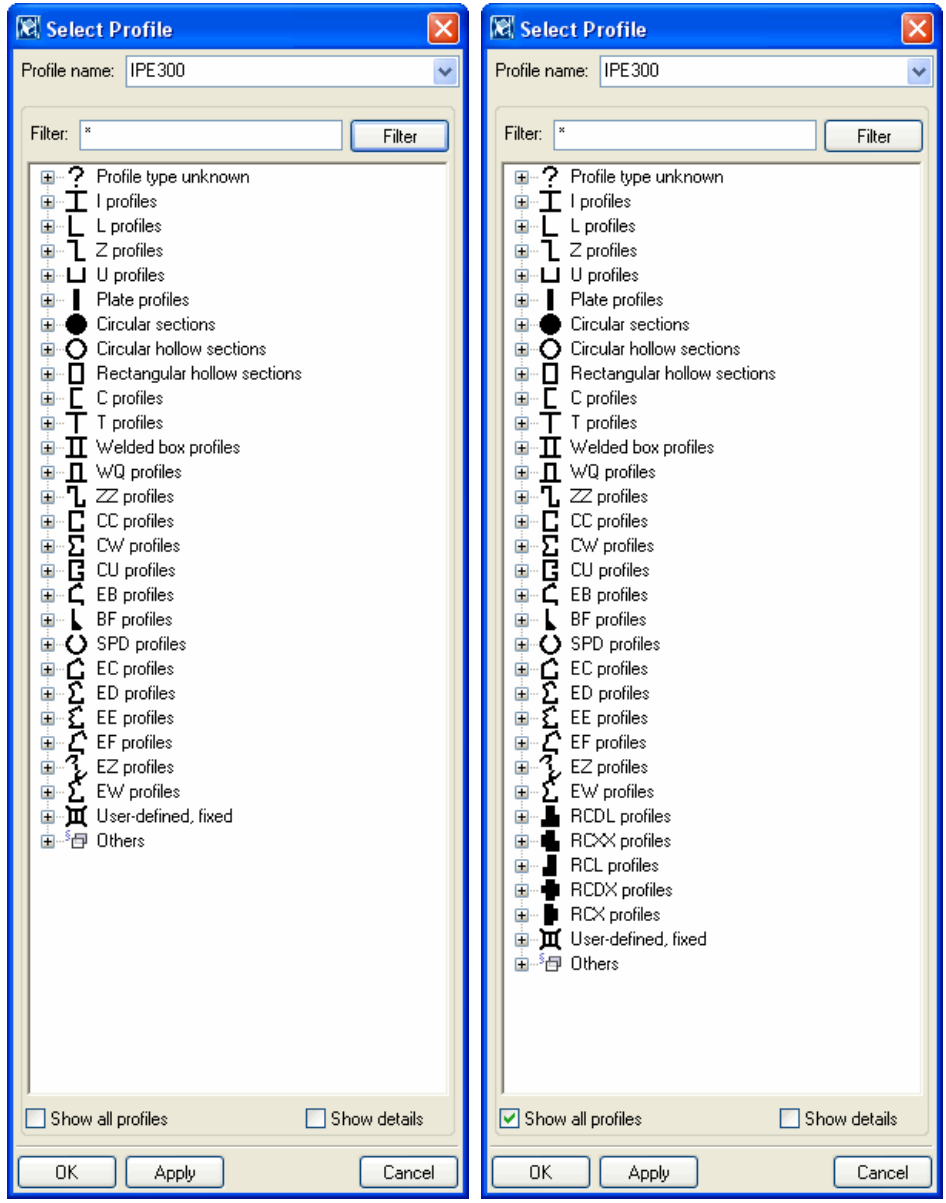
6. Repeat these steps for the other subdirectories

Showing empty rules

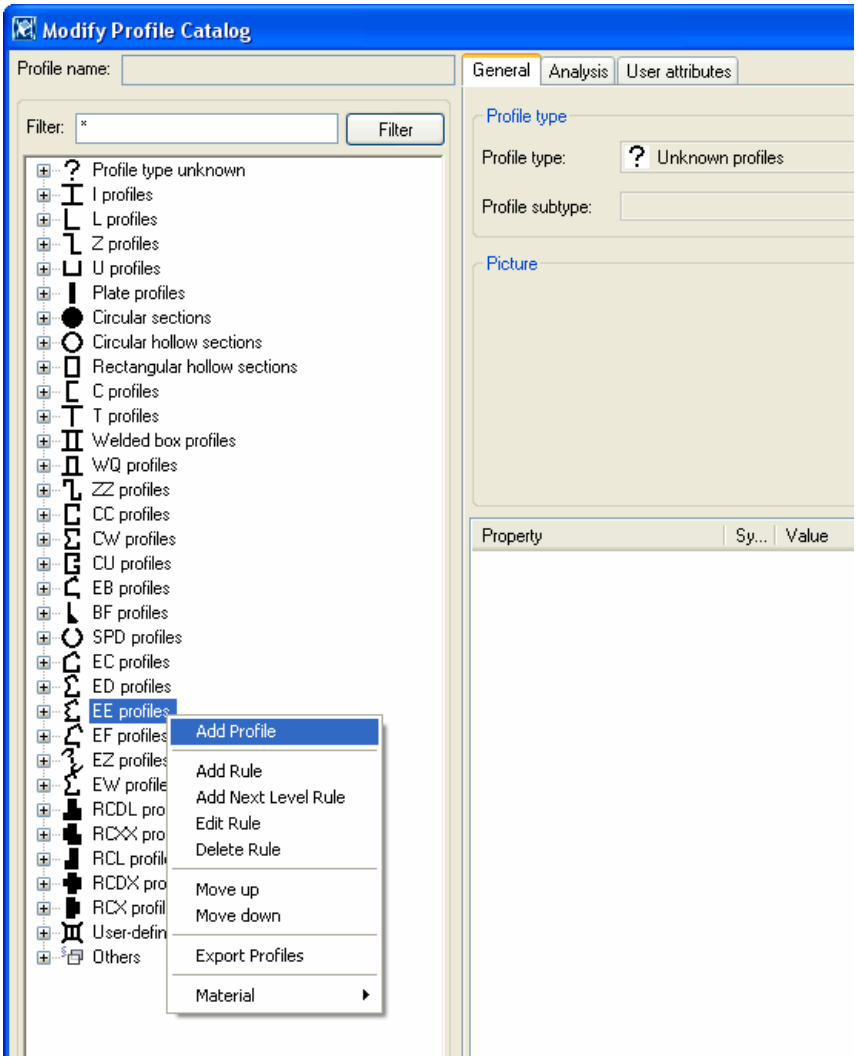
The dialog box **Modify profile catalog** includes a checkbox option for **Show empty rules**.

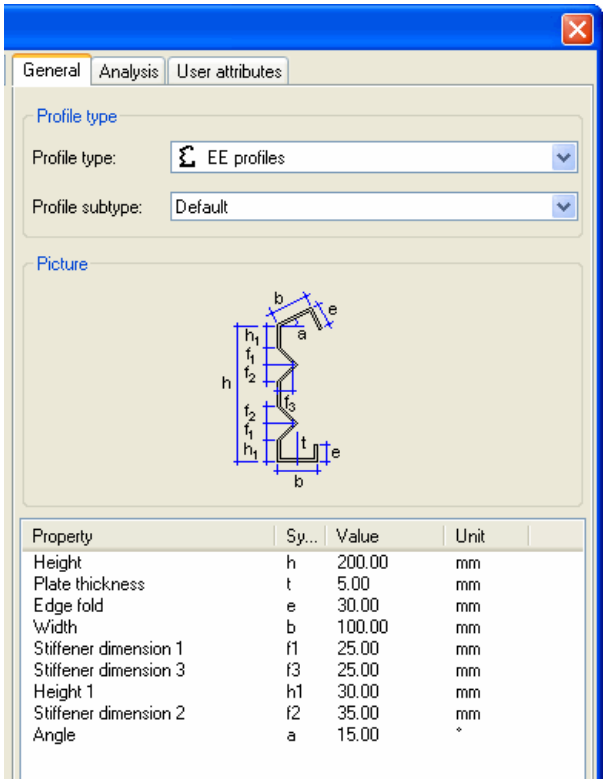
With this checkbox, it is possible to select whether only rules containing profiles have to be displayed, or all rules, even if the rule does not contain any profiles. These rules are created for assistance for future catalog additions, to add the profiles correctly in the tree.

For more clearance, see the picture below.



In case a parametric profile has to be added to the catalog, just right-click a profile shape and select **Add profile** from the context menu, see the next picture:


Modify Profile Catalog
Profile name:
General
Analysis
User attributes
Filter:
*
Filter
Profile type
Profile type:
Unknown profiles
Profile subtype:
Picture
Property
Sy...
Value
Profile type unknown
I profiles
L profiles
Z profiles
U profiles
Plate profiles
Circular sections
Circular hollow sections
Rectangular hollow sections
C profiles
T profiles
Welded box profiles
WQ profiles
ZZ profiles
CC profiles
CW profiles
CU profiles
EB profiles
BF profiles
SPD profiles
EC profiles
ED profiles
EE profiles
EF profiles
EZ profiles
EW profile
RCDL pro
RCXX pro
RCL profil
RCDX pro
RCX profil
User-defin
Others
Add Profile
Add Rule
Add Next Level Rule
Edit Rule
Delete Rule
Move up
Move down
Export Profiles
Material



Just like adding ‘standard’ profiles, just enter a new name for the profile and specify the values for the dimensions.

3.32 Profile length calculation

Topics

This chapter is divided into following sections:

Fitting (p.356)

For length calculation of profiles in templates and reports the variable **LENGTH** is used everywhere.

The benefit of the **LENGTH_GROSS** variable is only perceptible in case of double (or more) bevelled profile ends.

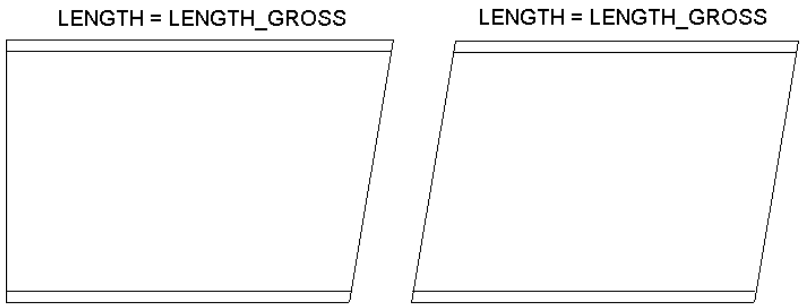
Fitting

At a fitting operation, the profile end is extended/ shortened towards the fitting line. A fitting line may occur once on each profile end. When fitting a profile, the shortest part of the profile will be removed automatically.

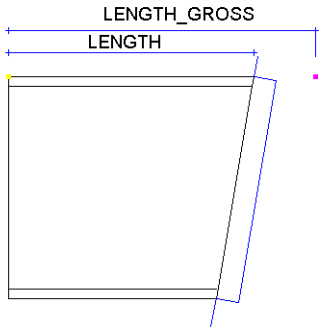


When shorten profiles, always use the *Fit* command. This because only fitting will return correct lengths on reports and in NC data. When the *Line cut* command is used for shortening profiles, these values may be wrong in certain situations.

If a profile-end needs two bevelled edges, use the *Fitting* command for the first bevelled edge (is the shortening operation), use the *Line Cut* command for the second bevelled edge!



Profile shortened with Line Cut command?

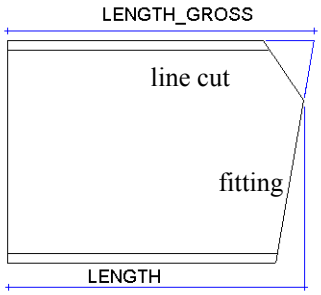


LENGTH=OK

LENGTH_GROSS= distance between the yellow and purple point

LENGTH_GROSS does not take Line Cut operations into account! This also counts for volumes which are created by part cuts or polygon cuts!

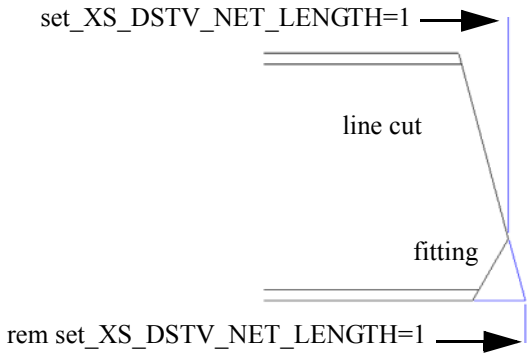
So, double Line Cuts on start/ endpoint returns the length between the profiles initial working points!!!



LENGTH returns the ‘short’ length

LENGTH_GROSS returns the ‘full’ length

See figure below:



Line cuts are not taken into account in NC-data! So the longest length is returned.

3.33 Save and load defaults

Topics

This chapter is divided into following sections:

Save defaults (p.361)

Load defaults (p.361)

A lot of settings, for example component settings, can be saved with a user definable name. These settings can be copied to the **ts** folder to make the settings available in other Tekla Structures projects.

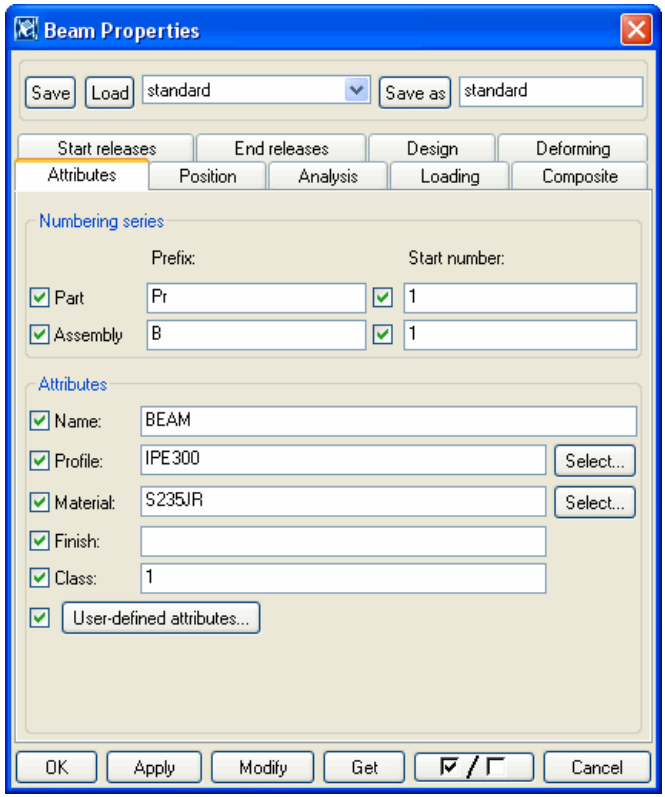
However, some settings can not be saved with a name, but are saved with a default name ‘standard’, with a accompanying file extension. See the table below for the file extensions. If a new Tekla Structures model is started, all standard file mentioned below are loaded into the model. If the **Load defaults** command is executed, the standard files from the **ts** folder are reloaded.

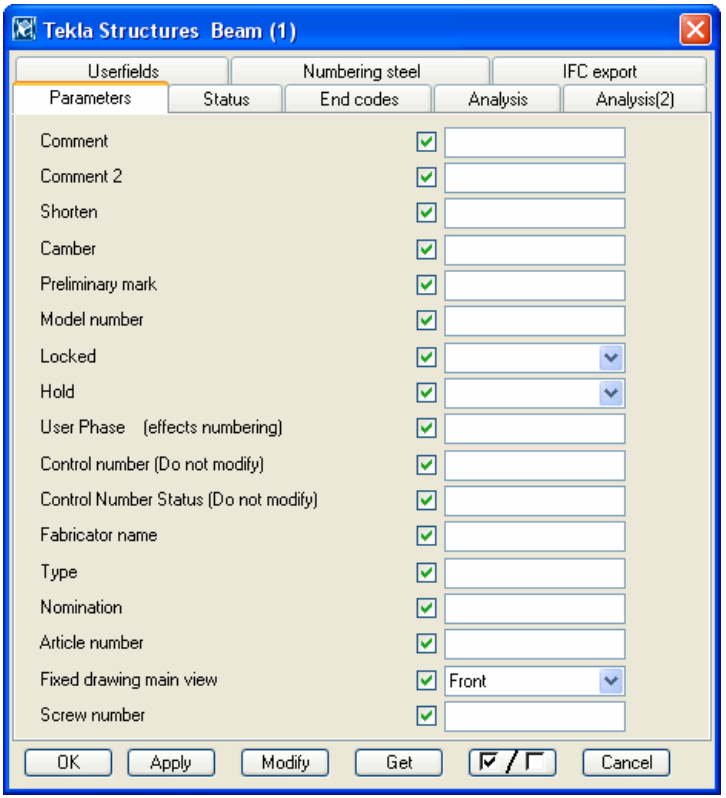
Standard files	File	In Tekla Structures
Column setting	standard.clm	Dialog box **Column properties**
Contour plate setting	standard.cpl	Dialog box **Contour plate properties**
Orthogonal beam setting	standard.crs	Dialog box **Orthogonal beam**
Twin profile setting	standard.dia	Dialog box **Twin profile properties**
Drawing frame properties	standard.fms	In Model and Drawing Editor: **Print Drawings > Frames**
Folded plate setting	standard.fpl	Dialog box **Folded plate properties**
Printer layer setting	standard.ler	In Model Editor: **File > Catalogs > Printers > Modify > Color table**
View setting	standard.mvi	In Model Editor: **Properties > View...**
Numbering setting	standard.num	In Model Editor: **Setup > Numbering...**
Project properties dialog	standard.prf	In Model Editor: **Properties > Project...**

Beam setting	standard.prt	Dialog **Beam properties**
Bout properties setting	standard.scr	Dialog **Bolt properties**
Weld properties setting	standard.wld	Dialog **Weld properties**

Some of the settings above do have an additional file with a extension, for example the Beam properties dialog box:

Standard.prt





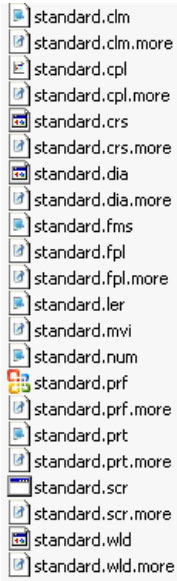
The extension *.*more* is intended for the *User Defined Attributes*.

If the setting file *standard.prt* is copied, the file *standard.prt.more* always has to be copied too!

Save defaults

Step plan

1. Go to *Setup* > **Save defaults**
2. In the Windows Explorer, open the model folder. In the ***Attributes***-folder, several files named ‘standard’ are stored.



3. Copy all setting files (or just the files which you want to use as a ‘default’) to the folder:

   disk:\TeklaStructures\version\environments\europe\ts

4. Restart Tekla Structures.

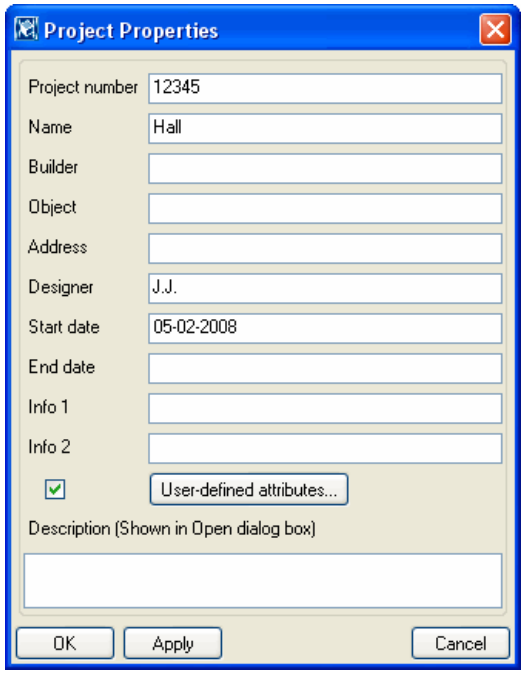
When this is done and a new project is started, the settings in the **ts** folder will be used.

Load defaults

To load a default setting (from the *ts* folder) into an **existing** Tekla Structures model, do the following:

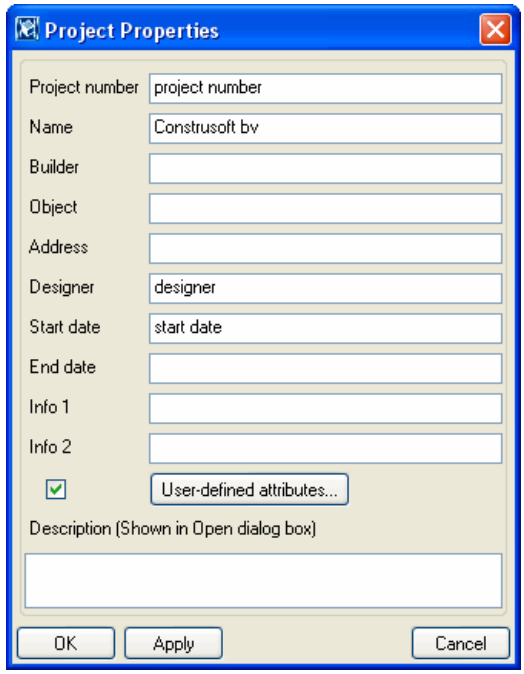
Step plan

1. Go to **Properties > Project Properties**
2. This opend dialog box contains the settings, and maybe they are changed manually.



3. Now go to **Setup > Load defaults**

   You will see the values in the dialog box will change to the default ones:





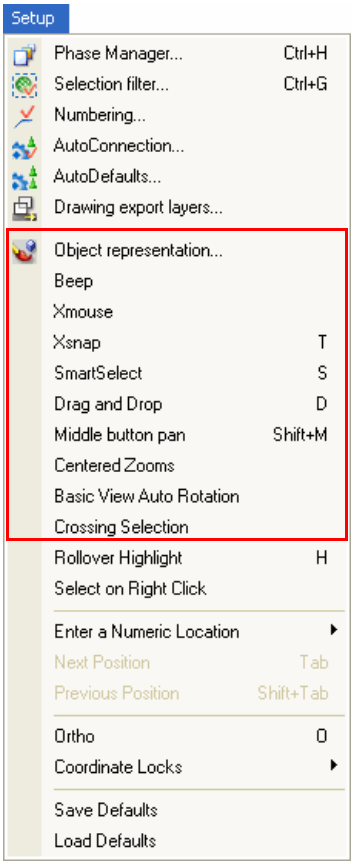
Make sure the dialog box for which you want to execute the Load Defaults command is opened. Otherwise the ‘old’ setting will be used.

3.34 Save Select switches and Snap settings

Some settings in the pull-down menu and the toolbars can be saved. E.g. the settings in **Setup** in the pull-down menu, the active selections in the toolbars **Select switches** and **Snap settings**, the current phase and selection by colors.



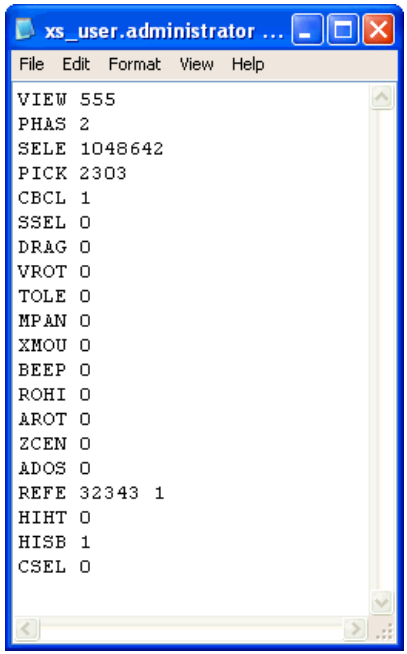




among other things the values of these settings can be saved.

These settings are saved per model in the file **xs_user.username**, so if you are logged on the system as “administrator”, the name of this file is **xs_user.administrator**.

When you close Tekla Structures or save the model, this file will be stored in the model folder. You can open this file using any text editor (e.g. Notepad).



VIEW	id number of each view
PHAS	current phase
SELE	active icons select switches
PICK	active icons snap settings
CBCL	color display
SSEL	smart select
DRAG	drag and drop
VROT	view rotation keyboard
TOLE	xsnap
MPAN	middle button pan
XMOU	xsnap
BEEP	beep
ROHI	rollover highlight
AROT	basic view auto rotation
ZCEN	centred zooms
ADOS	
REFE	reference model
HIHT	hidden lines
HISB	display bolts
CSEL	crossing selection

This file contains the settings. Rename this file to **xs_user.default** in the model folder and copy this file to the system folder. In every new model you now create, these settings will be used.

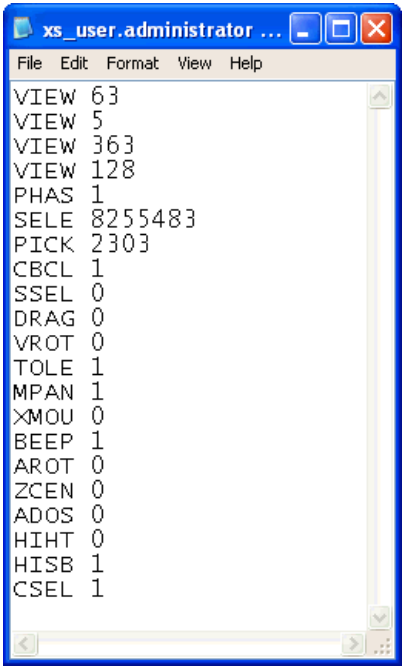
Most values in the file **xs_user.username** are ‘0’ or ‘1’, corresponding to a checkbox you can toggle on or off in the pull-down menu **Setup** menu.

The value ‘0’ means off, ‘1’ means on.

E.g. no autorotation for a part perspective view (in Create view > Part basic views or basic view) you enter a value by AROT. The same for Beep, Middle button pan, Xsnap, etc. Divergent lines are described in the following chapters:

VIEW

To each view in a model an id number is assigned. When closing a model all visible views will be saved in **xs_user.username**. If, for example, four views are visible, the file **xs_user.username** contains four lines starting with VIEW...



PHAS

This line contains the current phase (at the time you close the model).

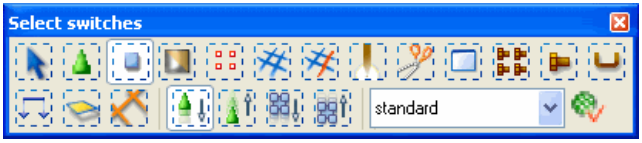
SELE

In this line the toolbar settings **Select switches** are stored. The numeric code behind ‘sele’ represents the number of active icons in this toolbar. See the pictures below.

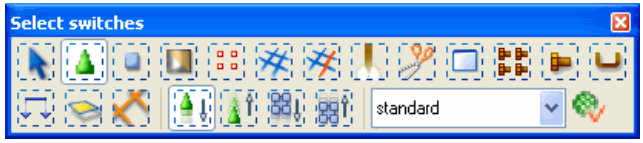
SELE 2095099



SELE 2090107



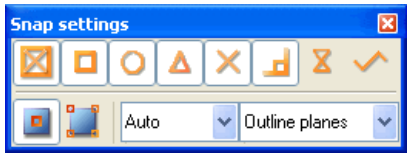
SELE 2095448



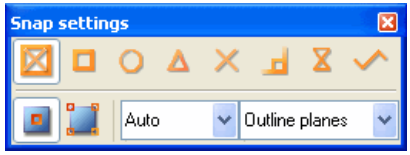
PICK

In this line the toolbar settings **Snap Settings** are stored. The numeric code behind ‘pick’ represents the number of active icons in this toolbar. See the pictures below.

PICK 2055

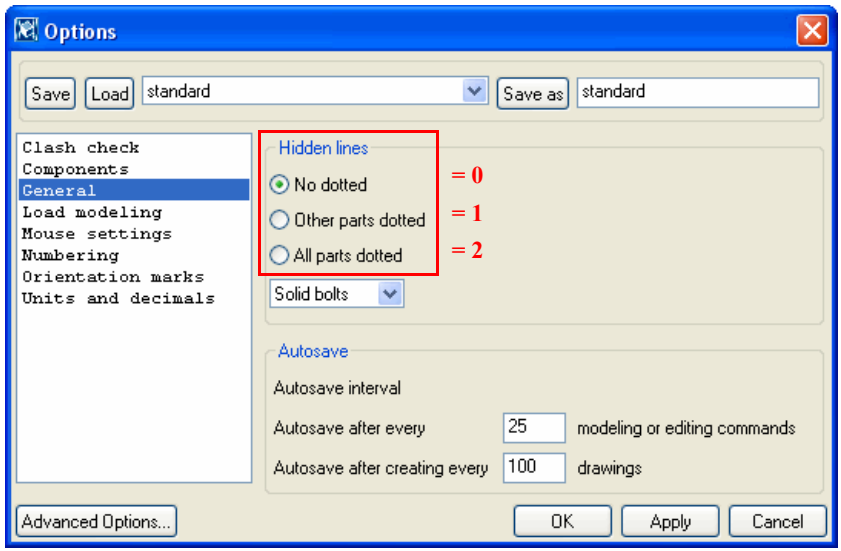


PICK 2303



HIHT

This line represents the value of the setting in **Tools** > **Options > General**



HISB

This line represents the value of the bolts in **Tools** > **Options > General**, see the dialog box above.

0 = Solid bolts.

1 = No bolts.

3.35 Sketching parametric profiles

Topics

This chapter is divided into following sections:

Create your own parametric profile (p.371)

Add dimension constraints to the cross section (p.375)

Modify the parametric profile - add chamfers (p.382)

Adding a picture of a cross section (p.386)

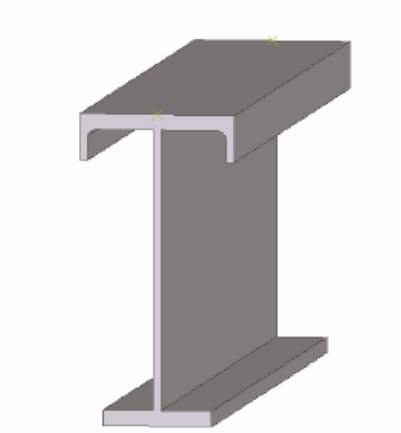
Extruded sketched polyline (p.386)

Part- and connection position planes (p.388)

In this lesson

In this lesson you will learn how to create a new parametric profile with the sketching tool.

After updating the profile catalog we will use the new profile in the model to create the crane beams.





For sketched profiles counts that only the values **H** (height) and **B** (width) are well-known so there are limitations regarding to the associativity and using the profile in custom components and applying system components to this profile.

Sketching tools

Tekla Structures includes the following tools for sketching parametric cross sections:

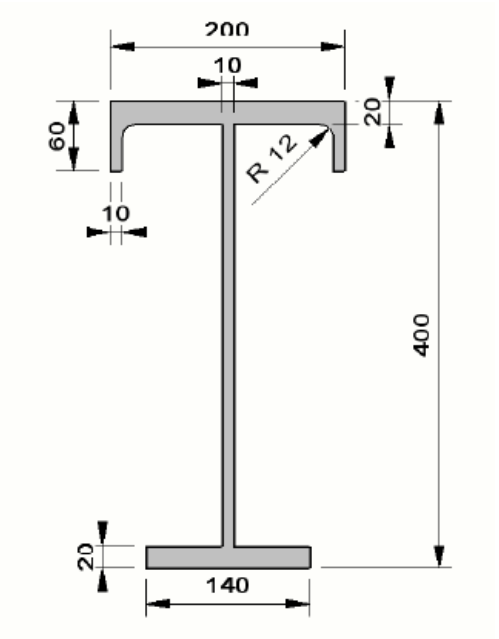
Command	Icon	Description
Sketch polyline		Tools to sketch the shape of the cross section.
Sketch arc		
Sketch circle		
Sketch free distance		Creates a dimension between two points you pick.
Sketch horizontal distance		Creates a horizontal dimension between the points you pick.
Sketch vertical distance		Creates a vertical dimension between the points you pick.
Sketch radial dimension		Creates a radial dimension for the arc or circle you pick.
Sketch angle dimension		Creates an angle dimension.
Parallel constraint		Forces lines to be parallel to each other.
Perpendicular constraint		Forces lines to be perpendicular to each other.
Coincident constraint		Forces two lines to start or end at the same point, by extending or shortening one or both lines.
Fixed constraint		
Horizontal constraint		Forces line to be parallel to the local x axis.
Vertical constraint		Forces line to be parallel to the local y axis.

Command	Icon	Description
Show or hide part positioning planes		To view the positioning planes that control a profile
Show or hide connection positioning planes		To view the positioning planes that control a profile
Display variables		To open the **Variables** dialog box, in which you can view and edit the parameters.
Display sketch browser		To open the **Sketch browser** dialog box, in which you can view and edit the model objects.
Save sketch as		Saves the cross section.
Close sketch		Prompts you to save the sketch. Option to close the sketch without saving

Create your own parametric profile

In this exercise we will define a new cross section to be used for a crane beam.

We will create the crane beam as a parametric profile using the Tekla Structures cross section sketch editor.



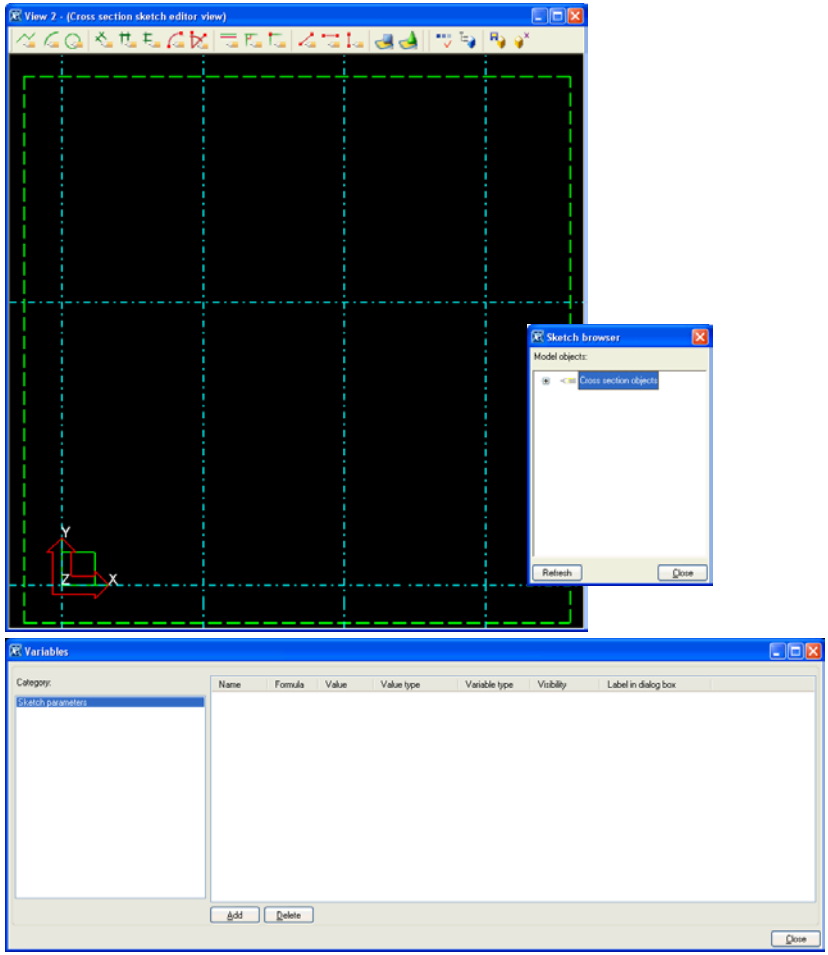
Sketch a cross section

We are going to open the Cross section sketch editor and then sketch a rough outline of the shape shown above. It doesn't matter if the lines are exactly horizontal or vertical, or if they touch at the ends. Neither the dimensions are important at this stage. We will connect the ends of the poly-line and force the lines to be horizontal or vertical later.

Open sketch editor

Select **Sketch parametric cross section** from **File > Catalog > Profiles > Sketch parametric cross section** to open the **Sketch editor**.

The sketch editor opens showing the **Sketch editor** dialog box with toolbar, **Variables** dialog box and **Sketch browser** dialog box.

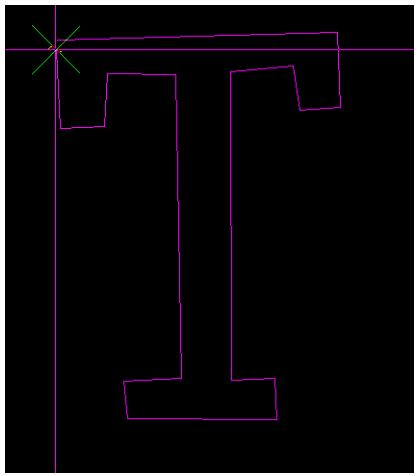


Sketch a cross section

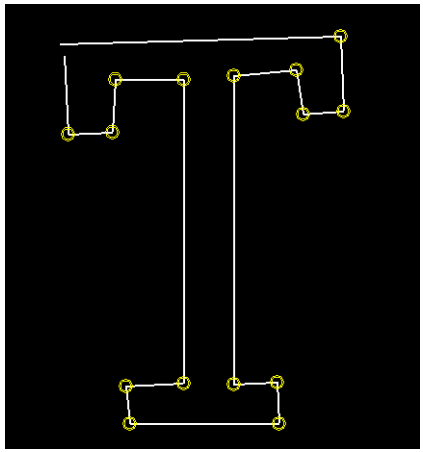
1. Click on the **Sketch polyline** icon



2. Sketch the profile shown below and click the middle mouse button.



The yellow circles represent "chamfers" in the **Sketch editor**



3. Click on the **Coincident constraint** icon



4. Pick the ends of the lines one by to connect the ends

Force the lines to horizontal/ vertical

1. Click on the **Horizontal constraint** icon

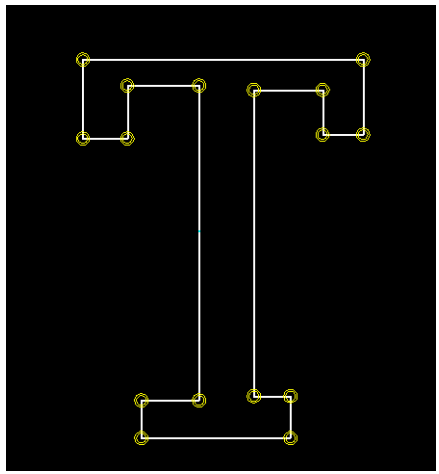


Select all the lines you want to be horizontal.

2. Click on the **Vertical constraint** icon



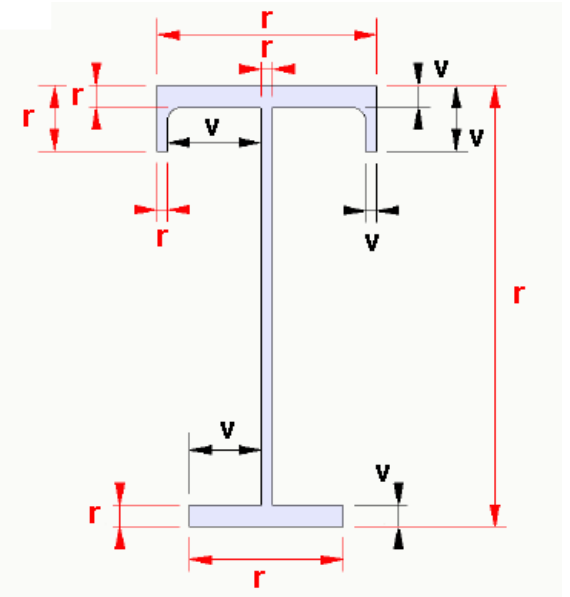
Select all the lines you want to be vertical



Add dimension constraints to the cross section

We will now add the needed dimension parameters to the cross section.

Some of the dimension parameters in the parametric profile (the red ones in the figure below) will be user definable and some (the black ones) will be tied to the user definable parameters with equations.



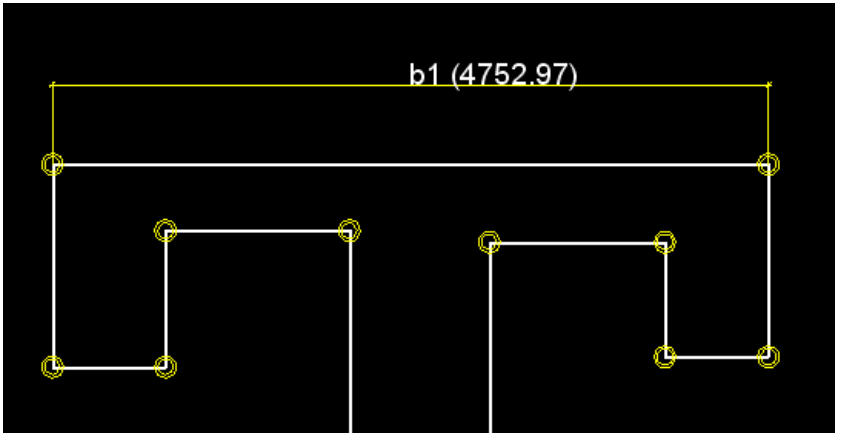
Add horizontal dimensions

1. Click on the **Sketch horizontal distance** icon

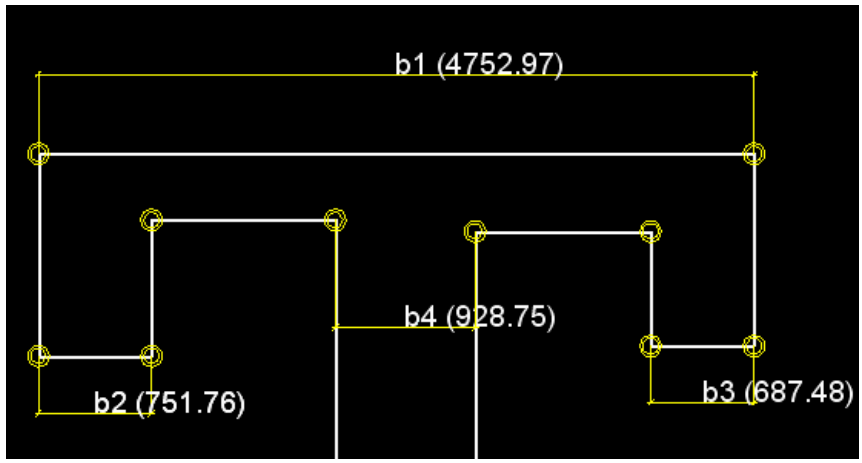


2. Select 2 points (shown in red) and then the dimension line position.

   A dimension is then added and a variable is added to the **Variables** dialog box.



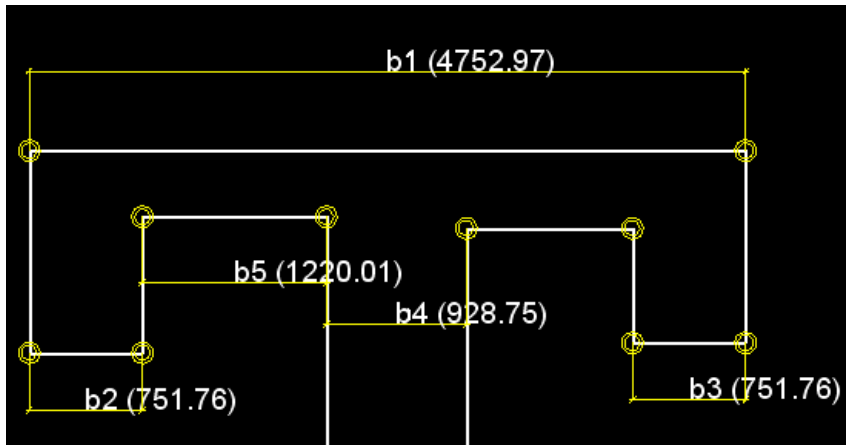
3. Repeat adding dimensions for the edge fold thicknesses (b2, b3) and the web thickness (b4).



4. In the dialog box **Variables,** change parameter **b3** to be **=b2**. B3 is the automatically hidden.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
b1	4752.975	4752.975	Length	Distance	Show	Width
b2	751.755	751.755	Length	Distance	Show	Width
b3	=b2	751.755	Length	Distance	Hide	Width
b4	928.747	928.747	Length	Distance	Show	Width

5. Add a dimension **b5** as shown.



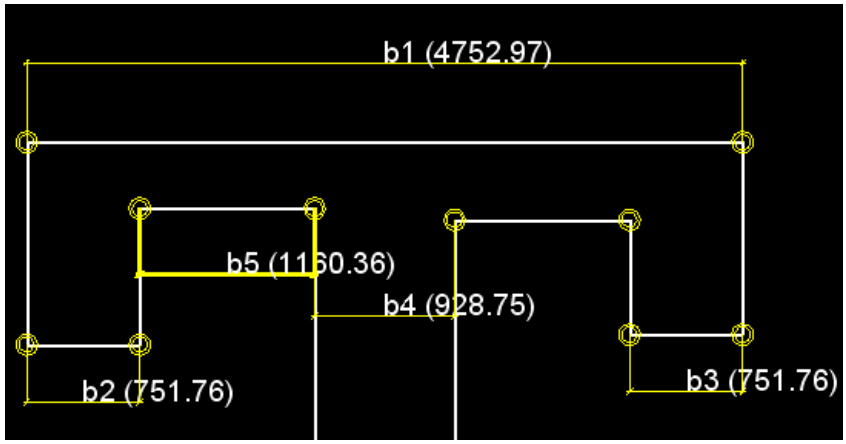


Take care not to add too many dimensions to the profile or the constraints will work against each other.

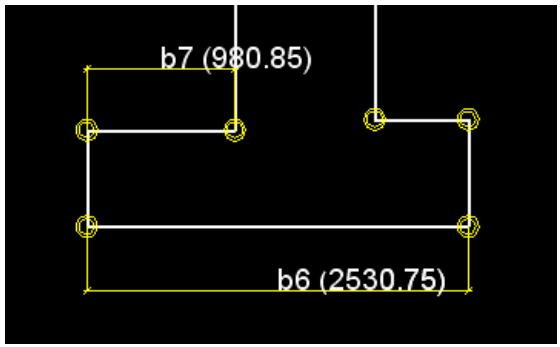
6. In the **Variables** dialog box, change parameter **b5**:

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
b1	4752.975	4752.975	Length	Distance	Show	Width
b2	751.755	751.755	Length	Distance	Show	Width
b3	=b2	274.653	Length	Distance	Hide	Width
b4	928.747	928.747	Length	Distance	Show	Width
b5	=b1/2-b2-b4/2	1160.359	Length	Distance	Hide	Width

This will result the upper flange to be horizontally symmetrical:



7. Add a dimension to the lower flange width (b6) and to the left cantilever (b7).



8. Add the equation =(b6-b4)/2 to parameter **b7**.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
b1	4752.975	4752.975	Length	Distance	Show	Width
b2	751.755	751.755	Length	Distance	Show	Width
b3	=b2	751.755	Length	Distance	Hide	Width
b4	928.747	928.747	Length	Distance	Show	Width
b5	=b1/2-b2-b4/2	1160.359	Length	Distance	Hide	Width
b6	2530.750	2530.750	Length	Distance	Show	Width
b7	=(b6-b4)/2	980.85	Length	Distance	Hide	Width

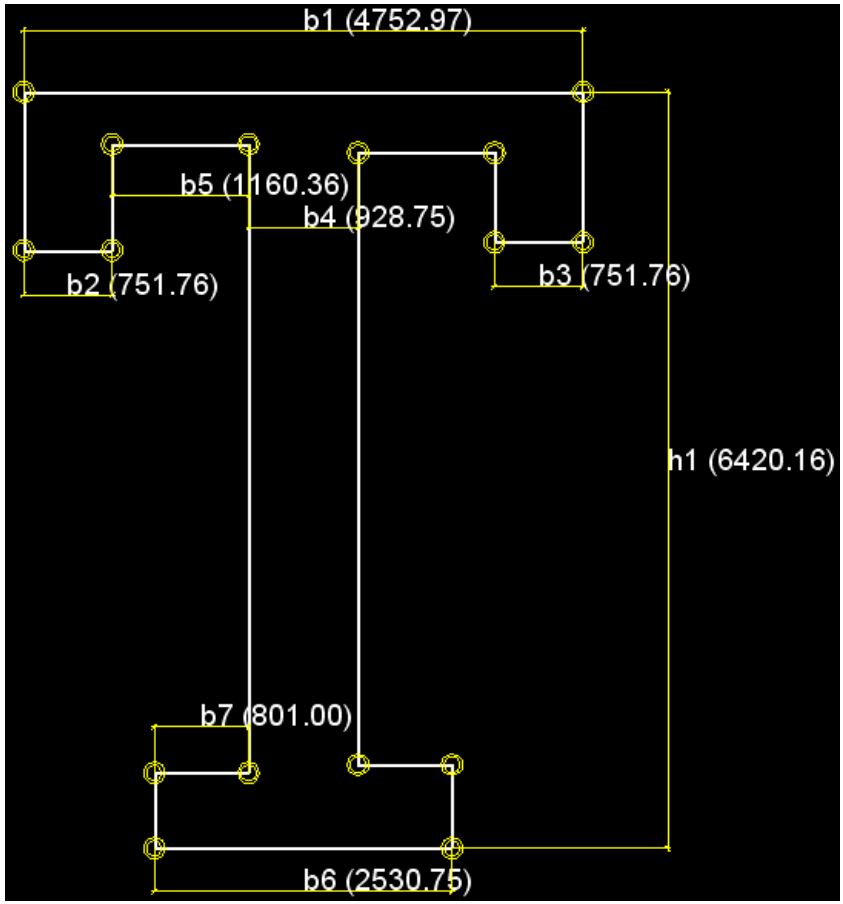
This will result in the left and right cantilevers being symmetrical.

Add vertical dimensions

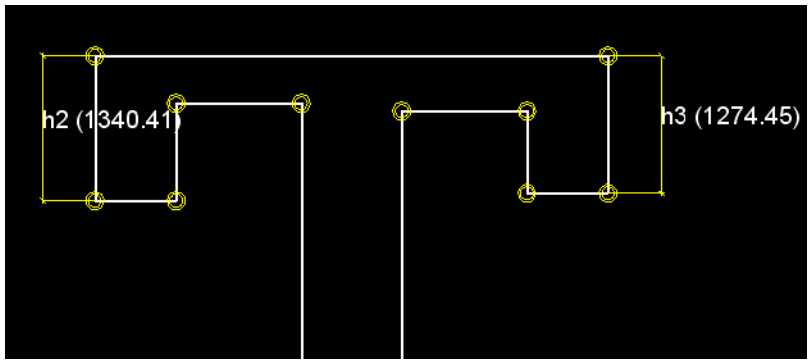
1. Click on the **Sketch vertical distance** icon



2. Add a dimension for the profile height.



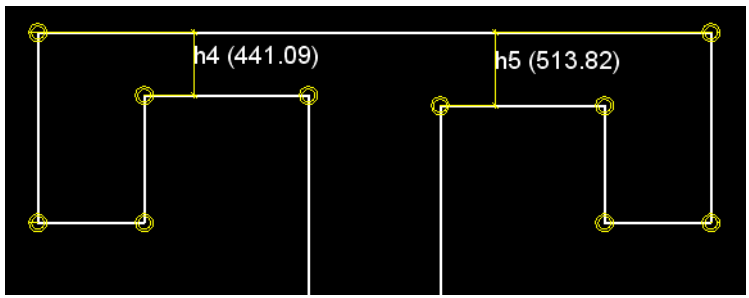
3. Add dimensions for edge fold heights.



4. Add the equation **h3=h2**.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
b1	4752.975	4752.975	Length	Distance	Show	Width
b2	751.755	751.755	Length	Distance	Show	Width
b3	=b2	751.755	Length	Distance	Hide	Width
b4	928.747	928.747	Length	Distance	Show	Width
b5	=b1/2-b2-b4/2	1160.359	Length	Distance	Hide	Width
b6	2530.750	2530.750	Length	Distance	Show	Width
b7	=(b6-b4)/2	801.001	Length	Distance	Hide	Width
h1	6420.164	6420.164	Length	Distance	Show	Height
h2	1340.412	1340.412	Length	Distance	Show	Height
h3	=h2	1340.412	Length	Distance	Hide	Height

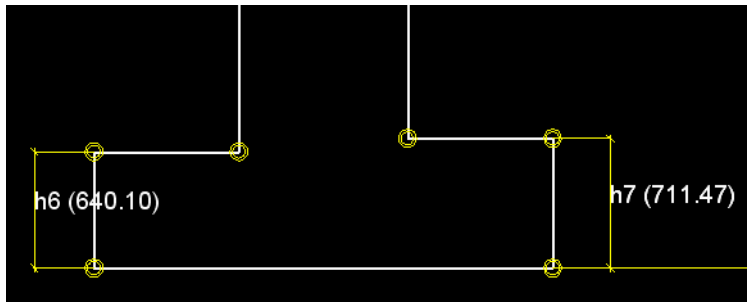
5. Add dimensions for the upper flange thicknesses.



6. Add the equation **h5=h4**

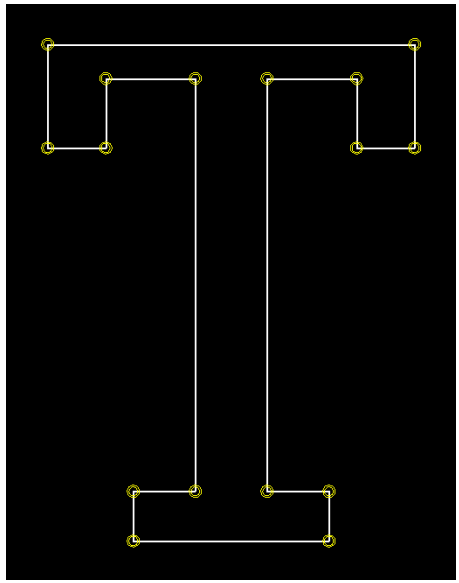
Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
b1	4752.975	4752.975	Length	Distance	Show	Width
b2	751.755	751.755	Length	Distance	Show	Width
b3	=b2	751.755	Length	Distance	Hide	Width
b4	928.747	928.747	Length	Distance	Show	Width
b5	=b1/2-b2-b4/2	1160.359	Length	Distance	Hide	Width
b6	2530.750	2530.750	Length	Distance	Show	Width
b7	=(b6-b4)/2	801.001	Length	Distance	Hide	Width
h1	6420.164	6420.164	Length	Distance	Show	Height
h2	1340.412	1340.412	Length	Distance	Show	Height
h3	=h2	1340.412	Length	Distance	Hide	Height
h4	441.088	441.088	Length	Distance	Show	Height
h5	=h4	441.088	Length	Distance	Hide	Height

7. Add dimensions for the lower flange thicknesses.



8. Add the equation **h7=h6**

   Now the profile is symmetric.



Edit the labels

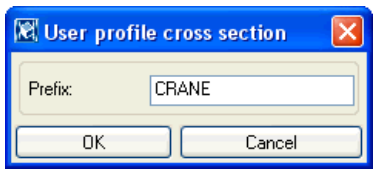
1. Select **Show** for Visibility for the variables that can be user defined. Edit the labels of the shown parameters.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
b1	4752.975	4752.975	Length	Distance	Show	Width
b2	751.755	751.755	Length	Distance	Show	Egde fold t
b3	=b2	751.755	Length	Distance	Hide	Width
b4	928.747	928.747	Length	Distance	Show	Web t
b5	=b1/2-b2-b4/2	1160.359	Length	Distance	Hide	Width
b6	2530.750	2530.750	Length	Distance	Show	Lower flange width
b7	=(b6-b4)/2	801.001	Length	Distance	Hide	Width
h1	6420.164	6420.164	Length	Distance	Show	Height
h2	1340.412	1340.412	Length	Distance	Show	Height
h3	=h2	1340.412	Length	Distance	Hide	Edge fold h
h4	441.088	441.088	Length	Distance	Show	Upper flange t
h5	=h4	441.088	Length	Distance	Hide	Height
h6	640.096	640.096	Length	Distance	Show	Lower flange t
h7	=h6	640.096	Length	Distance	Hide	Height

2. Save the profile.



3. Type the prefix "CRANE" into the **User profile cross section** dialog box.





You cannot store cross sections by making use of numeric values for the prefix in it's name en those names may not be equal to library profile catalog cross sections.

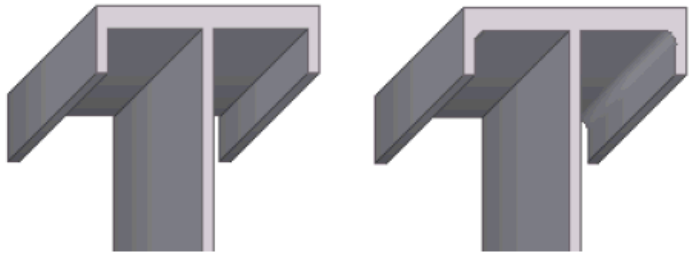
4. Click **OK**.
5. Close the Sketch editor.



The parametric profile is now ready to be used in the profile catalog.

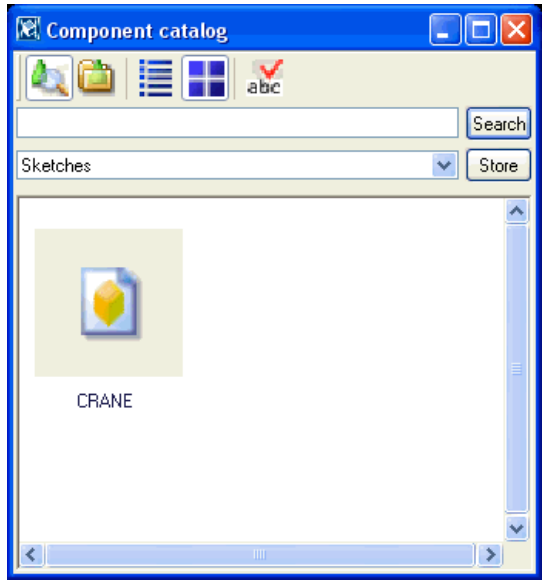
Modify the parametric profile - add chamfers

We will next complete the cross section shape by modifying two inner corners of the crane beam profile, which require a radius.



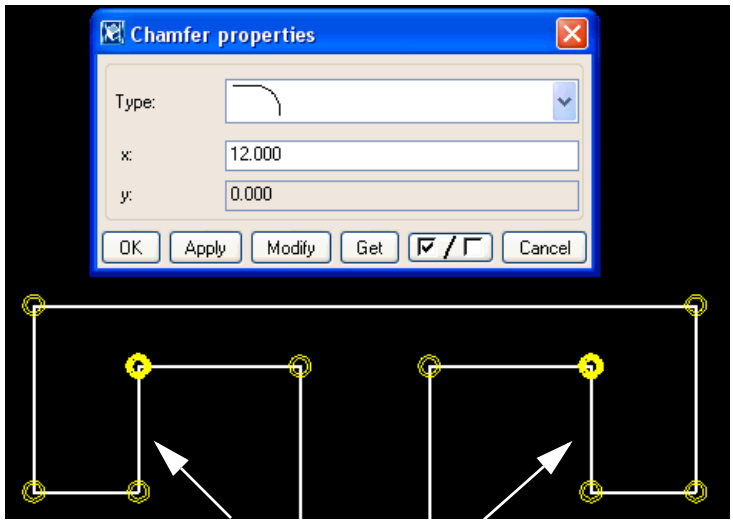
Reopen the profile for editing

1. Open the **Component catalog** (Ctrl + F) and select the category **Sketches** or select **File > Catalog > Profiles > Modify sketched parametric cross section** to access the profile you created in the catalog.

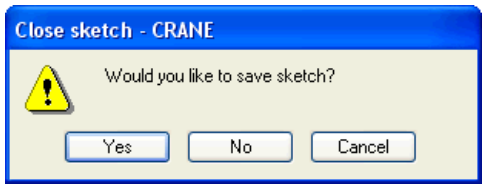


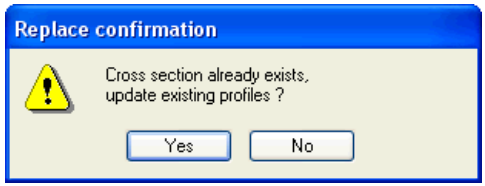
2. Double-click on the sketch CRANE that you just created.

3. Add chamfers to the profile: double-click on a circle highlighted in red, select curved chamfer type, and enter radius values as shown, **Modify**:

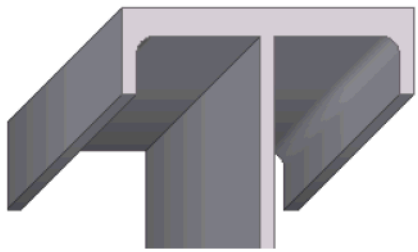


4. Close the editor and save the cross section.



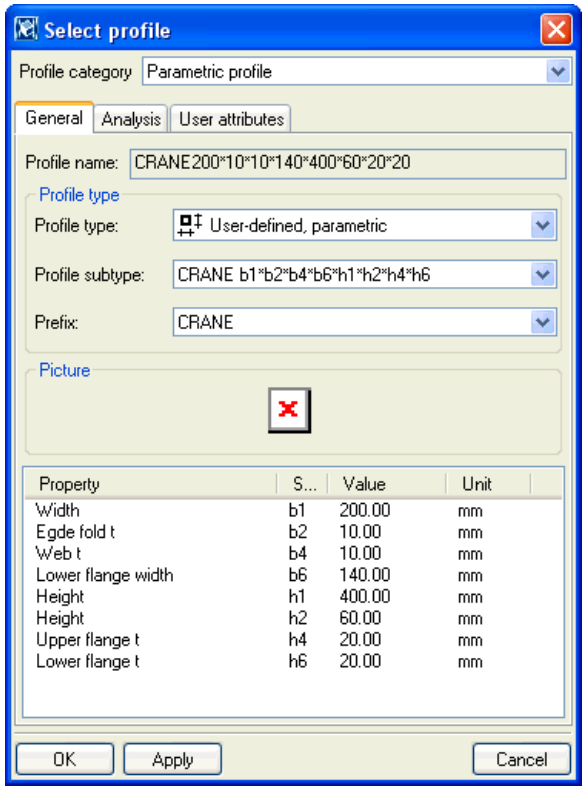


5. Click **Yes** to update the existing profiles in the model.



Test the parametric profile

1. Double-click on the **Create beam** icon. Tekla Structures opens the **Beam properties** dialog box.
2. Click on the **Select...** button on the right side of the **Profile** field. The **Select profile** dialog box opens.
3. Select **Parametric profile** as the category and **User-defined, parametric** as the profile type.
4. On the **Profile subtype** list, select the CRANE profile that you just created.
5. Create a beam to the model and test the profile with different parameters.



Adding a picture of a cross section

Library profiles include pictures which illustrate the shape and dimensions of each profile.

To add pictures of sketched parametric cross sections:

1. Using any image editor, e.g. **Paint**, create an image that shows the shape and dimensions of the profile.
2. Save the image in the folder ..\TeklaStructures*version*\nt\bitmaps, using the cross section name as the filename, with the file extension .bmp. For example, mysketch.bmp.

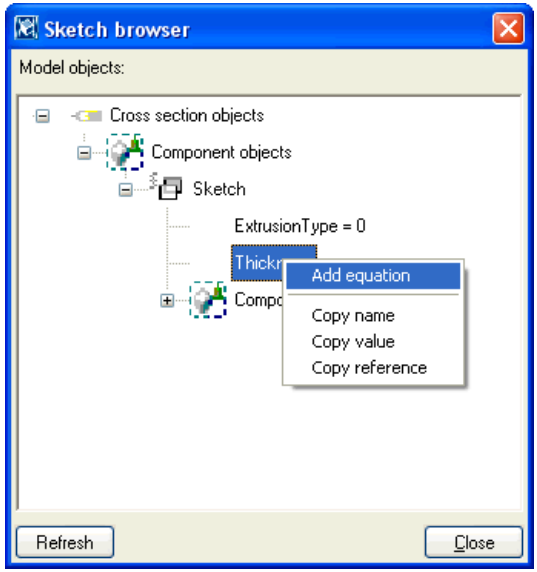
Using snapshots

To use a Tekla Structures snapshot as a picture:

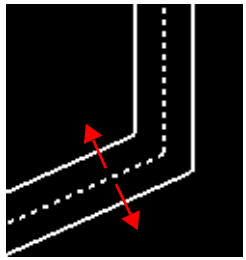
1. Open the sketch in the **cross section sketch editor** view. If necessary, click in the sketch to ensure it is the active view.
2. Press F12 on your keyboard to take a snapshot of the sketch, without the borders.
3. In any file manager, e.g. Microsoft Explorer, rename the snapshot file to have the same name as the sketch, with the extension bmp.
4. Copy the renamed snapshot file to the folder ..\TeklaStructures*version*\nt\bitmaps.
5. Tekla Structures displays a picture of the sketch when you browse for profiles.

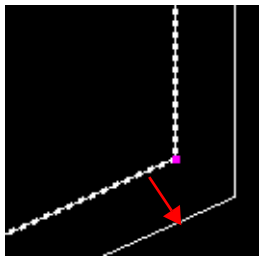
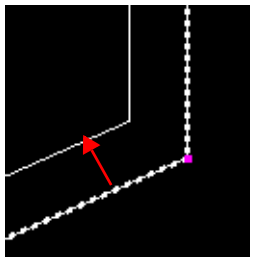
Extruded sketched polyline

To easily create a cold-rolled cross section (of a consistent thickness), sketch an open polyline, and define the extrusion type and thickness of the profile in the **Sketch browser**:



Define the thickness of the extruded line by entering a formula for the **Thickness** property. **Extrusion type** defines how the polyline is extruded. The options are:

Sketch editor view	Sketch browser dialog box	Description
	`Extrusion Type` `Extrusion Type = 0`	Default. Polyline is extruded symmetrically from the center of the sketched line.

Sketch editor view	Sketch browser dialog box	Description
	`Extrusion Type = 1`	Polyline extruded to the outside of the sketched line.
	`Extrusion Type = 2`	Polyline extruded to the inside of the sketched line.

Part- and connection position planes

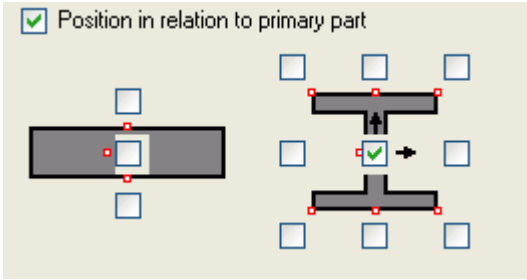
Positioning planes

Use the icons **Show or hide part positioning planes** and **Show or hide connection positioning planes** on the **Sketching** toolbar to view the positioning planes that control a profile:



Connection positioning planes

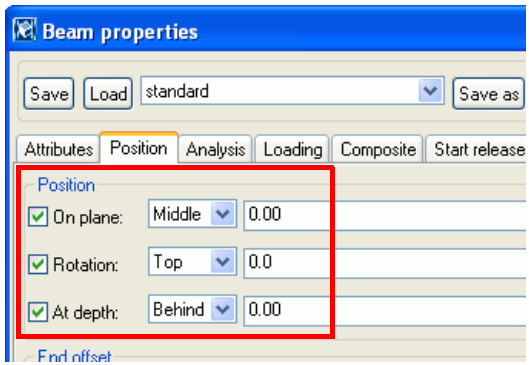
Tekla Structures uses connection positioning planes to define the position of components relative to parts. For example, the position of a custom component relative to a concrete slab. You must select the **Position in relation to primary parts** checkbox on the **General** tab in the component dialog box.



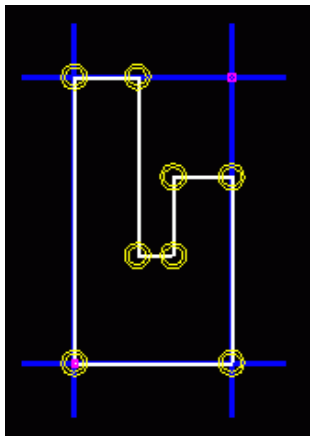
By default, Tekla Structures uses the solid edge planes of the main part as the reference points. If you have modified the part or connection positioning plane in **Cross section sketch editor view**, Tekla Structures uses the modified planes.

Part positioning planes

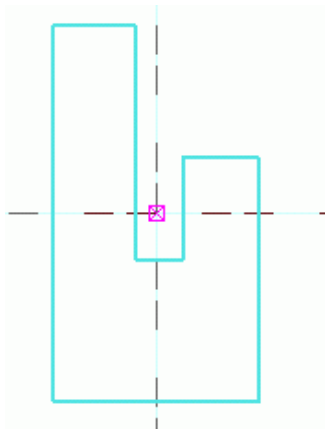
Part positioning planes affect the position of the part in the model (the properties on the **Position** tab, in the part properties dialog box).



This example shows the default part positioning planes in the **Cross section sketch editor view** and in the part end view:

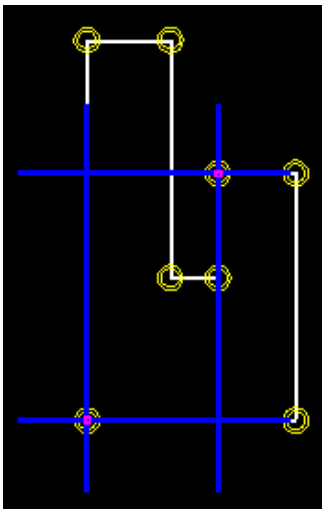


Cross section sketch editor view: Default part positioning planes.

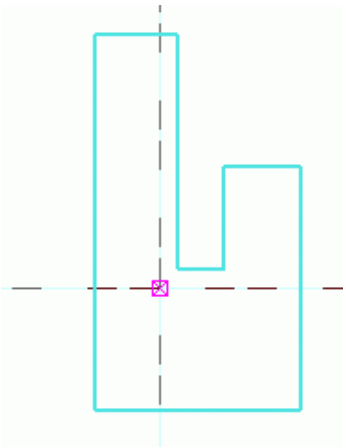


Part end view: The **On plane** and **At depth** properties are both set to **Middle**, with 0 offset.

This example shows modified part positioning planes in the **Cross section sketch editor view** and how those changes affect the position of the part reference line in the model:



Cross section sketch editor view: Modified part positioning planes.



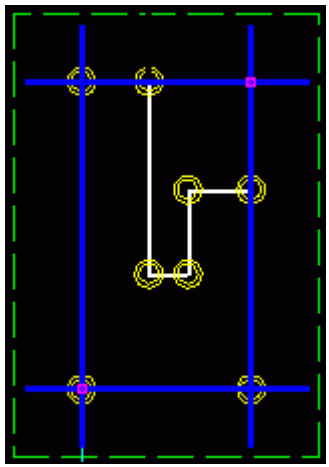
Part end view: The **On plane** and **At depth** properties are both set to **Middle**, with 0 offset.

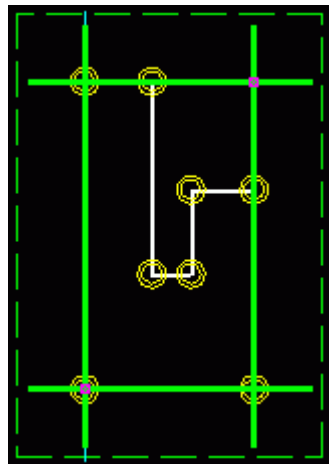
To view or modify part positioning planes:

1. Open the **Cross section sketch editor view**.

2. Click the icon  to show the default part positioning planes, or the  icon to show the connection positioning planes.

   - Part positioning planes appear as blue lines.
   - The connection positioning planes appear as green lines:





- To modify a plane, drag the lilac handles at the upper right and lower left corners of the plane. To save the changes, save the sketch.
- To revert to the default planes: select the planes, right-click and select **Delete** from the pop-up menu. Then click the **Show or hide part positioning planes** or **Show or hide connection positioning planes** icon again.



For more information about sketching parametric profiles, see the Tekla Structures Online Help.

3.36 Tekla Structures Import and Export

Topics

This chapter is divided into following sections:

Import in Tekla Structures (p.393)

Export from Tekla Structures (p.395)

Tekla Structures contains many user friendly interfaces for exchange of data with plant design software like Intergraphs PDS, PDMS, Plantview and calculation software answering the DSTV format.

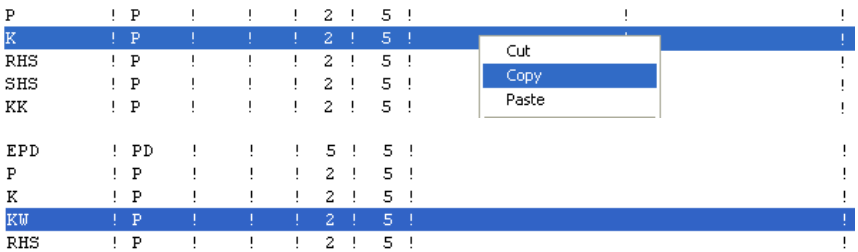
The import/export method is explained in this document. While importing/exporting conversion files can be used. These files are stored in the folder *disk:\TeklaStructures\version\environments\europe\profil*. The names of the files are:

	Material conversion file	Profile conversion file
DSTV	matexp_dstv.cnv	prfex_dstv.cnv
Plantview	matexp_plantv.cnv	prfex_plantv.cnv
SDNF	matexp_sdnf.cnv	prfex_sdnf.cnv
PML	matexp_pml.cnv	prfex_pml.cnv

While exporting to PML it is important that the profile conversion file contains all known files. Only those profiles with their attributes are known in Frameworks+. The other profiles can be imported in Frameworks+, but will only be known as Solid.

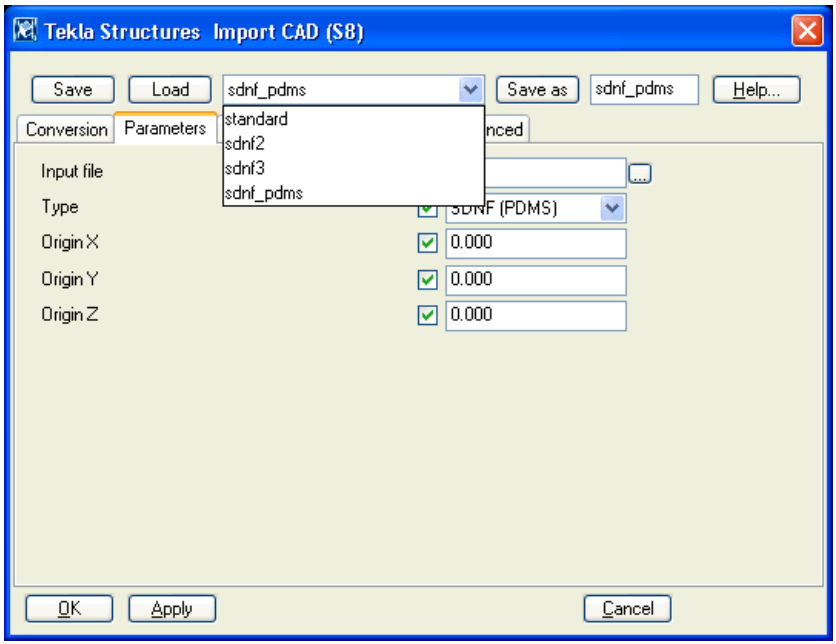
After importing or exporting files, the profile names and their possible converted names stored in the log file in the current model folder. Use this log file to check if the import/export was correct.

There is also an other way to import SDNF. For that, adapt the file *profitab.inp* in the folder *disk:\TeklaStructures\version\environments\europe\profil*. E.g. the prefix *kw.* Select the line *k,* copy the line and replace *k* by *kw.* See following pictures:



Developing SDNF is still goes on, SDNF (PDMS) not anymore!

In Tekla Structures there are some standard settings you can use while importing and exporting data:



The next two tables show some software and their accompanying formats you can import and export.

(However, it is possible that some more data that can be exchanged in Tekla Structures is not in the table).

Import in Tekla Structures

Import → / Software ↓	DWG/ DXF	FEM	CIMsteel	CAD	Other
ArchiCAD	2D, 3D			XML	
AutoCAD	2D, 3D				
BUS		BUS			
Cadmatic	2D, 3D				

Import → / Software ↓	DWG/ DXF	FEM	CIMsteel	CAD	Other
Calma				Calma	
ETABS			CIS2		
FabTrol					SteelFab
FrameWorks				SDNF	DGN (as a ref.model)
GT Strudl			CIS2		MicasPlus
MicasPlus					MicasPlus
MicroStation					DGN (as a ref.model)
ModelDraft					
Multiframe	2D, 3D				
Nastran		STAAD			
PDMS				SDNF	
PDS					DGN (as a ref.model)
Plantview				Plantview	
RAMSteel			CIS2		
ROBOT			CIS2		
RSTAB	2D, 3D	DSTV			STP
SACS		SACS		SDNF	
SDS/2			CIS2	SDNF	
SESAM		SACS			
S-frame	2D, 3D	S-frame			
SAP2000				SDNF	
SPACE GASS				SDNF	
Speedikon				HLI	
STAAD	2D, 3D	STAAD			
Stan 3d		Stan 3d			
StruCad				SDNF	
Structural for MicroStation TriForma					DGN (as a ref.model)
TriForma			CIS2 (Design)		

Export from Tekla Structures

Export → / Software ↓	DWG/ DXF	FEM	CIMsteel	CAD	Other
ArchiCAD	2D, 3D			XML	
AutoCAD	2D, 3D				
Calma				Calma	
FabTrol			CIS2 (Manufac tering)		MIS
EJE					MIS
EPC					MIS
Eureka LPM			CIS1 & 2		
FrameWorks			CIS1 & 2	SDNF, PML	3D DGN
GT Strudl		STAAD			
MicroSAS		MicroSAS			
MicroStation					3D DGN
ModelDraft				PML	
Multiframe	2D, 3D				
Nastran		STAAD			
PDMS				SDNF	
Plantview				Plantview	
RSTAB	2D, 3D	DSTV			
SCIA				SCIA	
SPACE GASS				SDNF	
Speedikon				HLI	
STAAD	2D, 3D	STAAD			
Steel 2000					MIS
Steelcad	2D, 3D			SDNF	
StruCad				SDNF	
Structural for MicroStation TriForma					3D DGN

3.37 ArchiCAD 9 - TS 11.2 im- and export

Topics

This chapter is divided into following sections:

General (p.396)

Workflow (p.396)

The I/O Process (p.397)

Exporting to Tekla Structures (p.398)

Importing from Tekla Structures (p.398)

General

ArchiCAD is integrated with Tekla Structures and carries the benefits of the Virtual Building™ and the 3D workspace into structural design and fabrication, all the way to the cutting machine.

The import-export link means that design elements can be carried directly into Tekla Structures for structural design tasks such as analysis and detailing, and returned to ArchiCAD retaining their critical parameters.

Because the intelligent building structural elements, such as parametric steel profiled beams, are interpreted the same way in both applications, these structures can be edited and detailed in Tekla Structures. All these changes and details will be automatically updated when imported back into ArchiCAD.

Workflow

The architect creates a basic (none detailed), or “wire”, model of the structure that he or she imagines. For this the architect chooses the spans and the profile (H, L etc.) that are appropriate from the Tekla Structures GDL library.

He/she exports this wire model through xml into Tekla Structures.

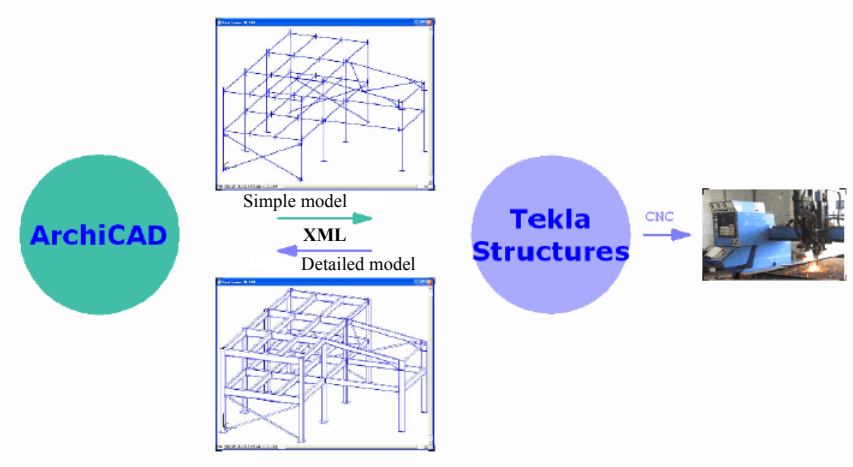
In Tekla Structures, the structural engineer completes architect's initial structure and continues with the analysis of the model, and chooses the right size profiles.

The structural engineer exports back this detailed model through xml into ArchiCAD without bolts or screws, but with the connecting plates.

The architect creates the architectural documentation based on this model in ArchiCAD.

The structural engineer creates the structural documentation based on this model in Tekla Structures.

The structural Engineer sends off the CNC data for the cutting machines.



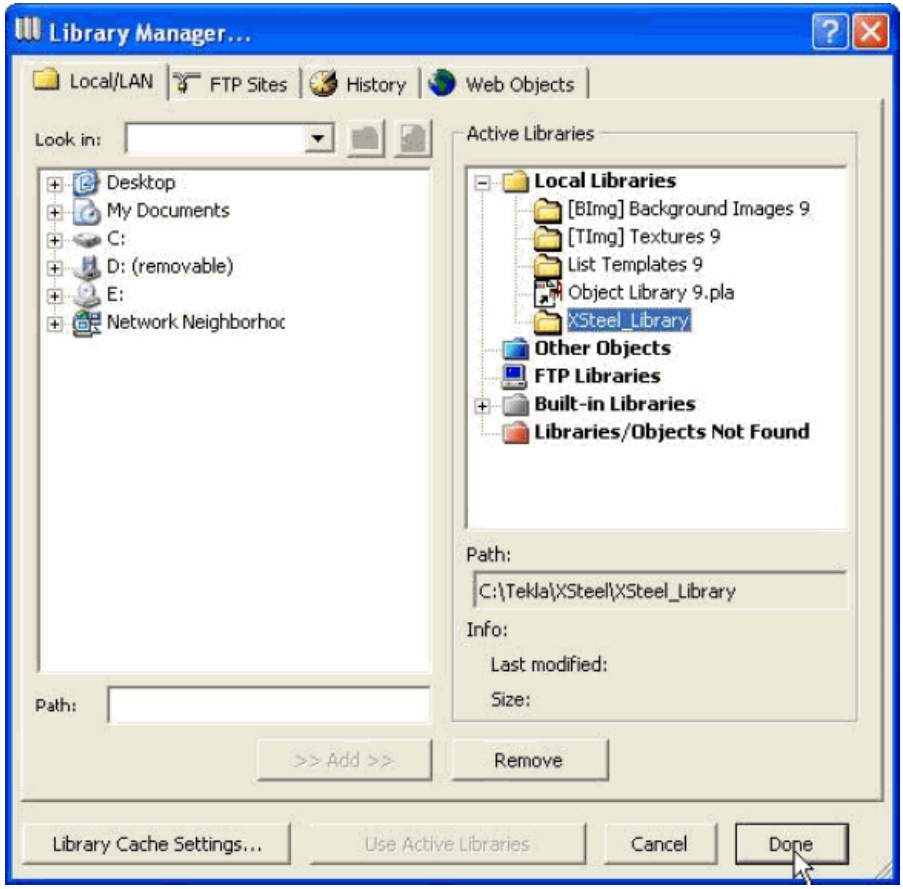
The I/O Process

To use the add-on and the library, you have to do the following:

1. Download the AC-TS_Metric.zip file (for metric EU standard profiles) or the AC-TS_Imperial.zip file (for imperial US standard profiles) file:

www.graphisoft.com/support/archicad/downloads/AC-TS, and uncompress it.

2. These files contain the TeklaStructures_Library (XSteel_Library) and the Tekla Structures add-on.
3. When ArchiCAD 9 is not running, put the Tekla Structures add-on into the C:\Program Files\Graphisoft\ArchiCAD 9\Add-Ons folder of your ArchiCAD.
4. It is advisable to create a separate copy of the library for every separate project, because when importing a model back from Tekla Structures maybe some custom profiles have been added during the detailing. The names of those profiles are not in the standard TeklaStructures_Library. The add-on in this case will offer you to set the different parameters of those profiles manually. Those profiles will be added automatically to the current library therefore the libraries may be different for every file that contains a different building.
5. After starting ArchiCAD 9 from ArchiCAD's File menu, open the Library Manager and "load" the TeklaStructures_Library (XSteel_Library).

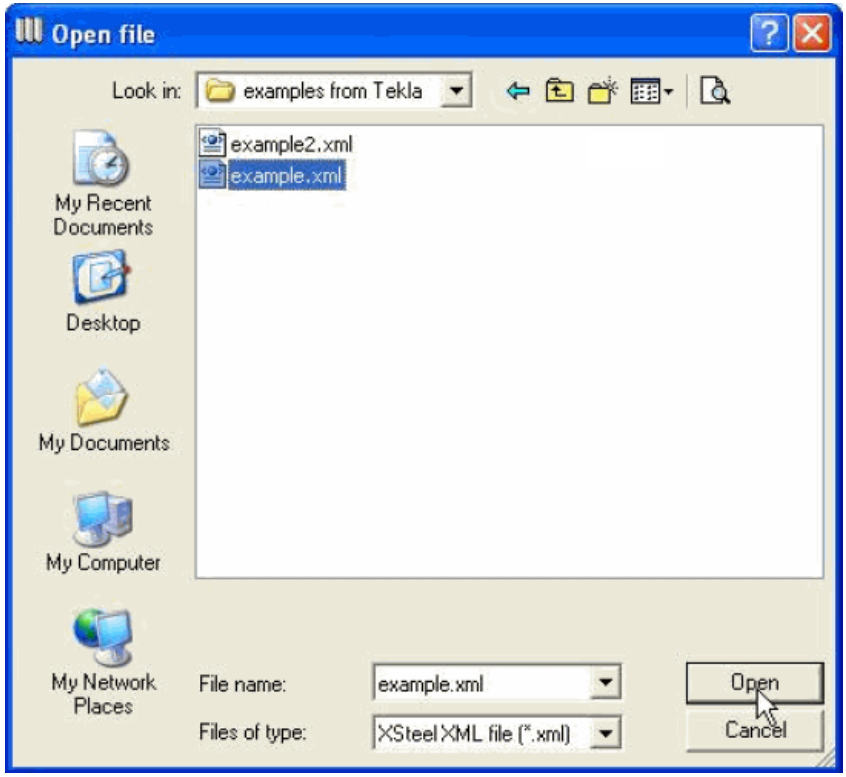


Exporting to Tekla Structures

You have to use the profiles of your Tekla_Structures_Library (XSteel_Library) for creating a basic (simple) model for detailing in Tekla Structures. After creating the model simply “Save as…” TeklaStructures (XSteel)XML file (.*xml)

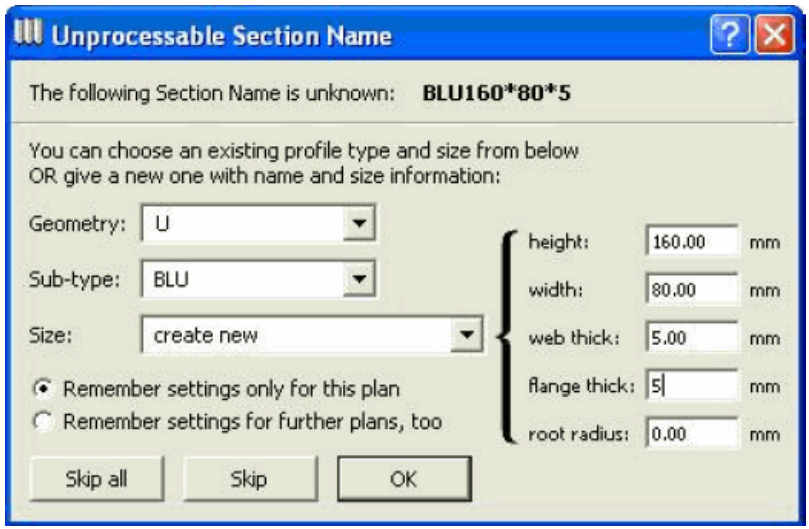
Importing from Tekla Structures

After loading the library, you can open any xml file from the ArchiCAD “Open dialog”.

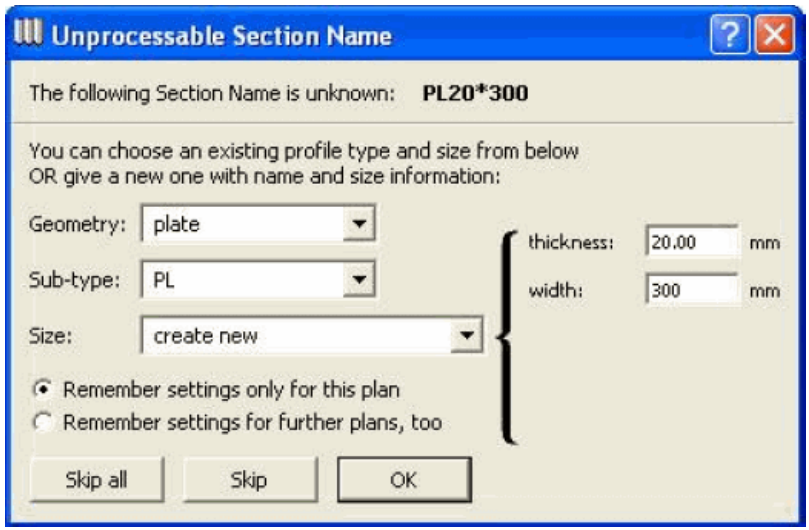


Now the add-on starts working in the following way:

The TeklaStructures_Library contains only the EURO or US Standard profiles of Tekla. If the add-on finds any custom profiles whose name is not in the TeklaStructures_Library, the dialog below comes up and the you will have to set the different parameters of those profiles manually. If you use the US imperial version, you can input the data in decimal inches instead of millimeters: It is important that one profile type can be named only once. You can not have different dimensions or names for the same profile type at the same time in the loaded library

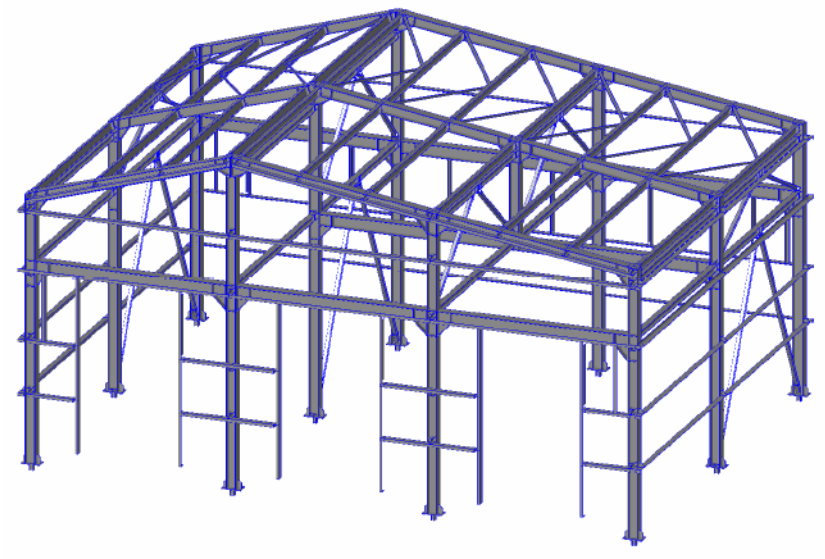


Like in this project above, there is a profile called “BLU160*80*5” which has U geometry with parameters that have to be set manually. When the parameters are keyed in, click OK. Than for, example, comes the PL (plate) or FL (flat) plates, where one parameter is the thickness, the other is the width, and the length is dynamic that is read from the xml:



Please note that there might be “Dxxx” profiles in Tekla Structures which are solid circular plates where xxx is the diameter in mm. This means that here you have to choose a circular profile, and set the wall thickness more than half of the diameter - so in this case it will not be a tube but a solid circular section.

After setting everything and clicking “OK” as many times as the add-on finds a custom profile, you will have a model that you can visualize with all the details within the 3D window of ArchiCAD 9.



3.38 ArchiCAD 11 - TS 12.0 im- and export

Topics

This chapter is divided into following sections:

Workflow (p.402)

The I/O Process (p.403)

Workflow

The architect creates a basic model of the structure that he imagines. For this he chooses the spans and load-bearing structures such as the beams, columns, slabs and walls.

He exports this model through IFC2x2 or 2x3 into Tekla Structures®.

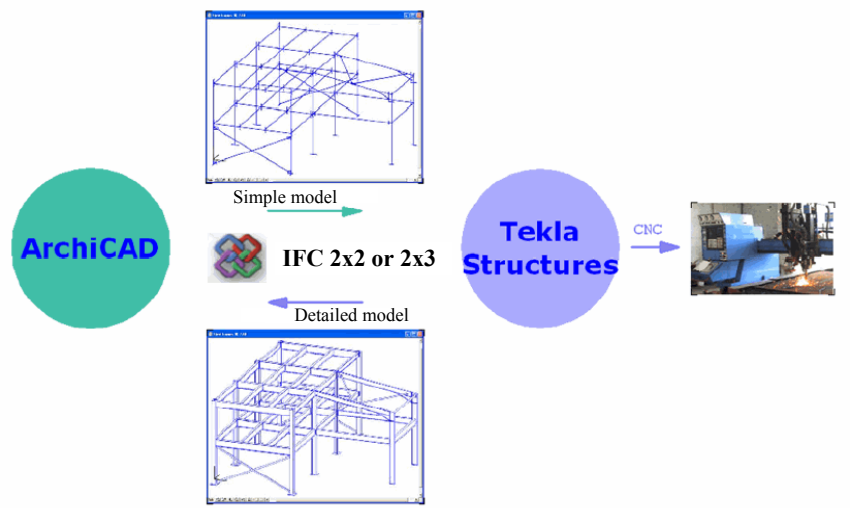
In Tekla Structures the structural engineer completes the architect's initial structure and continues with the analysis of the model, and chooses the right size profiles.

The structural engineer exports back this detailed model through IFC into ArchiCAD without bolts or screws, but with the connecting plates.

The architect creates the architectural documentation based on this model in ArchiCAD.

The structural engineer creates the structural documentation based on this model in Tekla Structures.

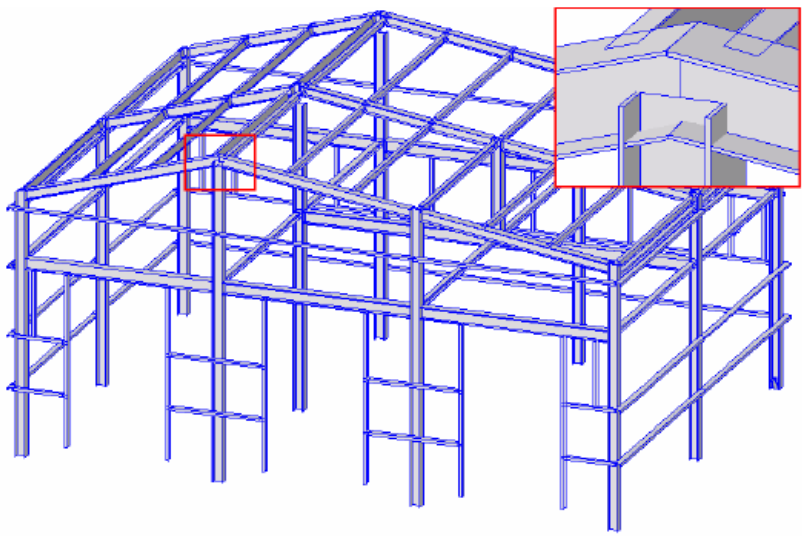
The structural Engineer sends off the CNC data for the cutting machines.



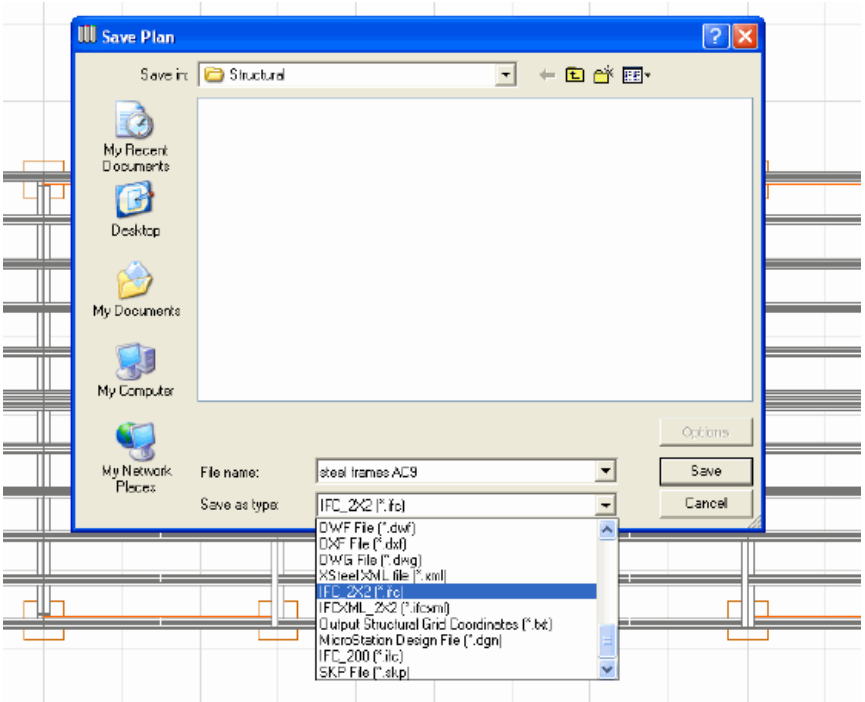
The I/O Process

To use exchange data, you have to do the following:

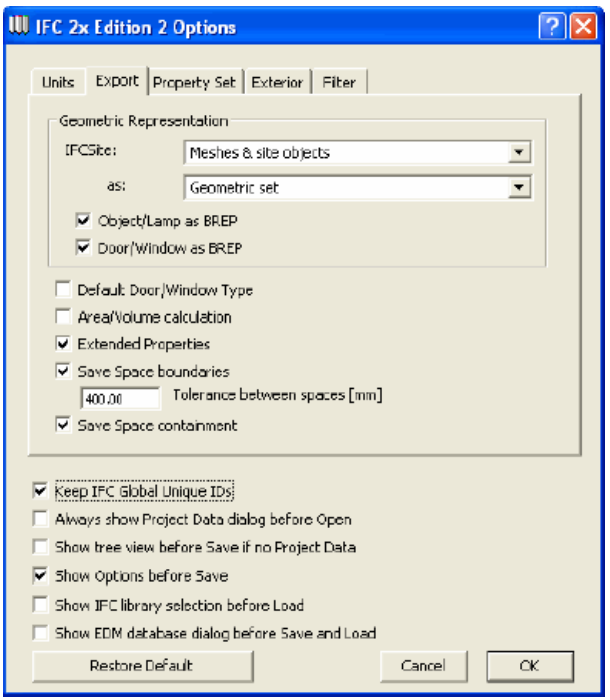
1. Download the latest version of Graphisoft's IFC 2x2 from http://www.graphisoft.com/support/ifc/downloads/public/ and uncompress it. The more recent IFC 2x3 add-on can be downloaded from this link: http://www.graphisoft.com/support/ifc/downloads/beta/
2. Install the add-on into your ArchiCAD (Add-Ons Folder) when ArchiCAD is not running by double-clicking on the uncompressed icon.
3. Start your ArchiCAD, and using the standard ArchiCAD Library and AC tools, assemble the structure you would like.



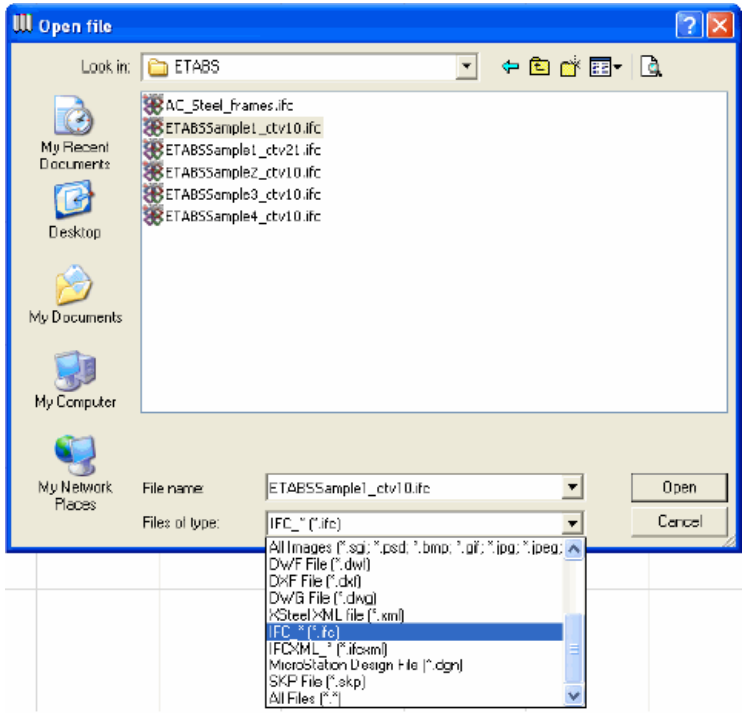
4. From the ArchiCAD Floor Plan view choose "save as IFC 2x2" or "save as IFC 2x3"

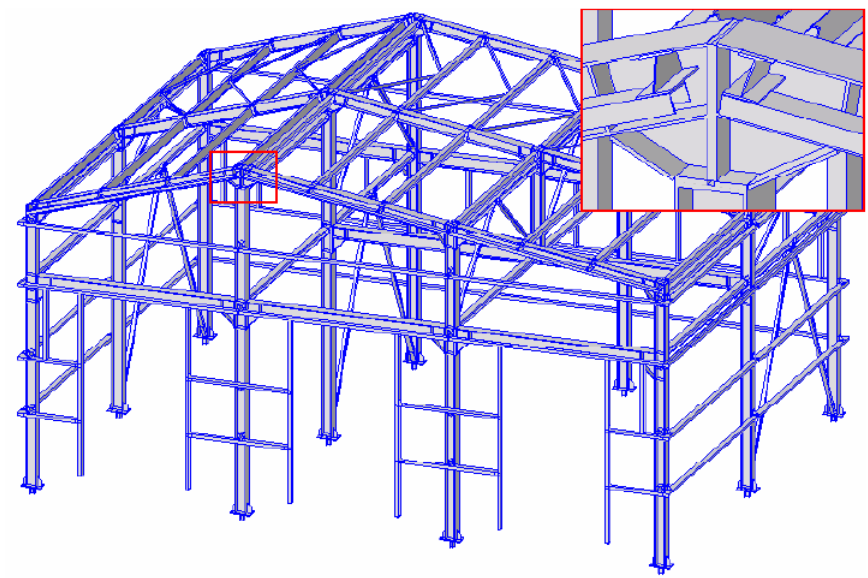


5. Don’t forget to have the following configuration on the Export tab page:



6. To import a model into ArchiCAD from Tekla Structures® through IFC 2x2 or 2x3 do the following:





3.39 Autodesk Inventor im- and export

Topics

This chapter is divided into following sections:

Importing in Tekla Structures (p.406)

How to link Tekla Structures with Autodesk Inventor (p.409)

If you need to export information from Autodesk Inventor and import it into Tekla Structures, you can do it. This hint provides you a solution for reference files, you can also transfer data by IFC format.

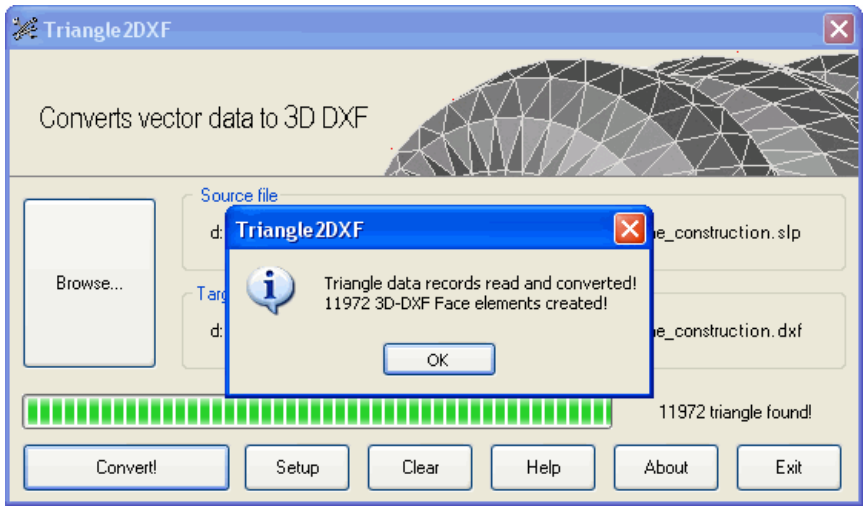
Importing in Tekla Structures

1. Export *.STL file from Inventor, download the video **Inventor_STL_Export** from the Construsoft Xtranet how to export

   In Autodesk Inventor you cannot export Inventor Assemblies to STL format, only Inventor Parts can be exported to STL format, hence the Inventor Assemblies must be converted to a Inventor Part, like this:

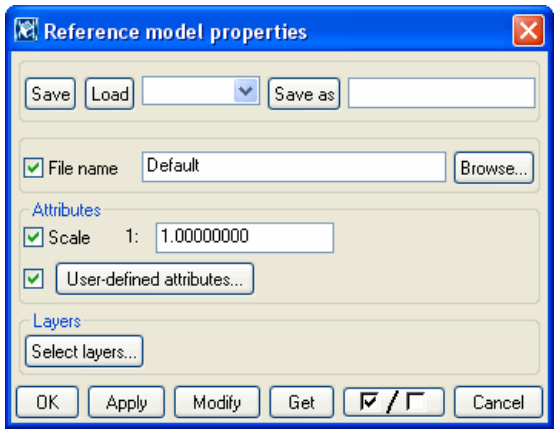
   Start a new Inventor Part and use the command 'Derived Component'. Select the original Inventor Assembly file (.iam) and load the file. You can choose the components you want to include on the target part and set other options. After this is done you can save the complete set of objects to a STL file.

2. Convert the exported *.STL to DXF with Triangle2DXF, you can download this tool from the Xtranet.





3. In Tekla Structures you import the DXF file as a reference object, go to **Parts** > **Reference object** or double-click the icon, the following dialog box will open:

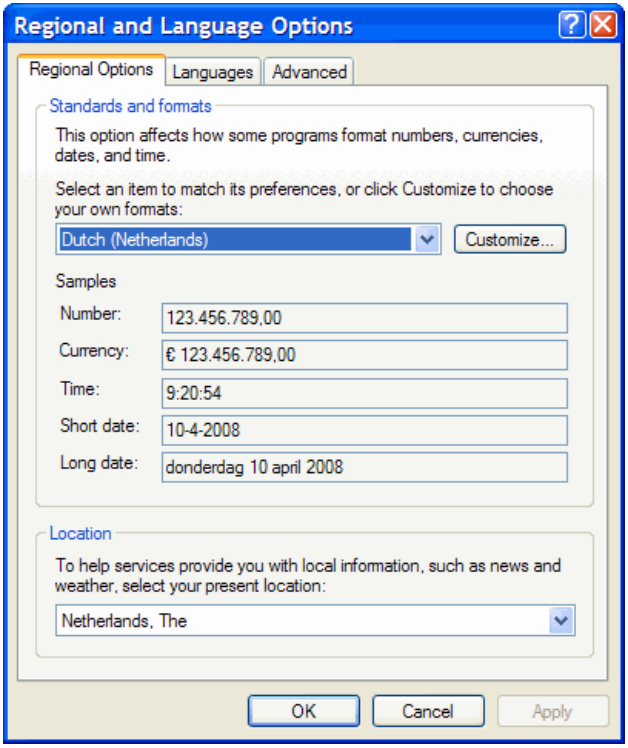


4. Click on the button <**Browse>** to select the DXF file
5. Click <**Apply>** or <**OK**> and pick a position in the model, the DXF file will be imported.

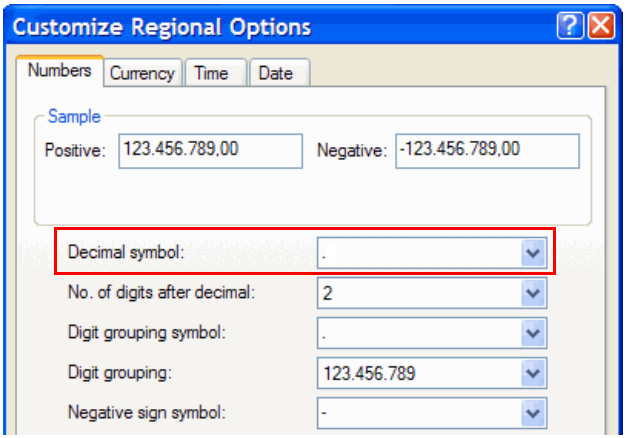


This only works if the option **Decimal symbol** is set to "." (point) in your regional- and language settings. Commas for example won't work.

In Windows, go to **Start** > **Settings** > **Control Panel** and double-click on the category **Regional and Language Options**, the following dialogbox appears:



Click the button <**Customize**>, the following dialogbox appears:



Set the option Decimal symbol to "." (point). Click <**Apply**> and <**OK**> and again <**OK**>.

How to link Tekla Structures with Autodesk Inventor

If you need to link Tekla Structures with Autodesk Inventor, you need to have Autodesk Architectural Desktop (ADT 2006) with IFC add-on. It is not part of the ADT 2006 standard package, but can be ordered as a third party add-on from http://www.inopso.com.

February 2007 Autodesk changed the product name from 'Autodesk Architectural Desktop' to 'AutoCAD Architecture 2008', this release includes also direct IFC import and export functionality. No further add-on needed.

Tekla Structures > Inventor

1. Create an IFC2X2 export in Tekla Structures from menu **File** > **Export** > **IFC**
2. Import it to ADT
3. Save and open with Inventor

Inventor > Tekla Structures

1. Open Inventor model with ADT
2. Create an IFC export
3. Edit the IFC file and change the following string

   ""FILE_SCHEMA(('IFC2X_FINAL'))"

   to

   "FILE_SCHEMA(('IFC2X2_FINAL'));"



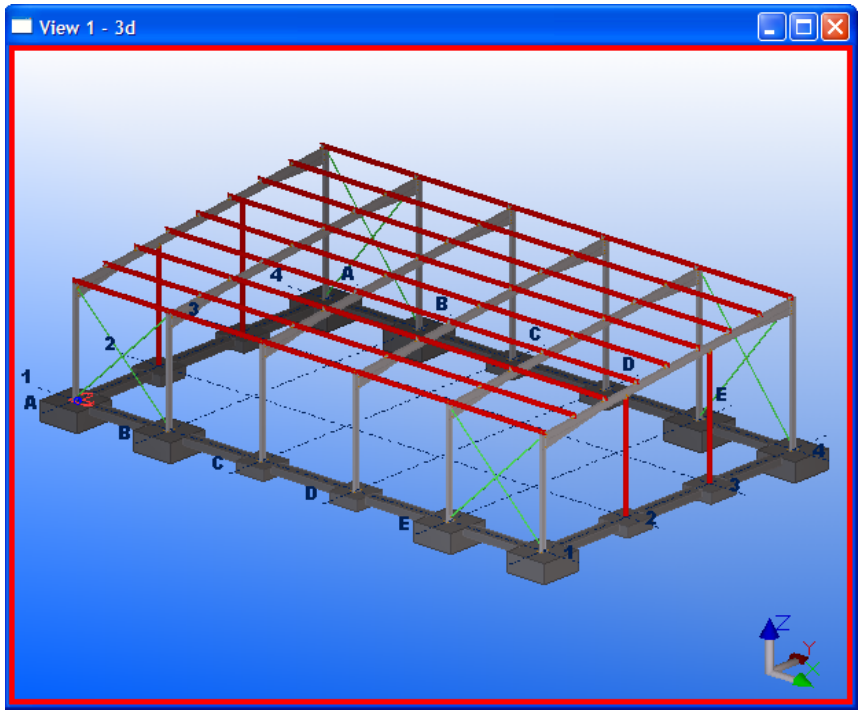
4. In Tekla Structures you import the IFC file as a reference object, go to **Parts** > **Reference object** or double-click the icon.

3.40 Create a 3D PDF from a Tekla Structures Model

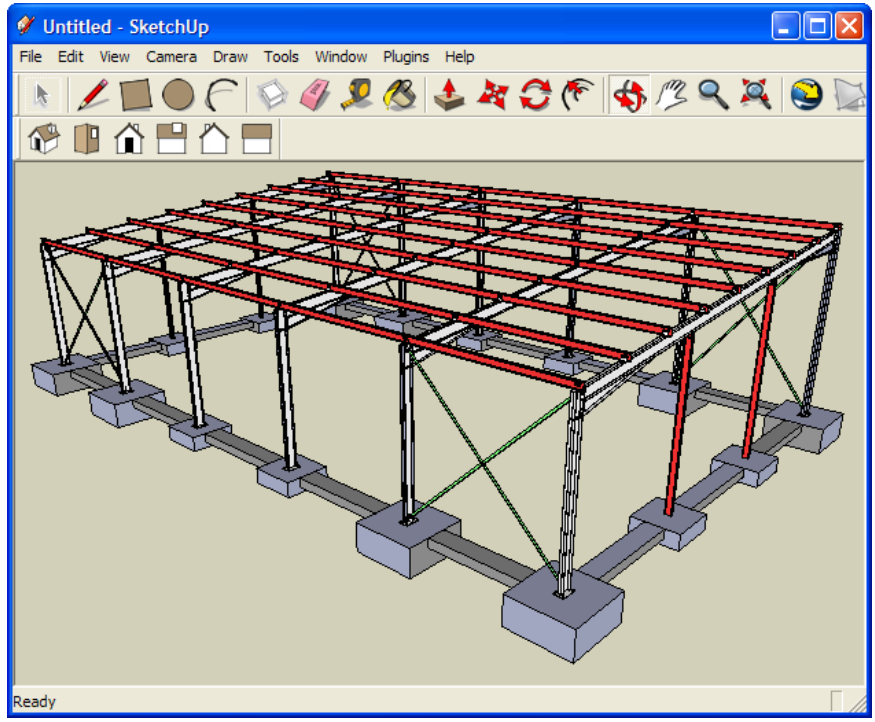
With Adobe Acrobat 3D you can create 3D PDF documents of Tekla Structures models (Tekla Structures model export as DWG/DXF/DGN). Adobe offers this package for about 1.200 EUR, a trial version is available from http://www.adobe.com/products/acrobat3d/tryout.html.

In case 1.200 EUR are too much then maybe the following hint might be of interest. Besides Tekla Structures you need Google SketchUp (Freeware and Pro version can be downloaded from http://sketchup.google.com/download.html) and a third party tool for Google SketchUp to generated 3D PDF documents (trial version can be downloaded from http://www.renderplus.com/pdf/pdf_product.htm).

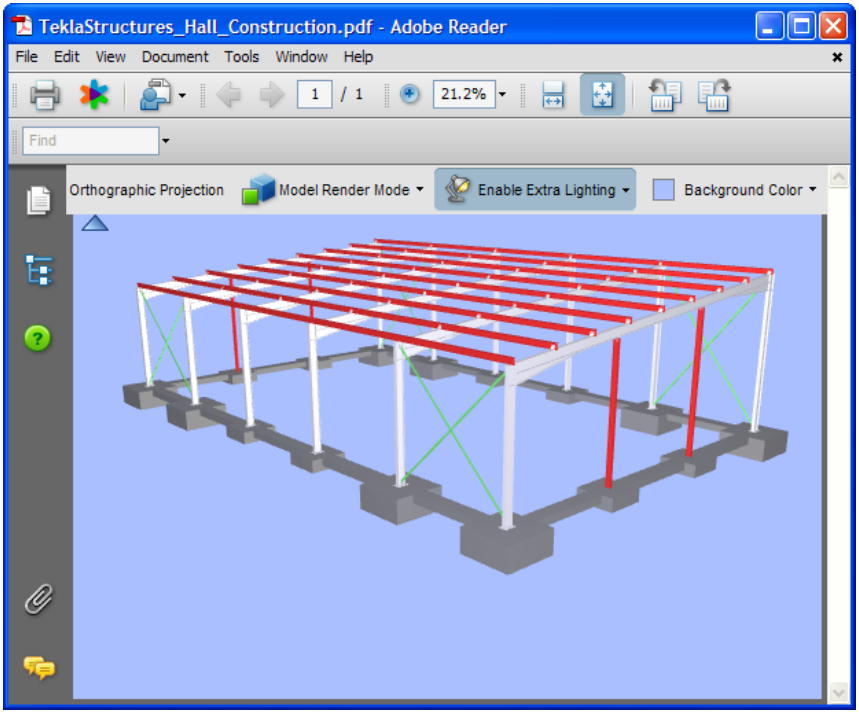
Google SketchUp is available as freeware edition, only the third party tool 'Render Plus System 3D PDF' is needed, single user license for about 190 US Dollar.



The above shown Tekla Structures Model we will use as an example to convert to 3D PDF. The model has been exported to 3D DWG. Next Google SketchUp need to be started to import the Tekla Structures 3D DWG file, the 3D DWG import is also available with the Google Sketchup freeware version. Here the imported Tekla Structures model.



Next the model need to be saved as Google SketchUp file (.skp). Now starting the installed 'Render Plus System 3D PDF' application and loading the saved Google SketchUp file, the file imported from Tekla Structures. This tool for generating 3D offers several settings to customize the output. The following snapshot shows the resulting 3D PDF, which you can download from the Construsoft Xtranet.


TeklaStructures_Hall_Construction.pdf - Adobe Reader
File Edit View Document Tools Window Help
1 / 1
21.2%
Find
Orthographic Projection
Model Render Mode
Enable Extra Lighting
Background Color

3.41 IFC im- and export

Topics

This chapter is divided into following sections:

What is IFC? (p.413)

Condition (p.413)

Exporting from Tekla Structures (p.414)

Importing in Tekla Structures as a reference model (p.415)

In Tekla Structures you have the possibility to export a model to an IFC file, you can also import an IFC file in Tekla Structures.

What is IFC?

The IFC system is a data representation standard and file format for exchanging 3D CAD data. This format is particularly supported by 3D CAD suppliers such as Tekla, AutoDesk, Nemetschek, Bentley, Archicad.

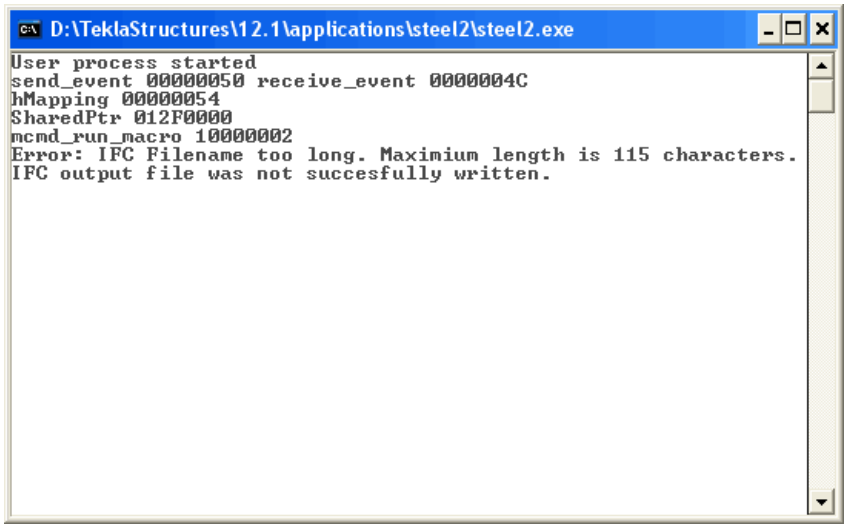
IFC supports the data exchange which effects the efficiency of the project in a positive way. Users can control the entire project and they can share information for several applications and disciplines.

The participants who were (and still are!) involved in developing the IFC format are leading software developers, research institutes, universities, project management agencies, etc.

It looks as if IFC will become the standard for data exchange more and more in the next years.

Condition

When you start exporting a model which is stored on a server, it may occur that the models' path name contains more then 115 characters. Because of this, the following dialog appears:



So it only concerns the “original” models’ path name, not the name that you can enter for the Output file, see the picture below. It is useless to enter **C:\ouput.ifc** e.g. for the Output file.

For a correct working, take into account the maximum of 115 characters.

To export the model, copy or move the model to any location on your computer that takes less then 115 characters, for example **C:\temp.**

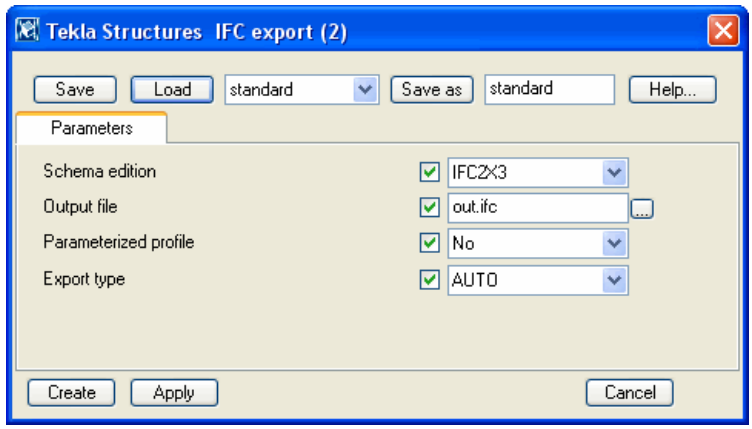
Exporting from Tekla Structures

To export a model or selected parts from a model into an IFC file you do next:

Step plan

1. Open a Tekla Structures model
2. Select a model or selected parts from a model you want to export.

3. Click **File** > **Export** > **IFC...** to open the IFC Export (2) dialog box:



Schema edition

Select the IFC version you want to export to. You can select IFC2X, IFC2X2, IFC2X3 en IFC2X3_mapping.

Output file

Enter the folder in which you want to store the IFC file. If nothing is entered, the file will automatically be stored in the model folder.

Parameterized profile

Select to export parametric profiles.

Export type

Use the option BREP to only use Brep presentation in IFC export.

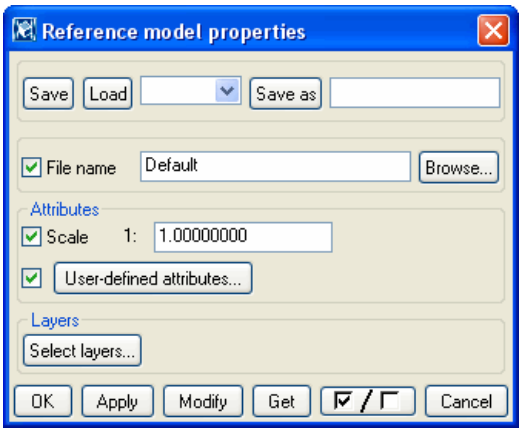
The option AUTO uses Brep presentation only for curved, tapered, warped etc. special cases.

4. Now click <**Create**>, Tekla Structures creates the export file.

Importing in Tekla Structures as a reference model



In Tekla Structures, IFC files can be inserted as a reference model to serve as a help (reference). To insert an IFC file, double-click the icon opposite. The following dialog box will appear:





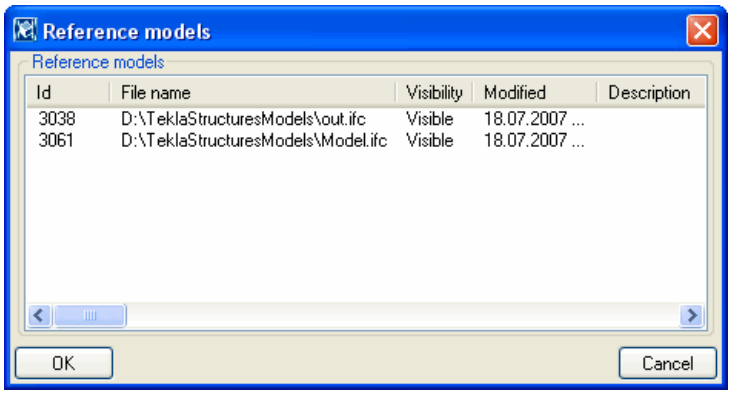
To be able to select the entire IFC file, activate switch **Select components**, to select the reference model at the highest level.

To switch on or off one or more layers in the used reference model, make use of the button **Select layers...**.

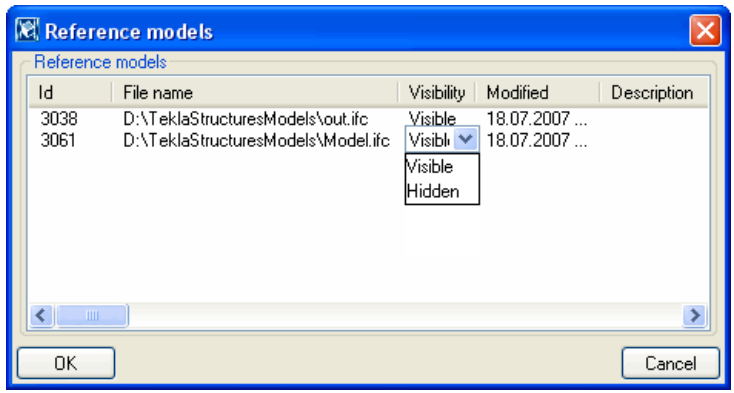
Reference models will keep the size of Tekla Structures model files relatively small, because the reference file(s) are considered as single objects. It is possible to snap to vertexes from the reference model.

If there are several IFC files in the Tekla Structures model, it can be useful to hide the reference models separately.

This can be done in dialog box *Reference models*. To open this dialog box, go to **File > Reference models**. The dialog box below will appear:



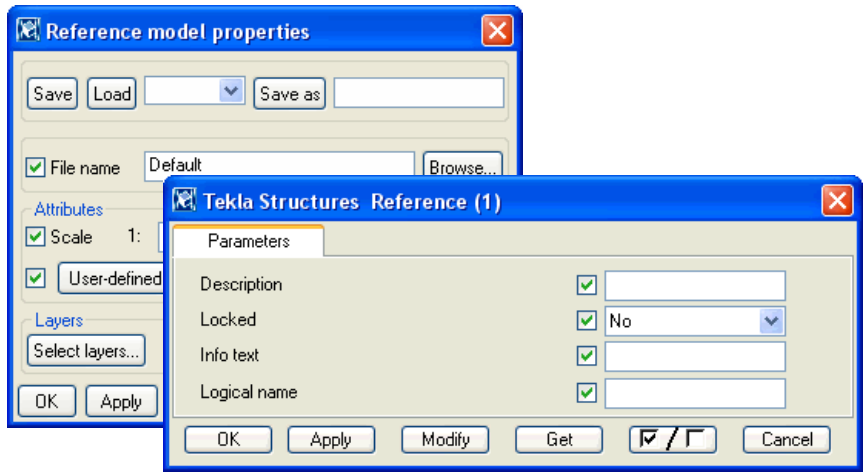
When selecting a line in this dialog box, the corresponding reference model will be highlighted in the model. The other way around, when selecting a reference model in the model will highlight the corresponding line in the dialog box.The visibility of several reference models can be set in column **Visibility**. By pressing the current setting in column **Visibility**, a list box will show up.



The options are **Visible** and **Hidden**.

If Tekla Structures is restarted, the reference model shall be invisible initially. To display this reference model again, check the option **Reference model** in **View Properties > Display**.

The displayed text in columns **Description, Name** and **Info text** in the dialog box above, can be entered in the User Defined Attributes of the reference model.



See also

For more information about importing and exporting IFC files, see Tekla Structures Help (F1).

3.42 Microstation im- and export

Topics

This chapter is divided into following sections:

Importing in Tekla Structures as a reference model (p.418)

Updating reference models (p.420)

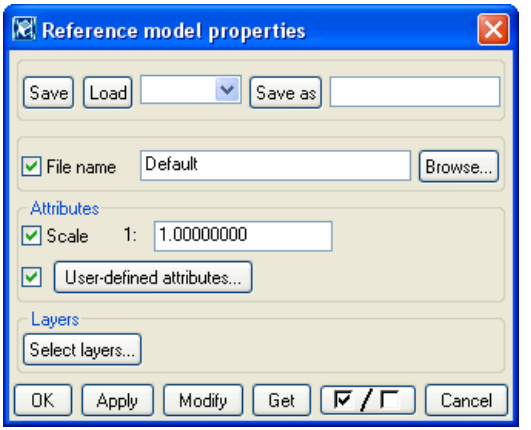
Exporting from Tekla Structures (p.420)

In Tekla Structures you have the possibility to import a model from Microstation in Tekla Structures, but you can also export a model from Tekla Structures to import it in Microstation.

Importing in Tekla Structures as a reference model



In Tekla Structures, a DGN file can be inserted as a reference model to serve as a help (reference) to model the structure. To insert a DGN file, double-click the icon opposite. The following dialog box will appear:





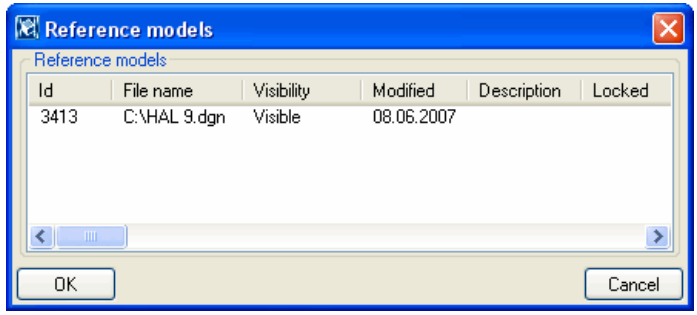
To be able to select the entire reference model, activate switch **Select components**, to select the reference model at the highest level.

To switch on or off one or more layers in the used reference model, make use of the button **Select layers...**.

Reference models will keep the size of Tekla Structures model files relatively small, because the reference file(s) are considered as single objects. It is possible to snap to vertexes from the reference model.

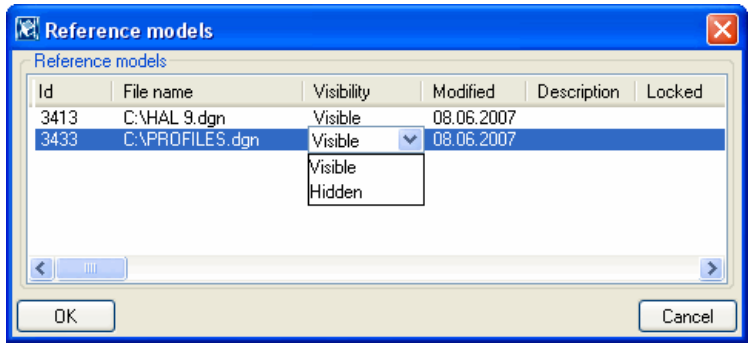
If there are several reference models in the Tekla Structures model, it can be useful to hide the reference models separately.

This can be done in dialog box *Reference models*. To open this dialog box, go to **File > Reference models**. The dialog box below will appear:



When selecting a line in this dialog box, the corresponding reference model will be highlighted in the model. The other way around, when selecting a reference model in the model will highlight the corresponding line in the dialog box.

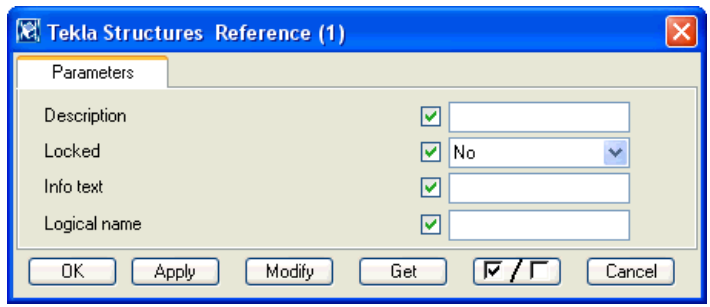
The visibility of several reference models can be set in column **Visibility**. By pressing the current setting in column **Visibility**, a list box will show up.



The options are **Visible** and **Hidden**.

If Tekla Structures is restarted, the reference model shall be invisible initially. To display this reference model again, check the option **Reference model** in **View Properties > Display**.

The displayed text in columns **Description, Name** and **Info text** in the dialog box above, can be entered in the User Defined Attributes of the reference model.



Updating reference models

You can easily update the reference models in your model in case there have been changes in the original reference models after you have brought them into Tekla Structures. To update all reference models:

- Click **File** > **Reference models...**
- Click <**Reload all**>.

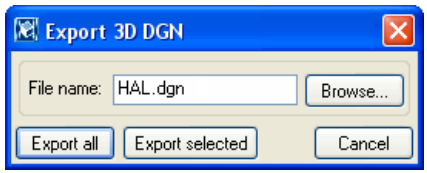
Tekla Structures will go through all the reference models and re-generates the reference models having newer modified date.

Exporting from Tekla Structures

Normally, you can export parts to Microstation using normal 3D DGN export. In this way you can not see the member properties in Microstation.

Step plan

1. Open a Tekla Structures model
2. Select the objects in the model to export.
3. Click **File** > **Export** > **3D DGN...** to open the Export 3D DGN dialog box:



4. Tekla Structures creates the HAL.dgn file in your current model folder.

Settings

You can also use the following variables to control DGN exports:

XS_DGN_EXPORT_PART_AS

Use to specify how Tekla Structures exports solids to DGN format:

Set to CELL to export solids as cells containing the solid faces as a DGN shape.

Set to SOLID to export solids as DGN solids defined by boundary elements

XS_EXPORT_DGN_FILENAME

Use to indicate the output file name in DGN exports. The default is model.dgn.

XS_EXPORT_DGN_INCLUDE_CUTS

Use to define which cuts to include in DGN exports. You can use the following switches:

Switch	**Use to**
FALSE	Exclude all cuts.
TRUE	Include all cuts (default value).
CLASH	Include all cuts but skip hole cuts' ends.
CLASH_NORMAL _PLATES	The same as TRUE for contour plates and CLASH for all other parts.

XS_EXPORT_DGN_INCLUDE_INNER_CONTOUR

Use to include or exclude inner contours in DGN exports. The options are:

TRUE to include inner contours (default)

FALSE to exclude inner contours

To view the exported DGN file, you can use the TC Reviewer or a similar reviewer with Microstation, you can download the TC Reviewer from the Xtranet.

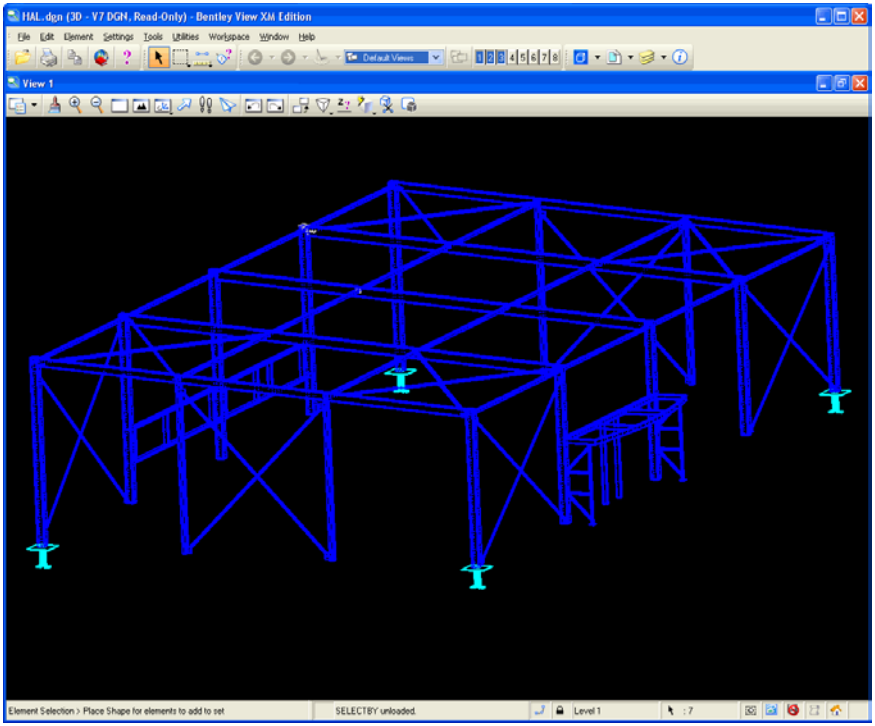
In case you want to see the member properties in Microstation:

Step plan

1. Make sure you have set XS_DGN_EXPORT_USE_LOCAL_ID=TRUE

   You can define this setting in the dialog box **Advanced options**, section **Export**

2. Create a 3D DGN export from the Tekla Structures model
3. AFTER the export, run the report dgn_attribute_info_file.drv.rpt, you can download this report from the Xtranet.
4. Microstation reviewers (like TC Review) are able to show member properties if you run this drv file in Microstation, you can download the tool TC Review from the Xtranet:


HAL.dgn (3D - V7 DGN, Read-Only) - Bentley View XM Edition
File Edit Element Settings Tools Utilities Workspace Window Help
Default Views
View 1
Element Selection > Place Shape for elements to add to set
SELECTBY unloaded.
Level 1

3.43 Pro/Engineer import

Topics

This chapter is divided into following sections:

Importing in Tekla Structures (p.423)

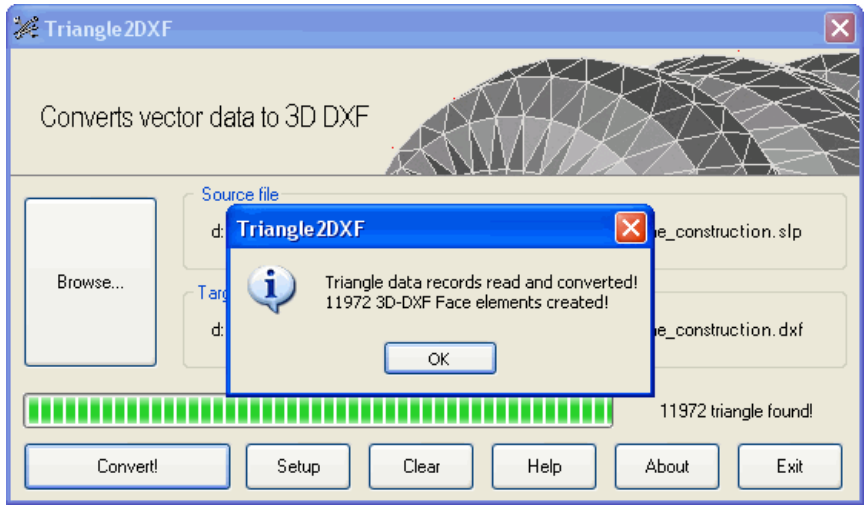
In Tekla Structures you have the possibility to import a model from Pro/Engineer.

Importing in Tekla Structures

1. Export *.SLP file from Pro/Engineer

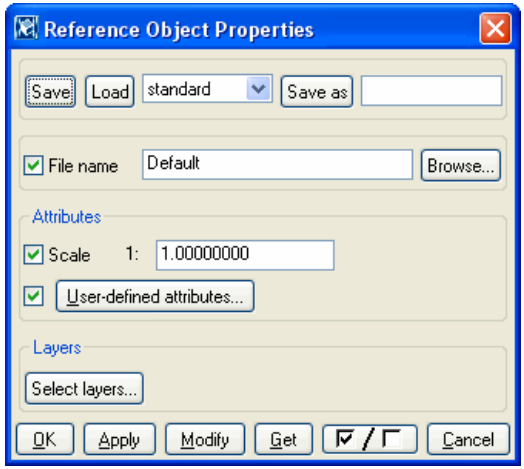
   Make sure you have selected the *.SLP ASCII export, **not** SLP Binary export.

2. Convert the exported *.STL file to DXF by using Triangle2DXF, you can download this tool from the Xtranet





3. In Tekla Structures you import the DXF file as a reference object, go to **Parts** > **Insert Reference Model..** or double-click the icon, the following dialog box will open:

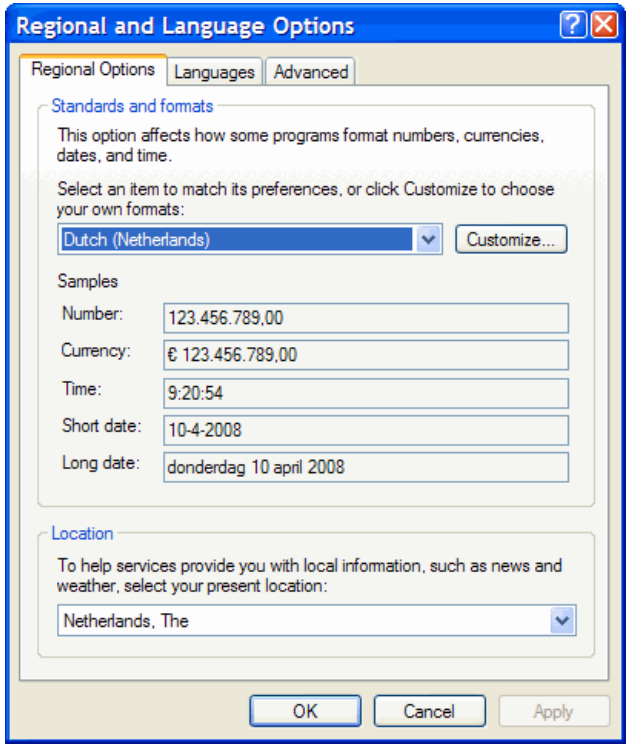


4. Click on the button <**Browse>** to select the DXF file
5. Click <**Apply>** or <**OK**> and pick a position in the model, the DXF file will be imported.

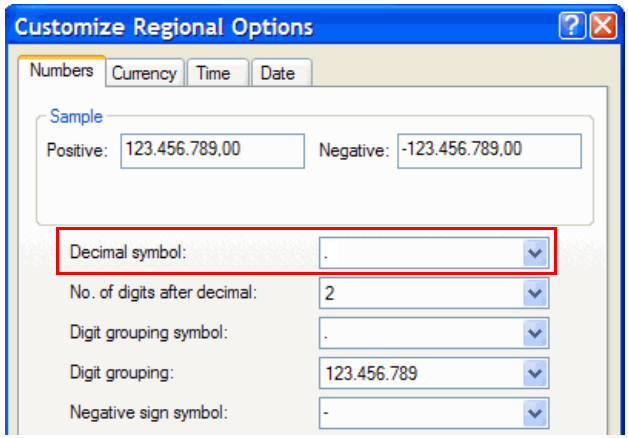


This only works if the option **Decimal symbol** is set to "." (point) in your regional- and language settings. Commas for example won't work.

In Windows, go to **Start** > **Settings** > **Control Panel** and double-click on the category **Regional and Language Options**, the following dialogbox appears:



Click the button <**Customize**>, the following dialogbox appears:



Set the option Decimal symbol to "." (point). Click <**Apply**> and <**OK**> and again <**OK**>.

3.44 SolidWorks im- and export

Topics

This chapter is divided into following sections:

Importing in Tekla Structures (p.426)

Exporting from Tekla Structures (p.429)

Using Tekla Structures model as reference in SolidWorks (p.429)

In Tekla Structures you have the possibility to import a model from SolidWorks in Tekla Structures, but you can also export a model from Tekla Structures to import it in SolidWorks.

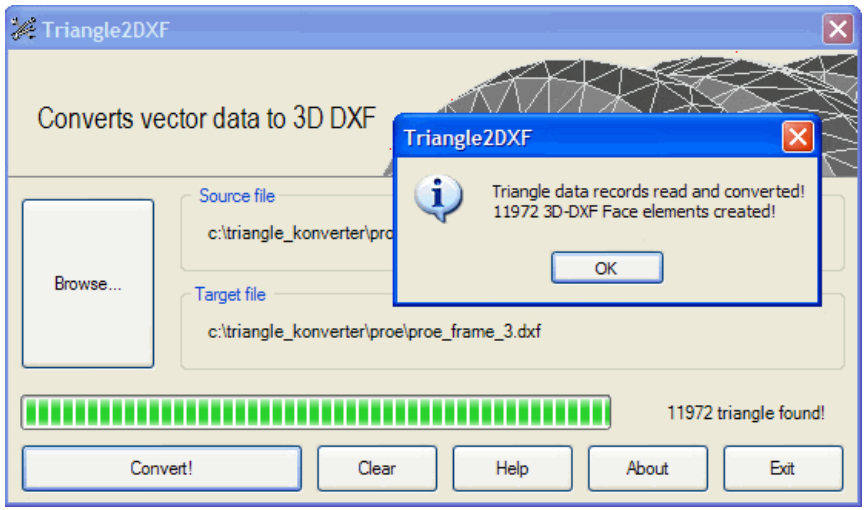
Importing in Tekla Structures

Option 1

1. You first need to create an ACIS file (*.sat) in SolidWorks
2. In the DWG Editor you create an new file and import the *.sat file, go to **File** > **ACIS IN**
3. In the DWG Editor click **File** > **SaveAs** to save the file to DWG version AutoCAD 14 (use this version to make it work better in Tekla Structures)

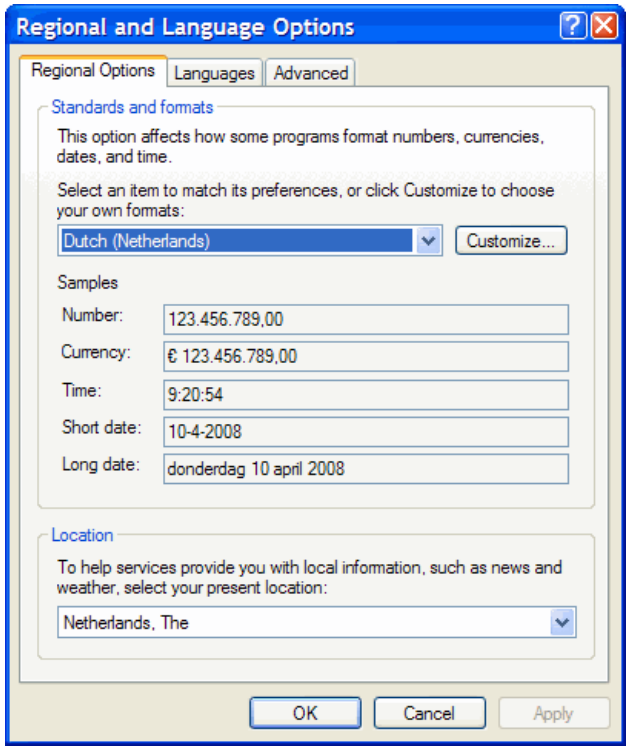
Option 2

1. In SolidWorks, export to an *.STL file
2. Click the button **Option** in the dialog box **Export** and select **Save all components of an assembly in a single file**
3. Convert the exported *.STL file to DXF by using Triangle2DXF, you can download this tool from the Xtranet.

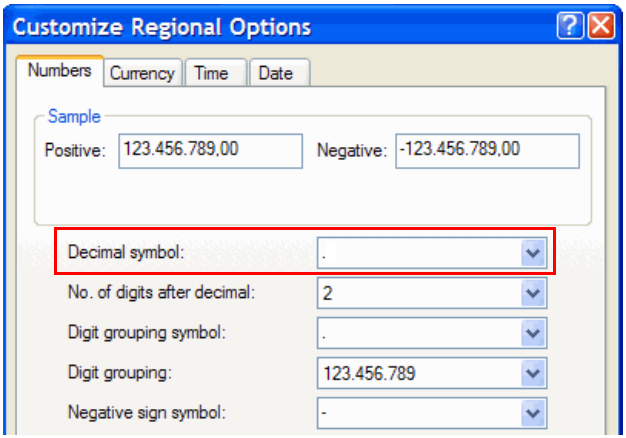


This only works if the option **Decimal symbol** is set to "." (point) in your regional- and language settings. Commas for example won't work.

In Windows, go to **Start** > **Settings** > **Control Panel** and double-click on the category **Regional and Language Options**, the following dialogbox appears:



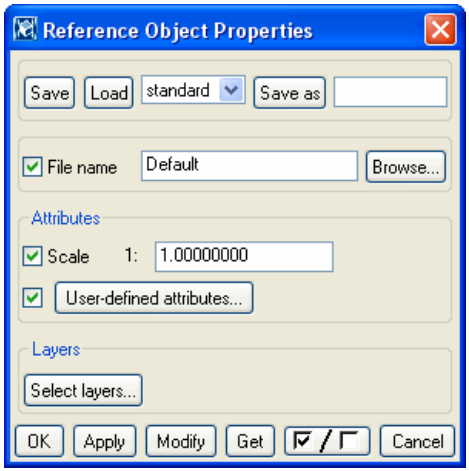
Click the button <**Customize**>, the following dialogbox appears:



Set the option Decimal symbol to "." (point). Click <**Apply**> and <**OK**> and again <**OK**>.



4. In Tekla Structures you import the DWG file as a reference object, go to **Parts** > **Insert Reference Model..** or double-click the icon, the following dialog box will open:

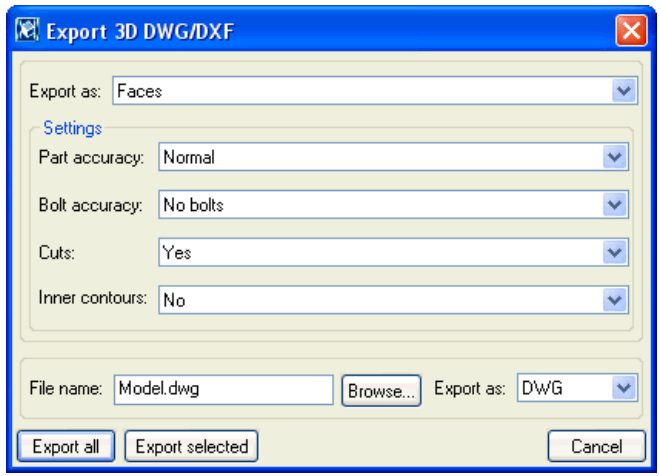


5. Click on the button <**Browse>** to select the DWG/DXF file
6. Click <**Apply>** or <**OK**> and pick a position in the model, the DWG/DXF file will be imported.

Exporting from Tekla Structures

Step plan

1. In Tekla Structures go to **File** > **Export** > **3D DWG...**, the following dialog box will open:



Tekla Structures parts can be exported in 2 formats: *faces* and *lines*.

- If *faces*-export is used, the profiles are exported in it's entirely, in other words, all lines that are visible in the Tekla Structures model, will be visible in the dwg. However, profiles are exported as 'blocks'.
- If *Lines*-export is used, the profiles are exported as a single line. The remainder options, like accuracy, cuts, inner contours, etc, speak for themselves. Take into account that every option affects the size of the export file.

In the **File name** input field, the name of the export file can be entered. Note that the file extension has to be added to the filename. By default, the file is stored in the model folder. By pressing the <**Browse>** button, an alternative location can be selected.

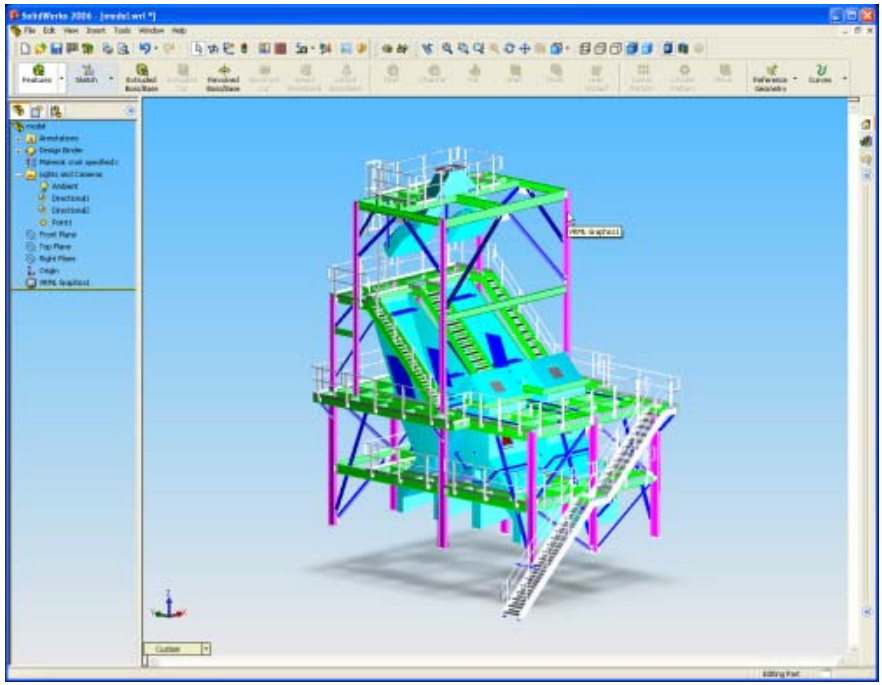
The last option is determining what has to be exported, the entire model or just a selection of it. This is controlled with the <**Export all>** and <**Export selected>** buttons.

2. In SolidWorks you can now import the 3D DWG file

Using Tekla Structures model as reference in SolidWorks

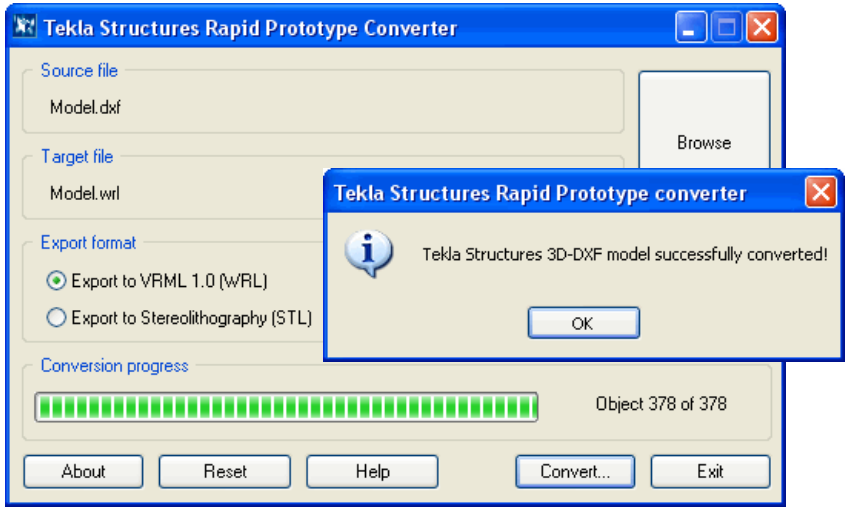
In case you want to use a Tekla Structures model in SolidWorks as a reference model, you can make use of the Tekla Structures Rapid Prototype Converter, which allows you to convert Tekla Structures 3D-DXF models to a VRML file.

Having the converted VRML model you can import this file format to SolidWorks. As you can see from the example (converted steel construction) you get the complete 3D model into SolidWorks.



That kind of reference model in SolidWorks is of type graphic, that means you cannot access the particular elements of the imported VRML file, but you get quite a fast and easy solution to check the Tekla Structures model with other SolidWorks generated models.

Before converting to VRML or STL format you need to export the Tekla Structures model, or parts of it, to 3D-DXF, using the Tekla Structures export functionality. The converter will then, according to user settings, convert the 3D-DXF data to VRML (option to generate compressed VRML) or to STL format.



You can download the Tekla Structures Rapid Prototype converter from the Xtranet.

3.45 Tekla Structures STEP IGES Converter

Topics

This chapter is divided into following sections:

Description (p.432)

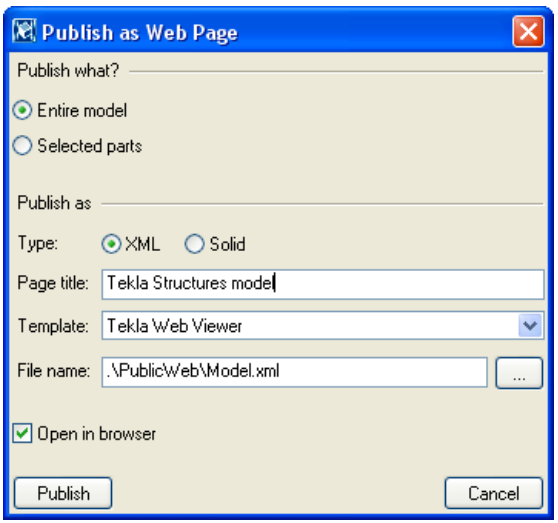
Steps (p.432)

Description

STEP and IGES are common file formats for Mechanical CAD applications, and commonly used in this branch. The document will explain how to generate STEP or IGES files bases on Tekla Structures WebViewer models, to be used with MCAD applications like Autodesk Inventor, SolidWorks, UGS SolidEdge, PTC PRO/Engineer, Dassault Systems Catia and others.

Steps

Based on Tekla Structures WebViewer XML models the Tekla Structures converter will generate STEP and IGES files. A WebViewer XML model will be generate within Tekla Structures, from the menu 'File' select 'Publish as Web page'.



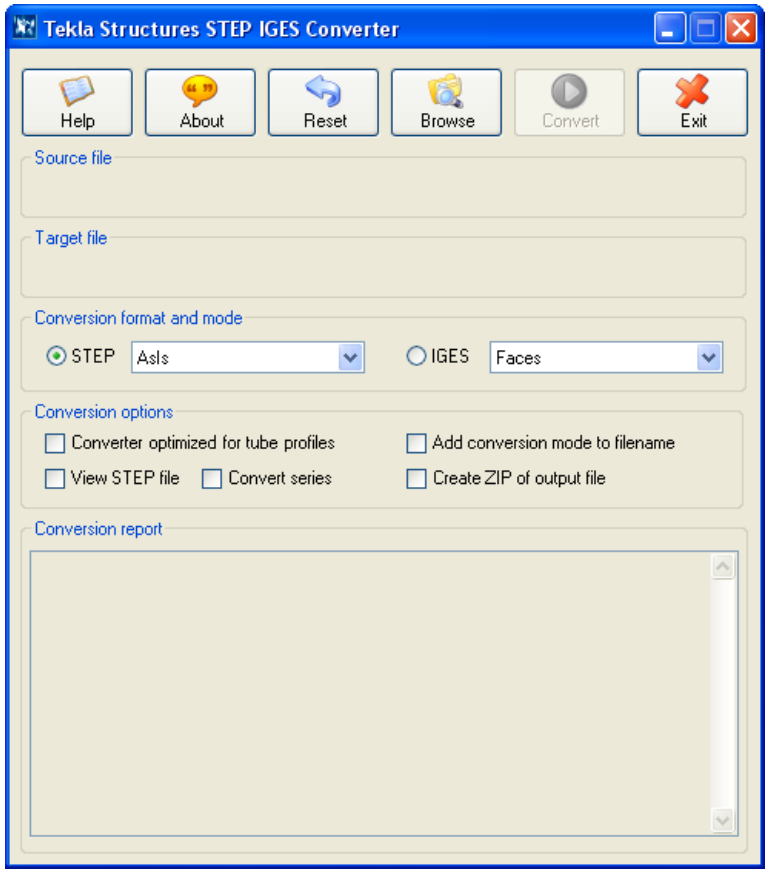


The complexity of the Tekla Structures model is very much related to the later file size of the STEP and IGES file. It is not recommended to export very large Tekla Structures models as one WebViewer model, rather than splitting the export into several single files.



Furthermore the accuracy settings in Tekla Structures will have a direct impact to the quality of the resulting STEP and IGES files. Eventually it is required to decrease the amount of segments of tubes in Tekla Structures, also the amount of vertex points of part cuts. Please see the Tekla Structures online help for the corresponding variables having influence to this or ask your local Tekla Structures support.

After the WebViewer XML export model has been generated the Tekla converter can be started to convert to STEP and IGES. Starting the application ' TeklaStructures_StepIges_Converter_GUI.exe' the following dialog will appear.



The handling of the converter is very much self-instructional, therefore here the essential information how to use it. To perform a conversion a Tekla Structures WebViewer XML file is needed, open the file selection dialog by clicking on the button 'Browse'. After a file has been selected, the button 'Convert' will be enabled. The box 'Source file' will show the selected Tekla Structures XML file for conversion, the box 'Target file' shows the name of the output file, according to the selected output format. Information on the actual conversion will be reported to the box 'Conversion report'.



The conversion to STEP and IGES format will take quite a long time when Tekla Structures models are rather big and complex. During the conversion the cursor will change to a hourglass symbol, please be patient, and do not cancel the conversion process.

Conversion format and mode

For STEP export the following modes are available:

- AsIs
- ManifoldSolidBrep
- FacetedBrep
- ShellBasedSurfaceModel
- GeometricCurveSet

For IGES export the following modes are available:

- Faces
- Brep

It depends very much on the target MCAD application which format and mode the most suitable will be, we recommend to test this issue with some example files also provided with the converter.

Conversion options

- **Converter optimized for tube profiles**

  In case the Tekla Structures model contains a lot of tube profiles the option 'Converter optimized for tube profiles' will generate better result in STEP and IGES format. It is not necessary to enable this option when having tube profiles to convert, but according to our tests certain MCAD application do handle tube profiles better when using this kind of optimization.

- **Add conversion mode to file name**

  This function will add to the selected mode to the output filename. For instance the Tekla Structures WebViewer XML model named 'Model.xml' shall be converted to STEP with mode 'AsIs', the resulting filename would be 'Model_AsIs.stp'.

- **View STEP file**

  In case the freeware STEP viewer CadFaster | QuickStep is installed on the computer the Tekla Structures STEP IGES converter is running, the resulting STEP files can be directly viewed after the conversion. The STEP viewer can be downloaded here: https://ssl.cadfaster.com/shop/quick-step.php.

- **Convert series**

  This is a fast way to convert to all kind of STEP and IGES format and mode with one click. For instance the Tekla Structures WebViewer XML model named 'Model.xml' will be converted to

  - 'Model_AsIs.stp'
  - 'Model_ManifoldSolidBrep.stp'
  - 'Model_FacetedBrep.stp'
  - 'Model_ShellBasedSurfaceModel.stp'
  - 'Model_GeometricCurveSet.stp'
  - 'Model_Faces.igs'
  - 'Model_Brep.igs'

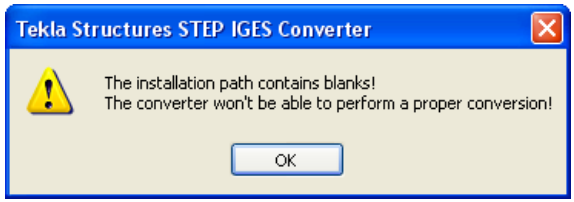
  This is very much of help when providing the complete set of STEP and IGES files as a package, much faster than performing the conversion step by step.

- **Create ZIP of outout file**

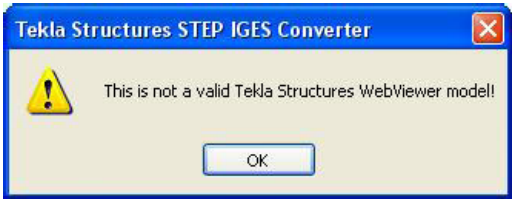
  The resulting STEP and IGES file will be compressed as ZIP file, this function creates a ZIP of each file to convert also when converting a series.

Additional information

It is recommended to install the Tekla Structures converter in a folder without blank in its path name, to ensure the conversion will work as designed, otherwise the following information will be shown:



The converter will also test if the selected XML file is a valid Tekla Structures WebViewer model, if not the following information will be shown and the conversion will be stopped:


Tekla Structures STEP IGES Converter
This is not a valid Tekla Structures WebViewer model!
OK

3.46 Stairs and railings

Topics

This chapter is divided into following sections:

Stairs (p.437)

Stringer details (p.443)

Railings (p.445)

General (p.471)

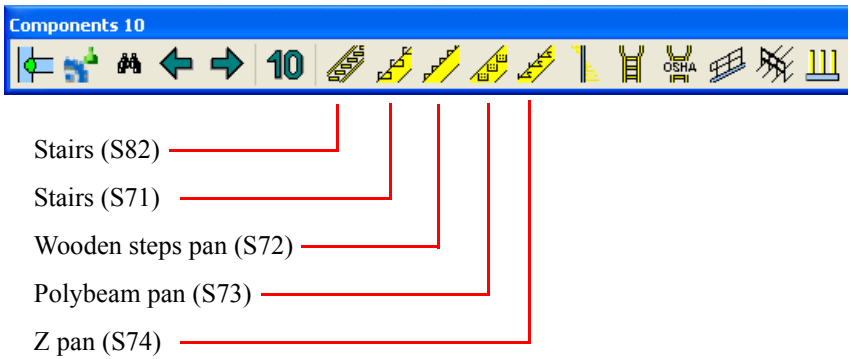
This document guides you through all available information about straight stairs and handrails.

You can also download a model from the Xtranet. This model is separated into phases. These phases can be made visible by using filters.

Lots of improvements have been added for stairs and handrails. There are also lots of settings added to work quicker and more effective with stairs and handrails.

Stairs

You can find these components by pressing the icon **Open component catalog** on the toolbar **Components** or you can use shortkey <**Ctrl** + **F**> to open the dialog box **Component catalog**:



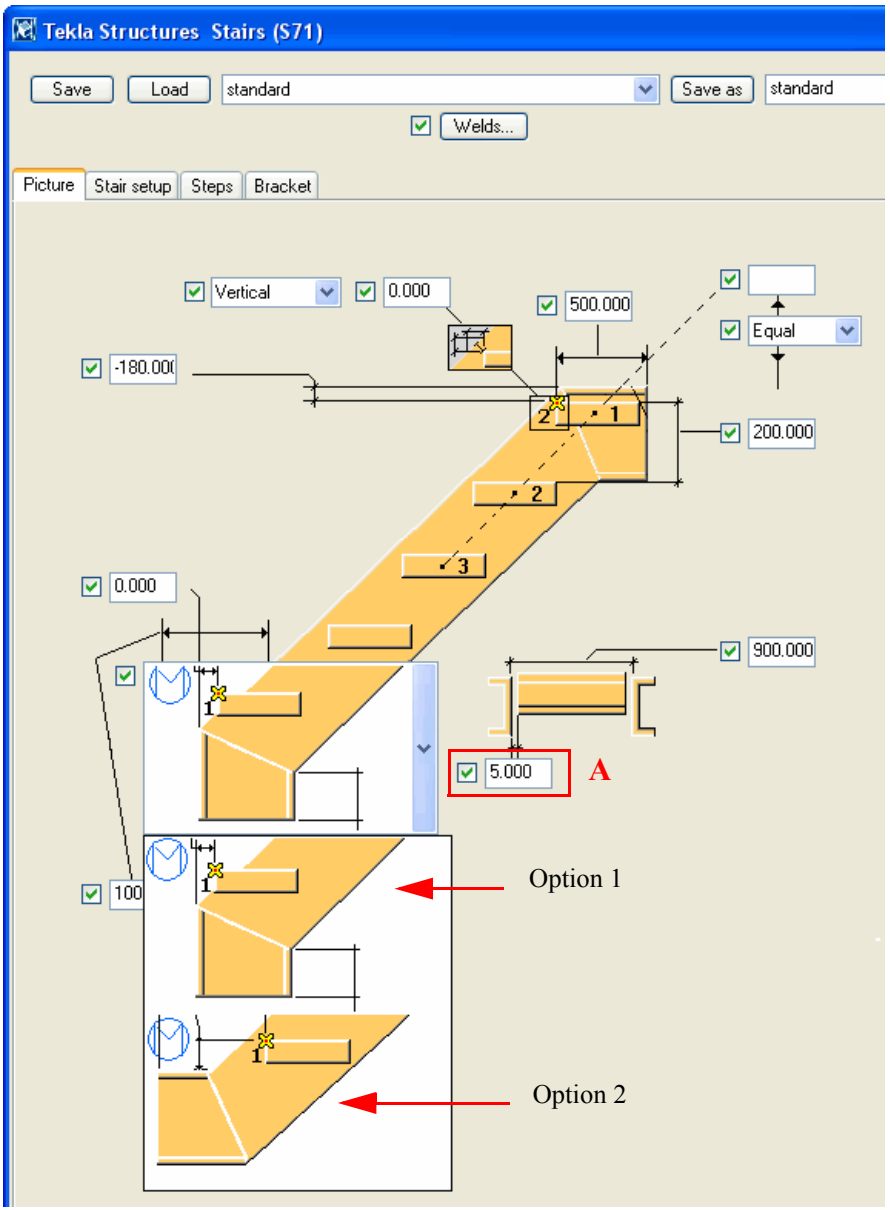
Select a start- and endpoint to place these stairs components.



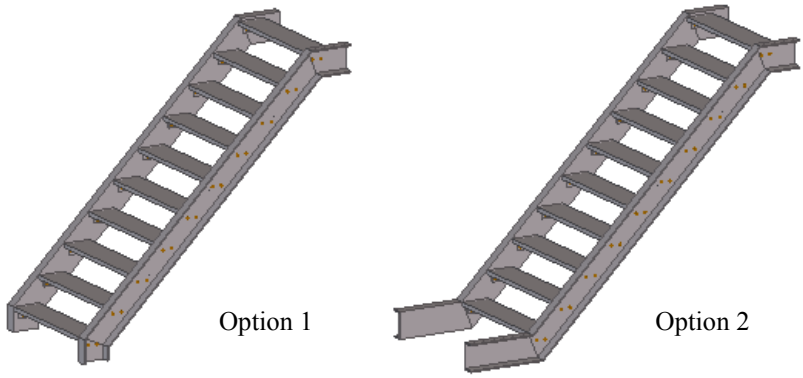
When you move a start- and endpoint of the stairs, the stairs component will be recalculated. In this way the stairs can be modified quickly.

Stairs (S71)

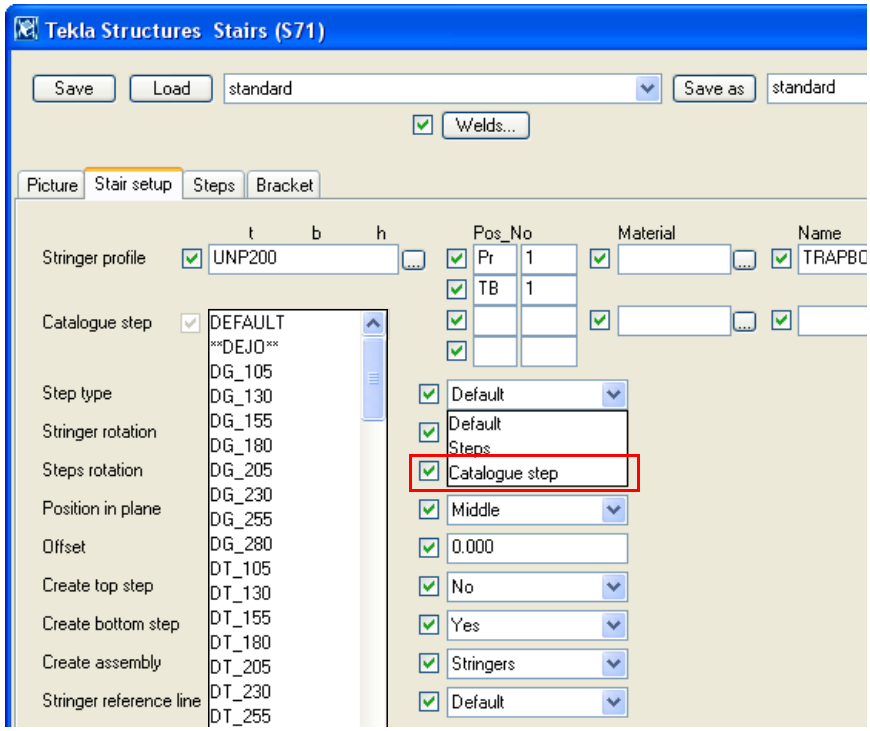
The tab **Picture** includes 2 de options to set the stair base:



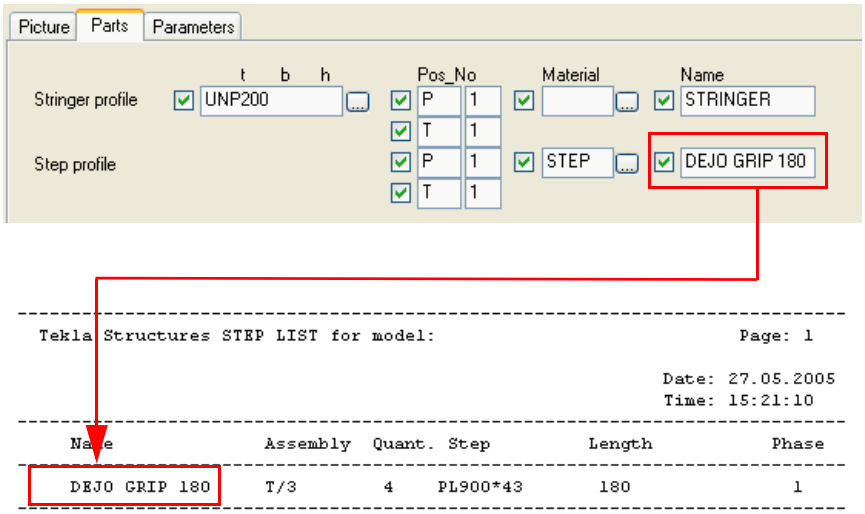
At input field **A** in this tab you can also enter the tolerance between the step and the stringer profile.



On the tab **Stair setup** you can select to use the catalogue steps, for this, select the option **Catalogue step** in the list box:



When you also enter the correct name for the material and the name of the steps in the component, you can create the report **step_list** which includes all steps:



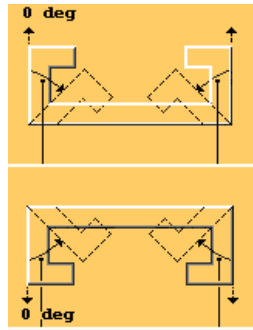
This report only displays the parts using the material STEP.



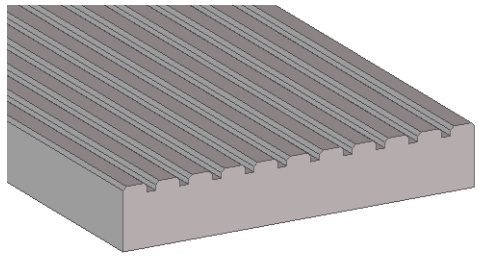
In the file **steps.dat** in the system folder, the step profiles of component **Stairs (S71)** are defined. **Never** copy this file to another Tekla Structures version!

When you select **Steps** in the list box *Step type* you can determine the performance and the dimensions of the steps by yourself on the tab **Steps**. See the following pictures:

2 options to create a step profile, using cross section U or C+, enter the values to define the contour.



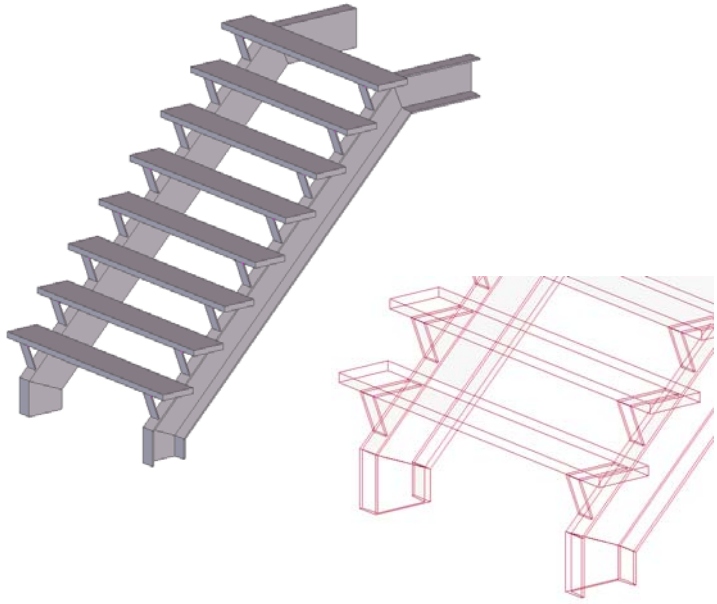
Option to create a wooden step profile, in the tab *Parts* you van select the step profile.



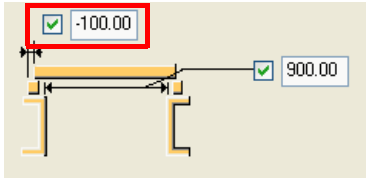
When you want to make use of a divergent step profile (see picture) you first add the profile to the profile catalog. Now you can select this step profile in the component.

Wooden steps pan (S72)

This component generates a stair which includes step brackets to connect the wooden step profile. The brackets contain holes to connect the wooden step profiles by means of head wood screws. See the following picture.



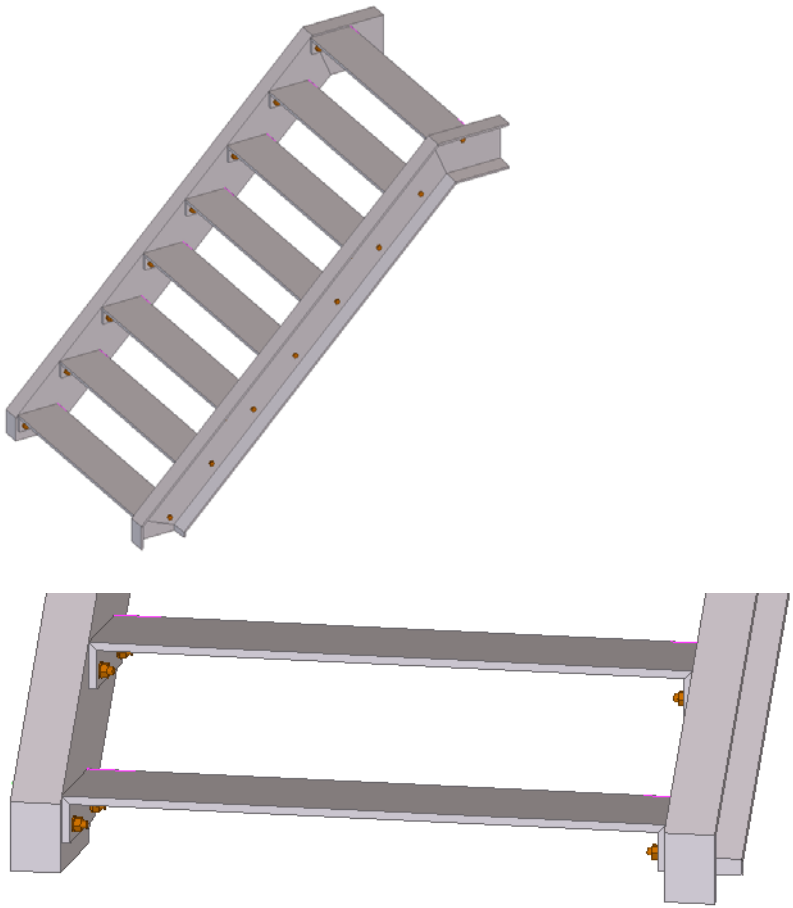
The way to make use of step profiles containing a divergent cross section is equal to component **Stairs (S71)**.



Make use of this parameter to add an extra length compared to the outside of the brackets

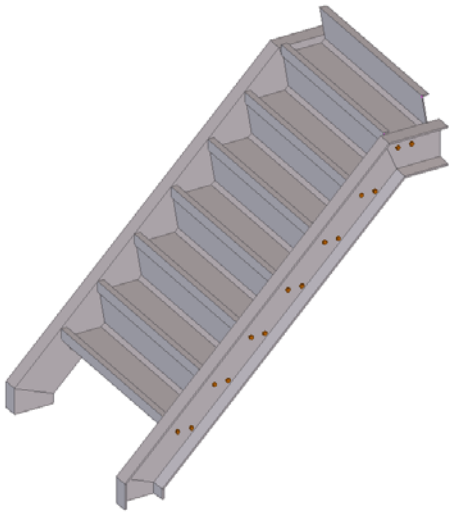
Polybeam pan (S73)

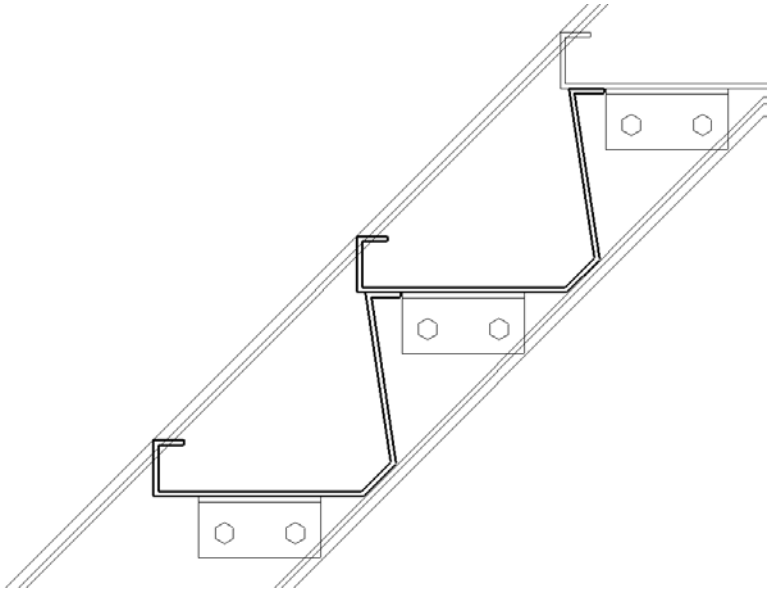
This component generates a stair containing a step profile as a folded plate with a U-shaped contour:



Z pan (S74)

This component generates a stair containing a step profile as a folded plate with a Z-shaped contour.

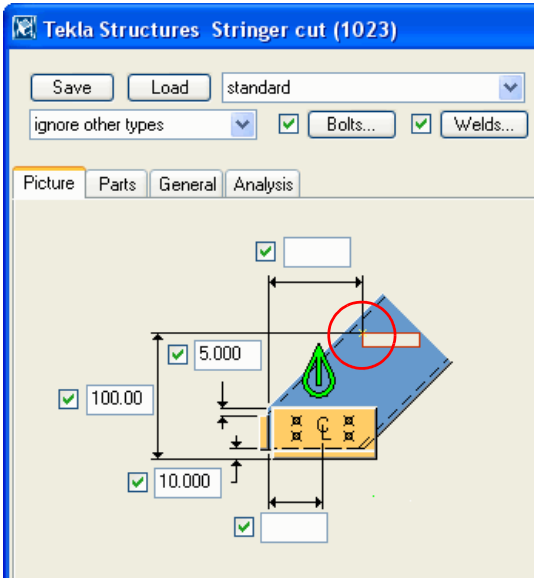




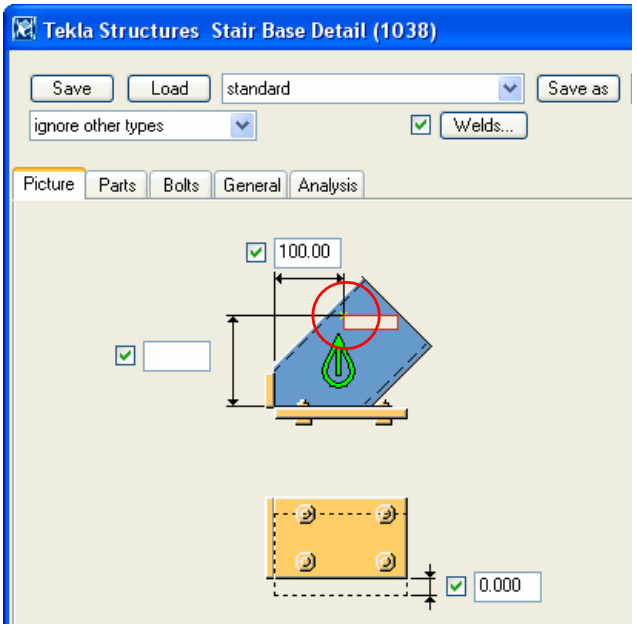
Stringer details

Lots of customer make use of stringer details in Tekla Structures. Use the point, shown in the picture, for positioning the components:

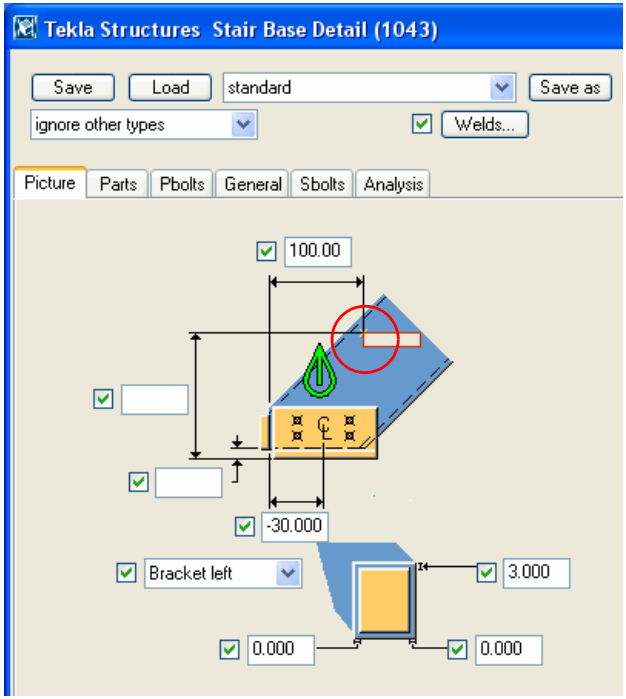
Stringer cut (1023)



Stairs detail (1038)



Stair Base detail (1043)



Improvements in functionality

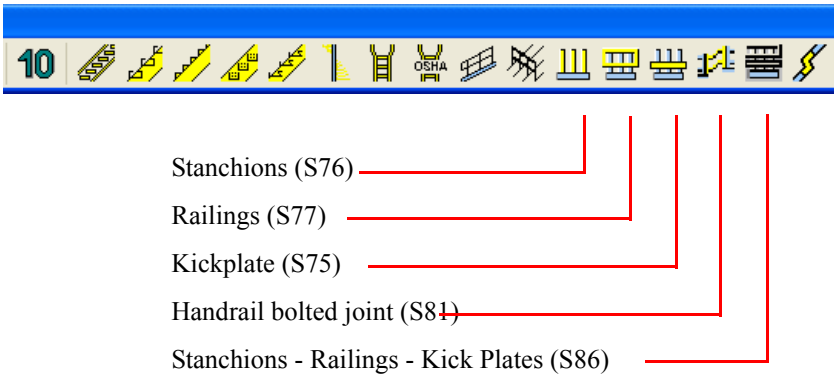
The improvements in the dialogs results in better stair details.

The selected point determines the side where the detail is being placed. If a point on the right side is picked, the detail is created on the main parts left side.



Railings

In Tekla Structures there are several components available to create a railing. You can find these macro's on page 10.

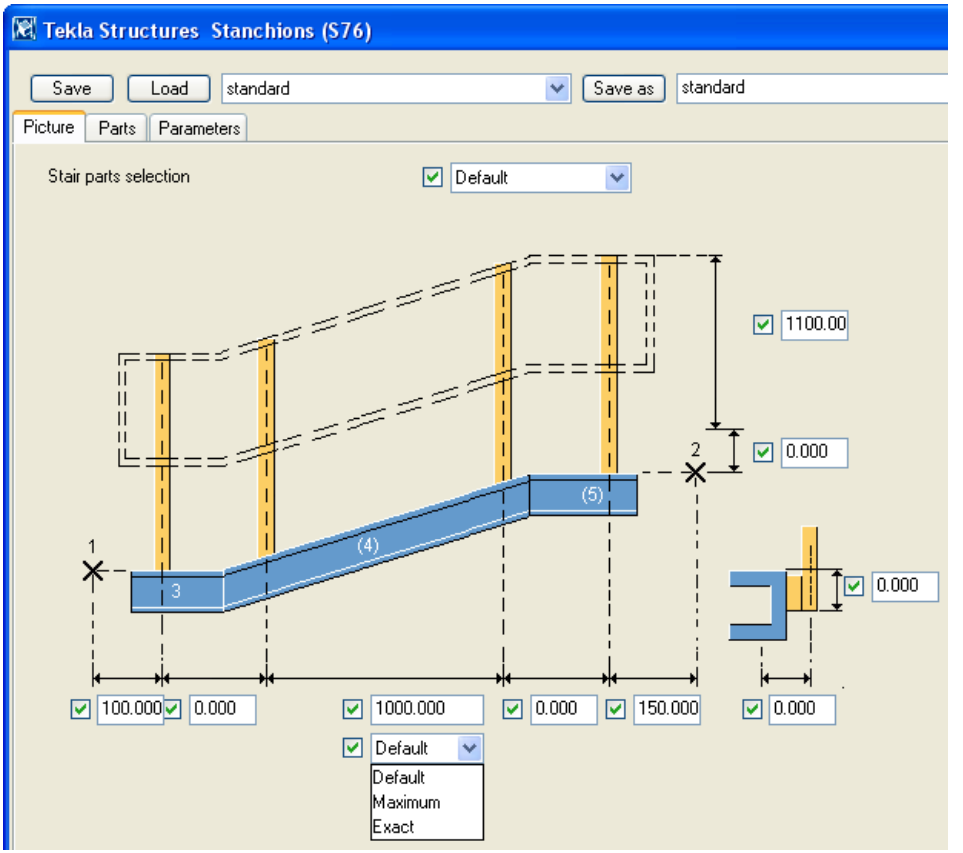


The components **Stanchions (S76)**, **Railings (S77)** and **Kickplate (S75)** can create one entire railing.

The components **Handrail bolted joint (S81)** and **Stanchions - Railings - Kick Plates (S86)** can create one entire railing or a complete connection between two railings. These components make use of the components **Stanchions (S76)**, **Railings (S77)** and **Kickplate (S75).**

Stanchions (S76)

This component only generates stanchions. With or without base plate depends on the selected setting.

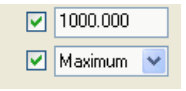


To place the component you need to follow the picking order in the dialog box, so first select a start and endpoint, then you select the profiles to divide.

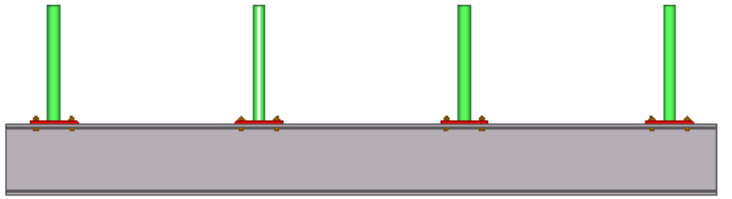
Depending on the option in the list box *Stair parts selection*, the stanchions will be divided over the selected parts when you make use of the setting **Use selected** or;

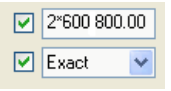
When you select a part by using the option **Use all** and this part belongs to a stringer which is created with one of the stairs components, all parts which belong to this stringer will be used to divide over the stanchions.

Dividing the stanchions can be set in two ways:

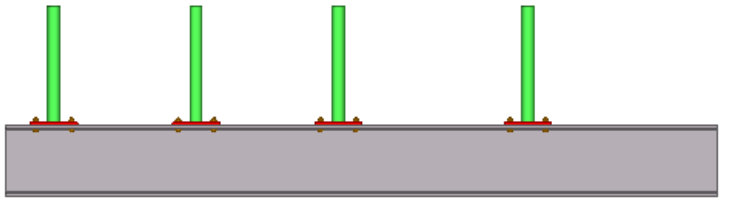


Maximum: Divides the distance between the first and the last stanchion in equal values, these values are smaller then the entered value.

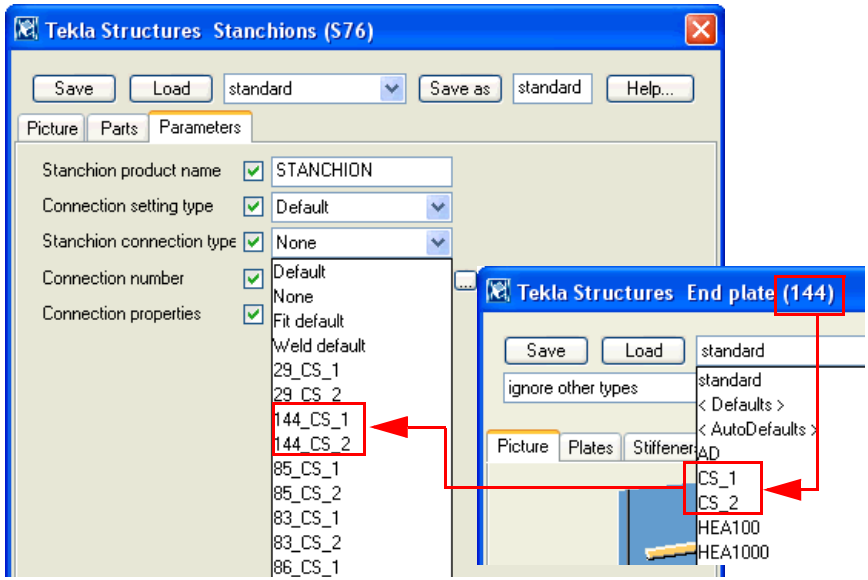




Exact: To place the stanchions by using the exact values from the input field.

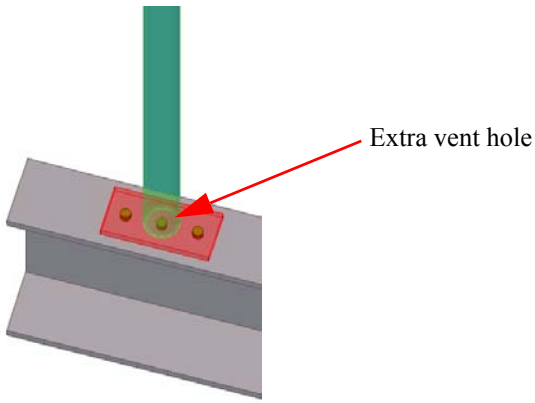


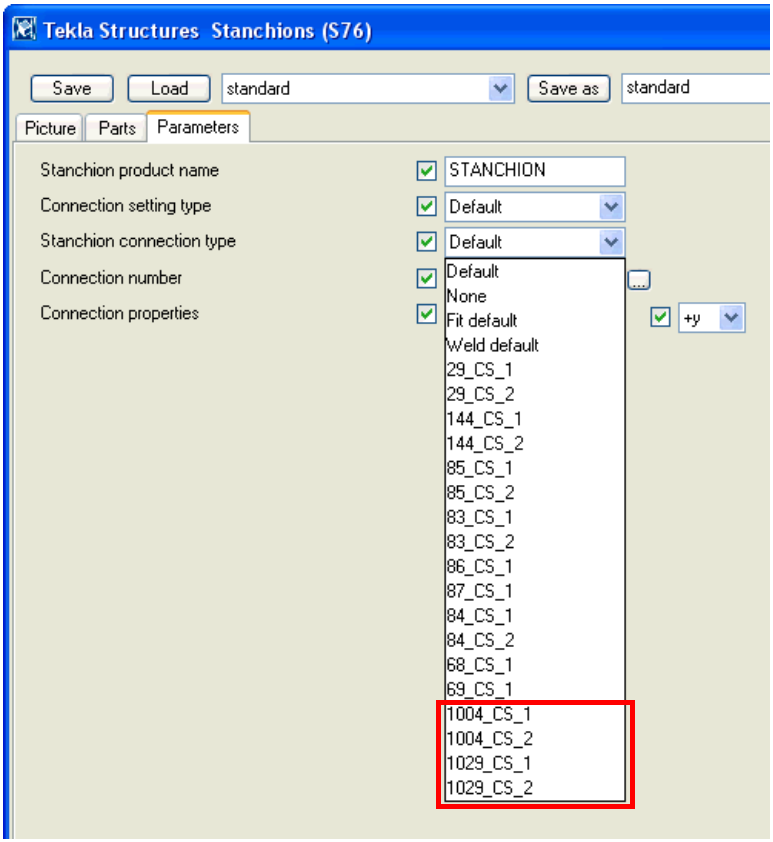
In this component you can call other components and details to connect the stanchions to the stringer. These other components are displayed in the model view. In case you explode a "main" component, the "detail" components will not explode.



When you desire other component settings CS_1 and/or CS_2, you can easily change these setting and save it by making use of the name CS_1 or CS_2.

When you make use of the setting 1029_CS_1 or 1029_CS_2, you can also create a vent hole.





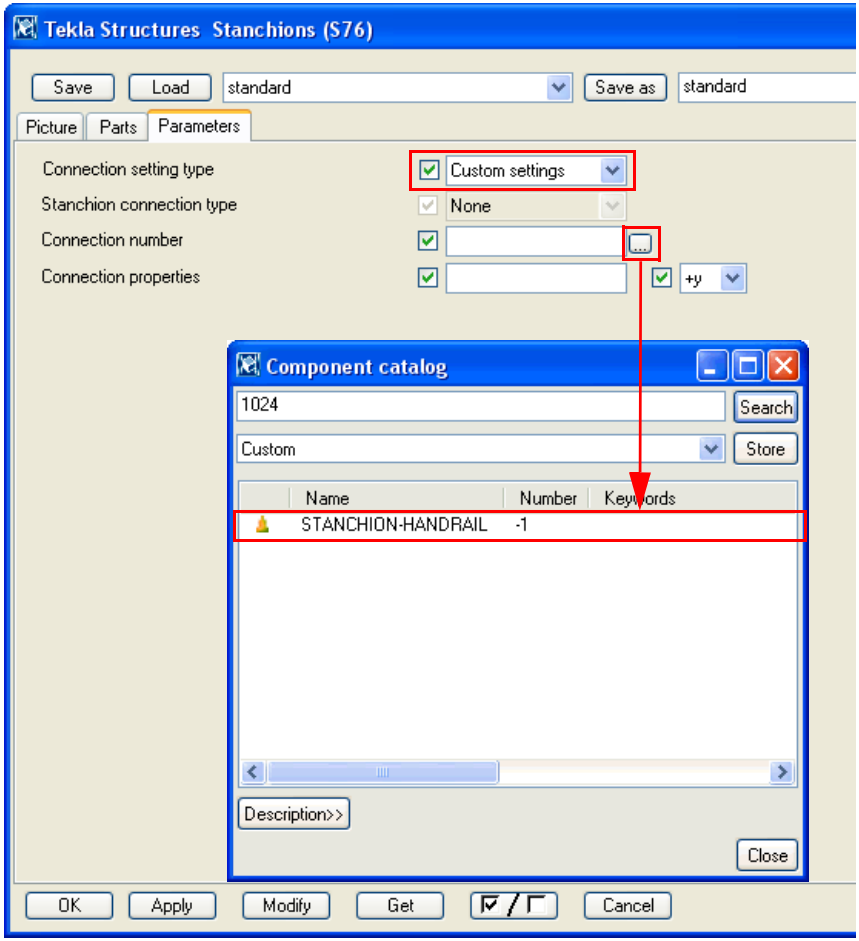


This component makes use of connections (components **End plate (29)**, **End plate (144)** and details (components **Base plate (1004)** en **Base plate (1029)**.

Details only work correct when the stanchions are positioned perpendicular to the stringer profiles.

You also use the setting from component **Base plate (1004)** and **Base plate (1029)** for stanchions placed on concrete parts.

You can also use custom components. For this, in the list box **Connection setting type** set the option to *Custom settings*.



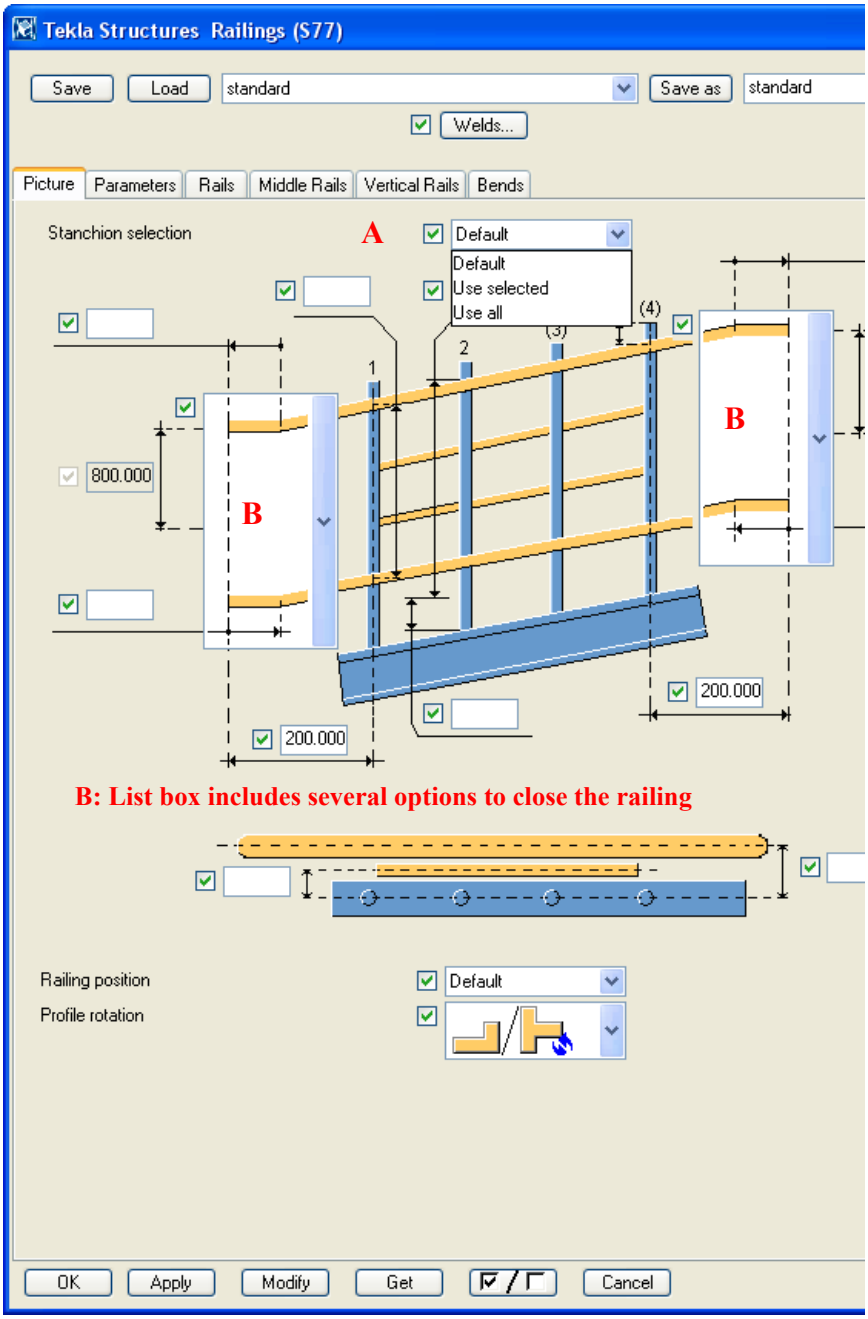
Railings (S77)

Component **Railings (S77)** makes use of already generated, possibly with component **Stanchions (S76)**, stanchions and creates hand- and knee rails.

Placing the component depends on the selected option for list box **A**.

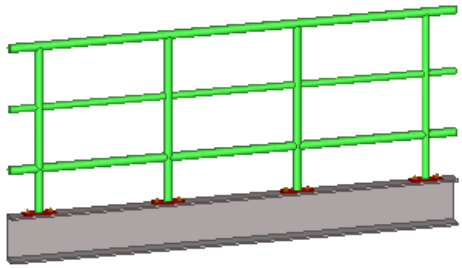
Using the option **Use selected** a railing will be created including stanchions, selected by the user.

Using the option **Use all** you only need to select one stanchion. If this stanchion was created with component **Stanchions (S76)**, all stanchions will automatically be used for the railing.

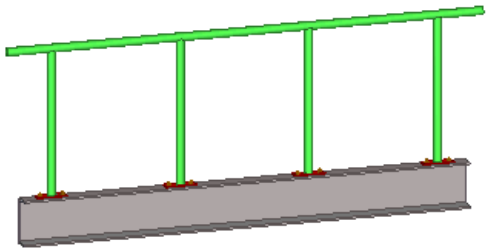


For component **Railings (S77),** the following settings are added by default:

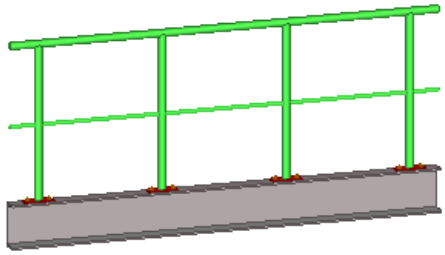
Standard



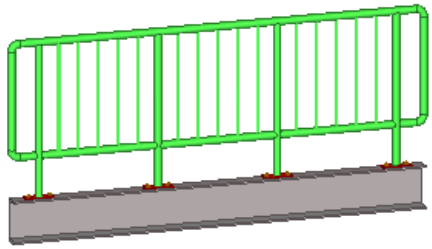
Handrail only



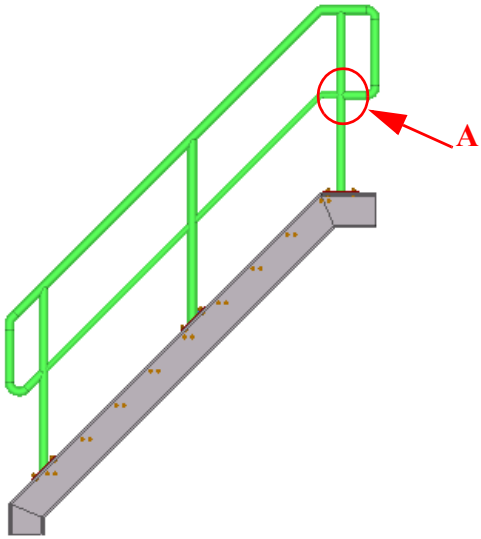
Handrail + knee-rail



Barred fence

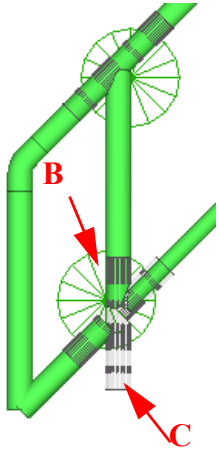


Akzo



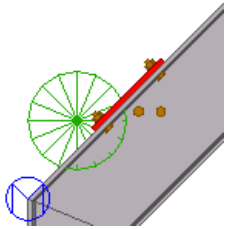
To create this type of railing correctly, you do the following:

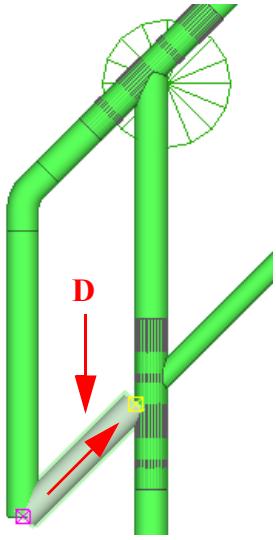
1. Place component **Stanchions (S76)** and next component **Railings (S77)** by using setting "Akzo".
2. Make use of component **Round tube (23)** and setting **S77** on the topside of the railing for the stanchion - middle rail connection (**A**):



Delete component **Round tube (23)** on the bottom side of the railing at the first stanchion (**B**) and one part cut (**C**) of the stanchion.

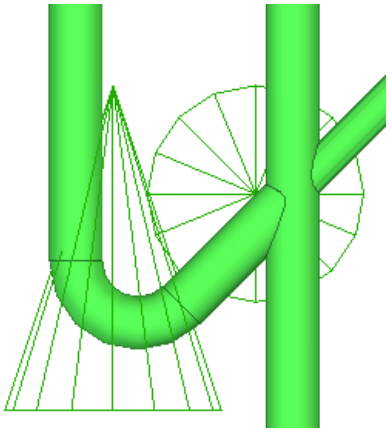
(It removes the solid error from the first stanchion).





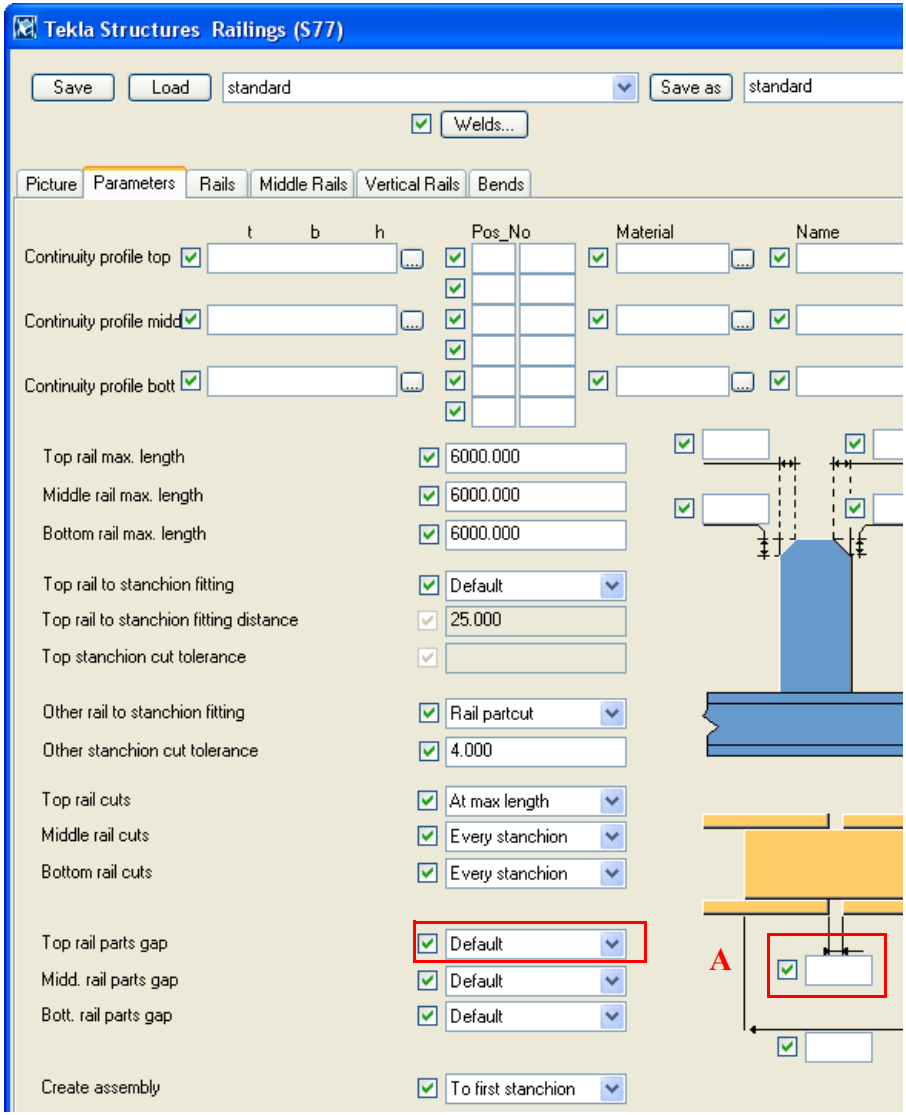
Next set the work plane to the knee rail (**D**) of the railing.

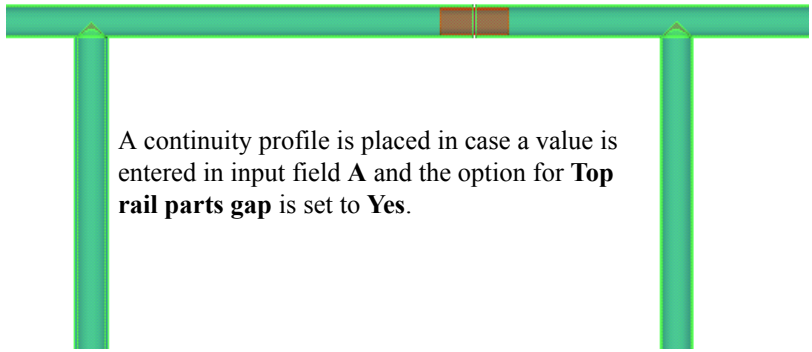
Move this little piece about 50 mm in the railings' direction.



Now use component **Railing plane to plane (90)** and setting **48.3-3S** for the end detail, and component **Round tube (23)** for the connecting the knee rail to the stanchion.

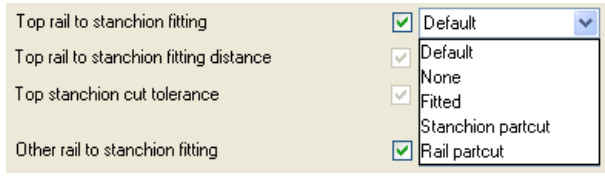
Tab Parameters


Tekla Structures Railings (S77)
Save
Load
standard
Save as
standard
Welds...
Picture
Parameters
Rails
Middle Rails
Vertical Rails
Bends
t
b
h
Pos_No
Material
Name
Continuity profile top
Continuity profile midd
Continuity profile bott
Top rail max. length
6000.000
Middle rail max. length
6000.000
Bottom rail max. length
6000.000
Top rail to stanchion fitting
Default
Top rail to stanchion fitting distance
25.000
Top stanchion cut tolerance
Other rail to stanchion fitting
Rail partcut
Other stanchion cut tolerance
4.000
Top rail cuts
At max length
Middle rail cuts
Every stanchion
Bottom rail cuts
Every stanchion
Top rail parts gap
Default
Midd. rail parts gap
Default
Bott. rail parts gap
Default
A
Create assembly
To first stanchion



A continuity profile is placed in case a value is entered in input field **A** and the option for **Top rail parts gap** is set to **Yes**.

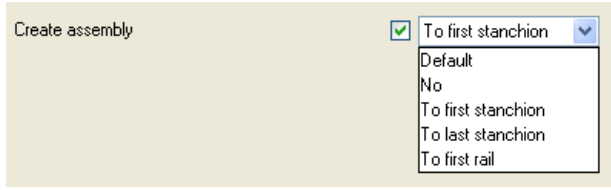
The top rail can be connected to the stanchions in several ways:

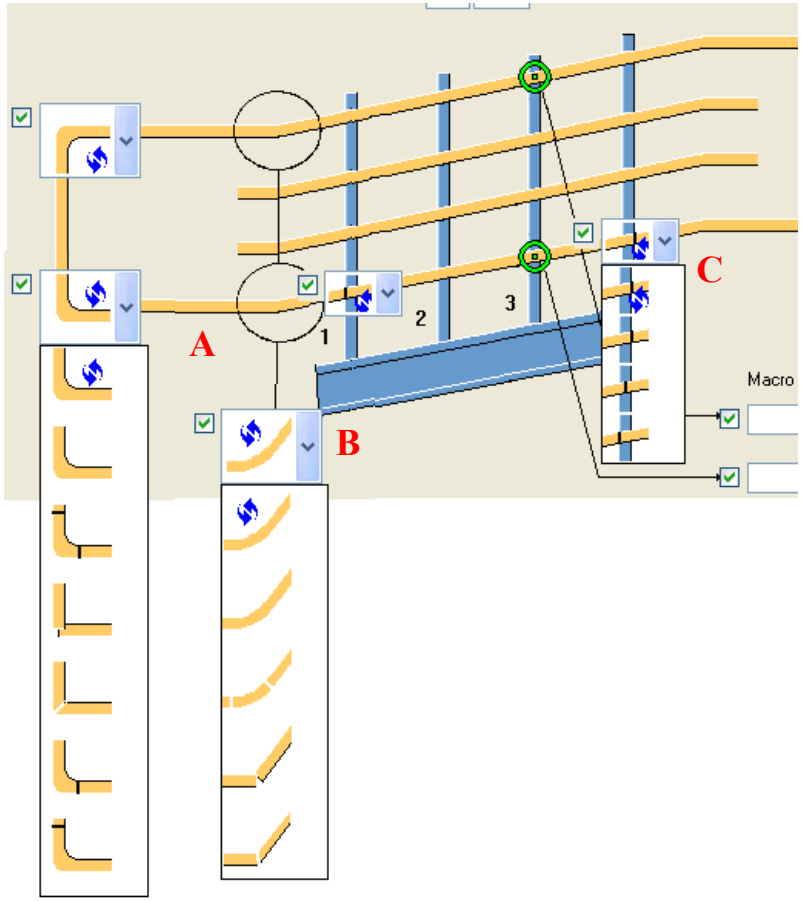


Also the setting **Other rail to stanchion fitting** includes several options:

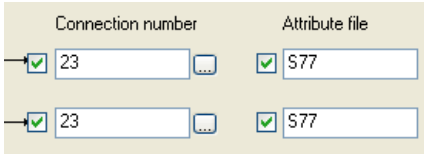


The option to define which part of the railing will be the main part of the assembly, affects how the railing will be positioned on the assembly drawing. If you define the stanchion as the main part of the assembly, it will also be positioned vertical on the drawing:



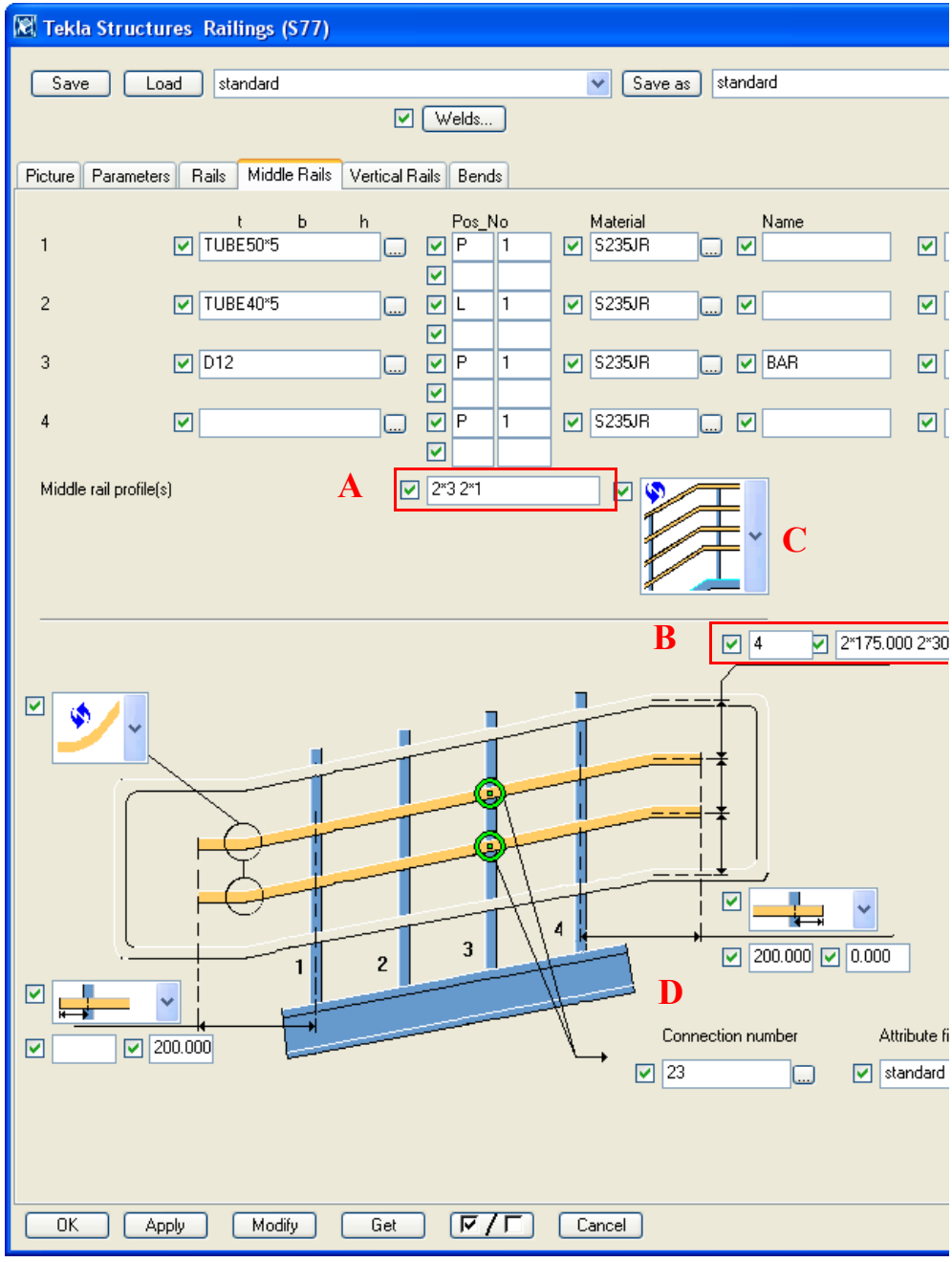


In the list boxes **A** and **B** you can set the bends for the top rail. In list box **C** you can define where the bottom rail stops referred to the stanchion.



This component includes an option to use other components to create the connections between the rails and the stanchions. E.g. component **Round tube (23)**. This component includes the setting **S77,** this to indicate that this setting is meant for component **Railings (S77)**.

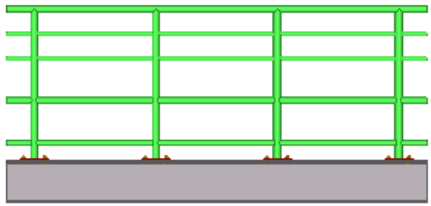
Tab Middle Rails



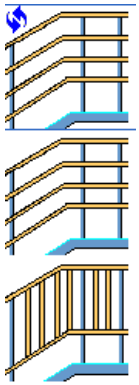
For **Middle rail profile(s)** at input field **A** you can set which profile you want to use. You can enter the profiles in input field 1 up to 4.

Example:	"3*2"	Creates 3 horizontal middle rails by using the profile entered in input field 2.
	"2*3 2*1"	Creates 2 horizontal middle rails by using the profile entered in input field 3 and 2 middle rails by using the profile entered in input field 1.

The options above depend on the numbers and the distances entered in input field **B**. You can also enter several centre-to-centre distances.



The example from the picture above results in 4 middle rails, 2 centre-to-centre distances of 175mm + 2 centre-to-centre distances of 300 mm



When you select the option **Vertical Rails** in list box C, you can define these rails on the tab **Vertical Rails**.

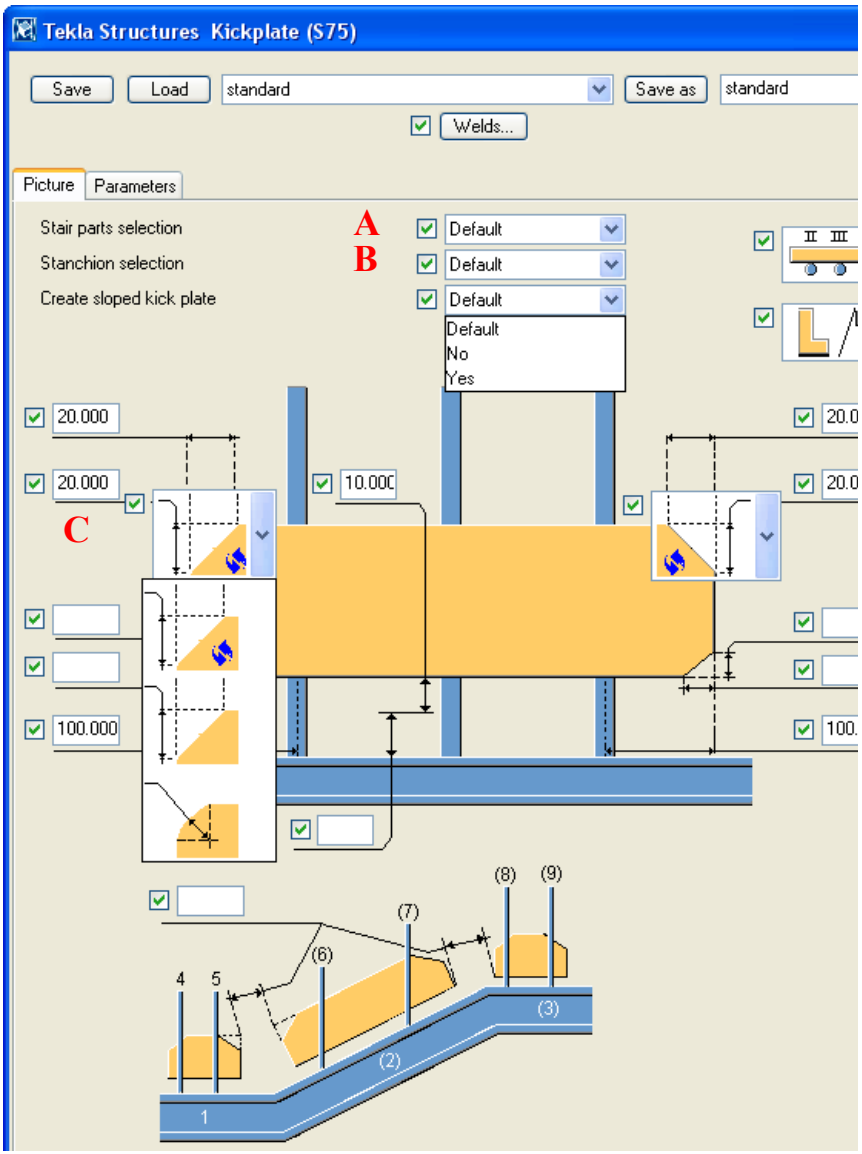


D: Here you can define a custom component. The picture shows an example of a railing including an extra hand rail, the bended tube between the stanchion and the rail is the connection (custom component).

Kickplate (S75)

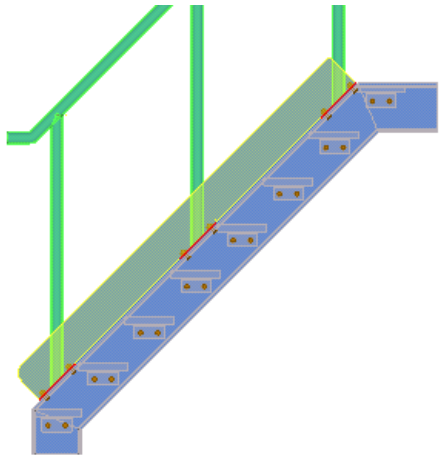
Component **Kickplate (S75)** uses existing stringers and stanchions, positioned manually or by using stairs- or stanchion components. A kick plate will be created by default if 1 part, belonging to a stringer (placed by using a stair component), is selected and 1 stanchion from component **Stanchions (S76)**.

In the list boxes **A** and **B** you can also select to use the selected parts. Follow the picking order in the component while creating the kick plate:



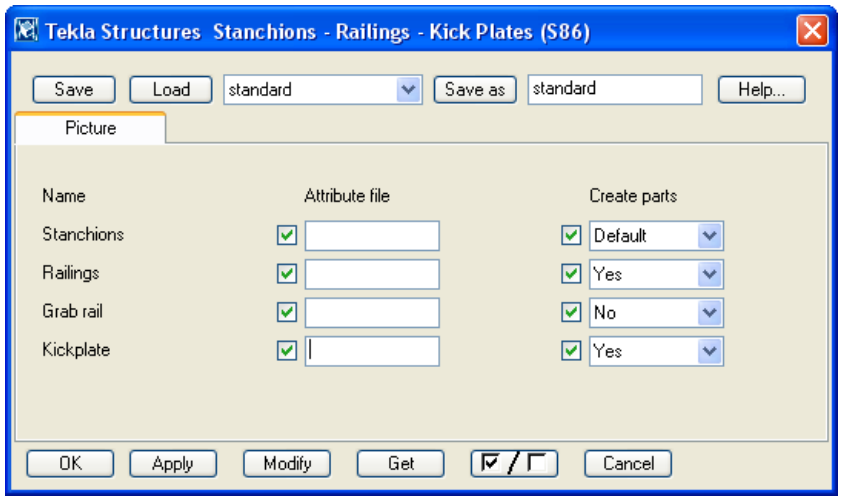
In the list box **Create sloped kick plate** you can select to create a sloped kick plate. When you select a sloped stringer profile, then the kick plate will follow this direction. Set to **Yes** by default.

In option **C** you can also select a chamfered kick plate:



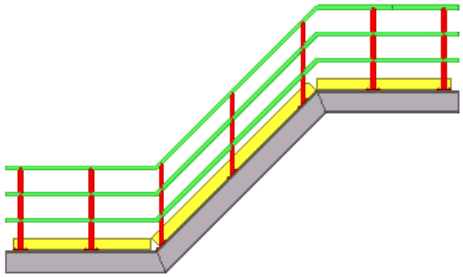
Stanchions - Railings - Kick Plates (S86)

Component **Stanchions - Railings - Kick Plates (S86)** creates a railing including stanchions, rails and a kick plate in one go.



You can place this component by picking a start- and endpoint. Next, you need to select the parts. The parts' direction must be equal!

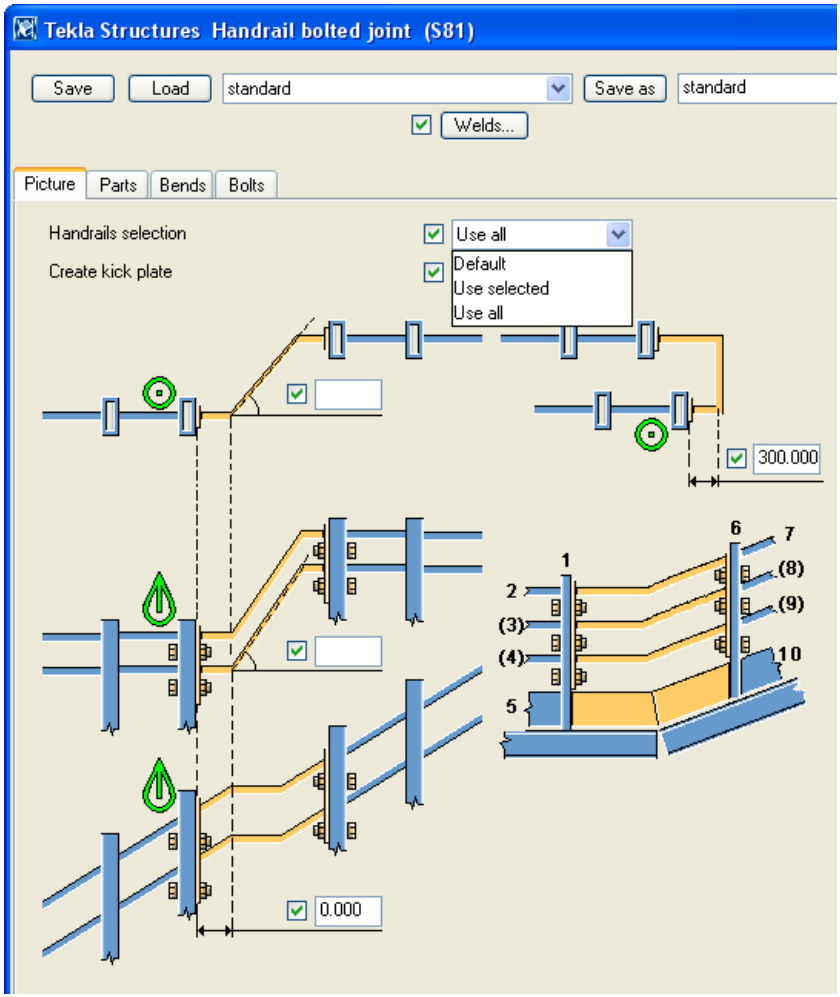
In this component you can use attribute files from the components **Stanchions (S76)**, **Railings (S77)** and **Kick Plates (S75)**.



If you want to modify the "loose" components later on, use <**Shift**> + Scroll (mouse wheel) to change the level of selecting so that you can double-click on the desired component.

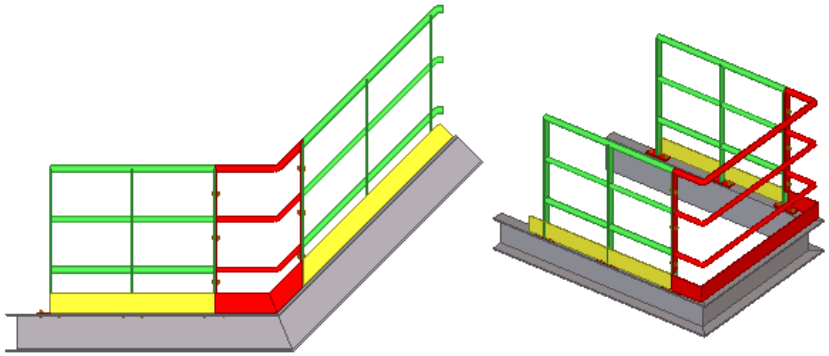
Handrail bolted joint (S81)

You can use this component to create a connection between two selected handrail- kick plate components by creating parts whore equal to the existing ones. These parts are welded between 2 vertical plates which are bolted to the existing stanchions. The handrails can be positioned in the same plane but also parallel or perpendicular to the plane.



Follow the picking order in the component. For **Handrail selection** you can select **Use selected** (with this only the selected rails are used) or **Use all** by which you only need to select one rail, and when this rail has been created with component **Railings (S77)**, the rails from that component will be used.

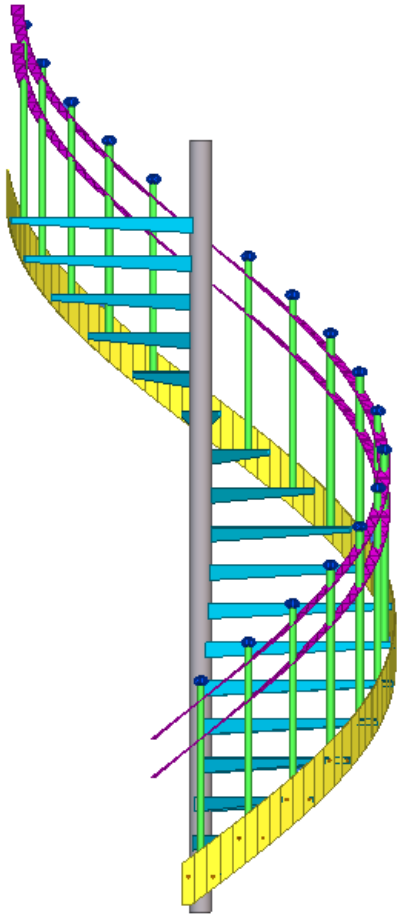
The following picture shows 2 different usages of this component:



This component does not suit railings including round stanchion profiles.

Spiral stair including a rolled stringer profile

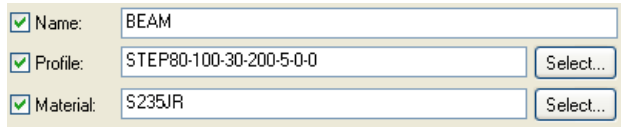
The following example shows how to model a spiral stair including a rolled stringer profile easily.



Step plan

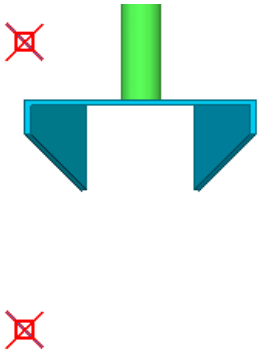
1. Place the main tube column and next the step profile by using the profile **STEP**; the length of the step determines the radius of the spiral stair. Then you can place a stanchion profile on the step



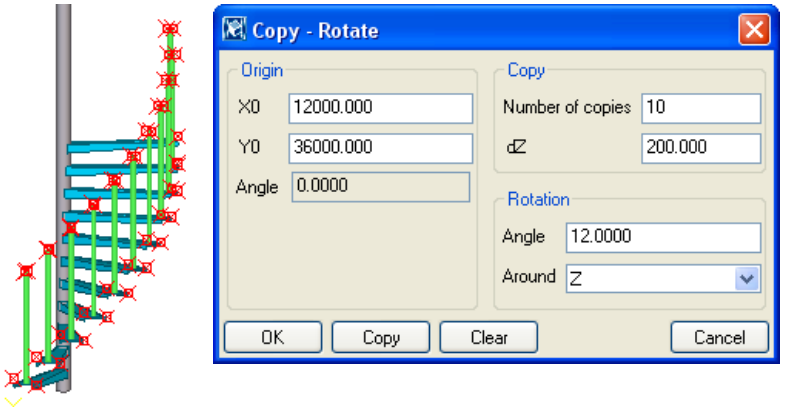


If needed, you can weld the parts together and you can use the command **Part cut** to cut the step profile in the main tube column.

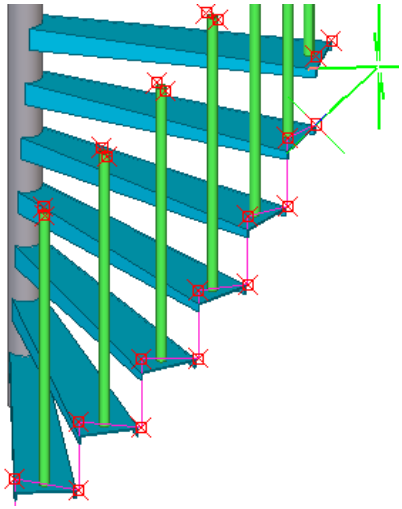
2. Pick a point at 50 mm above the top left corner of the step and copy this point 250 mm in the -Z direction, see the picture
3. Pick also 2 points on the upper side of the stanchion at 30 mm (left and right) from the centre of the stanchion, we will use these point later to position the railing and the stringer

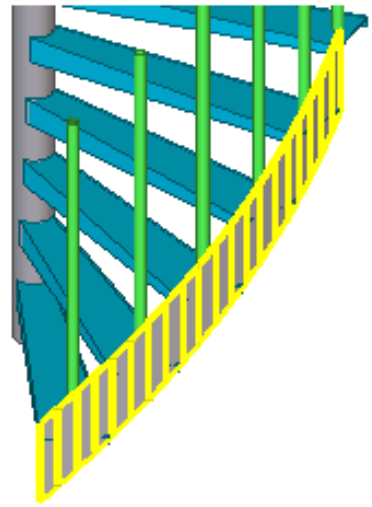


4. Copy and rotate the step, the stanchion and the picked points by using the command **Copy - Rotate** at the same time



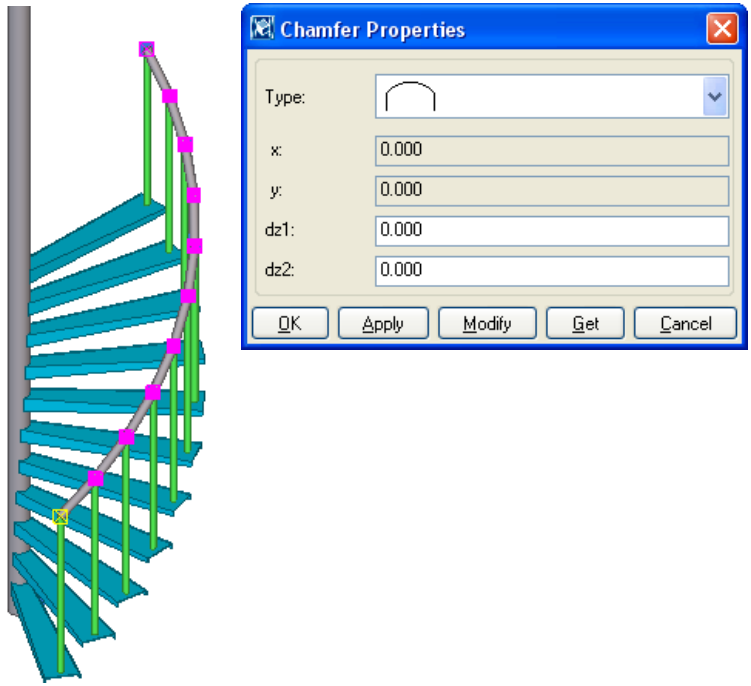
5. Now make use of component **Triangles generation (19)** to pick all copied points (zigzag), next select the middle mouse button



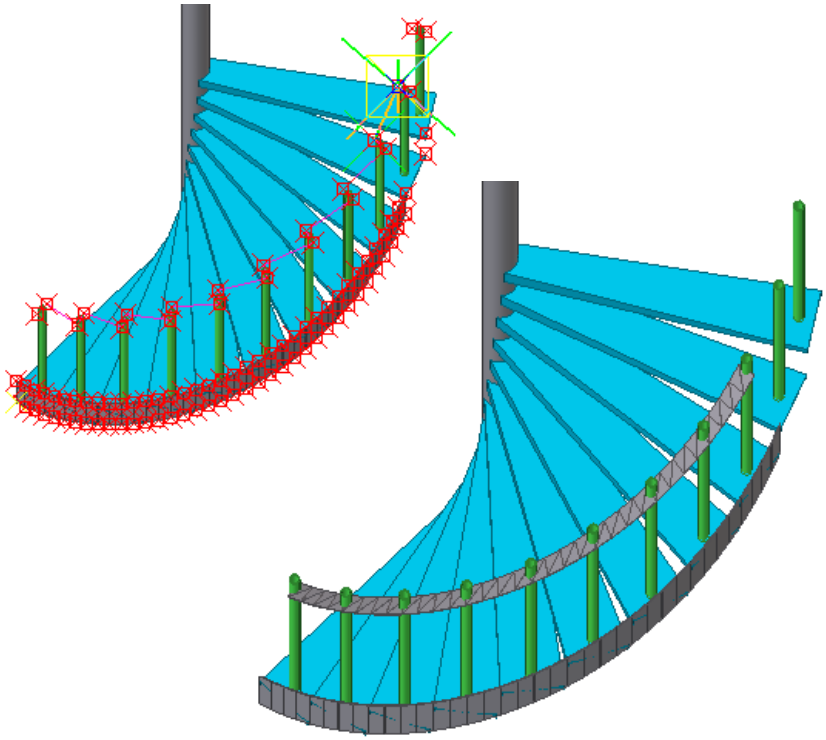


6. You can use a polybeam to create the handrail as a **tube**. Pick the centre of the stanchions and make sure that the tubes' position is set to **Middle**.

   After creating the polybeam you can chamfer the handles. The first and the last picked point don't need any chamfering, of course.



   When you use a **flat bar** to create the handrail you can again use component **Triangles generation (19)**. Make use of the picked points on top of the stanchions.



General

While modelling stairs and railings, the following components are handy:

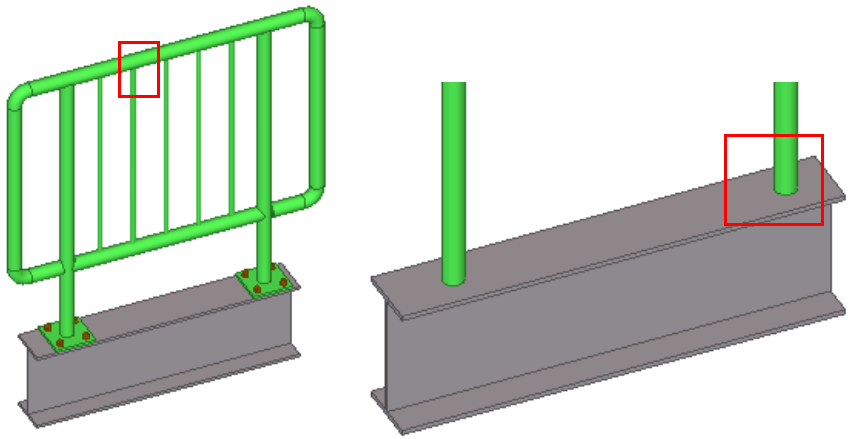


Component **Stanchion weld (85)**

Supplied settings:

- standard (fitting and welds)
- fit only: only fitting and no welds

You can use this component to connect vertical rails to the handrail or to connect the stanchion to a stringer.



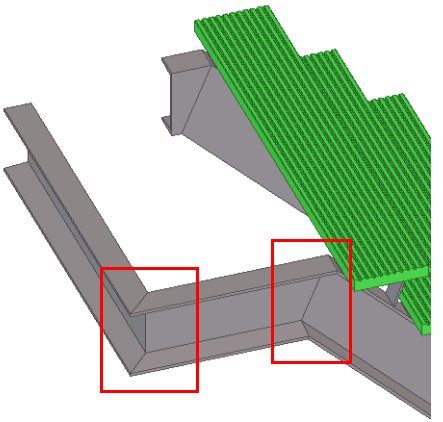


Component **Cranked beam (41)**

Supplied settings:

- cranked beam with weld
- cranked beam without weld

You can use these settings for situations as shown in the pictures, welded or not welded.





Component **Railing plane to plane (90)**

- several settings to create the correct bending, the generated parts are named “BENDING” automatically.

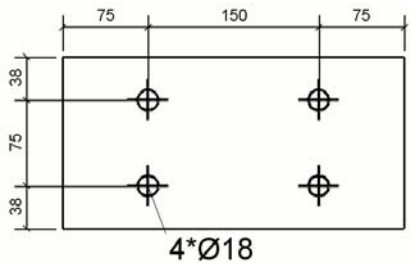
The wizard *all_workshopdrawings* creates an assembly drawing from parts with the name STRINGER. Also drawings from railings are now generated automatically. The filter in the wizards checks the part’ names STANCHION or RAILING.

3.47 Standard parts

It is possible to use standard parts (e.g. base plates, stubs). They have the same part number in each model they exist. This can be very useful, because this enables you to create a stock of parts.

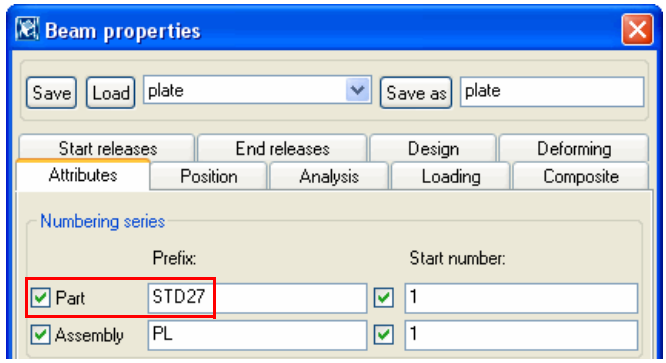
Example

We will add a plate (thickness 10 mm) as a standard part:



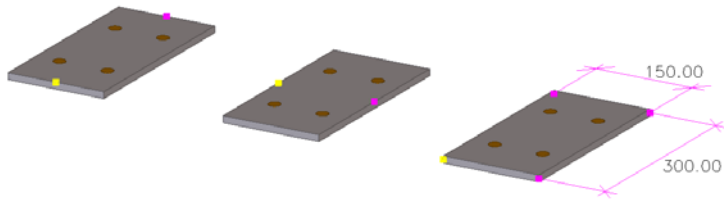
Step plan

1. Create a model with a specific name, e.g. **Std**
2. Create parts **without** using components. Set a unique prefix for each individual part:



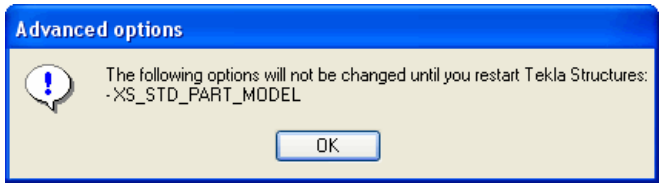
3. You need to model this part as follows to make sure that this part always will be "recognized":

Profile:	Profile:	Contour plate:
PL10*150 lg=300	**PL10*300 lg=150**	**PL10**

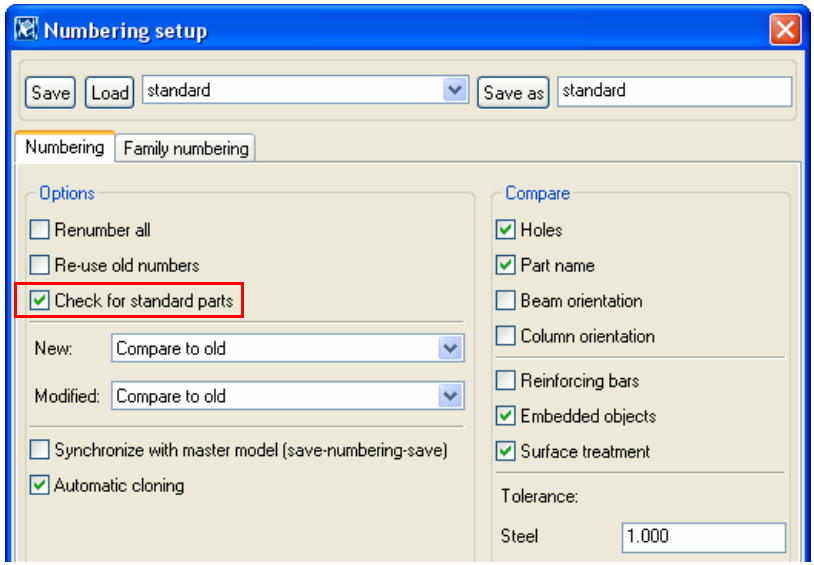


4. Go to **Tools** > **Advanced options...**

5. Select the category **Numbering** on the left pane**,** on the right pane, select the variable XS_STD_PART_MODEL
6. Modify the path name of the standard model in the field **Value**, e.g. *C:\TeklaStructuresModels\Std*
7. Click <**Apply**>, the following message appears:



8. Click <**OK**> and click <**OK**> again to close the dialog box
9. In the model folder, the file `options.ini` is stored. To apply this setting for all models, copy the file `options.ini` to the folder `ts` in the folder **disc:\TeklaStructures\version\environments\europe\ts**
10. Restart Tekla Structures and open any model
11. Go to **Setup > Numbering**, the following dialog box appears:



12. Check the option **Check for standard parts** and click <**Apply**> and **<OK**>.

With this option enabled, Tekla Structures will compare the current models' parts with the 'standard model' parts while numbering. If parts are identical, the part in the current model will get the same number as the 'standard models'-part.

3.48 Surface treatment

Topics

This chapter is divided into following sections:

Add surface treatment to a part face (p.477)

Add surface treatment to a selected area (p.479)

Cuts and recesses (p.480)

Add surface treatment to cut faces (p.481)

Cutting surface treatment (p.482)

Editing surface treatments (p.482)

Overlapping surface treatments (p.482)

Surface treatment select filter (p.483)

Numbering surface treatments (p.484)

Surface treatments options (p.484)

Add a tiled surface treatment (p.486)

Selecting surface treatment patterns (p.488)

Creating tile patterns (p.489)

Defining your own tile patterns (p.491)

Show tiled surface treatment in drawings (p.493)

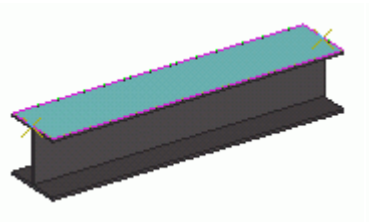
Surface treatment in templates and reports (p.496)

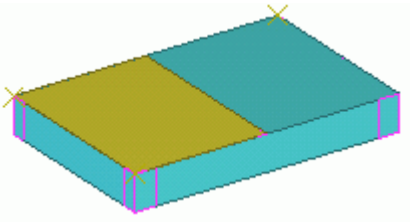
Tekla Structures includes tools to create surface treatments for concrete and steel parts, for example, unpainted areas, fire proofing, special finishes, or tile patterns.



Surface treatments are visible in models and can be included in drawings and reports.

You can create surface treatment for entire part faces, or to a selected area:







Surface treatment added to a part face.



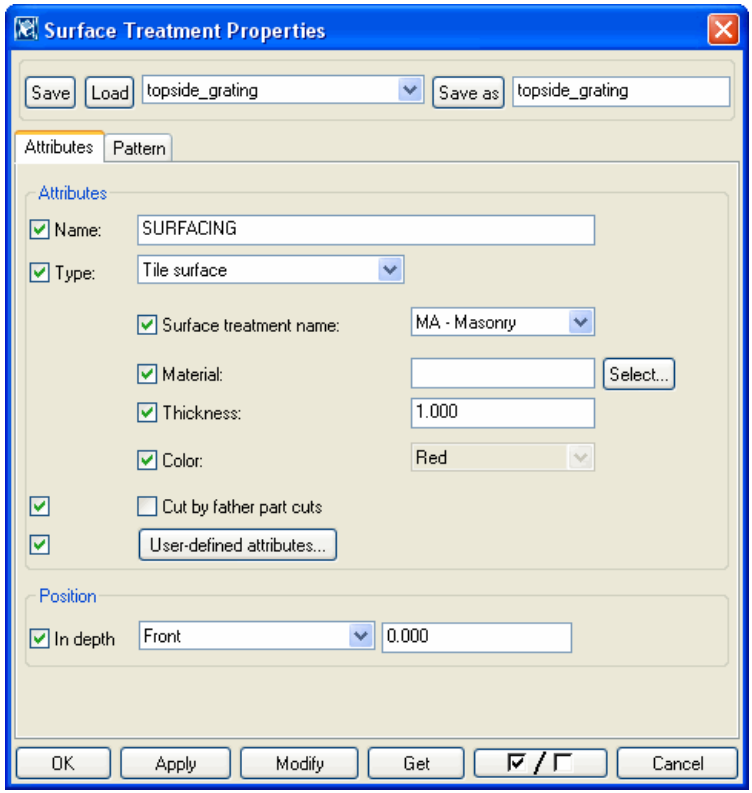
Surface treatment added to a selected area.

Add surface treatment to a part face

To add surface treatment to the face of a part:



1. Double-click the **Create face surfacing** icon to define the properties of the surface treatment in the **Surfacing properties** dialog box



The properties include:

- **Name** - used in drawing marks.
- **Surfacing type** and **Surfacing name** - used in reports and drawings.
- **Color** - color in rendered views.

- **Thickness** - thickness of the surface treatment.

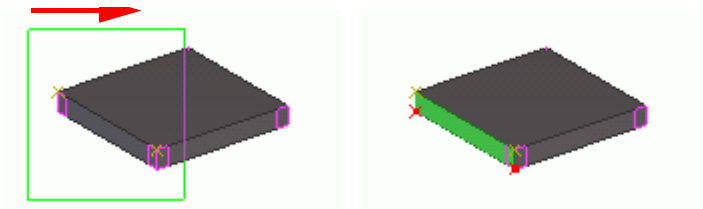
If necessary, a grade can be added.

If you add surface treatment to a part face that has openings, Tekla Structures uses cut parts and cut planes to cut the surface treatment.

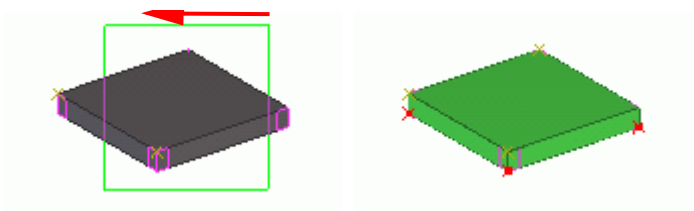
In the user defined attributes, several input field can be used to display the information in templates in drawings or in reports. You can define the pattern on the tab **Pattern** in the user defined attributes.

On the tab **Attributes** you can enter the depth and the offset of the surface treatment.

2. Click <**Apply**> or <**OK**> to save the properties and close the dialog box.
3. Select the part face to apply the surface treatment to. Ensure you are in a rendered view. Move the mouse cursor over a part, the faces that you can select appear in blue.
4. Select the part faces to apply the surface treatment to. Pick each face, or use the window selection technique:
   - To select all faces entirely within a rectangular area, drag the mouse from **left to right:**



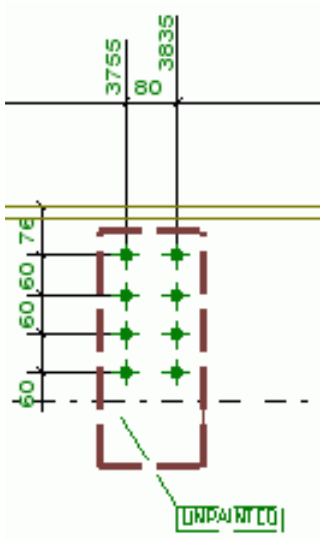
   - To select all objects entirely or partly within a rectangular area, drag the mouse from **right to left**:



5. Pick the origin and the direction of the surface treatment. For most of the surface treatment, the origin and the direction are not important, in those cases, you can pick the corner on the left as the origin and the direction of the part as surface treatment direction. For bricks, this is quite important. Zie *Add a tiled surface treatment*
6. Tekla Structures creates the surface treatment.

Add surface treatment to a selected area

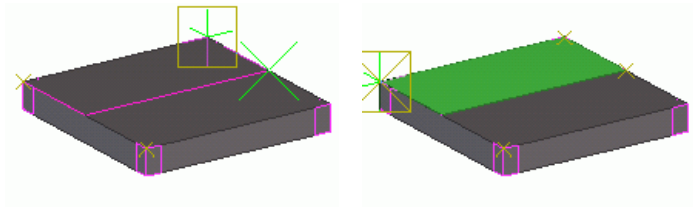
Use the **Create polygon surfacing** tool to add surface treatment to a selected area, for example you can specify unpainted areas of steel when you need to bolt an end plate to an unpainted surface:



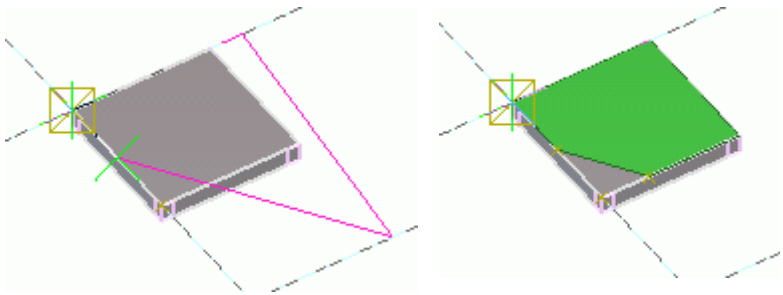
To add surface treatment to a polygonal area:



1. Double-click **Create polygon surfacing** icon to define the properties of the surface treatment in the **Surfacing properties** dialog box
2. Enter or modify the surface treatment properties to use.
3. Click <**Apply**> or <**OK**> to save the properties and close the dialog box.
4. Pick the origin of the surface treatment.
5. Pick the direction of the surface treatment.
6. Select the part face to apply the surface treatment to.
7. Pick points to indicate a polygonal area.



- If you pick points outside the part face, the surface treatment adjusts to fit the face:

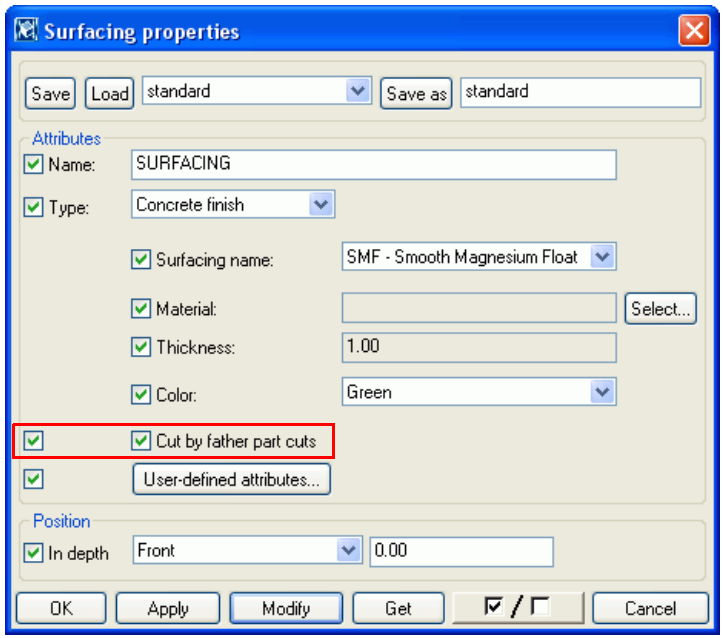


- If you pick corner points outside the part face plane, the corner points adjust to fit the part face.

Tekla Structures creates the surface treatment.

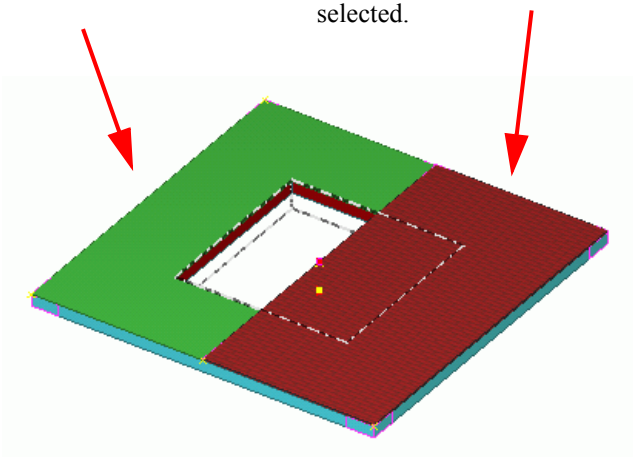
Cuts and recesses

To force Tekla Structures to consider openings and recesses in parts when adding surface treatment, select the **Cut by father part cuts** checkbox in the **Surfacing properties** dialog box:



The green surface treatment has check box **Cut by father cuts** is selected.

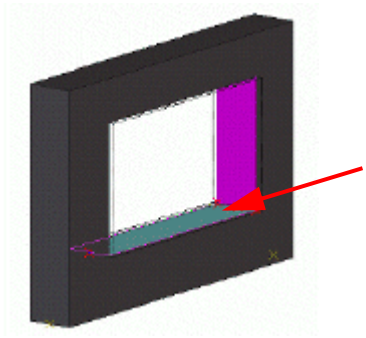
The tiled surface treatment is not cut by the cut in the part: **Cut by father cuts** is not selected.



Add surface treatment to cut faces

To add surface treatment to cut faces:

1. Click the **Create face surfacing** or **Create polygon surfacing** icon.
2. Pick the origin of the surface treatment.
3. Pick the direction.
4. Select the cut face to apply the surface treatment to:



5. For the polygon surfacing option, pick the points to define the area of the surface treatment.

Tekla Structures modifies the surface treatment when you change the part.

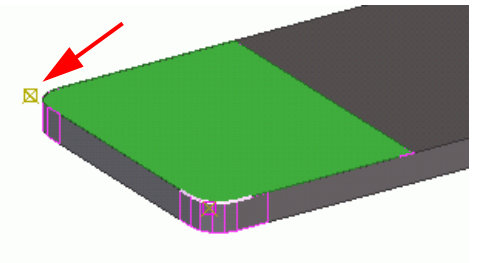
Cutting surface treatment

To define the cut depth of a polygon cut, for example, to cut thick surface treatment, use the variable `XS_POLYGON_CUT_EXTRA_THICKNESS`:

1. Click **Tools > Advanced options...**, and go to the **Modeling** category.
2. Set the cut thickness for the variable `XS_POLYGON_CUT_EXTRA_THICKNESS`. The default value is 5.0.

Editing surface treatments

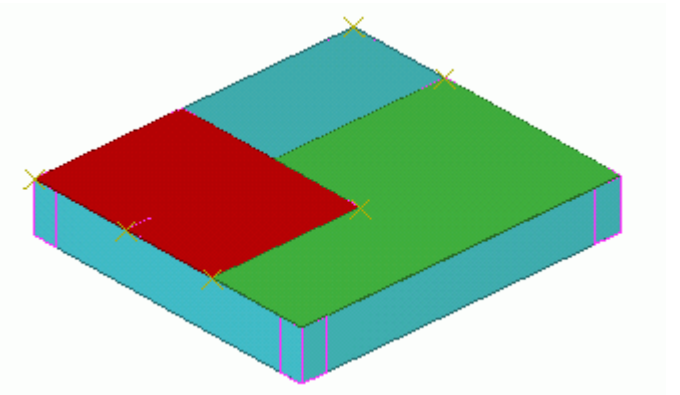
Use the chamfers on surface treatment corners to reshape and move the corners. You can also create cuts to add openings to surface treatments.



Overlapping surface treatments

When you create overlapping surface treatments, the smaller surface treatment overrides the larger one.

The overlapping area is recognized in reports: only the topmost (visible) surface treatment is calculated.

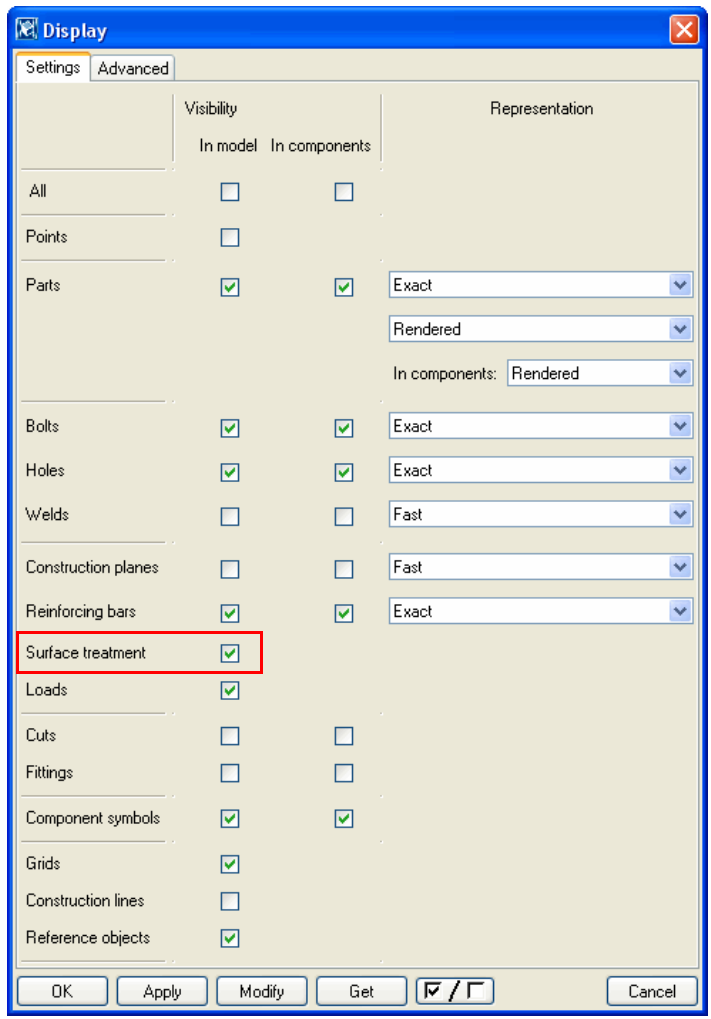


Surface treatment select filter

You can use the surface treatment filter to fast and easily filter surface treatments:

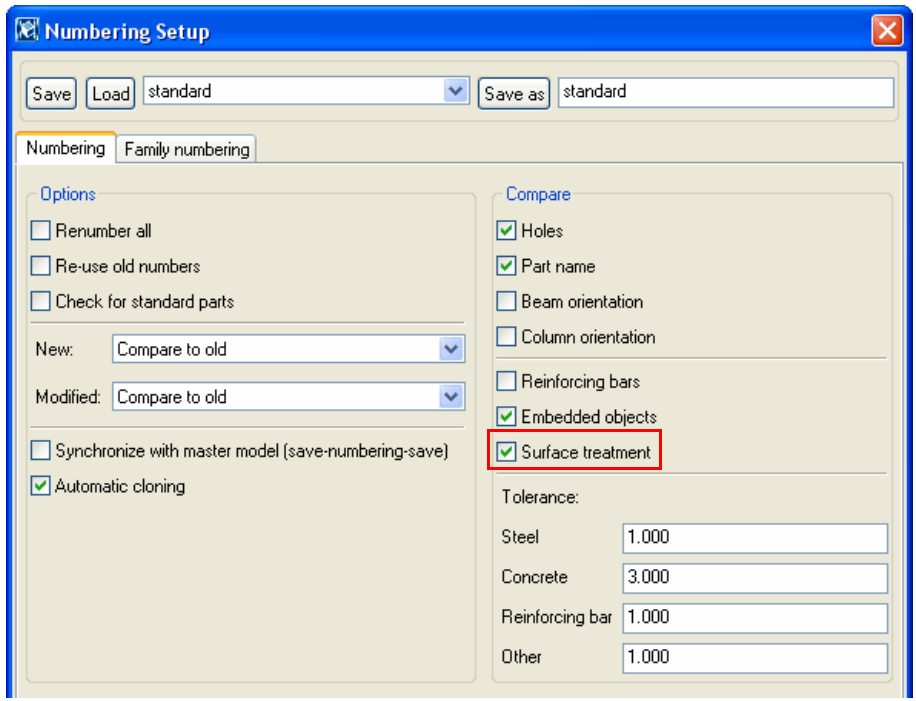


Besides you can set per view if surface treatment should be displayed or not:



Numbering surface treatments

To have Tekla Structures include surface treatment in numbering, click **Setup > Numbering...**, and select the **Surfacings** checkbox on the **Numbering** tab.



Surface treatments options

The **Type** and **Surfacing name** properties in the **Surfacing properties** dialog box, are defined in `product_finishes.dat` file, located in the folder *..environments\europe\system*.

You can edit this file to add surface treatments and save it. The modified file cab be stored in the folder ***europe/ts***.

The first section of the file defines the types of surface treatment available. Do not edit this section:

```
// Product finishes
// ------------------------
//
// Type         : Type of surfacing
//                  1 = concrete finish
//                  2 = special mix
//                  3 = tile surface
//                  4 = steel finish
// Code         : Abbreviation code used in drawings and reports
// Name         : This is the name used in reports
```

The remaining sections define the options for each surfacing type. This is where you add new options.

```
// ========================================
// *** Concrete Finish
// ========================================
// WET FINISH
// ----------
1            MF       "Magnesium Float"
1            SMF      "Smooth Magnesium Float"
1            WT       "Wet Trowel"
```

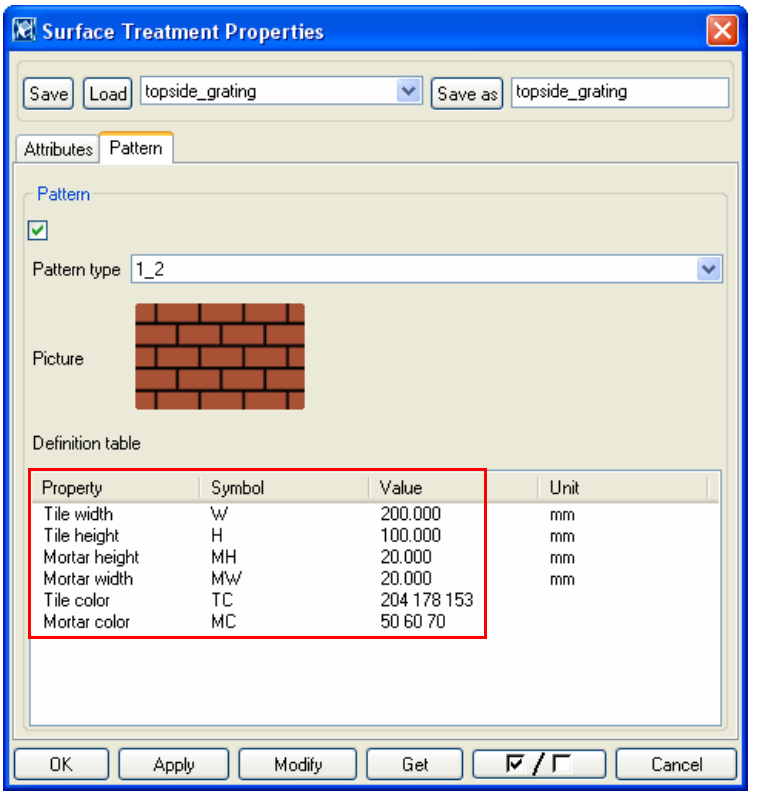
In the example above:

1. `1` is the surface treatment type (concrete finish)
2. `MF` is the code for the surface treatment option
3. `"Magnesium Float"` is the full name of the surface treatment option

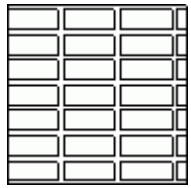
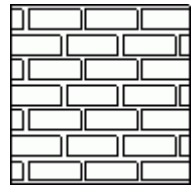
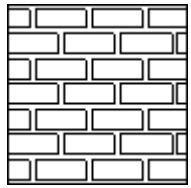
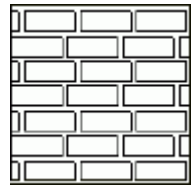
Add a tiled surface treatment

To add a tiled surface treatment to a part:

1. Double-click the **Create polygon surfacing** or **Create face surfacing** icon to open the **Surfacing properties** dialog box and go to the tab **Pattern**.
2. You can define or modify the tile pattern properties in the **Definition table**:



The options for bonding are:

Option	Bonding	Option	Bonding
1/1		1/3	
1/2		1/4	

- Define tile and mortar colors by assigning values for red, green, and blue, on a scale of 0 to 10. For example:

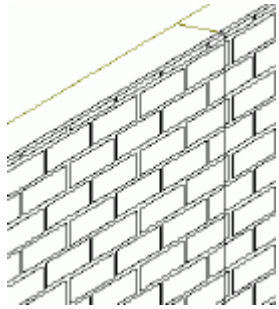
  Red = 10 0 0
  Green = 0 10 0
  Blue = 0 0 10
  White = 10 10 10
  Black = 0 0 0

3. Click <**Modify**> and <**OK**> to save the tile pattern properties.

To add the tiled surface treatment to a part, use the **Create polygon surfacing** or **Create face surfacing** command:

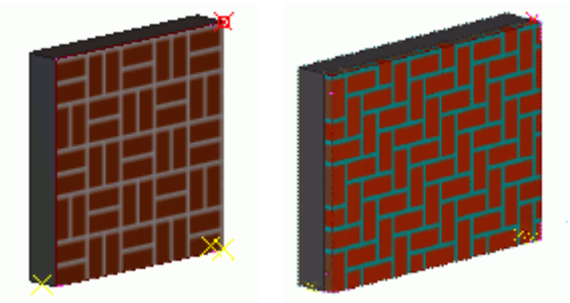
Pick the corners of the surface treatment or pick the part face or pick the origin and the direction of the surface treatment.

To create tiling that runs continuously from one part to the next, pick the same origin for the surface treatment for all the parts:



Selecting surface treatment patterns

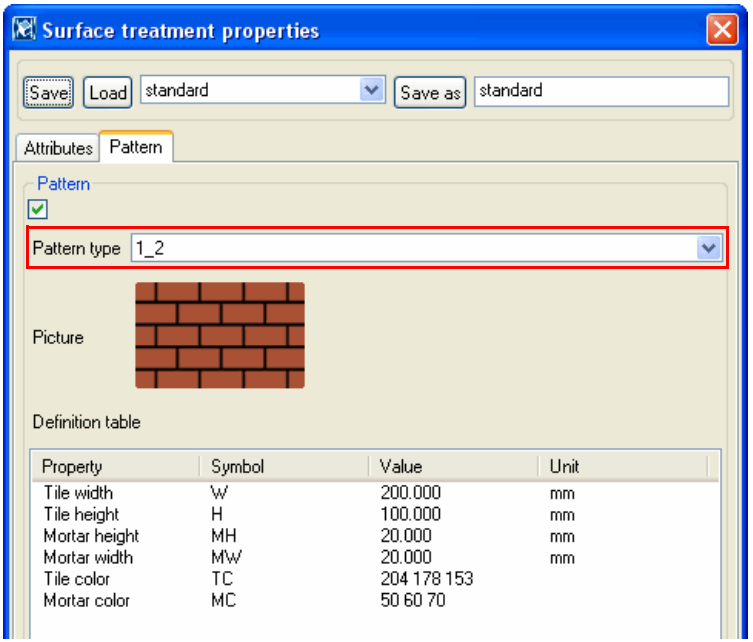
Tekla Structures includes complex tile and brick surface treatment options, such as basket weave and herringbone patterns:



Tiled surface treatment options are based on repeating tile patterns that are stored in XML format.

To define the properties of a tiled surface treatment:

1. Double-click the **Create face surfacing** icon or the **Create polygon surfacing** icon to open the **Surfacing properties** dialog box.
2. On the **Attributes** tab, go to **Type** and select **Tile surface**. Go to **Type > Material** and select **Miscellaneous > Tiles**.
3. On the **Pattern** tab, select the pattern from the **Pattern type** list box.



4. The **Definition table** lists the properties of the pattern type.
5. Click <**Modify**> and <**OK**>.

To apply the surface treatment to a part face:

1. Click the **Create face surfacing** icon.
2. Pick a point to indicate the origin of the surface treatment.
3. Pick a point to indicate the x direction of the surface treatment.
4. Pick the a part face to apply the surface treatment to it.

Creating tile patterns

Information about the predefined tile patterns in the **Surfacing properties** dialog box are stored in the following files:

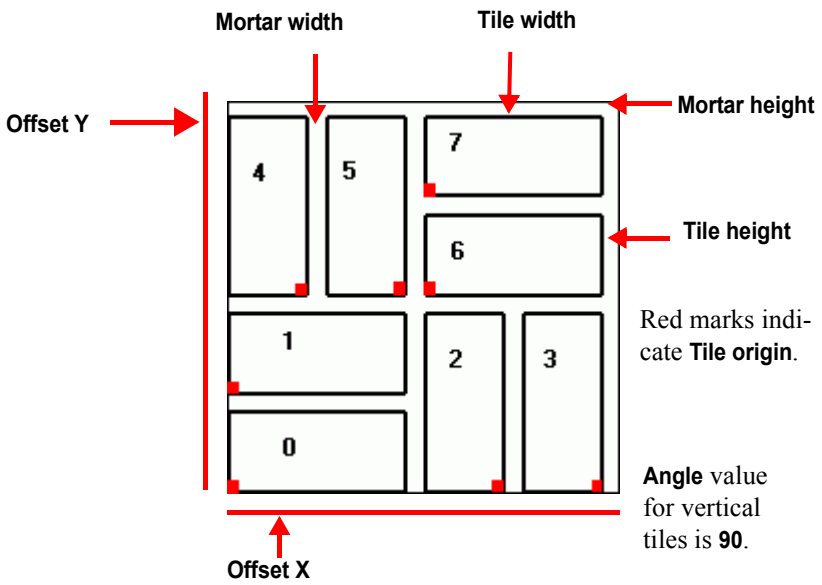
- `TilePatternCatalog.xml`
  - Contains the tile pattern definitions.
  - Located in the model's `attributes` folder, project folder, firm folder, and system folder.
- `TilePatternCatalog.dtd`
  - Document Type Declaration (DTD) file that defines the elements allowed in the `TilePatternCatalog.xml` file.
  - Located in the same folder as `TilePatternCatalog.xml` file
- Thumbnail images
  - The images that appear in the **Picture** field in the **Surfacing properties** dialog box.
  - Located in the `../nt/bitmaps` folder.
  - Filenames identify the pattern types, for example, `herringbone.bmp` illustrates the `herringbone` pattern type.

Tile patterns are defined by the following properties:

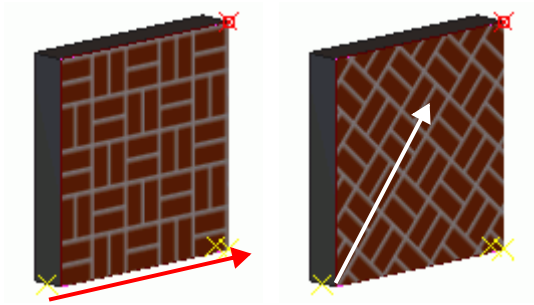
- **Mortar width**, **Mortar height, Mortar color R**, **G**, and **B**
  Mortar properties are the same in the whole pattern. The color scale is 0 - 255
- **Offset X** and **Offset Y**
  Define the point at which the pattern block is repeated in the x and y directions. Surface treatment runs in the x direction.
- **Tile width**, **Tile height**, **Tile color R**, **G**, and **B**
  Define the properties of individual tiles.
- **Tile origin**
  X and **Y** values that define the origin of each tile. The x direction is the same as for the surface treatment.
- **Angle**
  The angle of the surface treatment, calculated counterclockwise from the origin of the surface treatment.

Example

This example shows a basket weave pattern block made up of eight tiles:



The pattern is repeated in the x and y direction of the surface treatment, starting from the origin of the surface treatment. You can run this pattern in different x directions:



Defining your own tile patterns

To define your own tile patterns:

1. Open the file **`TilePatternCatalog.xml`** file using any text editor. The file is located in the folder:

   disc:\TeklaStructures\version\environments\europe\system

2. Add a new `<TilePattern>` element to the file. A `<TilePattern>` element must have `<HOffset>` and `<VOffset>` elements and at least one `<Tile>` element. Other elements are optional. You may find it easier to copy one of the existing elements, then change it suit your needs.

The `TilePatternCatalog.xml` file can contain the following elements:

- `TilePatternCatalog` - The container for tile patterns. Required.
- `TilePattern` - Tile pattern element. Required. This element can contain the following elements:
  - `HOffset` - Horizontal offset of the tile pattern. Required.
  - `VOffset` - Vertical offset of the tile pattern. Required.
  - `Tile` - The individual tile(s) used in a tile pattern. At least one required.
  - `Color` - Color of the tile or mortar. Optional.
  - `Parameter` - Creates an attribute for any element in the `TilePattern`. Optional.
  - `Label` - The label that identifies a parameter in the dialog box. Optional.
  - `TileOrigin` - The origin of an individual tile, defined from the origin of the pattern. Optional.

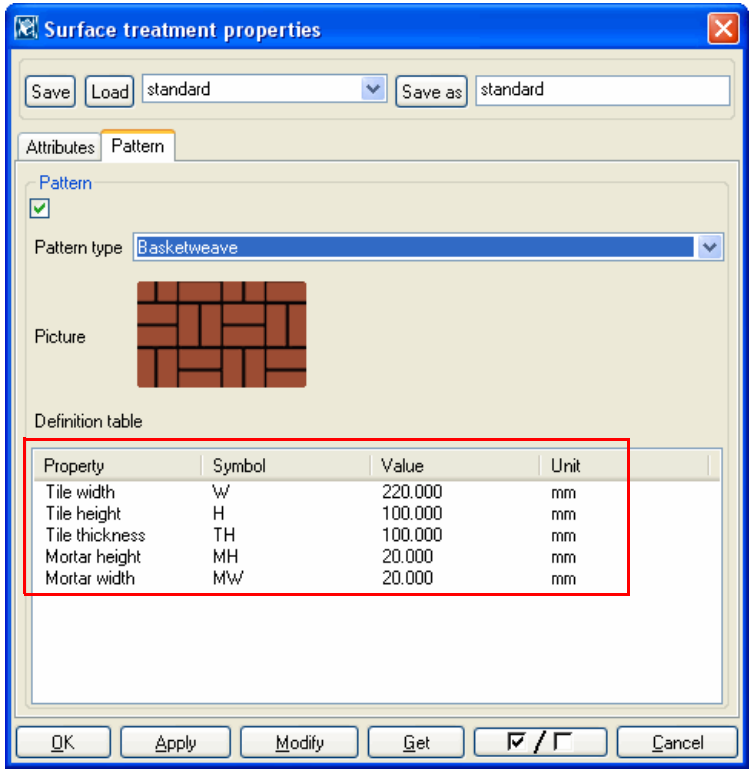
Example

This example explains how the **Basket weave** tile pattern is defined in the `Tile-PatternCatalog.xml` file and shows how the pattern options appear on the **Pattern** tab of the **Surfacing properties** dialog box:

```
<TilePattern Name="Basketweave">
        <Parameter Name="W" DefaultValue="220">
                <Label> _Tile_Width </Label>
        </Parameter>
        <Parameter Name="H" DefaultValue="100">
                <Label> _Tile_Height </Label>
        </Parameter>
        <Parameter Name="TH" DefaultValue="100">
                <Label> _Tile_Thickness </Label>
        </Parameter>
        <Parameter Name="MH" DefaultValue="20">
                <Label> _Mortar_Height </Label>
        </Parameter>
        <Parameter Name="MW" DefaultValue="20">
                <Label> _Mortar_Width </Label>
        </Parameter>
        <HOffset>
                <Vector2D X="W+2*H+3*MW" Y="0" />
        </HOffset>
        <VOffset>
                <Vector2D X="0" Y="W+MH" />
        </VOffset>
        <Tile Angle="0" Width="W" Height="H" Thickness="TH">
                <TileOrigin>
                        <Vector2D X="0" Y="0" />
                </TileOrigin>
                <Color Color="204 179 153" />
        </Tile>
        <Tile Angle="0" Width="W" Height="H" Thickness="TH">
                <TileOrigin>
                        <Vector2D X="0" Y="H+MH" />
                </TileOrigin>
                <Color Color="204 179 153" />
        </Tile>
        <Tile Angle="90" Width="W" Height="H" Thickness="TH">
                <TileOrigin>
                        <Vector2D X="W+MW+H" Y="0" />
                </TileOrigin>
                <Color Color="204 179 153" />
        </Tile>
        <Tile Angle="90" Width="W" Height="H" Thickness="TH">
                <TileOrigin>
                        <Vector2D X="W+2*(MW+H)" Y="0" />
                </TileOrigin>
                <Color Color="204 179 153" />
        </Tile>
</TilePattern>
```

The name of the pattern.

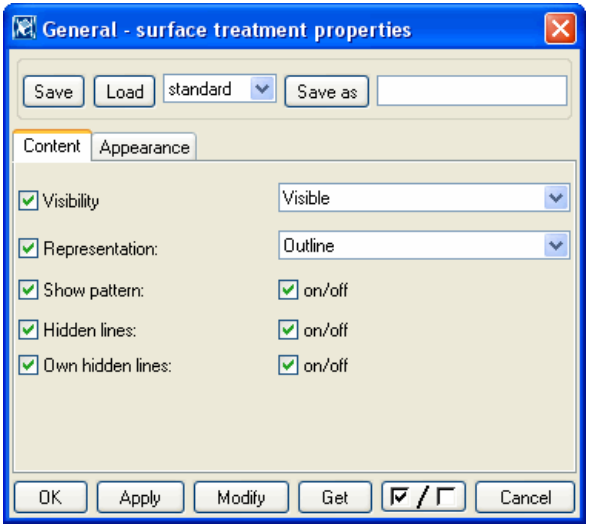
The size of the pattern block in the x and y direction, after which the pattern repeats



Show tiled surface treatment in drawings

In all types of drawings (single part-, assembly-, cat unit- and GA), you can define how tiled surface treatment appears in drawings:

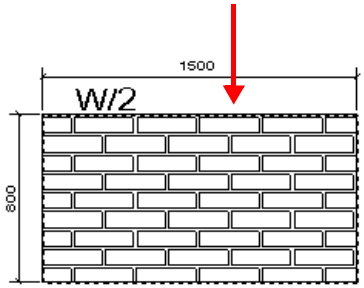
To open the **Drawing surfacing properties** dialog box, click the **Surfacing...** button in the drawing properties dialog box or double-click the outline of the surface treatment in the drawing:



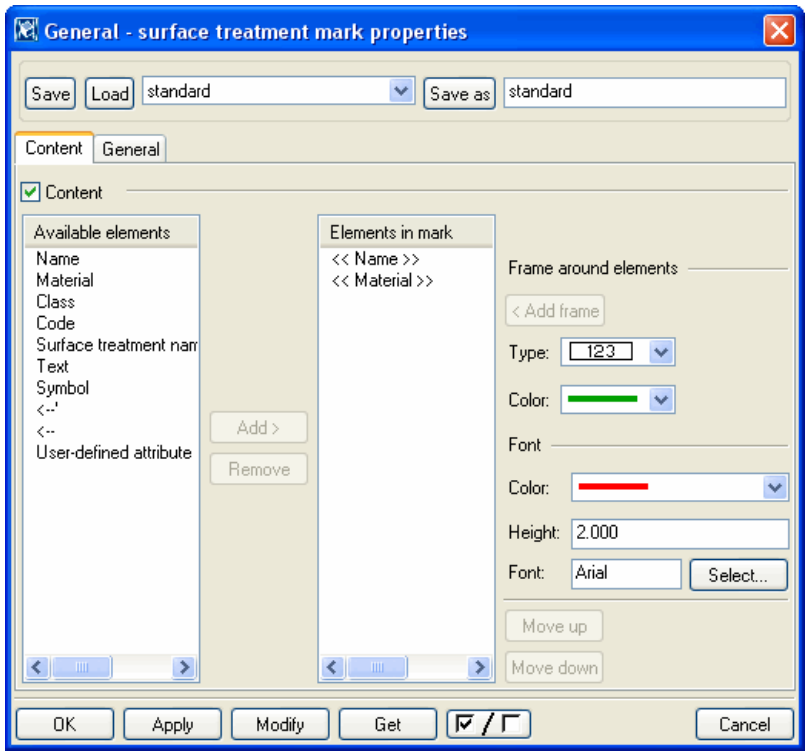
The display options are: **Outline**, **Exact**, **Workshop form**, **Symbol**, **Bounding box**, and **Base box**.

To show the tile pattern in the drawing, select the **Show pattern** checkbox. Click <**Apply**> or <**OK**> to save the drawing properties.

The drawing shows the tile surface treatment and the surface treatment mark:



To edit surface treatment marks, click the **Surfacing mark...** button in the drawing properties dialog box to open the **Surfacing mark properties** dialog box.



Modify surface treatment marks in the same way as part marks.

To show tile and mortar properties in surface treatment marks, use the following user-defined attributes: `TILE_WIDTH`, `TILE_HEIGHT`, `MORTAR_WIDTH`, `MORTAR HEIGHT`, `TILE_COLOR`, and `MORTAR_COLOR`.

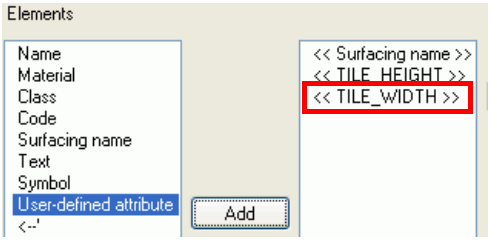
For example, to add tile width property to a surface treatment mark:

Select **User-defined attribute** in the **Elements** list and click **Add**.

In the **Mark content** dialog box, enter `TILE_WIDTH`:

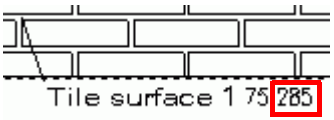


Click <**OK**>, to add `TILE_WIDTH` to the list of mark elements:



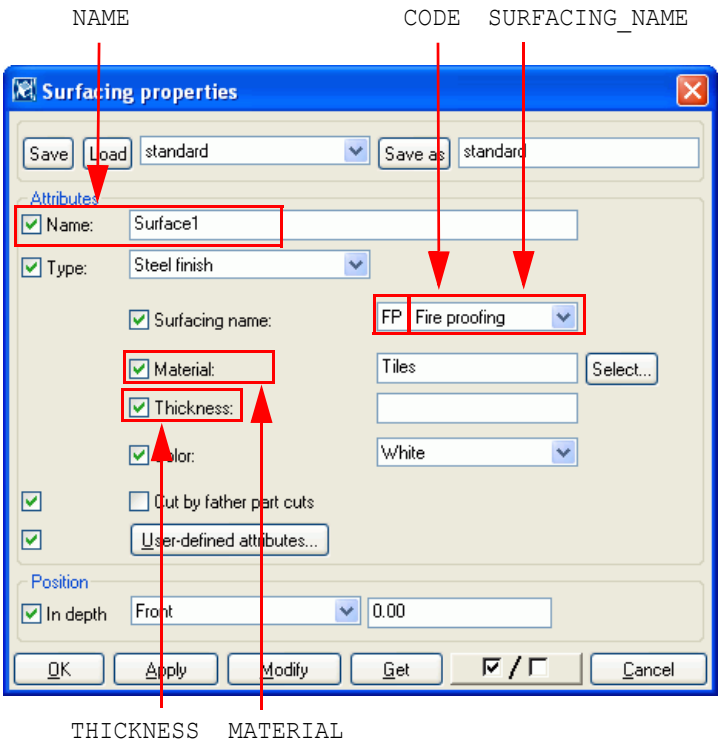
Add surface treatment marks to the drawing, or modify existing ones.

The surface treatment marks now include the tile width:



Surface treatment in templates and reports

The template attributes that relate to the **Surfacing properties** fields are shown below:



3.49 Tips Windows Explorer

Topics

This chapter is divided into following sections:

Customizing toolbars (p.497)

Apply a view for all files and folders (p.498)

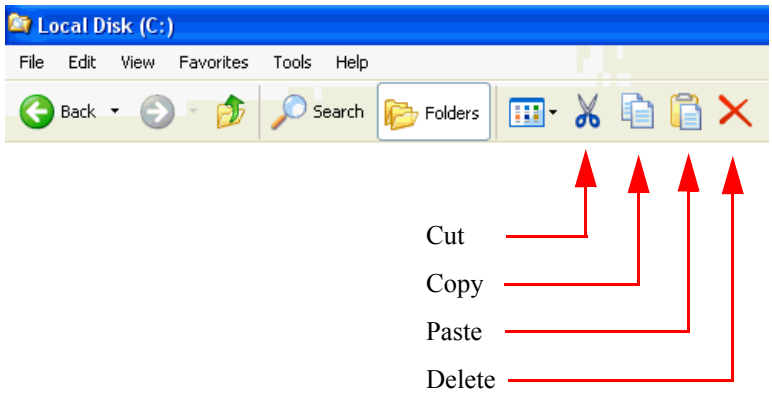
Choose details (p.500)

Hidden extensions (p.502)

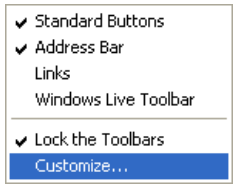
Shortkeys (p.503)

Customizing toolbars

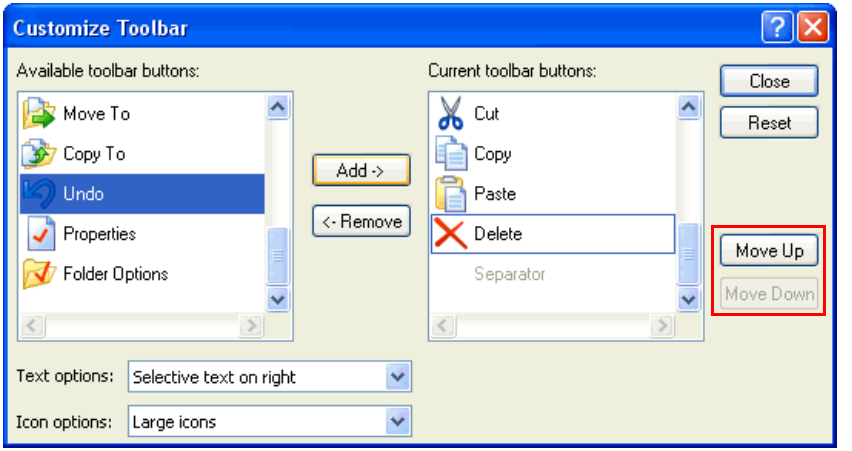
In the Explorer you can add some more commands to the toolbars. This can be handy while working in Tekla Structures.



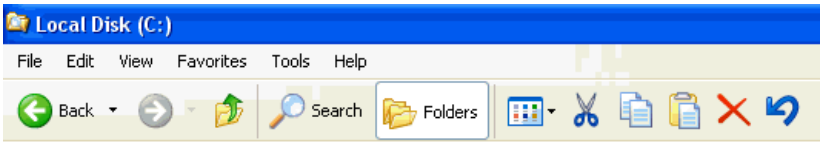
Go to **View** > **Toolbars** > **Customize...** or click the right mouse button on the toolbar and select **Customize..:**



You can now customize the current toolbar:



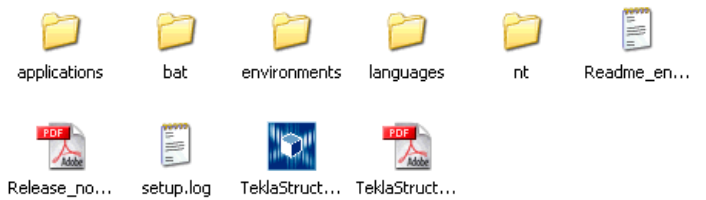
In case you have added a toolbar button, you can change it's position by using the buttons <**Move Up**> and <**Move Down**>. Next click **<Close>.**



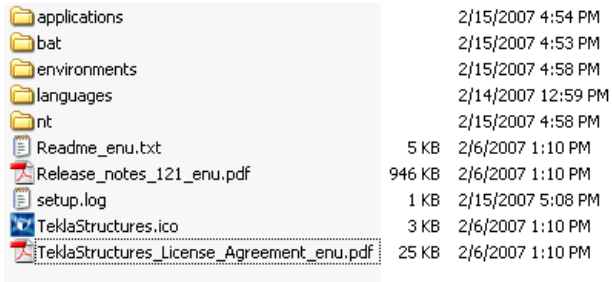
Apply a view for all files and folders

In the Explorer, you can apply the view that you are using (such as Icons or Details) for the current folder, to **all** folders.

Icons

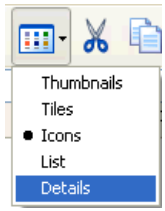


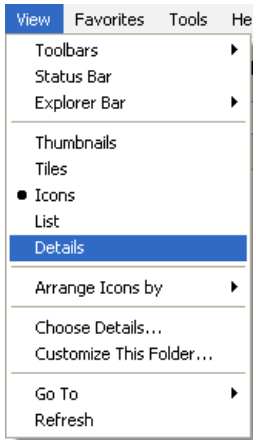
Details



Step plan

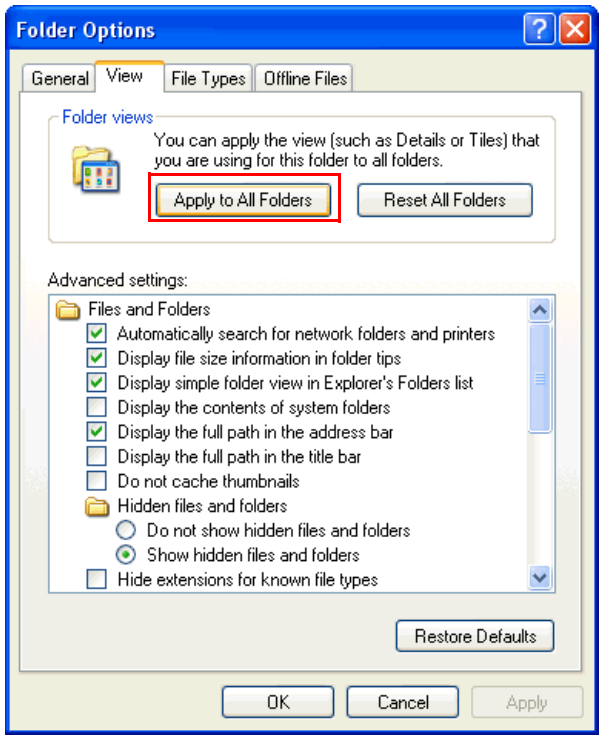
1. In the Explorer, go to the pull-down menu **View** or click on the icon and select Details:

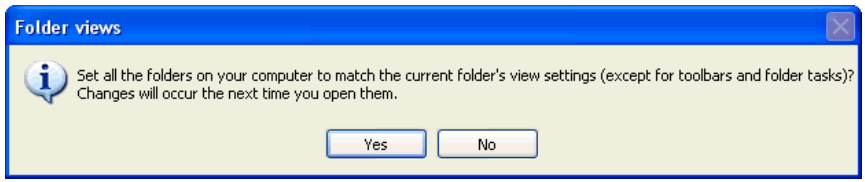




2. Select the option **Details**

3. Go to **Tools** > **Folder Options...** in the Explorer and select the tab **View.** Click "**Apply to All Folders**":

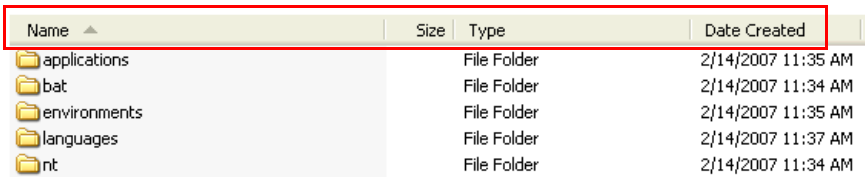




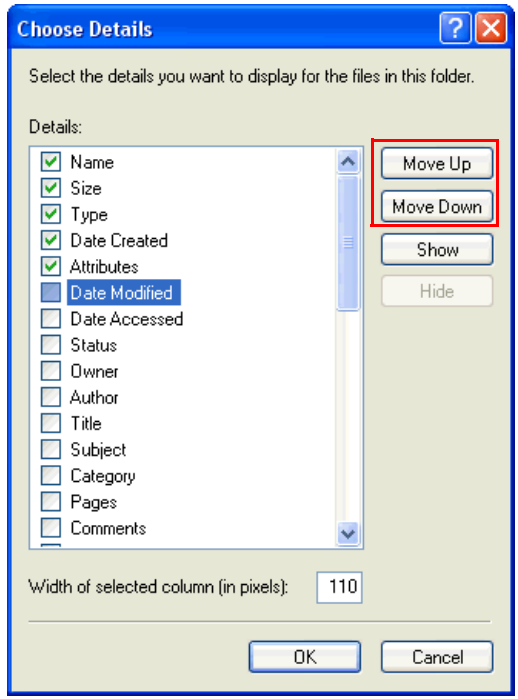
4. Click <**Yes**>

Choose details

In the Explorer you can select the details you want to display for the files in the folders:



Go to **View** > **Choose Details...** In the dialog box, select the details you want to display. Again, you can change the position of the columns by using the buttons <**Move Up**> and <**Move Down**>.

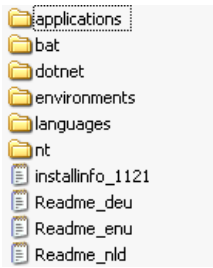


You can also select the details you want to display by clicking the right mouse button on the column:

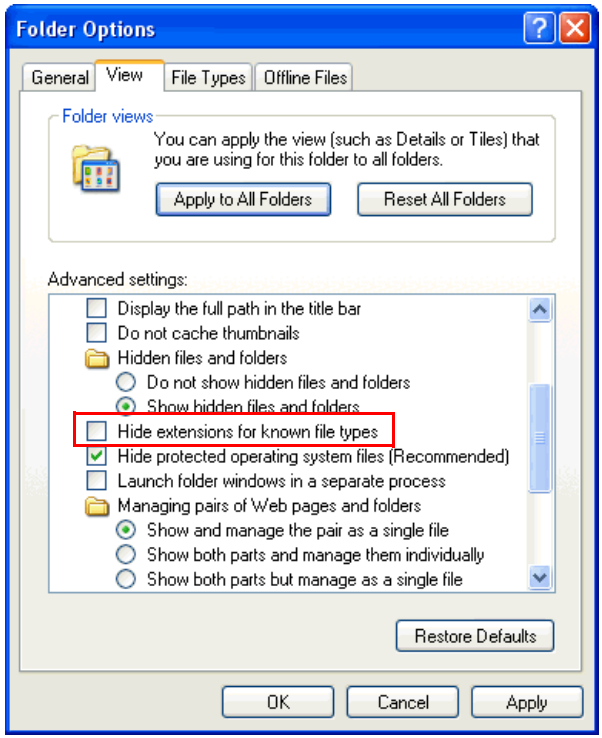


Hidden extensions

In the Explorer you can hide the extensions for the known file types. This is not handy in Tekla Structures because the file extension informs you about file type.

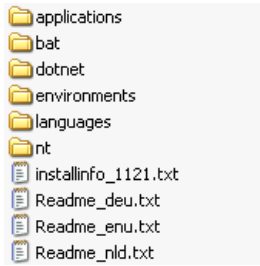


To display the file extensions for all file types, go to **Tools** > **Folder Options** in the Explorer. Toggle off the checkbox by "**ExHide extensions for known file types**" in the tab View:



Click <**Apply**> and <**OK**>.

All file extensions will be displayed:



Shortkeys

By using shortkeys in the Explorer you can work faster and more effective:

Command:	Action:
(⊞) + E	Open My Computer
(⊞) + D	Show the Desktop
F5	Refresh
F2	Rename selected item
<**Ctrl**> + A	Select All
<**Shift**> + <**Delete**>	Delete selected item permanently without placing the item in the Recycle Bin
<**Alt**> + arrow left	Go Back
<**Alt**> + arrow right	Forward
<**Backspace**>	Go Up One Level

3.50 Unfolding profiles and plates

Topics

This chapter is divided into following sections:

The file Unfold_corner_ratios.inp (p.505)

Bolt group in a polybeam (p.510)

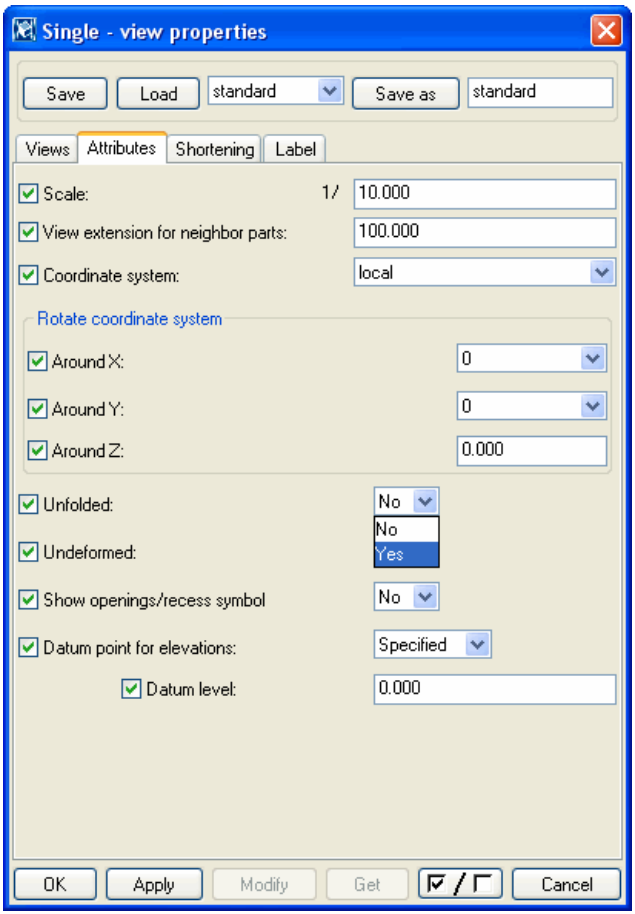
NC files from polybeams (p.511)

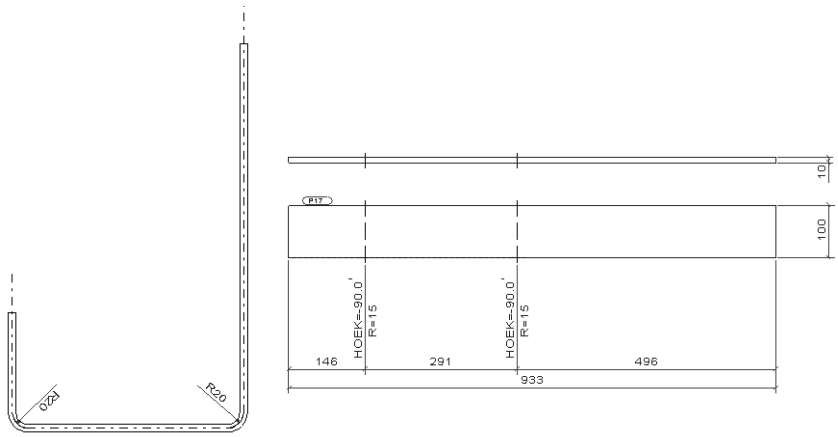


Profiles, created as **polybeams,** can be unfolded in Tekla Structures. This can be done for both profiles and plates (setting: FOLDED_PLATE).

In the drawing properties, a setting named **unfolded** is added. This setting can be used to unfold polybeam profiles.

Go to **Properties > Single part drawing > View > Attributes** to unfold the part, see picture below.





Folded plate in model Unfolded

The file Unfold_corner_ratios.inp

To create unfoldings in the new way mind the following conditions:

1 For the variables **XS_USE_OLD_POLYBEAM_LENGTH_CALCULATION** and **XS_CALCULATE_POLYBEAM_LENGTH_ALONG_REFERENCE_LINE** NOTHING should be entered in the **Advanced options...** in the category **Dimensioning: Unfolding**

The next variables must be set as follows:

1. The variable **XS_DRAW_BENDING_LINE_DIMENSIONS_IN_UNFOLDING** must be set to TRUE in the **Advanced options...** in the category **Dimensioning: Unfolding** to create bending line dimensions for unfolded parts
2. The variable **XS_DRAW_ANGLE_AND_RADIUS_INFO_IN_UNFOLDING** must be set to TRUE in the **Advanced options...** in the category **Dimensioning: Unfolding** to create angle and radius dimensions for unfolded parts.
3. The variable **XS_ANGLE_TEXT_IN_UNFOLDING_BENDING_LINE_DIMENSIONING** is set to HOEK= in the **Advanced options...** in the category **Dimensioning: Unfolding** as a prefix text for an angle dimension
4. The variable **XS_RADIUS_TEXT_IN_UNFOLDING_BENDING_LINE_DIMENSIONING** is set to R= in the **Advanced options...** in the category **Dimensioning: Unfolding** as a prefix text for a radius dimension

5. The variable **XS_DRAW_INSIDE_ANGLE_IN_UNFOLDING** is set to TRUE in the **Advanced options...** in the category **Dimensioning: Unfolding** for angle text dimensions, shows the interior angle instead of the exterior angle, i.e. 45° instead of 135°
6. The variable **XS_UNFOLDING_ANGLE_DIM_FORMAT** is set to 2 in the **Advanced options...** in the category **Dimensioning: Unfolding** as a format for angle text. These are the same values as on the *General* tab of the *Dimension properties* dialog box. Enter values as integers
7. The variable **XS_UNFOLDING_ANGLE_DIM_PRECISION** is set to 100 in the **Advanced options...** in the category **Dimensioning: Unfolding** and determines the accuracy of the angle text. These are the same values as on the *General* tab of the *dimension properties* dialog box. Enter values as integers, i.e. 1,2,3,4,8,16,32,10,100,1000

2 The configuration file `unfold_corner_ratios.inp`, which affects the unfold operations, is stored in the folder **system** in Tekla Structures.

This file is located in the **system** folder by default but it can also be stored in the **model** folder or in the folder **ts**.

The unfold-parameters define the position of the neutral sheet (stress and strain are 0) when a profile is unfolded.

The parameters are required for generating NC data, and also for unfolded representation on drawings. The content of file `unfold_corner_ratios.inp`:

```
# Example for unfolded platen
# Rules starting with '#'  are just comment
#
# PROFILE
#
# [type = 1] [profile] [material] [rotation min] [rotation max]
# [flag: 1 = sharp, 2 = curved] [angle/radius min] [angle/radius max] [ratio]
# 1 [type]
# 2 [profile]
# 3 [material]
# 4 [rotation min]
# 5 [rotation max]
# 6 [flag: 1 = sherp, 2 = radius]
# 7 [angle/radius min]
# 8 [angle/radius max]
# 9 [ratio]                              = length calculation alongs inside, hart or outside (.0, .5 of 1.0)

1 IPE* S235JR -180 180 1 -180  180 .5
1 B* S235JR -180 180 1 0 90 0.5
1 B* S235JR -180 180 2 -180  180 .5
1 PD* S235JR -180 180 1 0 90 0.5
1 PD* S235JR -180 180 2 -180  180 .5
1 ANCHOR* S235JR -180 180 2 -180  180 .5
1 D* S235JR -180 180 2 -180  180 .5
1 R* S235JR -180 180 2 -180  180 .5
1 UNP* S235JR -180 180 1 -180  180 .5
1 L* S235JR -180 180 1 -180  180 1.0

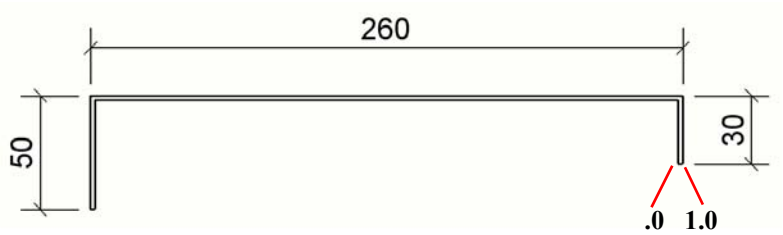
#1 IPE6* S235JR -180 180 2 0  1000 .3

#
# PLATE
#
# [type = 2] [profile] [material] [thickness min] [thickness max]
# [flag: 1 = sharp, 2 = curved] [angle/radius min] [angle/radius max] [ratio]
# 1 [type]
# 2 [profile]
# 3 [material]
# 4 [rotation min]
# 5 [rotation max]
# 6 [flag: 1 = sherp, 2 = radius]
# 7 [angle/radius min]
# 8 [angle/radius max]
# 9 [ratio]                              = length calculation alongs inside, hart or outside (.0, .5 of 1.0)


2 PL* * 1  3  1 -180  180 .0
2 PL* * 4 200 2    0 1000 .5
# [type = 3] [profile] [material]
```

Example1

A folded plate, thickness **2** mm:



The latter value in the file `unfold_corner_ratios.inp` is:

.0: Unfolded length: 48 + 256 + 28 = 332

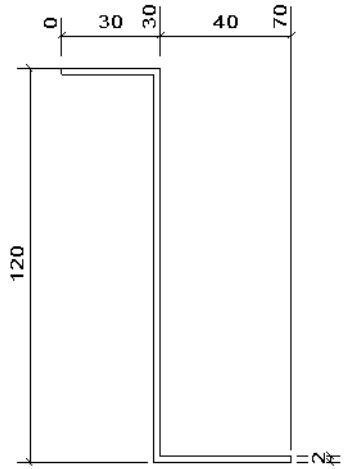
.5: Unfolded length: 49 + 258 + 29 = 336

1.0: Unfolded length: 50 + 260 + 30 = 340

For plates with thickness up to 3 mm, length is calculated across the **.0** line. For other plates, the length is calculated across the centre, the **.5** line.

```
2 PL* * 1   3   1 -180   180 .0
2 PL* * 4 200 2    0  1000 .5
# [type = 3] [profile] [material]
```

Example 2 Folded plate (Z shaped):



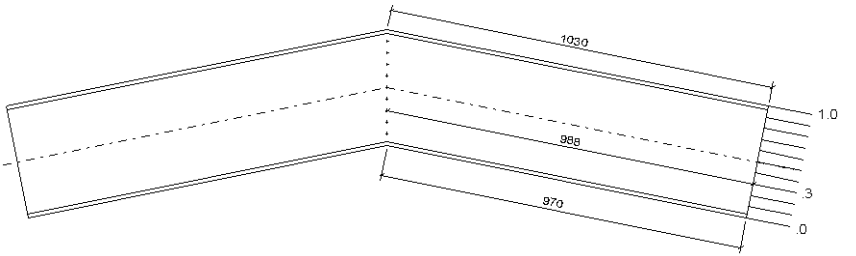
The latter value the file *unfold_corner_ratios.inp* is:

.0: Unfolded length: 28 + 116 + 40 = 184

.5: Unfolded length: 29 + 118 + 41 = 188

1.0: Unfolded length: 30 + 120 + 42 = 192

Example 3 Polybeam IPE300:



.0: Unfolded length: 2 x 970 = 1940

.3: Unfolded length: 2 x 988 = 1976

1.0: Unfolded length: 2 x 1030 = 2060

In the file `unfold_corner_ratio.inp` file, **never** replace the second option by just a *-character!

If modifications have been carried though to the file `unfold_corner_ratio.inp`, Tekla Structures needs to be restarted and all accompanying drawing have to be checked.

The last action is only needed in certain modifications to the file. Bear in mind that modifications (e.g. length in templates) are carried through immediately when opening an existing drawing.

In case a profile is not defined, the default value will be used for the length calculation. (= 0.5)

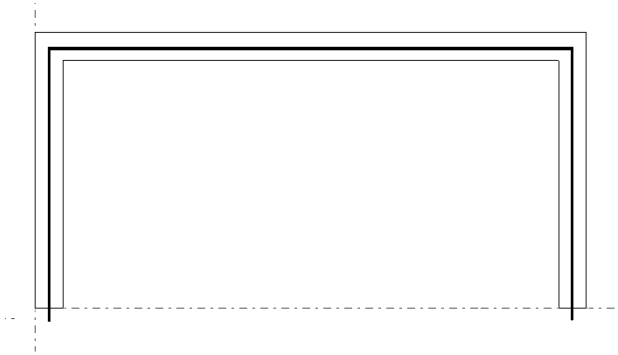


In the **Beam Properties** on the tab **Position**, you can set the **Position in plane** (Left, Middle, Right).

However, the position of the neutral line will **not** be moved with this.



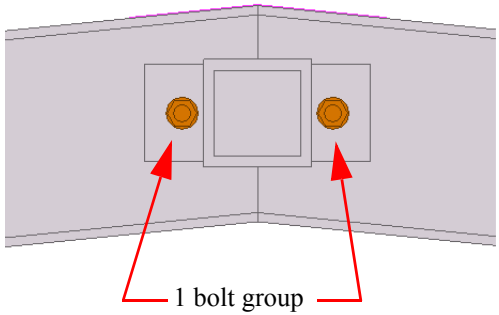
Position in plane: **Left** Neutral line on **0.5**



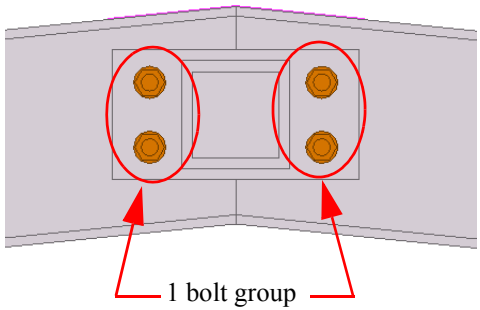
Position in plane: **Right** Neutral line on **0.5**

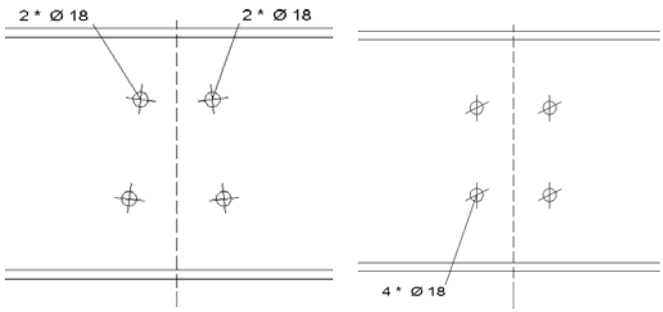
Bolt group in a polybeam

In case a bolt group is connected to the partition of a polybeam segment, the bolt group needs to be split up. So one bolt group for the first segment and 1 bolt group for the other segment.



so not one 2-bolt group





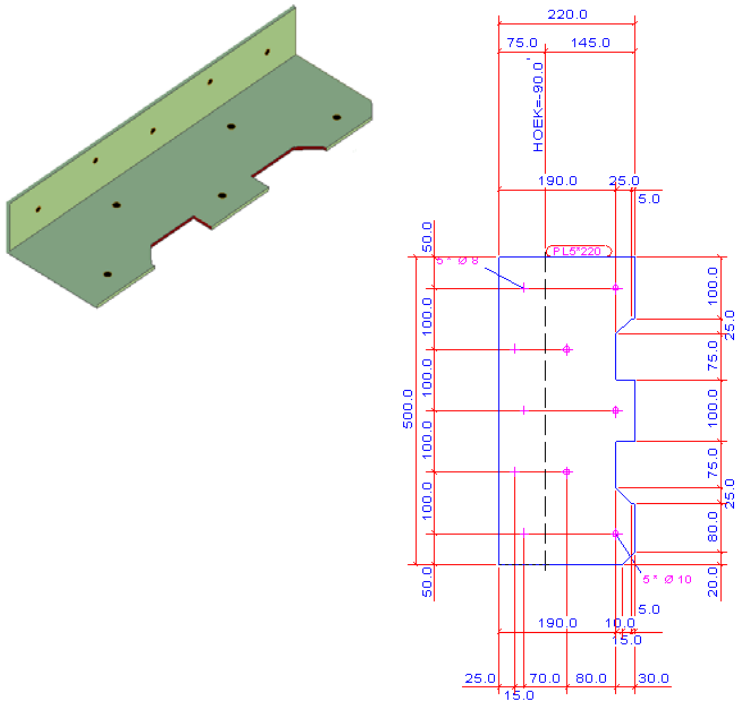
RIGHT **WRONG**

Unfolding profile

This means that if holes are drilled into the profile, the holes will be repositioned when unfolding the profile!

NC files from polybeams

Below there is an example of a NC file of a of a folded plate, setting.



This drawing has been generated with length calculation across the inside of the profile (i.e. .0 in the file `unfold_corner_ratio.inp`).

Also the NC-files are generated according to the settings in the file `unfold_corner_ratio.inp`. Change the value from .0 to .5 and the NC-file will be modified too.

3.51 Web models

Topics

This chapter is divided into following sections:

Conditions (p.512)

Create a web model (p.513)

Open a Web model (p.515)

Handling named views (p.515)

Hide objects (p.517)

Set clip planes (p.517)

Tool tips (p.519)

Full content rendering (p.520)

Pan (p.520)

Rotate (p.520)

Fly (p.520)

Center (p.521)

Home (p.521)

Snapshot to clipboard (p.521)

Change to perspective / orthogonal (p.521)

Background color (p.521)

Presentation (p.521)

Sending a web model (p.521)

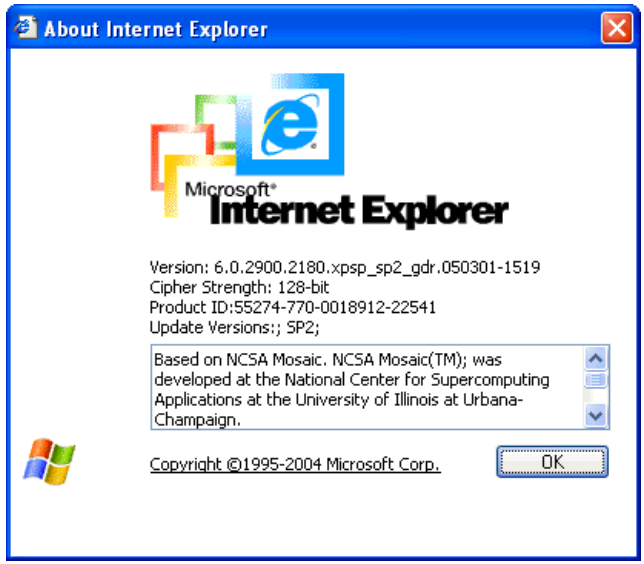
Notes (p.522)

In Tekla Structures you can create a Web model. You can open a web model by means of the Internet Explorer, so you don't need the Tekla Structures programme to view the model. This is very handy if you want to show your model to e.g. a builder or a customer, etc.

Conditions

If you want to open a web model, ***Internet Explorer version 6.0*** must be installed on your computer. Do the following to check this:

1. Open the **Internet Explorer**
2. In the pull-down menu, go to **Help** > **About Internet Explorer**



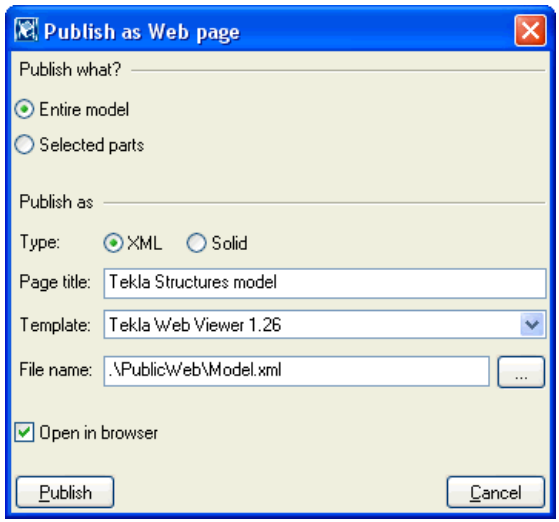
If you have not installed Internet Explorer version 6.0 on your computer, please update your Internet Explorer version (*www.microsoft.com)*.

Create a web model



To create a web model, you do next:

1. In the pull-menu, go to **File** > **Publish as Web page...** or click on the icon, the following dialog box appears:



2. You can now select what you want to publish, either the entire model or selected parts
3. Now select the type of web page:

XML	No bolts are published in the Web model
Solid	Everything is published in the Web model

   Do not change the lines <**Page title**>, <**Template**> and <**Filename**>.
4. Select a checkbox at <**Open in browser**> to open the Web model directly after it is created (Note: the condition is **IE 6.0**)
5. Click <**Publish**>, the Web model is now getting created

   A Tekla Viewer appears in the Task manager on your monitor, next click the Tekla Web Viewer to publish the web model.

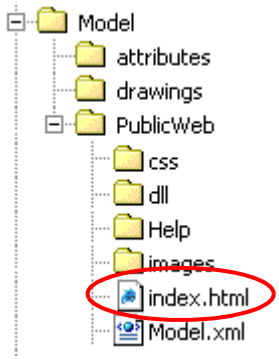




In case a webmodel won't open, import the created web model once as a reference model in your current model. If this is not possible, check your current model again.

Open a Web model

Once you have created a Web model, this web model is stored by default in the current model folder. The folder ***PublicWeb*** is added. This folder contains the file ***index.html***.

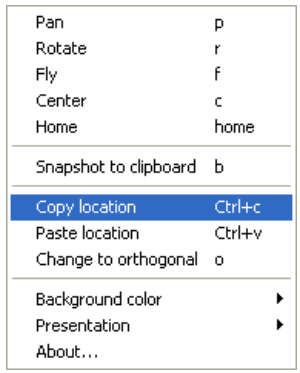


Double-click this file to open the Web model.

Handling named views

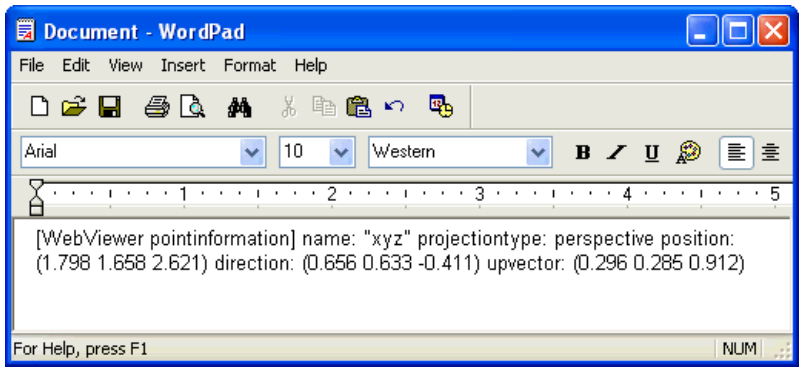
When you send a web model to a builder or customer, sometimes it is necessary that the customer has to take a look at some named views from a published model. This can be indicated in the e-mail, but the following method is easier:

1. Open the published model
2. Zoom in on the part of the model from which you want to create the view
3. Right-click and select Copy location or <**Ctrl**> + C



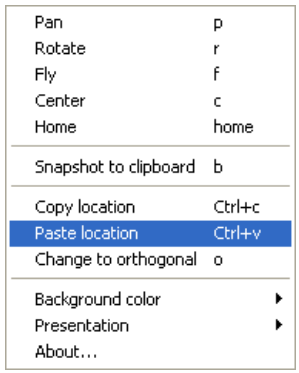
4. Create a new file in any text editor, for example Notepad

5. Paste or <**Ctrl**> + V:



The default name for the view is xyz

6. To change it, replace the text xyz with the view name that you want to use. e.g. “Detail A”. Note: the name starts with a letter!
7. Now copy the updated location information to the published model. For this, select all the text in the text editor
8. Click right mouse and select Copy or <**Ctrl**> + C
9. Go back to the WebViewer
10. Click right mouse on the model and select Paste or <**Ctrl**> + V



The view name appears on the Named views list:



Repeat these steps as many times as necessary. In the webviewer, you will see that the viewpoint automatically changes if you select another viewpoint!

WebViewer does not save named views with the published model. But, you can save the text file that contains the location information, then cut and paste the text to the published model in WebViewer the next time you want to use the view.

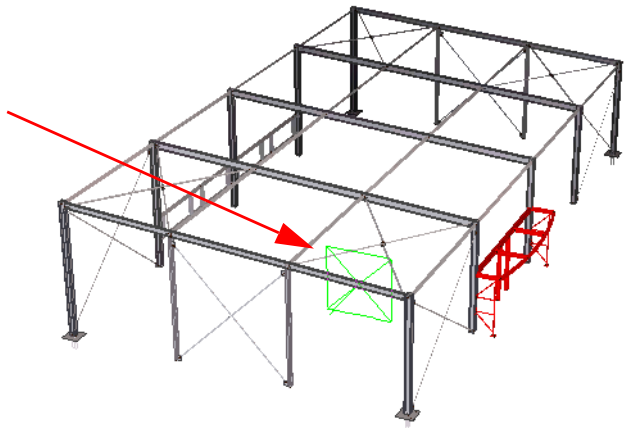
Hide objects

Use the mouse wheel to hide and unhide objects:

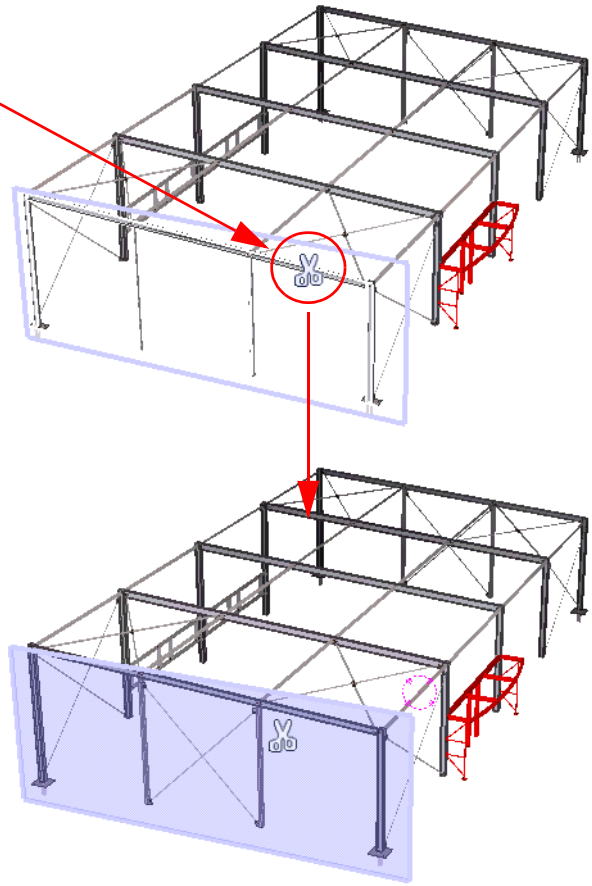
- To hide objects, move the cursor over the objects, hold down the <**Ctrl**> key and scroll forwards (or press the **Page Up** key).
- To unhide object, move the cursor over the hidden objects, hold down the <**Ctrl**> key and scroll backwards (or press the **Page Down** key).
- Press the **Esc** key to show all objects.

Set clip planes

To create a clip plane, click short-cut **p**, and select a part in the web viewer model:

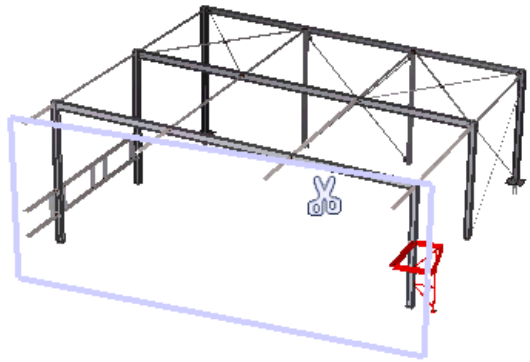


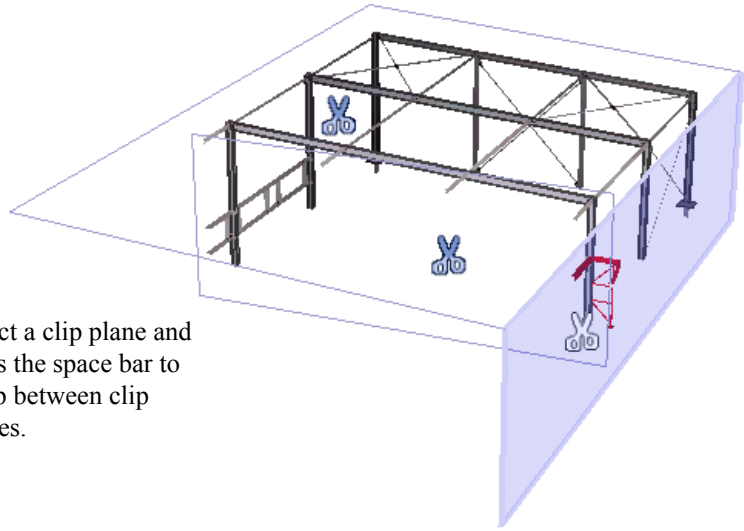
To select the clip plane, click the scissor symbol:



Move the clip plane by dragging the scissor symbol.

You can change the location of the symbol by holding down the **<Shift>** key and moving the symbol.

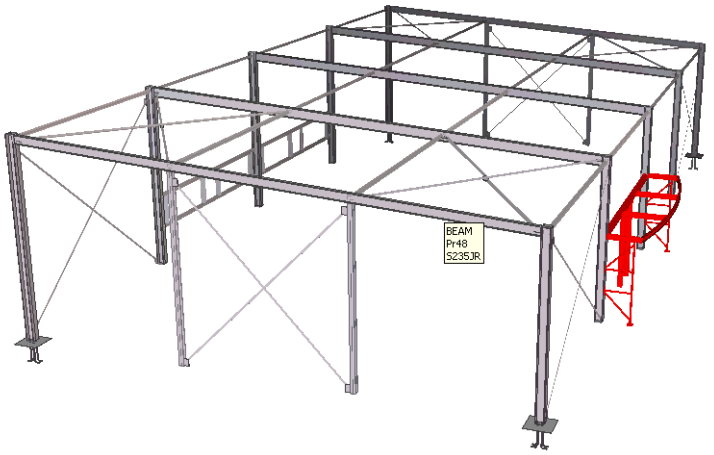




Select a clip plane and press the space bar to jump between clip planes.

Tool tips

To view the part name, position and material, move the cursor over the part:



The tooltips are in use only if the model was published as **XML**.

Full content rendering

To handle large models faster, disable full content rendering:

- Right-click and select **Disable full content rendering** from the popup menu.
- To enable full content rendering, select **Enable full content rendering** from the popup menu.

Pan

Shortcut for **Pan** command is **n**.

Rotate

Shortcut for **Rotate** command is **f**.

Fly

Shortcut for **Fly** command is **f**.

Center

Shortcut for **Center** command is **c**.

Home

Shortcut for **Home** command is **home**.

Snapshot to clipboard

To copy a snapshot to the Clipboard. Shortcut for the command **Snapshot to clipboard** is **b.**

Change to perspective / orthogonal

Shortcut for the command **Change to perspective** is **l.** Shortcut for the command **Change to orthogonal** is **o**. With this you can switch between the two commands.

Background color

To set the background color. You can select **Black**, **Blue** and **White**.

Presentation

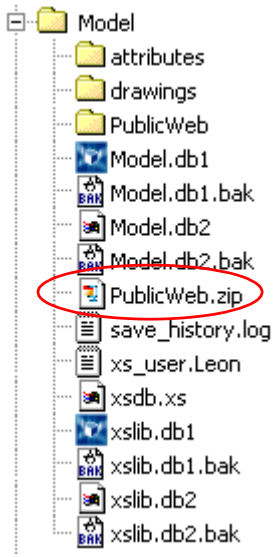
To set the representation of the profiles. You can select **Wire frame** (<**Ctrl**> + 1), **Shaded wireframe** (<**Ctrl**> + 2) en **Rendered** (<**Ctrl**> + 3).

Sending a web model

Sending a web model to e.g. a builder or an architect can be done very easily.

Step plan

1. Go to the model folder and zip the complete folder ***Publicweb*** by means of e.g. Winzip



2. You can now send this *.zip file by e-mail, for instance
3. The receiver must store the *.zip file, which is included as a subject in the e-mail, on e.g. the Desktop
4. Extract the *.zip file
5. Double-click on the file *index.html* to open the web model.



Never try to open a web model while it is in the *.zip file or in the e-mail, this won't work!

Notes

- In case you cannot open a web model, go to ***http://www.construsoft.com/tekla/web_models.php*** and click one of the following pictures:



- Now open a Web viewer models. Close this web model and open a **local** stored web model.

- Always store Web models on your local drive on the PC, **not** on any location on the network (except internet servers, see also the last note), to make sure that you can always open a Web model
- When the Internet Explorer crashes while opening a web model, there are incorrect profile cross sections in the model. This means that there are profiles used in the model containing incorrect cross sections like self defined cross sections. To check if these cross sections are incorrect, create a new web model of selected parts without the parts which possibly contain these incorrect cross sections
- When you copy the folder Publicweb to a internet server, so that everybody can view your web models online, the first time you visit this site, a programme will be installed our your computer to enable viewing such models. The folder ‘dll’ on the internet server must contain the file zkitlib.dll. When customers inform you about the fact that they cannot view web models, make sure that the file zkitlib.dll is uploaded correctly. (the size of this file on the server is not 0 kB)

3.52 Weight, area and volume calculation

Topics

This chapter is divided into following sections:

Weight (p.524)

Area (p.530)

Volume (p.534)

Weight

In Tekla Structures we have two types of profiles.

- Catalog profiles
- Parametric profiles

For calculating weights and surface areas, it is important to know what type of profile it's weight or surface area is asked for. Especially for profiles which look identical at first sight, it may cause differences.

A catalog box girder RHS80*5 may have a different weight than a parametric box girder RHS80/5.

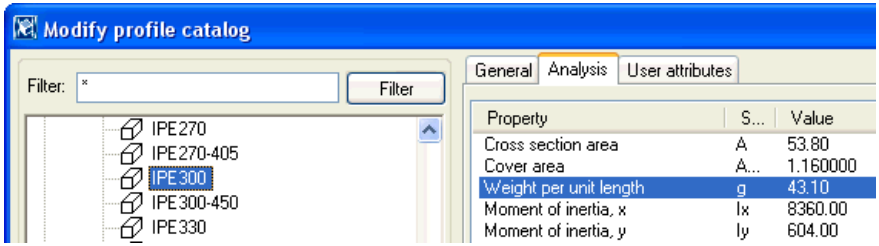
Profiles can be calculated in several ways in Tekla Structures. In the report and template editor are some variables available:

- **Profile_weight** (p.524)
- **Profile_weight_net** (p.525)
- **Weight** (p.526)
- **Weight_gross** (p.527)
- **Weight_net** (p.528)
- **Weight_m** (p.529)

Profile_weight

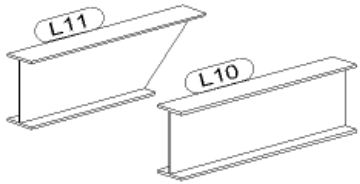
This method of weight calculation counts for all catalog profiles. In the profile catalog, weight is defined per linear meter.

For example, a IPE300 profile is set to 43.1 kg/mtr[1]



For a 1 meter straight IPE300 profile, and report variable **profile_weight**, a weight of 43,1 kg is returned on the list.

The length is calculated using variable LENGTH_GROSS. This is the gross length, which means that possible chamfers are taken into account for the weight calculation. See also the example below.

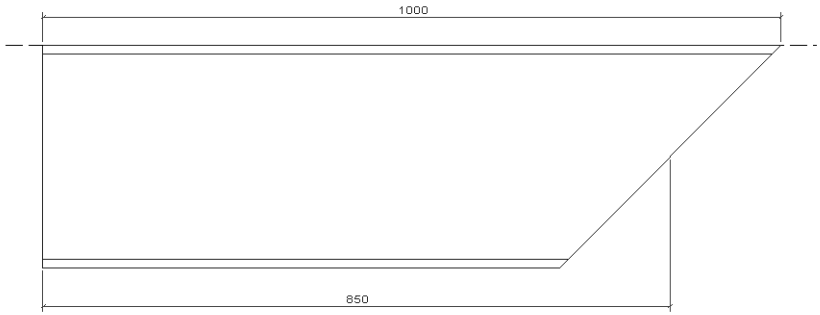



```
===================================
  Part Quant.  Profile    Profile
                          weight
===================================
  L10    1     IPE300     43.10
  L11    1     IPE300     43.10
```

Profile_weight_net

This method generally works the same way as **profile_weight** and therefore counts for catalog profiles.The difference is in the determination of the profile length.

For **profile_weight_net**, the average length is taken.



In the picture above, the IPE300 profile is fitted 45 degrees. The average length will be 850 mm.

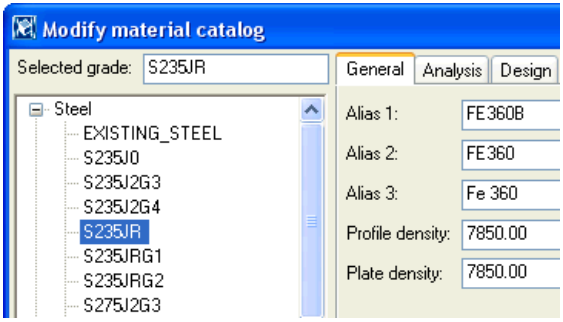
The weight for **profile_weigt_net** will be: 0.85 x 43,1 = 36,63 kg.

However, the gross length of this profile is 1 meter, so the **profile_weight** calculates 43,10 kg

Partno.	Quant.	Profile	Profile Weight	Profile Weight_net
Pr41	1	IPE300	43.1	43.1
Pr42	1	IPE300	43.1	36.6

Weight

For variable **weight**, Tekla Structures determines the profiles' volume. Next, this volume is multiplied with the specific gravity (as defined in the material catalog).



Example

A flat bar profile, size 20*240, lg.600: 0.02 x 0.24 x 0.6 x 7850 = 22,6 kg.

Partno.	Quant.	Profile	weight
PL2	1	PL20*240	22.61

This way of weight calculation is especially used for parametric profiles, because for these profiles no weight per meter is defined.

If the report or template contains the **profile_weight** variable, and a parametric profile is listed in the report (or template), than the weight will be calculated according to the **weight** variable.

Partno.	Quant.	Profile	Profile Weight	Weight
P35	1	PL20*240	22.61	22.61

If the variable **weight** is used in a report, and catalog profiles are listed in the report, the weights are calculated as follows: Tekla Structures takes the section area from the catalog and multiplies this value with the profiles' length and the profiles' specific gravity (material catalog).

Partno.	Quant.	Profile	Profile weight	Profile we_net	weight	gross
L10	1	IPE300	43.10	43.10	42.23	42.23
L13	1	IPE300	43.10	43.10	43.04	43.04

There is a difference in weight of assembly L10 between **profile_weight** and **weight**. This is because trade weights are used while calculating it's weight for **profile_weight**.

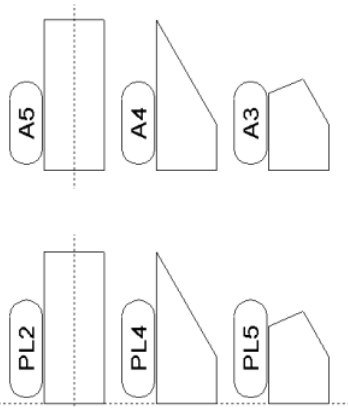
Calculating these weights, a specific gravity of 8000 kg/m³ is used. In the material catalog, a default specific gravity of 7850 kg/m³ is set. For assembly P35 - in the report on the previous page - this value has been increased to 8000 kg/m³, this makes that the weights become equal again.

Weight_gross

Weight_gross returns the gross weight for contour plates and folded plates. In other words, the weight of a rectangular plate which is initially needed for creating the part.

For plates modelled as a beam with setting **plate**, and afterwards edited using Fit-operation(s), **weight** and **weight_gross** variables will return the same weight values.

Weight_gross only returns a different value compared to variable **weight** in case of contour plates.



Plates modelled as a beam with plate setting

Pos	Aant	Profiel	Profile weight	Profile we_net	weight	gross
A3	1	PL20*240	11.64	11.64	11.64	13.75
A4	1	PL20*240	14.78	14.78	14.78	22.61
A5	1	PL20*240	22.61	22.61	22.61	22.61
PL2	1	PL20*240	22.61	22.61	22.61	22.61
PL4	1	PL20*240	14.78	14.78	14.78	14.78
PL5	1	PL20*240	11.64	11.64	11.64	11.64

Plates modelled as a contour plate

In the picture above is shown that **profile_weight** and **profile_weight_net** return the same value for all profiles; the difference for variable **weight_gross** is due to the way of modelling the plate.

For catalog profiles, there is no difference in variables **weight** and **weight_gross**.

It is not possible to inquire the gross weight for plates modelled as beam profiles. For these profile types, the same weight is returned for all weight-variables. (= the gross weight, exclusive the chamfered part)

If the gross weight is wanted, a formula can be entered in a report:

=MATERIAL.PLATE_DENSITY*LENGTH_GROSS*WIDTH*HEIGHT*0.00 0000001

Another option to inquire the gross weight of platen - modelled as beam -, is to add the plates as profiles to the profiles catalog. The weight (kg/m) can be entered as a fixed value in tab 'Analysis'.

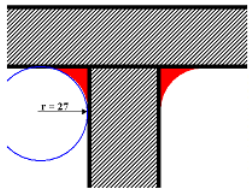
The variable **profile_weight** will return the parts‘ gross weight.

Weight_net

In case variable **weight_net** is used, the weight is calculated on the basis of the profiles’ volume (for both catalog profiles and parametric profiles). This volume will be multiplied with the specific gravity (material catalog).

For parametric profiles, the **weight_net** value will not differ from the **weight** value.

For catalog profiles, the calculated weights will differ, because the radius’ are not taken into account. For all previous described variables, the radius is included: or by weight per linear meter or by profiles’ cross section.



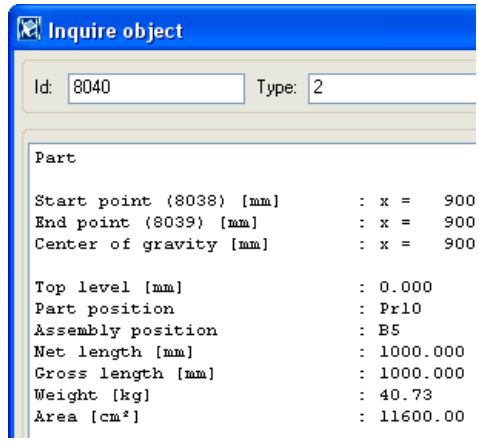
Example

IPE300, length 1 meter:

Flanges 2 x 0,0107 x 0,15 x 7850 = 25,20 kg + web (0,3 - 2 x 0,0107) x 0,0071 x 7850 = 15,53 kg

Total: 40.73 kg.

```
=========================================
Partno. Quant.  Profile   Profile  weight net
                          weight
=========================================
  L10     1     IPE300     43.10     40.73
```





The weights returned using the **Inquire** > **Object...** command, are the same as the weights returned by variable **weight_net**.

Because in most reports the variable **profile_weight_net** is used, it may happen that the weights returned by 'Inquire object' dialog box differ from the weights returned in reports.

Weight_m

The variable **weight_m** returns the proper weight (per meter) of catalog profiles. This can be used e.g. in reports, first display the profiles' total amount of meters and than multiplying that value with the **weight_m** value to calculate the weight.

Example: beam IPE300, length 1 meter, has a weight_m of 43.10 kg. A IPE300 beam length 3,5 meter too.

For parametric profiles, the weight of the profile is displayed, divided by the length. See following example the difference between box girder K80/5 (database) and box girder K80*5 (parametric)

For parametric plates, variable **weight_m** returns a '0' value.

Partno.	Quant.	Profile	Profile weight	weight_m
L10	1	IPE300	43.10	43.10
L15	1	IPE300	150.8	43.10
PL13	1	STRIP10*30	16.89	0.00
L38	1	K80/5	11.27	11.27
L39	1	K80*5	11.77	3.59

If the variable **weight_m** is used for plates, they have to be added as a catalog profile in the profile catalog. Enter also its' weight per linear meter. The variable **weight_m** does not apply to contour plates.

Catalog profiles and parametric profiles

Considering the different methods of calculating weights, for correct weights on reports it's very important that the profile is described correctly in the part properties dialog box. For HE* and IPE profiles this will not cause problems, because these profiles are not considered as parametric profiles by Tekla Structures. However, a box girder 80x80x5 can be entered as catalog profile as RHS80*5, while an input of RHS80/5 will display exactly the same profile, but now it is a parametric profile. The same counts for angle steel profiles. A L200/150/15 (catalog) profile will look the same as a L200*150*15 (parametric), but may have a different weight in reports and/or templates (depending of the defined variable)

Partno.	Quant.	Profile	Profile weight	Profile we_net	weight	gross	weight net
L20	1	K80*5	12.80	12.80	12.80	12.80	12.80
L18	1	K80/5	12.25	12.25	12.25	12.25	12.80
L21	1	L200*100*15	36.49	36.49	36.49	36.49	36.49
L19	1	L200/100/15	37.19	37.19	36.49	36.49	36.49

Area

Just like for weights, also for defining the profile area a number of variables are available. Some of the variables return a different value for catalog profiles and parametric profiles.

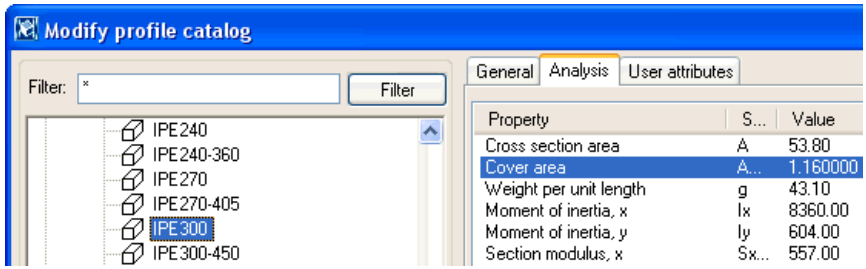
In the report editor, among others the next variables are available:

- **Area** (p.531)
- **Area_net** (p.531)
- **Area_gross** (p.532)
- **Area_plan** (p.532)
- **Area_per_ton** (p.533)
- **Profile_cover_area** (p.534)

Area

For variable **area** the **Cover area**-value is taken from the profile catalog, multiplied with the profiles' length.

The value set in the catalog is the so called paint area per square meter. This is exclusive the profiles' sides.




```
==============================
Partno. Quant.  Profile Area

==============================
  L10     1    IPE300     1.16
```

For plates (parametric profiles), the actual area (2x length x width + 2x length x height + 2x height x width) is calculated by Tekla Structures. In this case the profiles' sides are taken into account also.

Example

A plate 50x200, long 800 mm. The area of the profile will be: 2x 0.8 x 0,05 + 2x 0.8 x 0,2 + 2x 0,2 x 0,05 = 0,08 + 0,32 + 0,02 = 0,42 m²

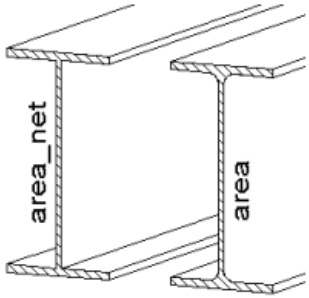
```
==============================
 Partno. Quant.  Profile    Area

==============================
  PL1     1    PL50*200      0.42
```

Area_net

This is the so called nett cover area. For catalog profiles as well as parametric profiles, the area is calculated from the model geometry.

For catalog profiles this may return in a higher value (radius included) and for parametric profiles this returns the same value as *area*, because the area was calculated in the same way in that case.



Partno.	Quant.	Profile	Area	Area _net
L1	1	IPE300	0.93	0.96
PL1	1	PL50*200	0.42	0.42

Area_gross

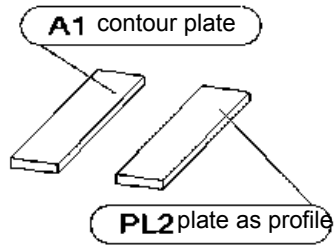
The returned value for this variable is the same as for **area**.



This concerns both catalog profiles and parametric profiles.

For contour plates, the area is calculated by multiplying the extreme length and width of the plate. So, not the area of the entire plate is returned (top face, bottom face, sides), but only one face.

Partno.	Quant.	Profile	Area	Area _gross
A1	1	PL30*150	0.30	0.12
PL2	1	PL30*150	0.30	0.30

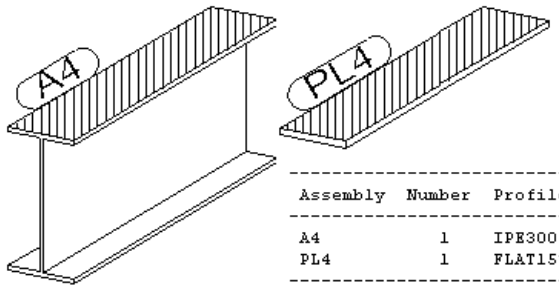


Area_plan

The variable **area_plan** returns the plane area of a profile (perpendicular to the global z-axis)

In other words: length x width of a profile.

For an IPE300, length 800 mm and a plate 150 x 800 (thickness makes no difference), the value **area_plan** is identical.

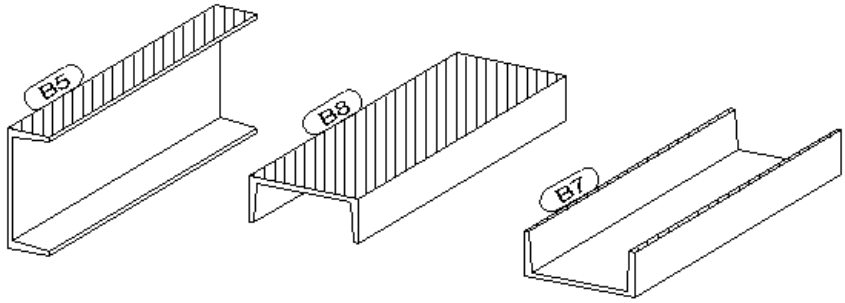


Assembly	Number	Profile	Area(m2)	Area_plan
A4	1	IPE300	0.93	0.12
PL4	1	FLAT15*150	0.27	0.12

For a UNP200, three **area_plan** values can be calculated, depending how the profile is positioned in the model.

For determining the **area_plan**, only the top plane is taken into account (perpendicular to the global z-axis).

For more clearness, see the following example:



Assembly	Number	Profile	Area(m2)	Area_plan
B5	1	UNP200	0.43	0.05
B7	1	UNP200	0.43	0.01
B8	1	UNP200	0.43	0.13



For profiles, which top planes are not perpendicular to the global z-axis, a value of 0.00 will returned for **area_plan**.

Area_per_ton

This variable returns the value *Area divided by weight x 1000*.

For catalog profiles this is a fixed value, irrespective of the length of the profile.

For parametric profiles this value is calculated per individual part. Because the profiles' sides are also taken into account, small differences may occur.

Assembly	Number	Profile	Length	Area_per_ton
B10	1	IPE300	800.00	27.44
B11	1	IPE300	1600.00	27.44
PL10	1	FLAT30*150	1600.00	10.35
PL11	1	FLAT30*150	800.00	10.51

Profile_cover_area

For this variable, a fixed value (mm² per linear meter) is returned for catalog profiles. This value is always equal, so independent of the profile length.

For parametric profiles, no value (i.e. 0) is returned.

Assembly	Number	Profile	Length	Prof_Cov_area(mm2)
B10	1	IPE300	800.00	1159000
B11	1	IPE300	1600.00	1159000
PL10	1	FLAT30*150	1600.00	0
PL11	1	FLAT30*150	800.00	0



Just as for the weights, the notation of profiles is important for area values also. By entering the profile name manually in the input field, distinction can be made between catalog profiles and parametric profiles (e.g. tubes and RHS-profiles). Although this is not noticeable in the model, this may affect the area values in templates and reports.

Volume

For calculating volumes, the following variables are available:

- Volume
- Volume_net
- Volume_gross

Catalog profiles and parametric profiles have their own method of calculating the volume. For catalog profiles, the required information is taken from the profile catalog. For parametric profiles the model parameters length, height and width are used for calculating the volume.

In the catalog, in tab *Analysis*, the cross section of catalog profiles is defined. This value is multiplied by the length of the profile to return the volume.

For parametric profiles counts: length x width x height.

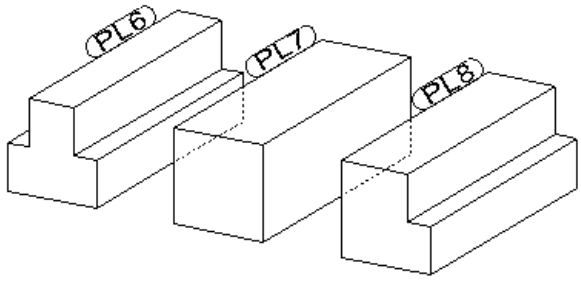
Example

A HEB600, length 5 meter: A= 270 cm².= 0,027 m² x 5m = 0.14 m³

Plate 400*400 length 4 m = 0.4 x 0.4 x 4 = 0.64 m³

Assembly	Number	Profile	Volume_net
B9	1	HEB600	0.16
PL5	1	PL400*400	0.64

For parametric profiles, the calculated volume is always correct. All chamfers, line cuts, negative volumes, etc. are taken into account.



Assembly	Volume
PL6	0.14
PL7	0.18
PL8	0.16

The output is equal for variables **volume**, **volume_net**, **volume_gross**.

For catalog profiles, the variable **volume_net** is also calculated in this way.

For HE- and I-profiles the nett volume shall (due to the fact that the radius is not taken into account) be a little lower than by **volume** or **volume_gross**.

Assembly	Number	Profile	Length	Volume	Volume_NET
B12	1	HEA1000	6000.0	0.21	0.20

Assemblies

In above paragraphs is described the weight, the area or the volume of a parts only. This can be both main parts as secondary parts.

It can be useful/ necessary to display the weight or area of an entire assembly on reports. To achieve this, a variable addition **ASSEMBLY_** has to be added.

Example: **ASSEMBLY_WEIGHT** returns the assembly weight just like the variable **WEIGHT** does. The same counts for **ASSEMBLY_AREA_PLAN**, this variable returns the **AREA_PLAN** of an assembly.

Total number

It is also possible to multiply the parts' weight, area, volume by the total amount of parts existing in the model.

To achieve this, a **N_** must be put in front of the variables.

Example

N_WEIGHT_NET returns the weight of the total number of parts listed in the report.

3.53 Xstwerkv

Topics

This chapter is divided into following sections:

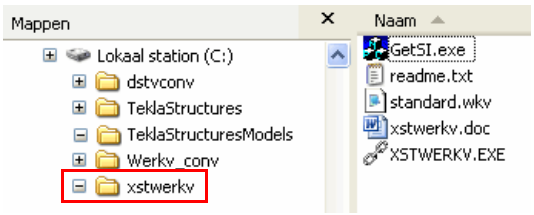
Exporting from Tekla Structures to Scia Cimsteel (p.536)

Exporting from Tekla Structures to Concreet and Matrix (p.540)

Instellingen algemeen (p.541)

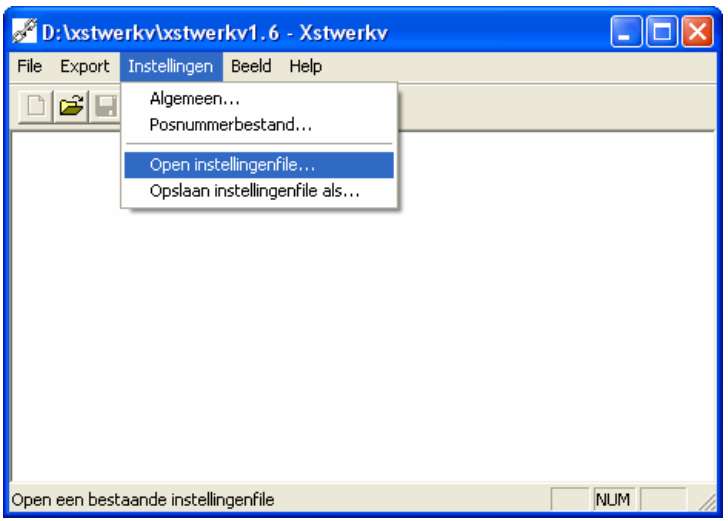
Exporting from Tekla Structures to Scia Cimsteel

Create a folder in which you can store the program Xstwerkv, e.g. the folder xstwerkv:

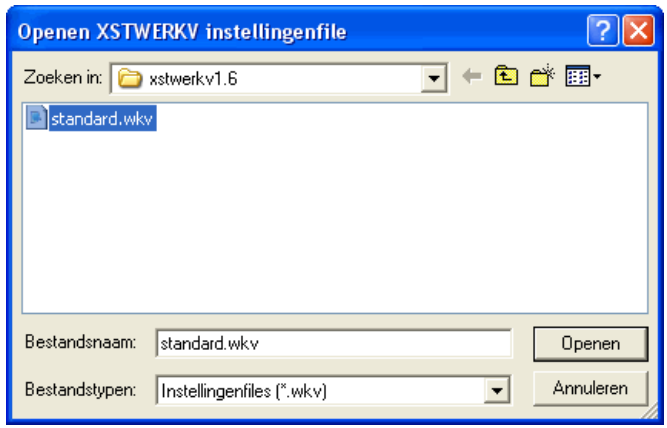


Step plan

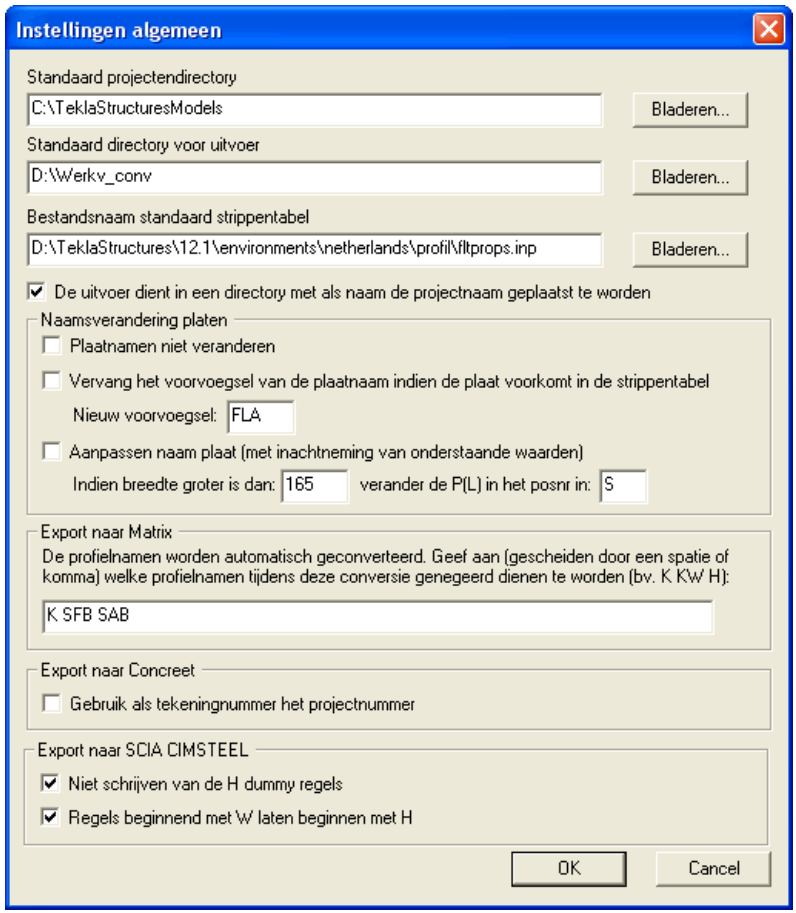
1. Create the report `werkvcim.xsr` in your model. The created report will be stored in the folder **Reports** in the model folder.
2. Double-click on the program **xstwerkv.exe**, the following dialog box appears:



3. Go to **Instellingen**> **Open instellingenfile..**

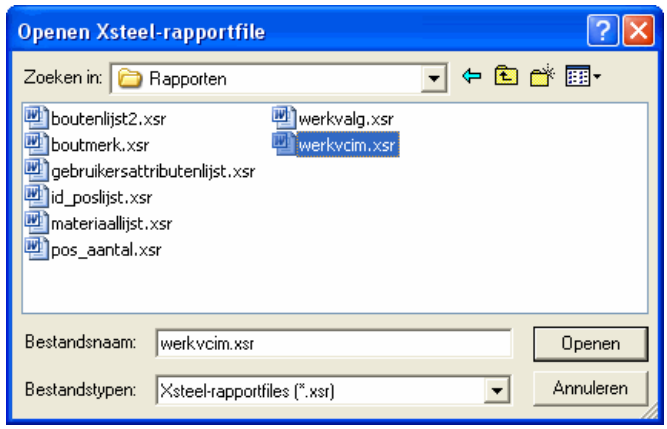


4. Select the file **standard.wkv** and click on the button <**Openen**>, the following dialog box appears:

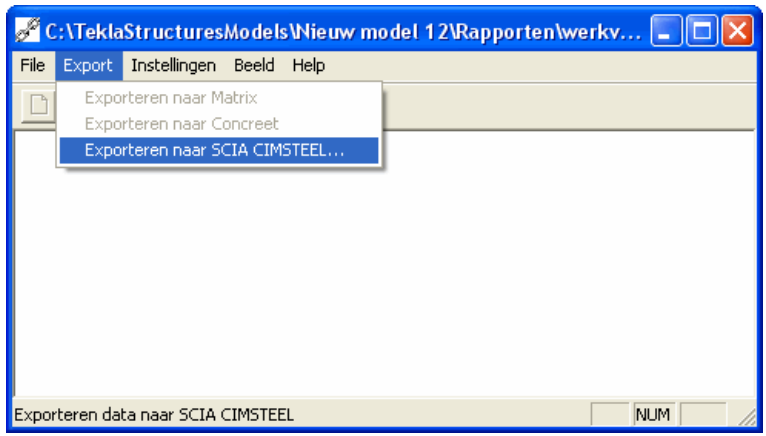


5. Modify, if needed, the settings, for example the flat bar table that should be used and click <**OK**>
6. Go to **File** > **Open...** or click on the icon

7. Browse to the report `werkvcim.xsr`



8. Click <**Openen**>



9. Go to **Export** > **Exporteren naar SCIA CIMSTEEL...** and change, if desired, the filename



10. Click the button <**OK**>, the conversion will start running



Exporting from Tekla Structures to Concreet and Matrix

Exporting to Concreet and Matrix is almost like exporting to Scia Cimsteel. Instead of the report `werkvcim.xsr` you need te use the report `werkvalg.xsr`.

Summery

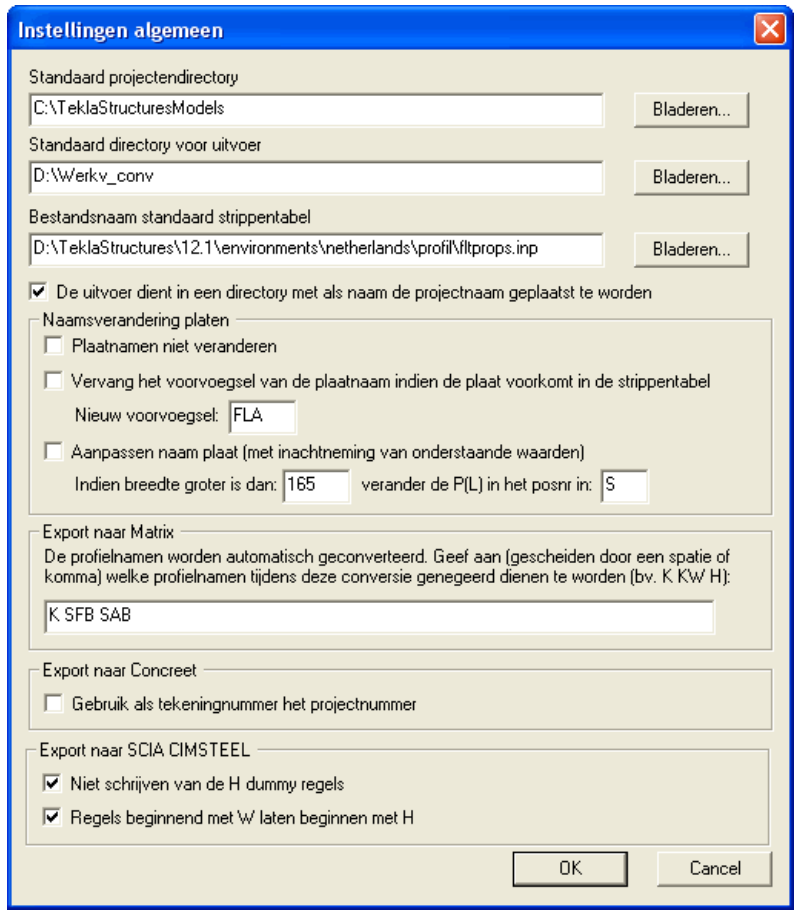
The output consist of the following files:

SCIA Cimsteel	Concreet	Matrix
Cimsteel.txt	Concrmerk.txt	Elem.csv
	Concrpos.txt	Elempos.csv
	Concrtek.txt	Pos.csv



Export to Fabtrol can be done from Tekla Structures directly.

Instellingen algemeen



Standaard projectendirectory

Here you can enter the folder in which you start browsing for the concerned report. Click <**Browse**> om de map aan te passen.

Standaard directory voor uitvoer

Here you can enter the folder in which you want to store the output.

Bestandsnaam standaard strippentabel

Here you can enter the location of the flat bar table.

Plaatnamen niet veranderen

Checkbox to change the plate names.

Vervang het voorvoegsel van de plaatnaam indien de plaat voorkomt in de strippentabel

Checkbox to replace the plate names' prefixes if it exists in the flat bar table.

Aanpassen naam plaat (met inachtneming van onderstaande waarden)

In case the with of the plate is larger then a specific value(e.g. 165), the letter P in the name will be changed into another letter (e.g. S).

Example

PL10*170 will become S170/10

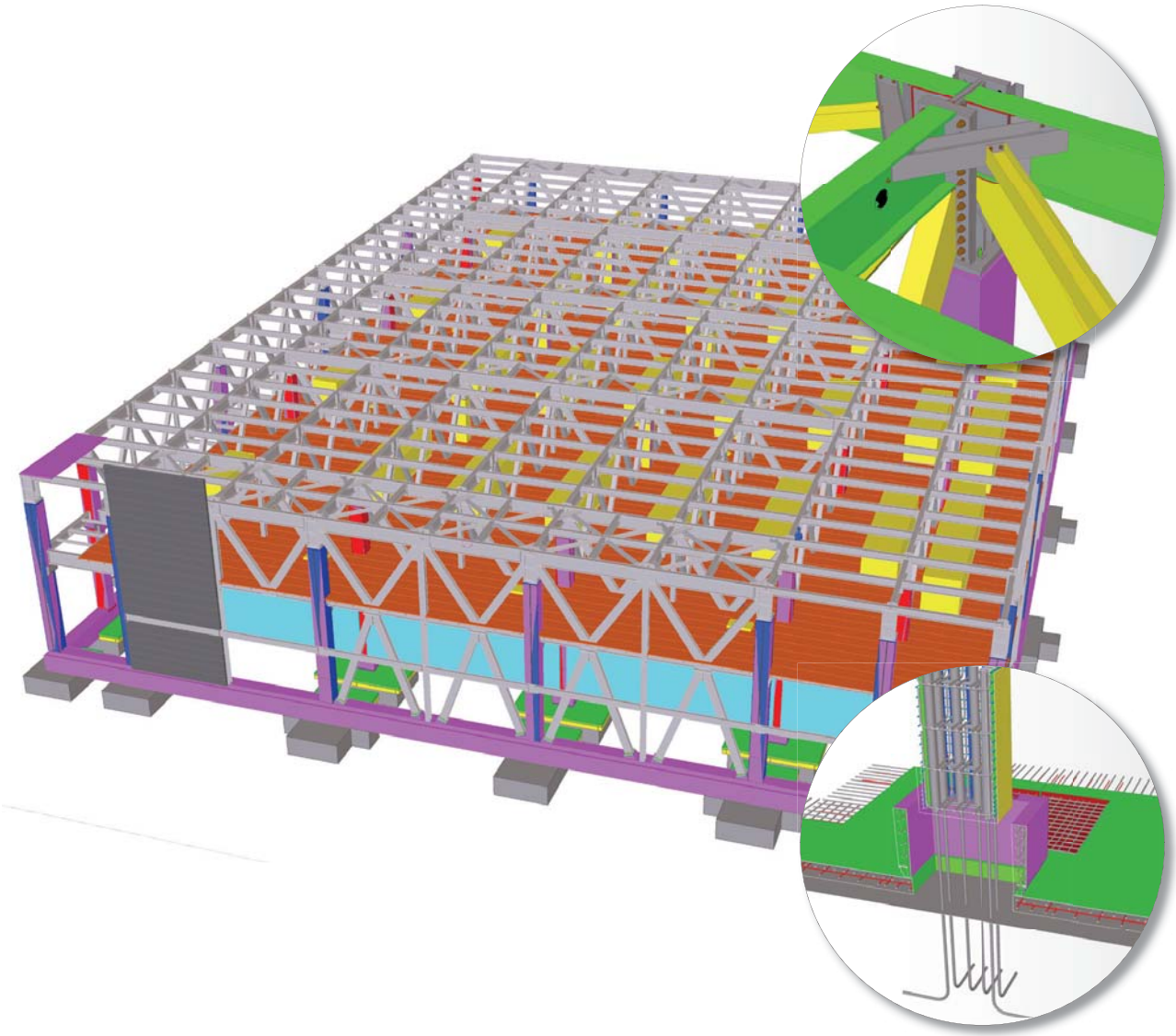
PL10*160 will become P160/10

construsoft

TEKLA Structures

Reinforcement

Basic training



Productversion 13.1

May 2008

All rights reserved. No part of the contents of this manual may be reproduced or transmitted in any form or by any means wihout the permission of Construsoft Inc.

Construsoft Inc. is not responsible for any consequences as a result of using Tekla Structures.

© 2008 Tekla Corporation and its licensors. All rights reserved.

This Software Manual has been developed for use with the referenced Software. Use of the Software, and use of this Software Manual are governed by a License Agreement. Among other provisions, the License Agreement sets certain warranties for the Software and this Manual, disclaims other warranties, limits recoverable damages, defines permitted uses of the Software, and determines whether you are an authorized user of the Software. Please refer to the License Agreement for important obligations and applicable limitations and restrictions on your rights.

In addition, this Software Manual is protected by copyright law and by international treaties. Unauthorized reproduction, display, modification, or distribution of this Manual, or any portion of it, may result in severe civil and criminal penalties, and will be prosecuted to the full extent permitted by law.

Tekla, Tekla Structures, Xcity, Xengineer, Xpipe, Xpower, Xsteel, and Xstreet are either registered trademarks or trademarks of Tekla Corporation in the European Union, the United States, and/or other countries. Other product and company names mentioned in this Manual are or may be trademarks of their respective owners. By referring to a third-party product or brand, Tekla does not intend to suggest an affiliation with or endorsement by such third party and disclaims any such affiliation or endorsement, except where otherwise expressly stated.

Portions of this software:

2D DCM © 1989-2004 D-Cubed Limited. All rights reserved.

EPM toolkit © 1995-2004 EPM Technology a.s., Oslo, Norway. All rights reserved.

XML parser © 1999 The Apache Software Foundation. All rights reserved.

Project Data Control Library © 2006 - 2007 DlhSoft. All rights reserved.

DWGdirect, DGNdirect and OpenDWG Toolkit/Viewkit libraries © 1998-2005 Open Design Alliance. All rights reserved.

Analysis Engine included in Tekla Structures uses a program copyrighted by and is the property of Research Engineers International ©. All rights reserved.

FLEXnet Copyright © 2006 Macrovision Corporation and/or Macrovision Europe Ltd. All Rights Reserved. This product contains proprietary and confidential technology provided by and owned by Macrovision Europe Ltd., UK, and Macrovision Corporation of Santa Clara, California, U.S.A. Any use, copying, publication, distribution, display, modification, or transmission of such technology in whole or in part in any form or by any means without the prior express written permission of Macrovision Europe Ltd. and Macrovision Corporation is strictly prohibited. Except where expressly provided by Macrovision Europe Ltd. and Macrovision Corporation in writing, possession of this technology shall not be construed to confer any license or rights under any of Macrovision Europe Ltd. and Macrovision Corporation's intellectual property rights, whether by estoppel, implication, or otherwise.

Elements of the software described in this Manual may be the subject of pending patent applications in the European Union and/or other countries.

4 Tekla Structures 13.1 Reinforcement 1

4.1 Modeling reinforcement manually 1
- Reinforcing bar 2
- Reinforcing bar group 10
- Curved reinforcing bar group 15
- Circular reinforcing bar group 16
- Reinforcement strand pattern 17
- Reinforcing mesh 18
- Attaching rebars to another part 19

4.2 Modeling reinforcement tips 21
- Reinforcing identical cast units 21
- Modeling reinforcement using circular parts 22

4.3 Reinforcement components 23
- Beam reinforcement (63) 23
- Rebar in beam (90) 25
- Examples Reinforcement components 26

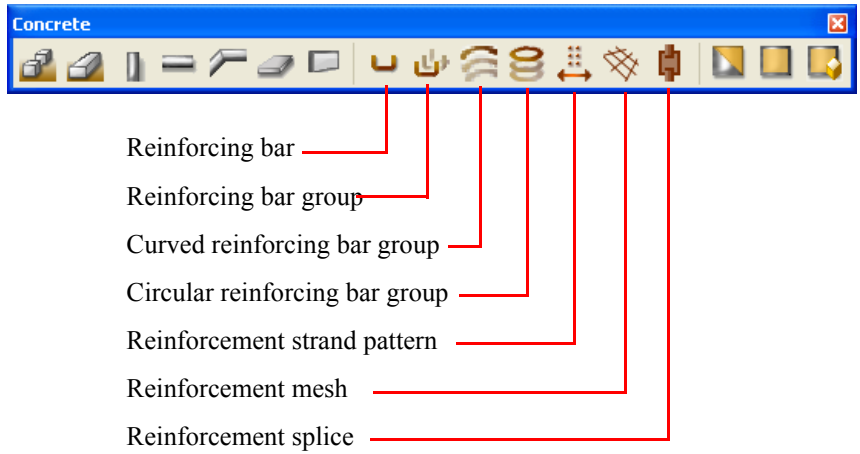
4.4 Component examples reinforcement 27
- Border rebar (93) 27
- Braced girders (88 and 89) 28
- Rebar in beam (90) 29
- Border rebar (92) 30
- Rebar conn 2 beams (94) 31
- Longitudinal reinforcement (70) 31
- Beam reinforcement (63) 32
- Pad footing reinforcement (77) 32
- Beam end reinforcement (79) 32
- Lifting anchor (80) 33
- Round column reinforcement (82) 33
- Rectangular column reinforcement (83) 33
- Hole reinforcement for slabs and walls (84) 34
- Hole creation and reinforcement (85) 34
- Starter bars for pillar (86) 34
- Starter bars for footing (87) 35
- Slab bars (18) 35

Reinforcement mesh array in area (89) 35
4.5 Reinforcement representation and marks in drawings 36
Reinforcement marks 38
Dimension marks 44
Create dimension line 45
Setting dimension marks in the drawing properties 45
Practical classifier example: Reinforcement mark 47
4.6 Appendix 48
Reinforcement related reports 48
Reinforcement related templates 51

4 Tekla Structures 13.1 Reinforcement

4.1 Modeling reinforcement manually

Concrete parts can be provided with reinforcement for higher strength. In the **Concrete** toolbar various commands are available which can be used to create reinforcement.

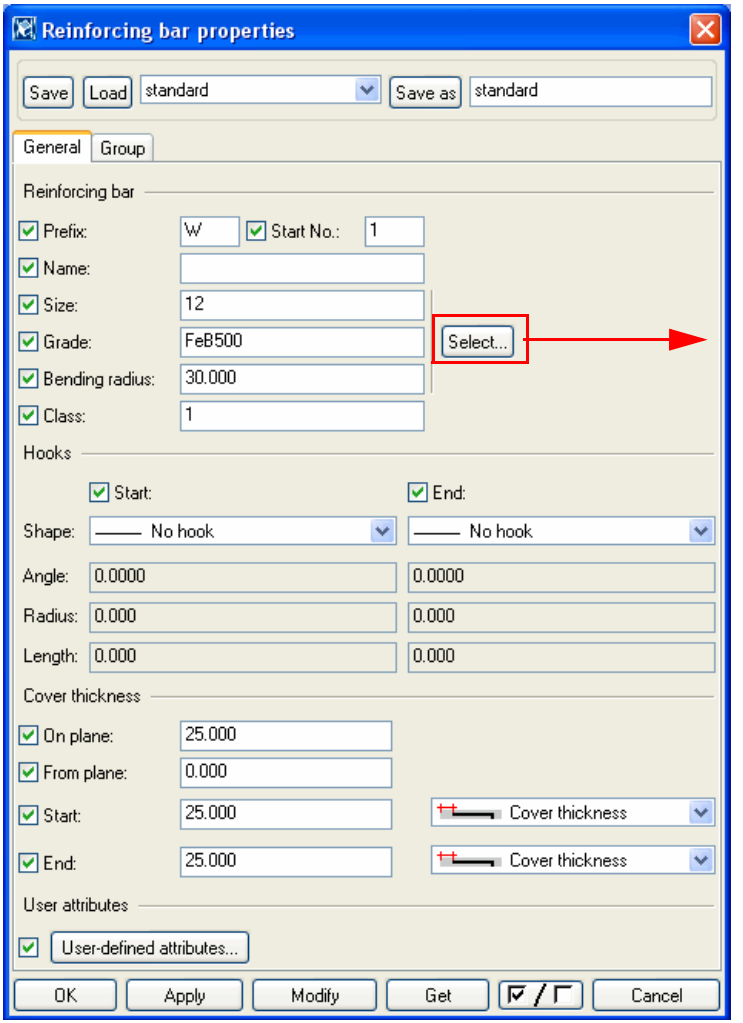


Reinforcing bar



With command **Create reinforcing bar** you can create a single reinforcement bar. This can be a straight bar, but also a hairpin, former or a bracket. Any shape with random dimensions can be modelled.

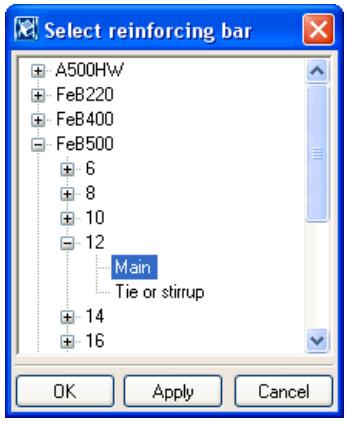
Double-click the icon **Create reinforcing bar** to open the dialog Reinforcing bar properties:



Link to *rebar_database.inp*

In this dialog box, the series and start number and possible names of the bars can be set. For **Size** (=diameter), **Grade** and **Bending radius**, the desired values can be entered. Remember that the grade and size of the bars have to be defined in the file *rebar_database.inp*, otherwise you will receive a error message. By clicking the **Select** button, you will be linked to the file *rebar_database.inp* (=reinforcement-database, this file is located in folder:

disk:\TeklaStructures\version\environments\europe\system.



With the tree structure, you can easily select a bar grade, diameter and application. For Grade you can choose between **Main** and **Tie** or Stirrup. This difference is made because in this way different bending radius can be defined for curved main reinforcement and stirrups.

These values can be modified in file *rebar_database.inp* in the folder:

disk:\TeklaStructures\version\environments\europe\profil

The file can be opened in any text processor

rebar_database - Notepad

File Edit Format View Help

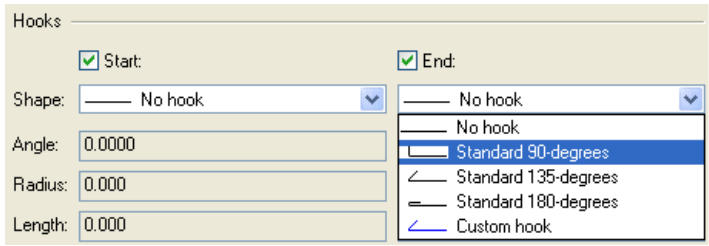
/* Code	Grade	Size	Usage	Nominal diameter	Actual diameter	Min bend radius	Weight [kg/m]	Area [mm2]
/*								
NEN	FeB500	6	tie/stirrup	6	8	15	0.222	28.300
NEN	FeB500	8	tie/stirrup	8	10	20	0.395	50.300
NEN	FeB500	10	tie/stirrup	10	12	25	0.617	78.500
NEN	FeB500	12	tie/stirrup	12	14	30	0.888	113.000
NEN	FeB500	14	tie/stirrup	14	16	35	1.234	154.000
NEN	FeB500	16	tie/stirrup	16	19	40	1.580	201.000
NEN	FeB500	20	tie/stirrup	20	23	50	2.470	314.000
NEN	FeB500	25	tie/stirrup	25	29	63	3.850	491.000
NEN	FeB500	32	tie/stirrup	32	37	80	6.310	804.000
NEN	FeB500	6	main	6	8	15	0.222	28.300
NEN	FeB500	8	main	8	10	20	0.395	50.300
NEN	FeB500	10	main	10	12	25	0.617	78.500
NEN	FeB500	12	main	12	14	30	0.888	113.000
NEN	FeB500	14	main	14	16	35	1.234	154.000
NEN	FeB500	16	main	16	19	40	1.580	201.000
NEN	FeB500	20	main	20	23	50	2.470	314.000
NEN	FeB500	25	main	25	29	63	3.850	491.000
NEN	FeB500	32	main	32	37	80	6.310	804.000
NEN	FeB400	6	tie/stirrup	6	8	15	0.222	28.300
NEN	FeB400	8	tie/stirrup	8	10	20	0.395	50.300
NEN	FeB400	10	tie/stirrup	10	12	25	0.617	78.500
NEN	FeB400	12	tie/stirrup	12	14	30	0.888	113.000
NEN	FeB400	16	tie/stirrup	16	19	40	1.580	201.000

For the various steel grades, every diameter is listed twice in the file (or can be added manually). Once as **main**, and once as **tie/stirrup** (respectively for main and tie/stirrups). The column *min bend radius* contains the bending mandrel that is used if the bar is bend in the model. In case that they will be bend with an angle of 90, 135 or 180 degrees (column alpha), the column "r" contains the accompanying mandrels.

You can edit the bars' diameter - if you want to use the correct bending radius - by using the button **Select**. If you only change the diameter in the field *Size*, the mandrel will keep it original diameter.

For **Class**, the bar color can be defined. We recommend to make use of this feature; if all bars have the same color in the model, there is more clearness in 3D-(rendered) views. In the different view- and select filters you can filter single 'Class'-values.

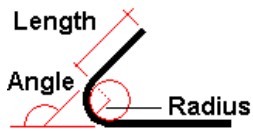
In the dialog, for **Hooks**, you can define a hook or a splice for the stirrups.

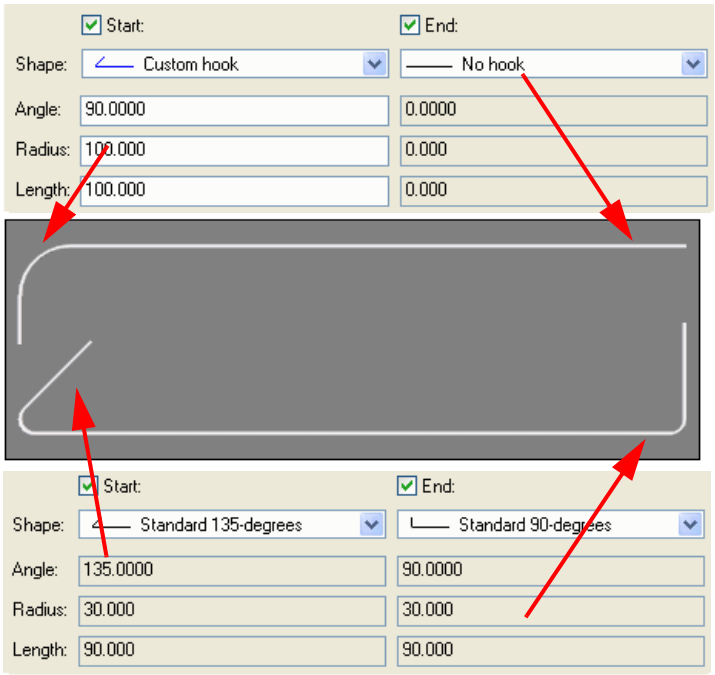


This hook doesn't have to be modelled; it will be added to the bar automatically. The options are:

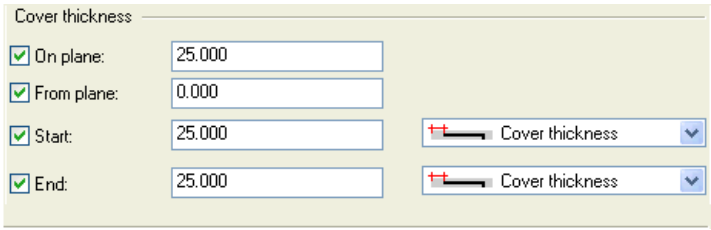
Option	Description
	No hook
	Standard 90° hook
	Standard 135° hook
	Standard 180° hook
	Custom hook

If a standard-hook is selected, then the fields **Hook**, **Radius** and **Length** will be greyed-out. These values will be read from the file *rebar_database.inp*. If Custom hook is selected, then the values can be defined.

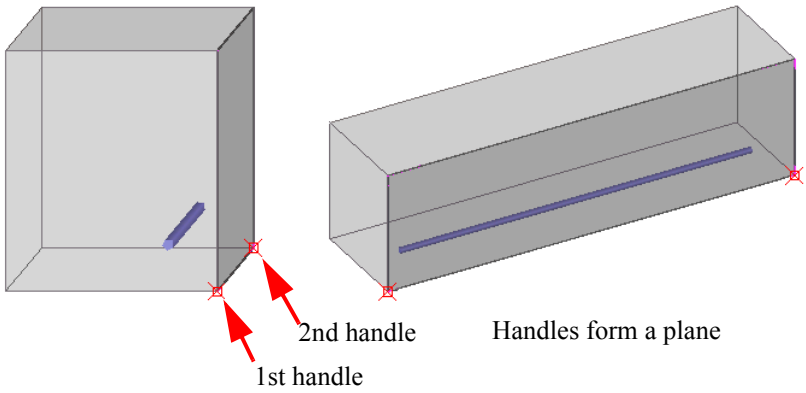
Field	Description	
Angle	Enter a value between –180 en +180 degrees.	
Radius	Internal bending radius of the hook.	
Length	Length of the straight part of the hook.	

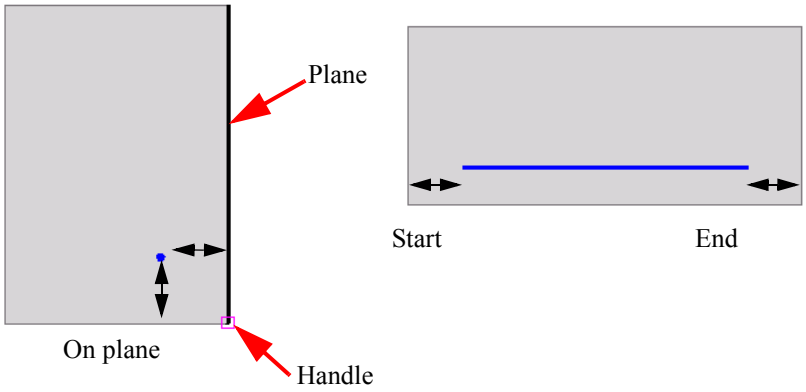


For **Cover Thickness**, the bar cover can be defined. This can be done in three directions: *On plane*, *From plane* and in longitudinal direction of the bar (start/ end:

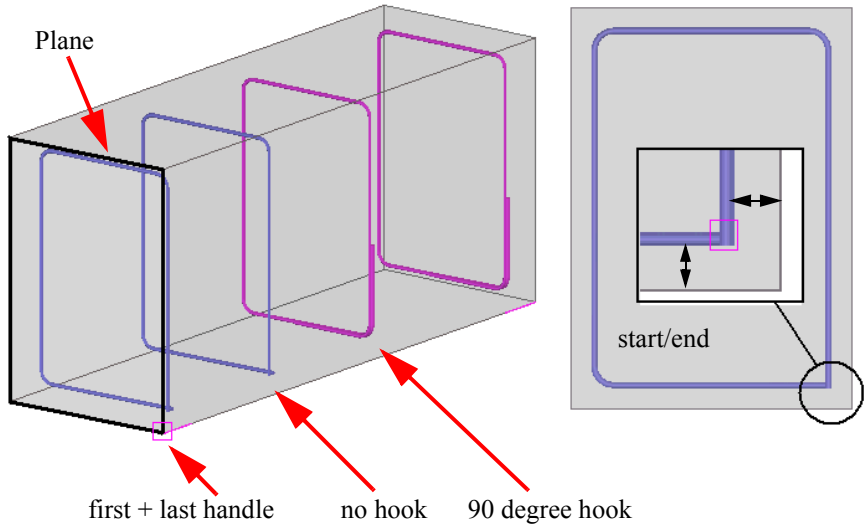


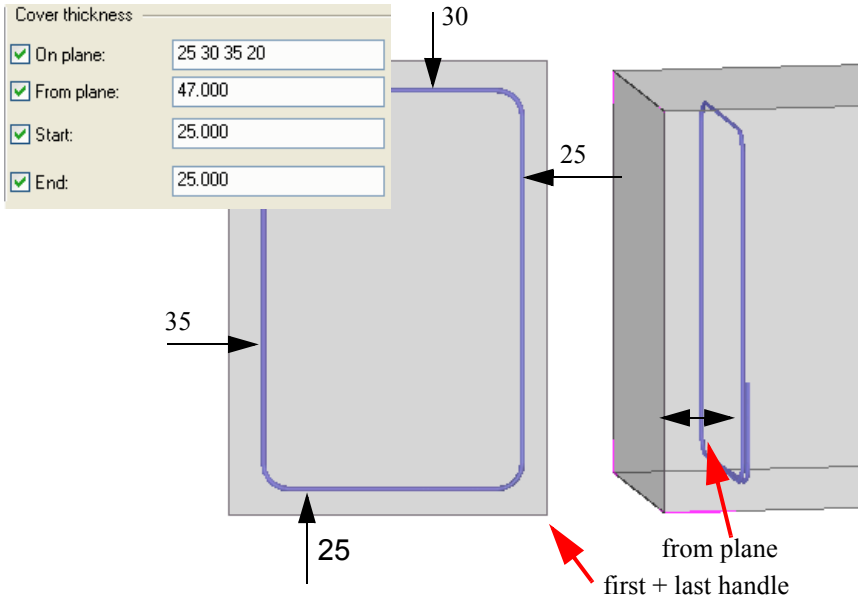
The shape of the reinforcing bar is displayed by clicking the contours of the concrete part. Handles are placed on these points. These handles are used to determine the cover - and by that the position of the bar. These handles form a plane in the global Z-direction and therefore determine what will be “On plane” and “From plane”. See the pictures below for more clearness:





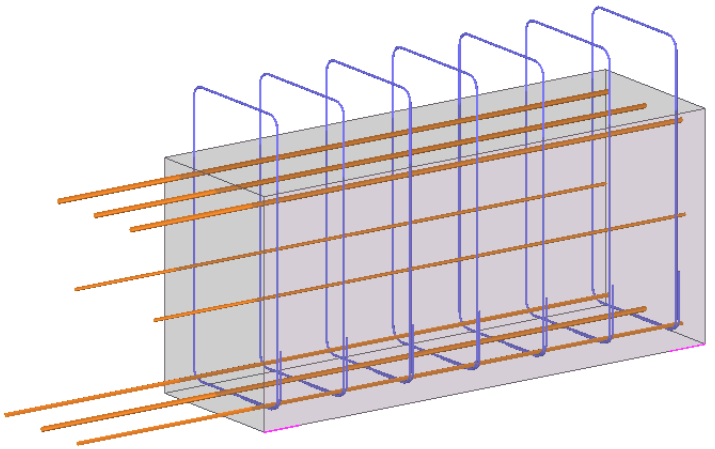
For stirrups, the shape of the bars (the positioning of the handles) will be represented in the concrete cross section. The plane, and by that also the features "On plane", "From plane" and "start / end" are different by that, see the next pictures:



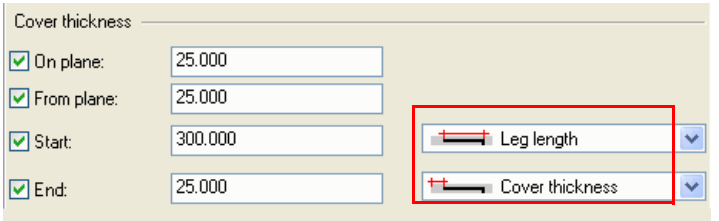


At cover thickness on plane several values can be entered. Make use of a space between the values. In the example above, for all sides a value is entered. If only 2 values would be entered, the first value would apply for the first and the second handle side, the second value would apply for the other sides.

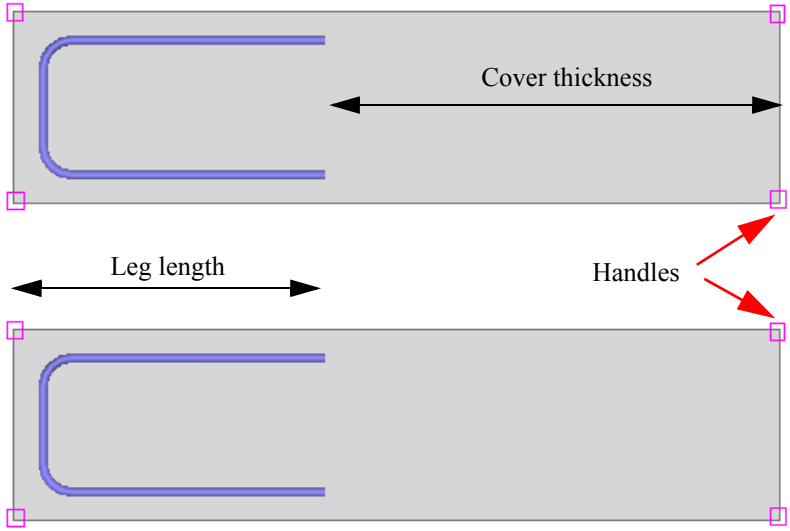
You can also enter negative values in the 4 fields for the **Cover thickness**. E.g. for modelling rebars or external stirrups.



Finally, there is the listbox to select the or **Cover thickness** or **Leg length.**



For bars that consist of one or more bending points, it may be useful to select a **Leg length**. The shape of the bar is then determined by the contour points of the concrete part (and by that also the direction), but the length of both ends of the bar is being calculated from the penultimate grip.



The input fields in the tab **Group** will be greyed-out when creating a single reinforcing bar. The settings in this tab only apply to a group of bars.

Close the dialog box after the correct settings are entered. The mouse pointer remains active, now select the concrete parts you want to add the reinforcement to. Now the handles can be selected. Finish the command by clicking in the middle mouse button.

The reinforcement bar properties can be modified by double-clicking on the rebar (or select the rebar, press right mouse and select Properties in the pop-up menu).

A reinforcement bar will highlight in the model if it is selected, including the handles. By moving these handles (or Drag and Drop) the shape of the bar can be changed.

The reinforcement will automatically be connected to the concrete part. The reinforcement will be displayed on that particular Cast unit drawing. It is possible to detach reinforcement from a cast unit part. Select the bar, click the right mouse and select **Detach from part**, now click on the concrete part.

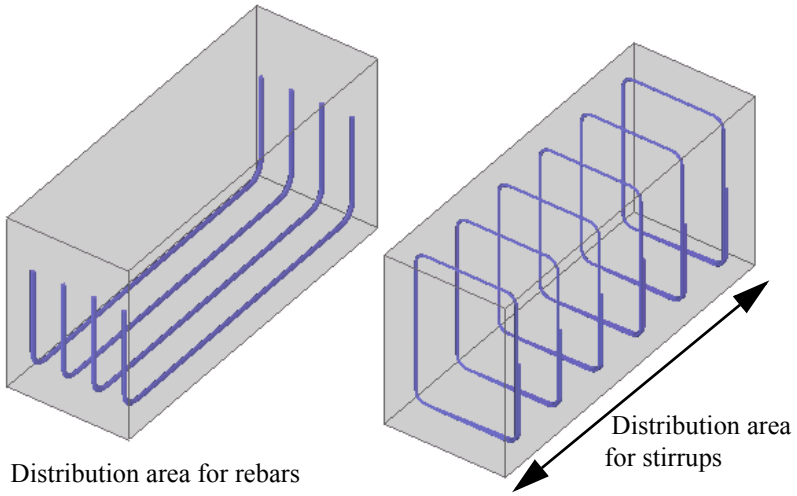
If a concrete part with attached reinforcement is moved or copied the reinforcement goes along. Therefore it is not necessary to select the reinforcement. If the shape (profile) of the concrete part will be changed, the handles will not follow the modification. In this case the handles need to be moved to the new outer contour.

Reinforcing bar group



For modeling a group of reinforcement bars, double-click the icon **Create Reinforcing bar group.**

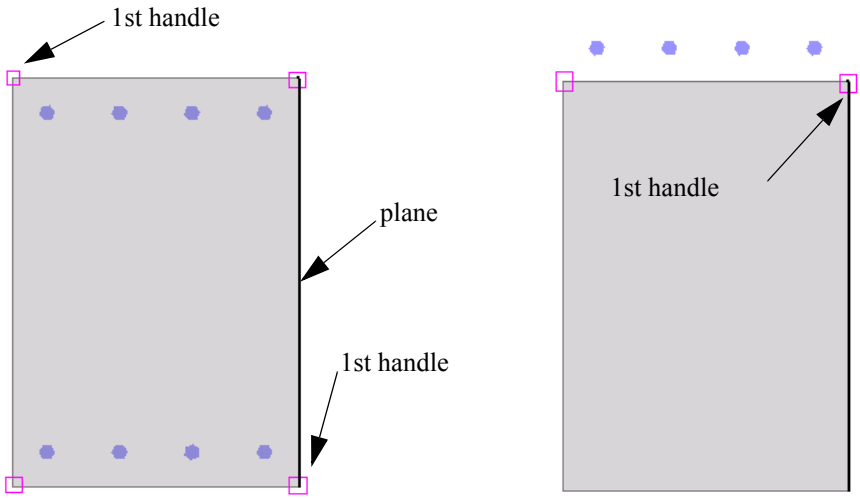
Besides the shape of the rebars (define the handles) you also need to define the distribution area for the rebars. After clicking the middle mouse button (to stop defining the shape of the rebars) this area can be selected by two points.



The distribution area points can be positioned on the outer contour of the concrete part again. Normally the distribution area will be perpendicular to the plane so the cover thickness on the sides can be defined with *Position from plane*.

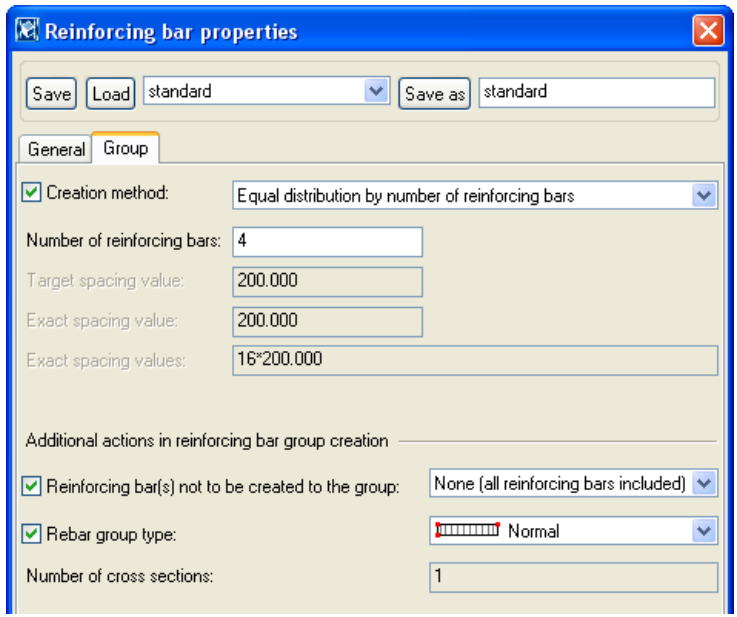
The select order of the handles which determine the cover area is important when placing the bars. The bars are placed on the right side of a line which is defined by the two picked handles.

See the picture below for the placing of the handles for bottom- and top reinforcement in a beam:



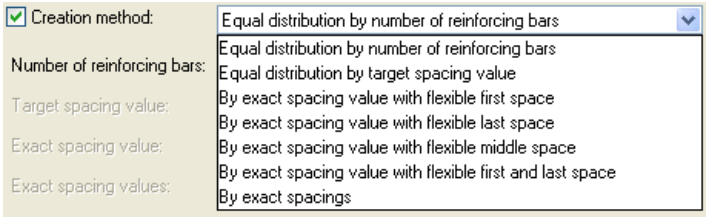
The picture on the right shows the rebars outside the concrete. This can be prevent by entering a negative value in *Position in plane*.

The tab **General** is equal to the tab for creating single bars. In the tab **Group**, several input fields can be set:



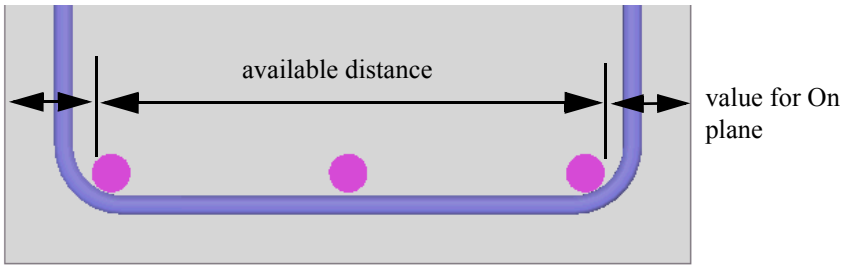
Distribution of bars

For *Creation method* you can define the way the bars will be distributed over the cover area.



Options

- ***Equal distribution by number of reinforcing bars***. This option will be used particularly for main reinforcement bars in beams. The defined number of bars is divided by the available distance (= cover area minus 2 x the value for 'On plane')



  This is the only option where the number of bars can be set. For all other options the number of bars is calculated by Tekla Structures. These ways are therefore most suited for determining the placing and distances of stirrups and wall- and floor reinforcement.

- ***Equal distribution by target spacing value***. Inhere, you can enter a spacing value. Next, Tekla Structures calculates the number of bars/ stirrups which will fit in the cover area and divides the (rounded-off) value by the available distance. The bar distances will rarely be a good, round value.

Example

Set '**Target spacing value**' to 150. The available distance for the bars is 1150 mm. This makes 1150:150=7,6 bars, rounded off that will be 8 bars. 1150:8=143,75. So 8 bars will be placed with distance 143,75.

- ***By exact spacing value with flexible first/ last/ middle space***. Often you probably want a fixed distance, resulting in a remainder distance. You can choose whether to add the remainder distance to the first, last or the middle distance.

- ***By exact spacing value with flexible first and last space***. Tekla Structures calculates the number of bars and places them with the defined distance. The remainder distance will be divided equally by the first and last bar distance.
- ***By exact spacings***. Exact distance values needs to be set. Always "number*distance" divided by space characters.



Equally distributed



exact spacing with flexible first



exact spacing with flexible first/last

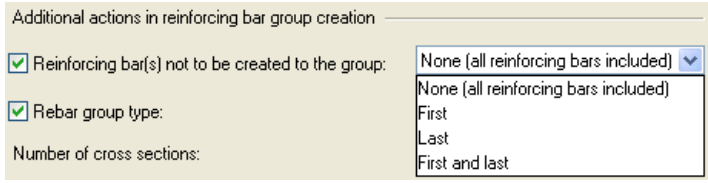


exact spacing, e.g.:

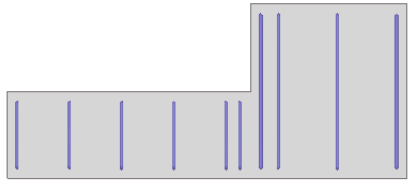
4*50 4*150 3*100

Reinforcing bar(s) not to be created to the group

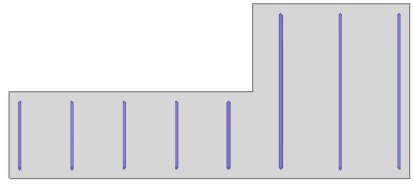
Possibility to exclude specific bars. The options are None, First, Last and First and last.



In the following example, two rebar groups are added to a concrete beam:



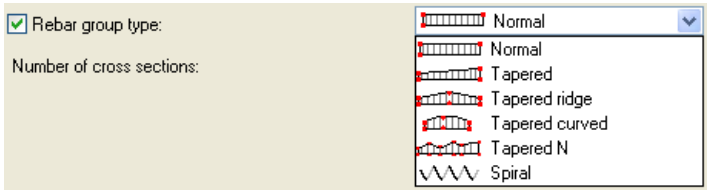
2 rebar groups with flexible end- and/or start value



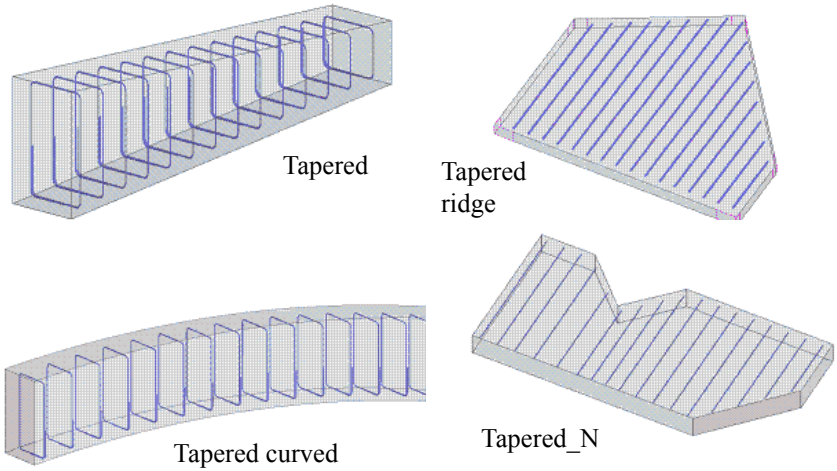
respectively the last and first bar removed

Rebar group type

For rectangular concrete parts it is sufficient to pick 2 points for defining the distribution area of the rebar group (= normal) If the shape is not rectangular, an alternative shape can be selected:



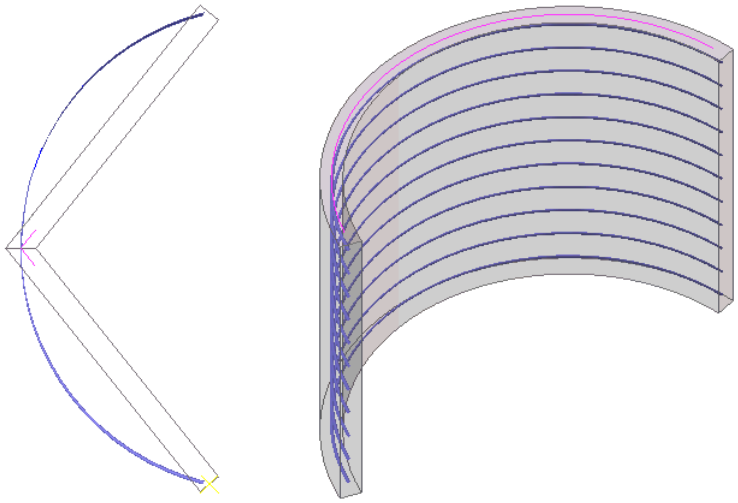
- ***Tapered***. To be applied in a tapered (height or width) beam. First click the object, define the grips in the first cross section, click middle mouse button, define the grips in the second cross section, again click middle mouse button. You don't have to pick the distribution area as this area is between the 2 defined cross sections.
- ***Tapered Ridge***. The tapered (height or width) cross section defined by three points. Create grips on the spot of the cross section and close the cross section with the middle mouse button.
- ***Tapered Curved***. For tracing a curve using three cross sections. Can be applied to create stirrups in a curved beam.
- ***Tapered N***. For creating free form reinforcements using multiple different cross sections. Close each cross section by clicking the middle mouse button. The command will remain active until the number of cross section is created set in the input field Number of cross sections:



Curved reinforcing bar group



For creating a curved reinforcing bar group, the same dialog settings are used as for straight bar groups. The only difference is that 3 points have to be picked for defining the rebar shape (start point, middle point, end point). Subsequently, do not click the middle mouse button, but continue with picking the distribution area:



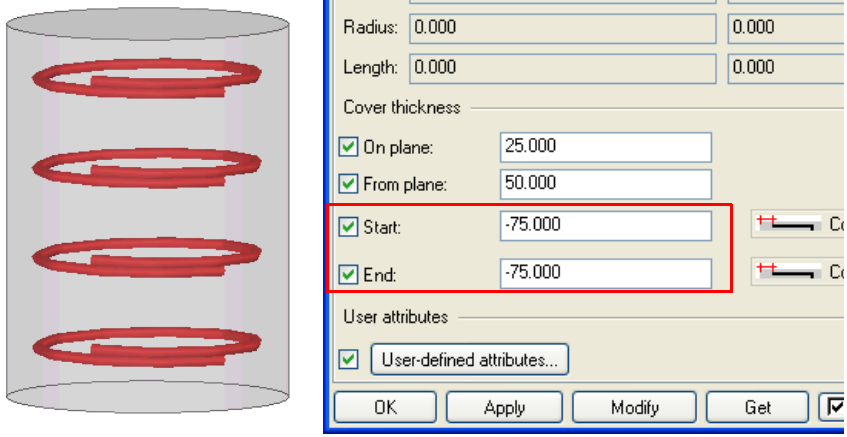
If the concrete part is created using multiple points and with the intention to create a curved part by changing the chamfer properties, then it is useful to create grip points for creating the curved rebars. Always place the curved rebars using the working points of the concrete part and set option “On Plane” to the correct position.

Circular reinforcing bar group



Use the command **Create circular reinforcing bar group** to create stirrups in round columns. Three points are needed for creating the rebar group, so pick three points on the outer contour of the concrete part. The radius is automatically calculated from these three points. (for accurate calculation, remind to switch on the first 2 **snap settings** icons). Subsequently, pick the distribution area (clicking middle mouse button is not necessary).

For the splice length of the round stirrups, do not set a length value at *Hooks*, but set a negative value for the “Cover”-options *Start* and/or *End.*

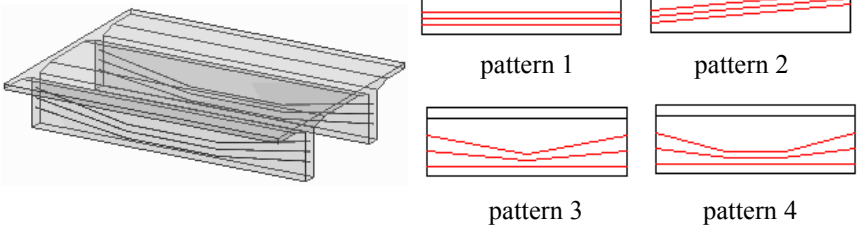


Reinforcement strand pattern

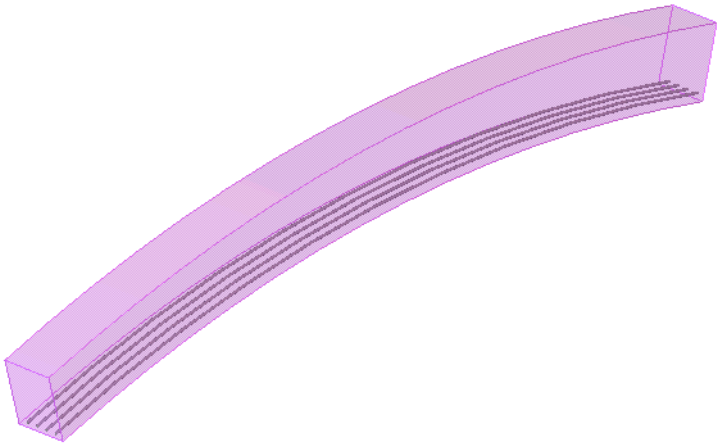


With command **Create reinforcement strand pattern** you can create pre-stressed reinforcement. Particularly for pre-stressed strands you can position the strands on any random position.

Besides the familiar rebar properties, you can also set the pre-stress load (in kN) and the desired pattern:



In some cases it can be handy to create "normal" rebars by using this command, especially for curved beams. Pick the position of the bars (no cover, you cannot define this while using this command), close by clicking the middle mouse button and select the start- and the endpoint for the bars.



Reinforcing mesh

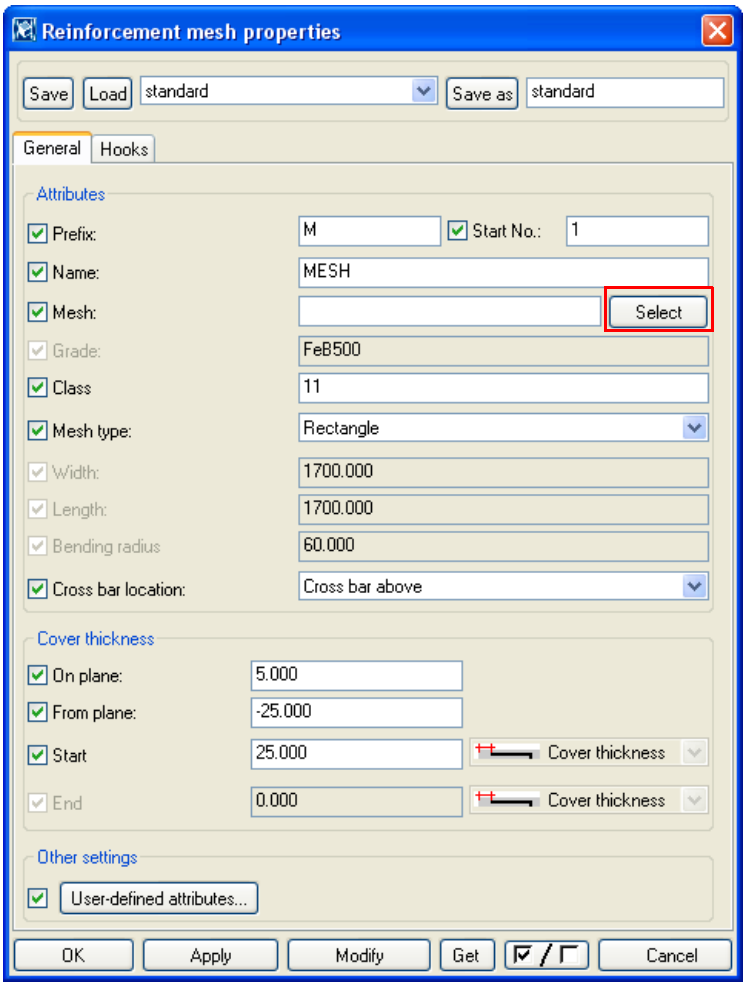


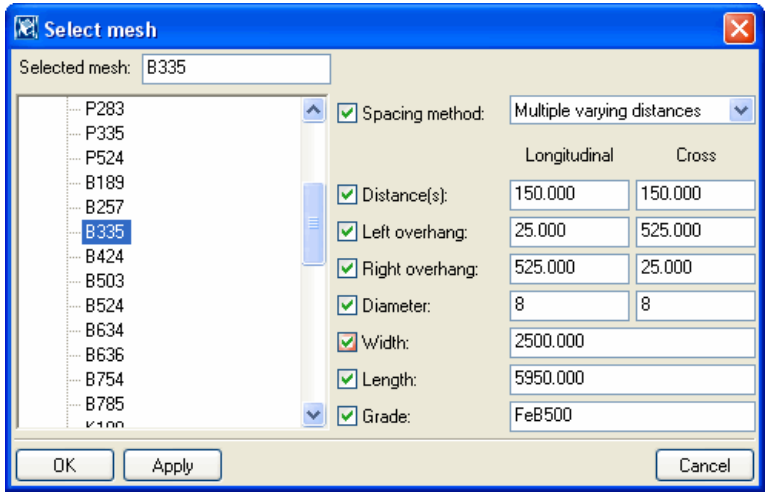
In Tekla Structures you can create **reinforcing meshes** which consist of two perpendicular bar groups. The length, width, diameters, mesh size and cover area can be set in the properties of the reinforcement mesh. In the model you have to select the concrete part, followed by an insertion point and the longitudinal direction of the mesh.

With the **Select** button, a link can be created to the reinforcement mesh database, the file *mesh_database.inp* in the folder:

disk:\TeklaStructures\version\environments\europe\profil

In this file you can you can add standard/stock meshes.





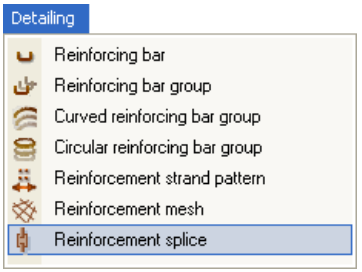
Attaching rebars to another part

When modeling rebars, a part has to be selected to attach the rebars to. It may occur that modeled rebars needs to be attached to another part at a later stage. To do this, select the rebar, click right mouse and select **Attach to part** in the pop-up menu**.** Next, select the new part the rebars should attach to.

Reinforcement splice



Use this command to create a splice between 2 rebar groups. Go to **Detailing > Create Reinforcement splice:**

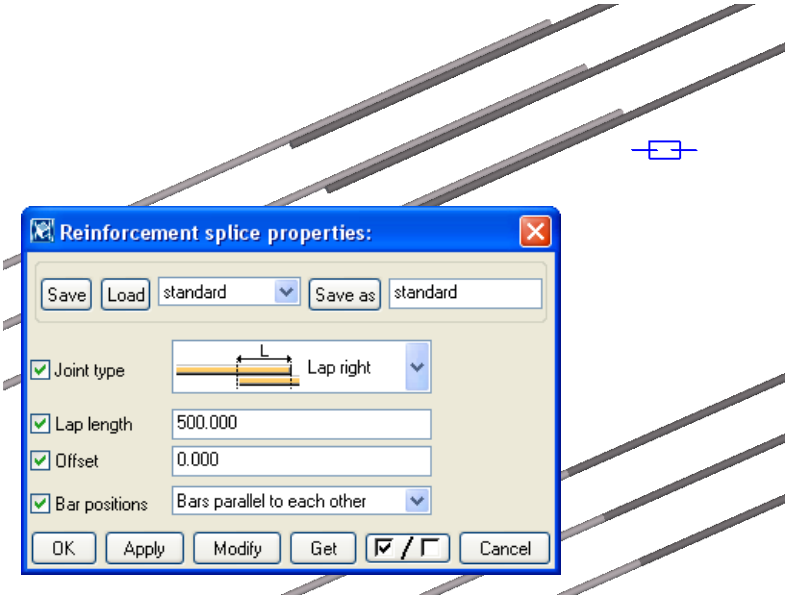


If you first want to edit the properties, then hold down the <**Shift**> key while you click the **Create Reinforcement splice** command in the pull-down menu.

Next, click 2 rebar groups, a splice is added between the 2 groups. The splice remains active and can be edited.



Double-click the blue rebar splice symbol in the model view.



Exercise

Now do exercise W1 and W2

4.2 Modeling reinforcement tips

Reinforcing identical cast units

Adding reinforcements to cast units is not always necessary. Example; a cast unit occurs 20 times in a model. Of course it is allowed to add reinforcements to all 20 cast units, but in some cases it is possible to add reinforcement to just 1 cast unit.

Pro's

- The model contains less reinforcement, this means a better organized model
- The model contains less reinforcement, this means a smaller database
- Less chance on numbering differences because of (by accident) different modeled reinforcement
- Numbering the model is faster
- Copying reinforcement is not needed and doesn't need to be put in a custom component
- Applying model changes is quicker (modifying reinforcement just once, instead of modifying a number of times)

Contra's

- Incorrect reinforcement weight
- Elements with identical shape but different reinforcement receive identical cast unit numbers (a difference in numbering can be achieved by added **Comment 2** in the User Defined Attributes)

If upper mentioned contra's are insuperable, then check the option Reinforcing bars in the Numbering settings (switched off by default)

The drawing must be created from the reinforced cast unit, and manually if possible, do not use the Wizard. If the drawing already has been created, and the reinforcements will be added in a later stage, then you can find the cast unit's ID-number (**Inquire** > **Object**) and add the reinforcing bars in that particular cast unit.

Modeling reinforcement using circular parts

Modeling circular section parts, e.g. cavity wall anchors, requires more computer capacity then modeling rectangular section profiles.

Circular section profiles are divided in segments (the number of segments is depending on the accompanying settings).

Bended circular section profiles requires even more capacity from your computer. Especially modeling (bended) cavity wall anchors, which are often used in large numbers per cast unit, may result in a less performing computer.

Larger projects can contain 10.000 or more cavity wall anchors easily, and for each select-, copy-, delete- and regenerate-command, these cavity wall anchors have to be re-build again every time.

To prevent this, we suggest to model these cavity wall anchors as reinforcement bars.

Rebar-database sections are less demanding and you can create multiple cavity wall anchors in a group, and the group of wall anchors are considered as one object.

This will result in a better performing computer, especially for larger Tekla Structures models.

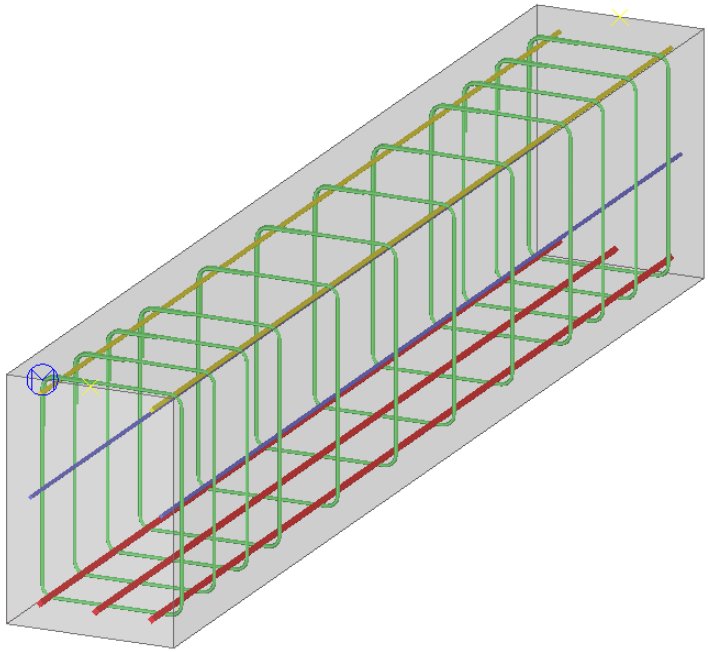
If you are using this method, you can also use the drawing template "cavity wall anchors", add this template to the layout for cast unit drawings.

4.3 Reinforcement components

Tekla Structures contains various reinforcing components. With these components you can create reinforcement rebars and -groups. After defining the properties (double-clicking the icon), you have to select the part to be reinforced. The reinforcement will be linked to the part automatically. Below some examples are described.

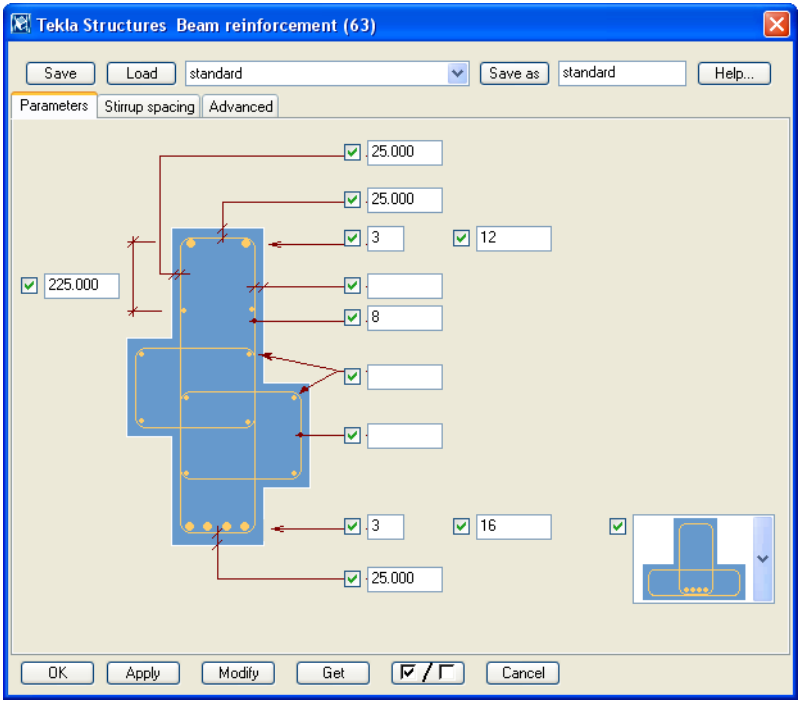
Beam reinforcement (63)

This component creates the stirrups, the top-, bottom- and side reinforcement in one go. If the size of the beam changes, the reinforcement will be adapted to the new dimensions:

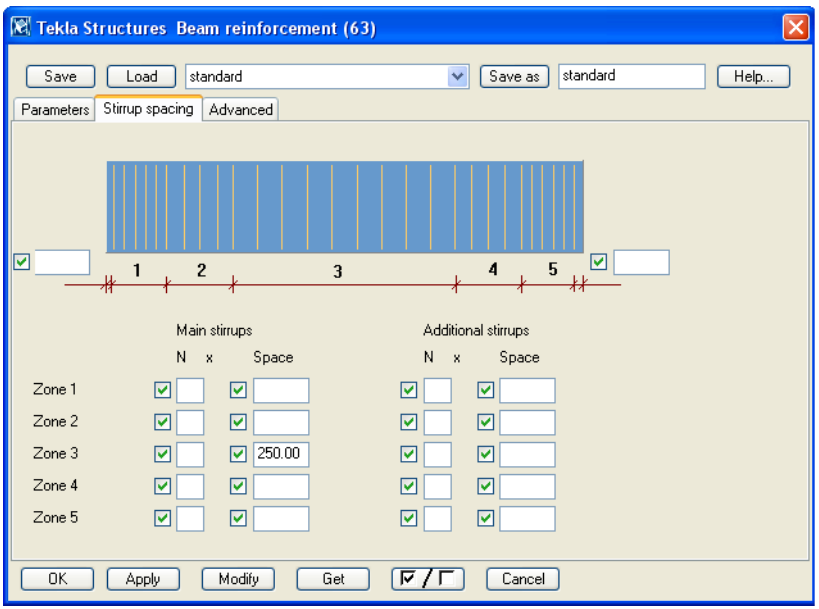


For the main reinforcement (top/bottom), the number of bars and diameter can be set individually. For T-profiles or beams with a ridge, multiple stirrup groups can be generated.

The cover thickness from the stirrups needs to be defined in the component; the positioning of the bars in longitudinal direction -independent of stirrup diameter- will be automatically determined by the component.

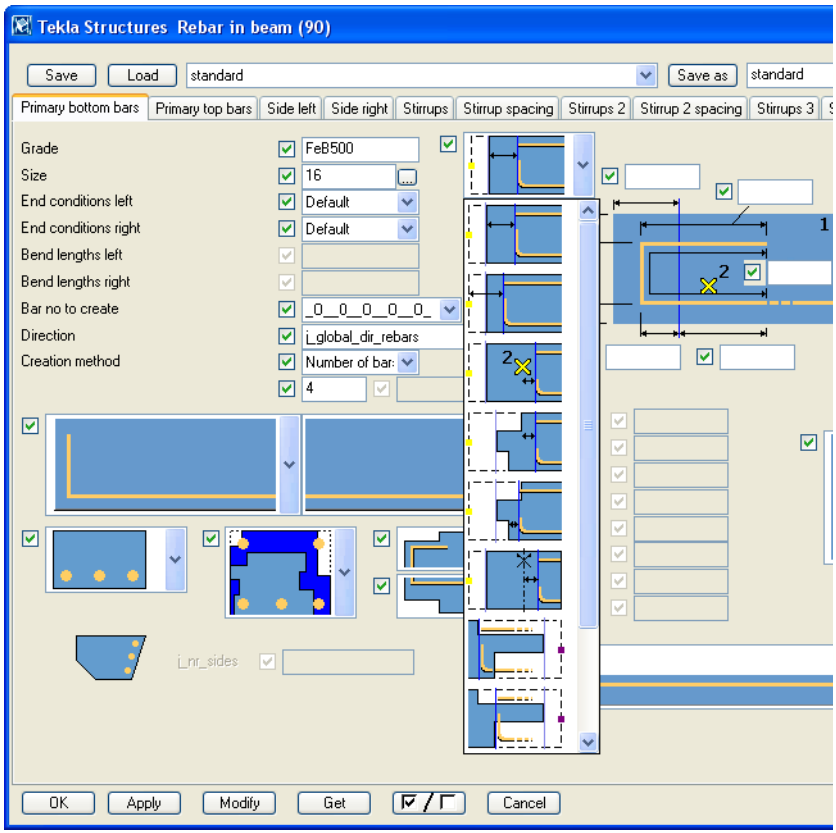


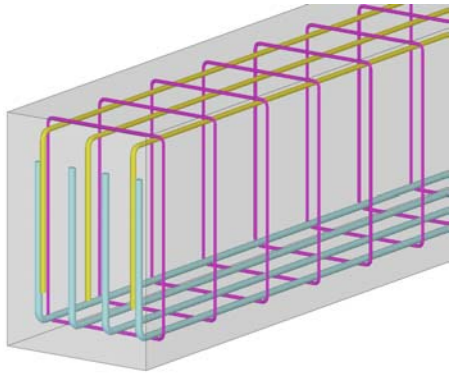
The stirrup distances can be extensively specified in tab **Stirrup spacing**. Consequently you can apply more stirrups on places where bigger shear forces occur. You can set the number of stirrups and stirrup distances for different zones.



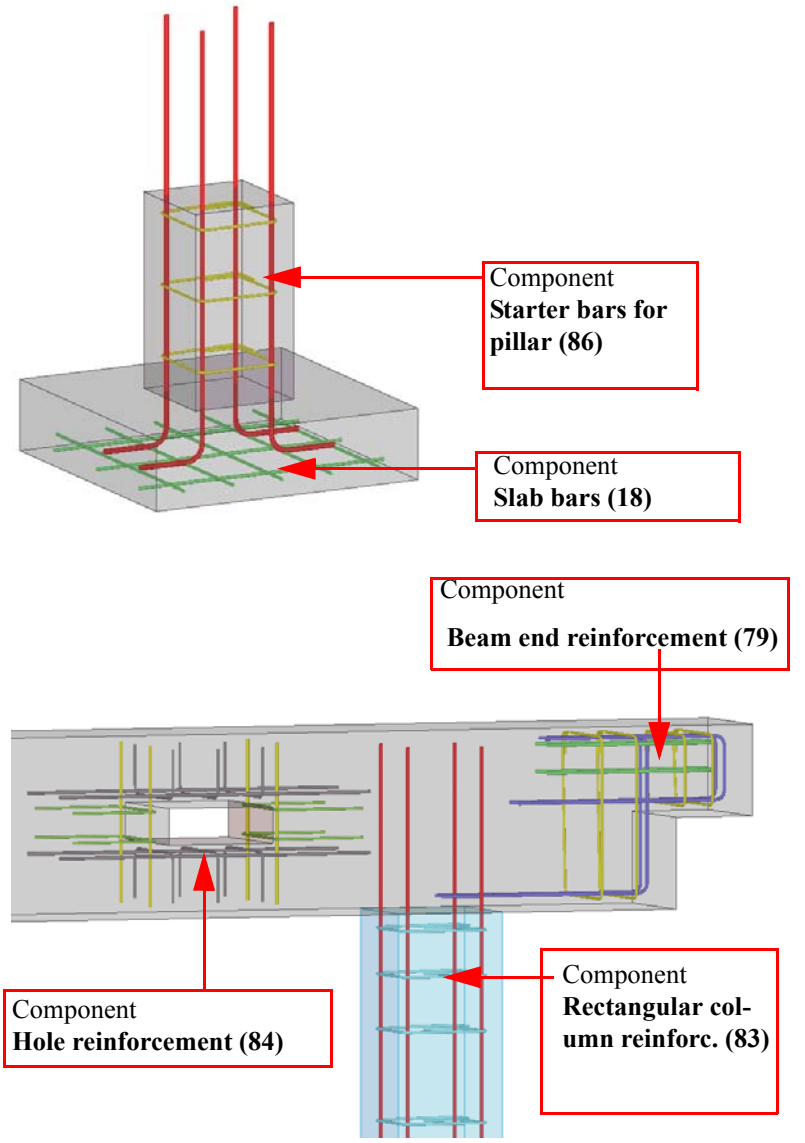
Rebar in beam (90)

This is a extensive component to create reinforcement in a beam. Besides all options which are available in component **Beam reinforcement (63)**, this component can also bend the bars at the start/end of the beam.





Examples Reinforcement components

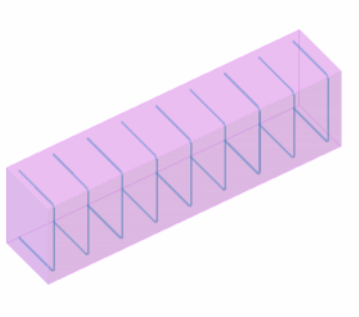
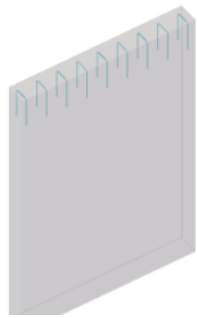
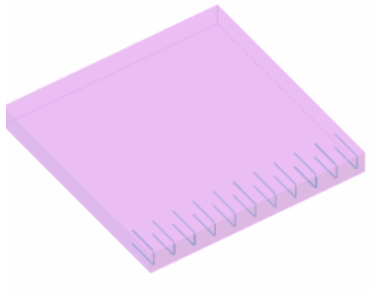
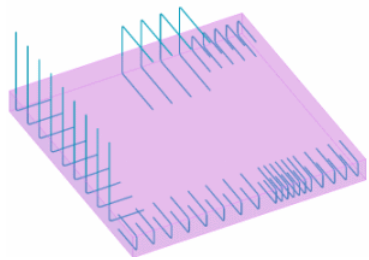


Exercise Insert reinforcements to all walls and beams in exercise B.

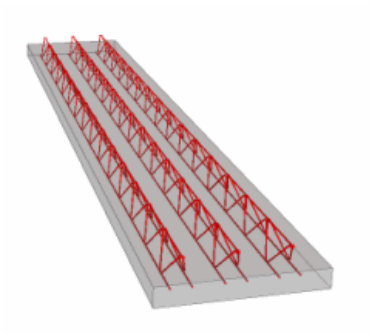
4.4 Component examples reinforcement

Below you will find a lot of very often used components for reinforcement:

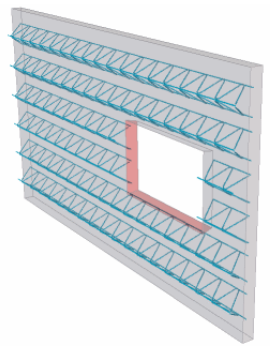
Border rebar (93)

 default beam	 default panel
 default floor	 border rebars

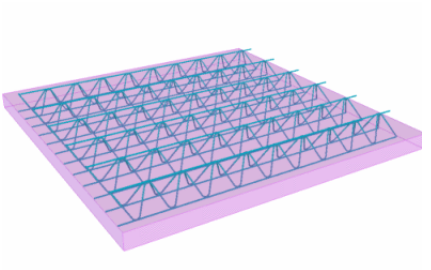
Braced girders (88 and 89)



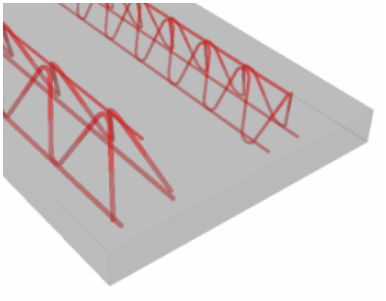
floor



panel

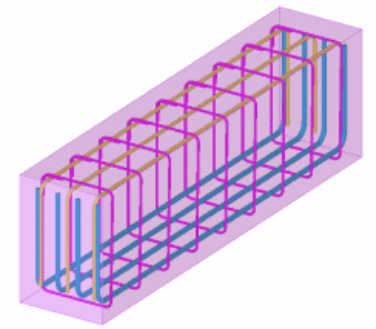
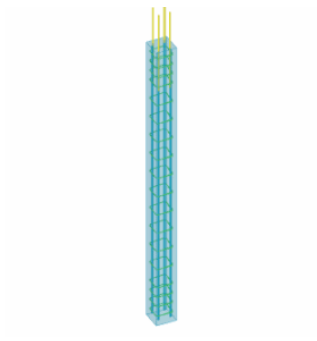
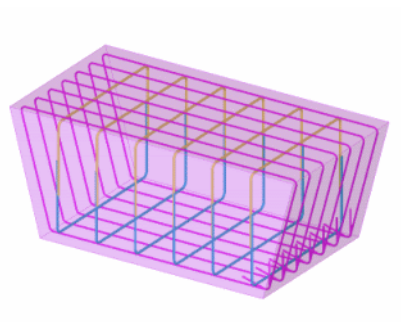
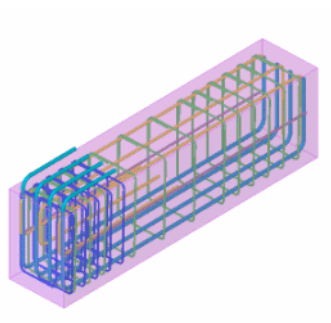
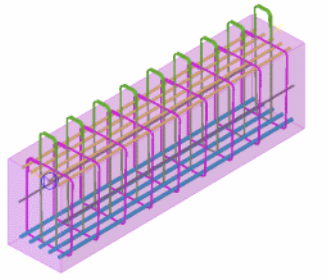
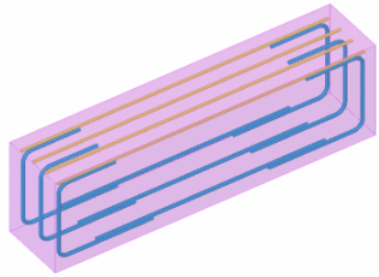


plate

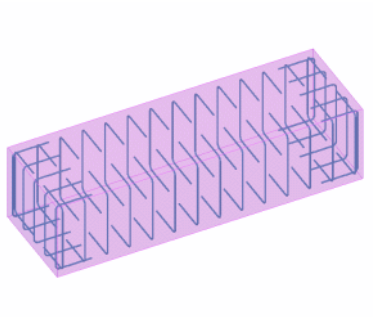
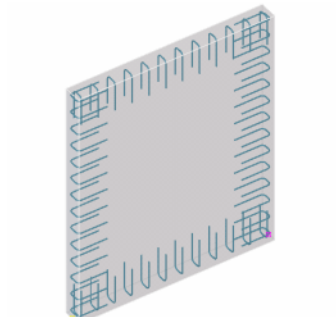
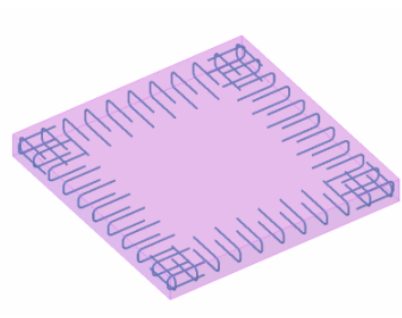
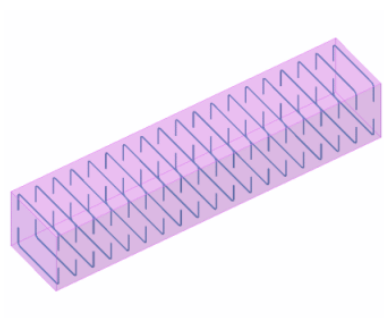


geometry

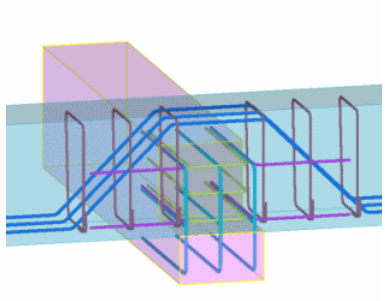
Rebar in beam (90)

	
beam	column + anchor bars
	
sloped angles	ledge beam
	
stirrup extension	reinforcement along the main axis

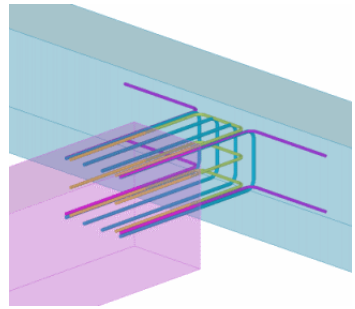
Border rebar (92)

	
default beam	**default panel**
	
default plate	**pins on top**

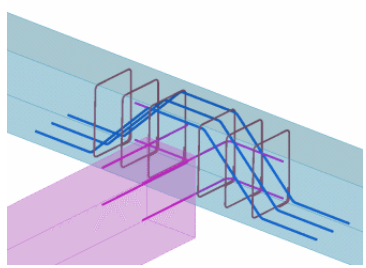
Rebar conn 2 beams (94)



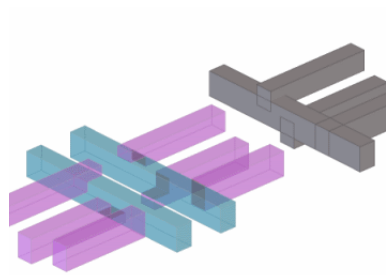
default



no overlap (bars)

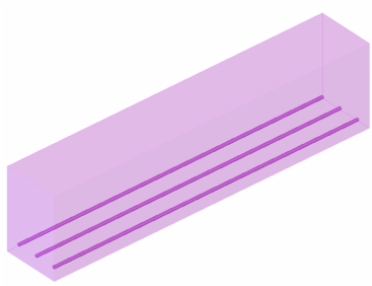


hang-up bars, pins and stirrups

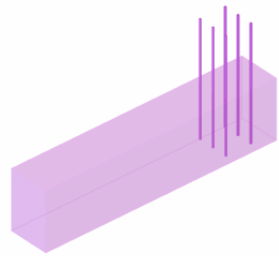


connections, no reinforcement

Longitudinal reinforcement (70)

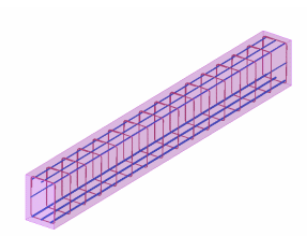


default longitudinal reinforcement

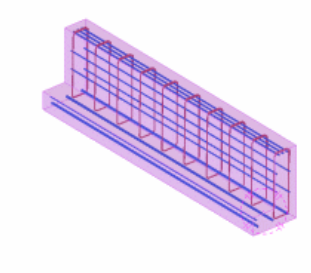


pins

Beam reinforcement (63)

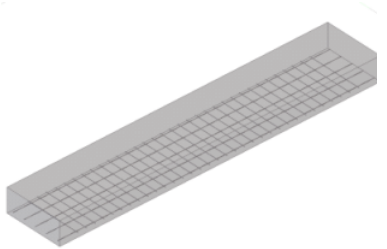


default reinforcement in beam

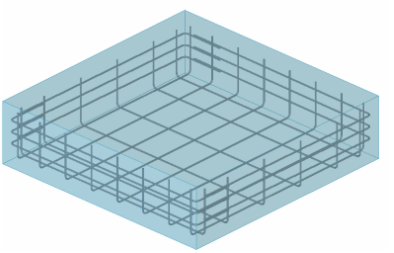


RCL profile

Pad footing reinforcement (77)

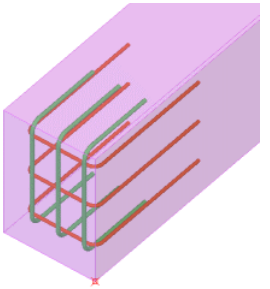


pad footing reinforcement below

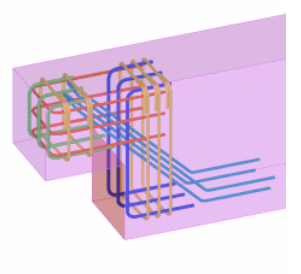


default footing plate + lacer bars

Beam end reinforcement (79)

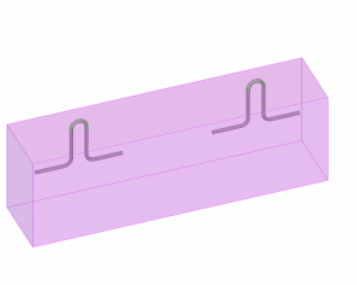


beam end

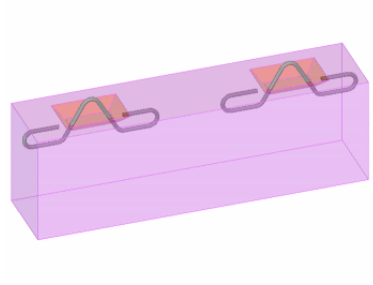


teeth

Lifting anchor (80)

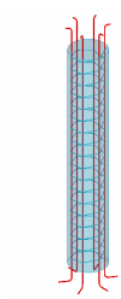


default lifting anchor

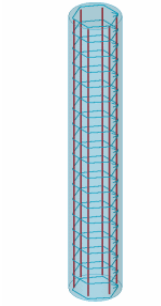


lifting anchor + trenching

Round column reinforcement (82)

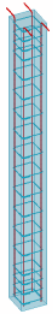


round column reinforcement + pins

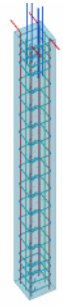


round column reinforcement

Rectangular column reinforcement (83)

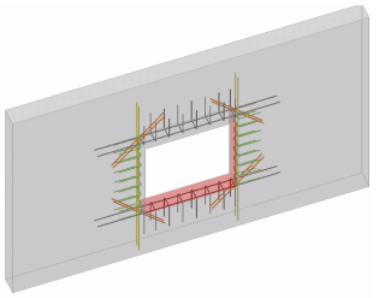


rectangular column reinforcement

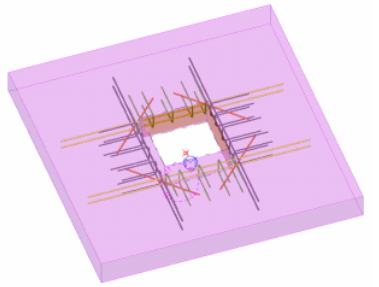


columns + pins

Hole reinforcement for slabs and walls (84)

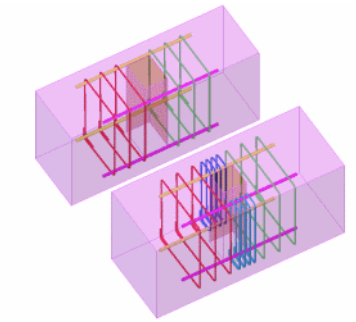


panel + reinforcement

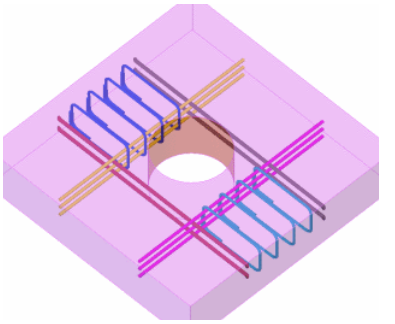


floor + reinforcement

Hole creation and reinforcement (85)

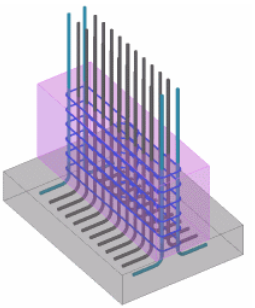


hole including reinforcement in beam

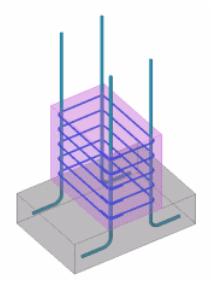


hole + reinforcement

Starter bars for pillar (86)

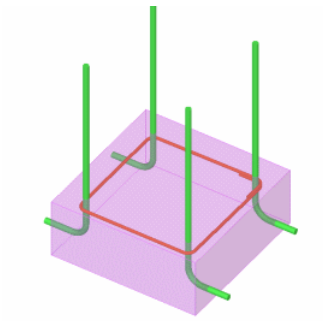


footing plate

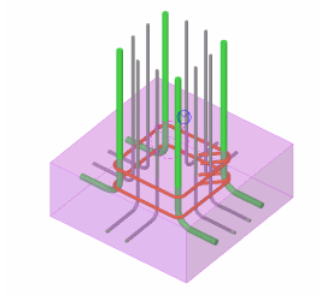


pins and border rebars

Starter bars for footing (87)

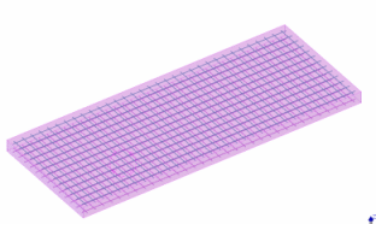


pins and starter bars

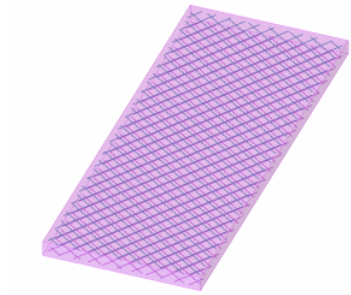


default footing plate

Slab bars (18)

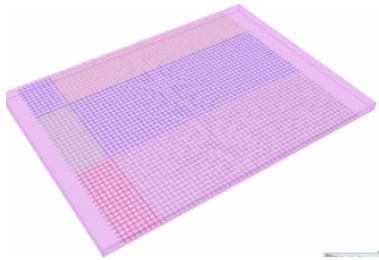


default bottom- and top bars

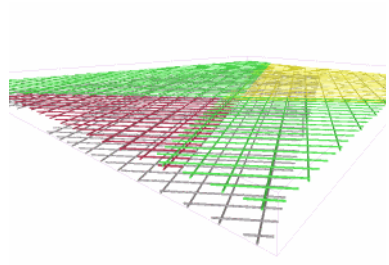


in a direction

Reinforcement mesh array in area (89)



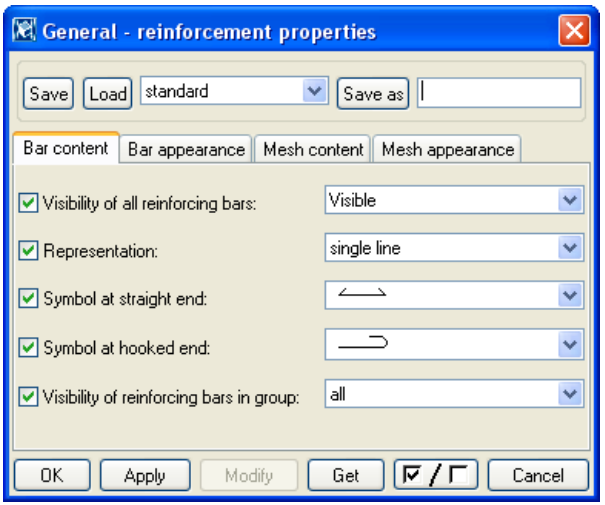
default meshes



reinforcement direction

4.5 Reinforcement representation and marks in drawings

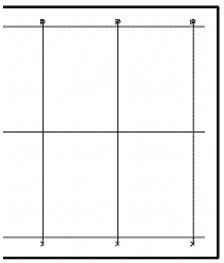
The modeled reinforcement can also be displayed on drawings. Open the drawing properties and click on the button **Reinforcement**. Now you can define the visibility, the representation and the lay out for the bars and meshes:



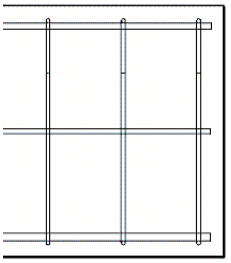
Visibility of all reinforcing bars

You can select the options **Visible** and **Not visible**

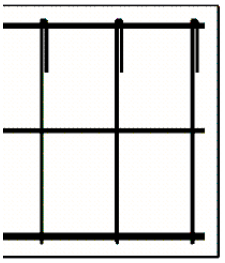
Representation



single line + stick

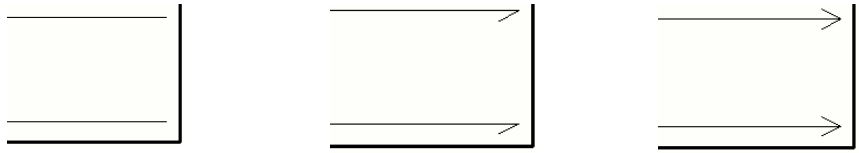


double lines



filled line

Symbol at straight end

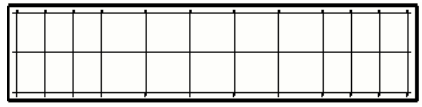


To represent the symbols set **Representation** to **single line**.

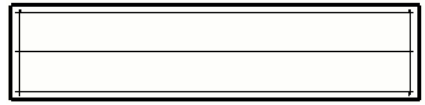
Symbol at hooked end



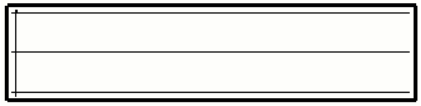
Visibility of reinforcing bars in group



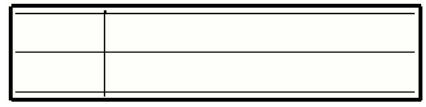
all



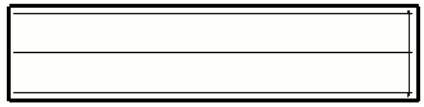
first and last bar



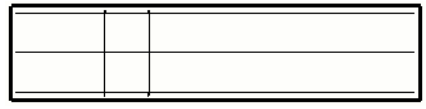
first bar



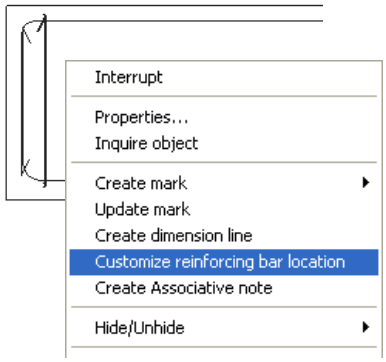
bar in the middle of group

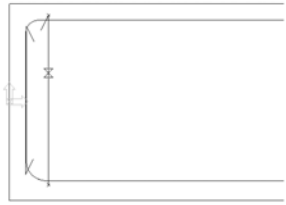


last bar



2 bars in the middle of group





Customized:

Select the bar, click right mouse, select **Customize reinforcing bar location**. Now pick the position.

Reinforcement marks

Defining the reinforcement mark content is equal to part- and/or bolt marks.

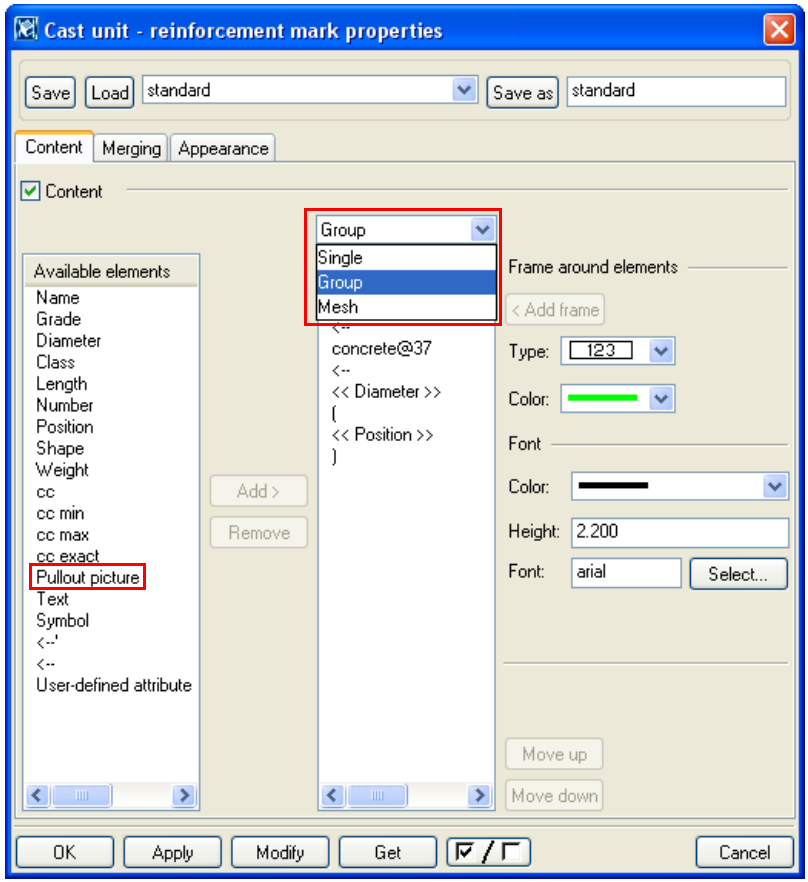
See also

For more information, see **Tekla Structures.pdf** in the folder:

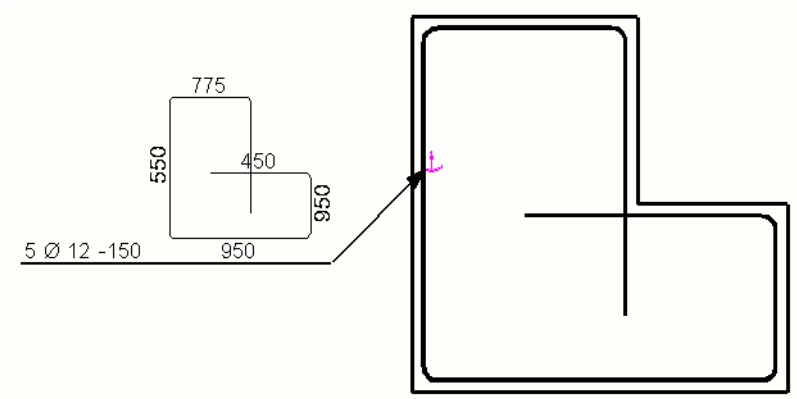
disk:\TeklaStructures\version\environments\europe\pdf.

Next, options only applying for reinforcement marks are considered.

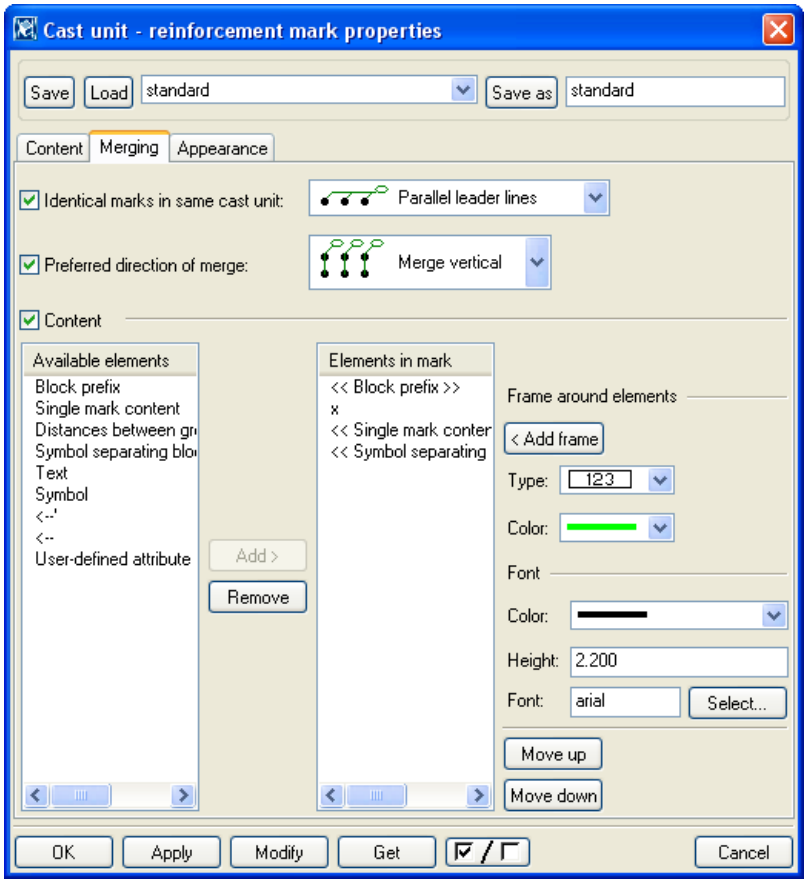
You can define separate reinforcement marks, depending on the way the reinforcing bars are modeled. You can create settings for single (loose) bars, for reinforcing bar groups and for reinforcing meshes.



You can use the element **Pullout picture** to display the shape of the reinforcing bars in the marks and you have the possibility to include the dimensions.

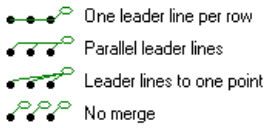


On the tab ***Merging*** you can set the following items:



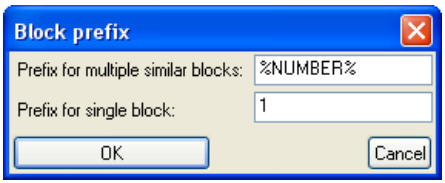
Identical marks in same cast unit

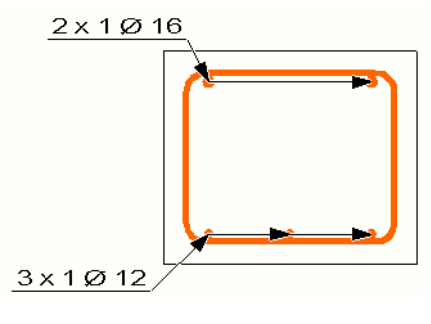
You can merge identical marks from reinforcing bars in the same cast unit. You have the following options:



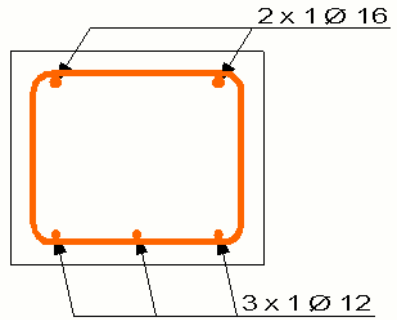
You can set the block prefix in the mark content when you merge reinforcement marks. Blocks combine similar single marks.

By default, the block prefix is set to first position the number of bars followed by the element <<single mark content>>.

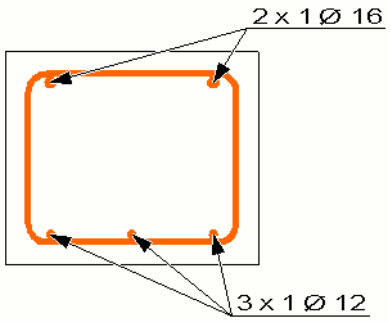




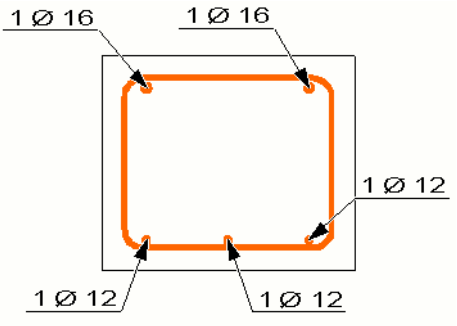
One leader line for a row



Parallel leader lines



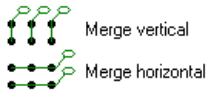
Leader lines to one point



Individual leader line for each mark

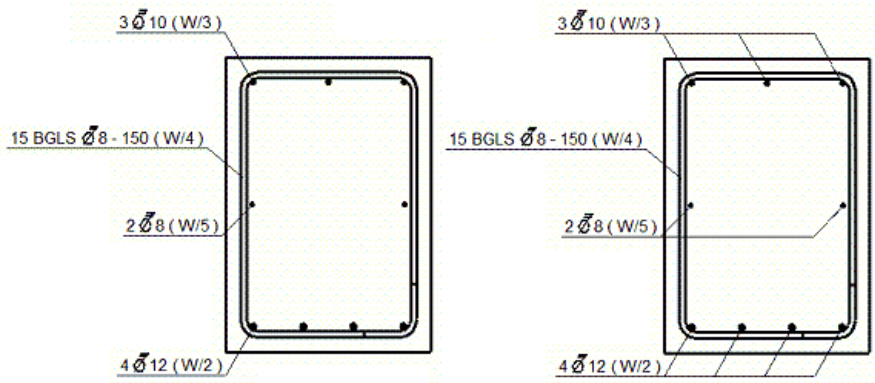
Preferred direction of merge

If there are several possible merge directions, you can select a direction:

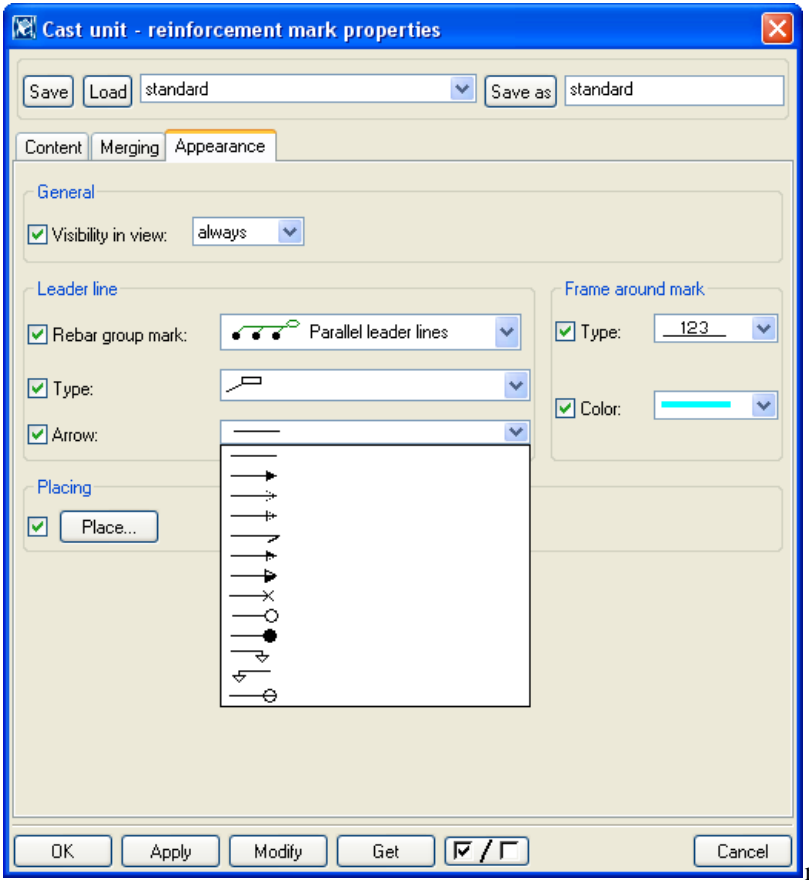


The placing and the minimum distance/search margin of the leader lines will also be determined by the checkboxes and the entered values in the button **Place** on the tab **Appearance**.

Setting for marks on the tab **Merging** refer to a reinforcing bar group (in the model defined as a group). A group can include several bars. You can set to display a leader line per reinforcing bar mark on the tab **Appearance** at *Rebar group mark*. You have the same options as you have for reinforcing bar groups, but now per bar.



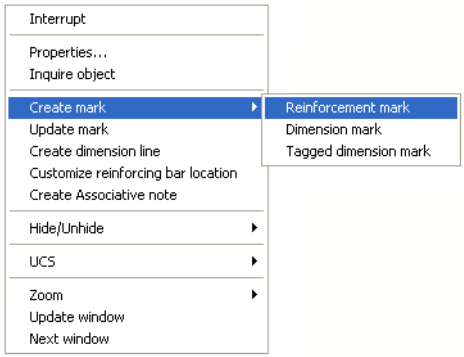
Tekla Structures includes several arrow types for the leader lines:



m

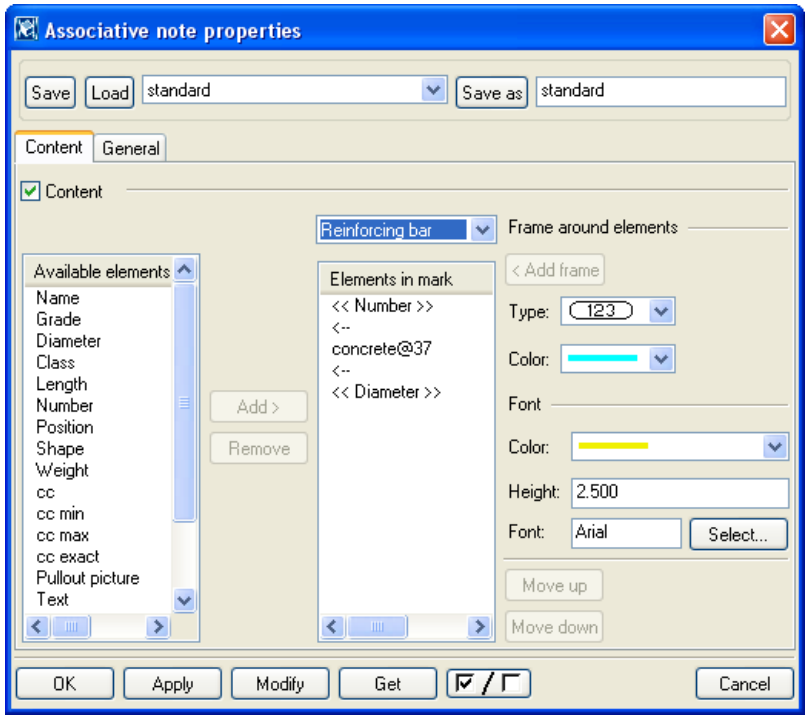
Creating reinforcement marks manually

You can add marks to reinforcing bars and/or groups manually just like you can for parts. Select the reinforcing bar or group(s), click right mouse, now select **Create mark** > **Reinforcement mark**



Associative reinforcement mark

You can also add associative marks to reinforcing bars and/or groups manually just like you can for parts. Creating associative marks is equal to part marks, Tekla Structures identifies the reinforcing bars or group and adds the mark(s).



Dimension marks

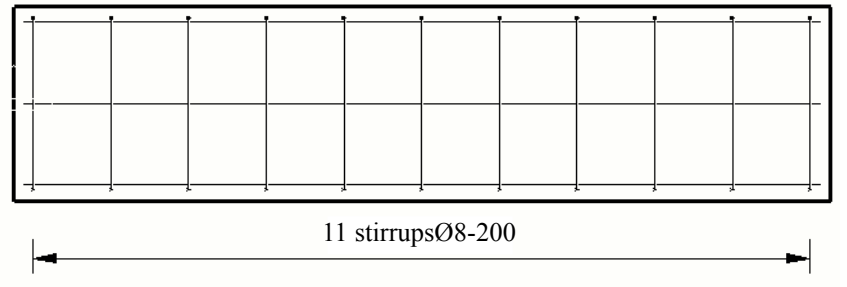
You can add a dimension mark to a reinforcing bar group. Select the group on the drawing, click right mouse, now select **Create mark > Dimension mark**. A dimension line will be created. Double-click the dimension line and load the setting *stirrup-group*. The contents of the dimension is defined in the line GroupBarMark in the file *rebar_config.inp* in the folder:

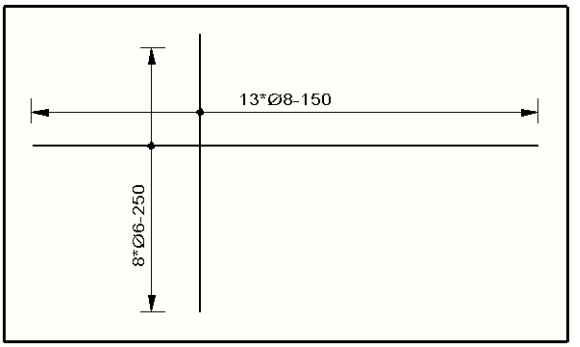
disc:\TeklaStructures\version\environments\europe\system

The line is set by default as follows:

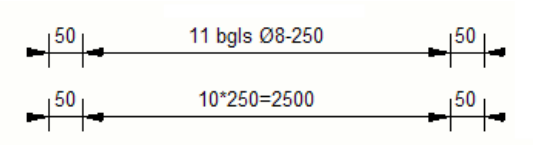
GroupBarMark="%NUMBER%%REBAR_NAME%%Ø%%SIZE%-%CC%"

For bars, where no name is entered (straight bars), will display 13Ø8-150. If there has been entered a name, i.e. stirrups, the dimension mark could be 11stirrupsØ8-200.





The option **Dimension mark** only displays the centre-to-centre distances from a reinforcing bar group, so the name and the numbers are not included.

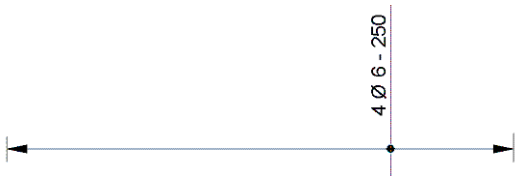


Dimension mark

Tagged dimension mark

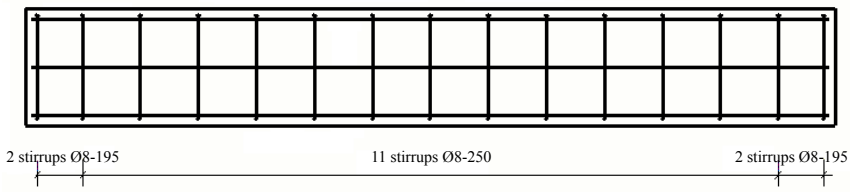
Create dimension line

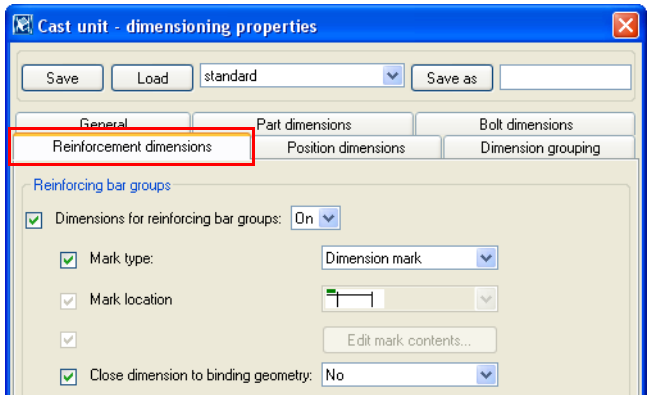
The command **Create dimension line** (select the bar > right mouse) creates a dimension line to determine the position of the bar. Possibly you can switch off the dimension text (setting *stirrups* in the dimension properties) to display the distribution area. The dimension intersection point is created automatically.



Setting dimension marks in the drawing properties

You can set the dimension marks as a setting in the Cast unit drawing properties, click the button **Dimensioning** and go to the tab *Reinforcement dimensions*. You can use this option to automatically set the distribution area for stirrups and hair-pins, for example. The contents of the mark is defined - just like manual added dimension marks - in the file `rebar_config.inp` in the folder system.



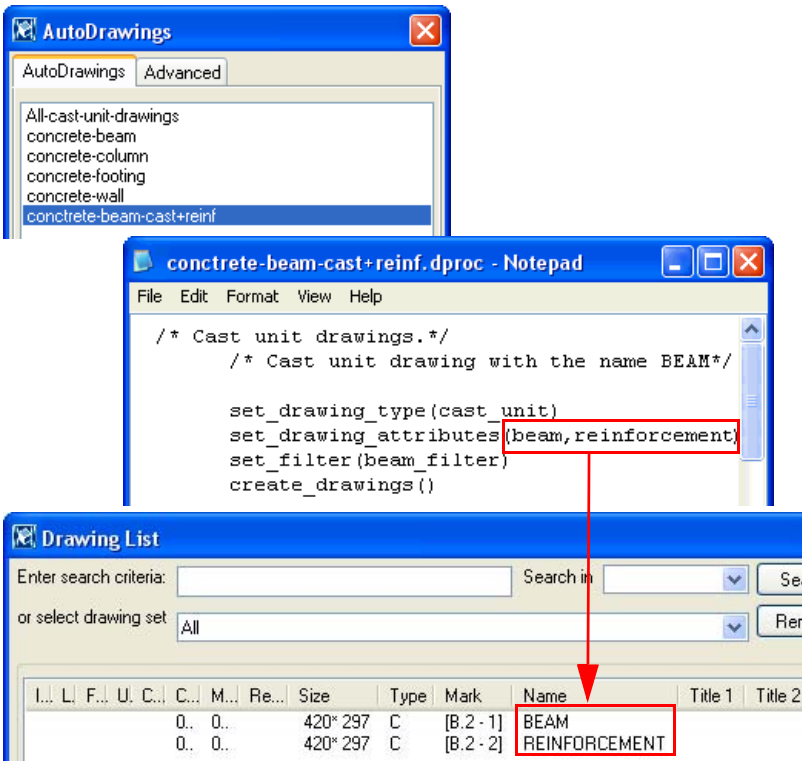


If you want to set the dimension mark manually, select the option **Tagged dimension mark** for the *Mark type*.

Creating multiple cast-unit drawings of the same part

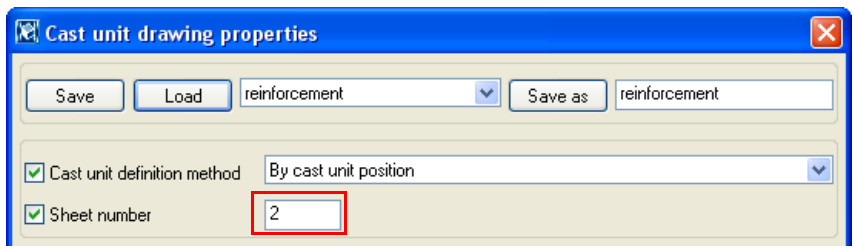
You can now create multiple cast-unit drawings of the same part by defining multiple drawing attributes in the drawing Wizard dialog box, for example for a beam to create a drawing of the beam and the reinforcement.

In the Wizard dialog box, give the names of the cast-unit drawing properties you want to use to create the multiple drawings. Give the names in parentheses in the set_drawing_attributes row as shown below:





Make sure that you have saved the cast-unit drawing properties by the names that you use as attributes in the row.



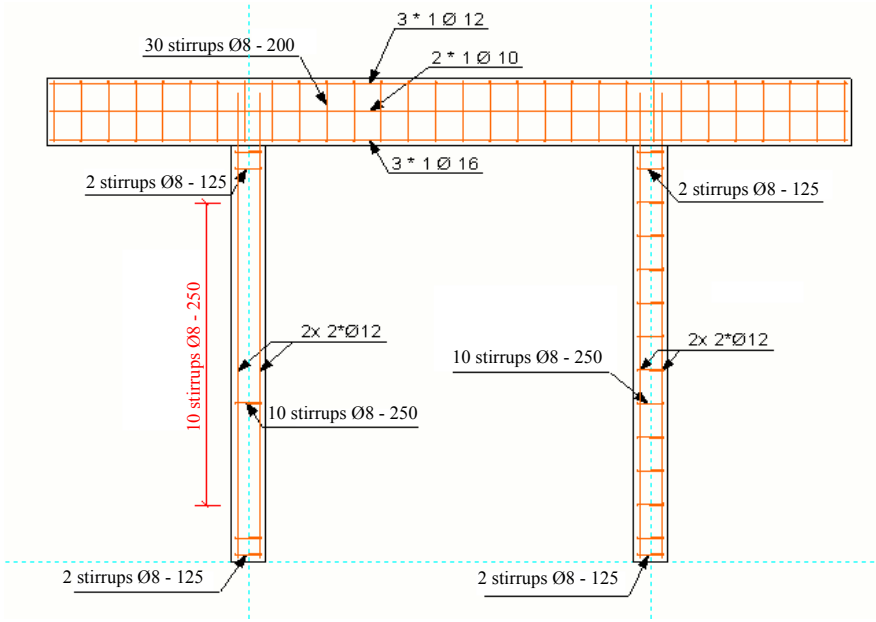
Practical classifier example: Reinforcement mark

Sometimes it is desirable to have different mark layouts for a stirrup groups and longitudinal bars, i.e. the text “stirrups” and centre-to-centre distances in the mark.

When you use the name "STIRRUP" while modeling stirrups (weather created by using a component or manually), you can filter on them. We can use the name "strirrup" for this filter.

Next you can add the text "strirrup" or the user-defined attribute ‘REBAR_NAME’ and the and centre-to-centre distance for the mark properties

Save this setting by using the name: **filter_cast unit_stirrups**.



- By using this classifier setting, the stirrup marks will differ from the bar marks.

The classifier possibilities are extensive and you can apply the classifier for many uses. You can create settings for anything you can filter on in combination with all the drawing settings.

You can copy select (filter) settings to the folder:

disc:\TeklaStructures\version\environments\europe\ts

Because of this you can use them in all projects and these settings will also be copied automatically to a newer Tekla Structures version while updating.

4.6 Appendix

Reinforcement related reports

Castunit-reinforcement

```
TEKLA STRUCTURES CASTUNIT REINFORCEMENT LIST                    Page:   1
Model:templates and reports                                     Project:Project number
                                                                Date:   04.01.2008
                                                                Time:   14:28:44

Unit        Qty.     Grade       Volume (m³)  Weight (kg)
B1           1        C45/55        1.05          2520

        Rebars:   Pos Qty.  Ø   Length  Shape   a    b    c    d   w/p      kg
                    2    3   16   5949    A   5949                    9.4   28.2
                    3    3   12   5949    A   5949                    5.3   15.8
                    4    2   10   5949    A   5949                    3.7    7.3
                    1   25    8   1654    U    142  450  300  450     0.7   16.3
                                             300  142
                                                                    +---------
                                                                Weight:      67.7
        Total Unit weight incl. reinforcement:  2588 kg

Unit        Qty.     Grade       Volume (m³)  Weight (kg)
W4           1        C45/55        2.60          6240

        Meshes:   Pos  Name        Qty.   Ø - c/c - size            w/p      kg
                    5 .              2   8/8-150/150-2520/4920     102.7  205.5
                                                                      +--------
                                                                Weight:     205.5
        Total Unit weight incl. reinforcement:  6445 kg
```

Bending-schedule

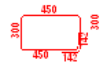
```
TEKLA STRUCTURES BENDING SCHEDULE                               Page:   1
Model:templates and reports                                     Project:Project number
                                                                Date:   04.01.2008
                                                                Time:   14:34:01

Number Ø   Length  Shape Part   Grade Wg./bar  Weight    Size          Comment
  25    8    1654   U    1      S500    0.65     16.3     142 450 300
   2   10    5949   A    4      S500    3.67      7.3    5949
   3   12    5949   A    3      S500    5.28     15.8    5949
   3   16    5949   A    2      S500    9.40     28.2    5949

                                    Total       68 kg
```

Bending-schedule-shape.html

BENDING SCHEDULE WITH SHAPES

Partnr.	Øk	Number	Length	Wg/bar	Weight	Shape
2	16	3	5949	9.40	28	
3	12	3	5949	5.28	16	
4	10	2	5949	3.67	7	
1	8	25	1654	0.65	16	

Total Weight (kg): 68

Phase-reinforcement-list

```
-------------------------------------------------------------------------------------
TEKLA STRUCTURES PHASE REINFORCEMENT LIST                    Page:     1
Model: templates and reports                                 Project:  Project number
                                                             Date:     04.01.2008
                                                             Time:     14:54:54

-------------------------------------------------------------------------------------
Phase  Name           Part Number Ø Length Shape  A    B  C   D   E    Wg/bar  Wg.
-------------------------------------------------------------------------------------
 3     Phase 3        1      25   8  1654  U     142 450 300 450 300    0.7   16.3
 3     Phase 3        2       3  16  5949  A    5949                    9.4   28.2
 3     Phase 3        3       3  12  5949  A    5949                    5.3   15.8
 3     Phase 3        4       2  10  5949  A    5949                    3.7    7.3
-------------------------------------------------------------------------------------
                              33 rebar in phase    3:Phase 3                 67.7
=====================================================================================

                              33 rebar in all phases                         67.7
-------------------------------------------------------------------------------------
```

Phase-reinforcement-mesh-list

```
-------------------------------------------------------------------------------------
TEKLA STRUCTURES PHASE REINFORCEMENT MESH LIST               Page:     1
Model: templates and reports                                 Project:  Project number
                                                             Date:     04.01.2008
                                                             Time:     14:57:49

-------------------------------------------------------------------------------------
Phase  Name           Part  Number     Ø    C to C        w x l       Wg./mesh  Wg.
-------------------------------------------------------------------------------------
 1     Phase 1        5       2      8/8-   150/150    2520 x 4920      65.3    130.6
-------------------------------------------------------------------------------------
                              2 meshes in phase    1:Phase 1                 130.6
=====================================================================================

                              2 meshes in all phases                         130.6
-------------------------------------------------------------------------------------
```

Id-reinforcement-mesh

TEKLA STRUCTURES ID REINFORCEMENT MESH
Model:templates and reports

Page: 1
Project: Project number
Date: 04.01.2008
Time: 16:28:53

ID	Part	Size	Width	Length	Weight
Id: 4999	5	8/8-150/150-2520/4920	2520	4920	65.3
Id: 5013	5	8/8-150/150-2520/4920	2520	4920	65.3

Id-reinforcement-bar

TEKLA STRUCTURES ID REINFORCEMENT BAR
Model:templates and reports

Page: 1
Project: Project number
Date: 04.01.2008
Time: 16:29:59

ID	Part	Size	Length	Weight	Grade	Cast unit
Id: 4819	1	8	1654	0.7	S500	B1
Id: 4883	4	10	5949	3.7	S500	B1
Id: 4891	4	10	5949	3.7	S500	B1
Id: 4865	3	12	5949	5.3	S500	B1
Id: 4873	3	12	5949	5.3	S500	B1
Id: 4878	3	12	5949	5.3	S500	B1
Id: 4841	2	16	5949	9.4	S500	B1
Id: 4855	2	16	5949	9.4	S500	B1
Id: 4860	2	16	5949	9.4	S500	B1

END

Reinforcement related templates

Bendingshapes

Type	Part.nr.	Number	Grade	Ø	Length	a	b	c	d	e	angle 1	angle 2	bend.radius	kg/p	kg
A	W/2	7	S500	8	1080	1080	144	312						0.4	15.4
J	W/3	11	S500	8	811	362	144	312			39		40	0.3	5.8
													Total weight reinf. (kg) :		21.1

Bendingshapes2

BENDING SHAPE

Partnr.	Number	Grade	Ø	Length	Bend. radius	kg/pcs	kg
1	25	S500	8	1654	64	0.7	16.3
2	1	S500	16	5949		9.4	9.4
3	1	S500	12	5949		5.3	5.3
4	1	S500	10	5949		3.7	3.7
					Total weight reinforcement (kg) :		**35**

Bending-schedule-rebar-shape

BENDING SCHEDULE WITH REBAR SHAPES

Partnr	Diameter	Number	Grade	Length	kg/p	Weight	Rebar shape
2	16	3	S500	5949	9.40	28.2	5950
3	12	3	S500	5949	5.28	15.8	5950
4	10	2	S500	5949	3.67	7.3	5950
1	8	25	S500	1654	0.65	16.3	450 300 300 142 450 142
					Total:	67.7	

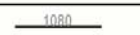
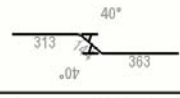
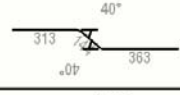
Bending-schedule-rebar-shape-no-text

BENDING SCHEDULE WITH REBAR SHAPES

Partnr	Diameter	Number	Grade	Length	kg/p	Weight	Rebar shape
1	8	25	S500	1654	0.65	16.3	
2	16	3	S500	5949	9.40	28.2	
3	12	3	S500	5949	5.28	15.8	
4	10	2	S500	5949	3.67	7.3	

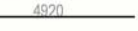
Bending-schedule-rebar-shape-cage-A

BENDING SCHEDULE WITH REBAR SHAPE PER ASSEMBLY

Partnr	Diameter	Number	Grade	Length	kg/p	Weight	Rebar shape
Assembly: I1		1 pcs.					
W/2	8	14		1080	0.43	6.0	
W/3	8	7		811	0.32	2.2	
Total weight per assembly:		8.2		Total weight for		1 assemblies:	8.2 kg
Assembly: I2		1 pcs.					
W/2	8	22		1080	0.43	9.4	
W/3	8	11		811	0.32	3.5	
Total weight per assembly:		12.9		Total weight for		1 assemblies:	12.9 kg
						Total over alll:	21.1 kg

Bending-schedule-rebar-shape-mesh

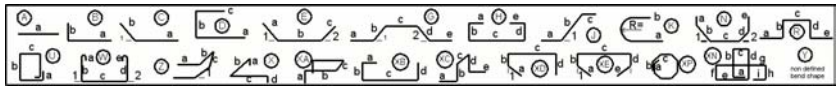
RODS WITH REBAR SHAPES

Partnr	Diameter	Number	Grade	Length	kg/pcs	Weight	Rebar shape
6	8	17	S500	4920	1.94	33.0	
7	8	33	S500	2520	1.00	32.8	
				Total rods:		65.9	

REINFORCEMENT MESHES

Partnr	Diameter - c to c - l*w	Number	kg/pcs	Weight
5	8/8-150/150-2520/492	1	65.3	65.3
		Total meshes:		65.3
	Total weight (kg) reinforcement per element:			131.2

Bending-schedule



Template_A2

REINFORCEMENT: S 500, UNLESS MENTIONED DIFFERENT
CONCRETE COMPRESSIVE STRENGTH FOR TRANSPORT: > 0.66*Fck
BENDING RADIUS REBARS: 2,5*Ø Dk
PROJECTION

Total-weight-reinforcement

Total weight reinforcement: 68 kg

5 Construsoft Newsletter TS 11.0

Topics

This newsletter contains information about the following subjects:

General

- **Adjusted toolbars** (p. 2)
- **Dialogbox NC files improved** (p. 3)
- **Templates and reports** (p. 3)
- **New rapports** (p. 3)

Modeling

- **Coordinate symbol in rendered views** (p. 4)
- **Sketch parametric profiles** (p. 4)
- **Clip planes** (p. 4)
- **Handles** (p. 5)
- **The <Alt> button** (p. 5)
- **Points in the statusbar** (p. 5)
- **Check database** (p. 6)
- **Batch file settings in the user interface** (p. 6)
- **Web viewer models** (p. 7)
- **Making construction lines magnetic in the model** (p. 7)
- **Reference objects in XML format** (p. 7)
- **Text in reference objectsmodels** (p. 7)

Components

- **Custom components** (p. 8)

Drawings

- **Going through the drawings** (p. 10)
- **Confirm save when closing a drawing** (p. 10)
- **Dimension line tags** (p. 11)
- **Dimensioning reference objects in drawings** (p. 11)
- **Combine equal dimensions** (p. 11)
- **Grayscale in hatching** (p. 12)
- **Select for user defined attributes in the drawing** (p. 13)
- **Associative drawings** (p. 14)
- **Family numbering** (p. 14)
- **Intelligent cloning** (p. 14)
- **Surfacing** (p. 14)
- **Create detail views** (p. 15)
- **Move drawing views** (p. 15)
- **Contour plate orientation** (p. 15)
- **Message** (p. 16)

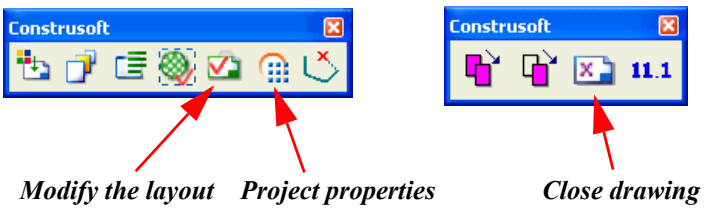
New features and fixes in Tekla Structures 11.2

- **A new dialogbox appears if the numbering is not up to date.** (p. 16)
- **Modifying numbering series** (p. 16)
- **GA dimensioning** (p. 17)
- **Short dimensions** (p. 17)
- **Resizing drawing view boundaries** (p. 18)
- **Default bolt diameter** (p. 19)

General

Adjusted toolbars

Toolbars in Tekla Structures have been improved. For example, new icons have been created for closing drawings and to modify the Layout (**.lay* files) and Project Properties.



Dialogbox NC files improved

The dialogbox to create NC files is improved. You can now create and store self-made settings, for example a setting for plates and a setting for profiles.

The NC files of plates have been improved. The switchingproblems, who were put right by DSTVCONV, are solved. So fixing this problem by using DSTV-CONV is not needed anymore.

For more information, see document Standard Documents.

Templates and reports

When you have choosen a certain Windows font (**.ttf*) in a template, an available font wil be choosen. When you open this template on another workstation it is possible that the templates' look different from the original. This because the choosen font is not available on the other workstation.

New rapports

Some new rapports have been added:

- ***id_boltlist_100***: This report only shows bolts with a lenght of 100 mm. Mostly these are bolts which are modelled incorrect. This is an "id" list so you can easily locate these wrong bolts in the model. Press the button ***Fit workarea by parts in selected views*** if you cannot find the bolts.

- ***id_length_gross***: This report only shows the profiles which differ in LENGTH and LENGTH_GROSS.

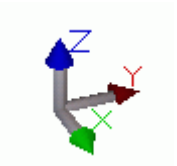
- ***id_loose_parts***: This report shows assemblies consisting of only one part. For example plates you forgot to weld.

- ***id_loose_parts_NOT***: This report is the opposite of the report id_loose_parts. So this report does not select loose plates, filler plates and wind braces consisting of one part.

Modeling

Coordinate symbol in rendered views

The coordinate symbol indicates the direction of the model. It is located in the lower right corner of the model view. The coordinate symbol follows the work-plane.

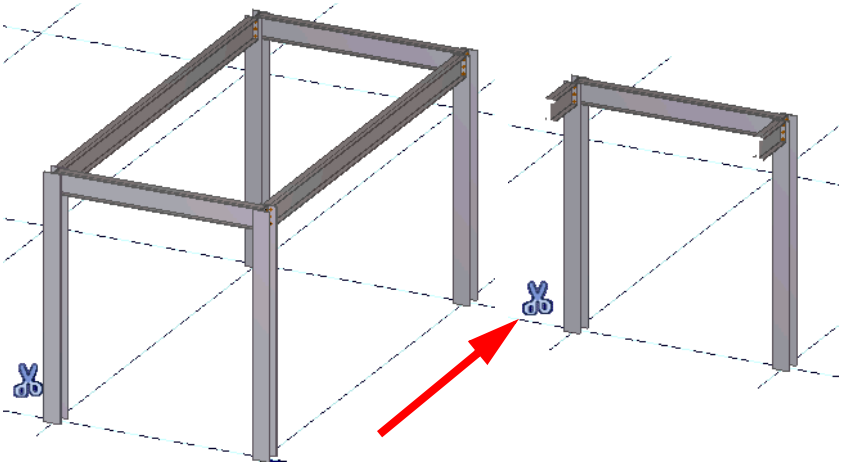


Sketch parametric profiles

Use the sketch editor to define parametric profiles. You can define the shape, dimensions, and parameters of a cross section. For more information, see Standard Documents.

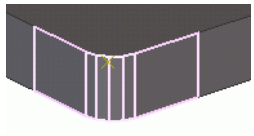
Clip planes

Now you can create up to six clip planes in any rendered model view and in web models:

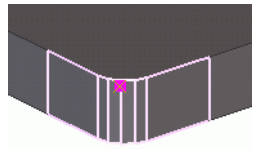


Handles

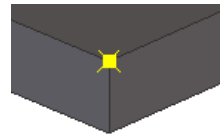
In Tekla Structures 11.x, contour plates contain so-called handles. These handles allow easier modifying the shape of the plate and the chamfers are not visible all the time anymore in the views like it was in earlier versions. So while creating a picture of a modelview(s): the magenta colored chamfers are invisible. The handles' chamfer appears when double-clicking the handle. There are several options:



CHAMFERS



CHAMFERS_AND_ HANDLES



HANDLES

Standard set to *XS_DRAW_CHAMFERS_HANDLES=HANDLES*

The <Alt> button

When you select a part in a model, the handles will highlight. You can eassily select handles by first selecting the part, then pressing the **Alt** key and enclose a window selection by dragging the mouse from left to right. Because of this you can modify the parts in a model much easier and faster.

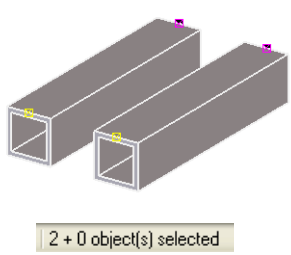


The <Alt> button is also used to select an entire assembly, therefore you have to switch of the crossing selection in the pulldown menu ***Setup***!

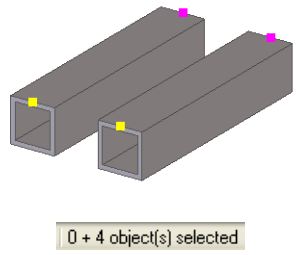
Points in the statusbar

In Tekla Structures selected points are now displayed in the statusbar.

Profiles are selected:

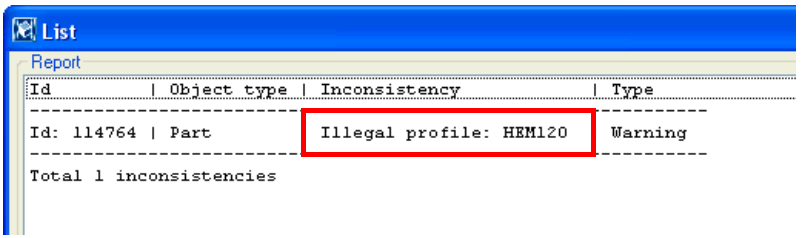


Profiles' points are selected:



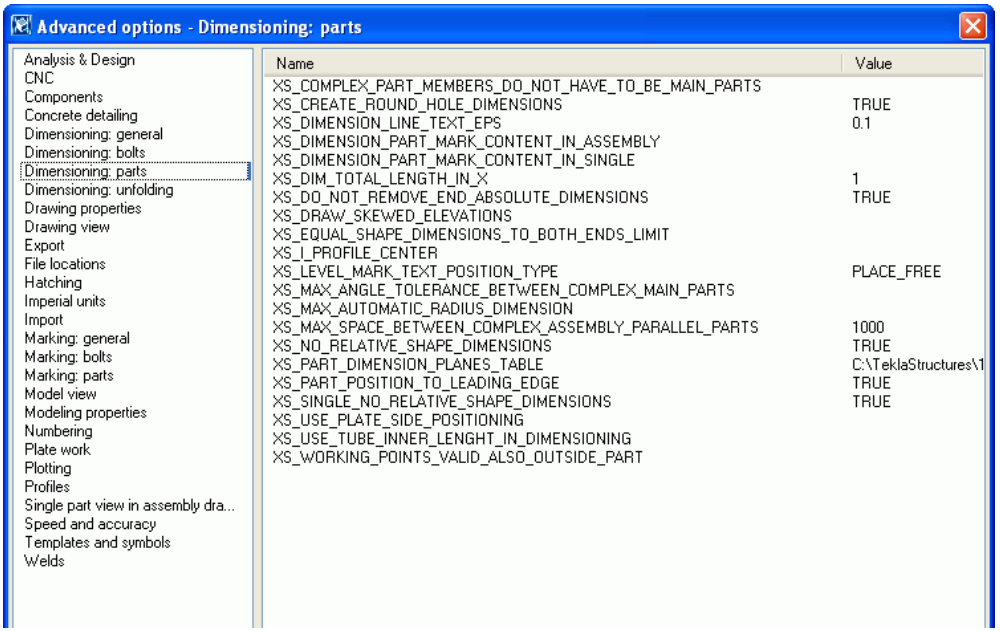
Check database

Check database now shows what are the illegal profiles. See the example below:



Batch file settings in the user interface

To allow easier managing batch file settings, the variables can now be used in a new dialog box. To view or change variable settings, click **Tools** > **Advanced options...** to open the **Advanced options** dialog box. Variables are grouped together in categories. The left pane in the dialog box displays the list of categories. Click a category to view the variables it contains in the right pane:



Changes you make in the **Advanced options** dialog box are stored in the model folder in the file ***options.ini.***

Web viewer models

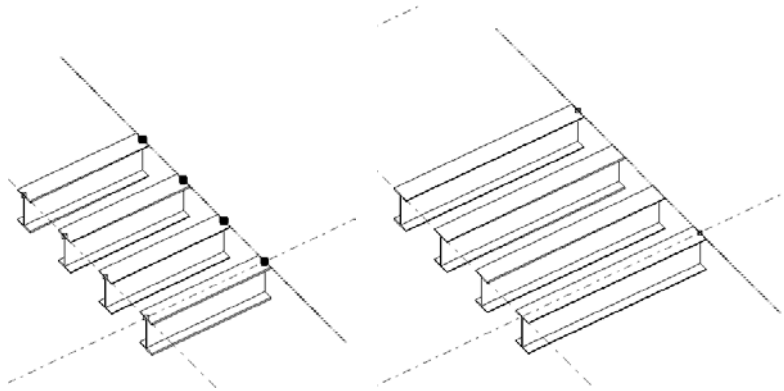
Towards web viewer models, a lot of improvements have been made such as:

1. Clip planes added
2. Profile information added

For more information, see Release Notes 11.0 and 11.1.

Making construction lines magnetic in the model

You can now easily make construction lines magnetic. Because of this, the profiles' handles on the line move with the construction line. Make sure that the profiles' handles are placed exactly on the construction line! This is useful, for example, when modeling facades. To make a construction line magnetic, open the **Construction line properties** dialog box and select the **Magnetic** checkbox:



Reference objects in XML format

Tekla Structures imports webviewer models in ***XML format*** as reference objects. This gives you an extra possibility to share the modeldata with other users.

Text in reference objectsmodels

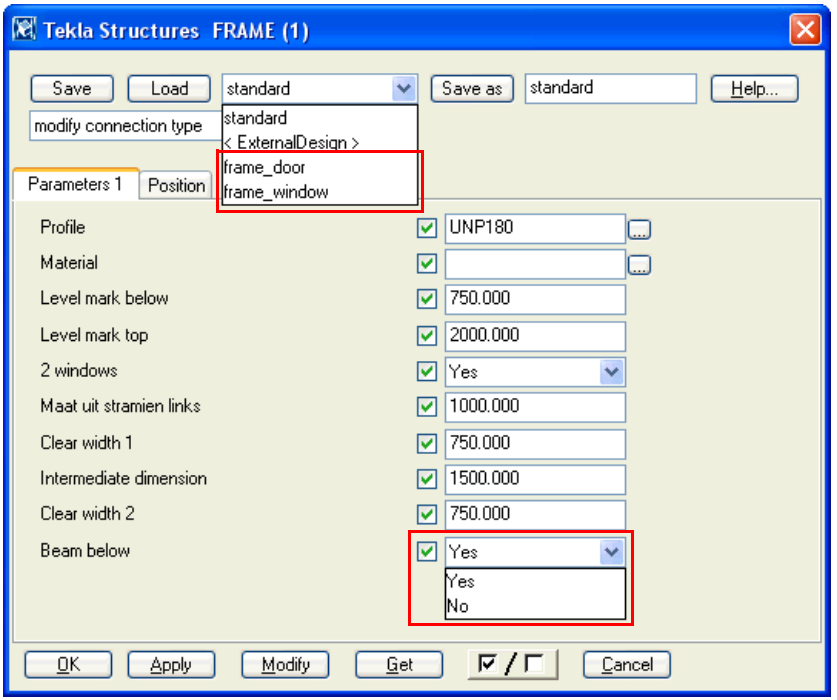
When you import a dxf or a dwg file as a reference object, the text in those files is now also displayed. This text can be saved in Autocad in a separate layer. In those cases you have to copy the so-called *.shx files from Autocad into the Tekla Structures model.

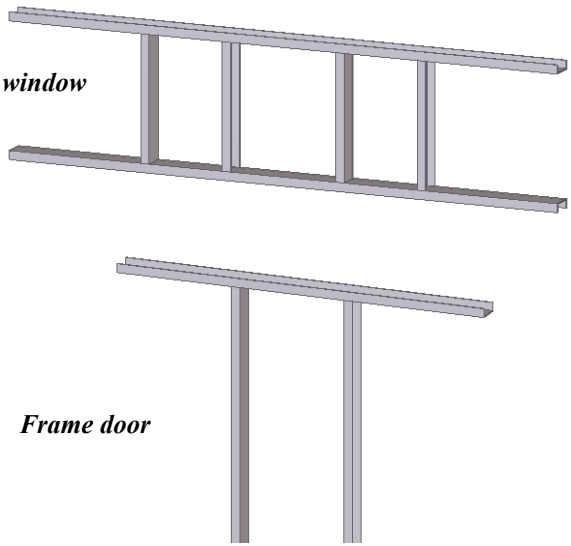
You can also make layers visible or invisible, arcs are displayed correctly. Also blocks are displayed.

Components

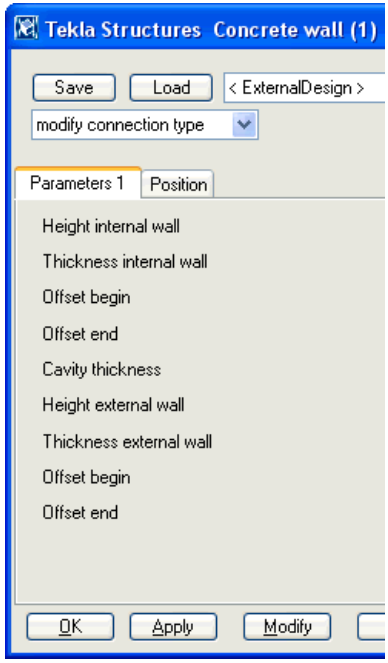
Custom components

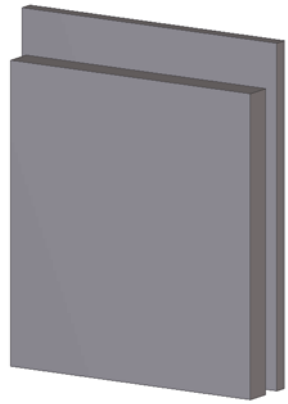
Custom component FRAME.uel is added to the folder *disk:\TeklaStructures\version\environments\europe\custom-components\steel*. You can now also use the option *Yes/No* in custom components.





Custom component CONCRETE_WALL.uel is added to the folder *disk:\Tekla-Structures\versie\environments\europe\custom-components\concrete*.





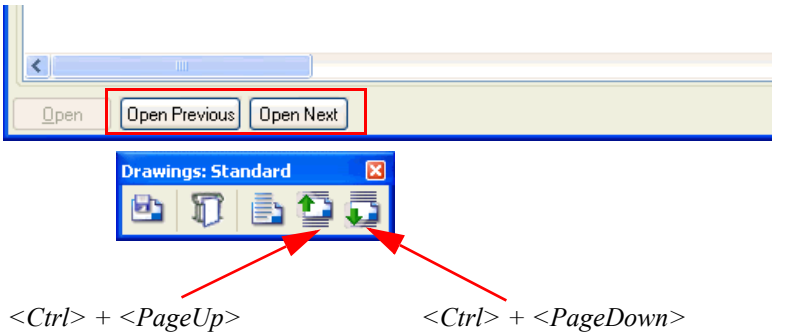
Concrete wall

In the ***Snap settings*** toolbar, the options *Center planes, Grid planes* and *Cut planes* are added. For more information, see Release Notes 11.1 and the document Custom Component.

Drawings

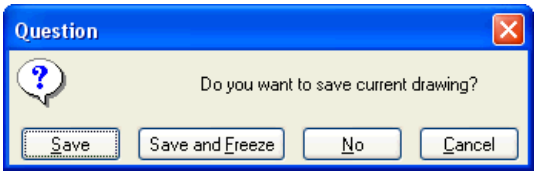
Going through the drawings

Checking of drawings is now made much easier and faster than before. From now on, you can keep the drawing list closed while going through the drawings, as you can **Open next** (default shortcut **Ctrl+Down**) or **Open previous** (default shortcut **Ctrl+Up**) drawing without having to pick them from the drawing list.



Confirm save when closing a drawing

If you close a drawing without making any changes, Tekla Structures no longer prompts you to save the drawing.

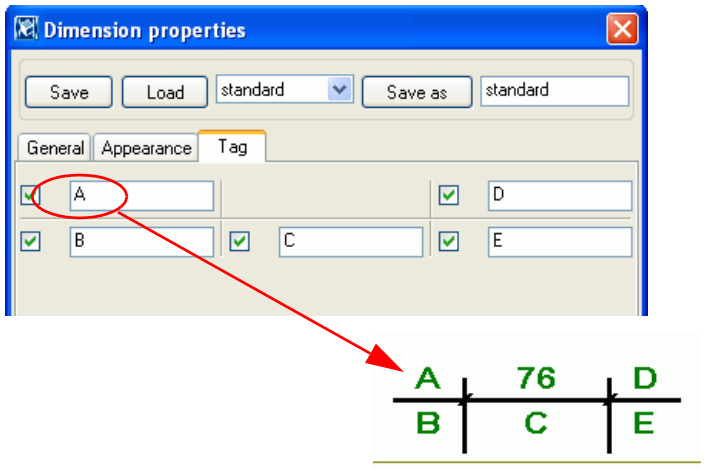


To have Tekla Structures always display the confirmation dialog box:

set XS_ALWAYS_CONFIRM_SAVE_WHEN_CLOSING_DRAWING=TRUE

Dimension line tags

In Tekla Structures you can add user-defined text tags to dimension lines:

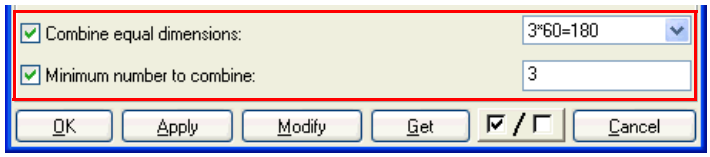


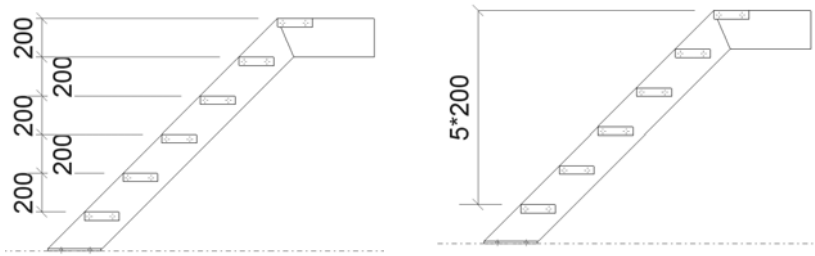
Dimensioning reference objects in drawings

Now you can snap to reference objects in drawings, so you have the possibility to dimension reference objects in drawings.

Combine equal dimensions

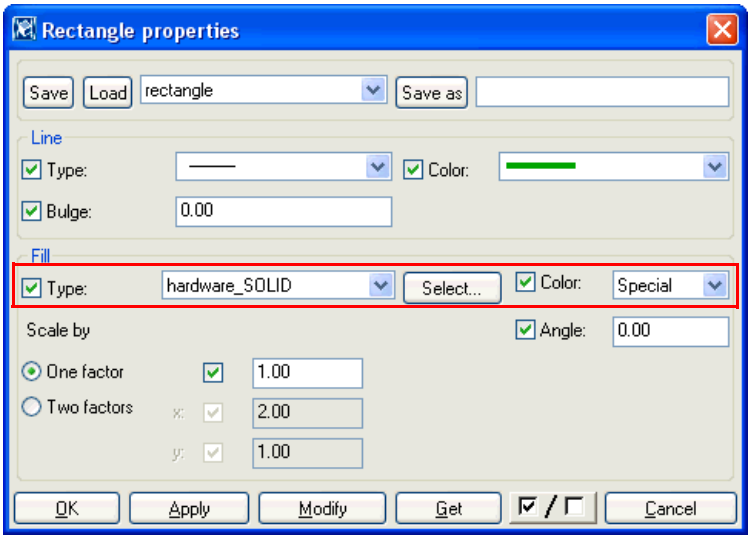
Tekla Structures contains a new option to combine equal dimensions in drawings. You can define how many equal dimensions must exist before they are combined, by entering a number:





Grayscale in hatching

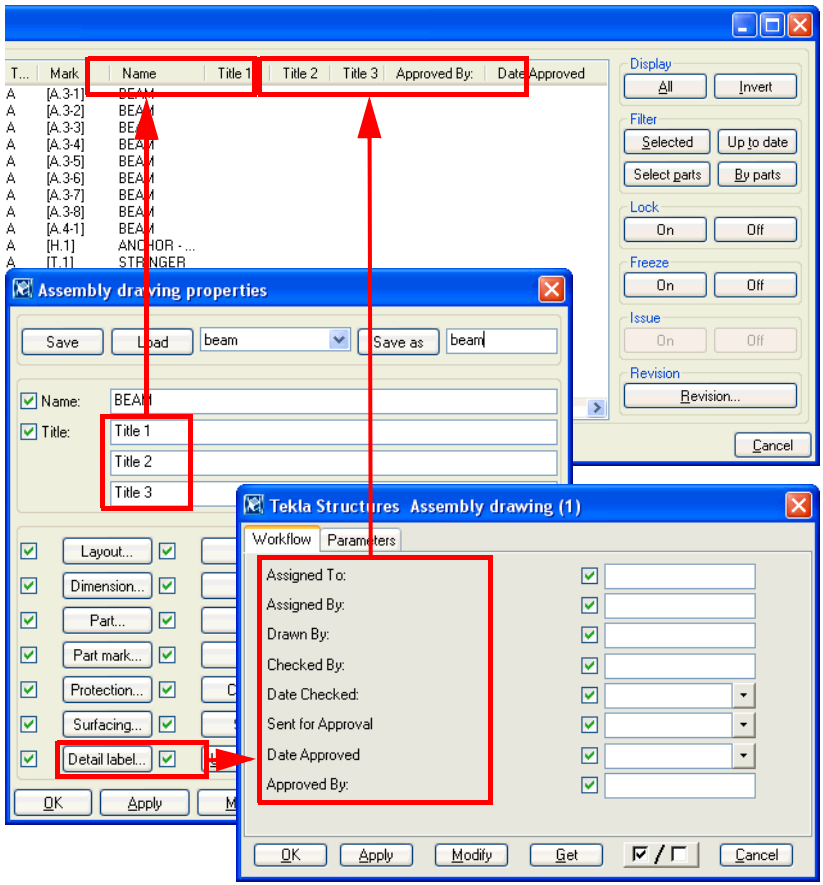
You can now select the option "grayscale" in the listbox. This color will be plotted as a grayscale, depending on the selected printer settings. The hatch color is defined using RGB (Red Green Blue) values on a scale of 0 to 255.



Select for user defined attributes in the drawing

Some columns are added to the drawinglist.

- 3 columns for Drawing Titles
- maximum number of 20 rows for user defined attributes



Associative drawings

All drawing types in Tekla Structures are associative, whether they are created manually or automatically.

This means that drawing objects are linked to model objects, so that most drawing objects automatically update if the model changes, but you won't lose manual changes made to drawings.



To keep manual changes made to single-part, assembly- and cast-unit drawings, freeze the drawings.

It is not necessary to freeze general arrangement or multidrawings, they are always associative.

Family numbering

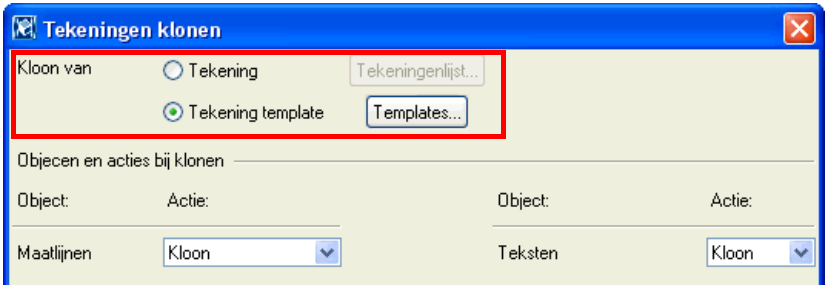
With Family Numbering you can group objects within the same numbering series into different "families". For more information, see Release Notes 11.1.

Intelligent cloning

In Tekla Structures you can now clone drawings and make use of drawings from an ***existing model*** or from a ***standard model***, a so-called drawing template.

You must define the location of the drawing template library by setting the variable XS_DRAWING_TEMPLATES_LIBRARY to point to the model folder that contains the template drawings, for example:

set XS_DRAWING_TEMPLATES_LIBRARY=C:\TeklaStructuresModels\CloningTemplate (where CloningTemplate is the model name).



Surfacing

For more information, see the Standard Documents.

Create detail views

For more information, see Release Notes 11.1.

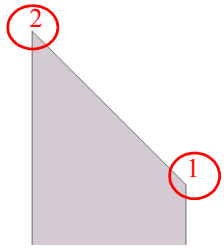
Move drawing views

For more information, see Release Notes 11.1.

Contour plate orientation

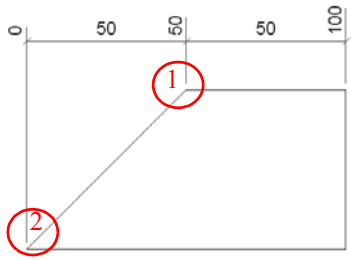
Instead of automatic plate orientation, you can now exert influence on the contour plates' main axis to follow the line created by the first and second picked points of the plate, regardless of the plate dimensions. This enables you to manually define the plate orientation in drawings or reports.





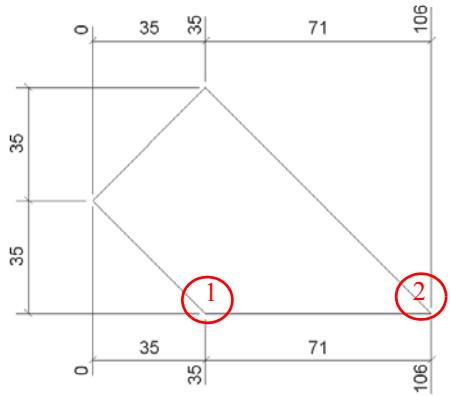
In the model:

First and second picked points



Single-part drawing:

User defined attribute ***Main axis direction*** is ***Automatic.***

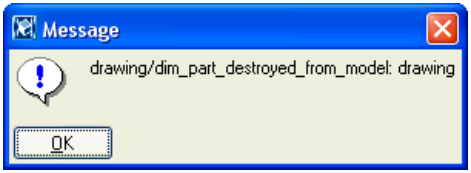


Single-part drawing:

User defined attribute ***Main axis direction*** is ***From 1st to 2nd creation point.***

Message

In Tekla Structures 11.x the following message can appear:



This message appears because the ID-number has changed. All objects in Tekla Structures have an ID-number, to wich the associative dimensioning is related.

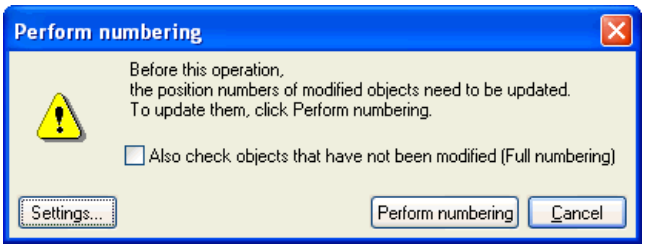
When an object is deleted from a model and a drawing dimension is (was) related to that object, and you then open a drawing, this message will appear.

Possible dimension lines who were related to the involved object are deleted automatically because the part does not exist anymore.

Tekla Structures 11.2

A new dialogbox appears if the numbering is not up to date.

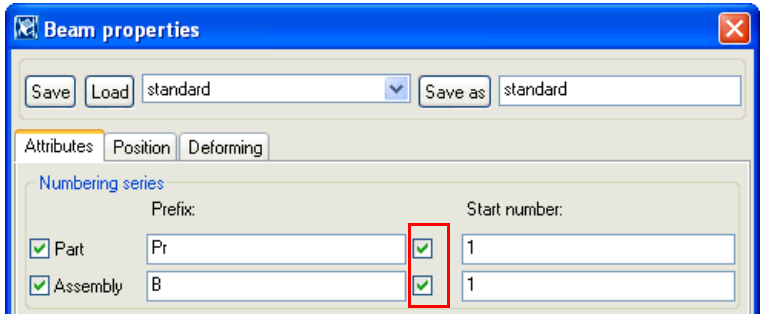
If you tried to create drawings but model numbering was not up to date, you had to cancel the drawing creation and could only continue after the model numbering was updated. Now when you create a drawing, and the numbering is not up to date, Tekla Structures 11.2 displays a warning message:



When you click Perform numbering, the numbering is being updated and drawing creating will be performed subsequently.

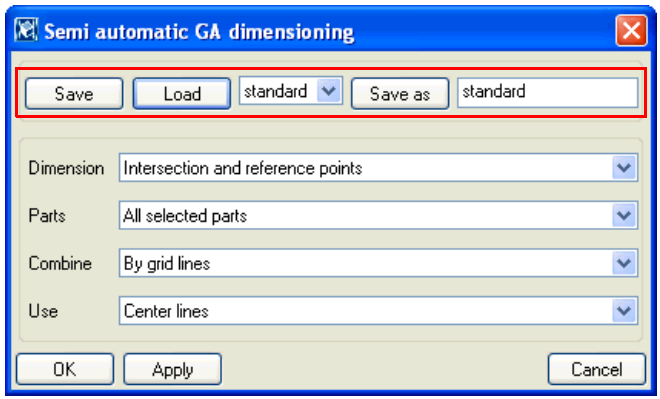
Modifying numbering series

Now modifying numbering series is a lot easier than before: You can select the checkboxes separately for **Prefix** and **Start number** in frame **Numbering series** in the dialog box Part Properties:



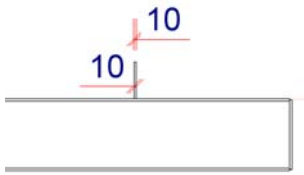
GA dimensioning

You can store settings for semi automatic dimensioning, for example to dimension anchors on GA drawings.



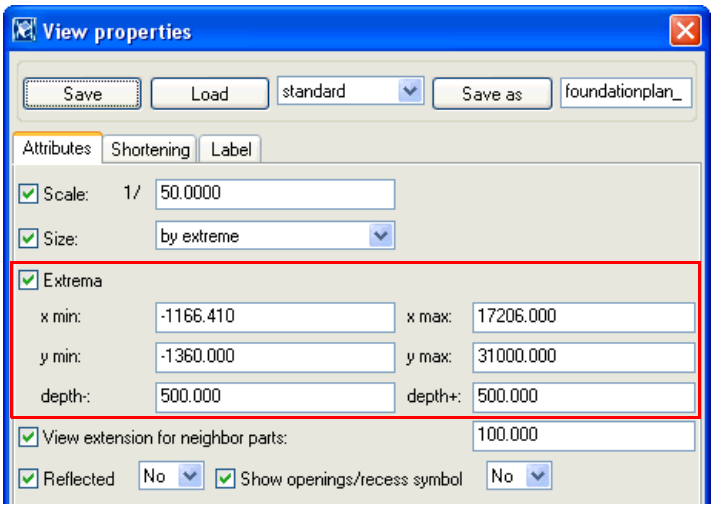
Short dimensions

When you select **Outside** as the **Short dimensions** type in the Dimension properties dialog box, Tekla Structures places the dimension on the side of the last picked point:

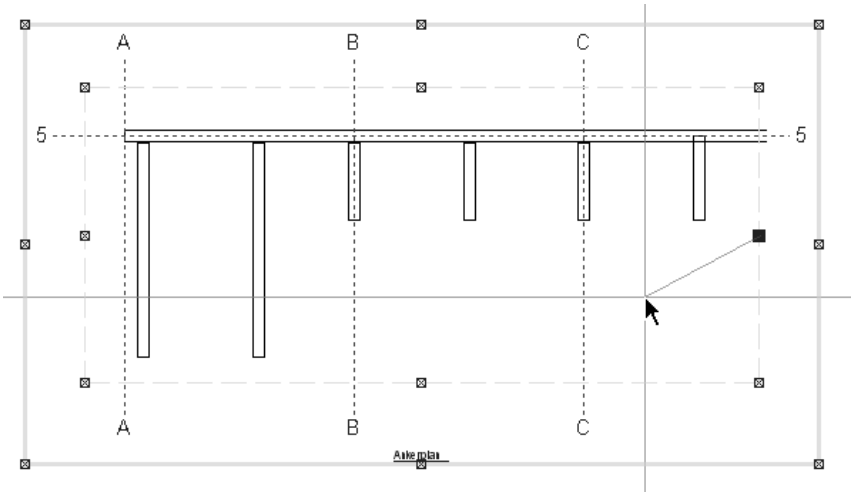


Resizing drawing view boundaries

Modifying drawing view boundaries is made easier in Tekla Structures 11.2:

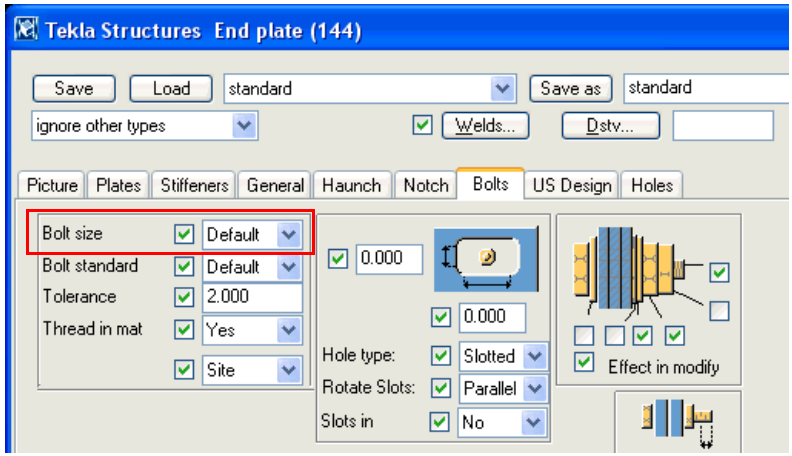


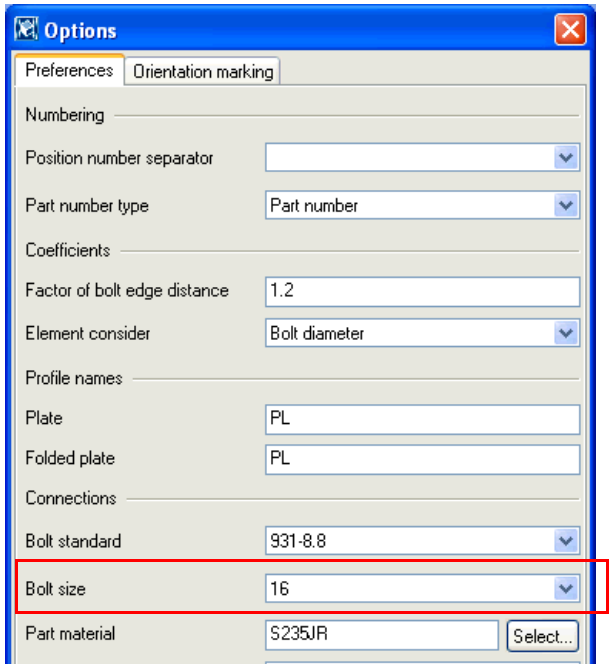
To modify the drawing view boundaries you can now drag the view plane:



Default bolt diameter

The projects' default bolt diameter can be stored in the modelfolder. Therefore in the macro the bolt size is set to ***default***.





Notes:

6 Construsoft Newsletter TS 11.2

Topics

This newsletter about Tekla Structures 11.2 contains information about the following subjects:

General:

- **Installing Tekla Structures 11.2** (blz. 3)
- **Construsoft Optimize Installation Tool (COIT)** (blz. 3)
- **3D dwg in installation added** (blz. 5)
- **Toolbars and shortcuts** (blz. 5)
- **NC files** (blz. 5)
- **Use relative file paths for NC files** (blz. 6)
- **Clipplanes** (blz. 8)
- **Opening reports automatically in Word** (blz. 9)
- **New templates** (blz. 9)

Modeling:

- **Mirroring boltgroups** (blz. 10)
- **Surface treatment select filter** (blz. 10)
- **Tiled surface treatment** (blz. 10)
- **Parametric profiles** (blz. 11)
- **Separator character in parametric profile names** (blz. 11)
- **Modifying numbering series** (blz. 12)
- **Define default bolt size in macro's** (blz. 12)
- **Haunch (40)** (blz. 13)
- **Web viewer models** (blz. 14)

Drawings:

- **Creating or updating and opening a drawing of a selected part or assembly automatically** (blz. 15)
- **Dimension planes** (blz. 15)
- **Open templates from within drawings** (blz. 18)
- **Level marks** (blz. 18)
- **Save drawings** (blz. 19)
- **Assign numbers** (blz. 20)
- **Update numbering** (blz. 20)
- **New copy and move options for drawing objects** (blz. 21)
- **Resizing drawing views** (blz. 21)
- **Hatches** (blz. 22)
- **New view settings** (blz. 23)
- **New setting for assembly drawings** (blz. 24)
- **Multiple lines of text** (blz. 25)
- **Angle dimension symbol** (blz. 25)
- **Plotting continuous lines** (blz. 26)
- **Create detail- and section views** (blz. 26)
- **Move drawing views** (blz. 27)
- **GA dimensioning** (blz. 27)
- **Short dimensions** (blz. 27)
- **Plot drawing to a file (PDF)** (blz. 28)
- **Associative drawings** (blz. 29)
- **Message** (blz. 29)

New features and fixes in Tekla Structures 11.3:

- **Hide and unhide objects in drawings** (blz. 30)
- **View name labels in assembly and single-part drawings** (blz. 31)
- **View orientation symbols** (blz. 32)
- **Add prefix and postfix text to dimension line text** (blz. 33)
- **3D views in assembly and single part drawings** (blz. 35)
- **Section view hatching** (blz. 36)
- **Central gusset (169)** (blz. 37)
- **Comb filler plates for Two sided end plate (142) and End plate (144)** (blz. 38)

New features and fixes in Tekla Structures 12.0:

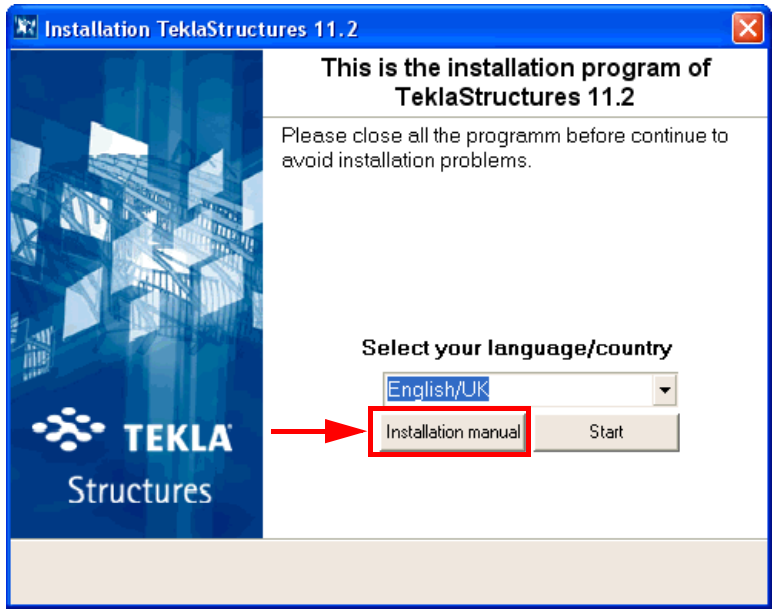
Level marks (blz. 39)

Notes: (blz. 40)

General

Installing Tekla Structures 11.2

When you start installing Tekla Structures 11.2, please press the button <Installation Manual> to open the Installation Manual. The pdf-file that will open, guides you through the complete installation.



Construsoft Optimize Installation Tool (COIT)

The default installation of Tekla Structures copies a large number of setting-files to the hard drive.

These are files like joint properties, column/beam properties, reports etc.

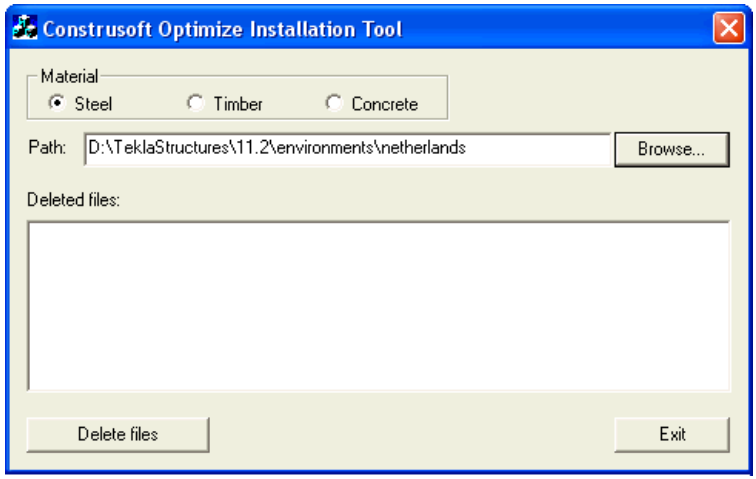
In previous versions of Tekla Structures you had to remove those files manually (using Windows Explorer) if you did not want to use the provided setting files.

This is not necessary anymore. With the “Construsoft Optimize Installation Tool” (COIT) you can configure which files you want to keep and which files have to be removed.

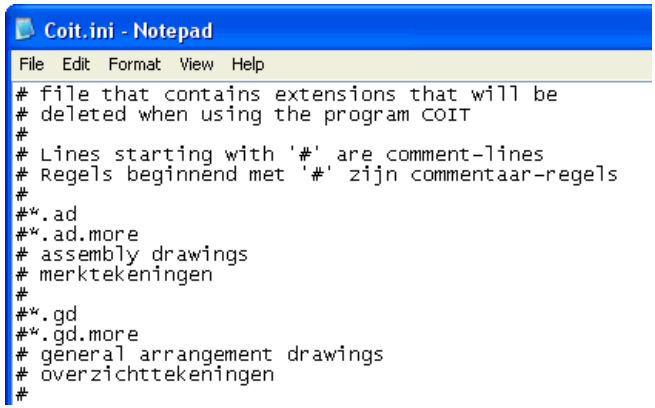
You can download the programm from the Xtranet; go to ***Downloads*** > ***Other’s***.

This program contains:

- ***COIT.exe***, this is the program to delete the files:



- ***Coit.ini***, the setting-file in which you can enter the types of files you want to delete:



- ***COIT_ENU.pdf***, this is the complete English manual about the program.



The files will be deleted from your computer and will **not** end up in the Recycle Bin!

3D dwg in installation added

Some new 3D dwg files have been added, you can use these files as a reference object.

These ***.dwg*** files are added to the folder:

disk:\TeklaStructures\11.2\environments\europe\system

Toolbars and shortcuts

The registration entries in the folder:

disk:\TeklaStructures\11.2\environments\europe\system

are added and extended:

- TS11.2_steel_detailing_W2K_XP.reg
- TS11.2_standard_design_W2K_XP.reg
- TS11.2_concrete_detailing_W2K_XP.reg
- TS11.2_project_manager_W2K_XP.reg
- TS11.2_viewer_W2K_XP.reg

By adding one of these registration entries, depends on the module you use, to the register of your computer, you can make use of optimized toolbars and shortcuts which are supplied by Construsoft.

When you already work with Tekla Structures and you made use of the Construsoft toolbars and shortcuts, then they will be automatically copied to Tekla Structures 11.2.

Keep in mind that the ***.reg*** files mentoined above, supplied by Construsoft, serve Tekla Structures 11.2 much better than the toolbars and shortcuts who are automatically copied from a previous version.

If you want to make use of the new ones, you must delete the branch from the registry in which the toolbars and shortcuts are stored.

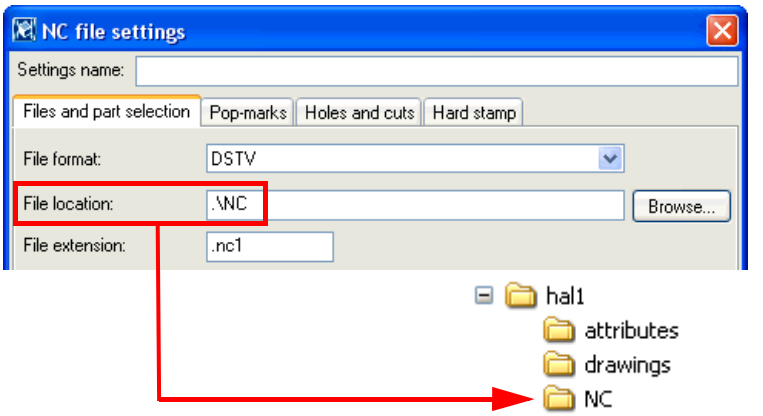
For more information, see the Basetraining and the Tekla Structures 11.2 Installation Guide.

NC files

In Tekla Structures 11.2 and Tekla Structures 11.1.2, all known issues at Construsoft about NC files for plates, are solved.

Use relative file paths for NC files

You can now enter relative paths in the **File location** fields of the **NC file settings** dialogbox:



The variabele XS_MIS_FILE_DIRECTORY affects where files are created:

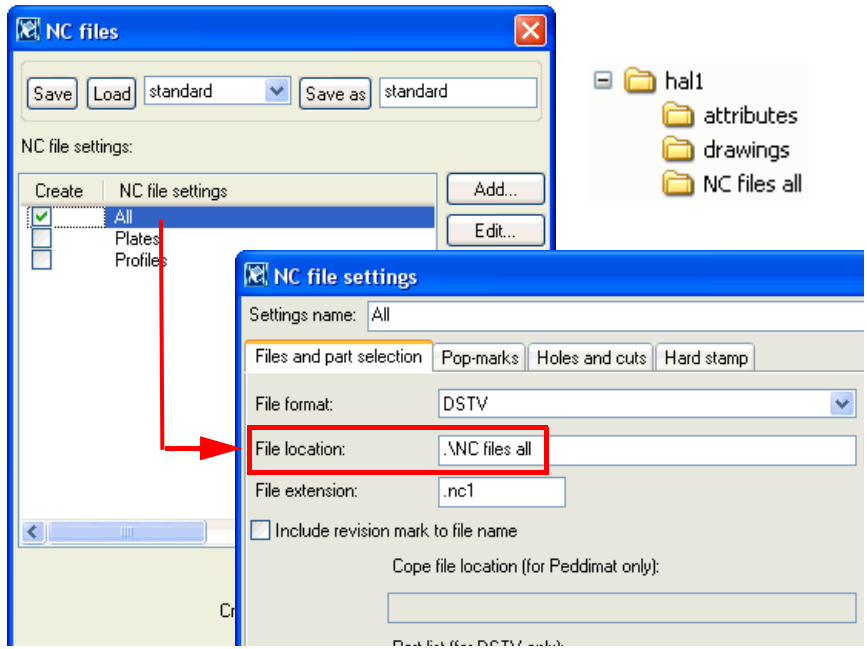
When XS_MIS_FILE_DIRECTORY is set to C:\NC, files are created as follows:

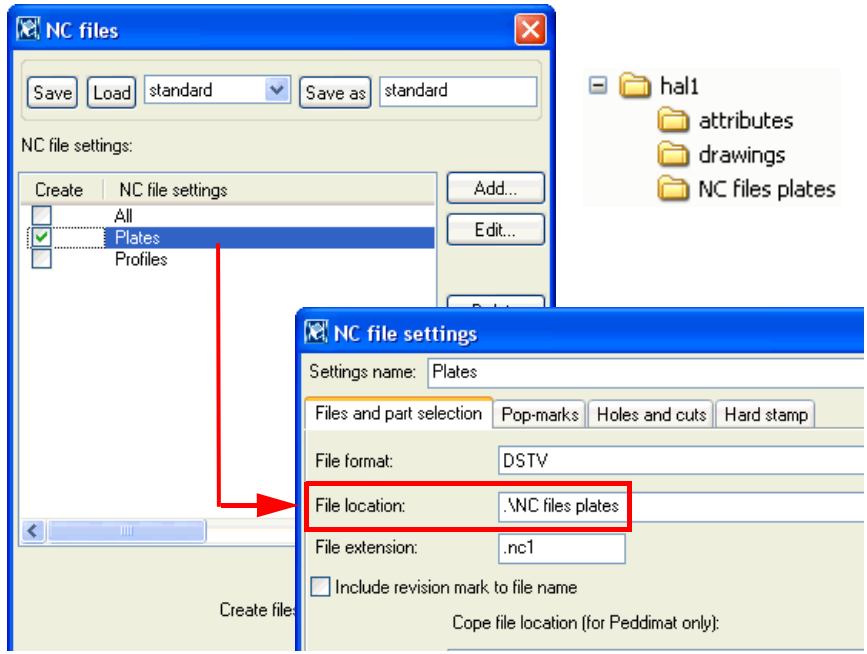
File location	NC files created in:
	C:\NC
.\	C:\NC\ModelName
.\MyFiles	C:\NC\ModelName\MyFiles
C:\TEMP	C:\TEMP

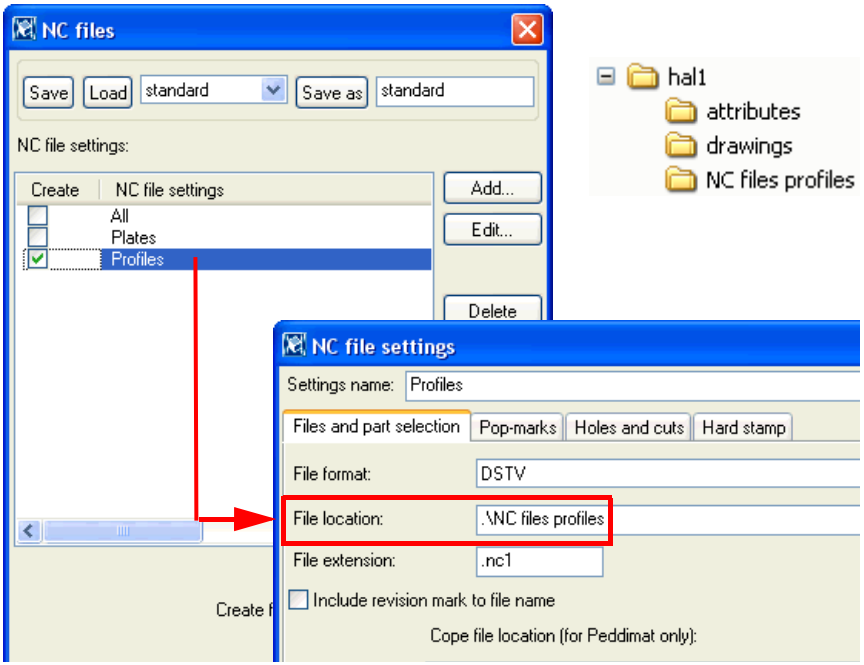
When the variabele XS_MIS_FILE_DIRECTORY is not set, files are created as follows:

File location	NC files created in:
	model folder
.\	model folder
.\MyFiles	model folder\MyFiles
C:\TEMP	C:\TEMP

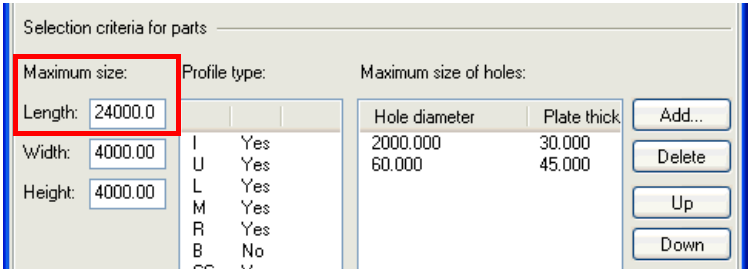
In Tekla Structures 11.2 the created folder is entered as follows:







Also for all NC file settings, the length for the maximum size has been changed from **15000** to **24000**:



Clipplanes

When you make use of clipplanes (Model Editor or Web model), you have to make sure that no other objects are selected when you delete a clipplane-symbol. Otherwise they will be deleted too.

Opening reports automatically in Word

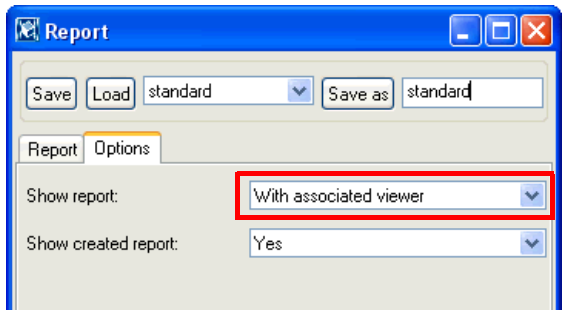
Reports, created in Tekla Structures, can be opened in Word automatically. Because of that, the name of the report must be changed:

Go to the folder:

disk:\TeklaStructures\11.2\environments\europe\firm\steel\template

Select a report, e.g. **phase_manager.rpt** and rename this file to **phase_manager.doc.rpt**

Set the associated viewer in the tab <Options> in the dialogbox *Report* in Tekla Structures. When you now generate a report, it will be opened in Word. In the modelfolder, the file **phase_manager.doc** is stored.



Of course, reports in Tekla Structures can be automatically opened in Excel too. For more information, see the Standard Documents. Make sure that Word (or Excel) is installed on your computer!

New templates

Some new templates have been added, the templates' output for ***assembly_lot_number*** and ***lot_assembly_number*** is equal, only the sort order differs, this also counts for ***assembly_phase_number*** and ***phase_assembly_number***:

ASSY	LOT	NUMBER
A3	1	1
A3	2	2

assembly_lot_number

LOT	ASSY	NUMBER
1	A3	1
2	A3	2

lot_assembly_number

ASSY	PHASE	NUMBER
A3	1	1
A3	4	2

assembly_phase_number

PHASE	ASSY	NUMBER
1	A3	1
4	A3	2

phase_assembly_number

Modeling

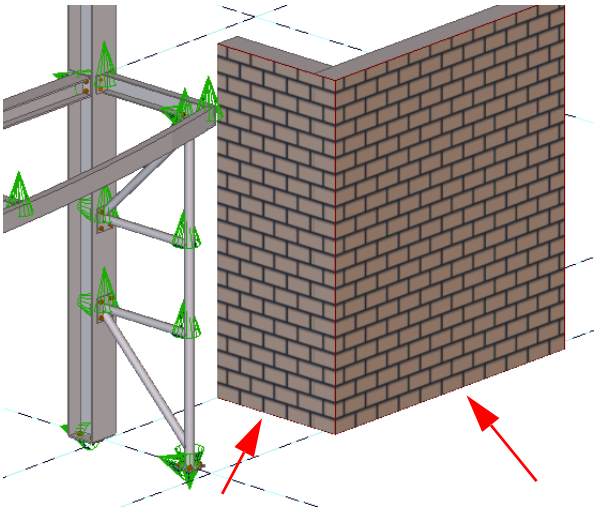
Mirroring boltgroups

Mirroring a-symetric boltgroups, and countersunk bolts, and offset bolts is now correct.

Surface treatment select filter

You can now use a new surface treatment filter to fast and eassily filter surfacings:





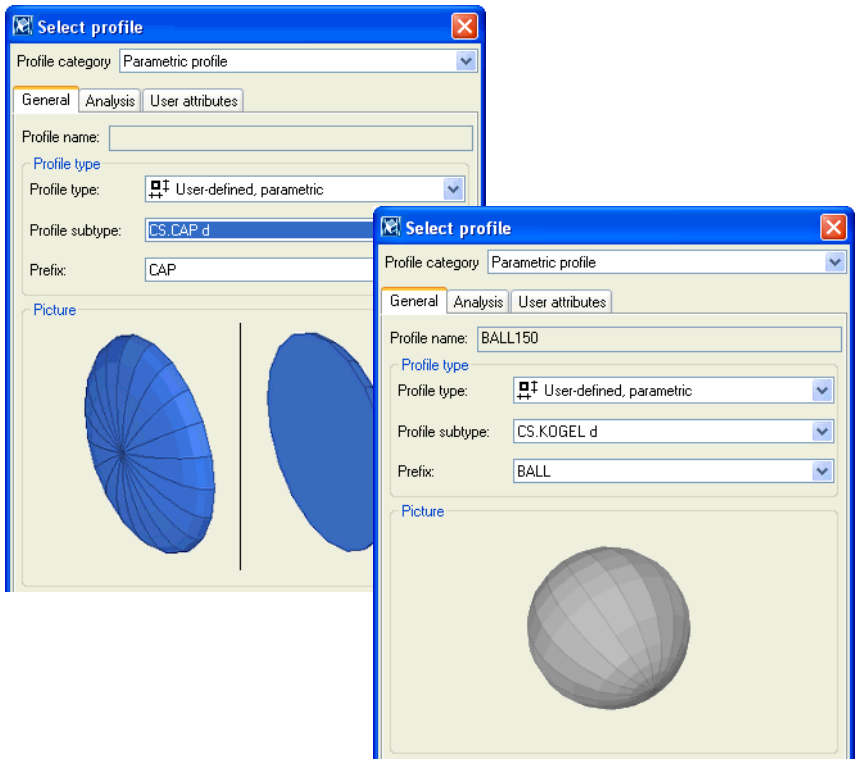
Tiled surface treatment

You can now define your own tile patterns. For the time being, this setting can not be stored.

For more information about defining tile patterns, see the Release Notes 11.2.

Parametric profiles

Two parametric subtypes have been added to the parametric profiles.



Separator character in parametric profile names

The character / is not accepted anymore as a separator character for parametric profiles, it caused problems using the commands copy from model or importing in a model in which these parametric profiles were stored. These profiles were recognized as other profiles.

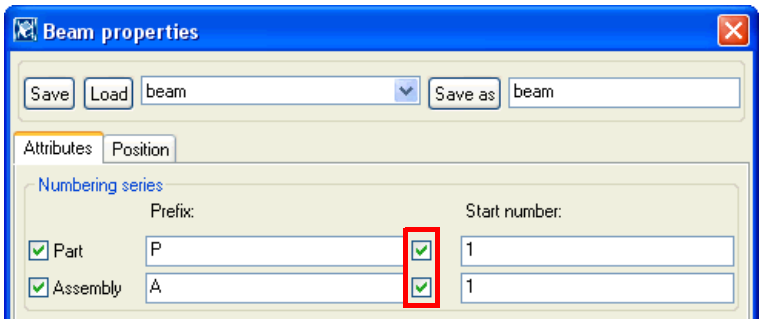
You can only use the characters -, X and * to separate dimensions in parametric profile names. You can set the variable:
XS_PARAMETRIC_PROFILE_SEPARATOR to accept the character / as a separator character, however, the output, like templates and reports, also gets the character / and not the character * like it is now.



When opening a model that is created in a previous version, **always** check the database with the command **Extra** > **Check database**! When the notice "Illegal profile" appears, you must always fix the problem.

Modifying numbering series

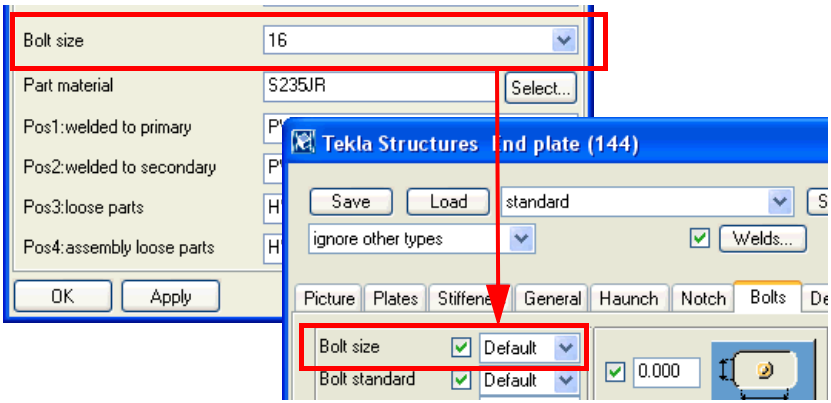
You can now modify numbering series by selecting the checkboxes separately for **Prefix** and **Start number** in the dialogbox Part Properties:



The advantage is to modify startnumbers for beams and colomns, who have different prefixes, at once.

Define default bolt size in macro's

To define the default bolt size used in macro's, click **Setup > Options** and select a bolt size in the listbox. When the bolt size in de macro is set to **< Default>,** the value, defined in the dialogbox **Options**, is used:



To use this setting for all models:

- Go to **Setup** > **Save defaults**
- Open the modelfolder and browse to the folder *attributes*
- Copy the file *standard.stp* to the folder *ts* from your current version and restart Tekla Structures

In all macro settings containing the name **standard.j***, the bolt size is set to <Default> and not to any value anymore. The other settings are not changed.

Haunch (40)

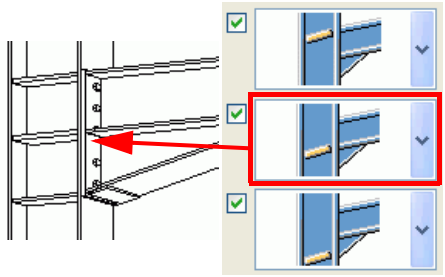
Macro **Haunch (40)** on macropage *7* contains a lot of improvements such as:

Stiffeners welded to the column

Stiffeners are now welded to the column. Sometimes the stiffeners are not placed correctly, this is fixed in Tekla Structures 11.3. When you want to weld stiffeners to the column, make us of macro **Haunch (40)** on macropage ***CS5***.

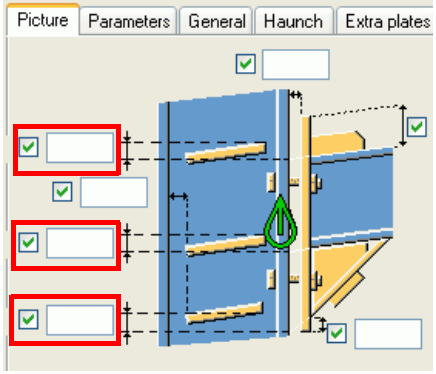
Lower rafter stiffener

You can now create a column stiffener at the lower flange of the rafter:



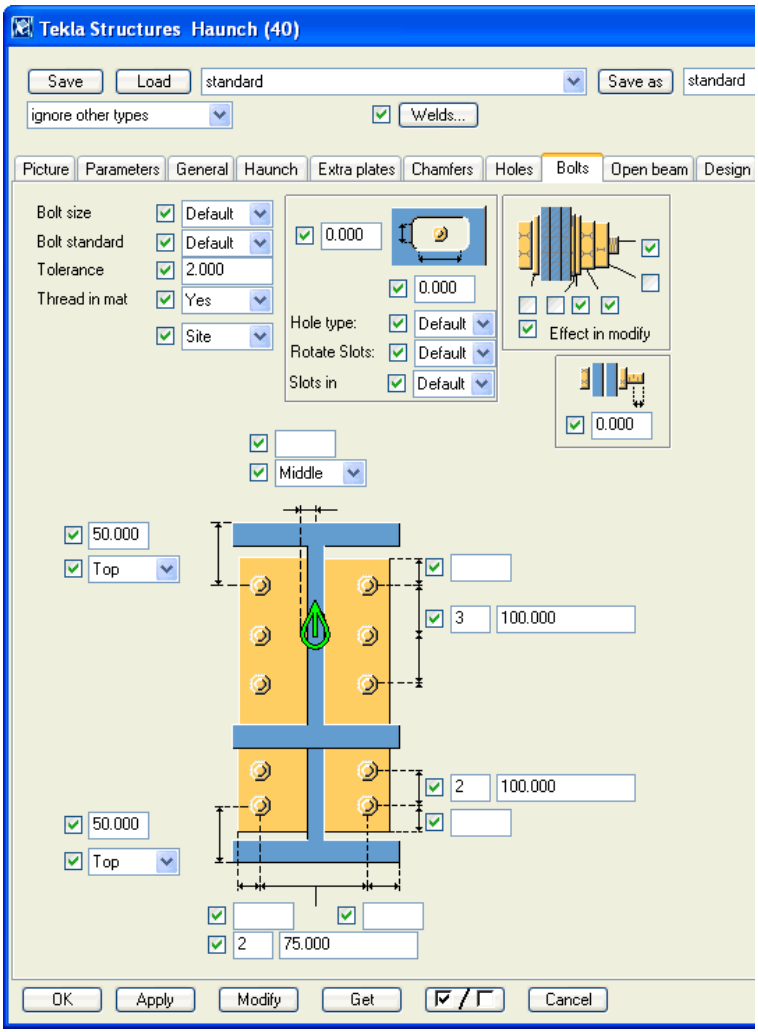
Stiffener z position

You can adjust the position of column stiffeners on the **z** plane:



Tab Bolts

The dialog <*Bolts*> is replaced by the tab Bolts:



For more information about improvements in macro **Haunch (40)**, see the Release Notes 11.2.

Web viewer models

On the Xtranet you can find a new document about web viewer models. You can now use the variable XS_SOLID_USE_HIGHER_ACCURACY=TRUE to view profiles more accurate, so including the rounding radius in profiles, in the Web Viewer and in XML export files. Formerly this was only possible in the Model Editor.

Drawings

Creating or updating and opening a drawing of a selected part or assembly automatically

To automatically open a drawing after creating or updating it, press down the <Ctrl> + <Shift> buttons.

- In the pull-down menu go to **Drawing** > **Single-part drawing, Assembly drawing** or **Cast-unit drawing**
- or press the right-mouse button and select **Drawing** > **Single-part drawing, Assembly drawing** or **Cast-unit drawing** in het pop-up menu

Tekla Structures now:

- Creates a drawing or updates an existing drawing
- Opens a drawing



This only goes for 1 part selected in the model!

Dimension planes

In Tekla Structures you can define how different profiles in drawings are dimensioned. You can define these so-called dimension planes in the file *dim_planes_table.txt.*

This file is located in the folder:

disk:\TeklaStructures\version\environments\europe\profil

In this file is defined that Tekla Structures always dimensions I-profiles smaller than 300 mm from the outside of the profiles, larger I-profiles from the middle of the profile:

dim_planes_table.txt - Notepad

File Edit Format View Help

```
/*** DIM PLANES TABLE for different profiles and dimensions ***/
/***
//value: -1.0: applies for all profiles regardless of dimension!
                          FLANGE                                  WEB
ProfType,MaxDim, middle,right, left,                    middle,right, left,
================================================================
***/

// I-Profiles - t/m 300mm from outside of the profile, the rest from the hart
1,        300.0,    FALSE, TRUE*, TRUE,        FALSE,  TRUE*,  TRUE
1,         -1.0,     TRUE*, FALSE, FALSE,       TRUE*, FALSE, FALSE

// L-Profiles - always from the outside
2,         -1.0,     FALSE, TRUE*, TRUE,        FALSE,  TRUE*,  TRUE

// U-Profiles - always from the outside
4,         -1.0,    FALSE,  TRUE*, FALSE,       FALSE,  TRUE*,  TRUE

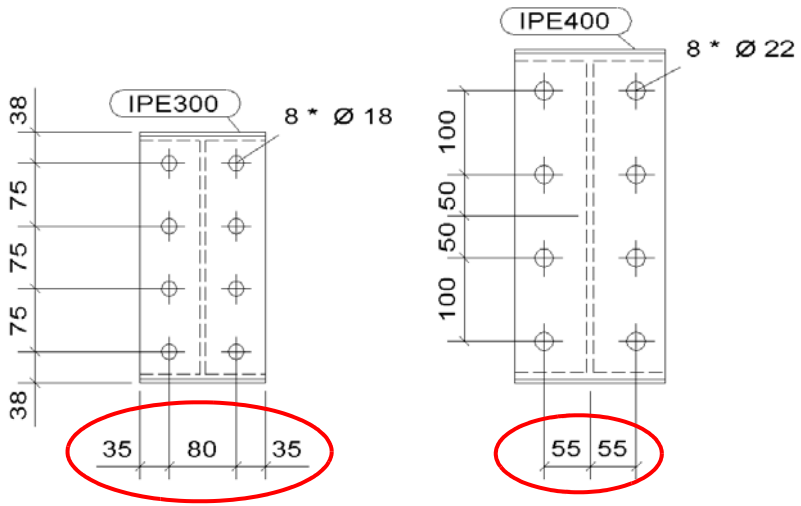
// Timber - always from the center
// 5,         -1.0,    TRUE,  FALSE, FALSE,          TRUE,  FALSE, FALSE

// Massief rond - altijd vanuit het hart
6,         -1.0,    TRUE*,  FALSE,  FALSE,      TRUE*,   FALSE,   FALSE

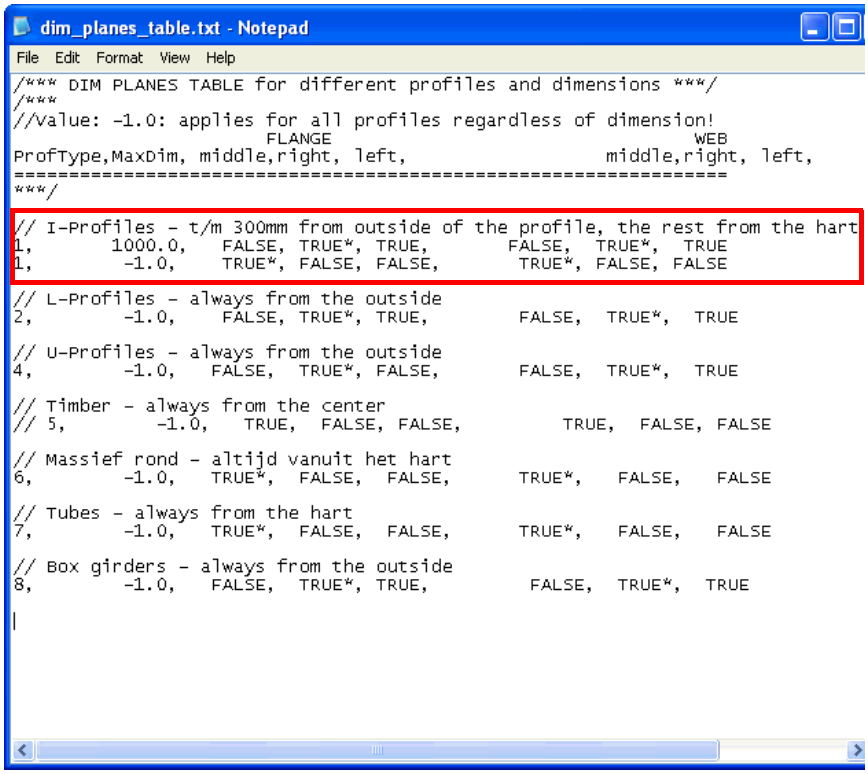
// Tubes - always from the hart
7,         -1.0,    TRUE*,  FALSE,  FALSE,      TRUE*,   FALSE,   FALSE

// Box girders - always from the outside
8,         -1.0,    FALSE,  TRUE*, TRUE,         FALSE,  TRUE*,  TRUE
```

Below an example for an **IPE300** and **IPE400**:



If you want Tekla Structures to dimension all profiles from the outside, you need to change the file:



When the file *dim_planes_table.txt* is modified, you need to store the modified file in the folder:

disk:\TeklaStructures\version\environments\europe\firm\ts

Now open the file **user.bat** (Rightmousebutton > Edit) in the folder:

disk:\TeklaStructures\version\bat\user

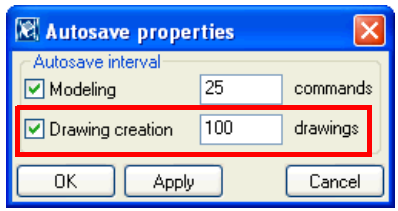
and add the following line to your own settings:

set XS_PART_DIMENSION_PLANES_TABLE=%XS_DIR%\environments\%2\firm\%6\dim_planes_table.txt

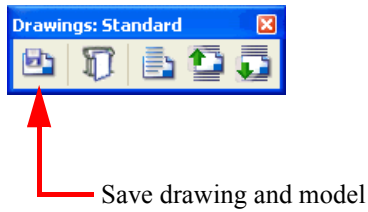
For more information about Dimension planes, see the Basetraining.

Save drawings

In the dialogbox "Autosave properties", a value is entered for Drawing creation. So while generating drawings, making use of the Wizard, after 100 drawings an "Autosave" file is created.



This function only works for generating drawings, making use of the Wizard, **not** for editing drawings in the Drawing Editor. In that case you must save the drawing by clicking the icon:



Now both drawing (*.dg-file) and model (the *.db1 and the *.db2-file) will be saved, so frequently save the drawing (the *.dg-file)!

When you regularly cannot open a drawing anymore, copy the variable

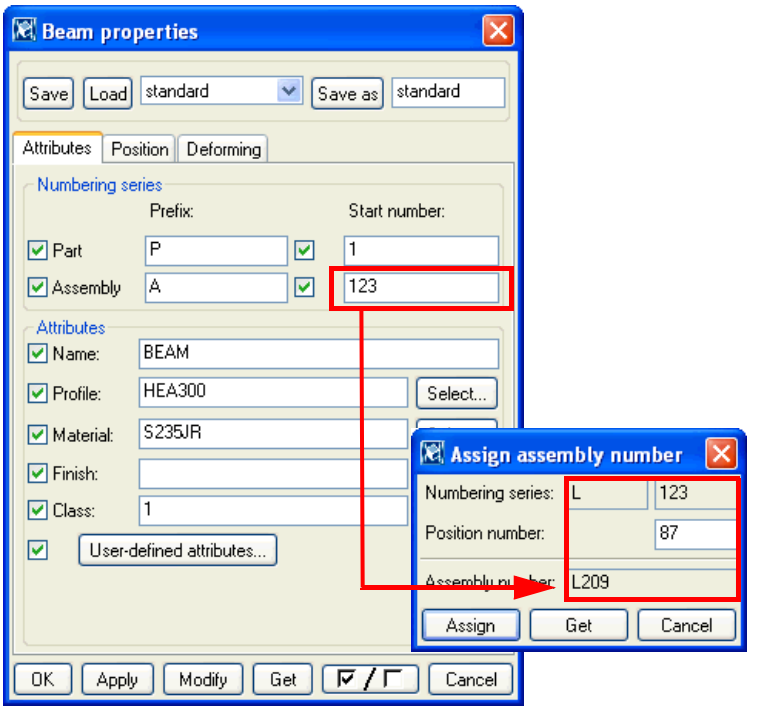
set XS_DO_NOT_DELETE_UNNECESSARY_DG_FILES=TRUE

to the file *user.bat* so that Tekla Structures will not delete *.dg-files anymore.

Lots of *.dg-files are stored now in the folder "drawings", you can solve this issue by setting the variable to FALSE. Now open the model once and close the model, the unnecessary *.dg-file are deleted.

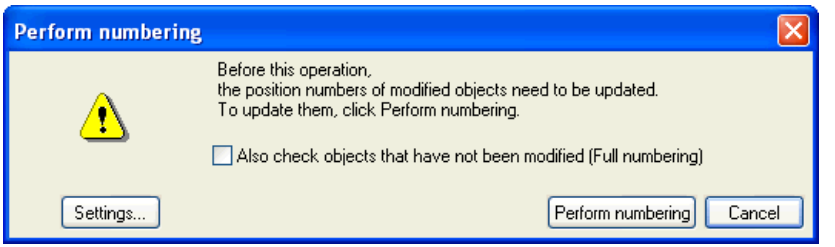
Assign numbers

When you assign numbers by yourself, the start number always must be set to 1. If not, the startnumber and the position number you assign will be added up, decreased by 1, see the following example:



Update numbering

When the numbering is not up to date, Tekla Structures displays the following warning:



In the dialogbox, click **Preform numbering** to update the numbering and to create a drawing, for instance. The command **Preform numbering** is equal to the command **Numbering modified**.

New copy and move options for drawing objects

The **Translate**, **Rotate**, and **Mirror** options have been added to the **Copy** and **Move** commands in the Drawing Editor:

This goes for dimensions, texts, welds, graphic drawing objects and dwg/dxf files.

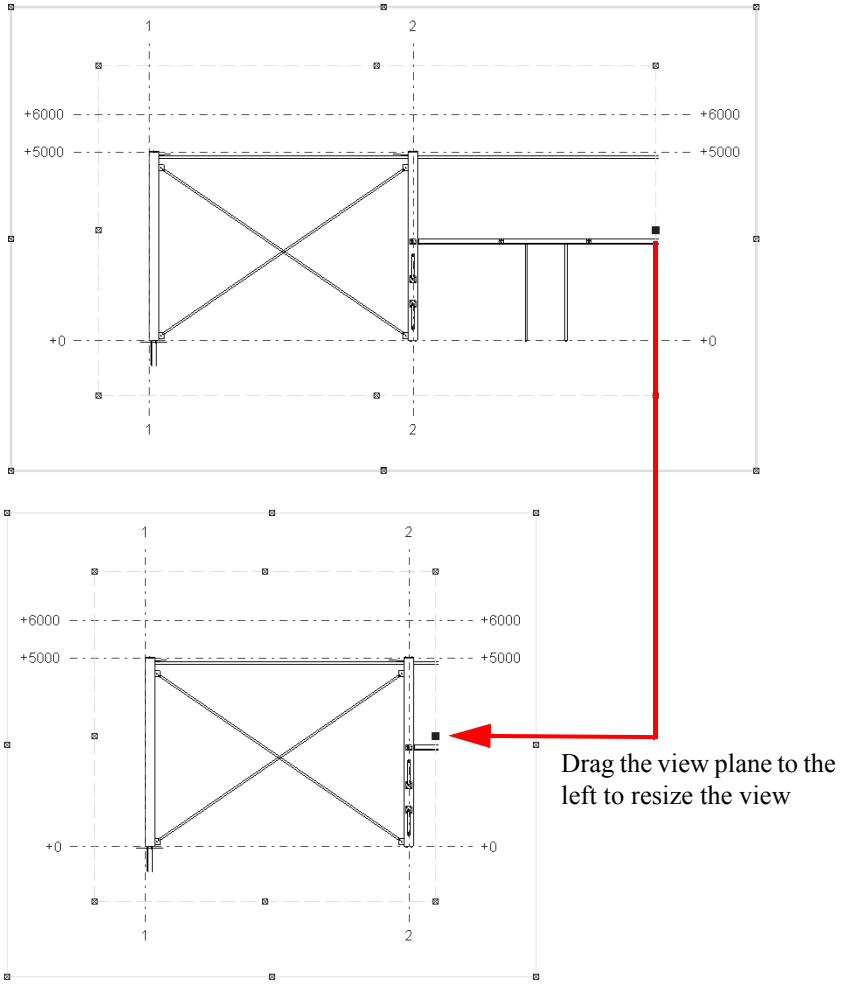
Resizing drawing views

To resize a drawing view in earlier versions, you needed to change the drawing view boundaries (**extrema**) by changing the **x min**, **x max**, **y min**, and **y max** properties in the **View properties** dialog box.



From Tekla Structures 11.2 you can select the view and drag the grips of the view plane, indicated by dashed lines, along the x and y axes of the view:

Select the view plane:



Hatches

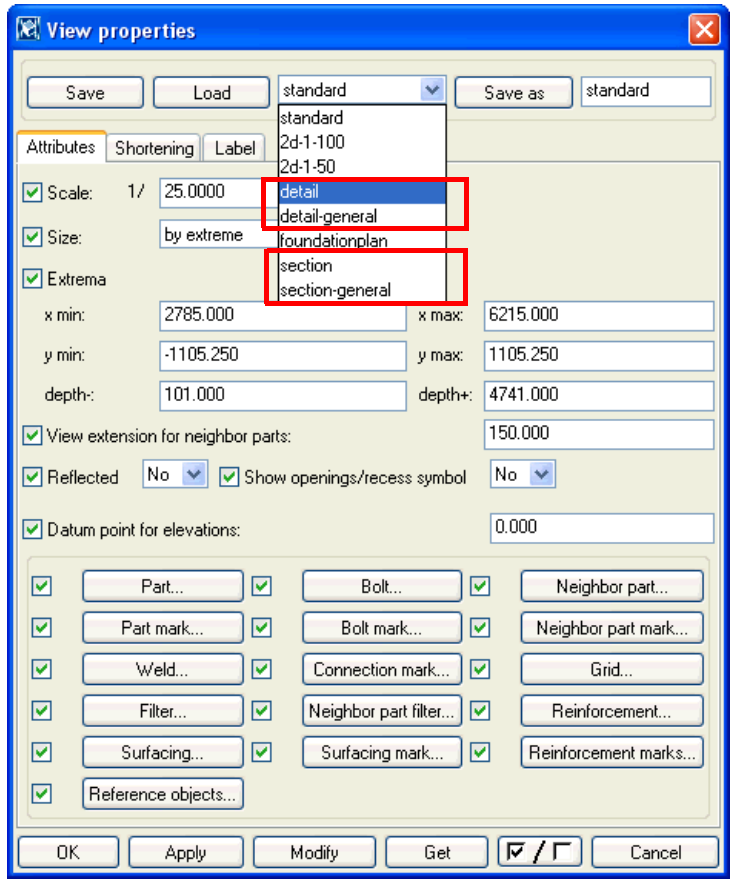
Three new patterns for hatching are added to Tekla Structures:

- HARD_INS1
- SOFT_INS
- SOFT_INS2

For more information about hatches, see Release Notes 11.2.

New view settings

Some new view settings have been added to the Drawing Editor:



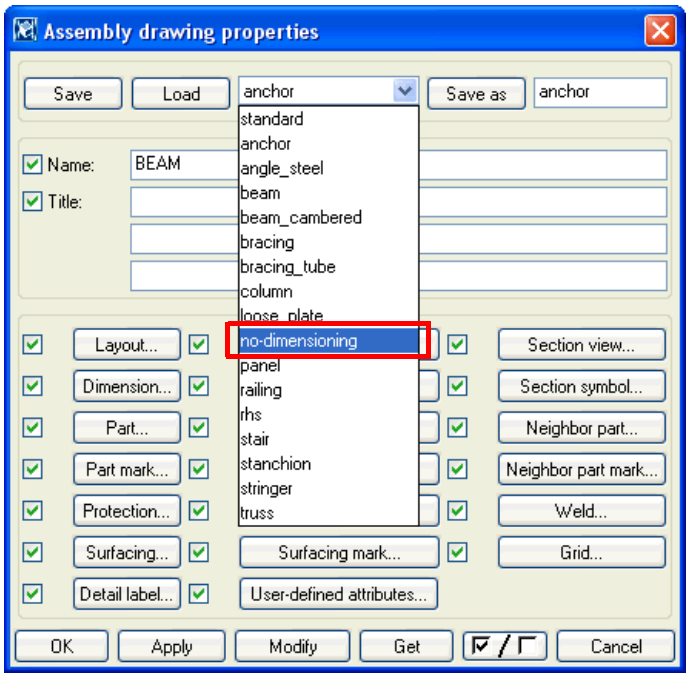
- ***detail***, this setting can be used for creating details on workshop drawings. The parts are displayed in a yellow color.
- ***detail-general***, this setting can be used for creating details on general drawings. The parts are displayed in a white color.
- ***section***, this setting can be used for creating section views on workshop drawings. The parts are displayed in a yellow color.
- ***section-general***, this setting can be used for creating details on general drawings. The parts are displayed in a white color.

New setting for assembly drawings

A new setting "**no-dimensioning**" has been added for assembly drawings.

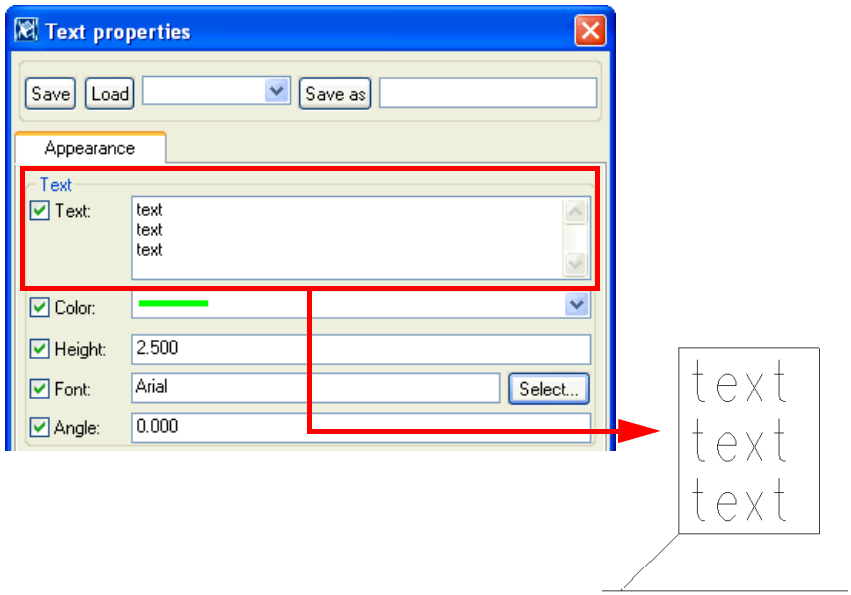
You can use this setting for lattice trusses, trusses and complex frames. This setting is also suitable for stairs and handrails.

The main advantage of this setting is that you do not have to delete the dimensions manually and generating and opening an assembly drawing is much faster.



Multiple lines of text

The **Text properties** dialog box has been improved, you can now enter single or multiple lines of text:





Settings, created in this dialogbox, are stored as **.drtxt* files, not as *.txt files.

So when you copy setting from an earlier version you must rename them!

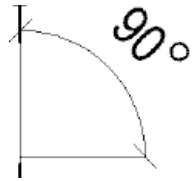
Angle dimension symbol

Use the variable `XS_ANGLE_DIMENSION_SYMBOL_SIZE_FACTOR` to define the size of angle symbols:

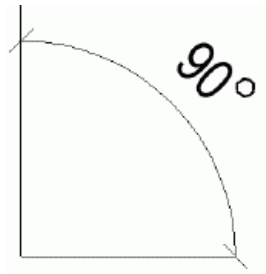
Set to **`1.0`**



Set to **`3.0`** (standard)



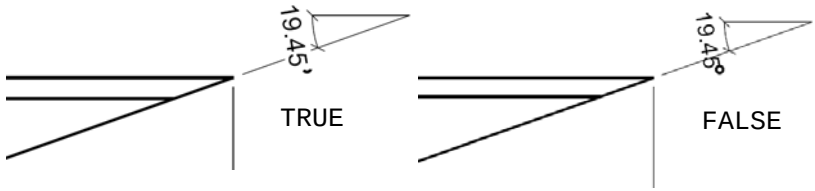
Set to **`5.0`**



Plotting continuous lines

To display continuous lines and symbols in drawings you plot, set the following variables off:

- `XS_USE_LINECLIP_IN_EXPORT` for DXF / DWG export.
- `XS_USE_LINECLIP` for plotted drawings.

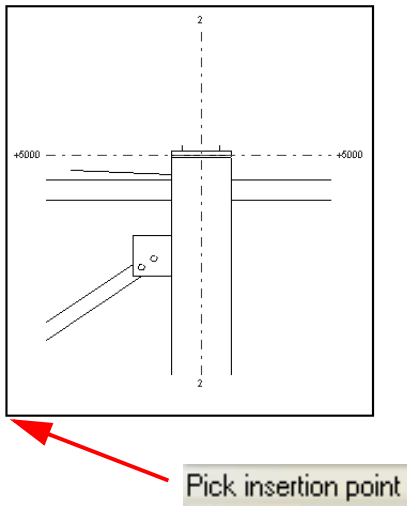


These variables are standard set to FALSE.

Create detail- and section views

When you create a detail- or a section view, you can now place it anywhere in the drawing. To create such view in a drawing:

- You pick the first and the second point of the cut plane
- You now pick a **third** point, a so-called insertion point, to indicate the location of the view in the drawing, this point is the bottom left corner of the view.



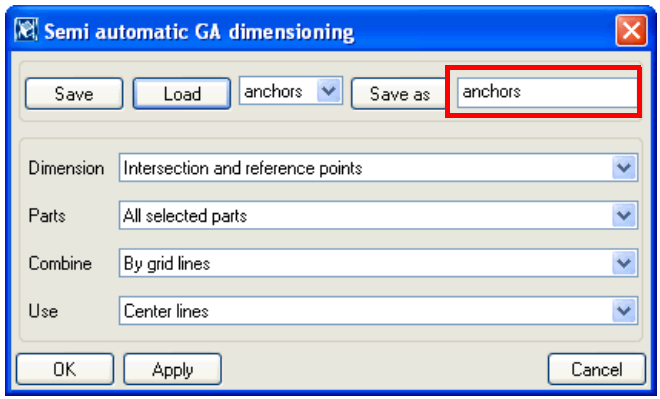
Previously when you created such view, it was automatically placed in the bottom left corner of the drawing.

Move drawing views

Moving drawing views is eassier when you temporary change the drawings'color to black and white. Go to ***Setup > Black and white drawing*** or make use of the shortcut "***B***".

GA dimensioning

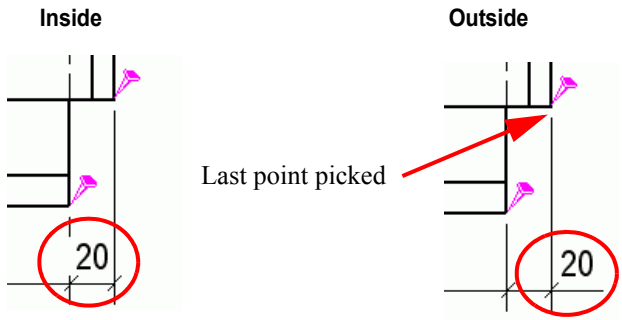
Options to **Save**, **Load** and **Save as** are added to the **Semi automatic GA dimensioning** dialog box**:**



Because of this you can use default settings for semi automatic dimensioning, e.g. anchors, so that dimensioning general arrangement drawings is much faster.

Short dimensions

When you select **Outside** as the **Short dimensions** type in the dimension properties dialog box, you can now specify on which side the dimension must be created. Now Tekla Structures creates the dimension on the same side as the last point picked:



Changing the side of the dimension afterwards is not possible!

Plot drawing to a file (PDF)

When you plot drawings to pdf, you have the possibility to add, among other things, the date and the time to the pdfs' name. Because of this you can always check when the drawings were plotted.

Copy the following line(s) to the file ***user.bat*** and add the variable ***%%UDA:DATE%%*** and/or ***%%UDA:TIME%%***:

set XS_DRAWING_PLOT_FILE_NAME_A=A.....

set XS_DRAWING_PLOT_FILE_NAME_G=G.....

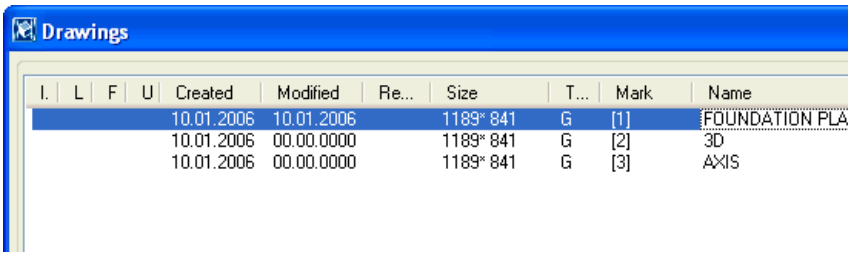
set XS_DRAWING_PLOT_FILE_NAME_M=M.....

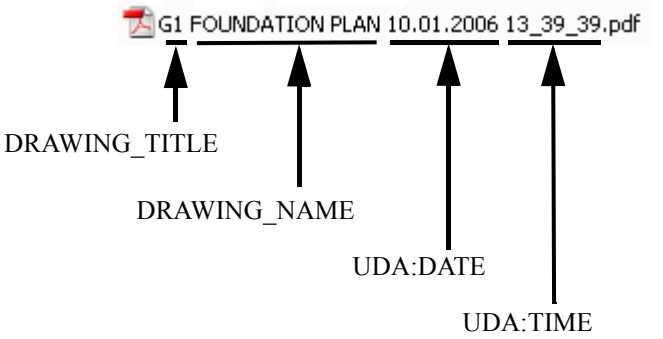
set XS_DRAWING_PLOT_FILE_NAME_W=W.....

set XS_DRAWING_PLOT_FILE_NAME_C=C.....

In the following example, the line in the file ***user.bat*** is set as follows :

set XS_DRAWING_PLOT_FILE_NAME_G=G%%DRAWING_NAME%% %%DRAWING_TITLE%% %%UDA:DATE%% %%UDA:TIME%%





Associative drawings

All drawing types in Tekla Structures are associative, whether created manually or automatically.

This means that drawing objects are linked to model objects, so that most drawing objects automatically update if the model changes, but you do not lose manual changes made to drawings.



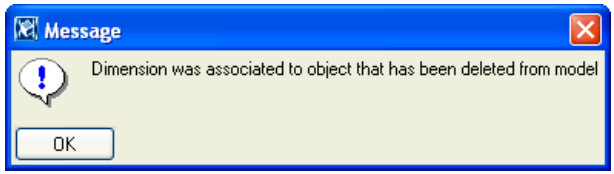
To keep manual changes made to single-part, assembly, and cast-unit drawings, ***freeze*** the drawings. It is not necessary to freeze general arrangement or multidrawings, they are always associative.

To switch off the associativity, with which you provide Tekla Structures from automatically updating drawings when the model changes, adjust the following variable:

rem set XS_INTELLIGENT_DRAWING_ALLOWED=TRUE

Message

In Tekla Structures 11.x the following message can appear:



This message appears because the ID-number has changed or the ID does not exist anymore. All objects in Tekla Structures have an ID-number, to wich the associative dimensioning is related.

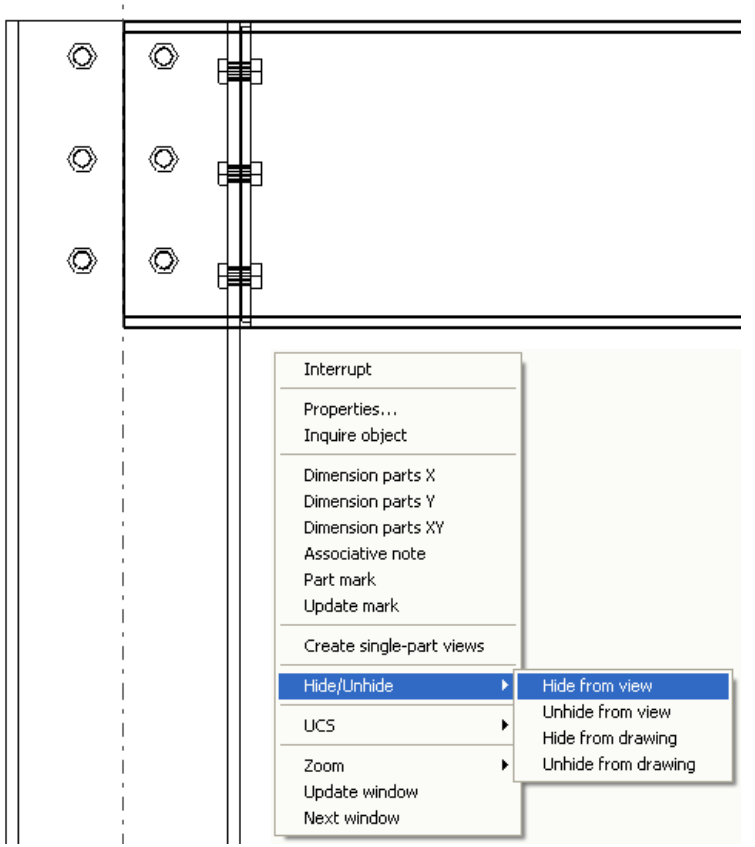
When an object is deleted from a model and a drawing dimension is (was) related to that object, and you then open a drawing, this message will appear.

Possible dimension lines who were related to the involved object are deleted automatically because the part does not exist anymore.

Tekla Structures 11.3

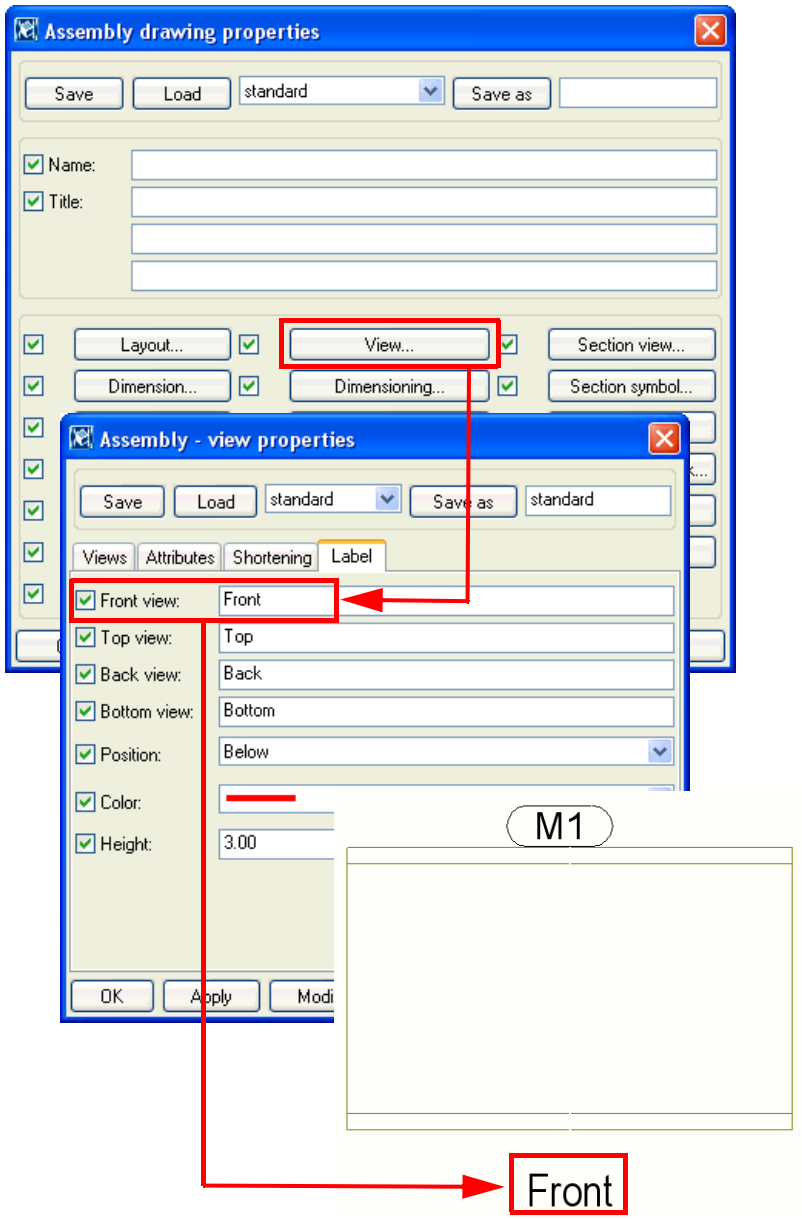
Hide and unhide objects in drawings

Now you can hide and unhide objects in drawing views. With the hide/unhide feature the editing of drawings is easier and faster.



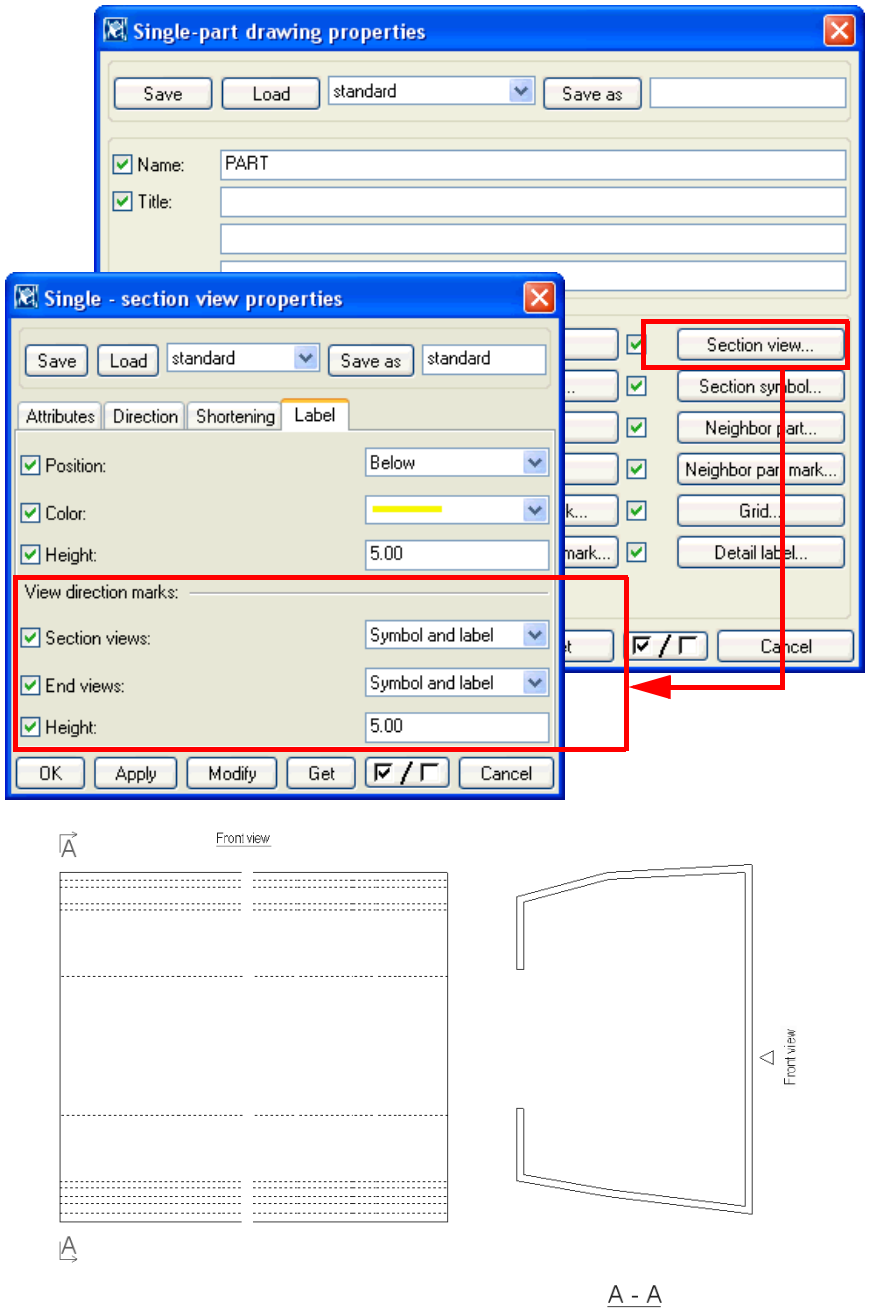
View name labels in assembly and single-part drawings

You can define the view names for front, top, back and bottom views in the drawing properties dialog.



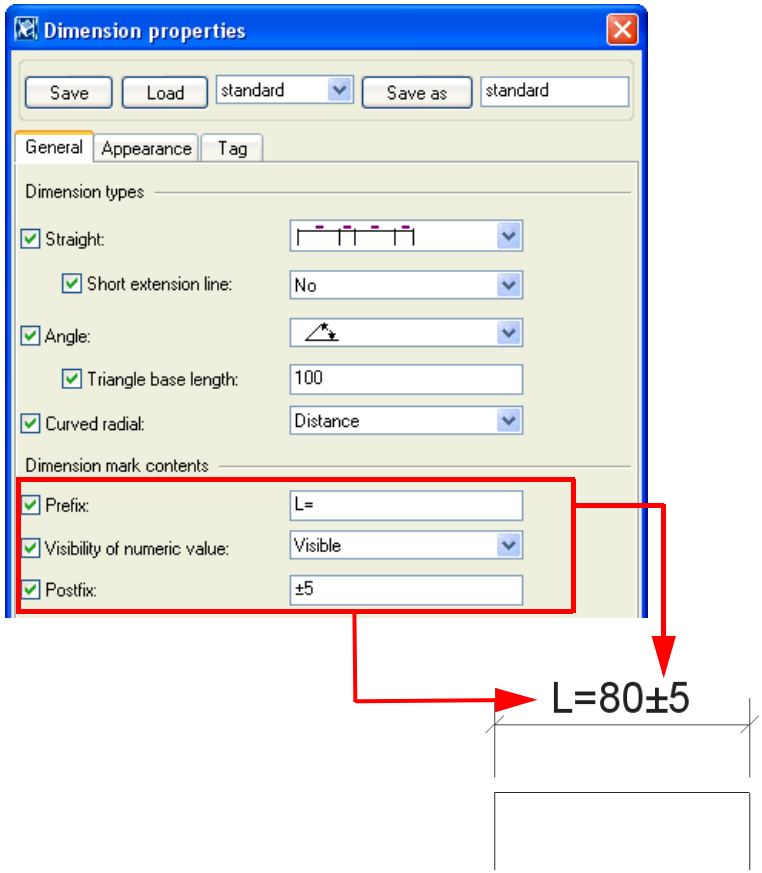
View orientation symbols

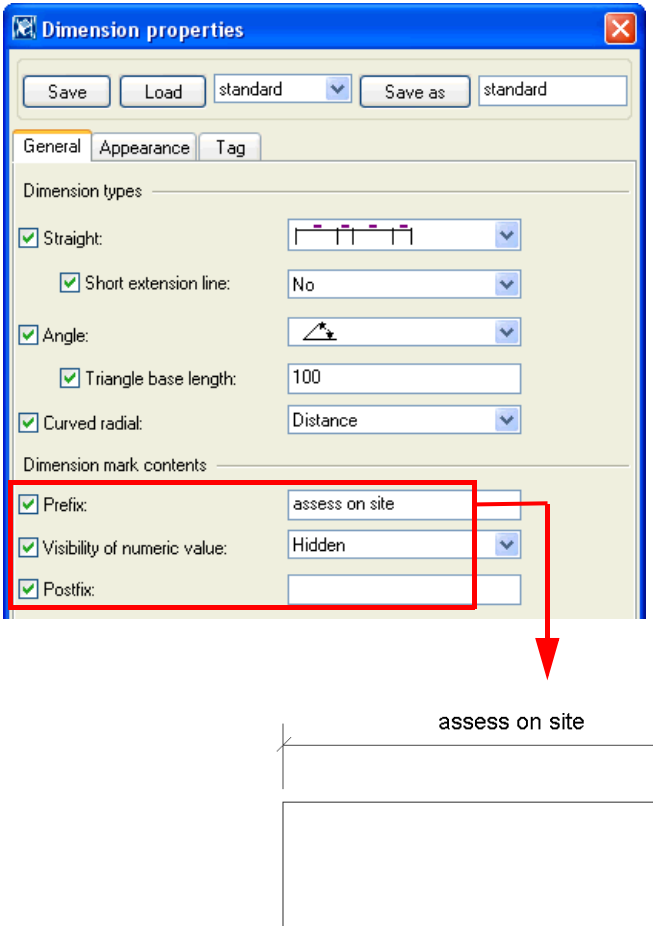
View orientation symbols can now be displayed in section views. This clarifies the orientation of views and section views.



Add prefix and postfix text to dimension line text

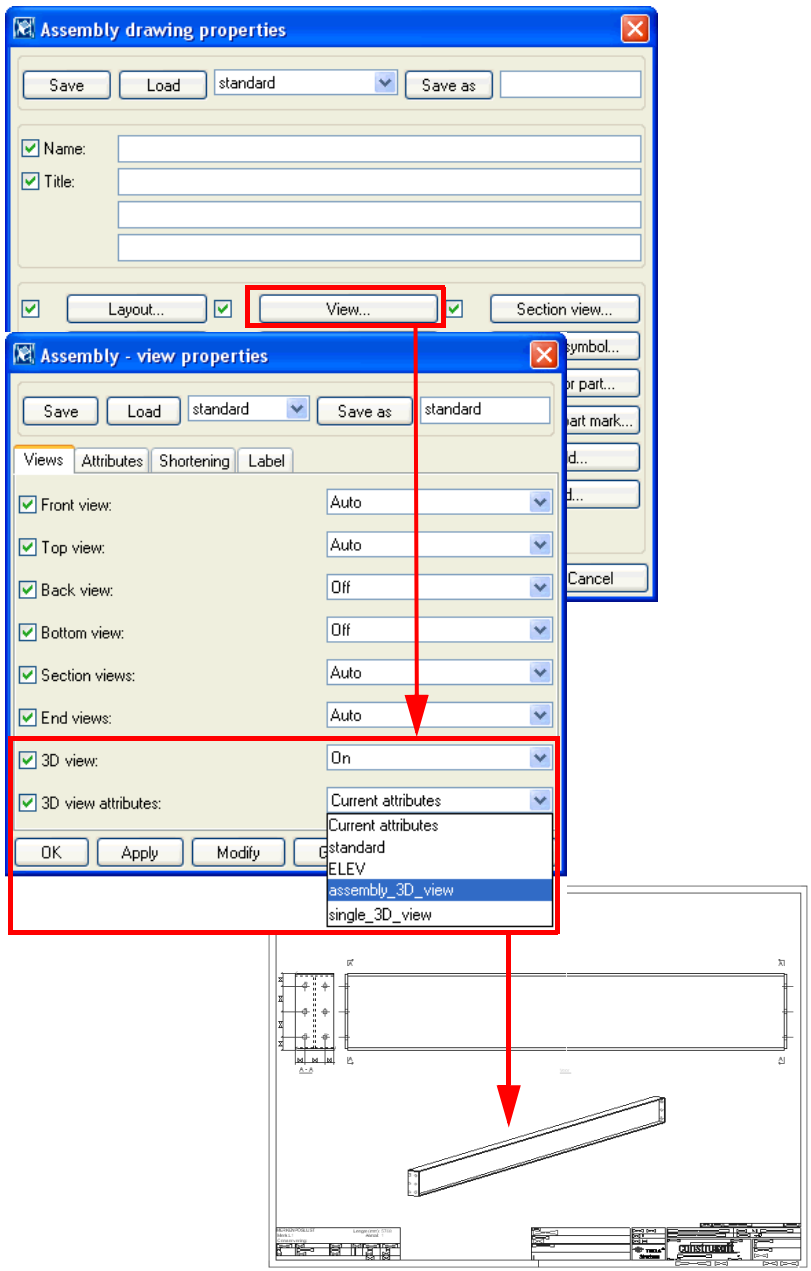
Adding prefix and/or postfix text to dimension line text is easy with Tekla Structures 11.3. Also later modifications to such dimension lines are faster. You can select Visible or Hidden, see following examples:




Dimension properties
Save
Load
standard
Save as
standard
General
Appearance
Tag
Dimension types
Straight:
Short extension line:
No
Angle:
Triangle base length:
100
Curved radial:
Distance
Dimension mark contents
Prefix:
assess on site
Visibility of numeric value:
Hidden
Postfix:
assess on site

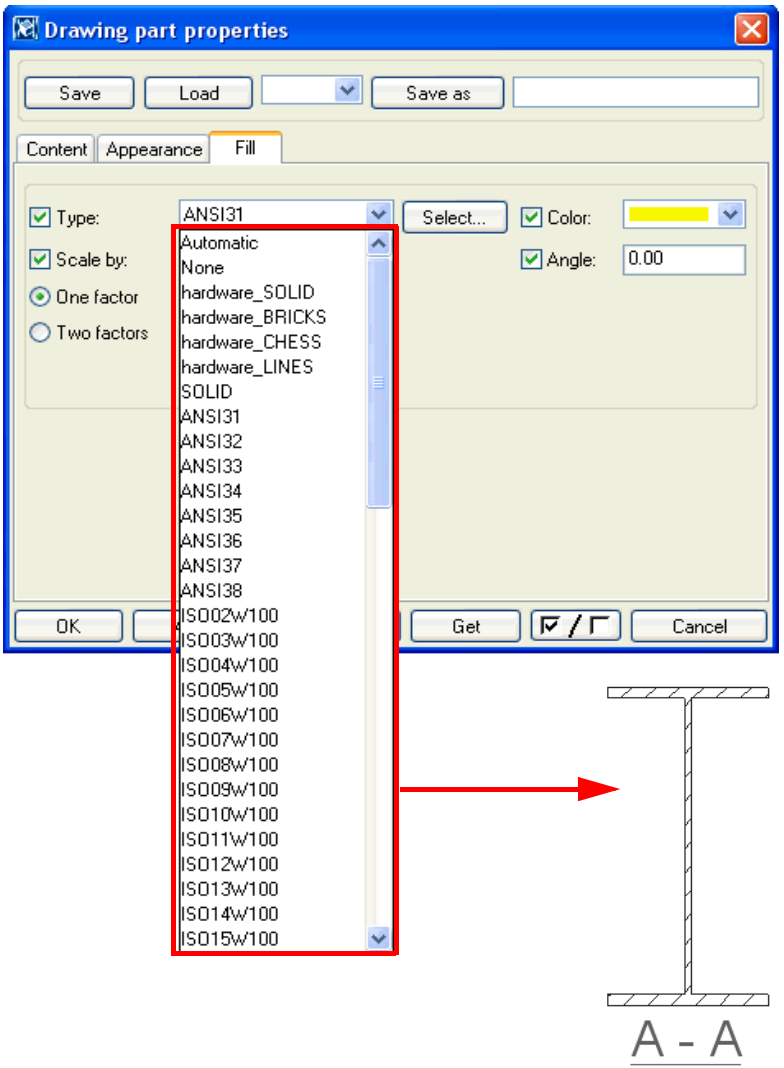
3D views in assembly and single part drawings

It is possible to create 3D views in assembly and single part drawings. You can create 3D views interactively also after the drawing has been created. This feature saves you from a lot of manual editing work: you can get the 3D view automatically into fabrication drawings.



Section view hatching

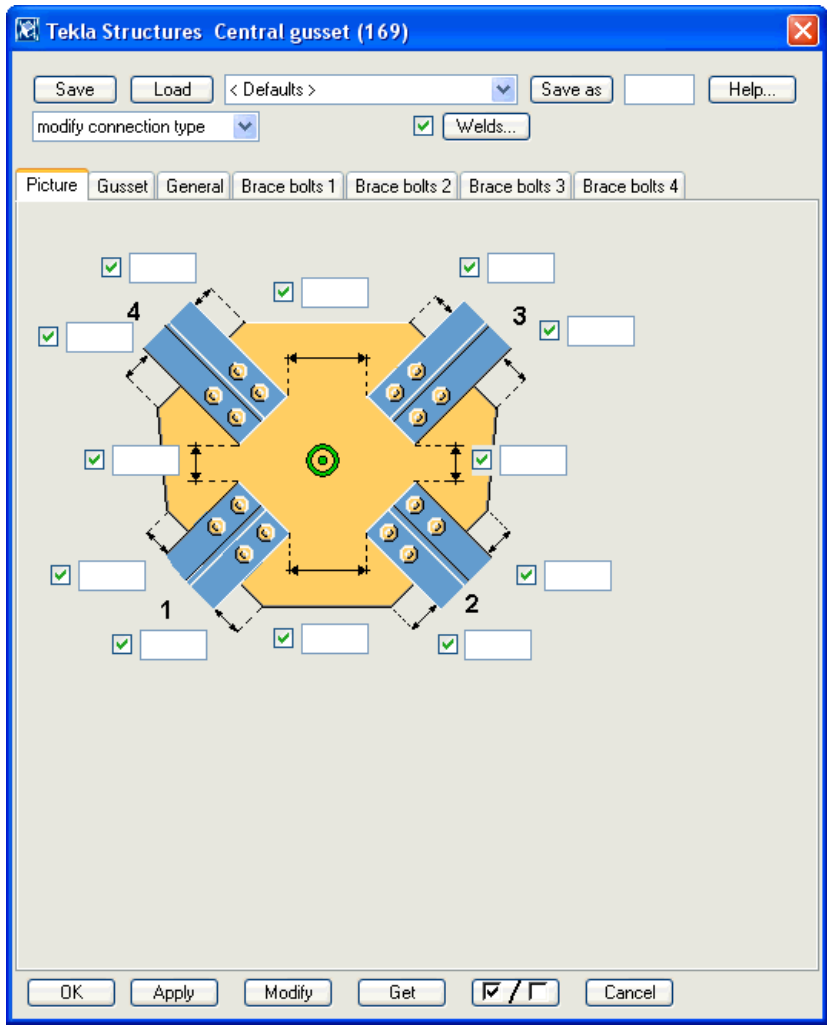
You can define different hatching properties for parts in section views.



Per profile you can edit the current hatching, switch it off or add one per part.

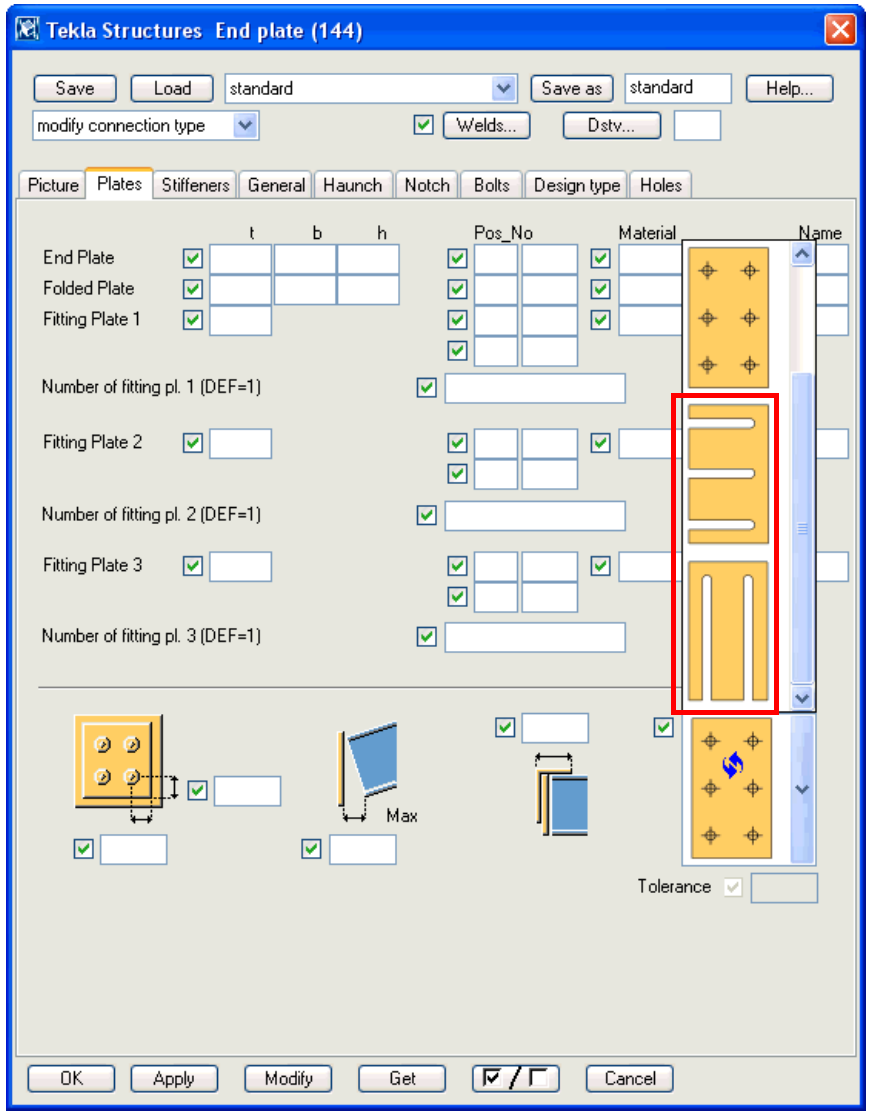
Central gusset (169)

This new connection **Central gusset (169)** connects for windbraces. A gusset is created and welded or bolted to the windbraces.



Comb filler plates for Two sided end plate (142) and End plate (144)

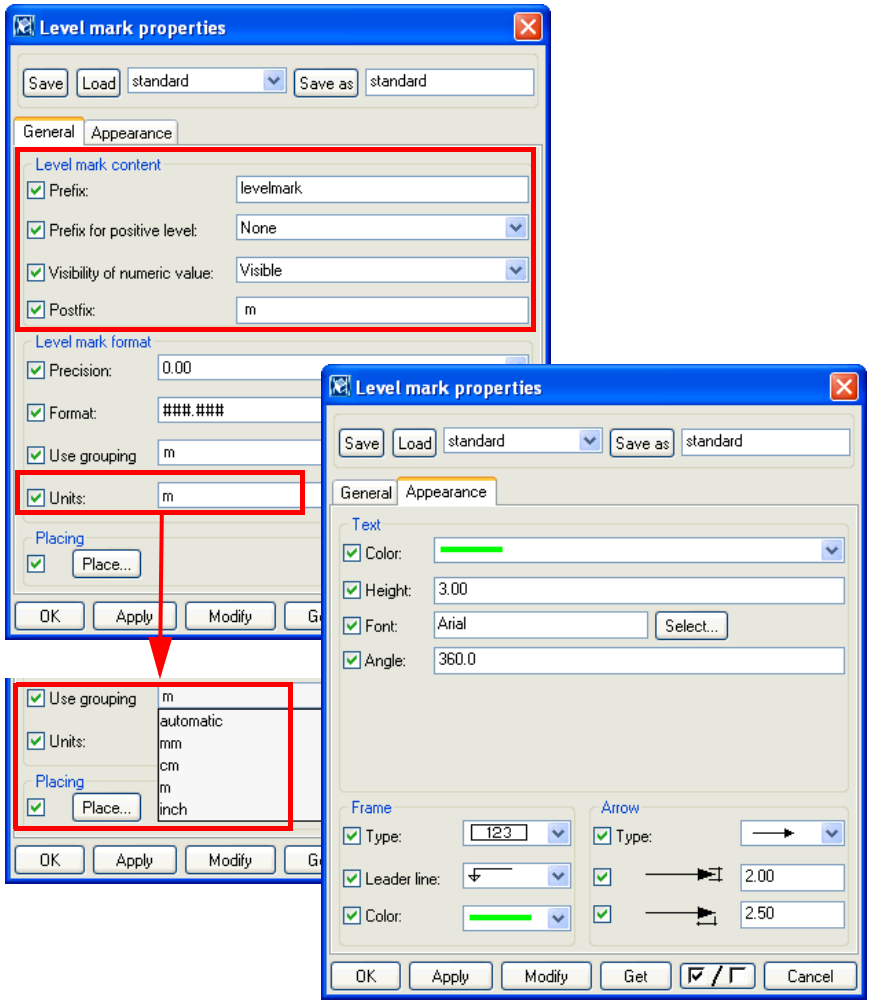
You can now create a connection with a comb filler plate using **Two sided end plate (142)** and **End plate (144)** connections:

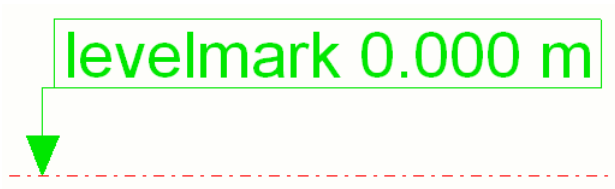


Tekla Structures 12.0

Level marks

In Tekla Structures 12.0 you have more options for level marks such as prefixes and postfixes and units.





7 Construsoft Newsletter TS 11.3

Topics

This newsletter about Tekla Structures 11.3 contains information about the following subjects:

General:

- **Locked file is deleted after "Saved as..."** (page 3)
- **Solid buffer memory increased** (page 3)
- **Netviewer standard installed** (page 4)
- **A new shortcut to Tekla Structures.pdf** (page 4)

Modeling:

- **Reference line of profiles visible in model views** (page 5)
- **Copy weld symbol** (page 5)
- **STACO steps added** (page 6)
- **Ajusted parametric profile shapes for concrete panels** (page 6)
- **Structo precast slab added** (page 7)
- **Connecting bracing crosses** (page 8)
- **Improved snap function in rotated rendered axis views** (page 12)
- **Adjust automatically work area by copying over large distances** (page 12)
- **Axis views at an angle: improved display** (page 13)
- **Locking objects** (page 14)
- **Adjusting DWG object rendering** (page 16)
- **Construsoft component Tensioner brace (13): new options on behalf of positioning** (page 17)
- **Construsoft component Rectangle to - Circle (17)** (page 19)
- **Construsoft component 3D Cut (10)** (page 21)

Drawings:

- **3D view on A- en W-drawings** (page 22)
- **Creation of unfolded views in W-drawings possibly disturbed by variabele** (page 23)
- **New symbol file "North.sym"** (page 23)

Reports

- **Reports in Tekla Structures 11.3** (page 24)
- **Using 'old' Reports and Templates in Tekla Structures 11.3** (page 24)
- **Delete brackets from drawing name on drawings** (page 25)
- **Double-clicking a template: No-option added** (page 28)

NC data

- **Hard stamp in NC files of secondary parts** (page 29)
- **Add popmarks to NC files > Numbering necessary** (page 29)

Tekla Structures 12.0

- **Levelmarks** (page 31)
- **Sub-assemblies** (page 32)
- **Anchorboltplan using 2 scales** (page 33)
- **Gridtext always visible** (page 34)
- **Highlighting changes in associative dimensionlines** (page 35)
- **Rollover highlight** (page 35)
- **Object representation: transparency** (page 36)
- **Custom components available in all models** (page 36)
- **Construsoft component Baseplate (1029)** (page 37)
- **Tekla component Joining plates (14)** (page 38)
- **Tekla component Stairs (S71)** (page 38)

RAM	XS_SOLID_BUFFER_SIZE
512 Mb	225000
768 Mb	330000
1024 Mb	**450000**
1536 Mb	675000
2048 Mb	900000
3072 Mb	1350000

Netviewer standard installed

The program Netviewer is used by Construsoft to look at your your computer or -when necessary- to take over your computer in order to give directions or make changes. This enables the Construsoft Helpdesk to give you the best possible support. The technology used by Netviewer is based on a 128-bit encryption so it is impossible for others to gain access to your computer.

Netviewer is automatically copied into the system folder of Tekla Structures 11.3.

In the system folder of Tekla Structures 11.3 you find the file **netviewerK.exe**.

In case you have not placed Netviewer on your computer, we advise you to create a shortcut on your desktop so that you can quickly start up this program if this is requested by the Construsoft Helpdesk.

To start a new Netviewer session you have to fill in a session number. This number will be given by the Construsoft Helpdesk by phone.

A new shortcut to Tekla Structures.pdf

During the installation of Tekla Structures 11.3 a shortcut is automatically placed on your desktop to the file **Tekla Structures.pdf**.

This file was previously known as **Standard_documents.pdf**.

This document contains several subjects of FAQs.

The location of this file is:

disk:\TeklaStructures\11.3\environments\europe\pdf.

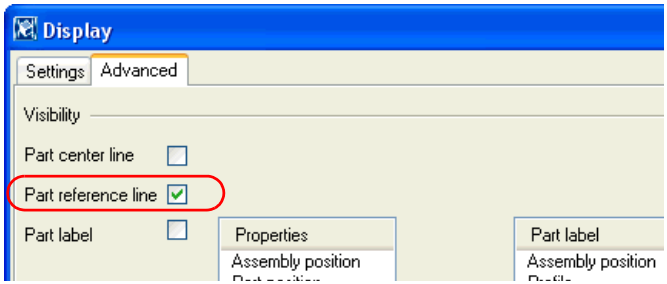
Modeling

Reference line of profiles visible in model views

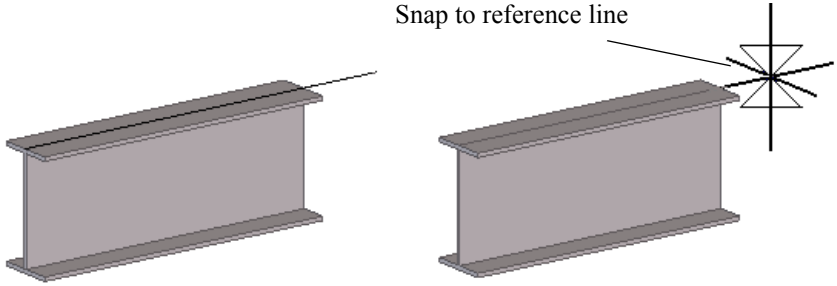
In Tekla Structures 11.3 reference lines of profiles in model views can be made visible. You can also snap to the part reference line.

Show reference lines

Open te properties of the view > Display > tab Advanced > check "Part reference line".



Snap to reference line



Copy weld symbol

In previous versions the weld symbol was not copied with the profile correctly when the ’target’-profile was not identical to the ’source-profile. (when the two profiles were identical there was no problem).

For example: a stub, welded to mainpart IPE300. The stub has to be copied to another part, HEA200. In previous versions the weld symbol, even when selected, was not copied to the next part. The stub had to be welded manualy to the ’target’-part after finishing the copy-comand.

In Tekla Structures 11.3 this is solved and welds are copied from ’source’ to ’target’ profile regardless the size or shape of the profiles.

STACO steps added

Three Staco steps are added:

Tekla Structures Profile name	Staco description
240_25	SP 600*240 carry bar 25*3
270_30	SP 800*270 carry bar 30*3
270_40	SP 1000*270 carry bar 40*4

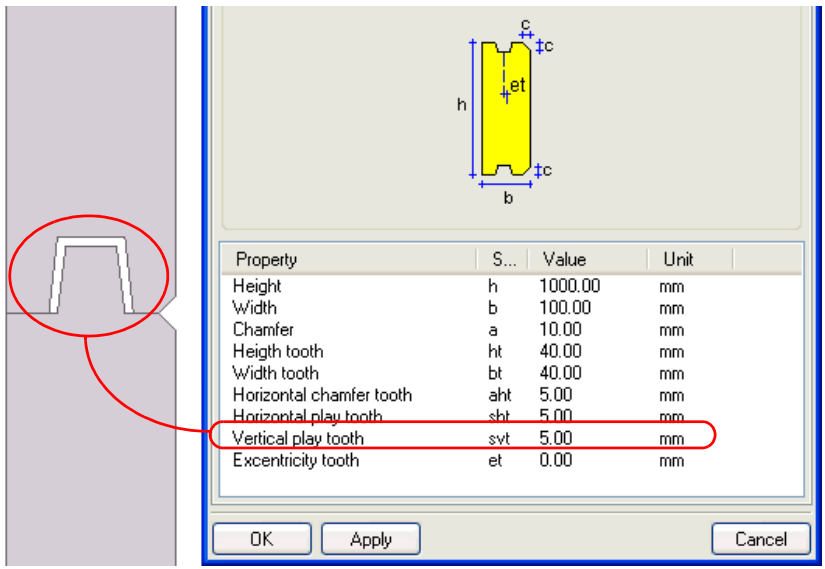
Ajusted parametric profile shapes for concrete panels

Some concrete panels had a shortcoming: the vertical tooth play could not be set. This ment that no play was made between the panels.

In Tekla Structures 11.3 this vertical play can be set.

In the folowing concrete panels is the vertical play added:

- CS_panel.asymm_t_h*b-a-ht*bt-[aht-[sht*[svt*[et]]]]
- CS_panel.asymm_t_b_h*b-a-ht*bt-[aht-[sht*[svt*[et]]]]
- CS_panel.asymm_t_g_h*b-a-ht*bt-[aht-[sht*[svt*[et]]]]



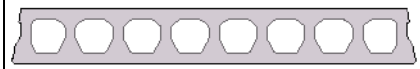
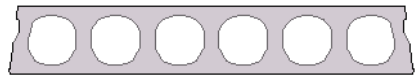
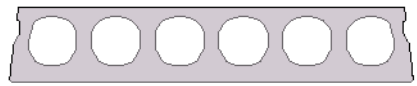
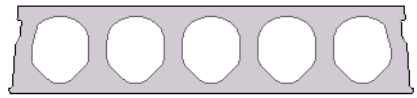
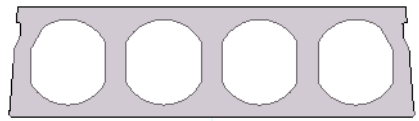
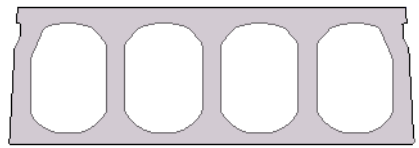
The second last variable is the vertical tooth play.

Structo precast slab added

Precast slab profiles of Structo are added in Tekla Structures 11.3.

The profiles are not avalible directly. A *.lis file has to be imported into the profile database. You can find the *.lis file at:

disk:\TeklaStructures\11.3\environments\europe\profil\lis\NLD\profiles

Tekla Structures Profile name	Structo name	Example
STRUCTO_130	EC/130/1200	
STRUCTO_160	DC/160/1200	
STRUCTO_200	DC/200/1200	
STRUCTO_220	DC/220/1200	
STRUCTO_260	DC/260/1200	
STRUCTO_320	DC/320/1200	
STRUCTO_400	DC/400/1200	

Connecting bracing crosses

A frequently asked question is how to connect the crossed bracing profiles, while positioned in the same plane.

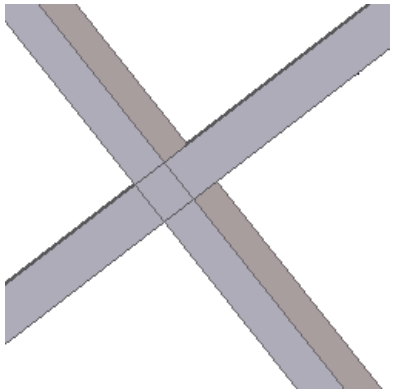
This connection can be made with component 11. This component is primary made to connect bracing profiles to main parts but can also be used to connect crossed bracing profiles.

Start situation

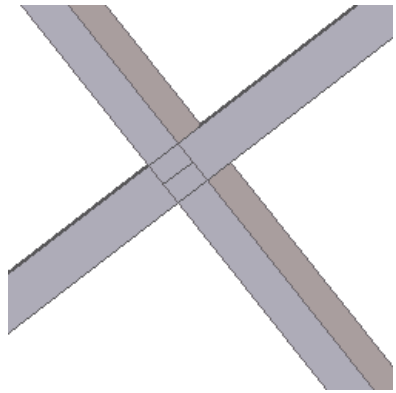
To connect a cross of which the profiles are situated in the same plane, one of the profiles has to be splitted at the crossing. To split the profile:

Edit > Split

Designate one profile and click at the position where the profile has to be split (= crossing of the two profiles).



Original situation

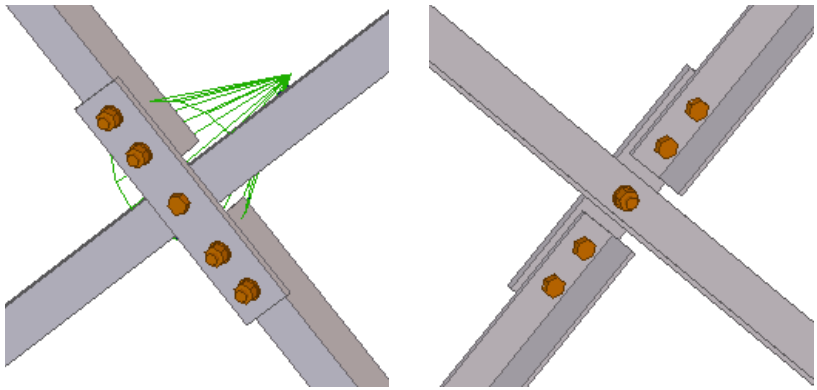


Situation after splitting one of the profiles

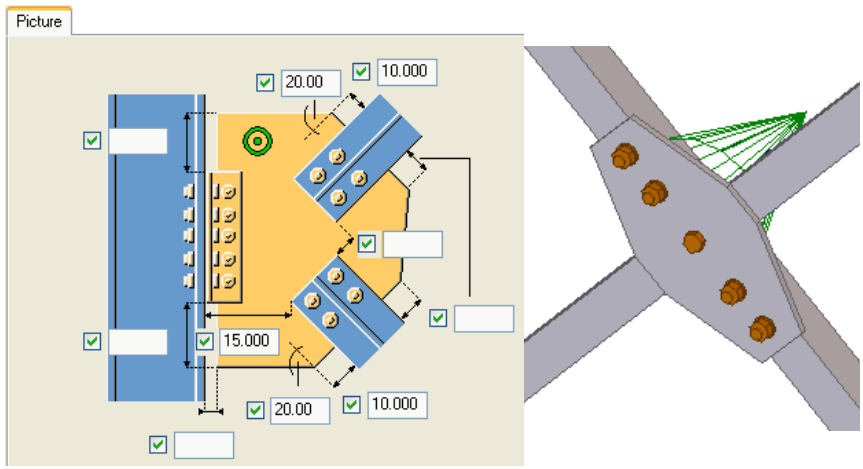
Bolted gusset

In the component Bolted gusset (11) the setting **cross_bolted** is available.

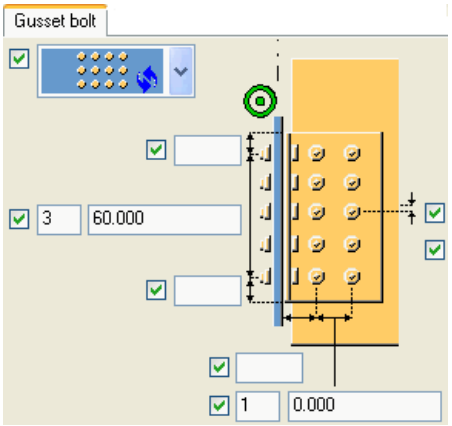
Load this setting and apply the component. First select the through profile and second the two other profiles. Finish the process by clicking the center mouse button. The connection will be made.

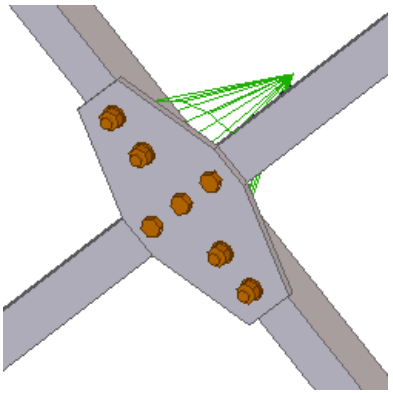


In case you want to adjust the shape of the gusset you can, for example, fill in angles at tab **Picture**.



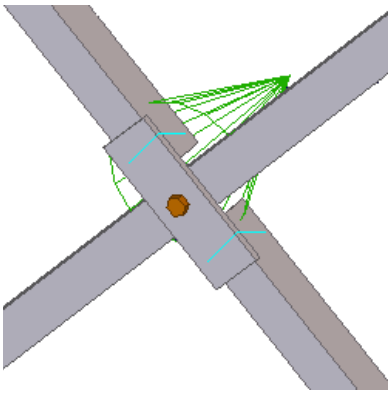
In case you want to use more bolts in the through profile, you can adjust the number of bolt rows at the tab **Gusset bolt**.

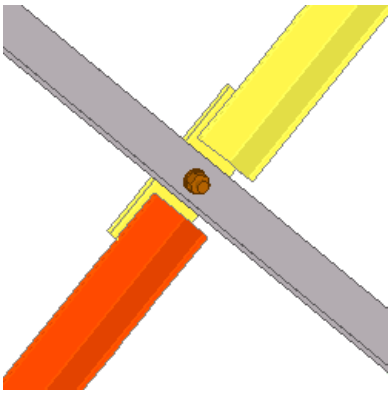




Bolted + welded gusset

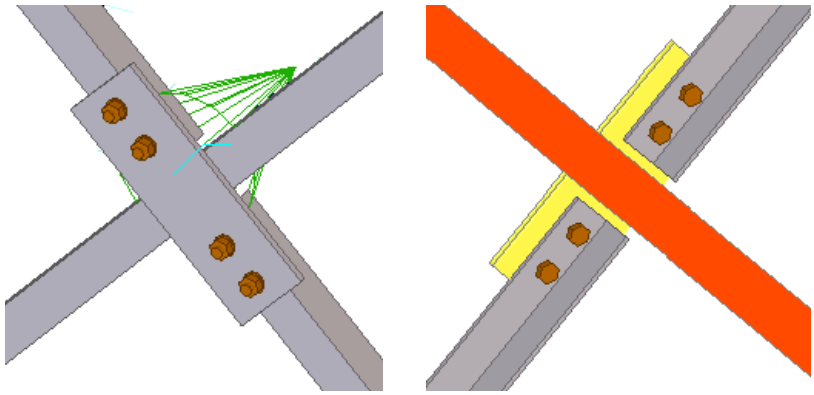
A cross connection by which the gusset is welded to the two split profiles and bolted to the through profile can be made by using the setting: **cross_welded**.





Welded cross

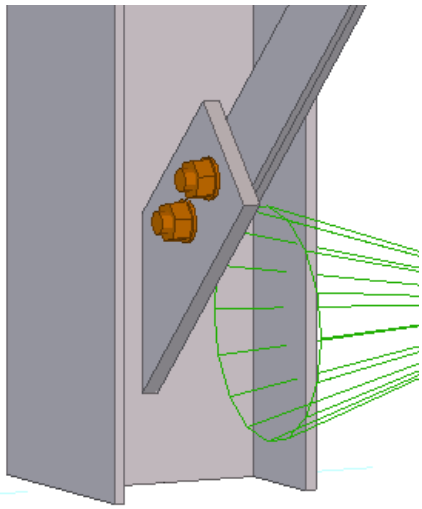
A cross connection by which the gusset is welded to the through profile and is bolted to the spilt profiles can be made using the setting **cross_welded_bolted**.



Flat

This is a setting to connect the bracing profile using a gusset with flat dimensions. Standard two bolts are placed.

The gusset is modelled using the flat dimensions because both angle-fields (tab **Picture**) are set to 0.

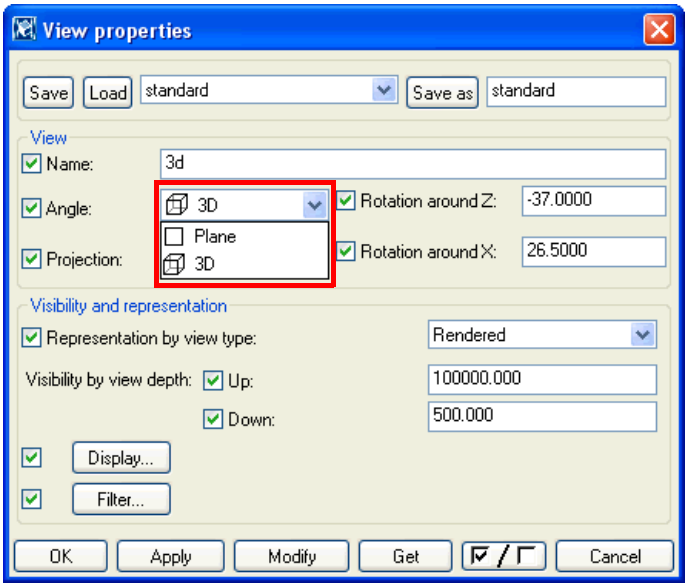


Improved snap function in rotated rendered axis views

In Tekla Structures 11.3 the snap function is improved in rotated rendered axis views.

In previous versions, for example, was snapping to profile edges impossible as soon as a rendered view was rotated.

This was caused by the Angle setting in the view properties: the setting was fixed to **Plane**.

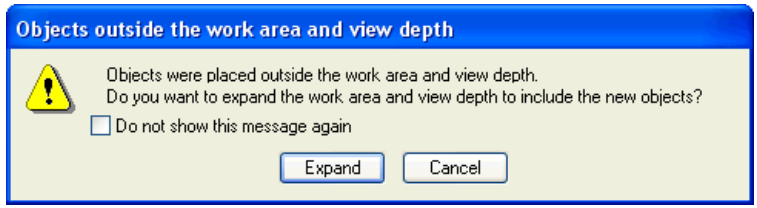


In order to snap, the setting Angle had to be changed to **3D**. To change the axis view back to **Plane**, the setting angle had to be changed again to **Plane** (or shortkey <Ctlr> + P).

In Tekla Structures 11.3 the setting Angle is automatically changed to **3D** when a axis view is rotated. Snapping to profile edges will still be possible. By pressing the shortkey <Ctrl> + P a rotated view can be switched back to **Plane**.

Adjust automatically work area by copying over large distances

In Tekla Structures 11.3 you will be warned when you are copying or moving an object over a very large distance so that the object would be placed outside the set work area and/or view depth. Tekla Structures will give you the option to enlarge the work area so that the copied or moved objects are visible.



Do not show this message again

By checking the box "do not show this message agian", in the future this message wil not be shown again when an object is copied or moved outside te work area.



By future Copy/Move commands the work area automatically will be adjusted so that all objects are visible.



By future Copy/Move commands the object will be placed on its new position. The work area and view depth will not be adjusted when this new position is outside the work area.

Display message again

If you have used the option "do not show this message agian" once, the message will never be shown again.

In order to change te setting you need to let the message appear again. Press <Shift> while activating the Copy-command and the message will appear.

Axis views at an angle improved display

The display of views at an angle are improved.

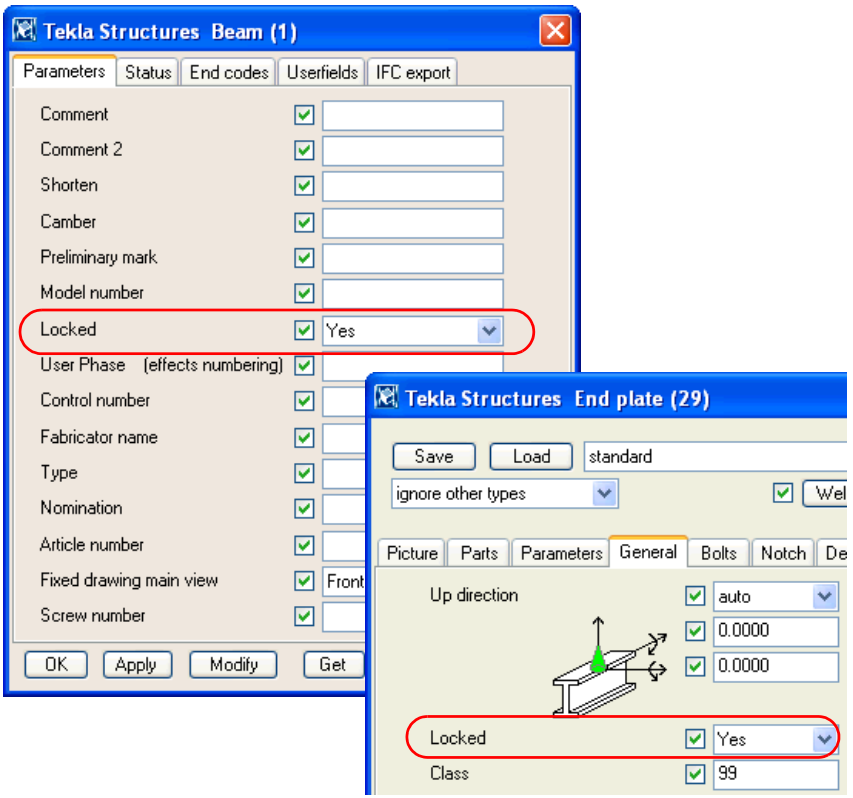
In previous versions it could happen that more objects were visible when axis views were put at an angle (axis views not according the global coordinate system) then should be visible according the set view depth. This could result in wrong selection of objects.

In Tekla Structures 11.3 this is solved: objects outside the set view depth no longer become visible when the axis view is put at an angle. Note: in case a small part of an object is situated in the set view depth, the entire object will be shown. This has not changed.

Locking objects

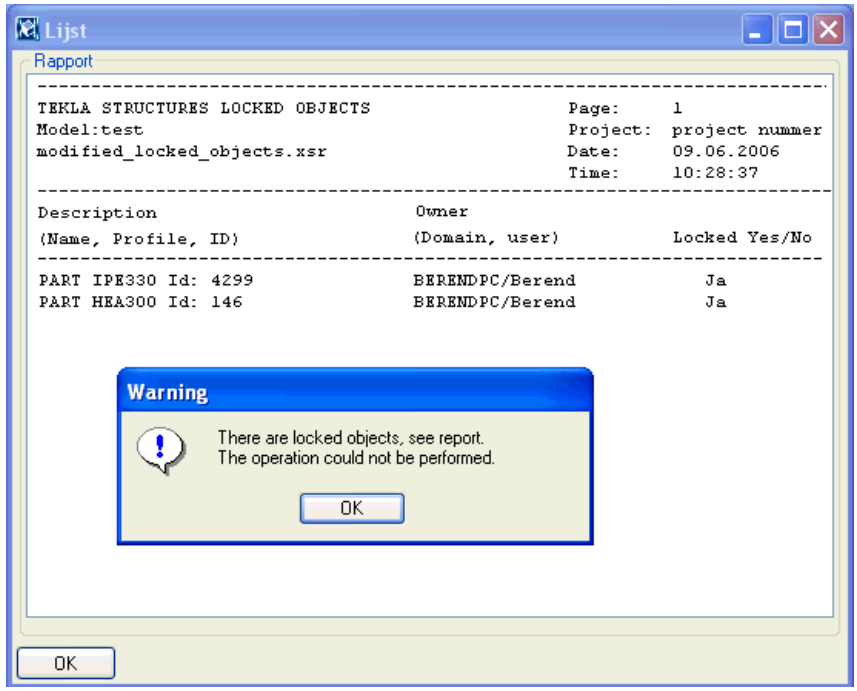
For safety reasons, sometimes you want to protect the model for model changes. This can be done by locking objects.

You can lock profiles by selecting 'Yes' for the 'Locked' option in te **User Defined Attributes** - tab **Parameters**. You can lock components bij selecting 'Yes' for option 'Locked' in the tab **General**.



If you want to modify a locked object (e.g. moving the object or changing parameters) , a message shall appear.

Also an id-list will be displayed which lists the locked objects. By selecting a line in this list, the accompanying locked object highlights in the model.



To find out which profiles are locked, you can generate a report named **locked_objects_report**.

Examples of locking:

A beam is connected to a column using component Endplate (29).

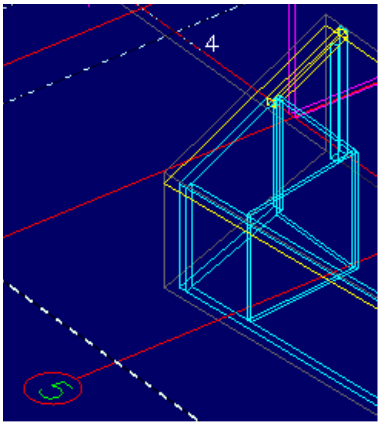
If only the beam is locked:

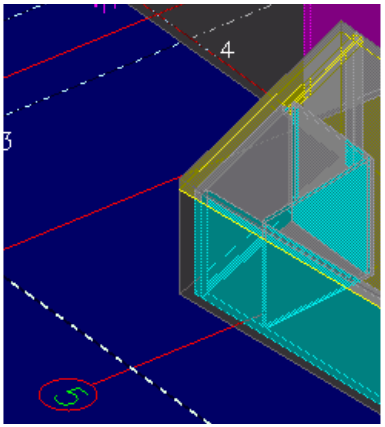
- the beam can not be moved/ modified.
- the Endplate (29) component can not be modified, because the secondary part (the beam) is locked.
- the column can not be moved/ modified, because the column is connected to the locked beam by component Endplate (29).

However, a possible Detail component, for example Stub (1011), on a locked beam can be moved/ modified.

Adjusting DWG object rendering

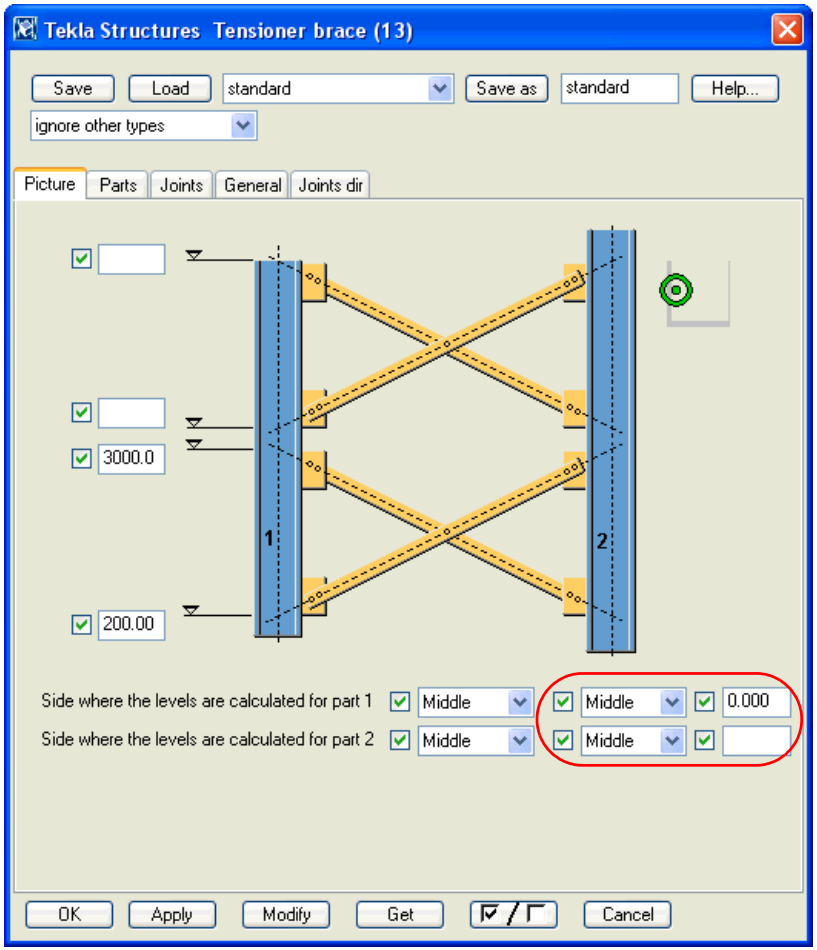
Shortkeys <Shift> + 1through 5 are used to adjust the rendering of component-parts. This shortkey now also adjusts the rendering of Reference objects.



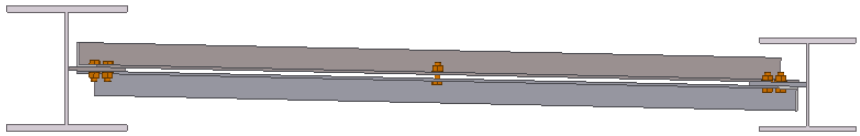


Construsoft component Tensioner brace (13): new options on behalf of positioning

A couple of options are added to position the bracing profiles. This reason why is that it is easier now to position the bracing profiles in case the columns are not of the same size.

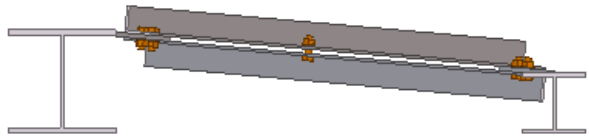
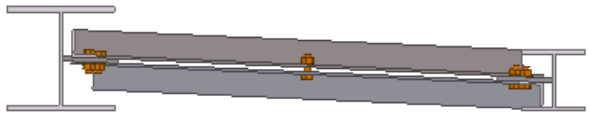
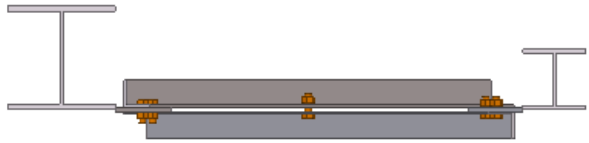
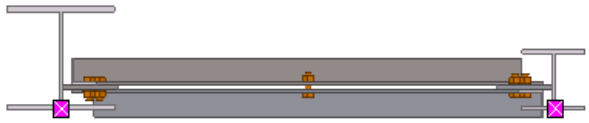


In previous versions Tekla Structures this component lacked the new position options. By unequal main parts the bracing profiles were always placed in the middle of the columns. Example:



Positioning in Tekla Structures 11.2

New positioning options in Tekla Structures 11.3:

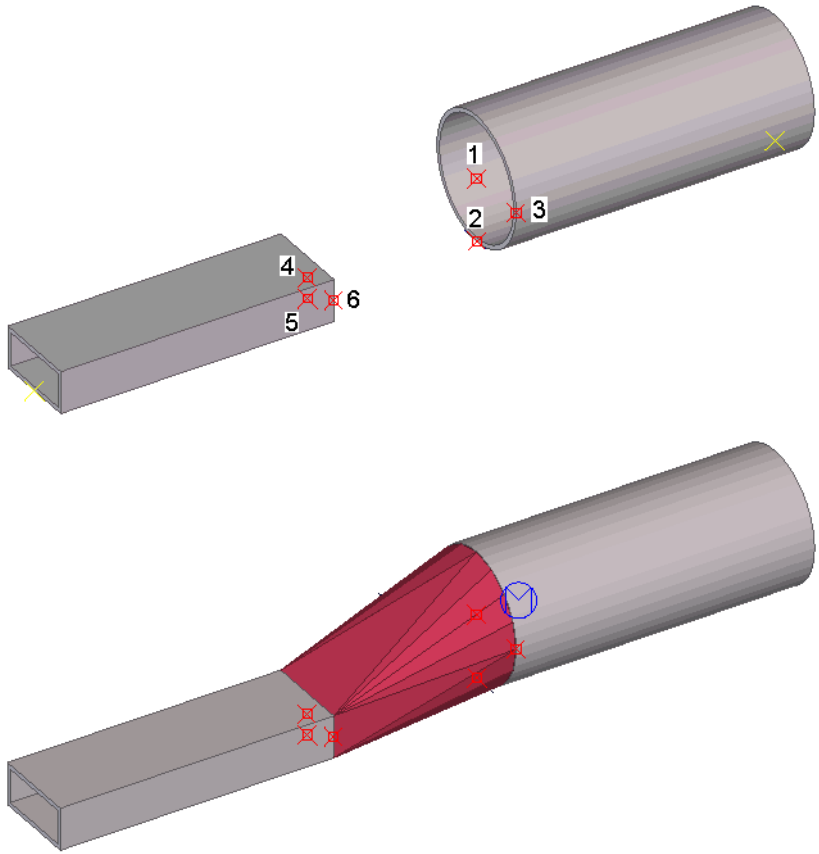
Option	Example
Up	
Middele	
Down	
Reference plane (+50mm offset)	

Construsoft component Rectangle to - Circle (17)

This component is added as a replacement for the Generation Conus (18). The improvements are especially in the applicability of the component. The old component could only connect rectangle and round profiles, the new component can also connect ellipse and multi angle profiles.

There are two ways to connect the profiles.

- select both profiles > Middle mouse button
- select 6 points: three points at the end of the first profile and three points at the start of the second profile > Middle mouse button



In case the connection is made by selecting the profiles, the reducing piece will autmatically adjust when the profiles are changed (when the connection is made by defining the 6 points the reducing piece will stay the same).

Tab Picture

At the tab **Picture** you can define the number of cuts in the reducing piece. It is only possible to use this option when the parts are welded (tab **Parameters**).

Tab Parts

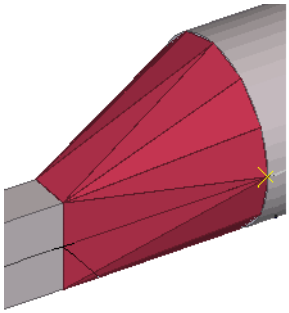
At this tab you can define the thickness of the profile used by the reducing piece. You can also set the position (Middle, Front, Behind).

Tab Parameters

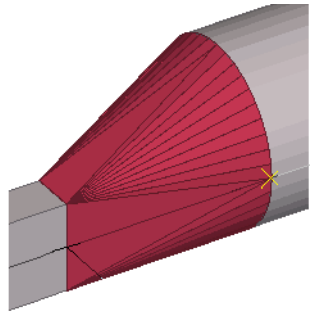
The parameters **Shape 1** and **Shape 2** determine the shape of the ends of the reducing piece. You can choose: **Circle**, **Ellipse** and **Rectangle**. These parameters only have effect when you make use of the second option (creating the reducing piece by indicating 6 points). If the reducing piece is made by selecting the two profiles, the shape will automatically be determined from the two profiles.

Number of segments

Number of segments (per angle) of the reducing piece. Example:



Number of segments = **3**



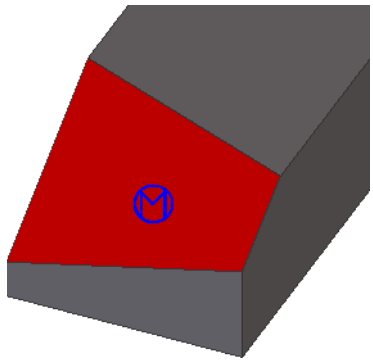
Number of segments = **10**

Welds

You can choose whether the reducing piece profiles should be welded or not.

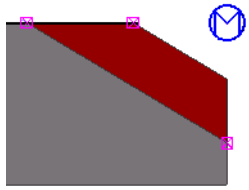
Construsoft component 3D Cut (10)

The Construsoft component **3D Cut** (10) creates a 3d cut to a (concrete) part. The cutplane is determined by picking three points on the parts' edges. The cut direction and cut offset can be defined.

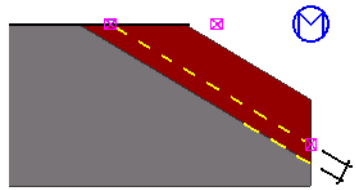


Offset

An offset can be defined for the cut. The offset is always perpendicular to the cutplane.



No offset



50mm offset

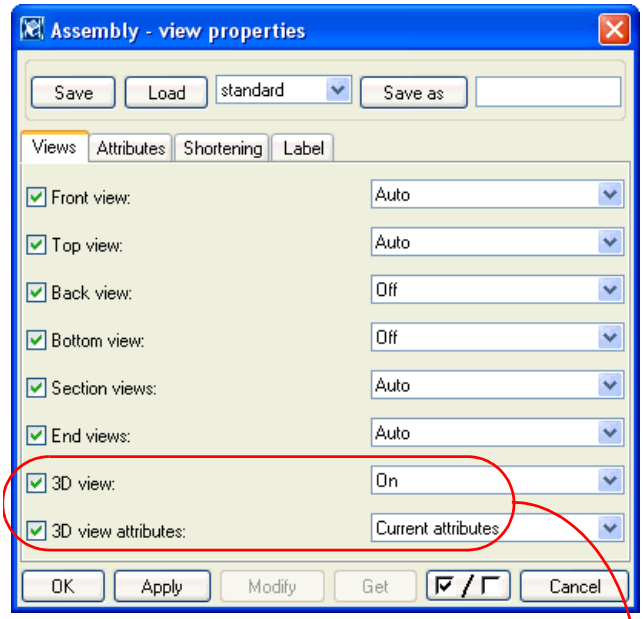
Drawings

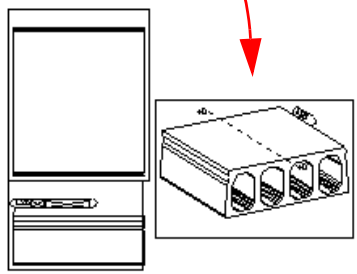
3D view on A- en W-drawings

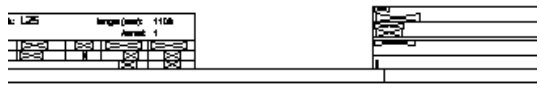
From Tekla Structures 11.3 on it is possible to place automatically a 3D view on the assembly (A) and workshop (W) drawings. In the view properties of the drawing properties two options were added:

3D view - to place a 3D view or not.

3D view attributes - to set te view properties for the 3D view.







Warning: by using drawing settings from (a) previous version(s)

A drawing setting from Tekla Structures 11.2 or an earlier version has no information regarding to **3D view** and **3D view attributes**, simply because this option was not avalible in those versions.

When you load an old drawing setting into Tekla Structures 11.3, the settings for **3D view** and **3D view attributes** will be unchanged.

Example

You open a drawing > open the drawing properties and add the 3D view. When you open a different drawing setting, the setting "3D view and 3D view attributes will not be reloaded.

Solution

Load every old setting > set the 3D options > save the setting > copy to the folder **ts**.

Creation of unfolded views in W-drawings possibly disturbed by variabele

In Tekla Structures 11.3, a new variable is introduced which makes it possible to create unfolded views of tubes containing holes:

XS_TUBE_UNWRAP_WITH_CUT_HOLES

However, it turns out that this variable disturbs the creation of unfolded views of tubular profiles which do not contain any holes.

By deafult, the variable is disabled (rem). We recommend to enable the variable and set it to FALSE. This means that the variable should look like this in the file **user.bat**:

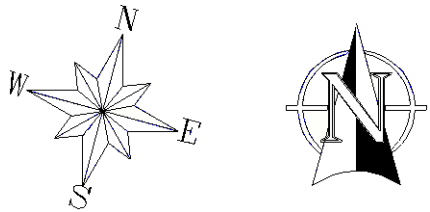
set XS_TUBE_UNWRAP_WITH_CUT_HOLES=FALSE

New symbol file "North.sym"

A new symbol file is added. This file includes a number of pictures. These symbols can be used on General drawings to point out the geographical north. This new file is located in the folder:

disk:\TeklaStructures\11.3\environments\country-independent\symbols

Example



Reports and templates

Reports in Tekla Structures 11.3

All reports in Tekla Structures 11.3 are converted and adjusted to the Template Editor v.3.3.

The converted reports are no longer editable in the Template Editor v.2.2.

A difference between the old and the new reports is that the width and the length is adjusted. This change will result in more information on every page. In case you still want to use 'old' reports, it could happen that the report does not fit properly on a A4 paper. This is caused by the following variables:

XS_PRINT_REPORT_LINE_WIDTH_PORTRAIT

XS_PRINT_REPORT_PAGE_HEIGHT_PORTRAIT

In Tekla Structures 11.3 the variables are set as follows:

set XS_PRINT_REPORT_LINE_WIDTH_PORTRAIT=90

set XS_PRINT_REPORT_PAGE_HEIGHT_PORTRAIT=70

To use 'old' reports (Tekla Structures 11.2 and previous versions), these variables have to be set as follows:

set XS_PRINT_REPORT_LINE_WIDTH_PORTRAIT=80

set XS_PRINT_REPORT_PAGE_HEIGHT_PORTRAIT=62

Using 'old' Reports and Templates in Tekla Structures 11.3

'Old' reports and templates (i.e. reports and templates created in 'old' Template Editor v.2.2) can be used in Tekla Structures 11.3. However, Construsoft does not guarantee 100% correct output of old templates in Tekla Structures 11.3. Verify de output of old templates Tekla Structures 11.3.

Note 1

If you want to use 'old' reports in Tekla Structures 11.3, and you do not have an older Tekla Structures version instelled on your computer to copy the old reports from, you can get them on this location:

disk:\TeklaStructures\11.3\environments\europe\firm\steel\template\old_reports.

This folder contains a zip-file including all 'old' reports. Unzip the file in:

disk:\TeklaStructures\11.3\environments\europe\firm\steel\template

and overwrite all files. The Zip-file contains report files only and no template files!

Note 2

If you want to use old reports and templates (i.e. reports/ templates created in 'old' Template Editor 2.2) in Tekla Structures 11.3, make sure all old reports are placed in one folder .

It appears that when reports/ templates are placed in more than one folder, only reports and templates are displayed from the folder which has the highest priority.

Example:

- an old report 'shortenlist' is copied to the folder **ts** of Tekla Structures 11.3.
- another old report 'boltlist' is located in the Tekla Structures 11.3 **template** folder.

In Tekla Structures 11.3, only the 'shortenlist' will be listed in the reportlist (among other new reports of course). The old boltlist will not be available.

Solution:

Store all old reports in one folder. This means storing all old reports and templates in the folder **ts** ór in the folder **template** of Tekla Structures 11.3. Only in this way all old reports and templates will be available in Tekla Structures 11.3.

Note 3

Reports and drawing template files are now searched on multiple loctaions. (in preveous versions these files had to be stored in the folder defined by variable XS_TEMPLATE_DIRECTORY).

The search-order (priority) is equal to other setting-files in Tekla Structures:

-**ts**-folder

- **steel**-folder

  - folder definined by variable XS_TEMPLATE_DIRECTORY

Delete brackets from drawing name on drawings

The drawing name usually is placed between two brackets (example **A [L.5]**).

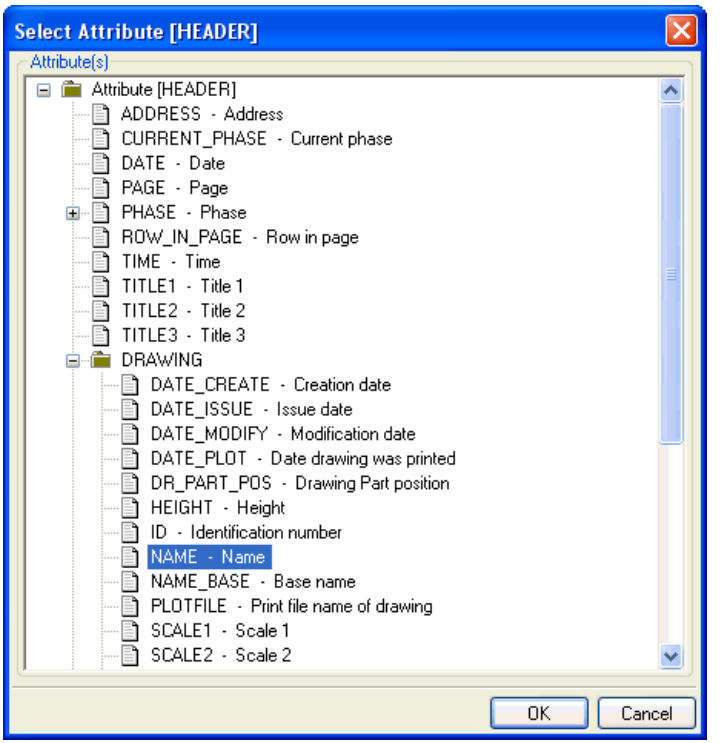
The Value Field resposible for this information on the drawing is **DRAWING.NAME**. This Value Field also shows the drawing type (A,W, G, M).

Brackets only can be left out when the Value Field **DRAWING.BASE_NAME** is used. This Value Field does not show the drawing type.

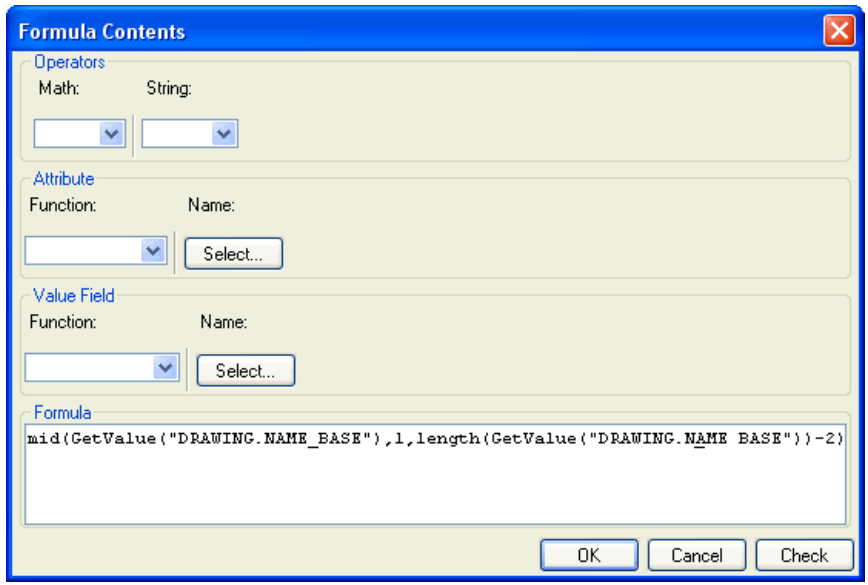
By means of a formula, the brackets can be left out.

Procedure:

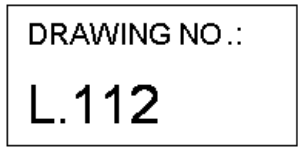
- Open the template in the new Template Editor
- Dubbleclick the Value Field. The dialogbox "Value Field Properties" will be opened.
- In "Formula" is written: GetValue("DRAWING.NAME"). Click on the button "Attribuut".
- Select **NAME_BASE** en click **OK**.



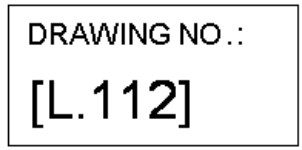
- Go back to the dialogbox "Value Field Properties and click on the button "Formula..." the dialogbox "Formula Contents" is opened.
- Rewrite the formula to: mid(GetValue("DRAWING.NAME_BASE"),1,length(GetValue ("DRAWING.NAME_BASE"))-2).



- Click OK.
- Save the template.
- Open a drawing with the following template.



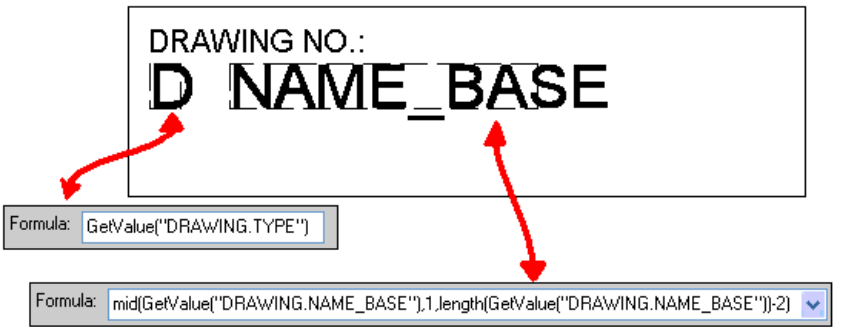
After



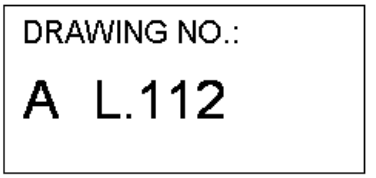
Before

The drawing type code (W, A, M or G) will not be displayed in case you use the Value Field: DRAWING.NAME_BASE.

In case you want to add the drawing type code, you have to add te Value Field: DRAWING.TYPE on the template.

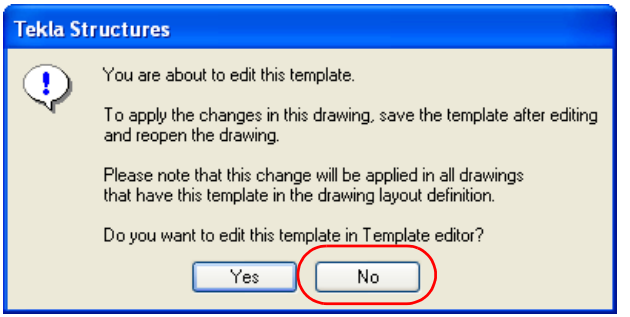


Example: on a drawing the drawing number will appear as follows:



Double-clicking a template: No-option added

When double-clicking a template, you now have the option to cancel the opening of the template by clicking the 'No' option in the dialog. (e.g. when you double-clicked the template by accident).



For comparison

In Tekla Structures 11.2, the dialogbox only contained a 'OK'-button to confirm opening the template in the Template Editor; it was not possible to prevent the template to be opened after double-clicking it.

NC files

Hard stamp in NC files of secondary parts

From Tekla Structures 11.1 on it is also possible to put a hard stamp for secundary parts in the NC files.In the hard stamp you can place information that can be engraved by the CNC machine into the beam.

Which information is put into the hard stamp, can be set at the NC files settings, tab "Hard stamp".

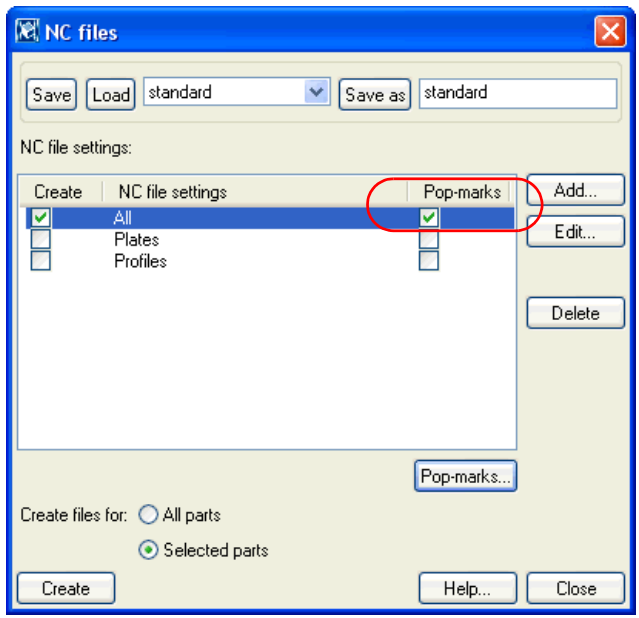
Note

In Tekla Structures 11.3 it is only possible to make the hard stamp for secondary profiles not for secondary plates.

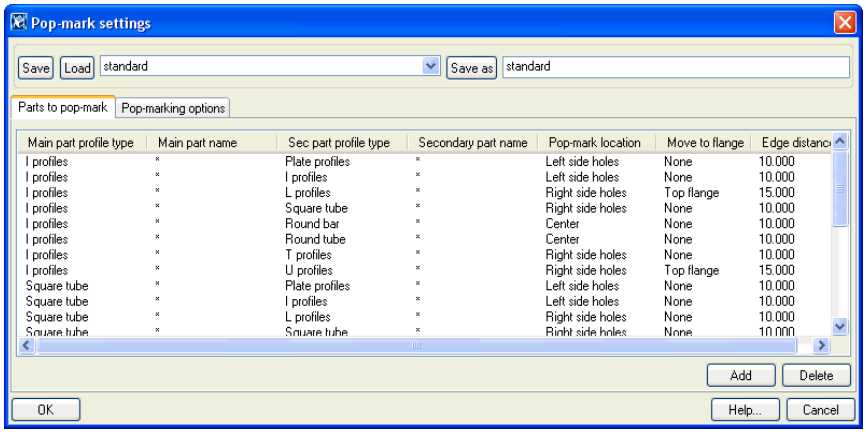
It is planned to make it possible to place hard stamps on secondary plates in future versions of Tekla Structures.

Add popmarks to NC file > Numbering necessary

To add pop marks (drill holes of 0 mm diameter) is simplified. In Tekla Structures 11.3 in the dialog box NC files, it is possible to check a box if you want to generate pop marks.



When and how the pop mark have to be placed, can be set in the dialog box "Pop marks". You can open this dialog box by clicking on the button "Pop-marks..."



This dialog box replaces the pop marks configuration file *pop_marks_parts.inp*. The same method is used to define the pop marks, it is only easier to configure

Pop mark settings can be saved in this dialog box, the file extension is *.ncp.*

Note

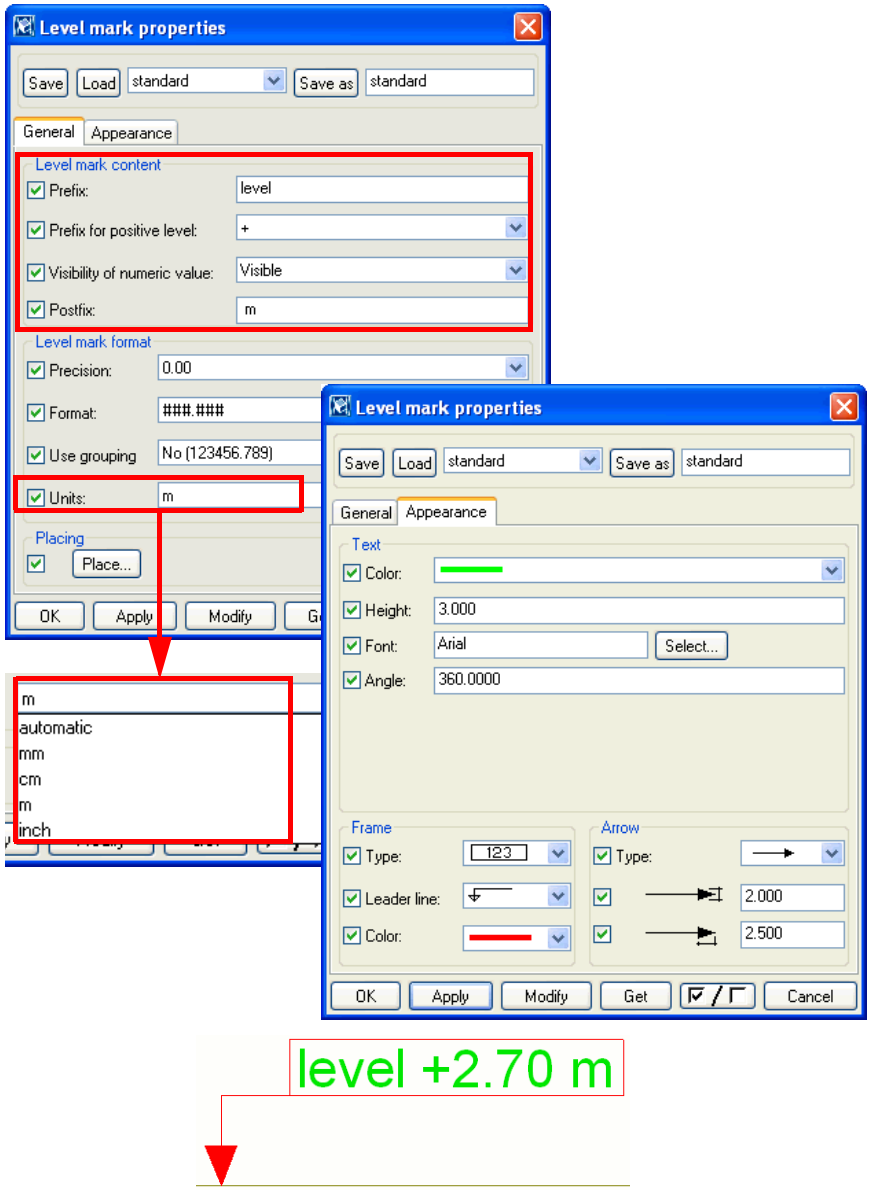
In case the generation of pop marks is switched on after all profiles already have been numbered, the model has to be numbered again, because pop marks have an influence on the part numbering.

Tekla Structures gives **NO** warning when it necessary to number after the option to generate pop marks is switched on. Please be aware of this!

Tekla Structures 12.0

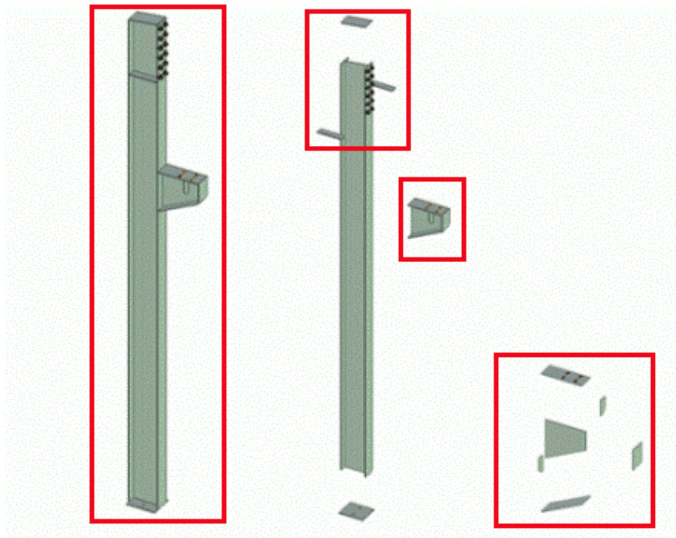
Levelmarks

In Tekla Structures 12.0 levelmarks do have more options, like units, prefix and postfix.



Sub-assemblies

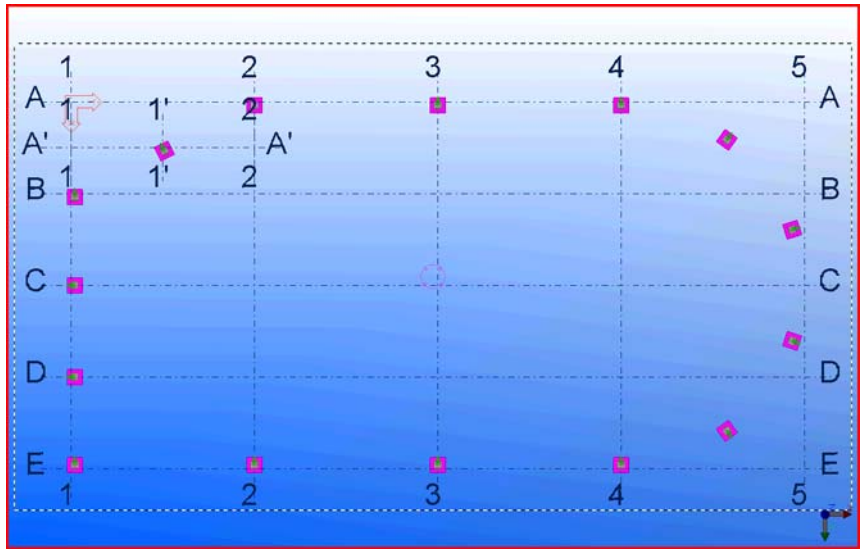
In Tekla Structures 12.0 sub-assemblies can be used. This means that an assembly can exist out of more assemblies. From every assembly can be an assembly drawing created. And of course an assembly drawing of the complete assembly.



Anchorboltplan using 2 scales

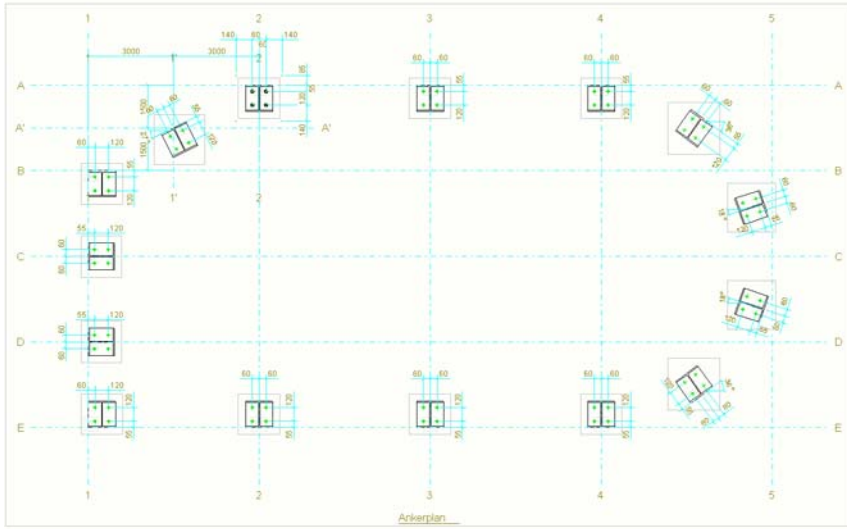
In Tekla Structures 11.2 it will be possible to represent the anchorbolt plan using 2 scales: one scale for the grid and one scale for the profiles.

Model

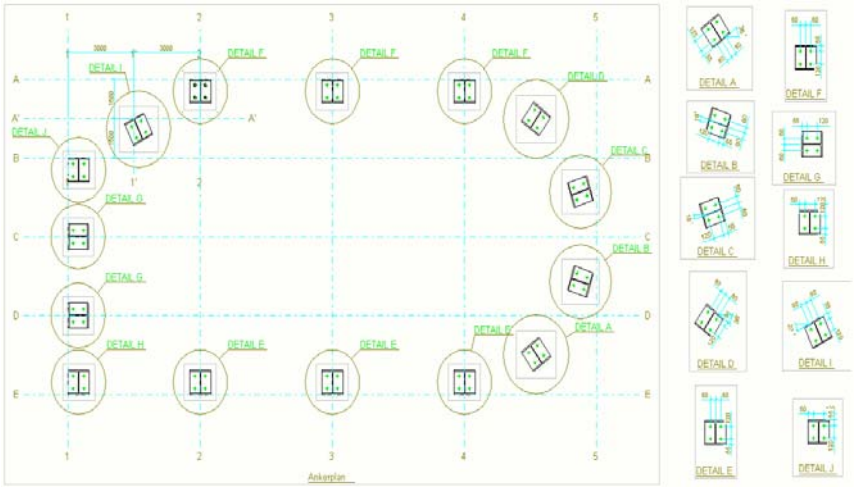


Drawing

Gridscale 1:100, column/baseplate 1:25 (separate scale definition)

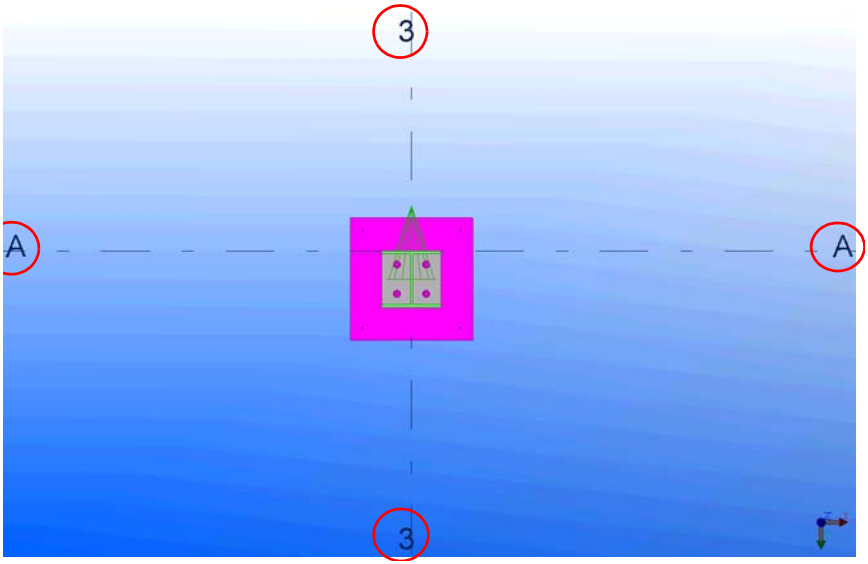


Drawing Gridscale 1:100, column/baseplate 1:25, details 1:25 (separate scale definition)



Gridtext always visible

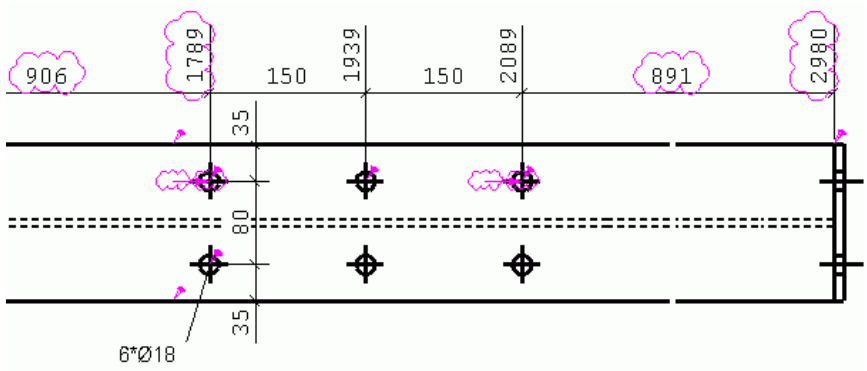
In Tekla Structures 11.2, grid-labels will always be visible in rendered modelviews. When the view is zoomed in, the labels will remain visible at the edges of the view.



Highlighting changes in associative dimensionlines

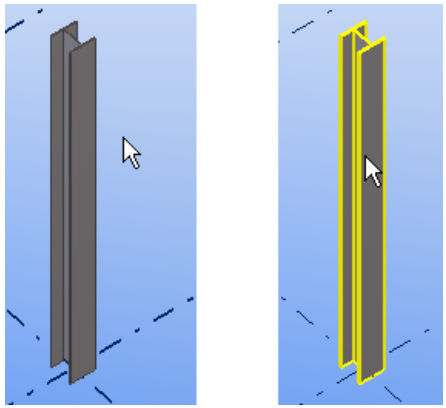
If associative drawings (single part-, assembly- and General Arrangement drawings) are updated in Tekla Structures 12.0, it will be possible to hightlight changed dimensiontexts en moved dimension points using clouds. These clouds will not be printed.

The visibility of the clouds can be disabled.



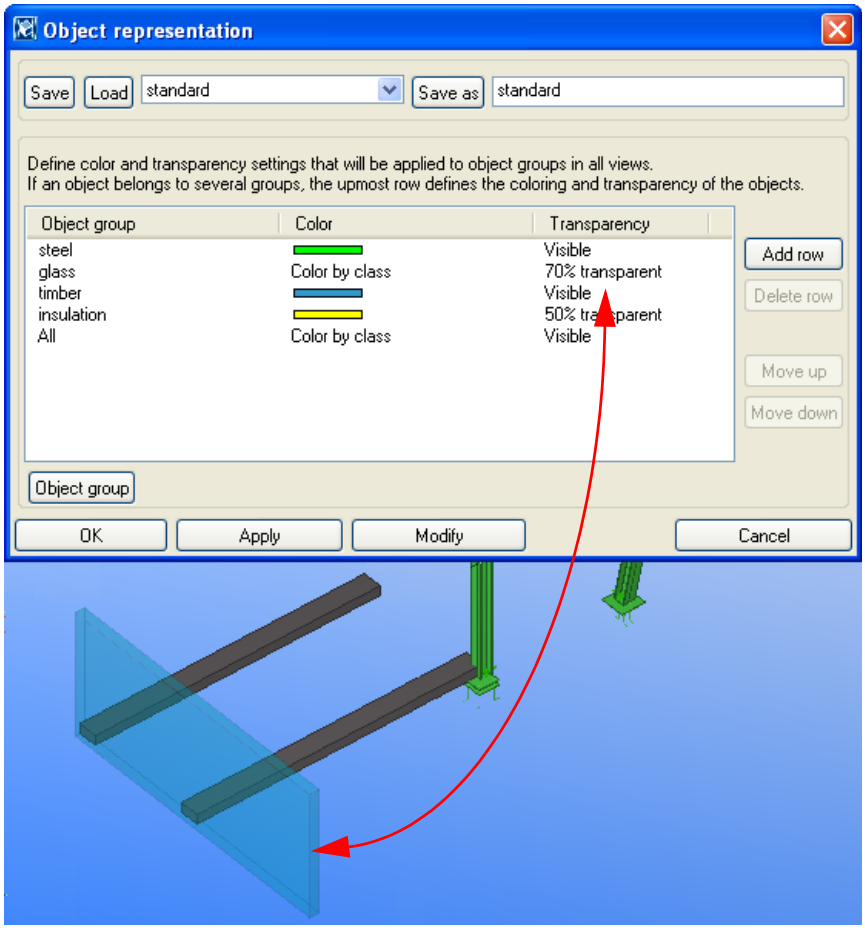
Rollover highlight

The rollover highlight makes it easier to select objects, by highlighting each part in yellow as the pointer moves over it



Object representation: transparency

In Tekla Structures 12.0, objects can be represented as transparent. Using rules you can set which parts should be represented transparent and which gradation should be applied.



Custom components available in all models

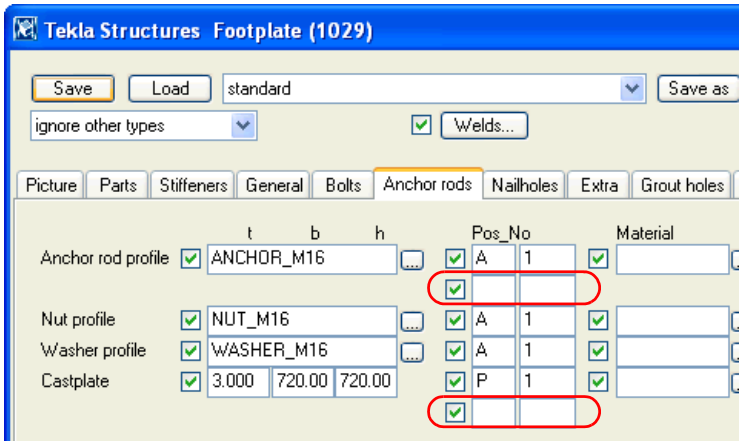
As from Tekla Structures 12.0 it will be possible to use Custom Components in all models without having to import them in each model separately.

To achieve this, make sure all *.uel files are placed in folder ***...europe\custom-components\steel***.

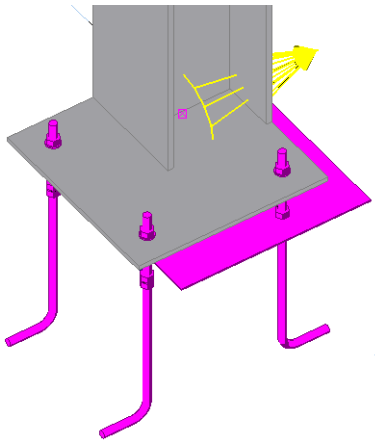
Construsoft component Baseplate (1029)

Several improvements are applied to Construsoift component baseplate 1029.

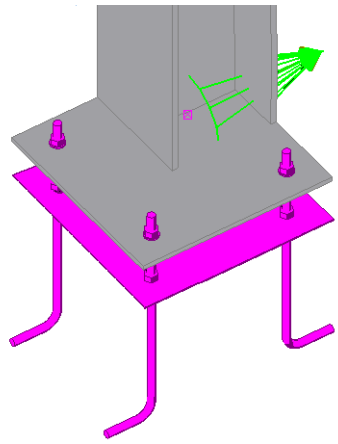
- In tab *Anchors*, Assembly Prefix and - Startnumber fields are added for anchor bars and cast plate.



- The castplate offset now works correctly in horizontal direction. Previously, the horizontal offset for castplate/ nailholet did not work (verticale offset worked properly). In TS 12.0, the vertical castplate/ nailhole offset is corrected.



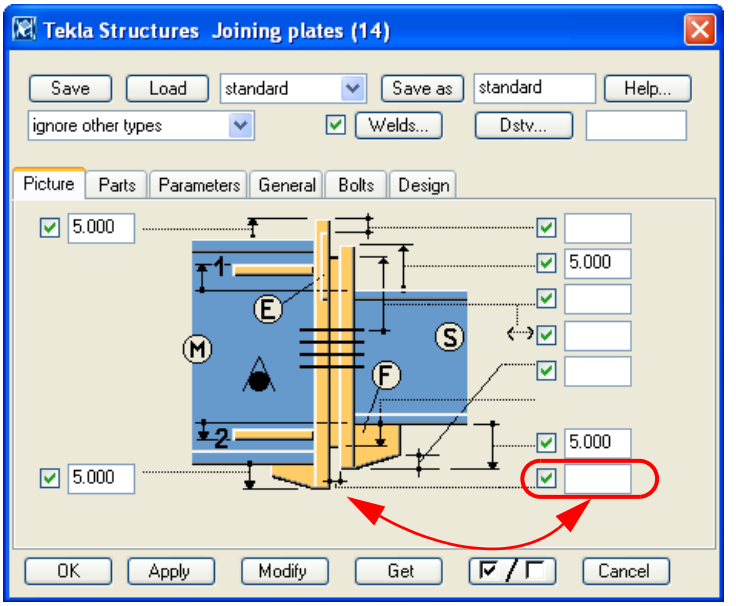
TS11.2 and previous



TS12.0

Tekla component Joining plates (14)

In Tekla component Joining Plates (14), an option is added which determines the clearance between the two plates. Previously this type of connection could only be created with Construsoft Customs Component "CS_14.uel". Now Tekla Component Joining Plates (14) can be used in multiple situations.



Tekla component Stairs (S71)

Tekla component Trap (S71) will be modified: in Tekla Structures 12.0 this component will be able to create the same step profiles as component Stais (S80). so besided the U-steps, in Tekla Structures 12.0 this component can also create standard step profiles of manufacturers like Dejo, Staco and Thielco.

Besides the other advantages in component Stairs (S71), this makes that this component will be the preferred component for stair creation.

Notes:

8 Construsoft Newsletter TS 12.0

Content This Tekla Structures 12.0 newsletter contains information about:

General

Sending models to the Construsoft Support desk (page 3)
E-mailing large files (page 3)
Construsoft Optimize Installation Tool (COIT) (page 4)
New functions: Interactive tour (page 4)
Mousepointer (page 5)

Modeling

Warning when starting Tekla Structures twice (page 6)
Creating, opening and saving models (page 6)
Background color in model views (page 10)
Locking the Grid (page 11)
Gridlabels always visible (page 11)
Fit Workspace to parts (page 12)
Changes to toolbars (page 14)
Rollover highlight (page 15)
XSnap (page 15)
Snap to reference objects (page 15)
New icons in Select Switches toolbar (page 15)
No points to profiles (page 17)
Deforming steel parts (page 20)

Round base plate (1052) (page 21)
Base plate component (1029) (page 22)
Component Stairs (S71) (page 23)
Editing a contour plate or a polybeam (page 24)
Assembly: set new mainpart (page 24)
Object representation manager (page 26)
Object representation: limitations (page 28)
Profile catalog: catalog- and parametric profiles merged (page 29)
New parametric profiles: GUTHER and AFSCHOT (page 30)

Drawings

Anchor bolt plan on a general arrangement drawing (page 31)
Diagonal check dimension after cloning incorrect (page 34)
Improvements in cut symbols (page 35)
Setting "Model" works correct again (page 36)
Saving en Freezing associative drawings (page 37)
Indicate changes to associative dimensions and marks (page 40)
Associativity of graphical objects (page 42)
Associative notes (page 43)
Datum point for elevations (page 44)
More clear representation large dimansion values (page 45)

Custom components

Availability of Custom Components and Sketched profiles (page 46)

Tekla Structures 12.1

Multicopy (page 49)
Speed improvements (page 49)
Bolt/ hole marks (page 50)
Short dimensions: switch sides (page 50)

Tekla Structures 13.0

Drawing list (page 51)
Aligning Multi text (page 52)

General

Sending models to the Construsoft Support desk

If you want to send an email to Construsoft Support, possibly including a Tekla Structures model, there are some regulations you have tou comply with. This helps Construsoft to analyse the problem faster so we can give you a quick solution.

- Give a clear description of the problem. Also report in which version the problem occurs, and possibly notify with whom the problem has been discussed.
- If the model folder is quite large, try to extract the problem situation into a smaller model. Make use of the command ***Copy from model***.
- Split up the problem in a phase, so the Construsoft Support Engineer can easily locate the problem.
- If possible, re-adjust the workspace so that only the parts in question are visible is the view.
- If you made changes to one of the catalogs (profile-, bolt- of material catalog) attach the correct catalog files.

E-mail which fulfils these requirements can be handeled much faster!

Use the correct email address:

Questions about	E-mail address
Tekla Structures general	ts-support@construsoft.com
Licenses	licenties@construsoft.com
Custom Components	component-support@construsoft.com

E-mailing large files

If you want to send a large file to the Construsoft Support desk, but your email provider refuses to send the files because they exceed a given maximum filesize, you can use the webservice www.yousendit.com to get the files at the desitation.

This webservice allows you to send files up to **100 Mb**.

If you want to use this service, go to http://www.yousendit.com and follow the instructions on the website.

Construsoft Optimize Installation Tool (COIT)

A new version of COIT is available.You can download this version from Construsoft Xtranet; go to ***Downloads*** > ***Other's***.

You can use this tool to delete superfluous files from the Tekla Structures installation folder. This can be useful, because the Tekla Structures installation copies all settings and default files in the installation folders.

Only if you do ***not*** want to use the defaults, you can use the COIT program to delete specific files easily.

New functions: Interactive tour

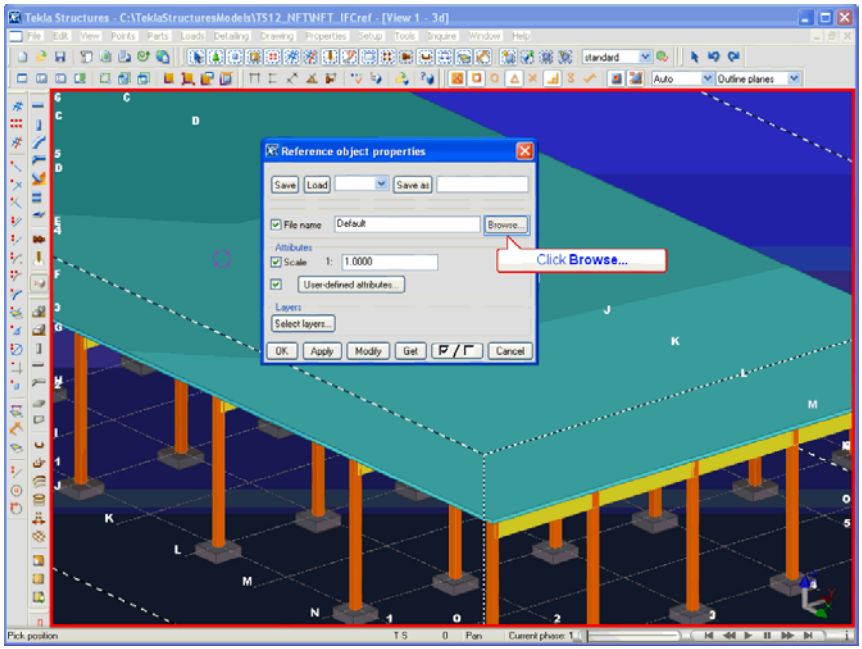
From the Xtranet pages, you can download an interactive tour which shows the new features in Tekla Structures 12.0.

Some functions have interactive videoclips showing you how to use new features and commands.

For the tour, go to ***Versions*** > ***Tekla Structures 12.0***.

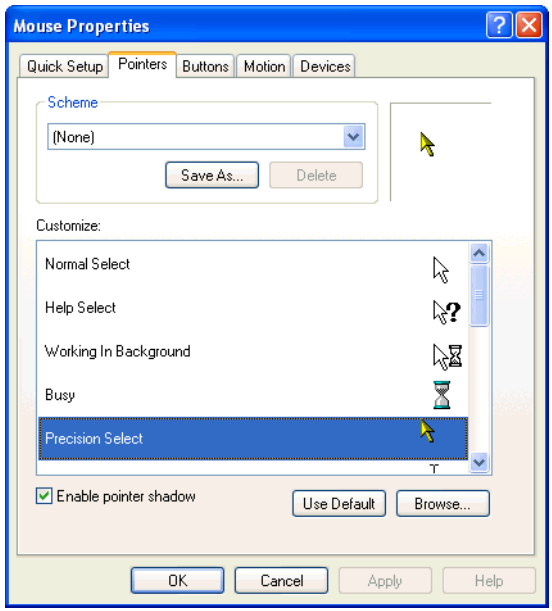
Step plan

- Extract the file **new_in_120.zip**, a new folder **NewFeaturesTutorial** will be created.
- In the folder, doubleclick file **NewFeaturesTutorial.htm** to start the tour.



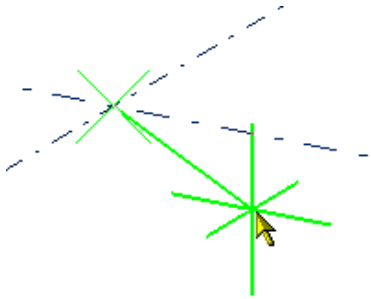
Mousepointer

When working with Tekla Structures, it is recommended to use the **Precision Select** mousepointer:



The advantage of this pointer is that the color changes to yellow when a command is activated.

This pointer is better visible/ recognisable in rendered views:



Modeling

Warning when starting Tekla Structures twice

If you start Tekla Structures and another Tekla Structures session is already running, a warning will be displayed:



This warning indicates that the dotnet-link is already in use. Click <**OK**> to start the program.



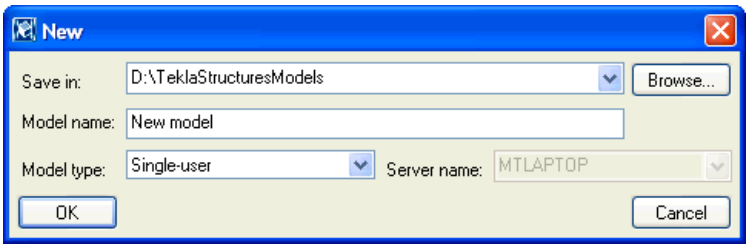
Be careful with starting multiple Tekla Structures sessions at the same time.

Creating, opening and saving models

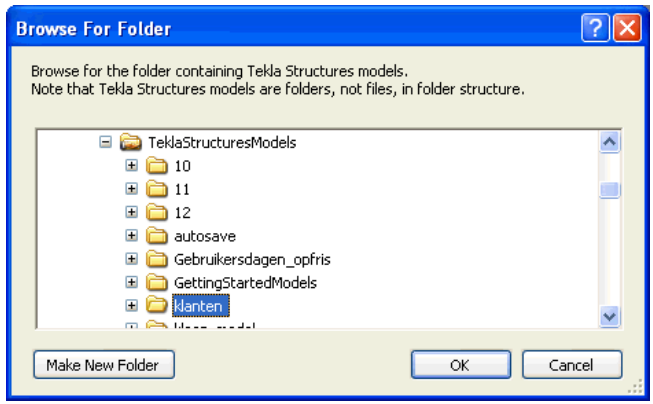
In Tekla Structures 12.0, creating and opening models is a lot easier than before.

Creating a new model

Specify the model location in field **Save in**, or use the default location which is specified during the installation, for example `C:\TeklaStructuresModels`.



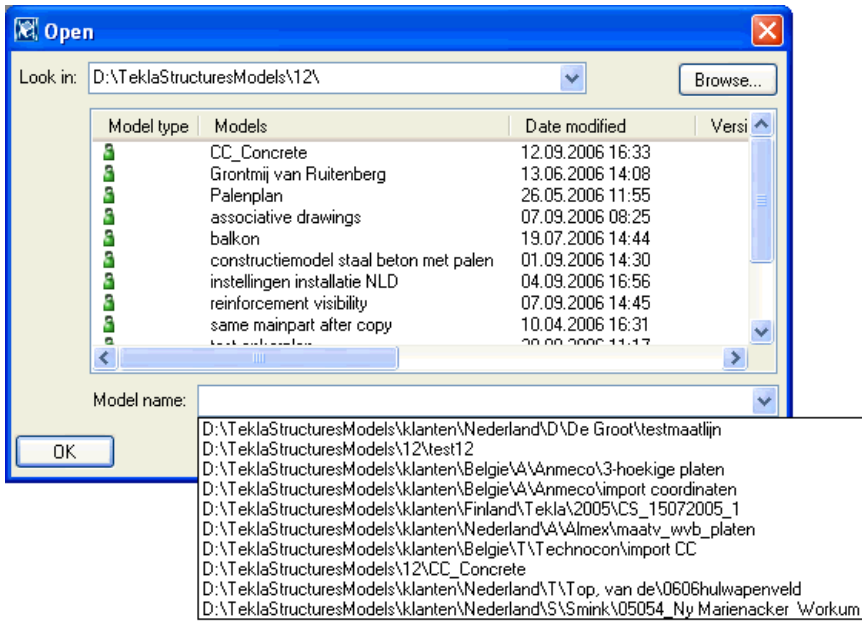
Click the **Browse** button to open dialog **Browse for folder**, where you can browse to another folder. It is also possible to create new folders.



Use the field **Model Type** to define the model to be a single-user- or multi-user model. For a multi-user model you need to specify the server in field **Server name**.

Opening a model

To open a model, click **File** > **Open**. To browse for a model, click the **Browse** button to open dialog **Browse for folder**. The information in the columns **Designer** en **Description** are derived from dialog **Project properties**.



If you create or open a model, both **Open** and **New model** dialogs contain a list with 10 recent opened models.

Single / multi-user

In the Open model dialog, you can recognise if a model is created in single-user mode or in multi-user mode. This is displayed by icons in the most left column:



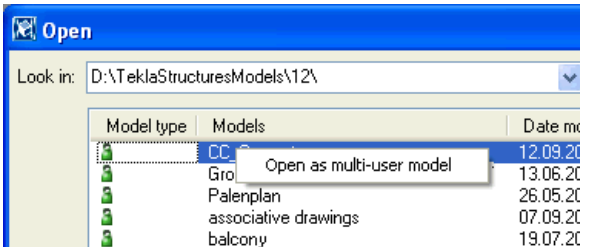
Multi user



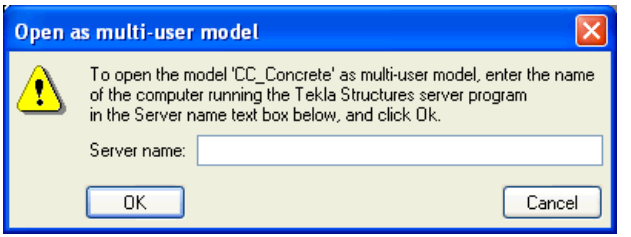
Single user

To open a single-user model in multi-user mode

1. Rightmouseclick a model in the **Open** dialog.
2. Select the option **Open as multi-user model**.

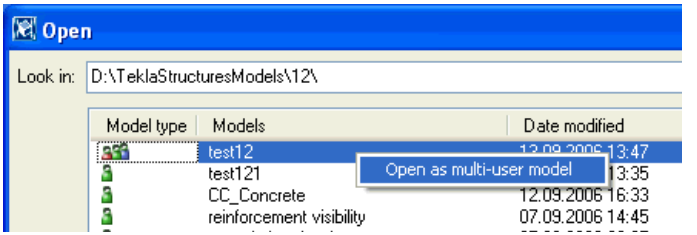


3. Enter the server name in field **Server name**.Then click **OK**.



To open a multi-user model in single-user mode:

1. Select the model in the **Open** dialog, then rightmouseclick.



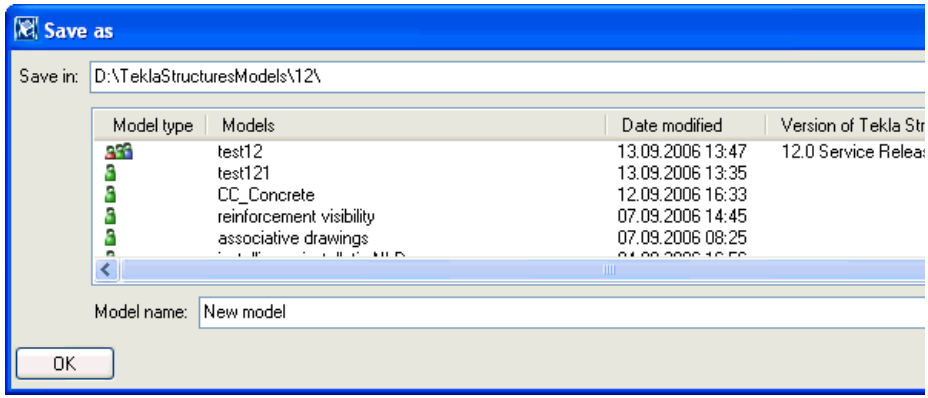
2. Select the option **Open as single-user model**.

See also

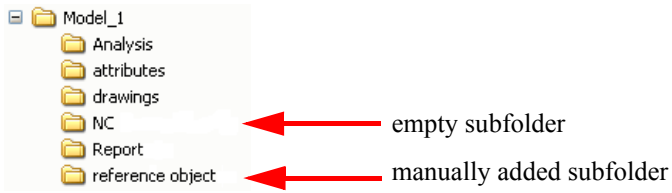
For more information about working in multi-user mode and installing multi-user, seee **Tekla Structures.pdf** in folder: **disc:\TeklaStructures\version\environments\europe\pdf.**

Save as

To save a model with another name, click **File** > **Save as** and enter a name in field **Model name**. Then click **OK**.



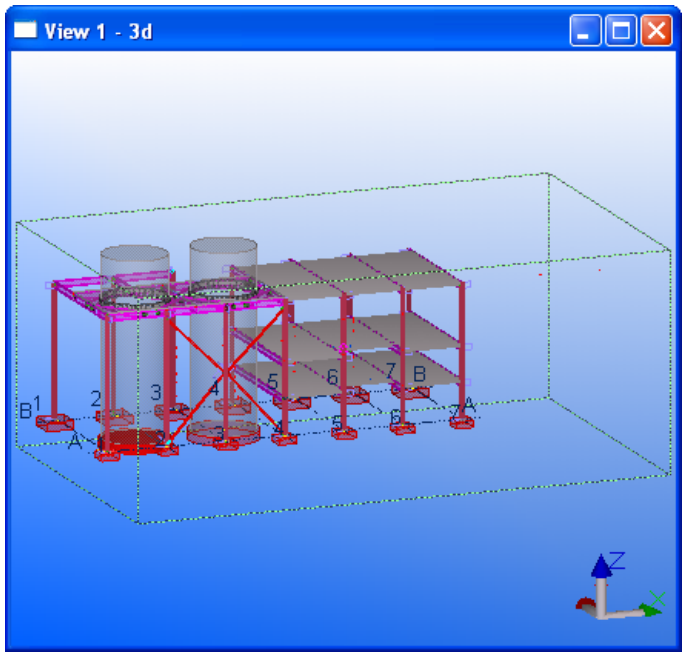
If you save a model in Tekla Structures 12.0 using command **Save as...**, all subfolders will now be copied correctly. This counts for both empty subfolders and not-empty subfolders. Also manually added folders are copied:



Background color in model views

The backgroundcolor in rendered model views has been changed in Tekla Structures 12.0.

In previous version Tekla Structures 11.x, the default backgroundcolor was black, in Tekla Structures 12.0 the backgroundcolor can be gradient:



This gradient background has been chosen because research has demonstrated that this background setting minimises headaches. The grid- and dimension color in modelviews is black and can not be changed.

You can still modify the backgroundcolor, use the following variables to do this:

- `XS_BACKGROUND_COLOR1`: left-upper corner
- `XS_BACKGROUND_COLOR2`: right-upper corner
- `XS_BACKGROUND_COLOR3`: left-lower corner
- `XS_BACKGROUND_COLOR4`: right-lower corner

These values define the default color transition:

- `XS_BACKGROUND_COLOR1`: 0.98828125 0.98828125 0.99609375
- `XS_BACKGROUND_COLOR2`: 0.99609375 0.99609375 0.99609375
- `XS_BACKGROUND_COLOR3`: 0.0000 0.37109375 0.99609375
- `XS_BACKGROUND_COLOR4`: 0.21484375 0.4609375 0.875

If you do not want a gradient color transition, but a uniform color, then enter the same values for all 4 variables.

White backgroundcolor

If you want to change the background color to white, modify the next variables in **Tools** > **Advanced options** > **Model views** in:

The scale goes from 0 to 1. For all 4 variables, enter **1.0 1.0 1.0**. Close and re-open the view to activate the changes.

The modified variables are saved in file **options_username.ini** in folder ***disc:\TeklaStructures\12.0\nt\bin.***

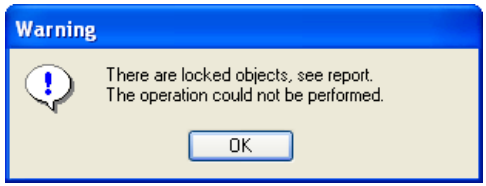
If you do not enter any value for these variables, the gradient default backgroundcolor is set again.

Locking the Grid

Grids can be locked in Tekla Structures 12.0. This can be used to prevent the grid from being removed unintentionally.

Step plan

- Doubleclick a grid to open the **Grid** dialog
- Click the button **User attributes**
- The **Grid(1)** dialog appears, set **Locked** to **Yes**.
- If you want to remove a locked grid, the follwing warning shall appear:



Gridlabels always visible

Gridlabels are always visible now (if the gridlines are visible too). Also when zoomed in in a view, you still can see your location in the model.

You can use special characters like umlauts and double accent aigu in grid labels, as well as combined characters like acute accent.

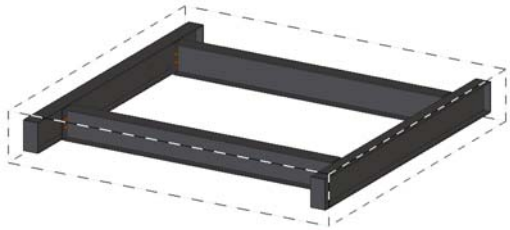
By the way, the dark-blue color of gridlines and -labels in rendered views can not be changed in Tekla Structures 12.0.

See also

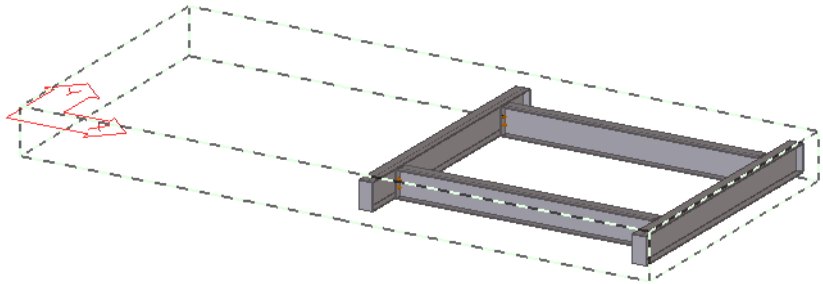
For more information, see chapter *Background color in model views on page 10*.

Fit Workspace to parts

In Tekla Structures 11.x, the Fit workspace command did not take a selected grid into account. The workspace only looked at the selected parts:



In Tekla Structures 12.0, the Fit Workspace commend will take a selected grid into account when fitting the workspace:

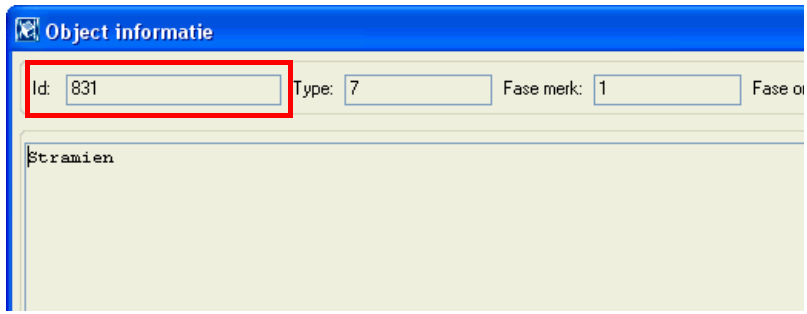


To prevent the grid affecting the Fit Workspace command, make sure the grid is not selected when activating the command:

You can deselect the grid manually, but a better way would be to filter the grid by its ID-number:

Step plan

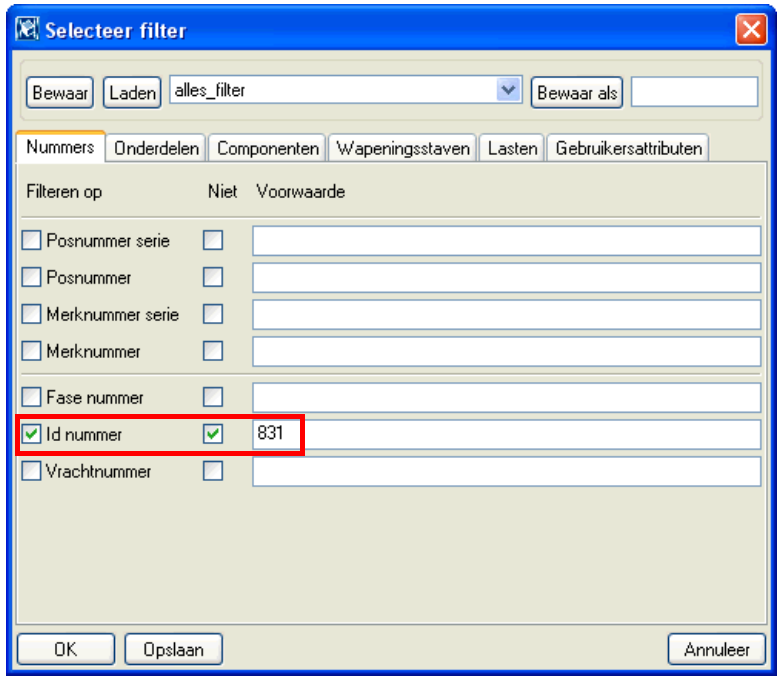
- Go to **Inquire** > **Object** or use shortkey **k** to inquire the grid information:



The ID-number is (in this case) 831, you can filter this ID number:



- Doubleclick the icon **Select filter** in toolbar **Selectiebox** or go to **Setup** > **Select filter....** The following dialog shall appear:



- Modify dialog and press <**Apply**> or <**OK**>

Now, if you fit the work area to the selected parts, you will see that the work area will be adjusted as before.

Changes to toolbars

Icon Create Clip plane added

The **Construsoft** toolbar has been changed:



The **Create Clip plane** icon has been added to the Construsoft toolbar.

The command is also accessible through **Window** > **Create Clip plane**.

U can change the position of the scissors-symbol of a clip plane by pressing the <**Shift**> key and moving the symbol.

See also

For more information about clip planes, see **Tekla Structures.pdf** in folder: **disc:\TeklaStructures\version\environments\europe\pdf.**

Construsoft toolbar

The **Construsoft** toolbar now has a fixed position in the left-lower side in the screen. The toolbar has got a fixed position because the grid labels are always visible. Because labels are invisible behind floating toolbars, we decided to give the **Construsoft** toolbar a fixed position.

Points-toolbar not visible by default

As from Tekla Structures 12.0, the **Points** toolbar is not visible by default. There are 2 reasons for this decision:

- Since the introduction of **Numeric Snap**, you have to create less points for positioning profiles. Some commands in this toolbar have become unnecessary.
- The **Points** toolbar needs pretty much space. Construsoft has decided to fill this space with a toolbar with more frequently used cimmands. The commands in the **Points** toolbar are still accessible through the **Points** pull-down menu.

The properties dialogse from all icons in toolbars can be opened quickly by holding down the <**Shift**>-key while activating the command.

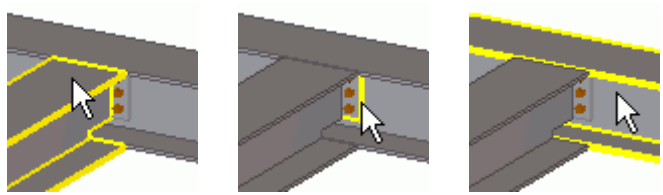
Of course you can make the **Points** toolbar visible again. To do this, go to **Window** > **Toolbars** and check **Points**.

See also

For more information about toolbars, see **Tekla Structures.pdf** in folder: **disc:\TeklaStructures\version\environments\europe\pdf.**

Rollover highlight

If you move the mouse pointer over objects in rendered model views, the objects will be hightlighted in yellow. This makes it easier to select the correct objects..



You can switch **Rollover highlight** on or off with shortkey **H**, or by clicking the command **Setup** > **Rollover hightlight.**

XSnap

Xsnap supports snapping to the correct points. While moving the mousepointer over an object, Tekla Structures marks the available snap positions.

You can switch **XSnap** function on or off using shortkey **T** or by clicking command **Setup** > **XSnap.** This feature is switched on by default.

Snap to reference objects

Snapping to reference objects is possible since Tekla Structures 11.x. However, for high detailed reference objects it may happen that a large number of snap points is found. This may result in a delay.

In such cases, it is advisable to switch off the select switches which are not necessary at that moment. For example, is an endpoint has to be found, it may be useful to switch off the other select switches.

New icons in Select Switches toolbar

The icons in the **Select Switches** toolbar are used to define which objects can be selected. In Tekla Structures 12.0, 4 icons are in this toolbar have been replaced:





If you activate switch **Select components**, you can select every level of an component, from the highest level downwards to loose parts in components.



If you activate switch **Select objects in components**, the select order is opposite: from the lowest level (loose parts in components) to the highets level.



If you activate switch **Select assemblies**, you can select every level of an assembly, from the highest level, through sub-assemblies, downwards to the loose parts.

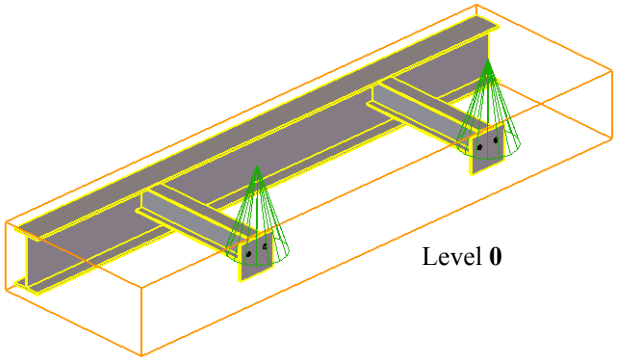


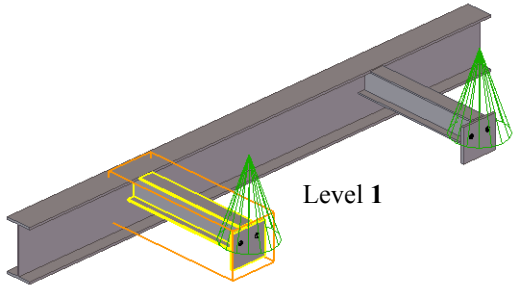
If you activate switch **Select objects in assemblies**, the select order is opposite:, from loose parts (lowest level) to complete assemblies (highest level).

The level of a selected assembly is displayed in the **Status bar**.

To shift to a specific level in an assembly, while the assembly is highlighted, hold down the <**Shift**> key and scroll the mousewheel.

The large assembly gets 0 (null), the smaller assembly gets 1, etcetera. You can continue to level 9:

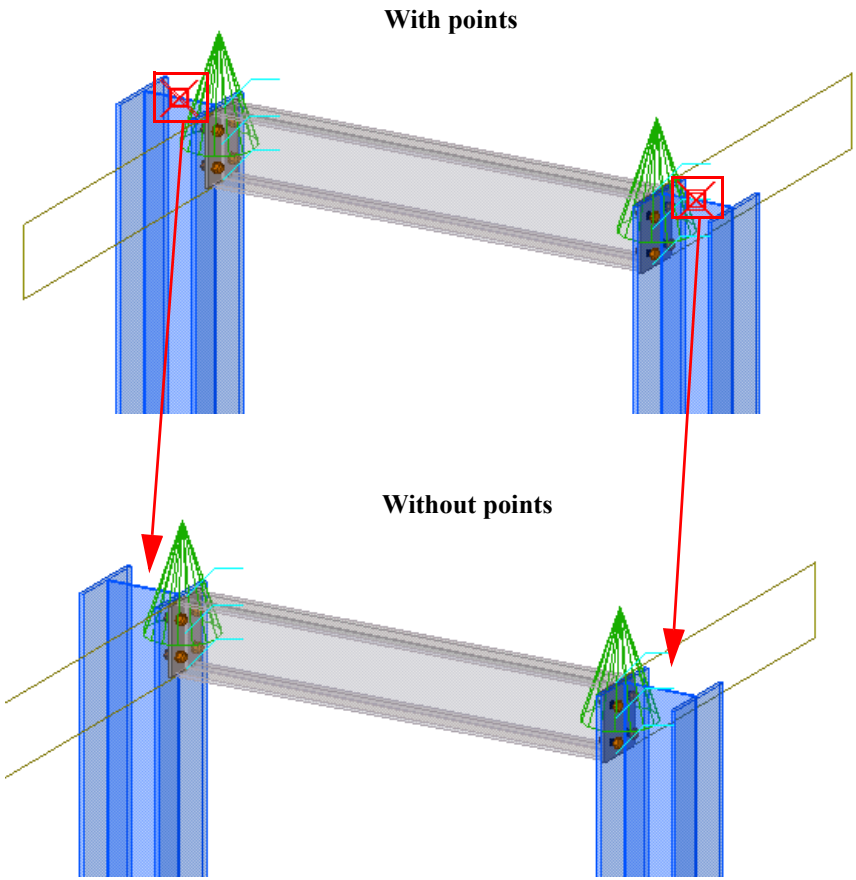




No points to profiles

Tekla Structures no longer creates points at profile ends. This reduces memory consumption and process-time. This means you have to redraw views less than before. If you need a point on a profile end, you can add a point manually.

Upto and including Tekla Structures 11.3 , points were automatically created on profile ends. These points could be used to, for example, move a profile end.

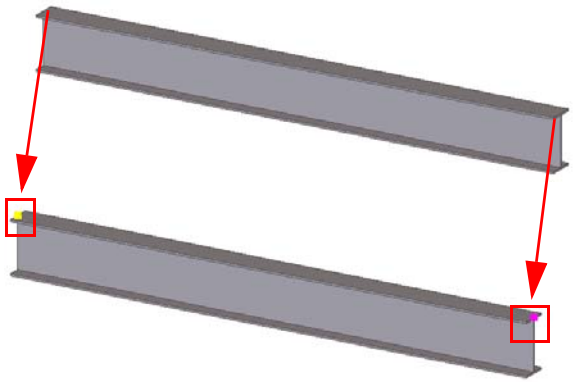


You can use profile handles and snap to position and move profiles. See next examples.

Select handles

To select a profiles' handle:

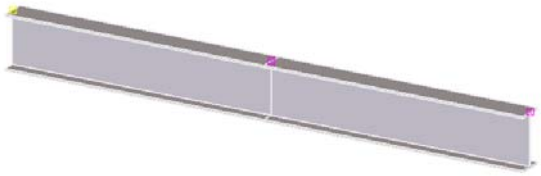
1. Select the object
2. Hold down the <**Alt**> key and drag the mousepointer from left to right to select the profile. Tekla Structures will show the profiles' handles. The handle of the first picked point will be yellow, the other point will be purple.:



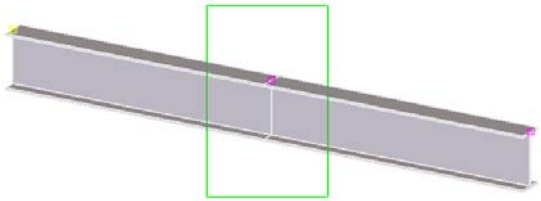
Moving objects

By selecting handles, you can move objects more easily.

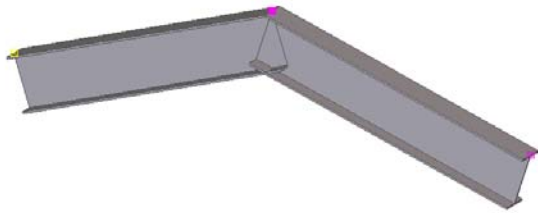
1. Select both objects by holding down the <**Ctrl**> key and picking the objects.



2. Hold down the <**Alt**> key and drag the mousepointer from left to right to re-select both objects.

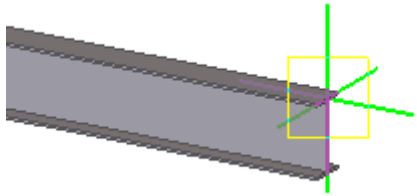


3. Select the handle in the middle of the two objects and then rightmouseclick (or rightmouseclick on the icon).
4. Select command **Move** > **Translate** and enter coordinates to move the two objects.
5. Click **Move**. The two objects will be moved.



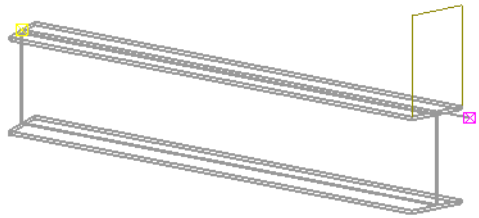
Snap to reference points

To be able to snap to parts' handles, make sure that switch **Snap to reference lines / points**  (**Snap settings** toolbar) is enabled. If you move the mouse pointer over part' handles, the snap symbol shall appear. Then leftmouseclick to select the parts' handles.



Part reference lines

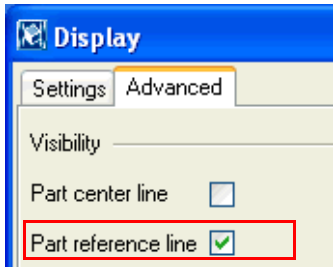
If you pick two points to position a part, these points will form the reference line, and handles will appear on the parts ends. You can use these handles for snapping:



You can adjust the visibility of parts' reference lines in the **Display** dialog. To display a parts' reference line:

1. Go to **Propertie** > **View..**.
2. Click **Display...** to open the **Display** dialog.

3. In tab **Advanced**, check option **Part reference line**.



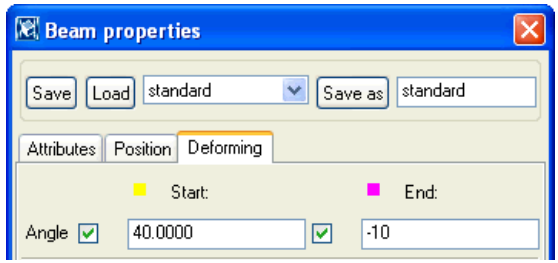
4. Click **Modify** and **OK**.

Deforming steel parts

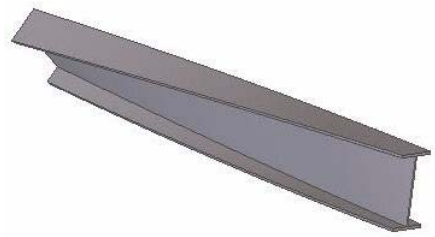
In Tekla Structures 12.0, you can use Deforming functionality in Steel Detailing also. To deform a steel beam:

1. Doubleclick a beam to open dialog **Beam properties** and go to tab **Deforming**
2. In field **Angle**, enter a value for 'Start', relative to the parts' reference line.
3. Possibly, in field **Angle** enter a value for 'End', relative to the parts' reference line.

For example: a beam needs a deformation angle of 40 degrees at the parts' start-point and 10 degrees at the parts' endpoint, In the Angle-field, enter '40' in **Start**-field and '-10' in the **End**-field



4. Click <**Modify**> to deform the beam, press <**OK**> to close the dialog.



You can create an undeformed drawingview of a deformed beam:

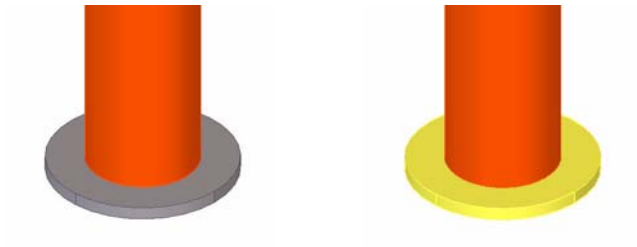
Step plan

1. In the Drawing Editor, open dialog **Drawing properties** and press **View**, or doubleclick a view and go to tab **Attributes 2**.
2. Set option **Undeformed** to **Yes**



Round base plate (1052)

As from Tekla Structures 12.0 SR1, the base plate in component Round base plate (1052) will be welded to the column if no bolts are used. This did not work in previous versions.

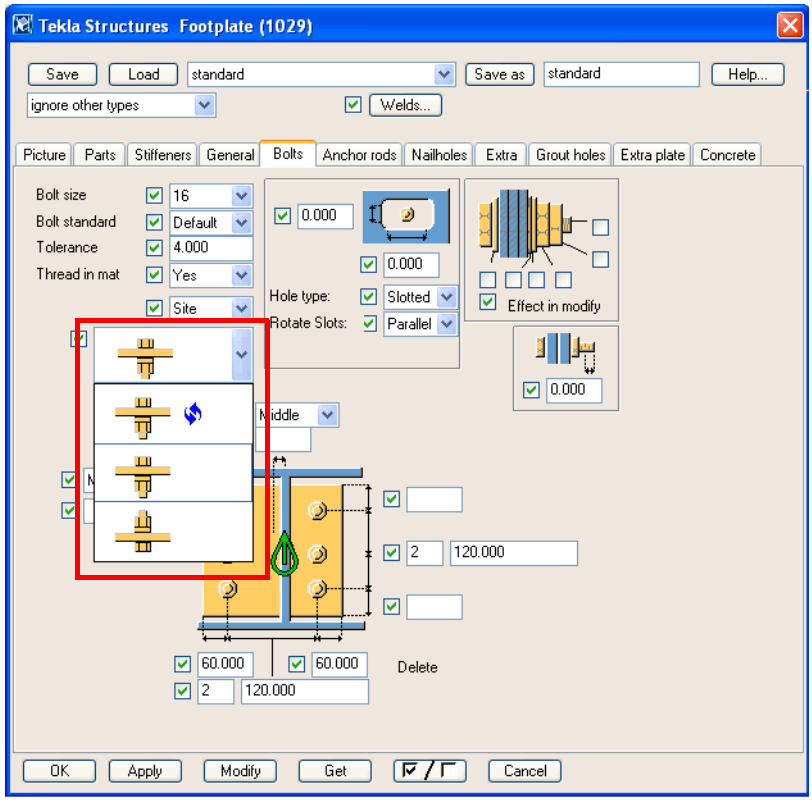


TS 12.0 and previous versions

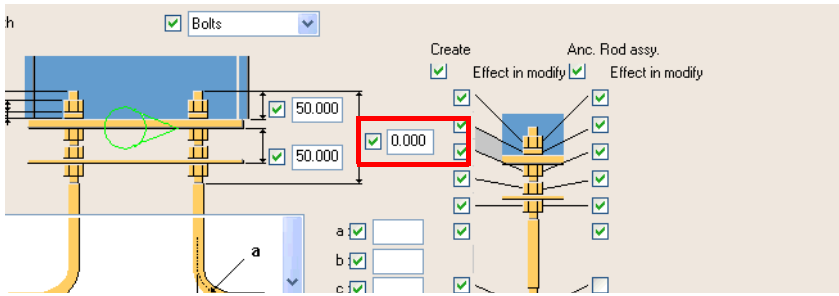
TS 12.0 SR1

Base plate component (1029)

In Base plate component (1029), a picklist is added to the **Bolts** tab for rotating bolts (no anchors). This options was already available in some other components, but not in Base Plate component1029:



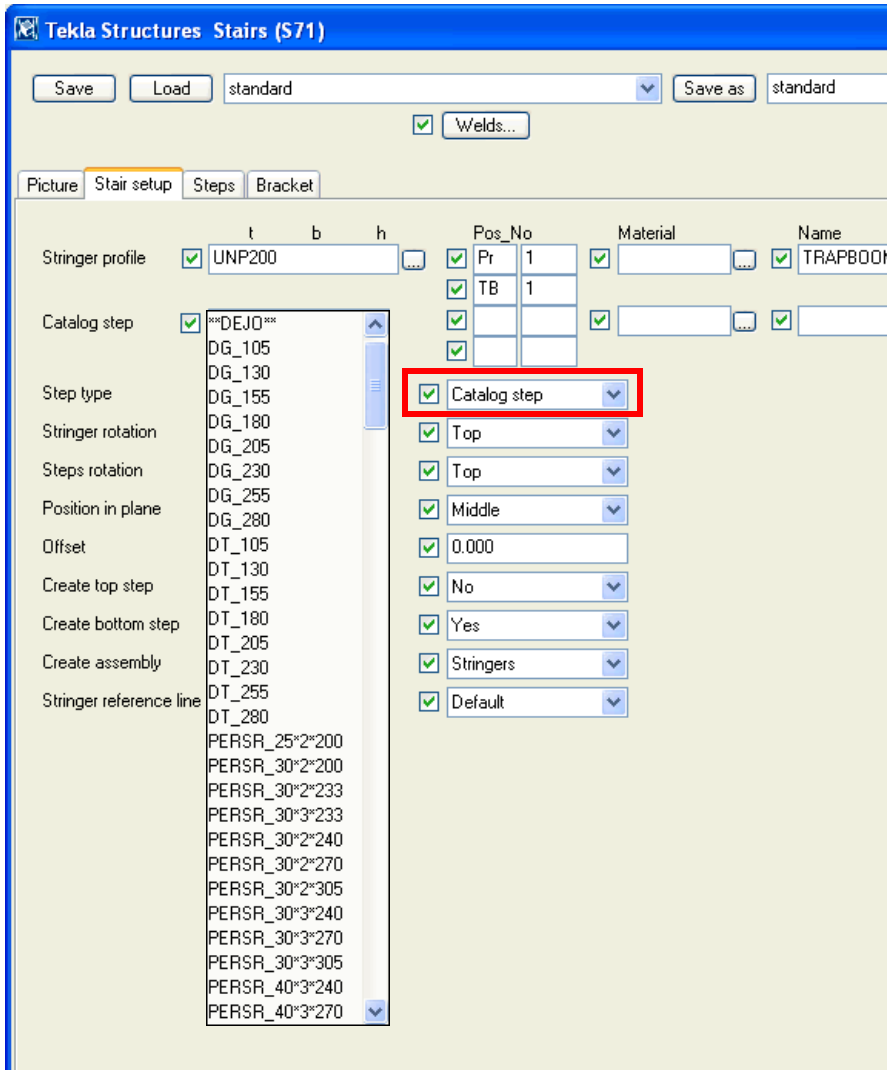
In some cases the field for anchor height (tab **Anchors**) was missing. This has been solved:



Component Stairs (S71)

In component Stairs (S71), you can use the same step profiles (Dejo, Arco, Thielco, ect.) as are used in component Stairs (S82).

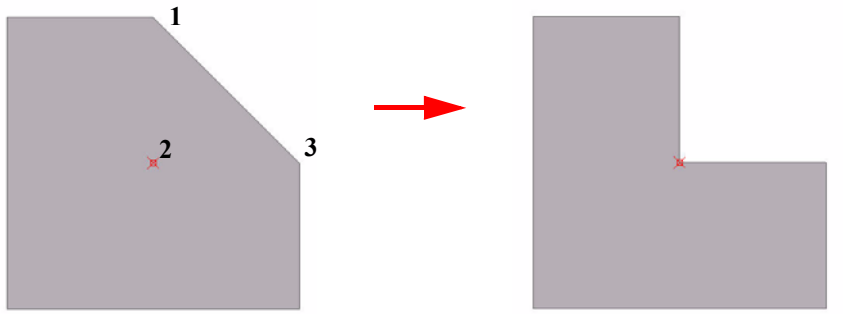
To do this, set picklist **Step type** to **Catalog step.** Now the step profiles list will be available.



Editing a contour plate or a polybeam

Adding a corner

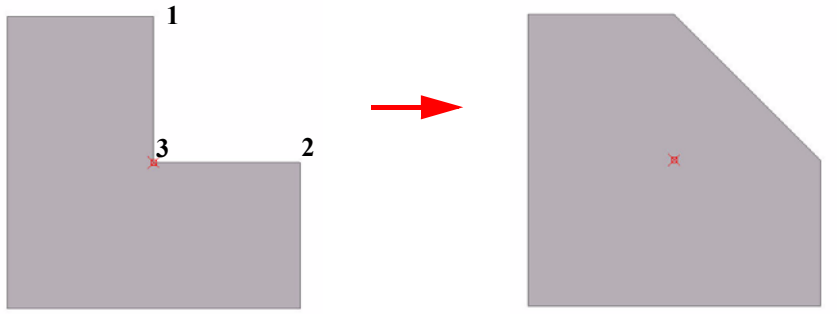
Adding a corner to a contourplate goes as follows::



1. Select the contour plate
2. Go to **Edit** > **Polygon shape**
3. Pick the corner positions (1 - 2 - 3)

Removing a corner

Removing a contour plate corner goes as follows:



1. Select the contour plate
2. Go to **Edit** > **Polygon shape**
3. Pick the corner positions (1 - 2)
4. Pick the corner to be removed (3)

Assembly: set new mainpart

Defining a new part as main part of an assembly Tekla Structures 12.0 is easier to do than before.

Somtime the main part changed in a copied assembly. Also the assembly prefix could change, which meant that the assembly got a new assembly number.

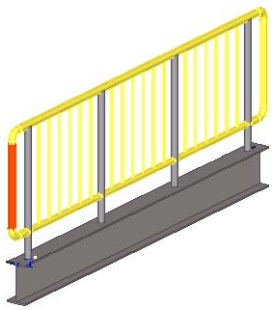
In previous versions, the only option to change the main part was by re-weld some or all parts, often prior to removing all welds. This was a time-consuming and complex activity.

Now the assemblies' main part can be (re-)defined very easy through a new option in the parts' contextmenu: **Set as new main part of assembly**:

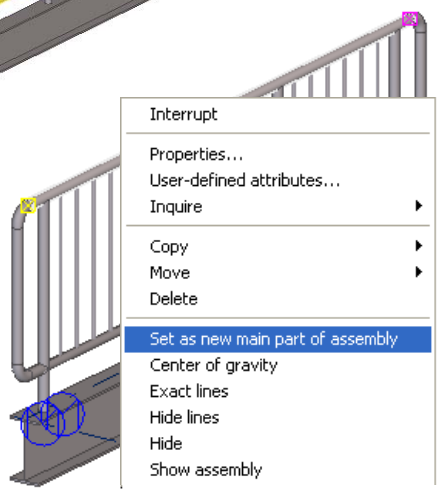
Step plan:

1. Select the part you want to become the assemblies' main part.
2. Rightmouseclick and select **Set as new main part of assembly**
3. Now, the selected part has become the new main part of the assembly.

Example

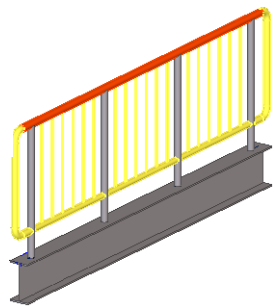


1. Original sitiuation: The vertical railing-part is the main part of the asembly.



2. Select the horizontal railing-part > RMK > Define as new main part of the assembly.

3. The horizontal railing-part is now the new main part of the assembly.





Now, when copying an assembly, the main part remains the same part!

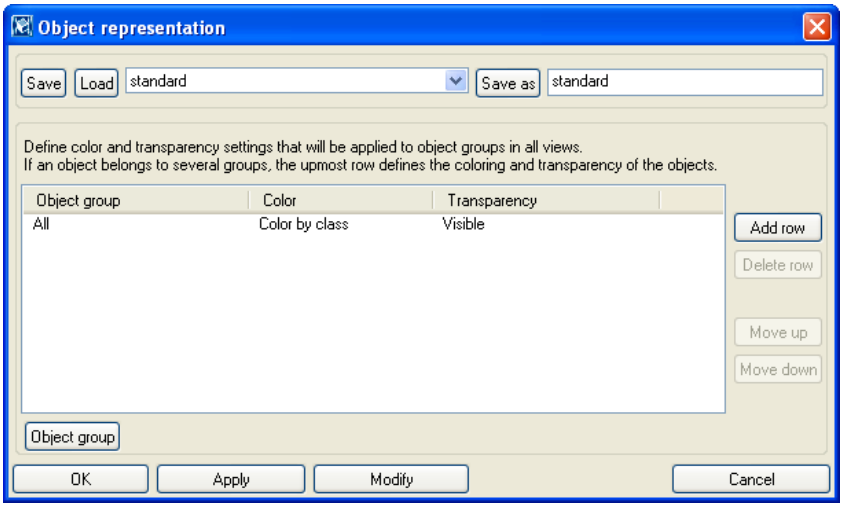
Object representation manager

An object representation manager has been developed to define the colors and transpareancy of parts. This Object representation manager replaces the old way of assigning colors (dialog **Colors**).

You can use the Object representation manager for the following:

- Change the color of objects in model views
- Change object visibility in model views
- Define new or modified ***object groups***

To open dialog **Object representation**, click **Setup** > **Object representation...**. The following dialog shall appear:

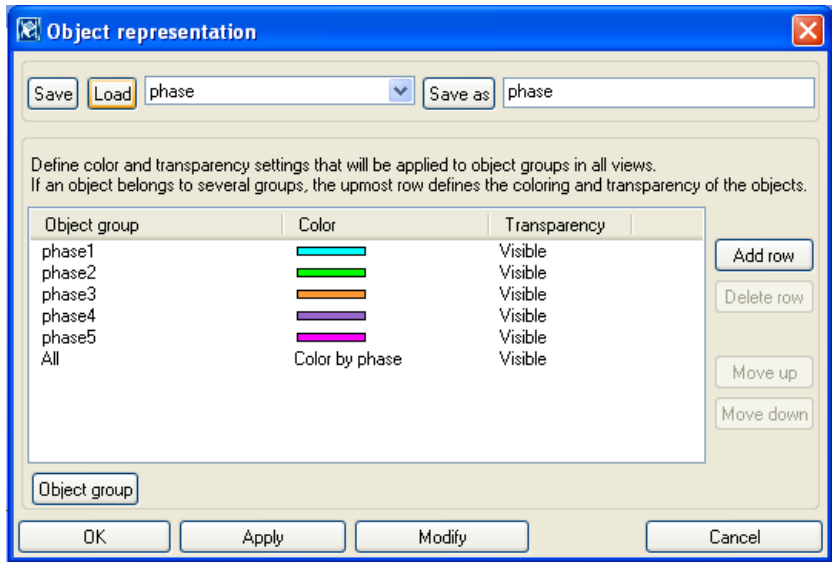


By default, 1 object group is defined: **All - Color by class.** This means that all profiles are colored according to their defined class.

A number of predefined settings are available. For example, by loading setting **phase**, 5 Object groupen wil be added.

When pressing the <**Modify**> button, the colors of the profiles will be modified in a way that they are 'grouped' per phase.

At the bottom, the **All** object-group remains. This is important, because the Object representation manager starts "assinging" colors at the top of the list. For a good working, it is essential that the most specific Object groups are on top of the list and the most common Object Groups at the bottom.



You can create extra Object groups for defining unique colors for parts, or to display parts with a certain degree of transparency, or to hide parts.



A big advantage of the Object Representation manager is that a setting will be applied for **ALL** opened model views.

Example: if you want to hide a phase from the model, you can define this by putting that phase group to "Hidden" (column Transparency).

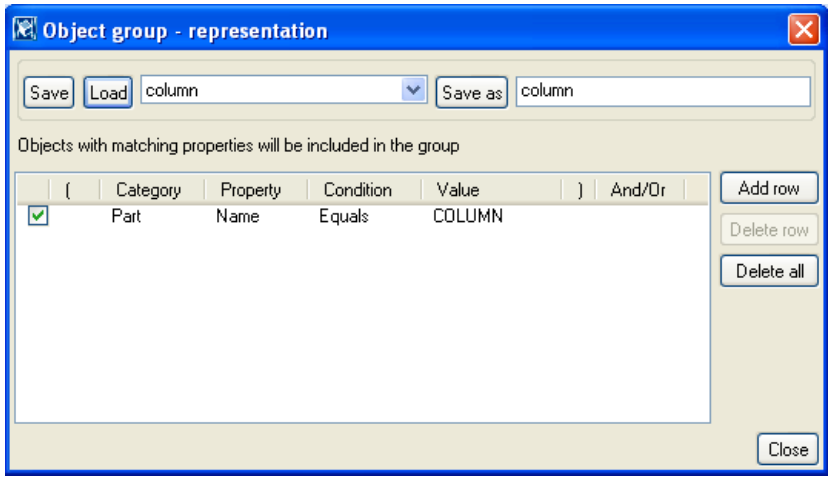
In previous versions you had to set View Filters for every single view to hide the phase in the model.

Example:

We will create a new Object Groep where all columns will be represented in blue - 70% transparency.

- In dialog **Object Representation**, click **Add row**. A second Object group will be added, this group initially has the same properties as the first Object group.
- Click **All**. A picklist will appear containing a number of options. These are the default options for Object groups. There is no option for hiding columns, so pick **Create new group...**
- Dialog **Object group - representation** appears.
- Leave the **Category**-option to Part.
- Leave **Proprty** to **Name**, we will use the partname to select columns.
- Leave **Condition** to **Equal to**.

- For **Value**, enter COLUMN (note: Capitals!). The dialog should be defined as follows:



- Save the setting as **column** to be able to re-use the new Object condition in other models.
- Click **Close**.

Object representation: limitations

During tests it turned out that a value defined in the **Shorten**-option in User defined attributes for bracings is not effective in Tekla Structures 12.0.

This means it is not possible to color profiles with a specific Shorten-value.

If you want to know which profiles do have a Shorten-value, you can use modified report **id_partlist**. A column is added in this report which represents the Shorten-vlaue. Because of the characteristic of id-reports is will be easy to locate profiles with Shorten-values.

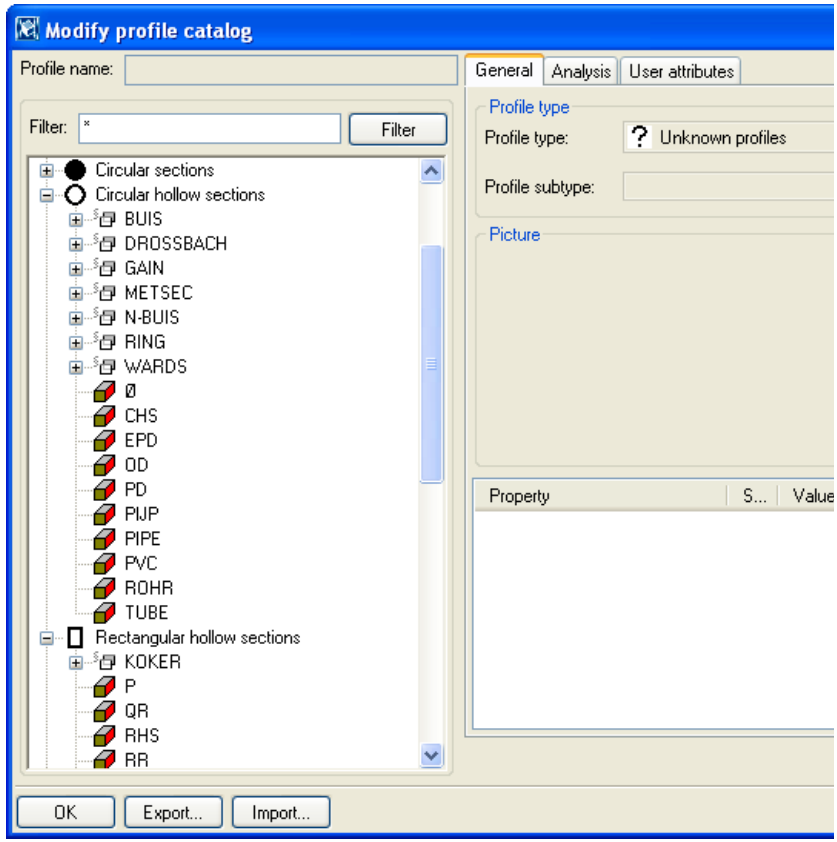
See also

For more information about id reports in Tekla Structures, see **Tekla Structures.pdf** in folder: **disc:\TeklaStructures\version\environments\europe\pdf.**

Profile catalog: catalog- and parametric profiles merged

The structure of the profile catalog is changed. The catalog and the parametric profiles are now stored in one list.

Colors are used to indicate if a profile is a catalog profile or a parametric profile: a blue icon represents a catalog profile, a red icon is a parametric profile.



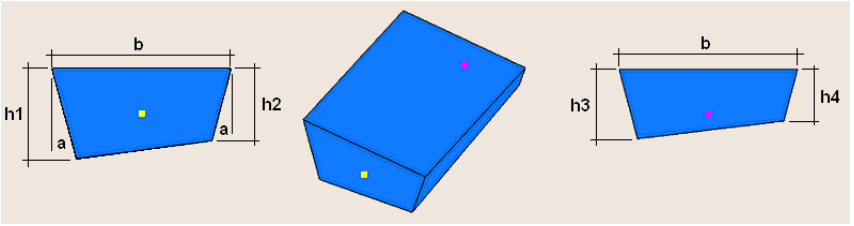
New parametric profiles: GUTHER and AFSCHOT

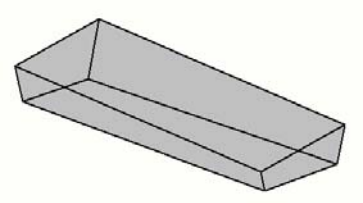
Two new parametric profile sections are added: Guther and Afschot. Both profiles can create a slope.

The difference between the 2 profiles:

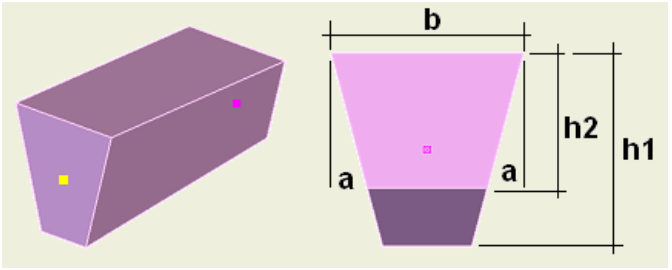
- for the AFSCHOT-profile, the height of every singlecorner can be defined
- for the GUTHER-profile, only the height of the two ends can be defined

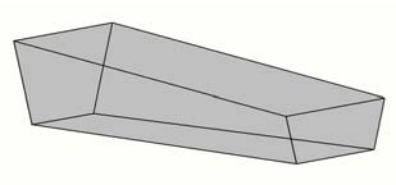
Afschot





Guther





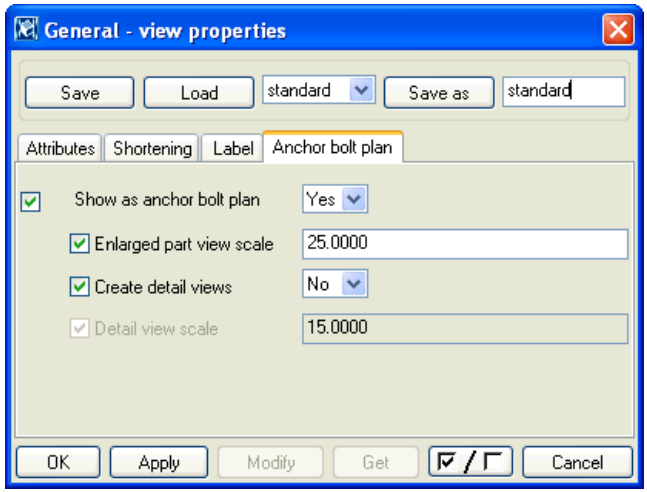
Drawings

Anchor bolt plan on a general arrangement drawing

In Tekla Structures 12.0 it is possible to combine two scales in anchor bolt plans in general arrangement drawings.

To enable this feature, go to dialog **Genaral arrangemant drawing properties** > **View** > **Anchor bolt plan**.

Set picklist **Show as anchor bolt plan** to **Yes**:

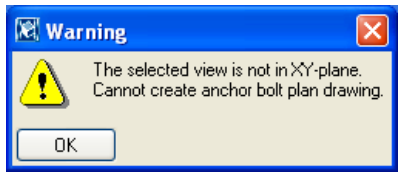


Also the scale of detail views can be set in this dialog.

Tekla Structures expects that only 2D top views will be placed on the drawing.

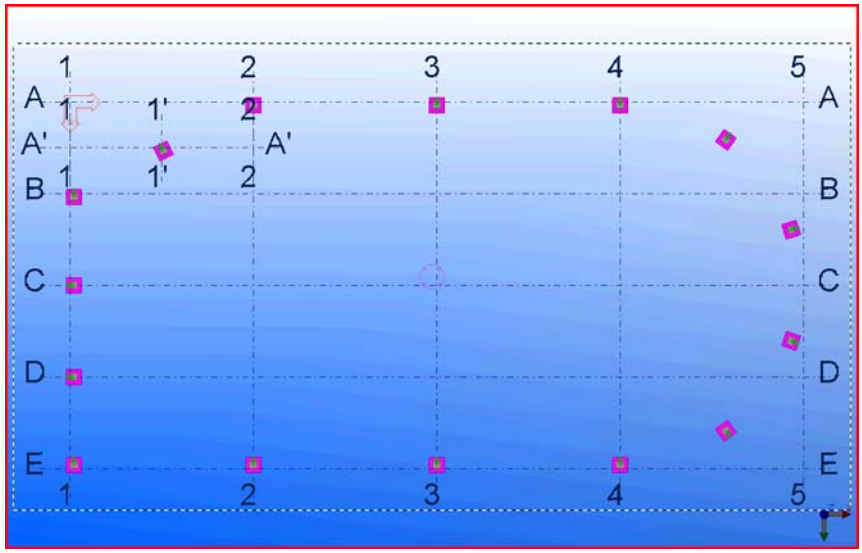
Warning

If no 2D anchor bolt plan view is selected, but a 3D view or a 2D Axis view, the following warning will be displayed:



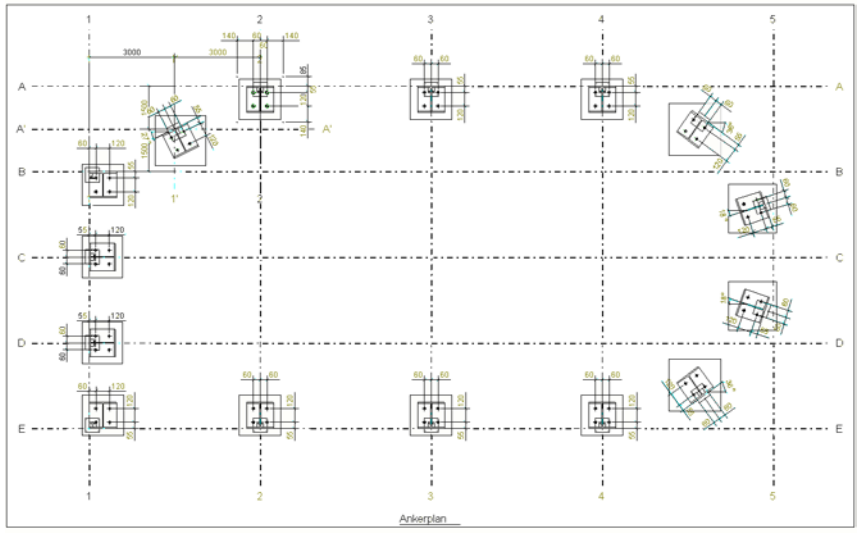
Click <**OK**> to create the drawing. Select a 2D anchor bolt plan view and the drawing will still be generated.

Model



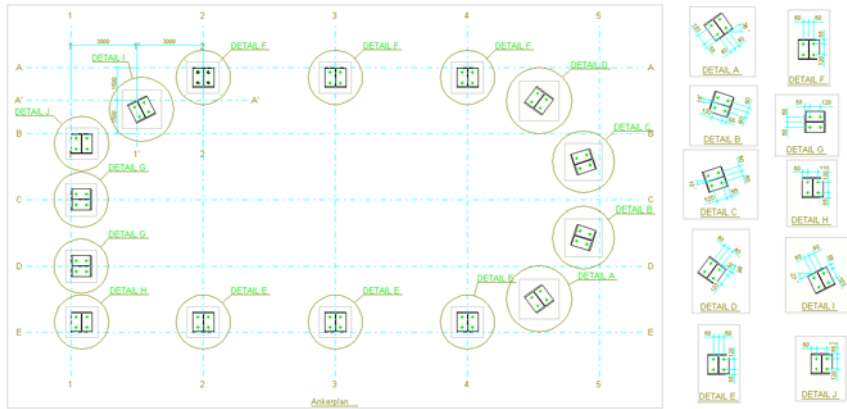
Drawing

Grids 1:100, column/ base plates 1:25:



Drawing

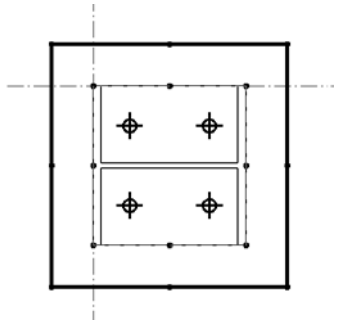
Grids 1:100, column/ base plates, details 1:25:



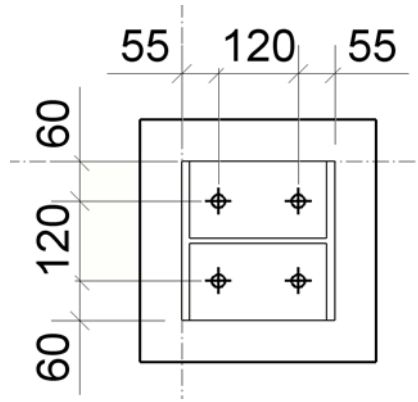
If you add a view to the anchor bolt plan drawing which is created with 2 scales, the view plane must also be in the XY plane!

Dimensioning anchor bolt plan

If you use 2 scales, you have to select the view first before you can start dimensioning the view:



Frst, select view,



then start dimensioning

In Tekla Structures, you can not dimension from one view to another view. This means, if you want to dimension the bolts/ holes, you have to use a 1-scale view.

Diagonal check dimension after cloning incorrect

In the drawings in Tekla Structures 12.0, the diagonal check dimensions are incorrect after cloning. This is caused by the slope of the dimension. This slope is not adjusted to the dimension and not adjusted to the new situation on the cloned drawing. The calculated dimension in the dimension line is not correct.



This dimension line must be deleted and replaced by a new diagonal check dimension !!

Example

Frame w*h: 1500*500. A check dimension has been added: 1118mm.

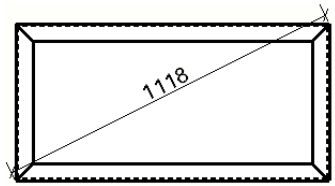
Then there is a second, similar frame, w*h = 2500*750.

The drawing of the small frame is cloned to the larger frame. The check dimension has a different value, but the value is not correct.

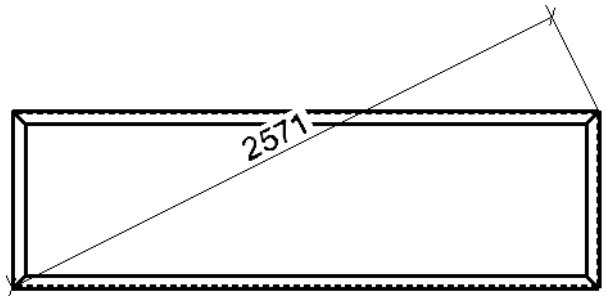
This because the slope of the dimension line has not been adjusted to the new shape of the frame, but the slope is identical to the original drawing.

To correct this you need to remove te dimension line and create a new check dimension. See example below:

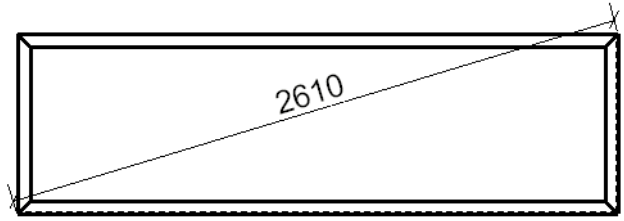
Original frame



Cloned drawing: check dimension incorrect! (slope is identical as in original)



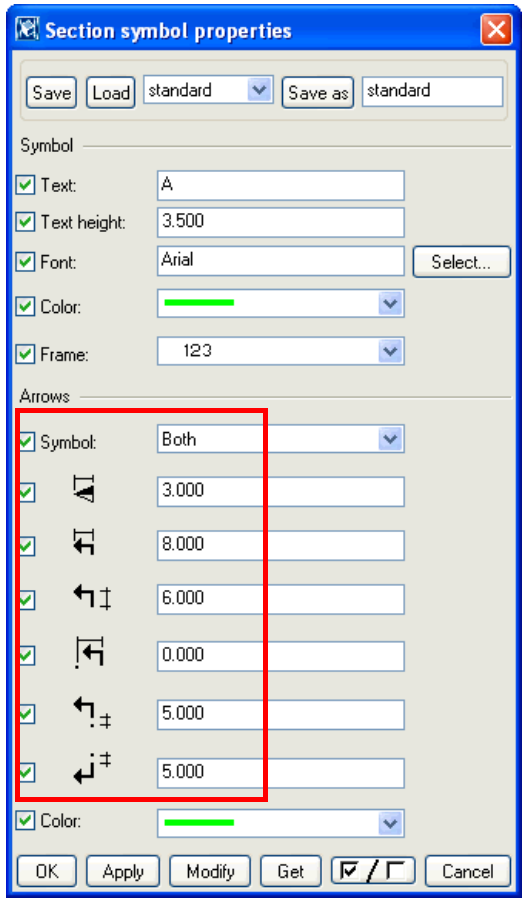
Cloned check dimension removed en replaced by a new created check dimension



Improvements in cut symbols

New options in dialog **Cut symbol properties** allow you to define the offset and size (length and heigth) of cut symbols:

The variable ALIGN_CUT_SYMBOL was used to control this offset: exactly on the location of the cut or a little offset from the cut position.



Old settings in Tekla Structures 12.0

These new options are important if you want to make use of drawing settings from an earlier version in Tekla Structures 12.0: these new options were, of course, not defined in the 'old' drawing properties.

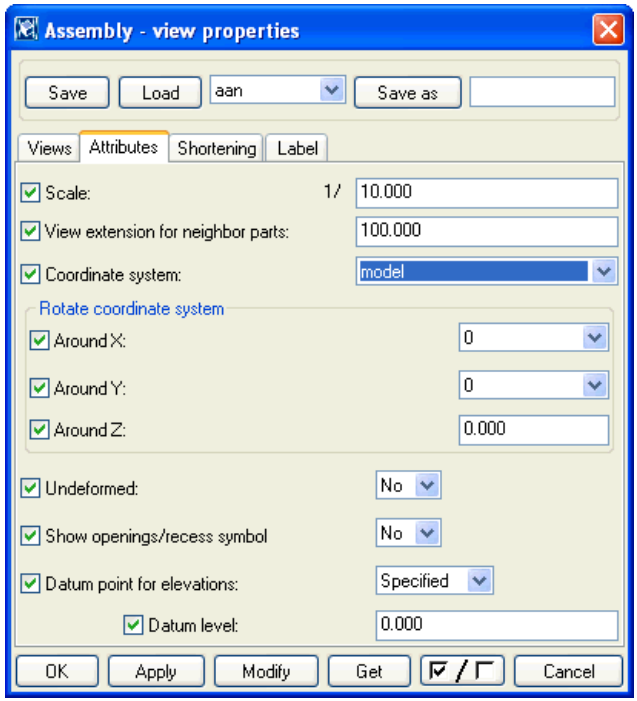
Because of that, these values will not be adjusted in the new fields when you load an 'old' setting in Tekla Structures 12.0.

So it is important to check your 'old' settings and overwrite them if necessary.

Construsoft advises to, if you make use of 'old' drawing settings (assembly- and single part) in Tekla Structures 12.0, you always should check these files and overwrite them if necessary. This is the only way to be sure that correct values are loaded in new fields.

Setting "Model" works correct again

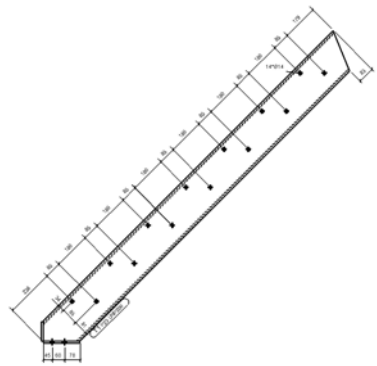
In Tekla Structures 12.0, the setting **Model** in **Coordinate system** picklist works correct again:



In Tekla Structures 11.x, this setting was affected by the rotation of the main part.

A possible result was that the part was displayed upside down on the drawing.

This has been corrected.



Saving en Freezing associative drawings

What is associativity?

All drawing types in Tekla Structures are associative, if they are created manually or automatically.

This means that drawing objects (labels, dimensions, welds, views and cut symbols) are linked to model objects. Because of that, drawing objects are updated automatically if the model changes.



In drawings, a nail symbol indicates an object to be associative. This symbol will not be printed. To hide associative symbols, click **Setup** > **Associative symbol** or use shortcut <**Shift** > + **A**.

Manual changes

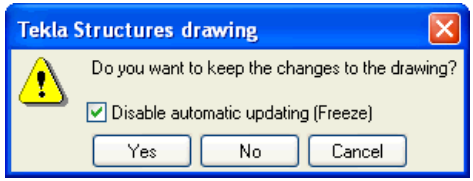
In previous versions, if you touched up a drawing manually, and later the model changed, the manual modifications to the drawing were lost for non-frozen drawings.

Freeze

In Tekla Structures 12.0, manual modifications will always be kept if the drawing is **not** frozen.

If you modify and save a drawing, Tekla Structures ask if you want to save the changes. Click <**Yes**>.

To prevent that a drawing will be updated automatically, check the option **Disable automatic updating**:



This will **freeze** the drawing. Freezing drawings causes drawing objects to be not associative anymore:



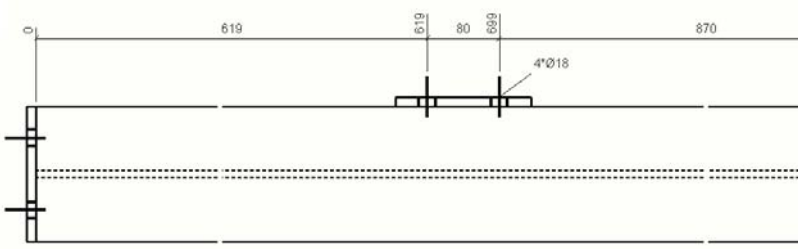
Frozen drawing: Not associative

Not frozen drawing: Associative

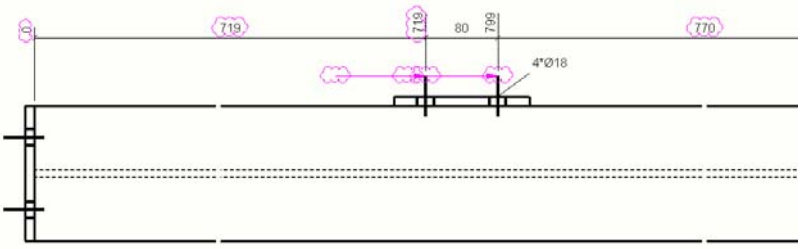
Example assembly drawing

Example

The drawing is **not** frozen:



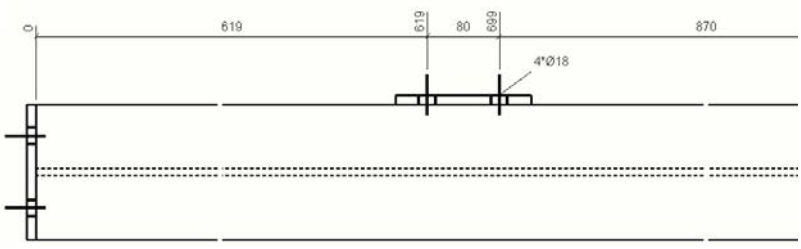
Before modification



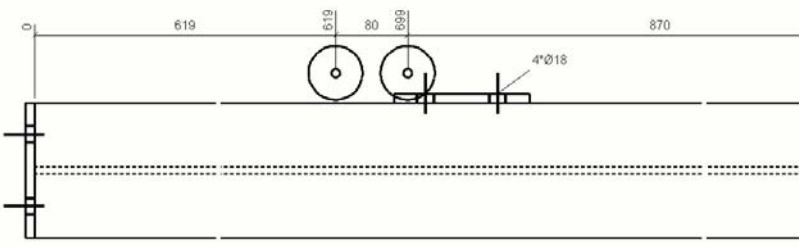
After modification

Example

The drawing is frozen:



Before modification

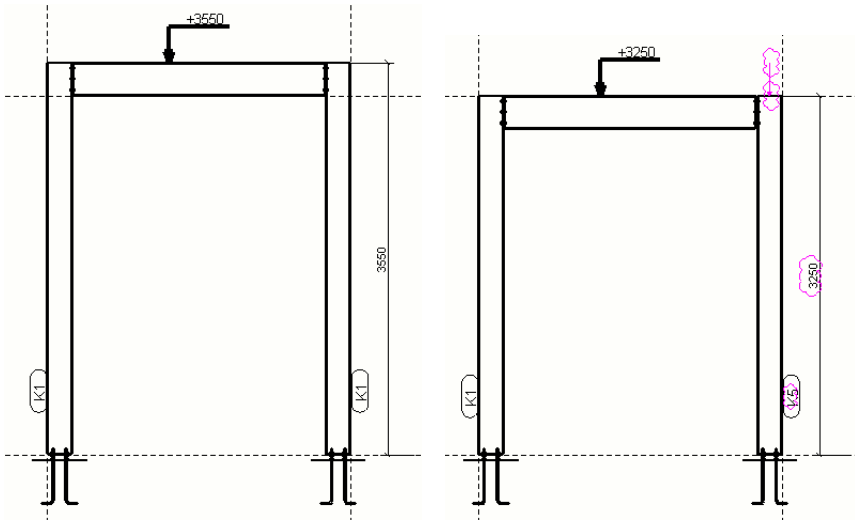


After modification

Example General arrangement drawing

Example 1

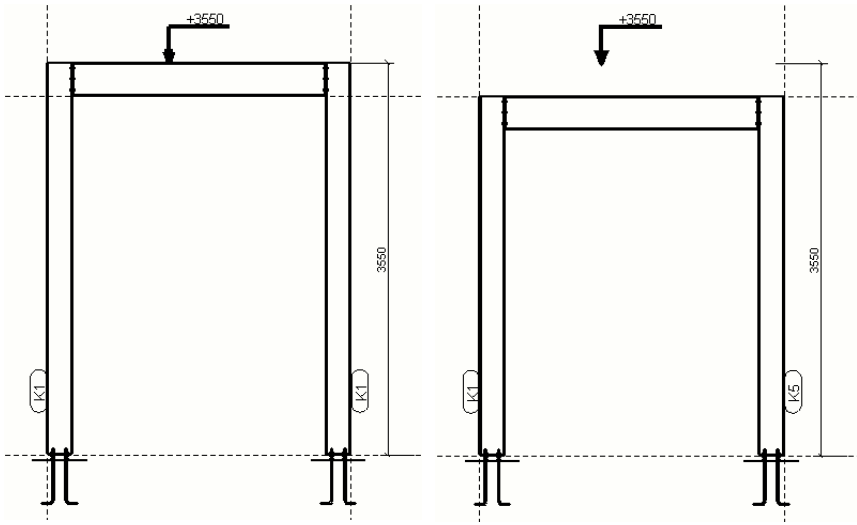
The drawing is **not** frozen:



Before modification | After modification

Example 2

The drawing is frozen:



Before modification | After modification

You see the level mark and dimension line are not updated, also the view size has not been changed.

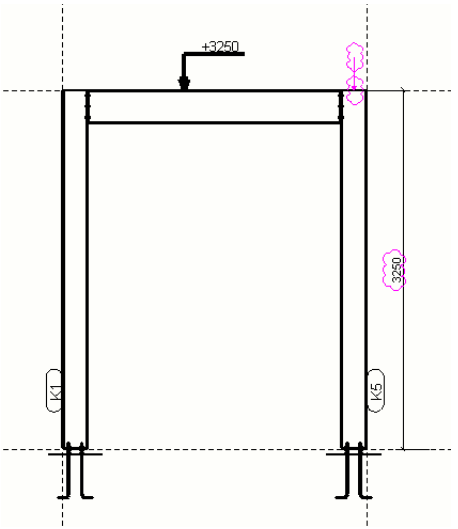
Updating a frozen drawing

You can still open a general arrangement drawing, unfreeze it, and then re-open it to automatically update the drawing with recent changes.

To re-enable automatic updating in a frozen drawing:

1. Go to **Drawing** > **Drawinglist...** to open the Drawing list.
2. Select the drawing.
3. Rightmouseclick and select **Freeze** > **Off**.

Now, clouds are placed to indicate the modifications:



See also For information about removing associative clouds, see **Indicate changes to associative dimensions and marks** (page 40)

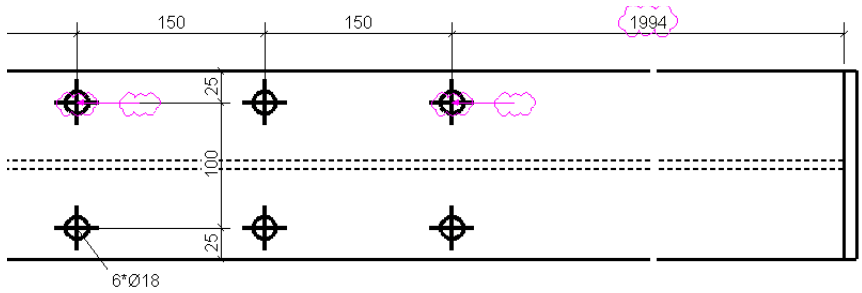
In previous versions, when you updated a single part-, assembly- or concrete drawing (Update) without freezing the drawing, the drawing was re-created. Now, the drawing is not being rebuild. This means you have to decide if changes should be displayed or not displayed when updating the drawing.

To rebuild a drawing:

1. Go to **Drawing** >**Drawing list...** to open the Drawing list.
2. Select the drawing you want to re-create.
3. Rightmouseclick select **Recreate**.

Indicate changes to associative dimensions and marks

Modified dimensions and dimension lines are indicated through clouds if you update associative drawings (single part-, assembly- and general arrangement drawings).



These clouds are not printed.



To remove clouds from part labels , go to **Edit** > **Remove mark change symbol** > **Single change symbol** or **All**, or click an icon.



To remove clouds from dimensions, go to **Edit** > **Remove dimension point change symbol** > **Single change symbol** of **Alles** of click an icon.

The 4 above mentioned icons for removing clouds are assimilated in toolbar **Drawings- Dimension** and are only available if you execute the correct Registry file.

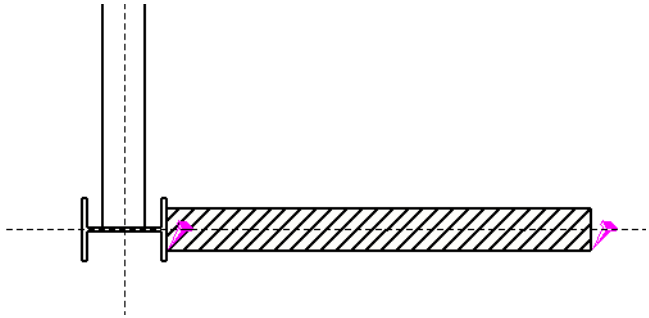


See also

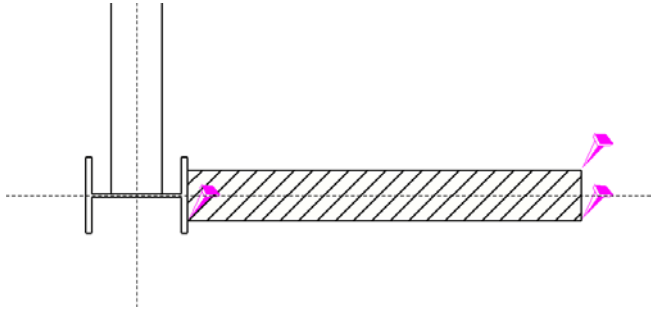
For more information about toolbars in Tekla Structures, see **Tekla Structures.pdf** in folder: **disc:\TeklaStructures\version\environments\europe\pdf.**

Associativity of graphical objects

The associativity of graphical objects has been improved in Tekla Structures 12.0. Graphical object are now updated correctly, see the pictures below (note the associativity symbols):



Previous Tekla Structures versions



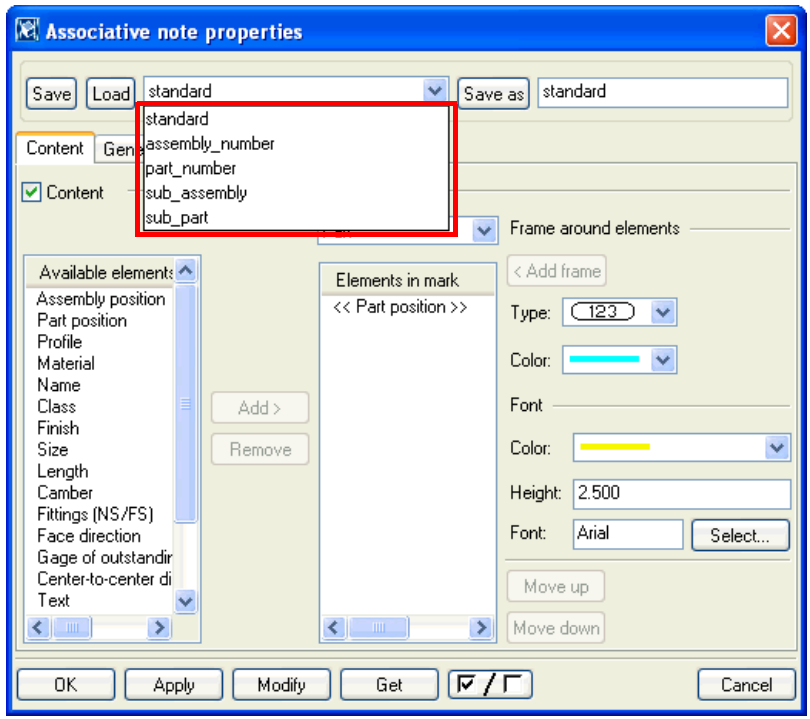
Tekla Structures 12.0

Associative notes

You can use associative notes in drawings as an additional marks. Associative notes are used like part marks and are updated automatically when a modification has been carried out to the object.

Step plan

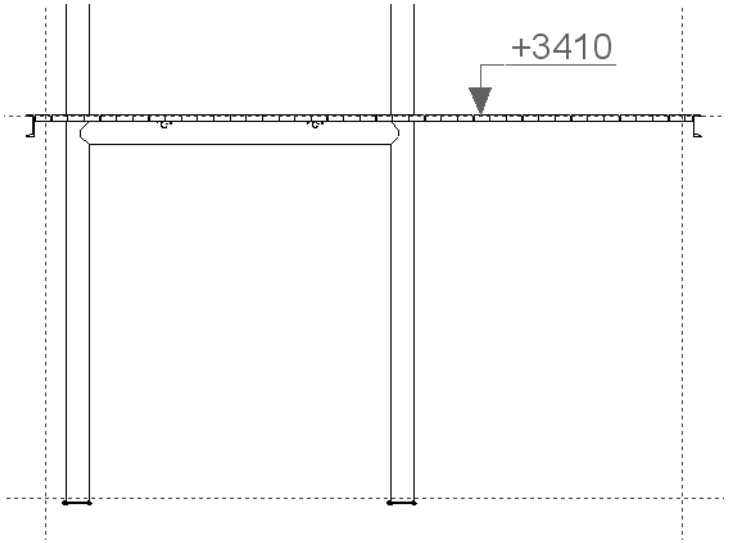
1. Open a general arrangement drawing
2. Go to **Properties** > **Associative note...**



3. Select a setting from the picklist
4. Click <**Load**> and click <**OK**>
5. Rightmouseclick and select **Associative note** or go to **Create** > **Associative note** > **With leaderline**, **Without leaderline** or **Along line.**
6. Pick part and position
7. The note is placed

Datum point for elevations

Tekla Structures determines the level marks in drawings using the reference points. These level marks can be manipulated in drawings. For example, if you want to display a level compared to Mean Sea Level (MSL) instead of the reference point level, you can set the difference between the MSL and the level of your model.:



Changing the reference point

1. In the Drawing Editor, open dialog **Drawing properties** and click **View**; or doubleclick a view go to tab **Attributes 2**.
2. Select the option **Specified** in picklist **Datum point for elevations:**

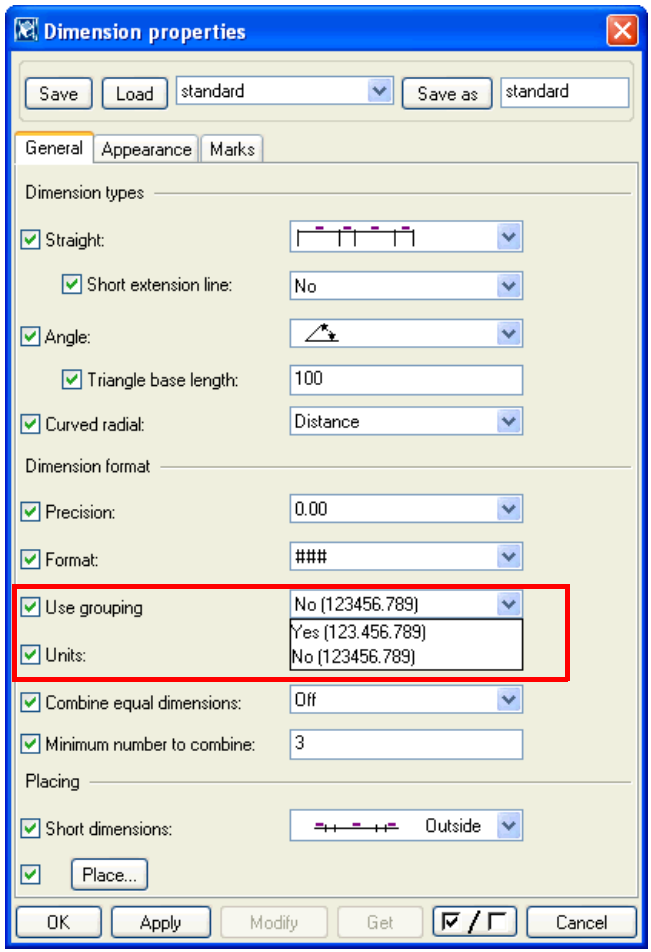


3. Enter a value in field **Datum level**.

More clear representation large dimansion values

You can now use several number notations for dimensions. This helps to show dimensions in a more clear way on workshop and assembly drawings. For example; a dimension of 15432 can be displayed as **15.432**, but also as **15 432** or **15.432.000**.

To do this, select option **Yes** in the **Use grouping** picklist (dialog **Dimension properties**):



Use the variables **XS_DIMENSION_DIGIT_GROUPING_CHARACTER** to set the separator and **XS_DIMENSION_DIGIT_GROUPING_COUNT** to set the number format.

From Tekla Structures 12.1 you can make use of a comma-sign (,) to also represent the value 15.432.000 as 15.432,000.

Custom component

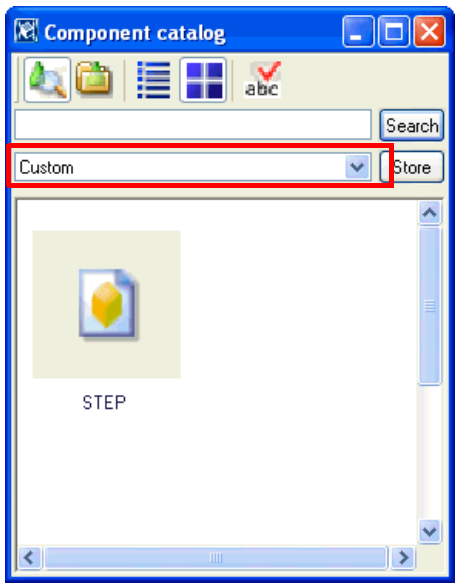
Availability of Custom Components and Sketched profiles

In Tekla Structures 12.0, all Custom Components and sketched profiles are available in all **new** models. This means you do not have to import them in every new model anymore.

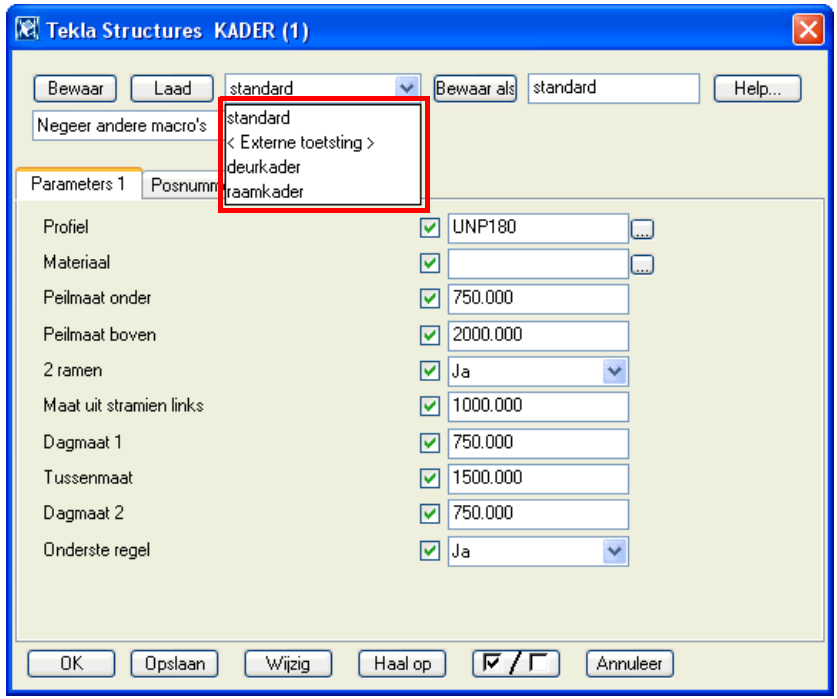
Custom component example

You want to have custom component "FRAME" and accompanying component settings to be available in all new models:

1. In Windows Explorer, go to the folder ***...europe\custom-components\steel*** and extract file **frame.zip**
2. Extract the file frame.uel into the folder ***...europe\custom-components\steel***, the three accompanying component-settings in folder ***...environments\europe\firm\ts***
3. Start Tekla Structures and create a new model
4. Click icon **Find a component** in the **Components**-toolbar, or use shortkey <**Ctrl + F**> to open dialog *Components catalog*
5. Set the picklist to **Custom** to display the Custom components



6. Now, doubleclick the custom component to load one of the settings from the picklist.

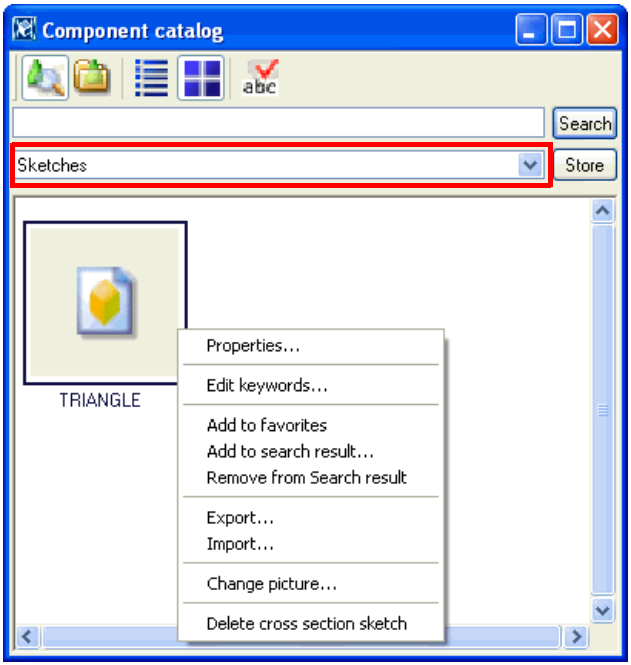


Example
Sketched profile

You have sketched a parametric profile, for example TRIANGLE100*100. You want this sketched profile to be available in all models:



1. In the model where the sketched profile has been created, click icon **Find component** in the **Components**-toolbar, or use shortkey <**Ctrl + F**> to open dialog *Components catalog*
2. Set the picklist to **Sketches** to display the Custom components



3. Click the sketched profile, select option **Export....** and save the sketched profile, for example as **triangle.uel**
4. Copy the file **triangle.uel** to folder ***...europe\custom-components\steel***
5. (Re-)start Tekla Structures and create a new model
6. Click icon **Find component** in the **Components**-toolbar, or use shortkey <**Ctrl + F**> to open dialog *Components catalog*
7. Set the picklist to **Sketches** to display the Custom components and you can see the sketched profile is available

Tekla Structures 12.1

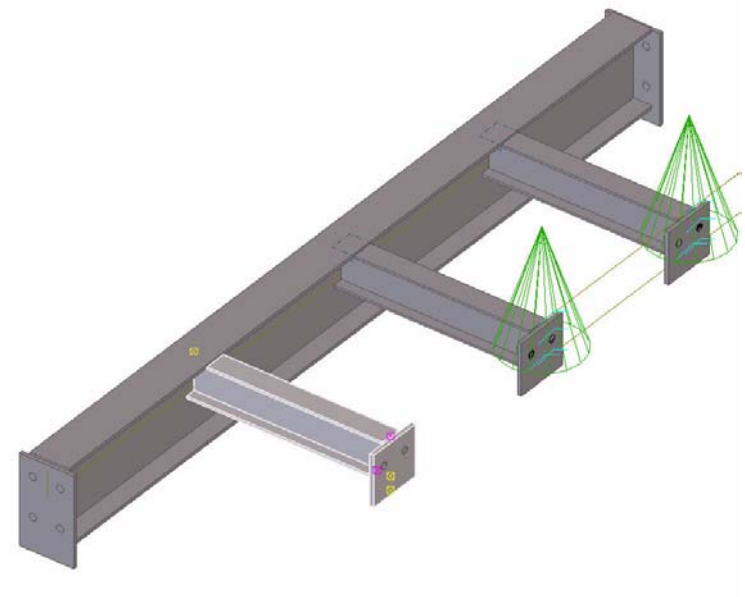
Multicopy

From Tekla Structures 12.1, you can use new functionality named Multicopy.

Using the <**Ctrl**>-key you can quickly create a copy of a drawing object (dimension, text, symbol, DWG ect.). You do not enter a translation distance.

Step plan

1. Select the object to be copied
2. Hold down <**Ctrl**>-key
3. Place the mousepointer on the object to be copied and drag it to the new position
4. Release the <**Ctrl**>-key. A copy van het object staat op nu de nieuwe positie.

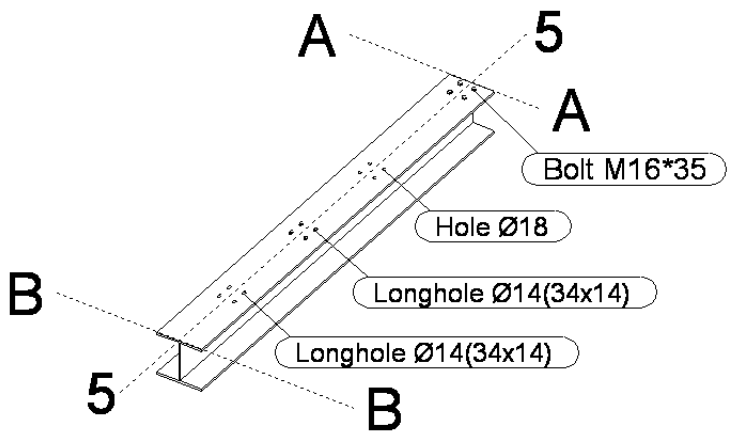


Speed improvements

In Tekla Structures 12.1, a large number speed improvements have been implemented which help you to work faster.

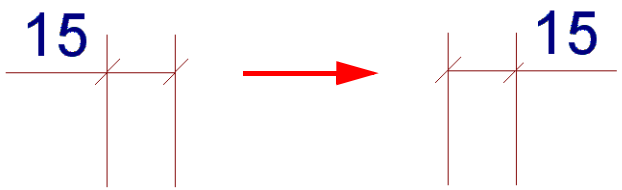
Bolt/ hole marks

The definition of bolt/ hole marks in general arrangement drawings is now correct:



Short dimensions: switch sides

In Tekla Structures 12.1 you can switch the side of the dimension value in short dimensions. This way you do not need to re-place the dimension:

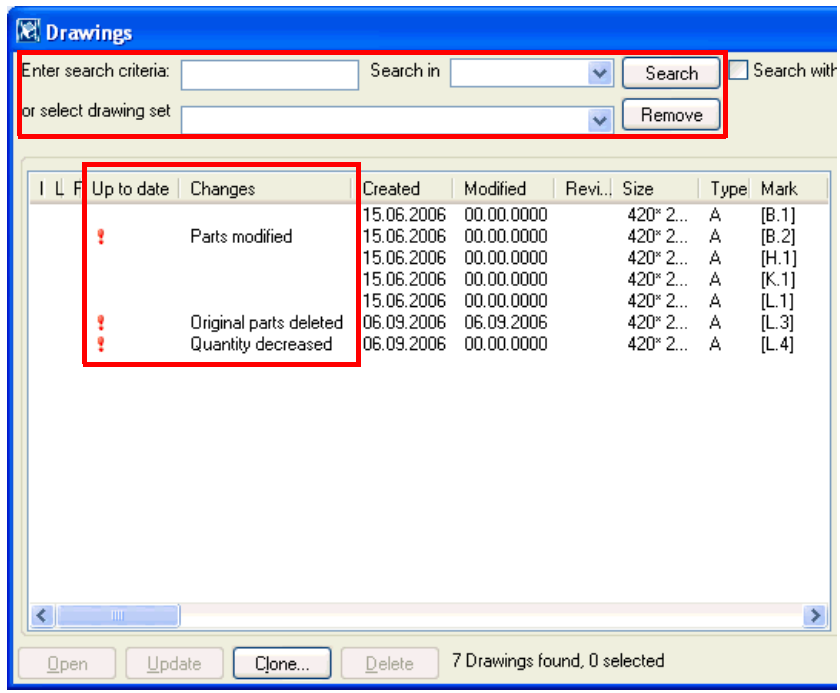


Tekla Structures 13.0

Drawing list

In Tekla Structures 13, the Drawing list will be radical modified and improved.

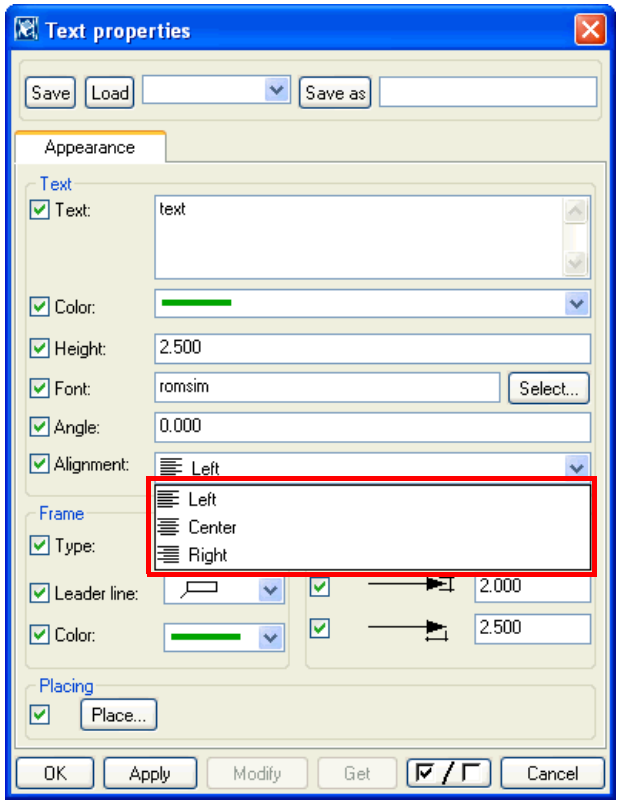
U can use a search function to filter workshop drawings or only drawings from a certain phase:



Model changes are idicated in the Drawing list with an exclamation mark (!).

Aligning Multi text

In Tekla Structures 13 you can align text in drawings (Left - Center - Right):



Notes:

9 Construsoft Release Info 12.1

Content This document about Tekla Structures 12.1 contains the following topics:

General

Releasing Tekla Structures versions (pag. 3)
Xtranet (pag. 3)
Folder structure in Tekla Structures 12.1 (pag. 4)
Shortkeys and the Construsoft toolbar (pag. 5)
Manage variables in the file options.ini (pag. 9)
Restore view settings in the Model Editor (pag. 12)

Modeling

Change the color of the grid (pag. 13)
Single bolts (pag. 14)
Workarea invisible (pag. 15)
Keys X, Y and Z (pag. 16)
Reference models (pag. 17)
Selecting (pag. 19)
Comment in user-defined attributes (pag. 21)
Modify user-defined attributes in the file objects.inp (pag. 21)
Improvements in copying and moving (pag. 24)
Changes in view and select filter (pag. 26)
Object representation manager (pag. 37)
Numbering (pag. 37)

New parametric profiles: TRZB and TRIA (pag. 38)
Haunch (40) improvements (pag. 38)

Drawings

Saving en Freezing associative drawings (pag. 39)
Representing bolts on GA drawings (pag. 39)
Classifier: No dimensions for zinc coating holes (pag. 40)
Associative notes (pag. 41)
Change the side of outside dimensions (pag. 42)
Improvements in leader line appearance (pag. 42)
New options in presenting section and detail view symbols (pag. 44)
Bolt/hole marks (pag. 46)
Grid label frame size (pag. 47)
Template pathname (pag. 47)
Templates "bottom side baseplate" and "top side castplate" added (pag. 48)
Plotter catalog (pag. 49)

Custom components

Autoplacing for custom component "part" (pag. 51)
Availability of Custom Components and Sketched profiles (pag. 52)

NC files

Variabele XS_DSTV_PRINT_NET_AND_GROSS_LENGTH (pag. 54)
NC files for plates (pag. 55)

Tekla Structures 13.0

Drawing list (pag. 56)
Bind handles to a plane (pag. 58)
Speed improvements while working with large construction circles (pag. 58)
Speed improvements while opening a large compont catalog (pag. 58)
Scale in views (pag. 59)
Aligning multiple lines of text (pag. 60)

General

Releasing Tekla Structures versions

From now on, Tekla releases two versions per year. The global planning for Construsoft will be as follows:

TS 12.1 February 2007.

TS 13.0 April/May 2007, depending when version will be final. Is only for customers who really want that version or if we think that it is usefull to work with that version (e.g. concrete customers).

TS 13.1 After the summer holidays: last week of September.

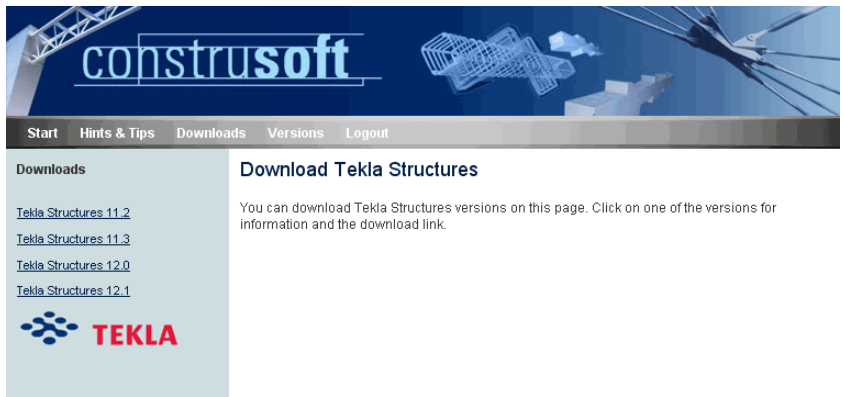
TS 14.0 Spring 2008.

TS 14.1 After the summer holidays 2008.

Xtranet

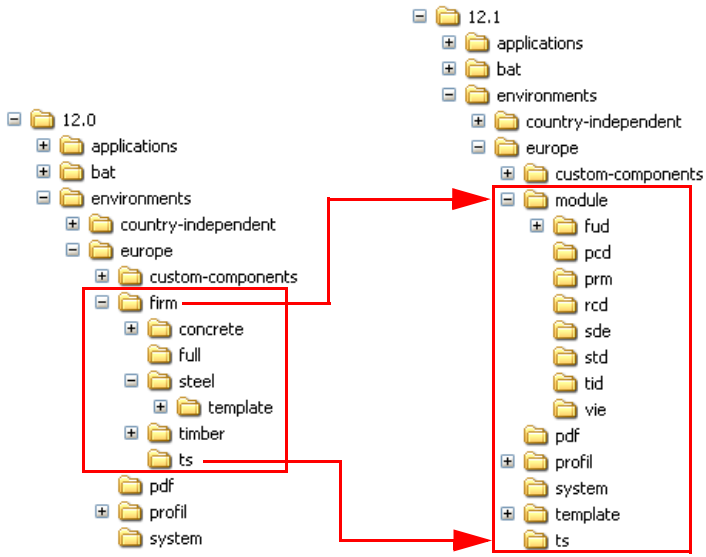
On the Construsoft website (www.construsoft.com), all customers can log on to the Xtranet section. All customers can submit a request for this. After submitting this request you will receive an email to indicate that you have been added to the database. Now you will have access to the Xtranet section; in this section are several useful documents and files available for download. You will also be kept informed to new downloads, changes and warnings by e-mail.

Also information about Frequently Asked Questions will be available. So log on frequently!!



Folder structure in Tekla Structures 12.1

The folder structure has been changed compared to Tekla Structures 12.0.



Tekla Structures 12.0 Tekla Structures 12.1

This to be able to differentiate the various software configurations, which are used by customers, such as STD (Steel Detailing) and SDE (Standard Design). The following modules can be used:

- Standard Design (SDE)
- Engineering (SDE-AD)
- Project Management (PRM)
- Steel Detailing (STD)
- Timber Detailing (TID)
- Precast Concrete Detailling (PCD)
- Reinforced Concrete Detailing (RCD)
- Full Detailling (FUD)

Specific settings for e.g. STD customers are now stored in folder `...europe\module\std`.

Also the `ts` folder has been moved and is now situated directly under folder `...europe`.

Shortkeys and the Construsoft toolbar

Depending on the used software configuration, you can choose for automatic adding of shortkeys to the Windows Registry, each time you start Tekla Structures 12.1.

This can be useful if you use multiple configurations, e.g. Steel Detailing and Standard Design.

Because of this, you always have icons and toolbars which are concentrated on a specific module and you can use the shortkeys, which are supplied by Construsoft, optimal.

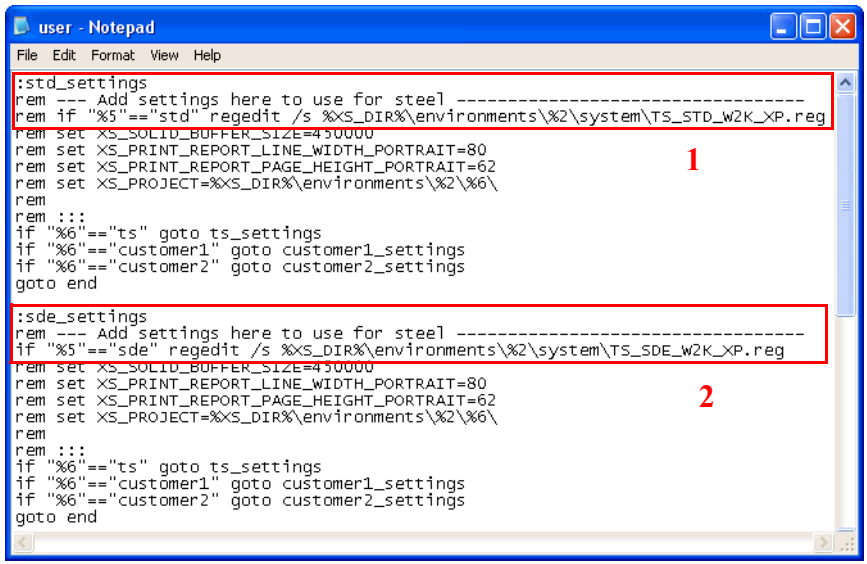
Also the toolbar Construsoft is now available in which often used commands are added.

If you do not add the Registry file (*.reg), in which the shortkeys are added, to the Windows Registry, the following icons appear on your screen when you start a module, not having access to some commands:



Adding a Registry file to the Registry goes as follows. Everytime you start version 12.1, depending on the module you use, the content of the file **user.bat** will be used from the folder *disc:\TeklaStructures\12.1\bat\user.*

By default the folder **user.bat** contains among other things the following lines:



When you start the module SDE, for example, the settings `sde_settings` will be used (see above).

This setting contains the line

if "%5"=="sde" regedit /s %XS_DIR%\environments\%2\system\TS_SDE_W2K_XP.reg

which will add the file `TS_SDE_W2K_XP.reg` to the Windows Registery.



For the module Steel Detailing, this option is swiched **off** by default. This has been done to protect customers, who only work with the module STD, from overwriting their customized settings wich include their own shortkeys and toolbar.

If you want to add the file `TS_STD_W2K_XP.reg` to the Windows Registery, set this line.

Several shortkeys in Tekla Structures 12.1 have been changed. Below is a list for both Model- and Drawing Editor:

Model Editor

	Command		Command
a	Customizing toolbars	**m**	Move > Translate
c	Copy > Translate	**n**	Numbering > Full
e	Open model folder	**o**	Ortho
f	Phasemanager	**q**	Create clip plane
g	Redraw all	**s**	Smart Select
h	Rollover highlight	**t**	Xsnap
i	Inquire assembly	**u**	Update all
k	Inquire object	**v**	Pick rotation point
l	Drawing list	**w**	Wizard

	Command		Command
1	Create single-part drawing	**3**	Create GA drawing
2	Create assembly drawing	**4**	Create cast unit drawing

	Command		Command
Enter	Repeat last command	**Alt + p**	Print
Alt + Enter	Properties	**Back space**	Undo last polygon pick
Home	Zoom original	**Spatie balk**	Finish polygon input

	Command		Command
Ctrl + b	Named views	**Ctrl + m**	Move
Ctrl + c	Copy multi	**Ctrl + p**	2D / 3D
Ctrl + f	Find a component	**Ctrl + w**	Weld

	Command		Command
Shift + a	Select all objects	**Shift + k**	Colomn properties
Shift + b	Bolt	**Shift + l**	Beam properties
Shift + c	Copy > With 3 points	**Shift + m**	Move > With 3 points
Shift + f	Fitting	**Shift + z**	Line cut
Shift + h	Construction line		

Drawing Editor

	Command		Command
a	Customizing toolbars	**l**	Drawing list
b	Black and white drawing	**m**	Move objects
c	Copy objects many times	**o**	Ortho
e	Open model folder	**s**	Smart Select
f	Phasemanager	**t**	Xsnap
j	Close drawing	**u**	Update all
k	Inquire > Object	**z**	Zoom in

	Command
Ctrl + b	Open named view list
Alt + Enter	Properties
Home	Zoom original
Enter	Repeat last command
Ctrl + Page Up	Open previous drawing
Ctrl + Page Down	Open next drawing
Shift + A	Associative symbol

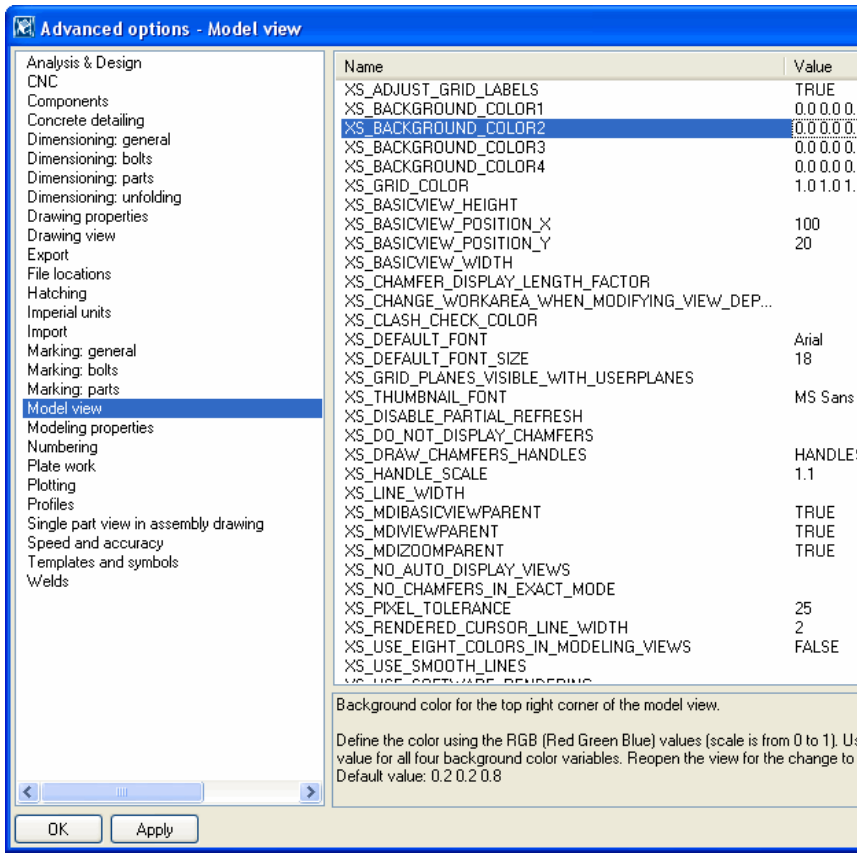
Manage variables in the file options.ini

To allow easier managing variables in Tekla Structures, the variables can now be modified at two levels:

- Model level
- User-specific level

You don’t need to change variables in the batch files anymore!

To view or change variables, click **Tools** > **Advanced options...** to open the Advanced options dialog box:

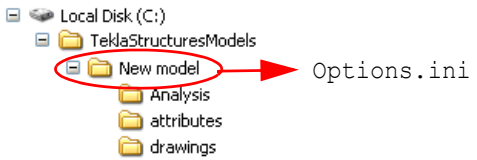


Variables are grouped together in categories. The left pane in the dialog box displays the list of categories. Click a category to view or change the variables it contains in the right pane

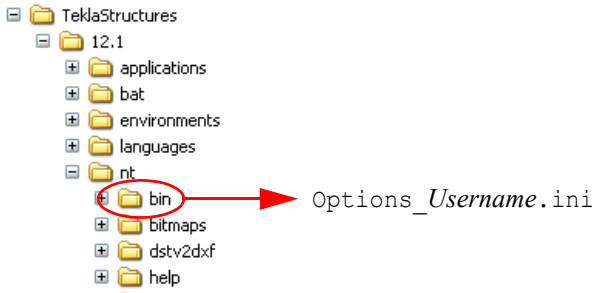
Some variables contain a discription to clarify the effect of that variable.

Changes you make are stored as follows:

1. This file contains modified variables at a model level.



2. This file contains modified variables at a user-specific level.



Method of working

All `options.ini` files inTekla Structures will be used consecutively. So when you have a specific line in several `options.ini` files, the line in the file with highest priority, in order of files, is being used.

The priority of the file `options.ini` in order of files is as follows:

1. Model folder: highest priority and will be used as last
2. The folder `ts`
3. The folder `std` (or another option from the module folder, e.g. `sde` or `pcd`)
4. The folder `system`

It is still possible to define variables in the batch files. Take into account that, when variables are also defined in the Advanced options, the following order of files will be used:

1. Advanced options (model folder) (highest priority)
2. Advanced options (the folder `ts`)
3. Advanced options (the folder `std` oor another option from the module folder, e.g. `sde` or `pcd`)
4. Advanced options (the folder `system`)
5. user.bat
6. europe_env.bat

Model dependig variables

Example

A model depending variable, for instance, is

XS_USE_ASSEMBLY_NUMBER_FOR=

Set this variable to assign the assembly number to the main parts of an assembly.

```
----------------------------------------
TEKLA STRUCTURES ASSEMBLY PART LIST
Model:


----------------------------------------
Assembly     Part      Number Profile
========================================
B1           B1            1   IPE300
----------------------------------------
             B1            1   IPE300
========================================

----------------------------------------
```

MAIN_PART

```
----------------------------------------
TEKLA STRUCTURES ASSEMBLY PART LIST
Model:


----------------------------------------
Assembly     Part      Number Profile
========================================
B1           Pr4           1   IPE300
----------------------------------------
             Pr4           1   IPE300
========================================

----------------------------------------
```

LOOSE_PART

This variable is set to LOOSE_PART by default.

To modify this variable in a model, you do next:

Step plan

- Go to **Tools** > **Advanced options...**
- Select the categories **Numbering** on the left pane
- On the right pane, select the variable XS_USE_ASSEMBLY_NUMBER_FOR
- Modify the setting LOOSE_PART in the field **Value** to MAIN_PART
- Click <**Apply**> and <**OK**>

The modification is stored in the file `options.ini` in the model folder.

Apply settings for all models

When you want to apply a setting for all models, you do next:

Option 1 If there is no file `options.ini` in the folder `ts` yet, you can copy the file from the model folder to the folder `ts`

Option 2 If there is a file `options.ini` in the folder `ts` already, open the file `options.ini` in the model folder by using Wordpad. Select the variable in the file `options.ini` in the model folder and copy it to the file `options.ini` in the folder `ts`.

User-specific variables

A user-specific variable, for instance, is

XS_SOLID_BUFFER_SIZE=

The value you enter in this variable, effects the buffersize of the memory which is used for the Tekla Structures-processes. So it is used to speed up your system.

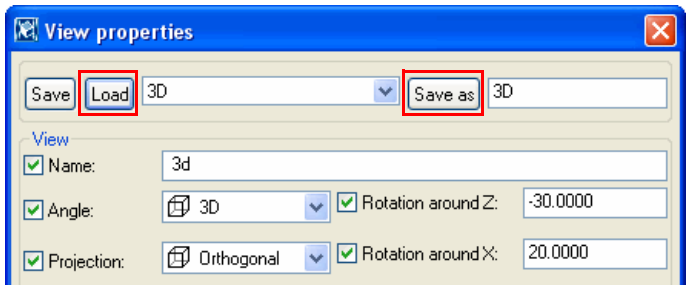
When you modify this variable in **Tools** > **Advanced options...** and store the changes, in the folder `bin` (disc:\TeklaStructures\version\nt\bin) the file `options_Username.ini` is created.

This modified variable will now be used for all models which are opened by that specific user on that computer.

See also To check a list of model- and user-specific variables, see **Tekla Structures.pdf** in the folder: **disc:\TeklaStructures\version\environments\europe\pdf.**

Restore view settings in the Model Editor

When you copy model view settings (file extension ***.mvi**) from an older version, you need to restore these setting in the new version. This, because of the changes that were made in the View properties.



Select the concerned setting(s), click on the button <**Load**> and next <**Save as**>. The settings are stored in the model folder.

Modeling

Change the color of the grid

You can use the variable `XS_GRID_COLOR` in **Tools > Advanced options > Model view** to change the color of the grid in the Model Editor.



Define the color of the grid using RGB values:

```
XS_GRID_COLOR = <Red> <Green> <Blue>
```

Define the values on a scale of `0` to `1`. Reopen the view for the change to take effect.

The changes are stored in the file `options_Username.ini` in the folder `disc:\TeklaStructures\version\nt\bin` and are used for all models.

See also

For more information about storing the settings in the file `options.ini`, see chapter **Manage variables in the file options.ini** (pag. 9)

Example

You want to use a blue backgroundcolor and white grid lines in Modelviews.

Set the following variables to use a blue backgroundcolor in **Tools** > **Advanced optios** > **Model view**:

- `XS_BACKGROUND_COLOR1: 0.21484375 0.4609375 0.875`
- `XS_BACKGROUND_COLOR2: 0.21484375 0.4609375 0.875`
- `XS_BACKGROUND_COLOR3: 0.21484375 0.4609375 0.875`
- `XS_BACKGROUND_COLOR4: 0.21484375 0.4609375 0.875`

The chages again are stored in the file `options_Gebruikersnaam.ini`.

To use white grid lines, set the variable `XS_GRID_COLOR` to `1.0 1.0 1.0`.

Again re-open the view for the changes to take effect.

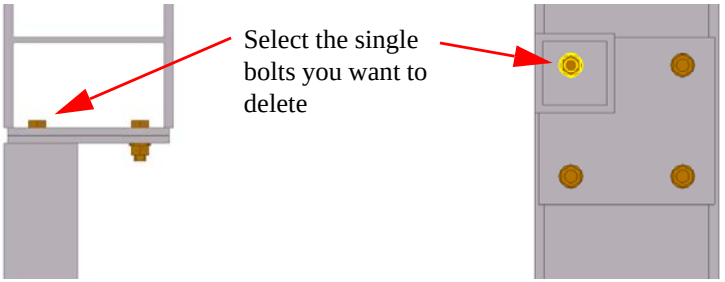


Keep in mind that it is not possible to change the color of dimensioning lines yet! These dimensions will be hard to see while measuring in a model view. They still are displayed in the status bar.

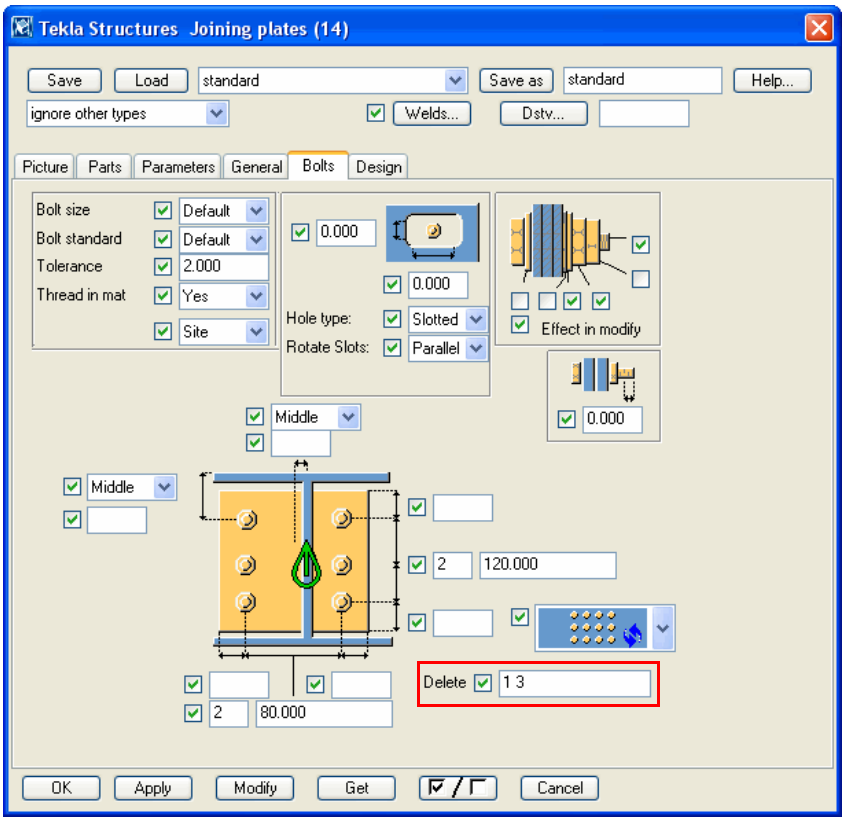
Single bolts

In Tekla Structures you can delete single bolts from a boltgroup. For this, you need to select the icon **Select single bolts** in the toolbar **Select switches**:



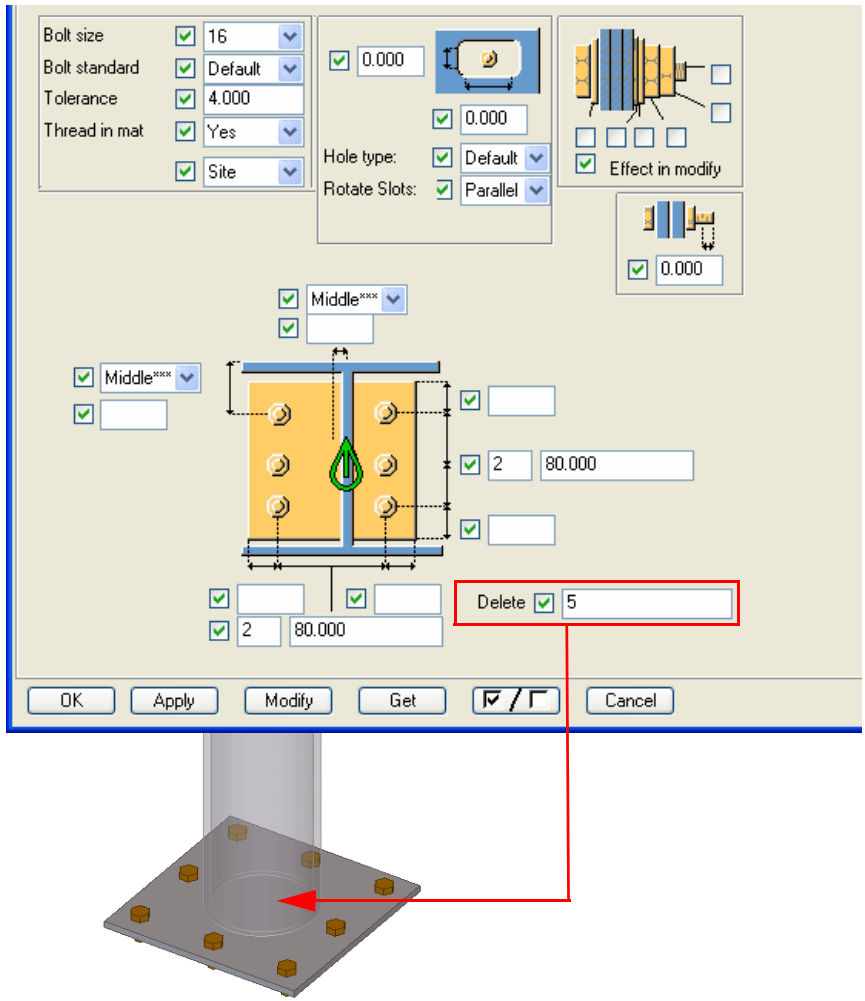


In some macro's you can make use of the bolt-delete option:



Example

Only the bolt (hole) in the centre of the column is deleted.



Workarea invisible

In Tekla Structures you can make the workarea invisible. Because of this you have an even better view on the model.

Invisible

To make the workarea invisible, keep pressed down the <**Ctrl**> and the <**Shift**> key and select in the pull-down menu **Window** > **Redraw all**. The workarea is now invisible.

Visible

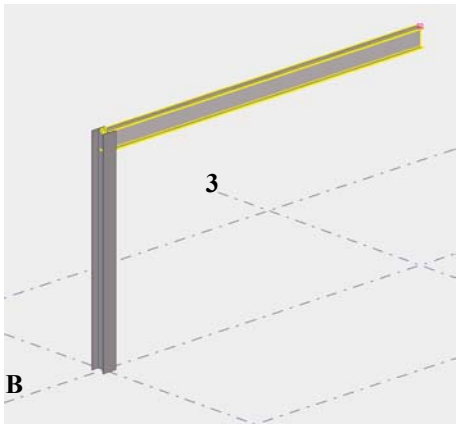
To display the workarea, select in the pull-down menu **Window** > **Redraw all** or press shortkey **g**. The workarea is visible again.

Keys X, Y and Z

In Tekla Structures you can eassily make use of the keys X, Y and Z to lock the mousepointer in a specific direction.

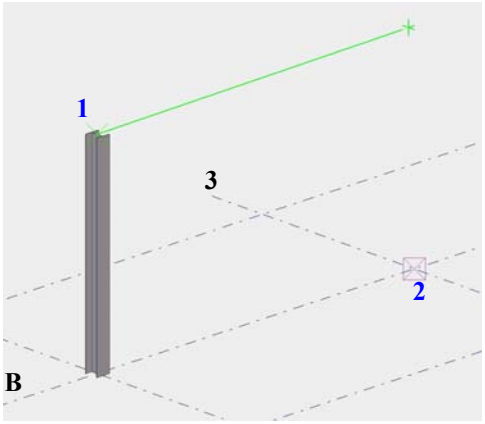
Example

You want to create a beam at a certain level:



Step plan

- Click on the icon **Create beam** and pick the first position (1)



- Snap at the first picked point and press the key **Z,** the z direction is now locked



You always need to snap on a certain point to be able to lock the mousepointer!

- Pick the second position (2) on grid B-3, now that the **Z** direction is locked, it automatically positions the second point of the beam at the locked **z** direction.

Reference models

In Tekla Structures, reference models can be inserted, they serve as help (reference) to model the structure. Reference models will keep the size of Tekla Structures model files relativily small, because the reference file(s) are considered as a single objects. If there are several reference models in the Tekla Structures model, it can be useful to hide the reference models separately.

XML files

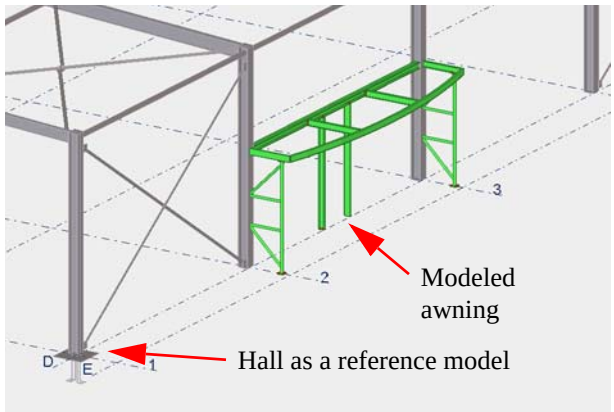
You can use DWG or DXF and also DGN (Microstation) files. You can also use webviewer models which are created in XML format.

Example

An existing model needs an addition, in this case an awning. First, save the existing model as a Webmodel. To do this, go to **File** > **Publish as Web page...**

In the model folder, in sub folder ..\PublicWeb, the file **Model.xml** is stored. This file will be used as a reference model.

Now delete all objects from the current model and add the file **Model.xml** as a reference model.





To be able to select a complete reference model, activate switch **Select components**, to select the reference model at the highest level.

Snapping to reference models

While snapping to high detailed, large reference models it may happen that a large number of snap points is found. This may result in a delay.

In such cases, it is advisable to switch off the select switches which are not necessary at that moment. For example, if an endpoint has to be found, it may be useful to switch off the other select switches.



It affects picking speed, because one dgn file is one solid object and it takes time to find correct picking point.

Split reference models

Split large reference models into smaller ones. Because of that only the parts you need in the modelview are represented. This of course effects the speed.

Speed and memory improvements

Working with reference models and reference model objects is now faster in Tekla Structures 12.1 . When you load a reference model for the first time, Tekla Structures creates a cache file of the source file, which is located in the folder ..\model folder\RefCache. Once the cache file is created, the reference model is considerably faster to read and to draw, so it opens now much faster.

You may sometimes want to change the default location of the cache file when you are working with multi-user models in order to reduce network traffic and disk usage in the server or to speed up the cache operation (if the local drive is faster than the server drive). To do this, use the variable `XS_REFERENCE_CACHE` in the batch file.



Reference models can no longer be displayed in old wire frame views.

Reference model scales



When you export files, always define the units used in order to have the correct scaling.

If you use files that have no units defined, Tekla Structures assumes that the model has been created in meters and, for imperial files, in inches. If this is not the case, you need to set the scale manually in the **Scale** option in the **Reference object** dialog box.

Selecting

The icons in the **Select Switches** toolbar are used to define which objects can be selected:





If you activate switch **Select components**, you can select every level of a component, from the highest level downwards to loose parts in components.



If you activate switch **Select objects in components**, the select order is opposite: from the lowest level (loose parts in components) to the highest level.



If you activate switch **Select assemblies**, you can select every level of an assembly, from the highest level, through sub-assemblies, downwards to the loose parts.



If you activate switch **Select objects in assemblies**, the select order is opposite:, from loose parts (lowest level) to complete assemblies (highest level).

The level of a selected assembly is displayed in the **Status bar**:

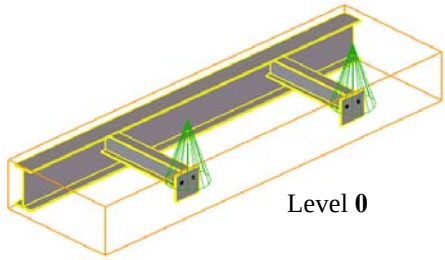
Level 0



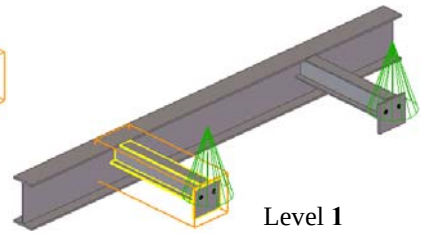
Level 1



To shift to a specific level in an assembly, while the assembly is highlighted, hold down the <**Shift**> key and scroll the mousewheel. The large assembly gets 0 (null), the smaller assembly gets 1, etcetera. You can continue to level 9:



Level **0**



Level **1**

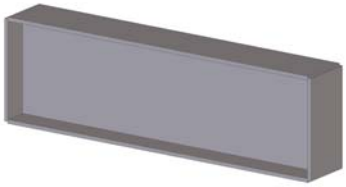
Generating output

While generating reports, drawings, NC files or other types of output, you **always** make use of the switch **Select objects in components** to select all parts!

The switch **Select components** selects at the highest level, because of that, stiffeners or endplates, created by a component, will highlight but will not be selected!!!

Example

A beam + 2 endplates:



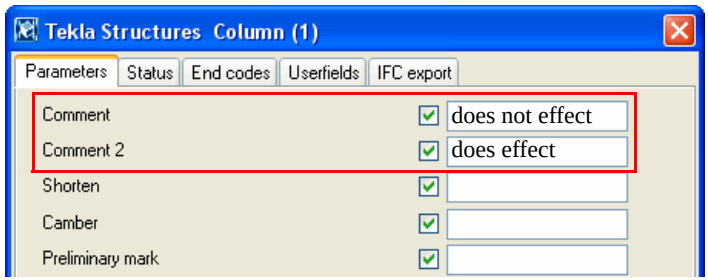
If you activate switch **Select components**, only the beam will appear on the list:

Profile	Part	Grade	Number	Length(mm)	Area(m²)	Weight(kg)
IPE300	Prl3	S235JR	1	980	1.14	41.4
				980	1.14	41.4
				Totaal:	1.14	41.4

If you activate switch **Select objects in components**, both endplates and beam will appear on the list:

Profile	Part	Grade	Number	Length(mm)	Area(m²)	Weight(kg)
FLAT10*150	P7	S235JR	2	290	0.10	3.4
				580	0.19	6.8
IPE300	Prl3	S235JR	1	980	1.14	41.4
				980	1.14	41.4
				Totaal:	1.33	48.2

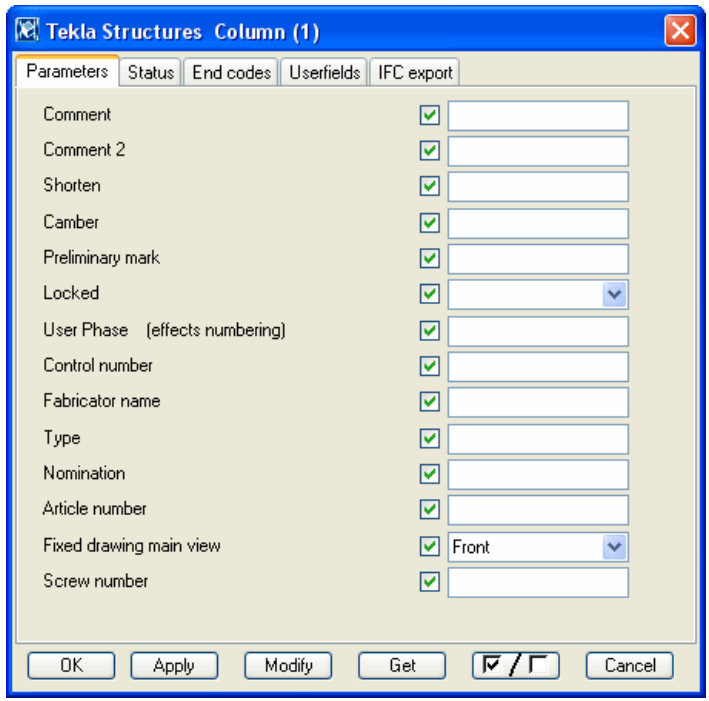
Comment in user-defined attributes



The field **Comment** in the user-defined attributes in profiles does **not** effect numbering anymore. **Comment 2** still effects numbering.

Modify user-defined attributes in the file objects.inp

Many dialog boxes contain user-defined attributes for various objects, such as beams, columns, bolts and drawings, in Tekla Structures.

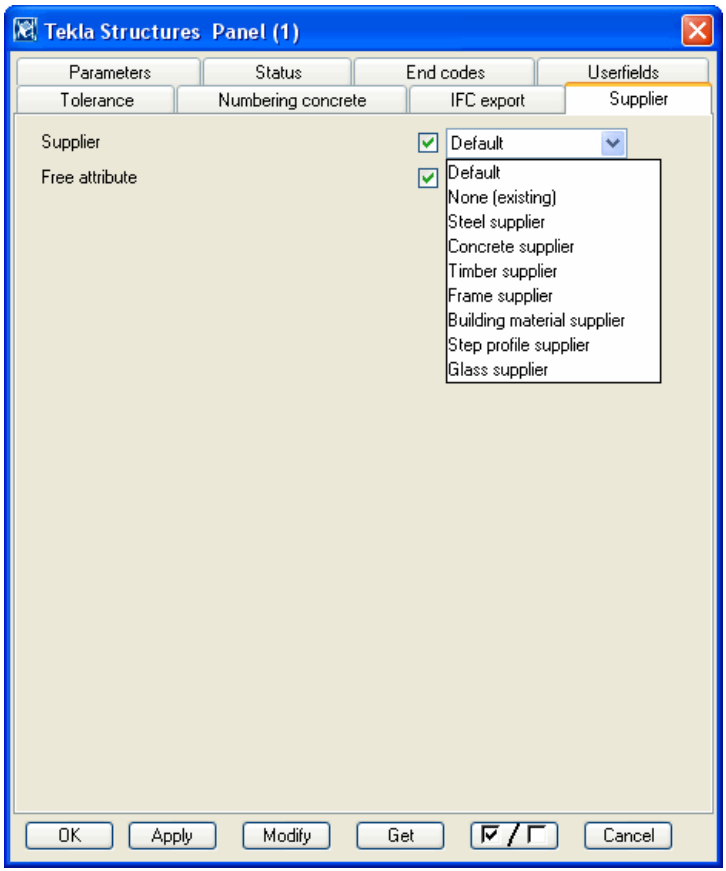


You can modify the existing user-defined attributes fields, which are displayed in the dialog boxes, in the file `objects.inp`. You can also add one or more user-defined attributes fields.

Example

You want to add the tab `Supplier` for all available profiles:

In here you want to have a listbox in which you can select who the supplier is from that specific part. This, to eassily select all parts which should be supplied by a glass supplier, for example.



You also want to make use of a free attribute.

As mentoined before, you can add user-defined attributes fields in the file `objects.inp`. This file is stored in the folder `..versie\environments\country-independent\inp` by default.

Step plan

Open the file `objects.inp` in the model "example objects.inp", the following appears:

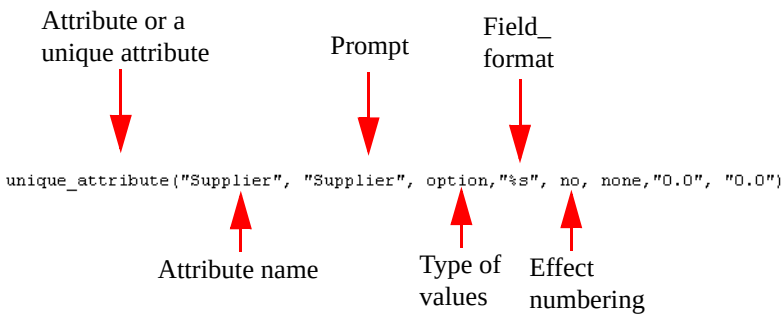
```
part(0,"Part")
{
 tab_page("Supplier")
  {
   unique_attribute("Supplier", "Supplier", option,"%s", no, none,"0.0", "0.0")
   {
       value("j_default", 2)
       value("None (existing)", 0)
       value("Steel supplier", 0)
       value("Concrete supplier", 0)
       value("Timber supplier", 0)
       value("Frame supplier", 0)
       value("Building material supplier", 0)
       value("Step profile supplier", 0)
       value("Glass supplier", 0)
   }
   unique_attribute("Free attribute", "Free attribute", string,"%s", no, none,"0.0", "0
   {
       value("0.0", 0)
   }
  }
}
beam(0,"j_beam")
{
tab_page("Supplier","Supplier",22)
modify(1)
}
```

1

2

3

1 The fields in the file:



Attribute	Regular attribute, which is copied with other part properties.
Unique attribute	Non-copyable attribute. The value of the attribute is never copied to another part. For example part checking status attributes are usually non-copyable.
Attribute name	Attribute name, used to find the attribute value.
Prompt	The text Tekla Structures displays in the dialog box.
Type of value	option is used for lists, string is used for text.
Field_format	"%s" for strings, "%d" for numbers
Effect numbering	You can set whether the user-defined attribute affects numbering or not. This is useful when you have members that are identical in all respects, apart from their user-defined attributes

2 The values which appear in the listbox

3 Defines in which parts this listbox appears

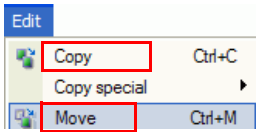
Improvements in copying and moving

You can now copy and move objects by translating more easily in the Model- and Drawing Editor with the new commands:

- **Copy** (shortcut **Ctrl + C**)
- **Move** (shortcut **Ctrl + M**)

Unlike with the traditional Tekla Structures **Copy** command, you can now pick multiple locations to easily create multiple copies of the selected objects to picked locations.

The new **Copy** and **Move** commands do not copy or move by mirroring, rotating, from model or with three points.

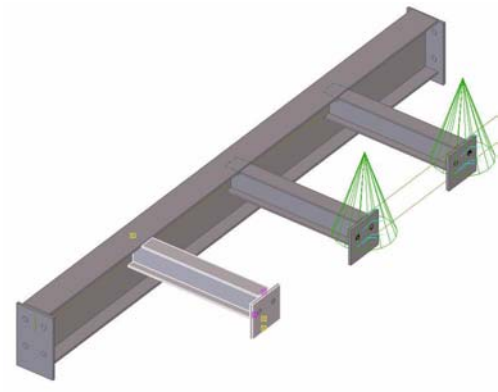


Copy

To copy objects to a new destination:

1. Select the objects to copy.
2. Click **Edit > Copy**.
3. Pick the origin for copying.
4. Pick the destination point. The objects are copied immediately.
5. You can continue and pick other destination points, and the objects are copied to all these locations.

   If you click **Edit > Undo**, the latest copy operation is undone, but the **Copy** command remains active.
6. When all copies have been made, right-click and select **Interrupt** from the pop-up menu.



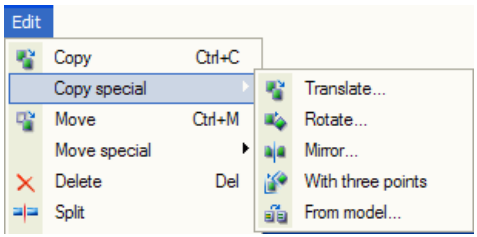
Move

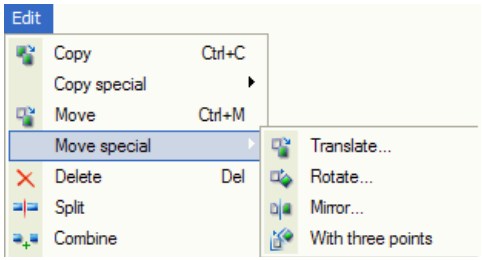
To move objects to a new destination:

1. Select the objects to move.
2. Click **Edit > Move**.
3. Pick the origin for moving.
4. Pick the destination point, and the objects are moved immediately. The **Move** command does not remain active.

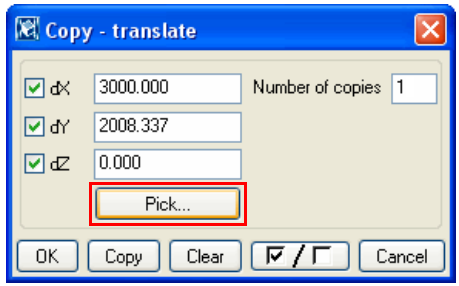
New location for and new features in the old copying and moving commands

The old copying and moving commands can now be found in **Edit > Copy special** and **Edit > Move special** submenus in the main menu and in the pop-up menu. The commands also have toolbar icons.





The **Copy - translate** and **Move - translate** dialog boxes contain a new **Pick** button. When you want to pick a new destination point for copying or moving, and you already have the dialog box open, but the command is not active anymore, you do not have to start the command again, just click the **Pick** button.



Changes in view and select filter

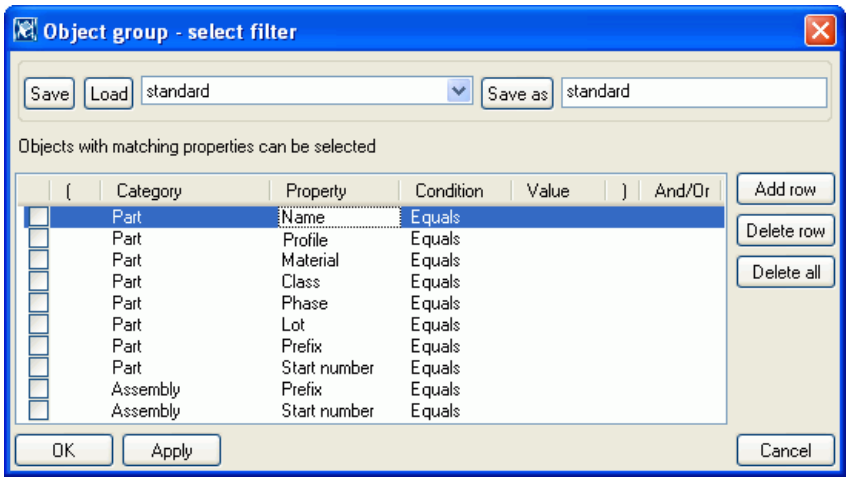
In Tekla Structures 12.1 object groups are used in view (file extension `*.VObjGrp`) and select filters (file extension `*.VObjGrp`) to define which objects are displayed in the selected view and which objects can be selected.

Because of this, the number of options to work faster and more effective in Tekla Structures 12.1 has increased.

Select filter

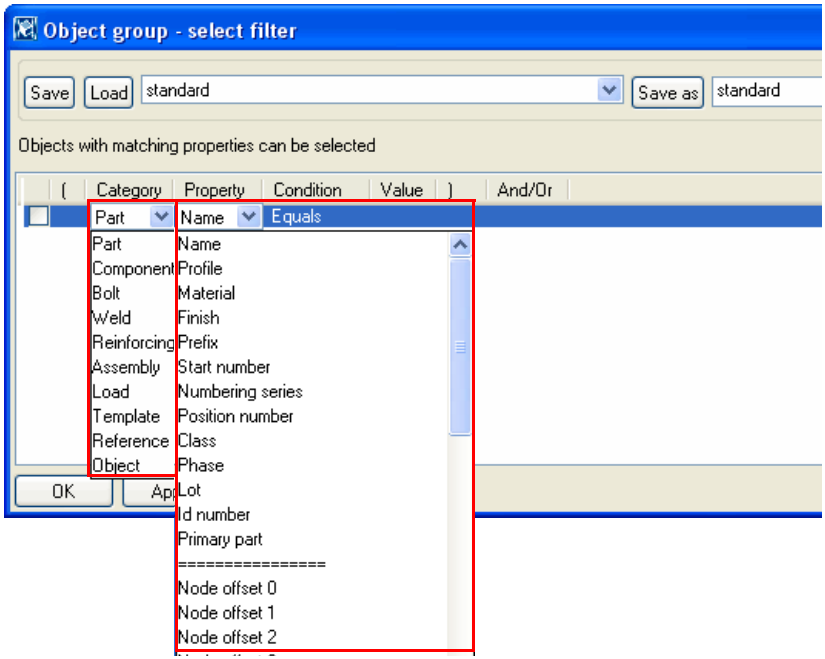


Use the new **Object group - select filter** dialog box to define which objects to select. To access the dialog box, click **Setup > Select filter...** or click on the icon.



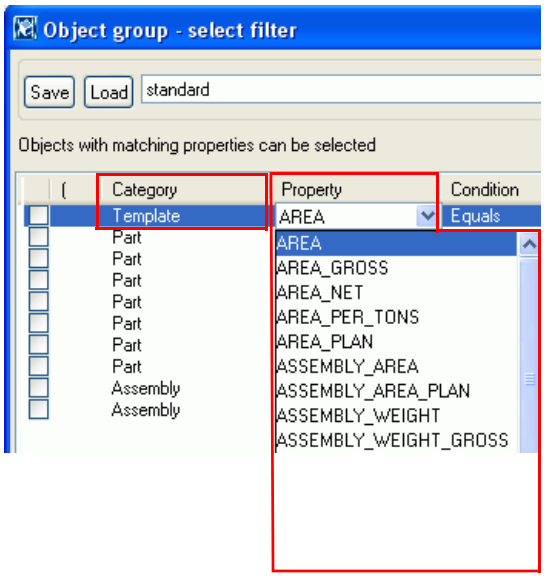
You now have more options in terms of categories and properties by which you can select objects.

To see all the available options, click the list boxes in columns **Category** and **Property**:



Template properties

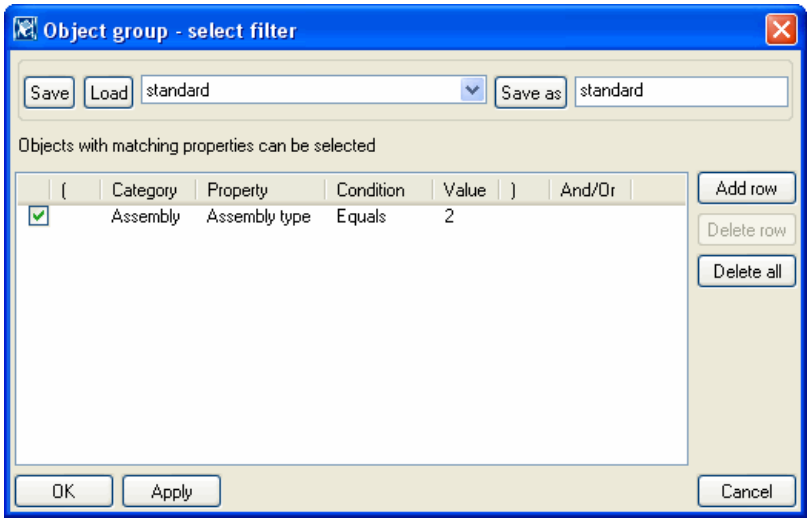
You can select objects according to template properties. To do this, use the category **Template**, and select the desired property in the list box in **Property** column:



See also

For an example of template properties, see **Shorten filter** (pag. 29).

When the category you have selected is **Assembly**, the property you have selected is **Assembly type** and you use the option **Select from model...** in the **Value** column, Tekla Structures returns a value in form of a number that indicates the type of assembly in question.

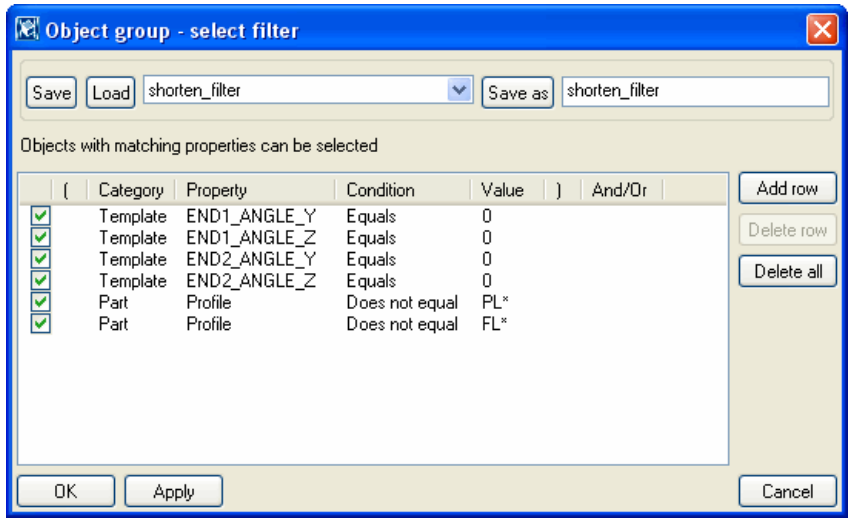


See the following table for information on which number corresponds to which assembly type:

Value	Assembly type
0	prefab
1	cast in place
2	steel
3	timber
4	reinforcing bar
5	reinforcing mesh
6	miscellaneous

Shorten filter

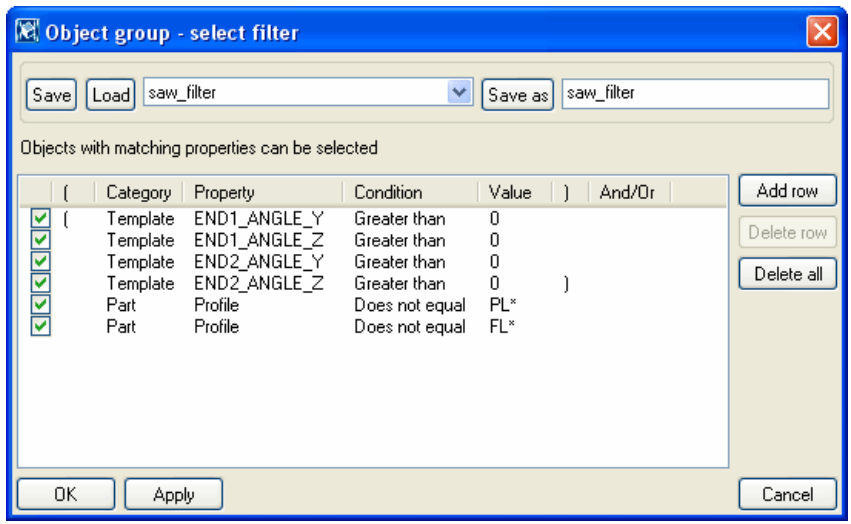
A **shorten filter** is added to the select filters. Use this filter to select profiles (and no plates) from the model to shorten, so only profiles from which the angle of the profiles is equall to 0°:



This filter uses template properties in the four rows, as discribed in chapter **Template properties** (pag. 27).

Saw filter

The saw filter has bee modified. Use this filter to select profiles (and no plates) from the model to saw, so only profiles from which the angle of the profiles is not equall to 0°:



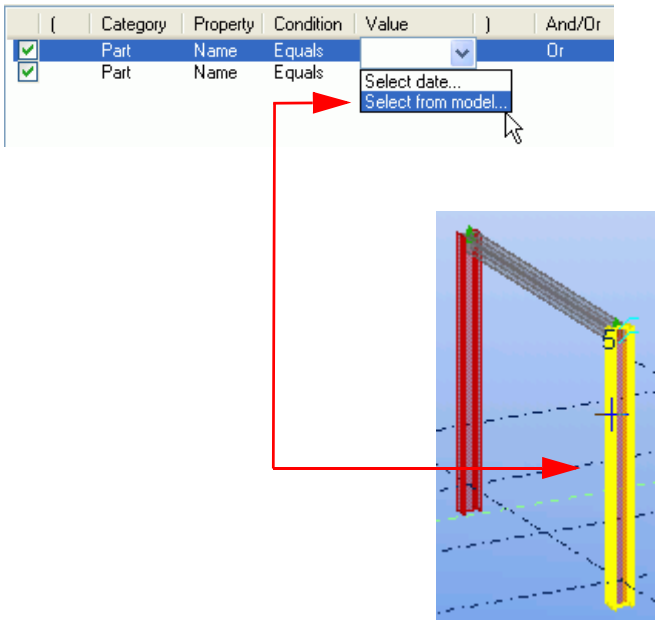
Defining select filters

The following two examples show you how to create object group rules in order to define select filters.

Example 1

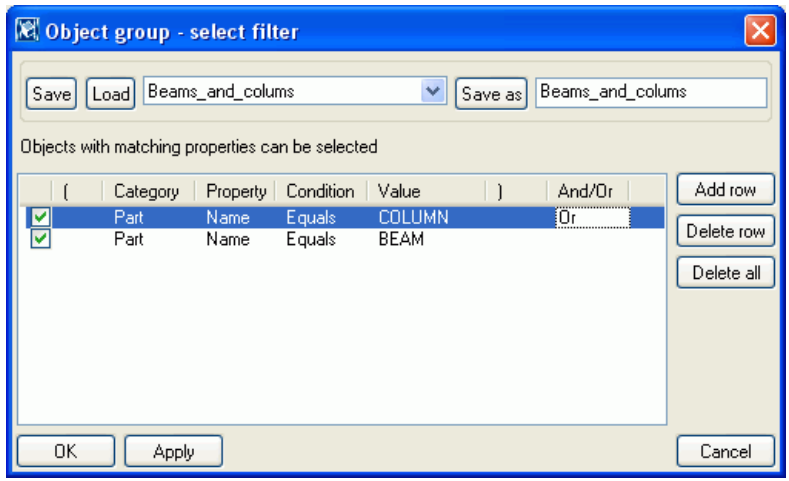
This first example shows you how to define a select filter for beams and columns.

1. Click **Setup > Select filter...** to open the **Object group - select filter** dialog box.
2. You need two rows for this filter. You can delete unnecessary rows by selecting the row and clicking the **Delete row** button.
3. In the column **Category**, select **Parts** for both rows. Select the checkboxes next to these two rows.
4. In the column **Property**, select **Name** for both rows.
5. In the column **Condition**, select **Equals** for both rows.
6. In the column **Value**, select **Select from model...** to select the parts directly from the model. For the first row, pick a column in the model. For the second row, pick a beam in the model.

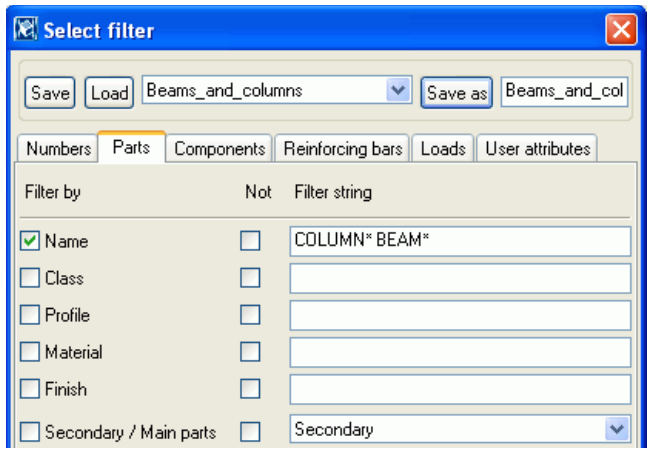


7. For the first row, select **Or** in the column **And/Or.** This enables the select filter to include both beams and columns (parts, whose name is 'beam' or parts, whose name is 'column') in the filter.
8. Enter a name for the select filter in the field next to the **Save as** button and click the button to save the select filter.

The settings should now look like in the picture below:



This corresponds to the following settings in the **Select filter** dialog box in previous versions of Tekla Structures:

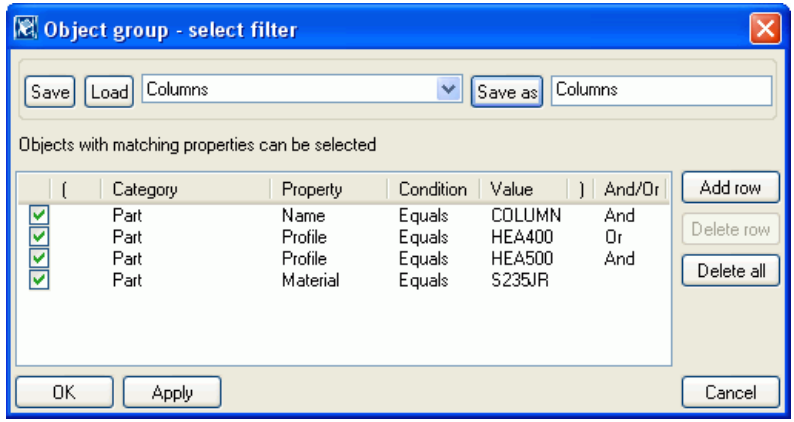


Example 2

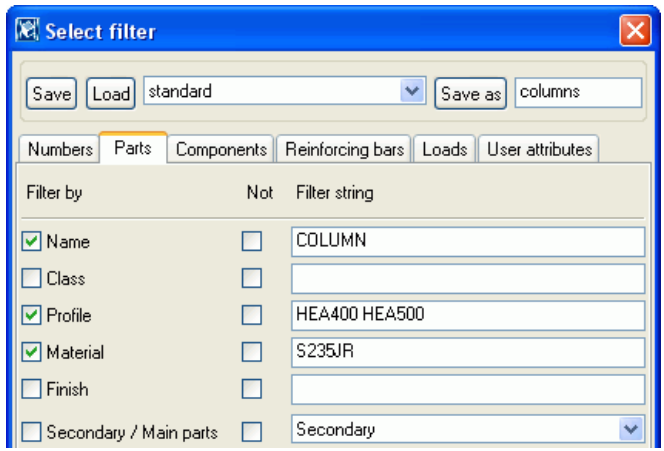
This second example shows you how to define a select filter for columns whose profile is `HEA400` or `HEA500` and the material is `S235JR`.

1. Click **Setup > Select filter...** to open the **Object group - select filter** dialog box.
2. You need four rows for this filter. You can delete unnecessary rows by selecting the row and clicking the **Delete row** button.
3. In the column **Category**, select **Parts** for all four rows. Select the checkboxes next to these four rows.
4. For the first row:
   - In the column **Property**, select **Name**.
   - In the column **Condition**, select **Equals**.
   - In the column **Value**, select **Select from model...** Pick a column in the model.
   - In the column **And/Or**, select **And**.
5. For the second row:
   - In the column **Property**, select **Profile**.
   - In the column **Condition**, select **Equals**.
   - In the column **Value**, select **Select from model...** Pick a column with profile `HEA400` in the model.
   - In the column **And/Or**, select **Or**.
6. For the third row:
   - In the column **Property**, select **Profile**.
   - In the column **Condition**, select **Equals**.
   - In the column **Value**, select **Select from model...** Pick a column with profile `HEA500` in the model.
   - In the column **And/Or**, select **And**.
7. For the fourth row:
   - In the column **Property**, select **Material**.
   - In the column **Condition**, select **Equals**.
   - In the column **Value**, select **Select from model...** Pick a column with material `S235JR` in the model.
   - In the column **And/Or**, select the option that leaves the field empty.
8. Enter a name for the select filter in the field next to the **Save as** button and click the button to save the select filter.

The settings should now look like in the picture below:



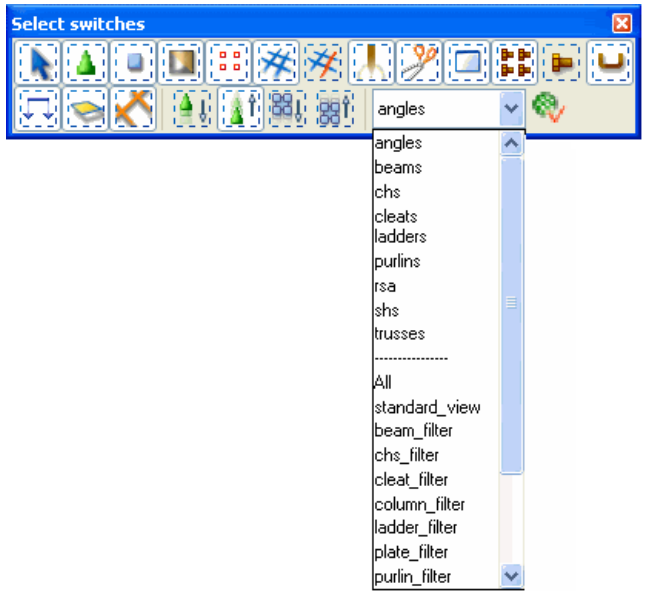
This corresponds to the following settings in the **Select filter** dialog box in previous versions of Tekla Structures:



Select filter list box

The select filter list box is divided into two parts:

- New select filters (file extension .SObjGrp), which you can use in select filtering on top of the list
- Old filters (file extension .msf)

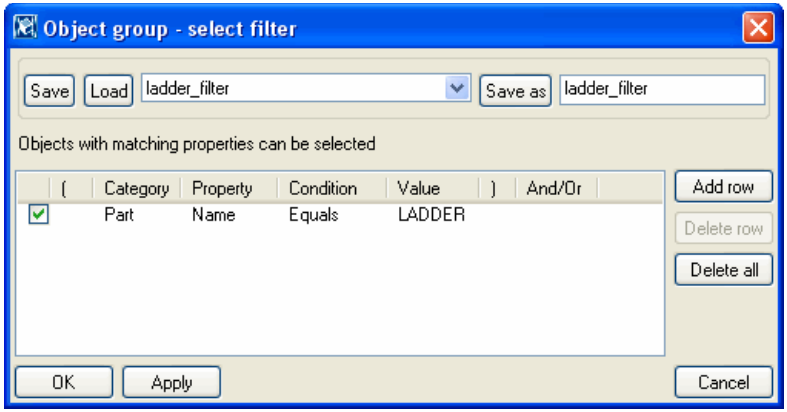


Converting old select filters

You need to convert old filters in Tekla Structures 12.1 (`*.msf files`).



Go to **Setup> Select filter...** or click the icon, the following dialog box appears:



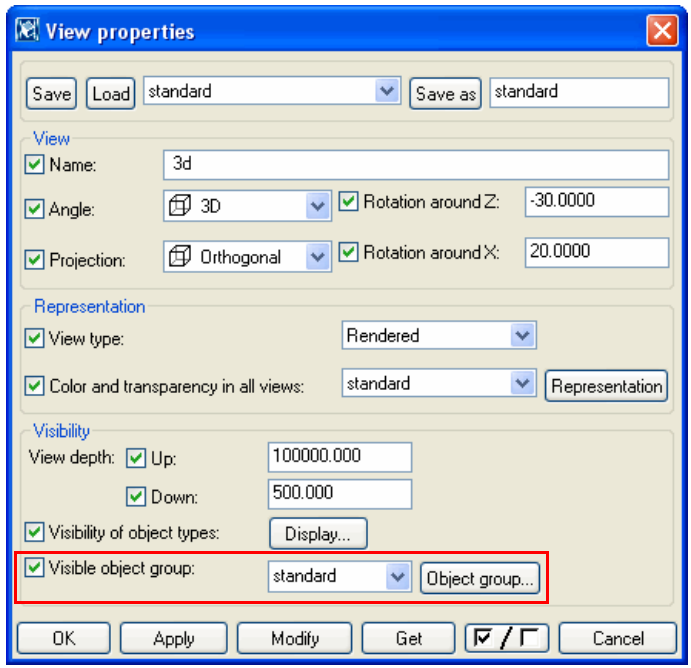
Select the filter you want to convert in the listbox and click the button <**Load**> and next <**Save as**>. Tekla Structures automatically converts this old filter to a new one and stores it. The old filter is now deleted from the listbox.



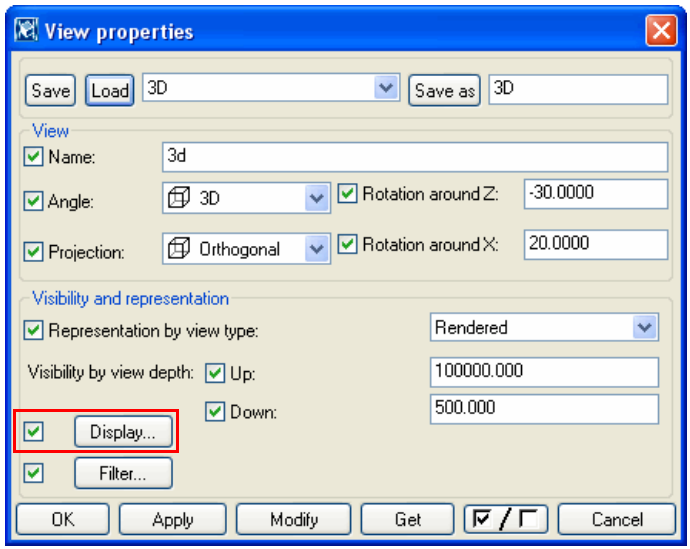
Mind that you need to check the working of the converted filters to see if it is equall to the old ones!

View filter

Also in model views, object groups can be used to define which objects are displayed in the selected views. Double-click on a modelview to open the dialog box **View properties**:



This corresponds to the following settings in the **View properties** dialog box in previous versions of Tekla Structures:

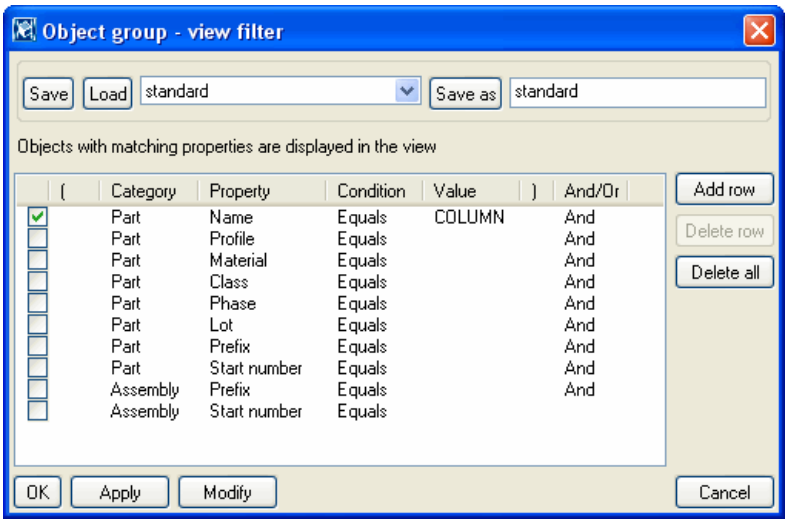


To view all available options, click the button **Object groep...** in the dialog box **View properties**.

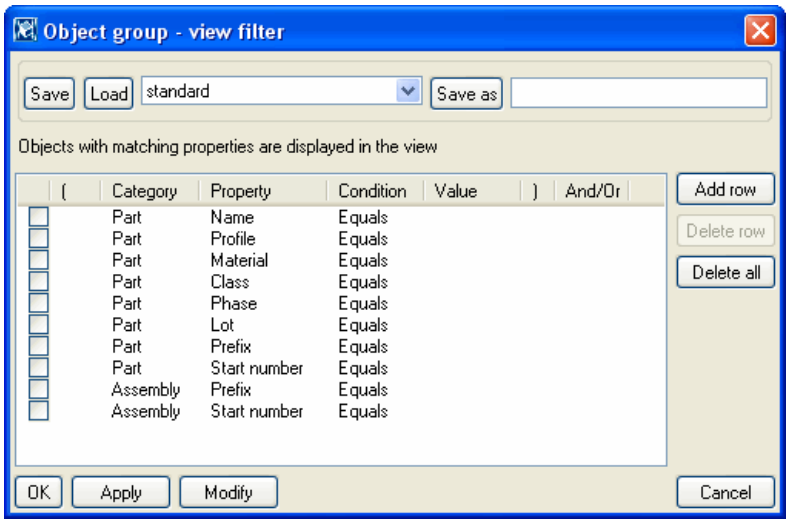
To use an option, select an object group in the listbox.

Store view filter settings

When you use a filter in the column **Value** in the model view, always store this modified setting by using the button <**Save as**>. The setting is stored.

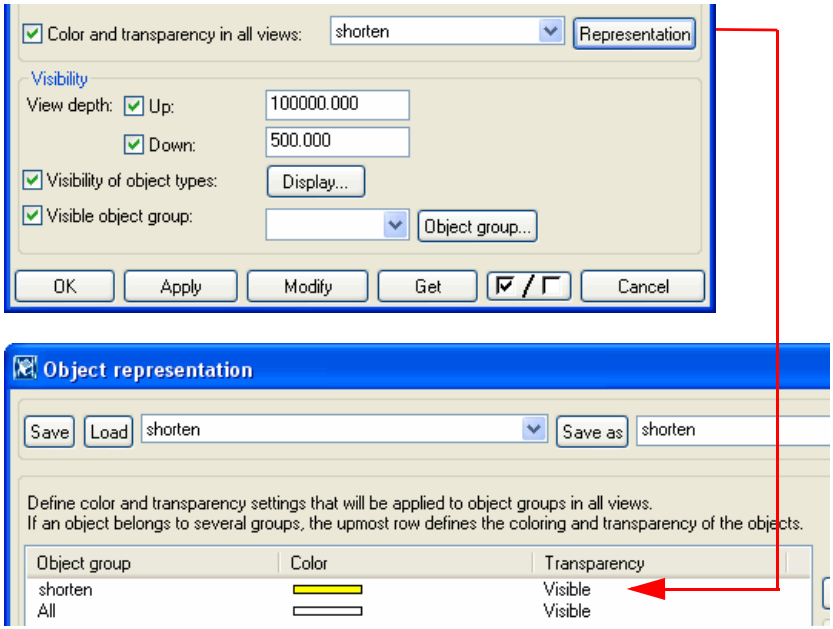


If you do not store the setting and re-open the dialog box later on, the default setting will be displayed and you cannot see what filter setting was used:



Option Shorten is fixed

In Tekla Structures 12.1 you can use the object group **Shorten** again to check which profiles are shortened:



Object representation manager

You can use the object representation manager to define the colors and transpareancy of parts, you can also define new or modify ***object groups***.

See also

For more information about the object representation manager, see **Tekla Structures.pdf** in folder: **disc:\TeklaStructures\version\environments\europe\pdf** and the model `Supplier` in the Tekla Structures 12.1 download.

Numbering

In Tekla Structures 12.1 a new file **numbering results** is stored in the model folder.



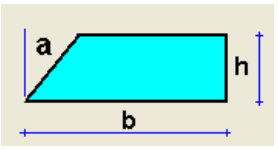
This file only shows the results of the last time you numbered the model.

New parametric profiles: TRZB and TRIA

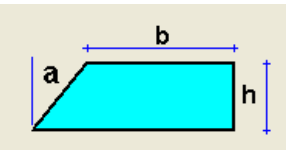
Two new parametric profile sections are added.

TRZB

Profile TRZB is almost like profile TRZA, only value **b** differs:

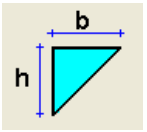


TRZA



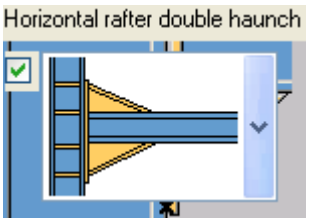
TRZB (new)

TRIA

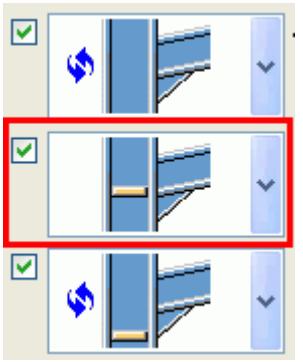


Haunch (40) improvements

Use the new options in **Horizontal rafter double haunch** on the **Picture** tab in cases where the haunch above is a copy of the haunch below and the rafter is horizontal.



Option on the tab **Picture**



Option on the tab **Parameters**

In addition to selecting an option in **Horizontal rafter double haunch** on the **Picture** tab, remember to select the stiffener option on the **Parameters** tab. For example, for the double haunch option shown in the picture above, select the stiffener option shown in the picture above.

Drawings

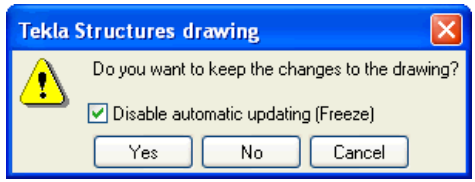
Saving en Freezing associative drawings

Manual changes

In previous versions, if you touched up a drawing manually, and later the model changed, the manual modifications to the drawing were lost for non-frozen drawings.

Freeze

From Tekla Structures 12.0, manual modifications will **always** be kept if the drawing is **not** frozen. If you modify and save a drawing, Tekla Structures ask if you want to save the changes. Click <**Yes**>. To prevent that a drawing will be updated automatically, check the option **Disable automatic updating**:



This will **freeze** the drawing. Freezing drawings causes drawing objects to be not associative anymore:

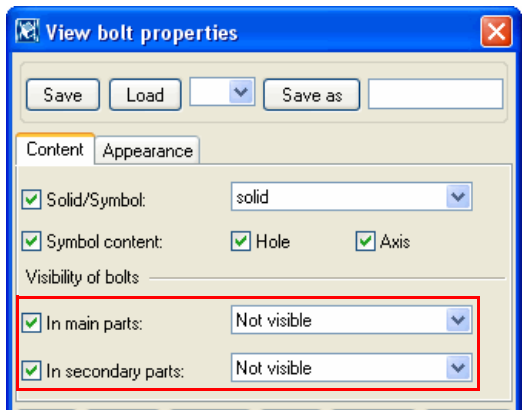
Frozen drawing: Not associative

Not frozen drawing: Associative

Representing bolts on GA drawings

It is recommended on GA drawings, when you use a very small scale (e.g. 1:100), to switch off the bolts.

Because of this, GA drawings will open faster and you can work faster in the Drawing Editor.



Classifier: No dimensions for zinc coating holes

General

By using the drawing classifier you can effect the layout in drawings. This enables you to create different drawing styles.

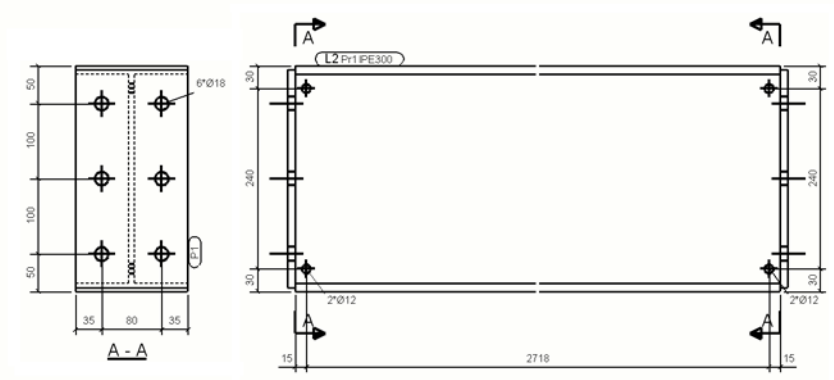
You can use the classifier to control the drawing properties for different types of model objects, for example:

- Different phases in separate colors
- Different parts in separate colors
- Different material grades in separate colors

You can set the classifier to automatically dimension boltholes and to not dimension zinc coating holes on workshop drawings:

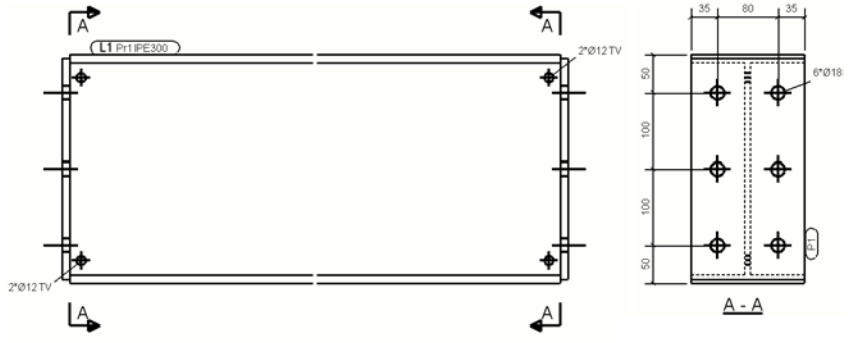
Without classifier

All holes are dimensioned:



With the classifier

Boltholes are dimensioned, zinc coating holes are not dimensioned:

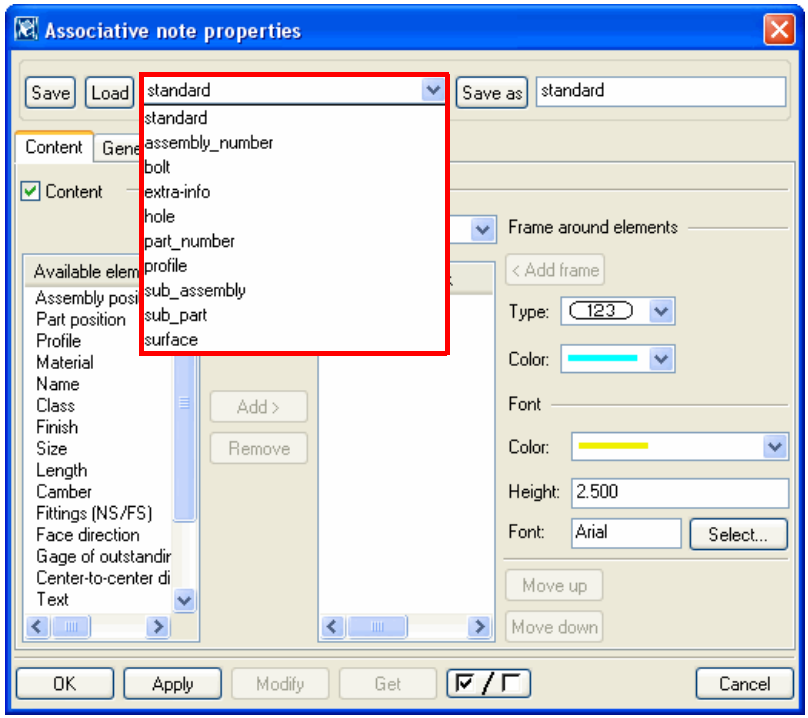


See also

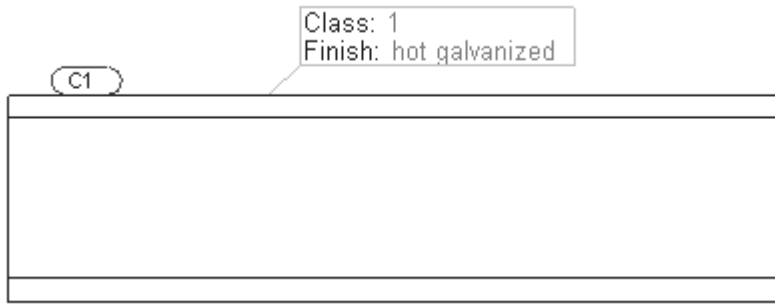
For more information about the classifier, see **Tekla Structures.pdf** in the folder: **disc:\TeklaStructures\version\environments\europe\pdf** and the model `no dimension for zinc coating holes` in the Tekla Structures 12.1 download.

Associative notes

Several settings are added to associative notes, you can use them in drawings as additional marks like part marks and there are updated automatically when a modification has been carried out to the object.

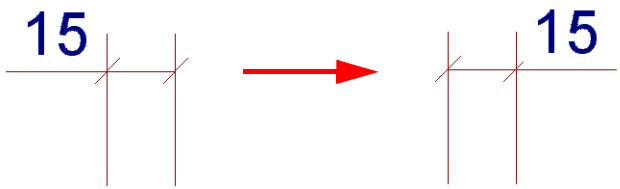


Example extra-info



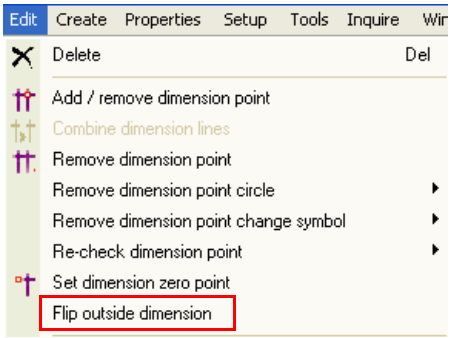
Change the side of outside dimensions

When measuring short dimensions in Tekla Structures 12.1and when the location of the dimension text is set to outside of the dimension lines, you can now select on which side of the extension line the dimension text is placed.



This makes drawing editing easier and faster. To change the side of outside dimensions:

1. Click **Edit > Flip outside dimension**.



2. Select the dimension.
3. Tekla Structures changes the side of the dimension text.

Improvements in leader line appearance

In Tekla Structures 12.1 you can create professional looking drawings with better readability with the help of the improvements in leader line appearance.

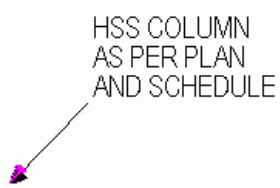
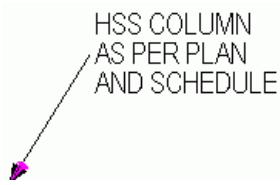
Leader line position options

There are new options to position the leader line. You can use the new positioning options for two types of leader lines:

- To use the different options for a leader line with a rectangular frame, use the variable `XS_MARK_LEADER_LINE_POSITION_TYPE_FOR_RECTANGULAR_FRAME` in **Tools > Advanced options > Marking: general**. The default is `0`.

- To use the different options for a leader line with no frame, use the variable `XS_MARK_LEADER_LINE_POSITION_TYPE_FOR_NO_FRAME` in **Tools > Advanced options > Marking: general**. The default is `0`.

The following table includes information on the new positioning options. The **Appearance** column includes examples of situations, where the variable `XS_MARK_LEADER_LINE_POSITION_TYPE_FOR_NO_FRAME` has been used.

Position	Appearance	Set the variable to
Nearest corner		0
In the middle of the text area		1
1/3 from the top of the text area		2
In the middle of the first row of text		3

Leader line extension length

To define the length of the leader line extension, use the variable `XS_MARK_LEADER_LINE_EXTENSION_LENGTH` in **Tools > Advanced options > Marking: general**. The default length is 0. Define the length in millimeters.

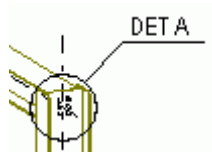
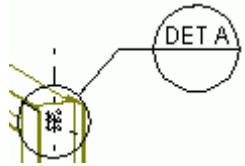
New options in presenting section and detail view symbols

You now have more options in presenting section and detail view symbols. You can define the type of the symbol according to your local customs and preferences. To do this, use the variables in **Tools > Advanced options > Drawing properties**:

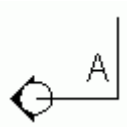
- XS_DETAIL_SYMBOL_TYPE to define the type of the detail symbol
- XS_SECTION_SYMBOL_TYPE to define the type of the section symbol
- XS_DRAWING_VIEW_REFERENCE_SYMBOL to define the appearance of the symbol
- XS_DETAIL_SYMBOL_REFERENCE_IF_TO_CURRENT and XS_SECTION_SYMBOL_REFERENCE_IF_TO_CURRENT to include references to the view in detail and section symbols if the view is located in the same drawing. Normally the reference is included only if the view is in a different drawing.
- XS_SECTION_SYMBOL_LEFT_ARROW_SYMBOL and XS_SECTION_SYMBOL_RIGHT_ARROW_SYMBOL to define the arrow symbol Tekla Structures uses in section symbols at the left and right end of sections

See the following tables for examples on the new options.

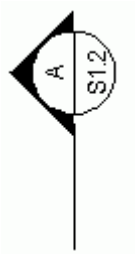
Detail symbol type

Set variables to	Description	Appearance
XS_DETAIL_SYMBOL_TYPE=0	Tekla Structures uses the traditional detail symbol type.	
XS_DETAIL_SYMBOL_TYPE=1 XS_DRAWING_VIEW_REFERENCE_SYMBOL=sections@66	Tekla Structures uses the symbol defined in the variable XS_DRAWING_VIEW_REFERENCE_SYMBOL as the detail symbol.	

Section symbol type

Set variables to	Description	Appearance
XS_SECTION_SYMBOL_TYPE=0 XS_SECTION_SYMBOL_LEFT_ARROW_SYMBOL=sections@0 XS_SECTION_SYMBOL_RIGHT_ARROW_SYMBOL=sections@1	Tekla Structures uses the traditional section symbol type.	
XS_SECTION_SYMBOL_TYPE=1 XS_SECTION_SYMBOL_LEFT_ARROW_SYMBOL=sections@2 XS_SECTION_SYMBOL_RIGHT_ARROW_SYMBOL=sections@2 XS_DRAWING_VIEW_REFERENCE_SYMBOL=sections@66 Adjust the arrow length and offset in the **Section symbol properties** dialog box if necessary.	Labels are drawn inside the symbol.	
XS_SECTION_SYMBOL_TYPE=2 XS_SECTION_SYMBOL_LEFT_ARROW_SYMBOL=sections@6 XS_SECTION_SYMBOL_RIGHT_ARROW_SYMBOL=sections@6 Adjust the arrow length and offset in the **Section symbol properties** dialog box if necessary.	Label is drawn to only one arrow and located in front of the symbol.	

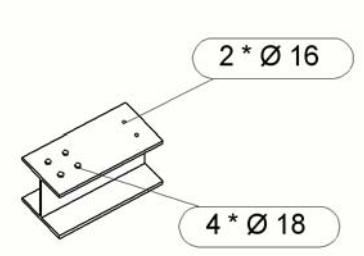
References in detail and section views

Set variables to	Description	Appearance
XS_SECTION_SYMBOL_TYPE=1 XS_SECTION_SYMBOL_LEFT_ARROW_SYMBOL=sections@2 XS_SECTION_SYMBOL_RIGHT_ARROW_SYMBOL=sections@2 XS_DRAWING_VIEW_REFERENCE_SYMBOL=sections@66 XS_SECTION_SYMBOL_REFERENCE_IF_TO_CURRENT=TRUE Adjust the arrow length and offset in the **Section symbol properties** dialog box if necessary.	References are included in the section view.	

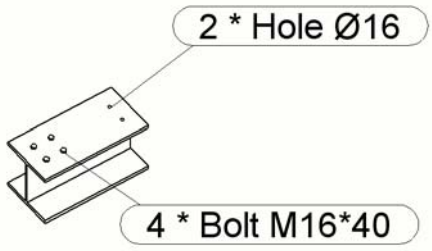
The section view label is now updated also in the other drawing when you move the section view from one drawing to another, and change the section symbol label in the original drawing.

Bolt/hole marks

The definition of bolt/ hole marks in general arrangement drawings is now correct, the marks are displayed correctly.



Tekla Structures 12.0

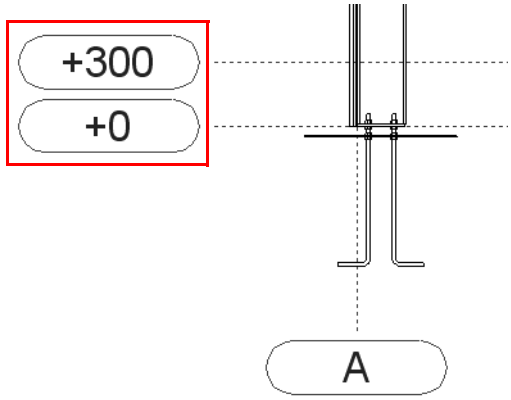


Tekla Structures 12.1

Grid label frame size

You can now define the frame size for the grid labels. This is useful for example when you want all the grid label frames to be the same size no matter whether there are one or two digits in the frame. This gives the grid a consistent appearance.

To define the grid label frame size, use the variable `XS_DRAWING_GRID_LABEL_FRAME_FIXED_WIDTH` in **Tools > Advanced options > Drawing properties**. The default value is zero, which indicates that the width of the grid label frame depends on the width of the grid label. To increase or decrease the width of the frame, enter the desired value in millimeters, in this example, value 12 has been used:



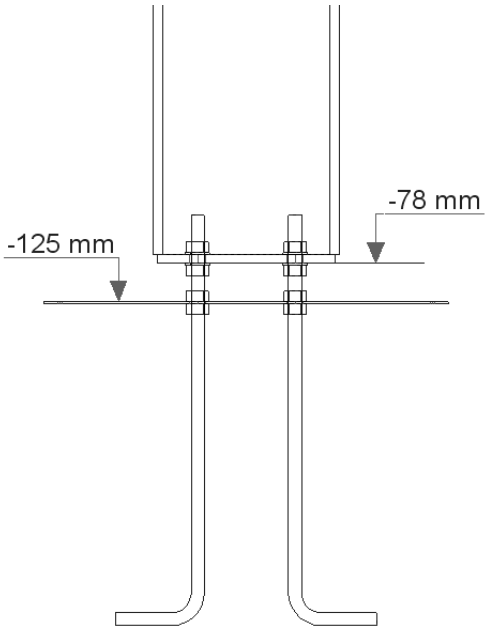
Template pathname

A new template **pathname** is added. This template displays the lokation and the modelname on the drawing, e.g.:

C:\TeklaStructuresModels\hal1\

Templates "bottom side baseplate" and "top side castplate" added

The templates **bottom side baseplate** and **top side castplate** can be used on the foundation plan.



The template **bottom side baseplate** is displayed as follows:

Grid	Profile	Bottom side baseplate
B/2	HEA240	-0.078

The template **top side castplate** is displayed as follows:

Grid	Profile	Top side castplate
B/2	PL3*500	-0.125

Plotter catalog

The system folder in Tekla Structures 12.1 now contains two files in which the settings for the printers/plotters are stored:

plotdev.bin

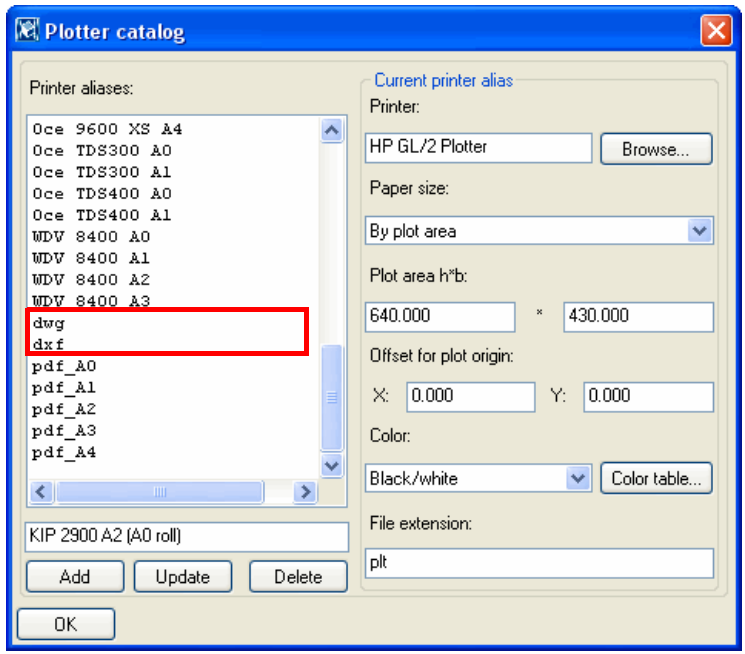
This file contains the regular printers and plotters. When you use this file, you only have to delete just a few or mayby no printers/plotters.

plotdev_org.bin

This is the master file containing all printers and plotters and their default settings. If you want to use this file, first delete the file `plotdev.bin` in the system folder and then rename the file `plotdev_org.bin` to `plotdev.bin`.

Dwg/dxf

It is recommended to delete the printer aliases dwg and dxf from the plotter catalog. They will be deleted by Tekla in the near future, so they cannot be used anymore.



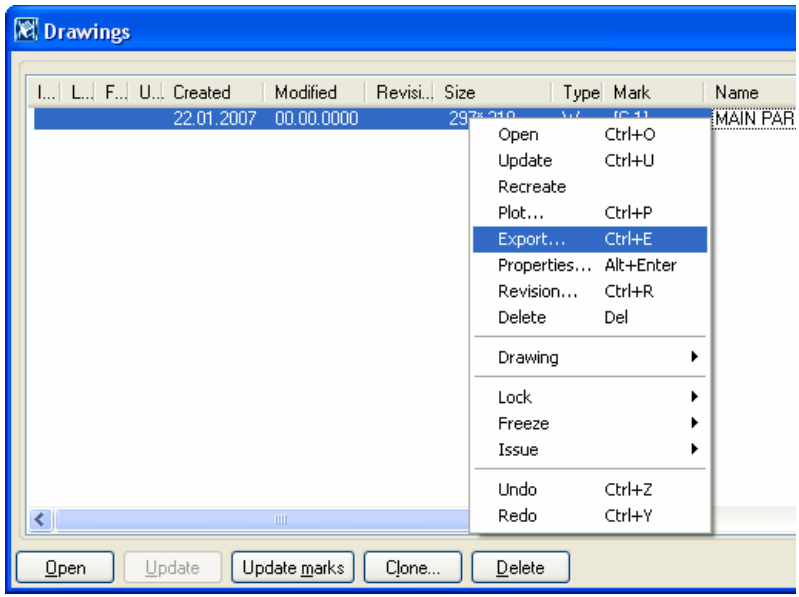
Exporting drawings

The best option for printing drawings to dwg/dxf is the command Export.

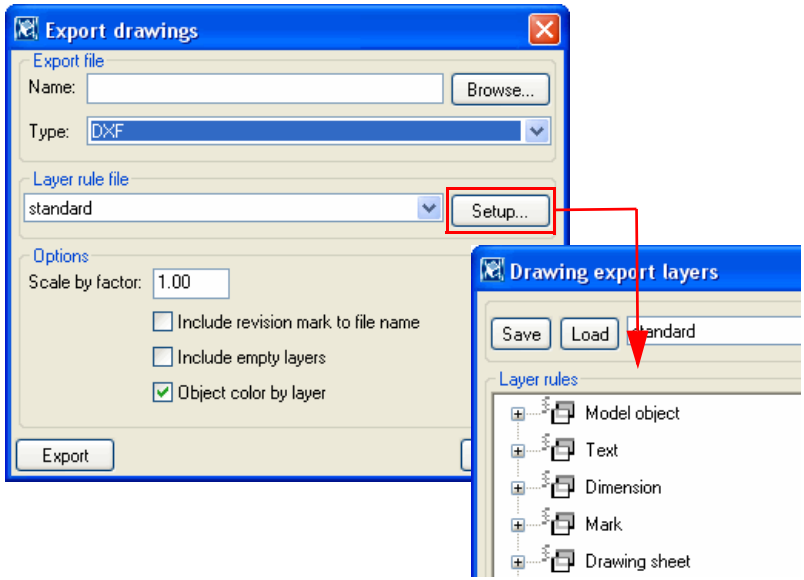
The main advantage is that layers can be defined beforehand for the several objects (profiles, bolts, marks ect.) on the drawing.

With this the original layer structure can be maintained. Also for possible finishing of drawings the filter and layer options can be used (freezing, layer on/off ect.).

The <**Export**> command can be activated from the drawinglist: right mouse click on the drawing(s), then choose **Export** from the contextmenu:



The dialog box **Export drawings** appears:



You can enter the name of the file, if there is no name entered, the export file shall get the same name as the Tekla Structures drawingname. You can also modify the layer rules.

See also

For more information about exporting drawing to dwg/dxf files, see **Tekla Structures.pdf** in the folder: **disc:\TeklaStructures\version\environments\europe\pdf.**

Custom components

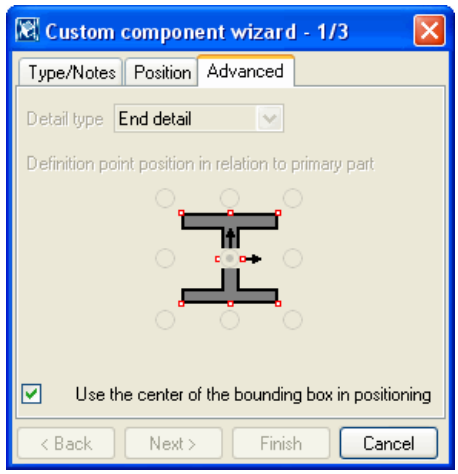
Autoplacing for custom component "part"

Up to Tekla Structures 11.3 the position of a custom component was determined in relation to the points you picked while creating a custom component.

In Tekla Structures 12.1 a so-called autoplacing is added. This means that, when a custom component changes, e.g. its height, also the position of the custom component in the model will be adjusted. In many cases, this could be really handy.

However, if you start from the points, it is impossible to add level marks.

Because of that, an option has been added while creating a component for the type part, you can toggle autoplacing on or off.

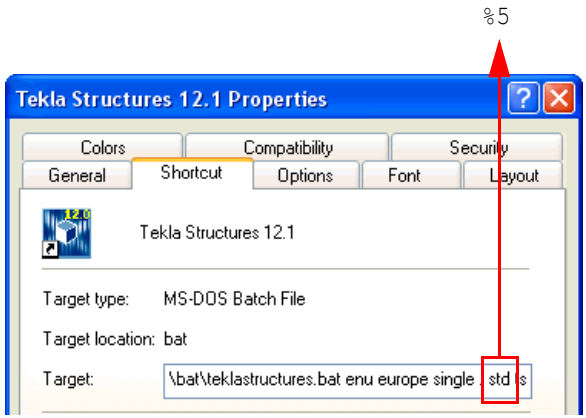


Availability of Custom Components and Sketched profiles

In Tekla Structures, all Custom Components and sketched profiles are available in all **new** models. This means you do not have to import them in every new model anymore.

For this, the variable `XS_UEL_IMPORT_FOLDER` is used. The variable refers to the folder `disc:\TeklaStructures\version\environments\europe\custom-components\%5\.`

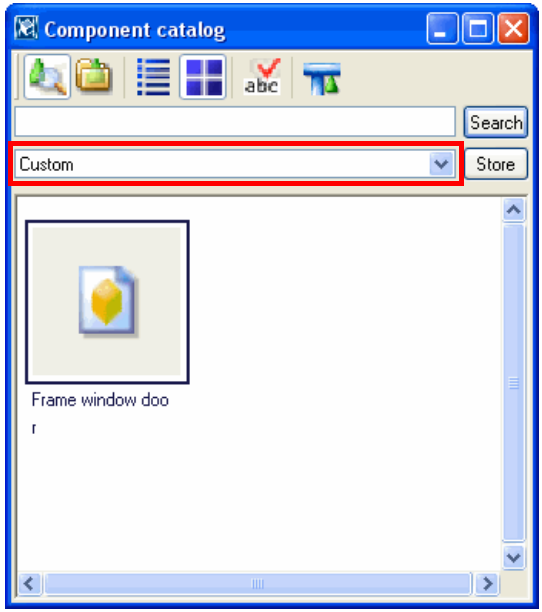
`%5` in the mentoined line above means that if you use the configuration STD (Steel Detailing), for example, the refering `*.uel` files from that folder are automatically imported in the new model. The reason for this is the shortcut on your desktop, refering to the folder `std`.



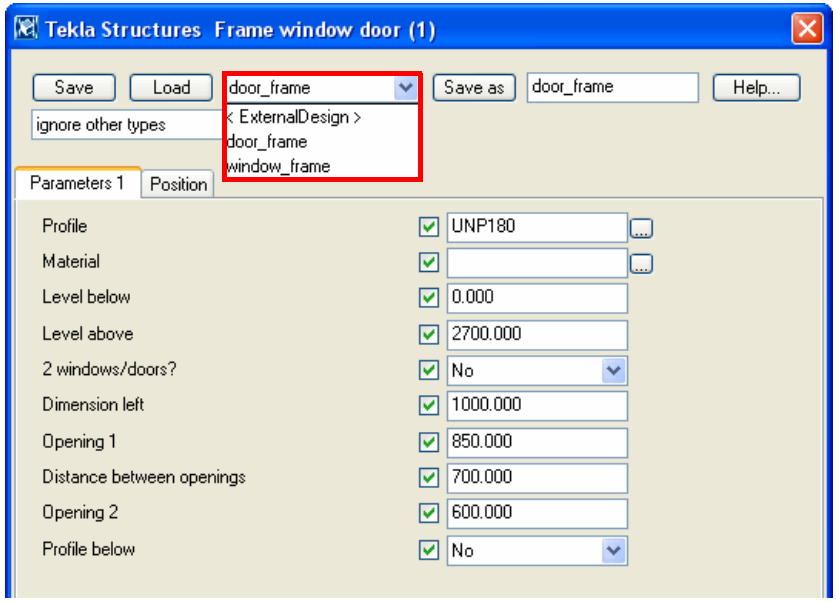
Example

You want to have custom component FRAME WINDOW DOOR and accompanying component settings to be available in all new models:

1. In the Windows Explorer, go to the folder `...europe\custom-components\std` and extract the file `frame window door.zip`, both `*.uel` file and accompanying component setting(s)
2. Start Tekla Structures and create a new model
3. Click icon **Find a component** in the **Components**-toolbar, or use shortkey <**Ctrl** + **F**> to open dialog *Components catalog*
4. Set the picklist to **Custom** to display the Custom components



5. Now, doubleclick on the custom component to load one of the settings from the picklist.

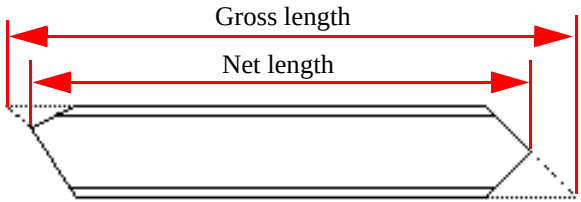


6. The custom component settings are stored automatically in the folder `attributes` in the model folder.

NC files

Variabele XS_DSTV_PRINT_NET_AND_GROSS_LENGTH

The variabele XS_DSTV_PRINT_NET_AND_GROSS_LENGTH is set to TRUE in NC files, by default.



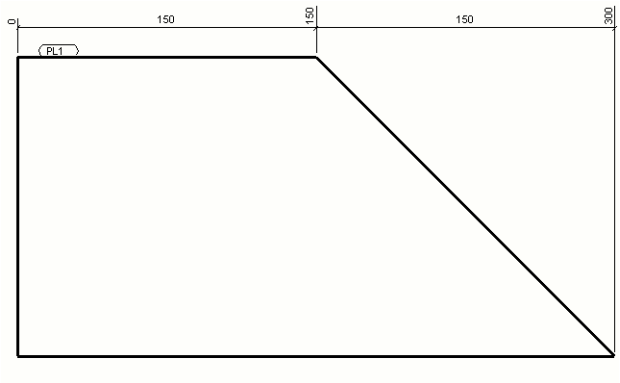
Because of this, both net- and gross length are included in the NC files' header, according to the DSTV standard:

In the NC file:

```
ST
** B1.nc1
  project number
  1
  B1
  B1
  S235JR
  1
  IPE300
  I
    2704.89,2710.64
     300.00
     150.00
      10.70
       7.10
      15.00
     42.200
      1.159
      0.000
     15.745
      0.000
      0.000
```

NC files for plates

For plates, the angle is no longer included in the NC files' header.



In the NC files:

Tekla Structures 12.0

```
ST
** PL1.nc1
  project number
  1
  PL1
  PL1
  S235JR
  1
  FLAT10*150
  B
     300.00,300.00
     150.00
      10.00
      10.00
      10.00
       0.00
     78.500
      2.241
      0.000
     45.000
      0.000
      0.000
```

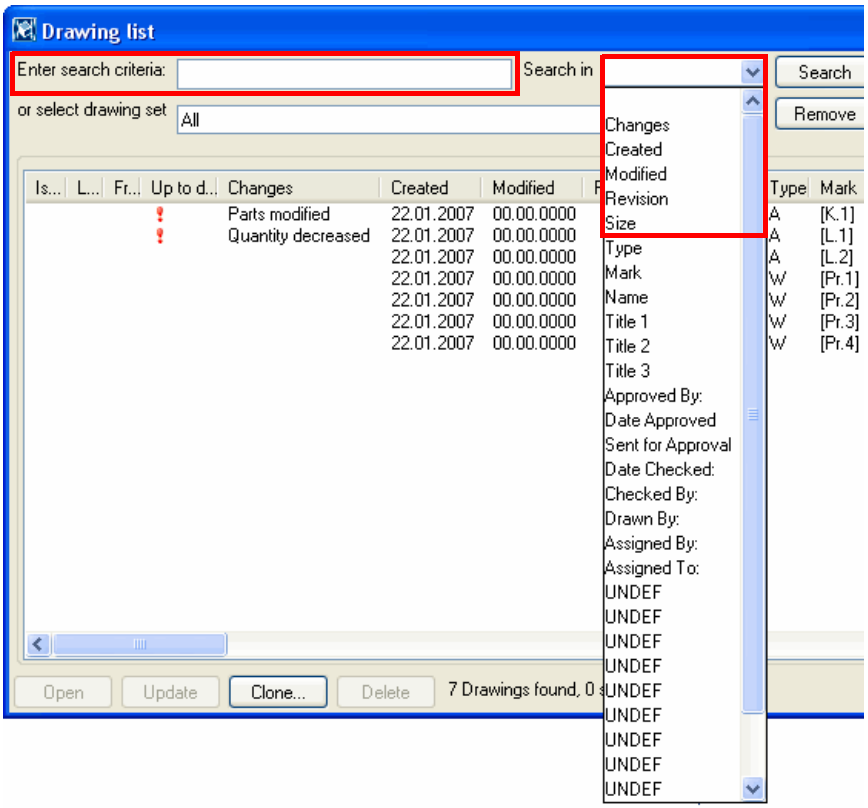
Tekla Structures 12.1

```
ST
** PL1.nc1
  project number
  1
  PL1
  PL1
  S235JR
  1
  FLAT10*150
  B
     300.00,300.00
     150.00
      10.00
      10.00
      10.00
       0.00
     78.500
      2.241
      0.000
      0.000
      0.000
      0.000
```

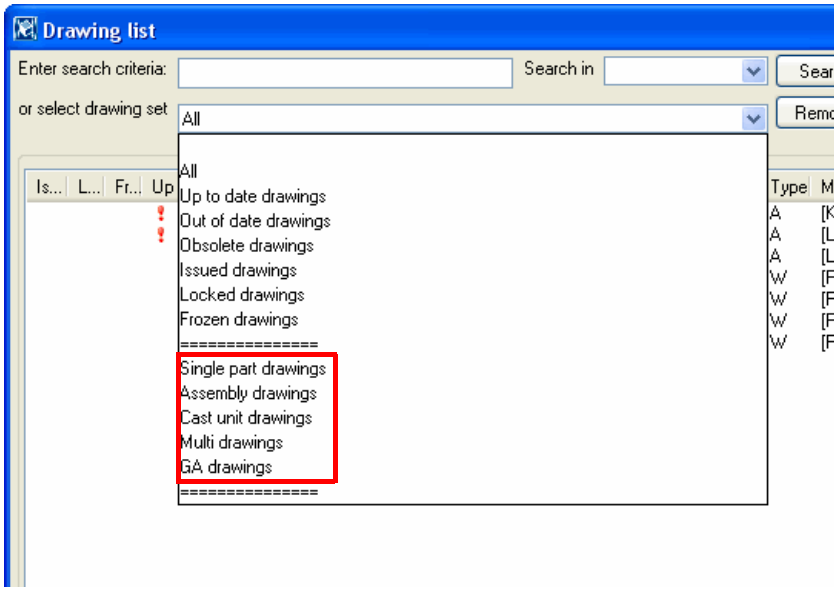
Tekla Structures 13.0

Drawing list

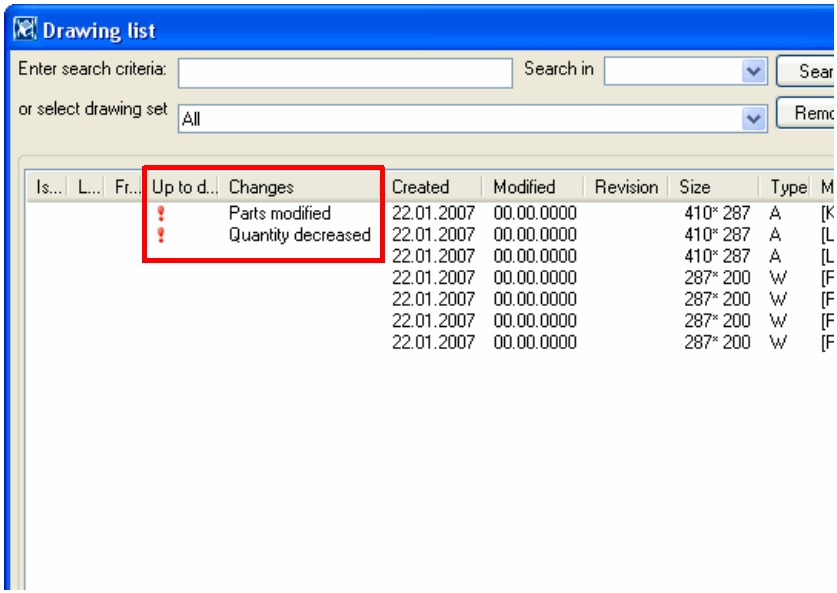
In Tekla Structures 13, the Drawing list is modified and improved. You can use a search function to filter drawings with a specific format or only drawings which are modified:



You can also filter on drawing types, e.g. all workshop drawings or all multi drawings.

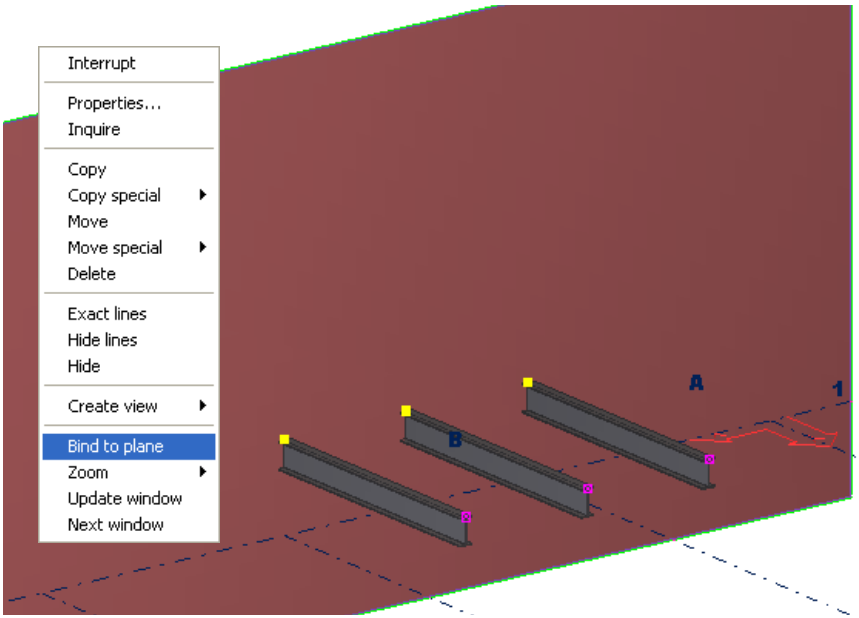


All the changes you make in the model are displayed in the drawinglist with the help of an exclamation mark.



Bind handles to a plane

You can bind more then one handles to a plane in one go, e.g. to a grid plane:



Speed improvements while working with large construction circles

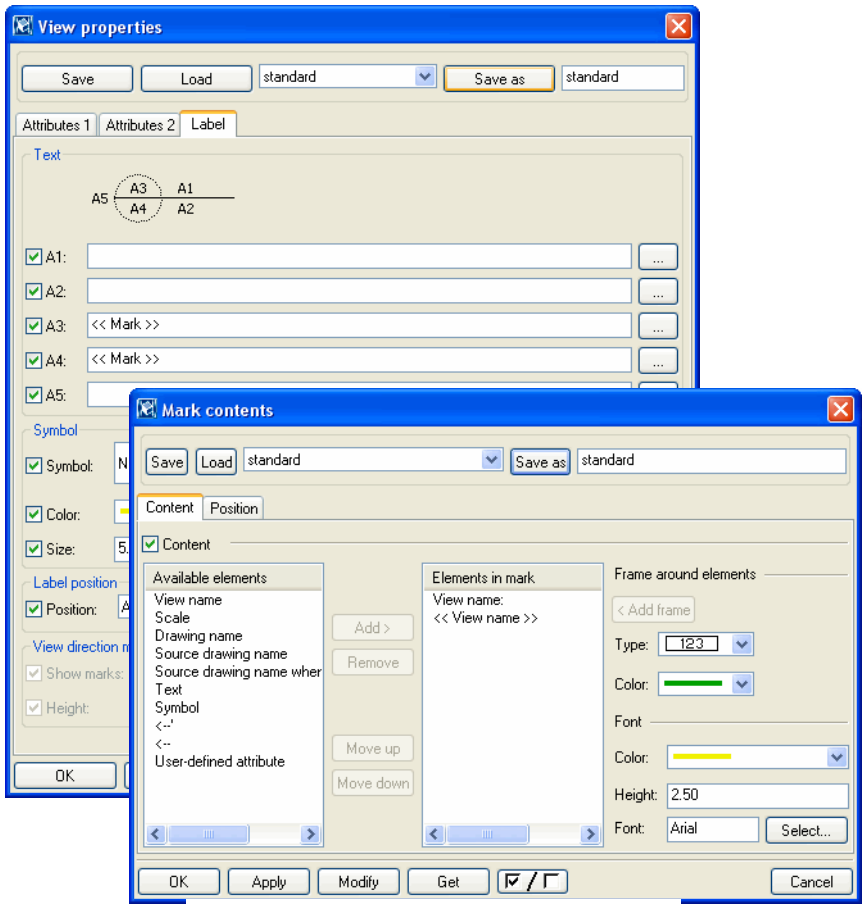
There is no loss in speed while working with large construction circles. In previous versions, the processes in Tekla Structures slowed down.

Speed improvements while opening a large compont catalog

There is no loss in speed while opening a large compont catalog. In previous versions, the processes in Tekla Structures slowed down.

Scale in views

You can display the scale per view:

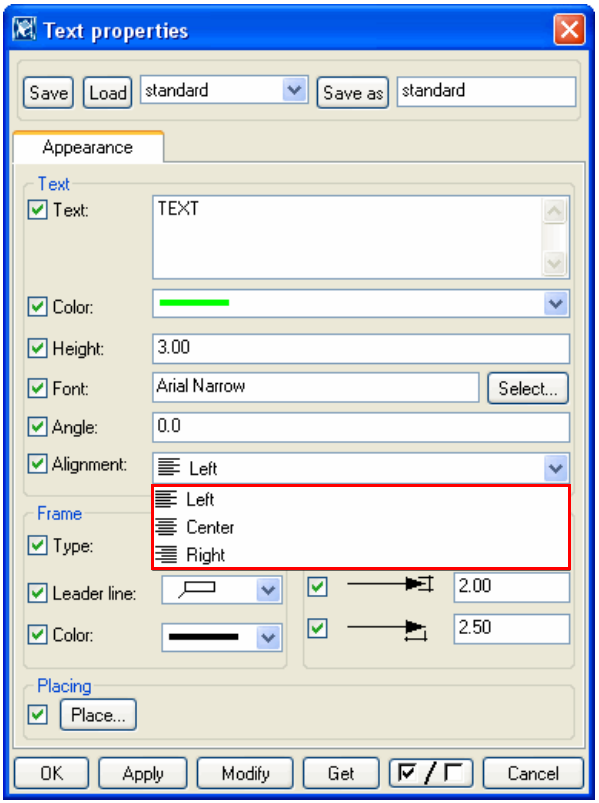


View name: 3D

scale:1/100

Aligning multiple lines of text

In Tekla Structures 13, you can align multiple lines of text in a drawing:



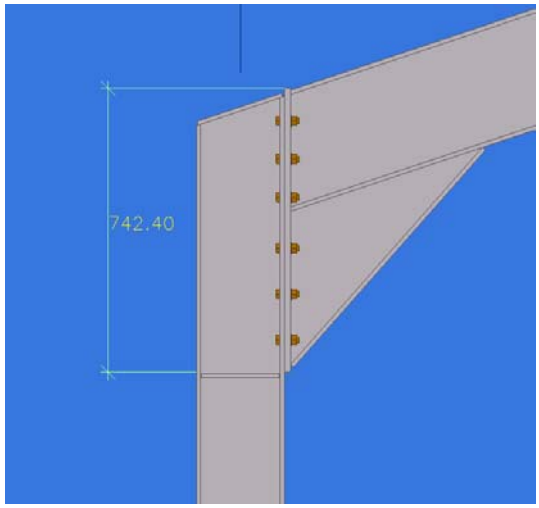
Modify the color of dimensions in the Model Editor

In Tekla Structures 13.0 you can modify the color of the dimension line and the dimension text in model views.

This is handy when you use dark backgroundcolors in the modelviews.

In the modelview

You can set the color of the dimension line and the dimension text seperately:

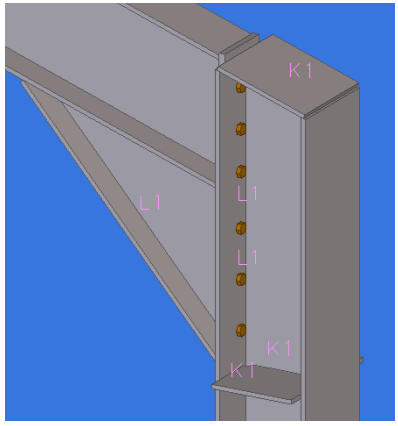


Modify the color of part labels in the Model Editor

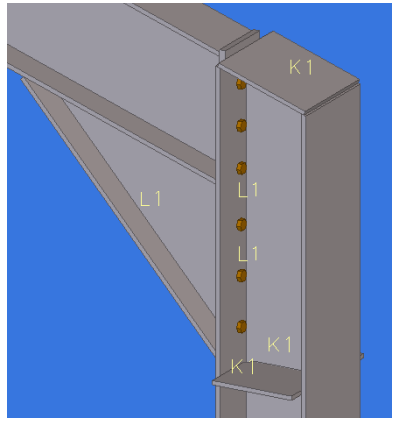
In Tekla Structures 13.0 you can modify the color of part labels in model views.

In the model view

Pink:



Yellow:



Notes:

10 Construsoft Release Info 13.1

Content This Tekla Structures 13.1 Release Info contains the following topics:

General

RHS and Angle steel profiles (pag. 5)
Fixed parametric profiles (pag. 5)
New parametric profiles (pag. 6)
Batch files replaced by ini files (pag. 7)
Mouse with scrollwheel (pag. 18)
Changing file Objects.inp (pag. 19)
Xtranet (pag. 21)
Binding multiple handles to a plane (pag. 21)
Check database and Full numbering (pag. 22)
Select on right click (pag. 24)
Representation of holes in rendered views (pag. 25)
Part Add command (pag. 26)
Report ID_assembly (pag. 26)
Report ID_original drawing (pag. 27)
Report Boltlist complete (pag. 27)
Report Multidrawing_content (pag. 28)
Unique numbers for assemblies (pag. 28)
Change language (pag. 28)
Large workarea in modelview (pag. 29)
Improved icon information (pag. 29)
View button (pag. 30)

Macros (pag. 31)
Reference models (pag. 34)
Im- and export (pag. 35)
TS folder on a server (pag. 36)

Modeling

Notification report and displaying project messages (pag. 37)
Copying loose gridlines (pag. 38)
Speed improvements for large construction circles (pag. 39)
Speed improvement when opening a large component catalog (pag. 39)
Editing colors of dimensions, dimension texts and part marks in the Model editor (pag. 40)
Background color in model views (pag. 41)
View filters keep there setting (pag. 42)
Filtering for bolt diameter, bout length or hole diameter (pag. 43)
Modeling gratings (pag. 45)
New dialog Options (pag. 47)
Displaying all parts objects (pag. 48)
Modeling bolts (pag. 49)
Changing Bolt Standard: diameter remains (pag. 50)
User-defined attributes: Fabricator name, Type and Nomination (pag. 51)
Variable XS_ASSOCIATIVE_REBARS (pag. 51)

Components

Bolted gusset (11) and Tube gusset (20) (pag. 52)
Stiffened end plate (27) (pag. 52)
End plate (29) (pag. 53)
Stairs (S71) and Stairs (S82) (pag. 53)
Panel stabilizer (114) (pag. 54)
Unfold rules surface (21) (pag. 55)
Baseplate (1029) (pag. 56)
Fitting plates in components (pag. 56)
Stringer details (pag. 57)
Components Imbeds (8) and (1008) (pag. 59)

Drawings

Improvements in dialog Drawinglist (pag. 60)
Selecting drawings using filter settings (pag. 61)
Searching and re-using search results (pag. 64)
Selecting all drawings (pag. 65)
New grafical symbols and status text (pag. 65)
Quicker UDA access (pag. 67)
Printing change symbols (pag. 67)
Angle dimensioning, level marks and associative notes (pag. 69)
New macro Remove Change Clouds added (pag. 70)
Displaying Tekla Structures version on drawings (pag. 70)
Scaling objects in drawing views (pag. 73)
Comment-variable in drawings modified (pag. 73)
Aligning Multi text (pag. 76)
New Symbol file "Details.sym" (pag. 77)
Angle dimensions on plate drawings (pag. 78)
Moving the leaderlines start point (pag. 79)
Template B (pag. 79)
Automatic dimensioning of GA drawings (pag. 80)
Displaying connecting parts on an anchorbolt plan (pag. 80)
Labels in drawing views (pag. 82)
Pop marks (pag. 85)
Alternative Level Mark (pag. 86)

Custom components

Password protection for custom components (pag. 87)
Component name and Component attribute file (pag. 87)

NC files

DSTV convert tool (pag. 90)

Multi user

Multi user server (pag. 92)
Opening a multi-user model (pag. 93)
Saving model revision notes (pag. 93)

General

Tekla Structures 13.1 contains many new features and fixes.

Compatibility

Tekla Structures 13.1 is compatible with all previous versions. You can open and work with existing models using Tekla Structures 13.1. Note the following issues:

- To avoid conversion problems, **always open a model created with an older version of Tekla Structures as a single-user model first**.
- Updating drawings created with an older version of Tekla Structures in Tekla Structures 13.1 may cause problems. We recommend that you complete any drawings you have started using your current version or recreate the drawings using Tekla Structures 13.1.
- We recommend that you always check that custom components created in older versions work correctly in the new version of Tekla Structures.

  When you open custom components made with an older version of Tekla Structures in the Custom component editor, and the new version contains improvements requiring update, Tekla Structures asks whether you want to update the component. If you do not update the component, it works in the same manner as in the version where it was originally made, but you do not gain the benefits of the improvements.

  If you choose to update the component, you need to check and sometimes even recreate dimensions depending on the improvements. When you delete a dimension and create a new one (even with the same name), also the equations containing the dimension need to be edited, because the dependency created by the equation is lost when a dimension is deleted. You can recreate the dimensions and edit the equations easily in the Custom component editor.

We suggest that you complete any models you have started using your current version.



Once you save a model in Tekla Structures 13.1, you can open it in Tekla Structures 13.1, but not in older versions.

Installing

Tekla Structures setup creates a new subfolder for Tekla Structures 13.1.

You need Windows Administrator rights to install Tekla Structures 13.1. This ensures that all Windows system folder *.dll files are updated.

RHS and Angle steel profiles

Parametric angle steel and RHS profiles (i.e. notation including "**x**" and "*****" characters) are removed from file `profitab.inp`.

E.g. L70/7 and L70*7 are equal profiles, but have different names. This means they will receive different position numbers. So this is a safety issue.



When opening old models in Tekla Structures 13.1, containing parametric RHS- and/or angle steel profiles, these profiles will not be displayed!!!

Go to **Tools** > **Check database**. An "Illegal profile" message will appear; add the missing profiles to the database.

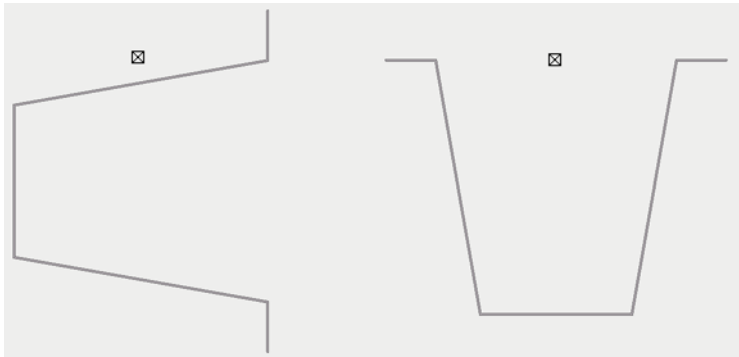
It is allowed to undo this removal by adding the lines again to file `profitab.inp`.

Fixed parametric profiles

In Tekla Structures 13.1, a large number of parametric profiles have been fixed. Both positioning and representation of the parametric profiles have been updated.

Example

Omega profiles:



Tekla Structures 12 Tekla Structures 13.1



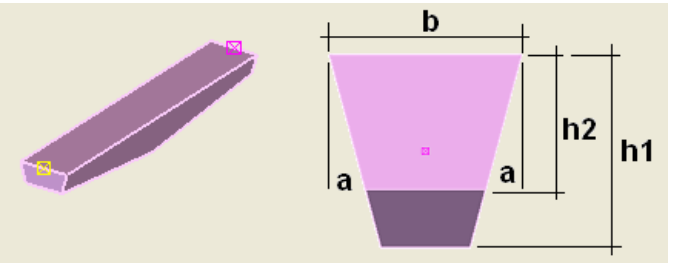
When opening old models in Tekla Structures 13.1, containing parametric profiles, these profiles will be positioned incorrect!!!

New parametric profiles

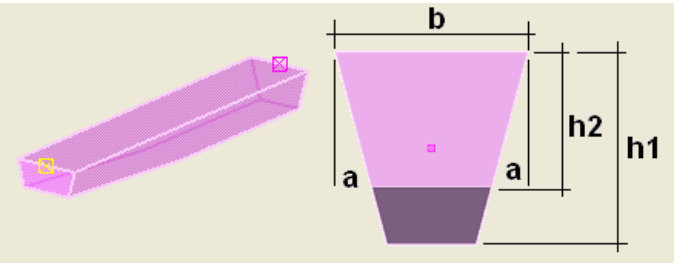
Some new parametric profiles have been added , like **Gutter** and **LT**. Take a look at the current available parametric profiles; new parametric profiles are added regularly (CMDR, CMBEUR, CMDOOR, etc).

Gutter

GUTTER:

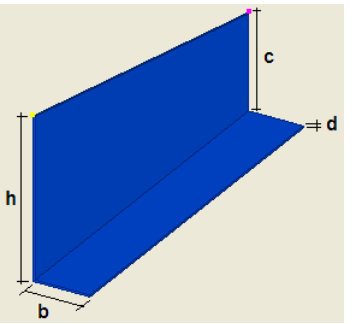


GUTTERTR:



LT

Conical L profile:



Batch files replaced by ini files

Ini files

Batch files are no longer used in Tekla Structures 13.1; `ini` files are used instead (*.ini).

This change was necessary to certify Tekla Structures for Windows Vista and to make it Windows Vista compatible.

`Ini` files can be modified in the same way as `batch` files.

`Ini` files are textfiles, which can be opened in any text editor.

The modifications in relation to Tekla Structures 12.1 are:

TS 12.1	TS 13.1
`europe_env.bat` in folder `..\bat\environment`	`env_europe.ini` in folder `..\environments\europe`
	`env_global_default.ini` in folder `..\environments\country-independent`
`nld.bat` in folder `..\bat\language`	`lang_nld.ini` in folder `..\13.1\nt\bin`
`user.bat` in folder `..\bat\user`	`user.ini` in folder `..\13.1\nt\bin`

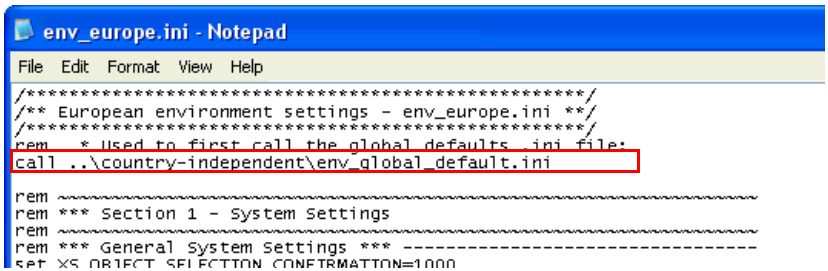


How are the new `ini` files being read? The Tekla Structures 13.1 shortcut contains the following lines:

C:\TeklaStructures\13.1\nt\bin\TeklaStructures.exe -i
C:\TeklaStructures\13.1\environments\europe\env_europe.ini -i
C:\TeklaStructures\13.1\environments\europe\std.ini -i
C:\TeklaStructures\13.1\nt\bin\user.ini

These `ini` files are being read as:

1. The file **TeklaStructures.exe** is being executed
2. The file **env_europe.ini** is being read



In this file the line started with `call ...` is being read, the file **env_global_default.ini** is being read.

3. The file **std.ini** is being read, dependable of the module being used:



This, to make difference between the various software configurations (e.g. STD (Steel Detailing) and SDE (Standard Design). Customer specific settings using the STD module are now stored in folder `...europe\module\std`.

4. The file **user.ini** is being read

The `ini` files are being read in this order. This means that lines in file **user.ini** have the highest priority and therefore will overrule identical lines in previous `ini` files.

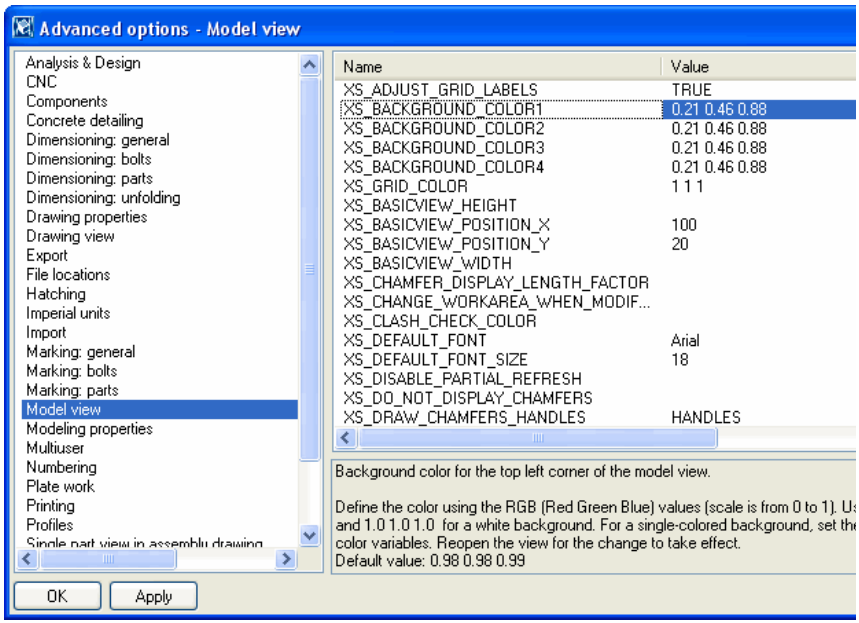
Modifying variables

You can modify variables on two levels:

- Model level
- User-specific level



Go to **Tools** > **Advanced options...** or press shortkey **<Ctrl> + E**. The following dialog will appear:

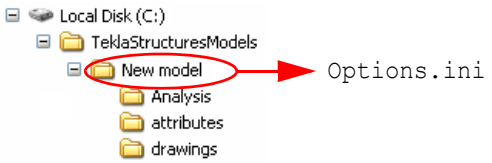


Variables are grouped in categories. Categories are listed on the left side. Click a category to view the accompanying variables.

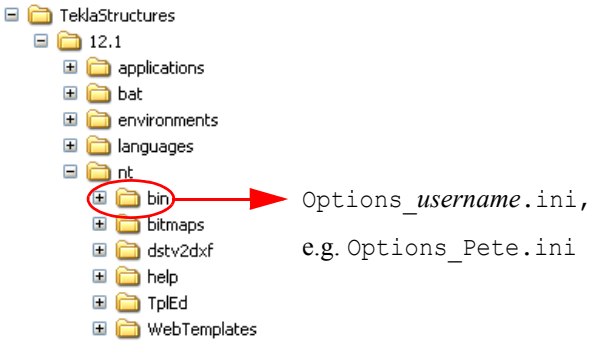
Several variables have a short description to decribe the working of the variable. We plan to have descriptions for all variables in a future version.

Modifications are stored as follows:

1. This file contains modified variables on model level; these changes apply to the current model only.



2. This file contains all modified user-specific variables which apply to all models which are opened by that particular user.



If you want to make changes which affect all models, we recommend to apply modifications in file **user.ini**, this file is stored in folder: `..\TeklaStructures\13.1\nt\bin`.



Note: when copying lines to **user.ini**, be sure that the lines start with `set`!

In this way the changes are automatically copied when installing a next version of Tekla Structures.

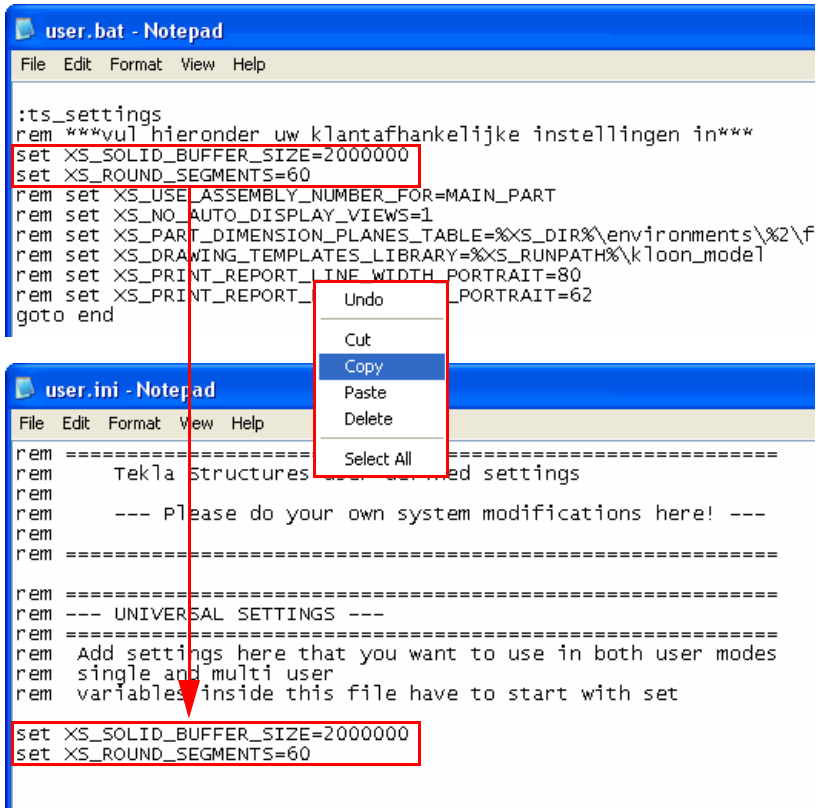
See also For more information about `ini` files, see **Tekla Structures F.A.Q.**.

Copying batch file settings to ini files

When you already use Tekla Structures you want to copy your old 'batch' settings to Tekla Structures 13.1.

Step plan

1. Open file **user.ini** , located in folder:
   `..\TeklaStructures\13.1\nt\bin.`
2. Open file **user.bat** from the previous version, located in folder:
   `..\TeklaStructures\version\nt\bin.`
3. Copy de desired lines from file `user.bat`.



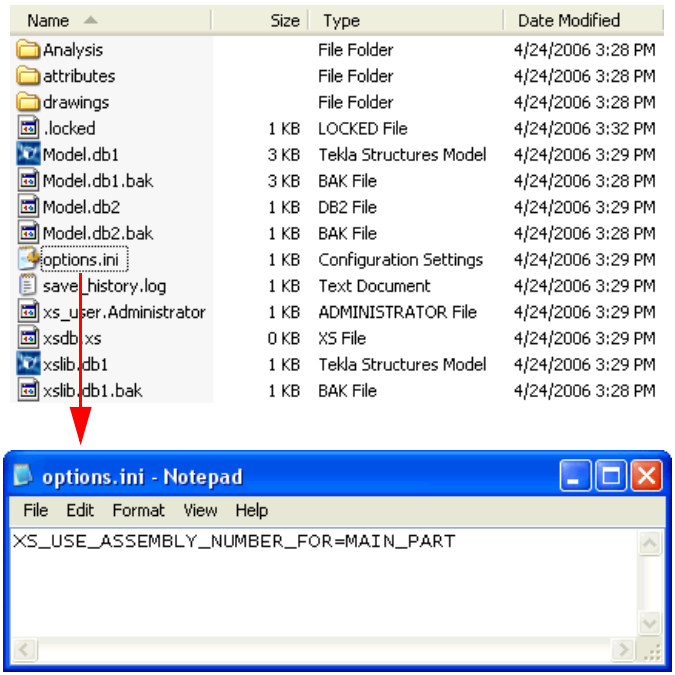
4. Paste the lines in file `user.ini`.
5. Save file `user.ini`.

Working for multiple contractors

Using a default model

If you work for multiple contractors, you can choose to use a standard model for each contractor.

You can do this by storing a **options.ini** file into a default model. This file contains the modified contractor-specific variables.



It is also possible to save the specific settings (e.g. drawing settings, templates and template-layouts) in the contractors default model. (`*.bin` files in the modelfolder, other files in `attributes` folder).

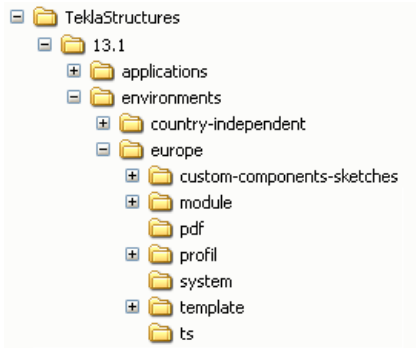
Each time you save the default model to a new name, all settings are automatically saved into the new model, so all model variables are used.

Notes

- If you want to modify the saved file `options.ini` (in the model-folder) , it is allowed to modify the file directly using a random text editor (e.g. Notepad). So it isn't necessary to use the **Advanced options.**
- Copying the file `options.ini` from one model to another model is allowed. If the target model is already opened, re-opening the model is required to apply the changes.

Using a contractor depended folder

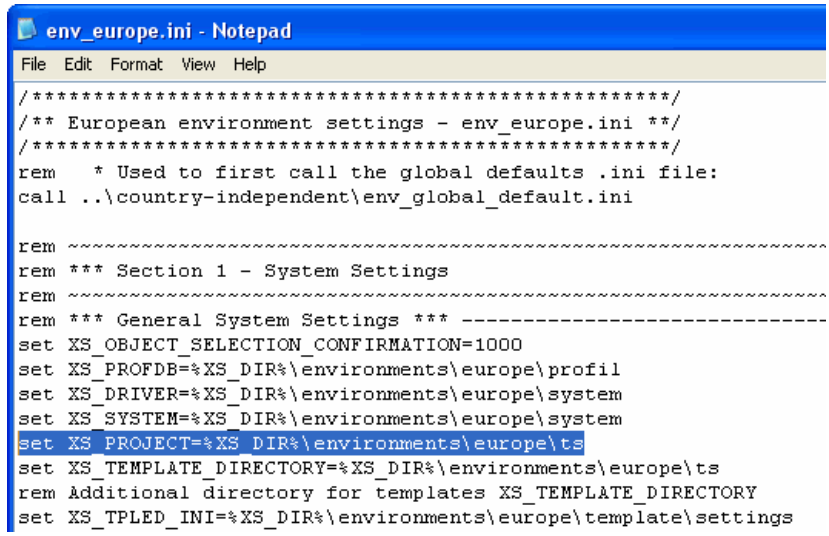
If you use a standard model, then the variables and setting files are stored in the model folder; the other method is to store these contractor-specific files in a special folder (like like storing your personal files in the `ts` folder.



Create multiple shortcuts, to which the variables and settings are linked.

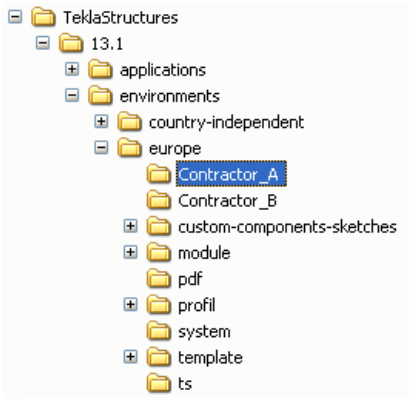
When starting Tekla Structures, several `ini` files are read, including `env_europe.ini` and `user.ini`.

By default the settings in the ts folder are used, because the file `env_europe.ini` contains a reference to that folder:



If you are going to use settings from a contractor (e.g. drawing settings, templates and layouts), then it is best to ask for the contractors complete `ts` folder; this folder contains all contractor specific settings.

Next, in the `europe` folder, create a new folder, name it after the contractors name and copy the content of the contractors `ts` folder to this folder:

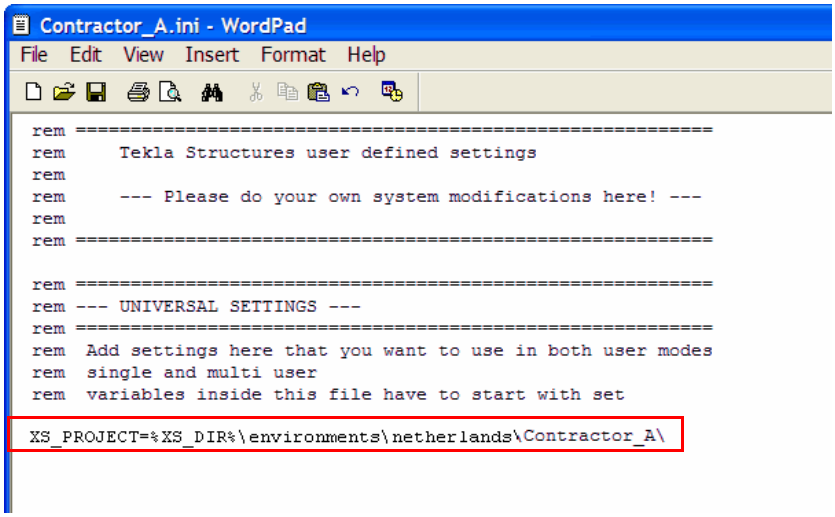


Now go to file `user.ini` (located in folder ..\TeklaStructures\13.1\nt\bin)and create a copy of this file. Rename the file to `Contractor_A.ini`.

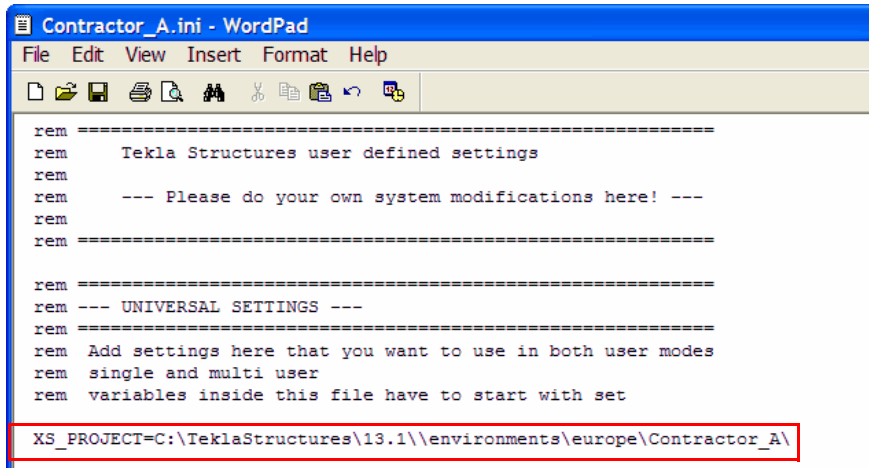
Next, copy the following line from file **env_europe.ini**

```
set XS_PROJECT=%XS_DIR%\environments\europe\ts\
```

and add this line to file `Contractor_A.ini:`



Change the path by replacing the link into the full path:

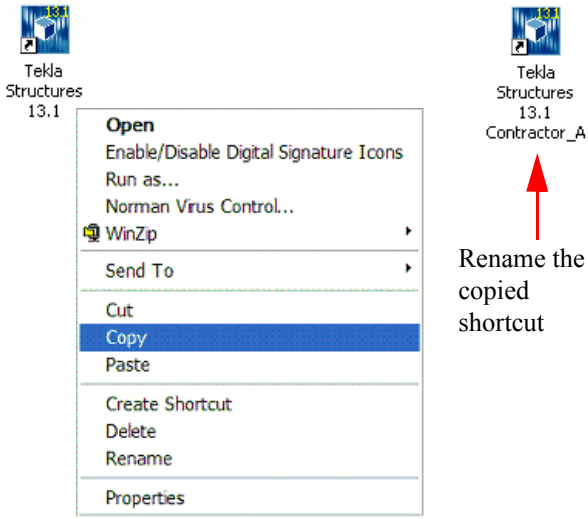




You **must** enter the full path.

Replacing `ts` into `Contractor_A` is not sufficient!

Copy the shortcut on the desktop and rename it:



Go to the shortcut's properties. For 'Target', replace the link to `user.ini` into `Contractor_A.ini`. This results in this file being read, including variable:

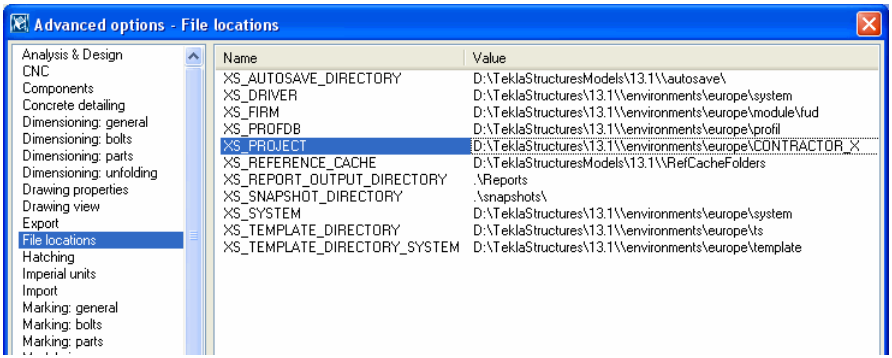
```
set
XS_PROJECT=C:\TeklaStructures\13.1\environments\europe\Cont
ractor_A\
```

When working for this contractor, always start Tekla Structures using this shortcut.

Using a standard model and Contractor specific settings

You can also use a standard model but have the settings in the contractor depended folder instead in the model folder.

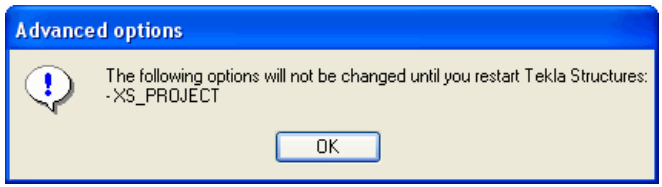
- Go to **Tools** > **Advanced options...**
- On the left side, select the **File locations** category



- On the right side, click line XS_PROJECT

Change the value to C:\TeklaStructures\13.1\environments\europe\Contractor_A

- Click <**Apply**>, the next dialog shall appear:



- In dialog Advanced Options, click <**OK**>

The file `options.ini` has been added to the standard models folder. This means that the folder Contractor_A will be read first when opening a model.

Reference to server

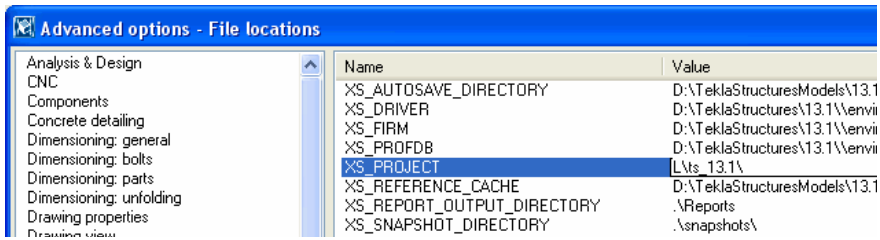
The `ts` folder is installed automatically. If multiple licenses are in use in one company, then it is useful that every user uses the same settings for profiles, drawings, templates, ect.

Therefore it is useful to place the `ts` folder of a serverdisc, so that every user uses the settings from that location.

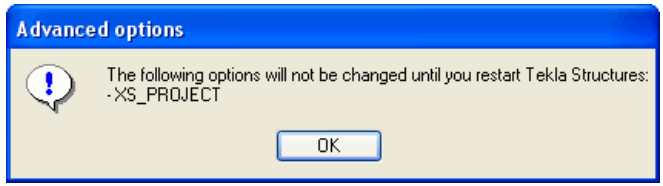
This increases uniformity and cuts maintenance time becasue only one folder needs to be maintained. Also upgrading towards a new Tekla Structures version is much easier. How to set-up the workstations?

Step plan

- Go to **Tools** > **Advanced options...** and select the **File locations** category
- On the right side, click XS_PROJECT
- Change the value to, e.g. `L:\ts_13.1\`



- Click <**Apply**>, the next dialog shall appear:



- In dialog Advanced Options, click <**OK**>
- Now, open the model folder en double click the file **options.ini**
- Copy the line `XS_PROJECT=L:\ts_13.1\`
- Open the file **user.ini** (located in folder **..\TeklaStructures\13.1\nt\bin)**
- Paste the line in file **user.ini**
- Save the file **user.ini** and restart Tekla Structures.

Autosave

We recommend to save the autosave files (set by variable XS_AUTOSAVE_DIRECTORY in **Tools** > **Advanced options > Files locations**) to a local disc, and **not** to a server disc. This decreases the network traffic.

Pro's and Con's: Settings (files from the folders `system`, `ts` and `profil`) are stored in the cache memory.

The advantage is that all settings have to be loaded just once. Especially if all data is stored on a server, or when running in multi-user mode.

There is also one disadvantage: speed.

Tekla Structures loads the settings, but this takes more time because reading from a network location is about 5 times slower then reading from a local disc.

Mouse with scrollwheel

There are two methods for zooming using the scrollwheel:

1. In the mouse-mode, you can zoom by turning the scrollwheel.
2. In the panmode, you can zoom by holding down the scrollwheel while scrolling.

The scroll zooming can be affected by three variables.



To modify these variables, go to **Tools** > **Advanced options > Model view...**

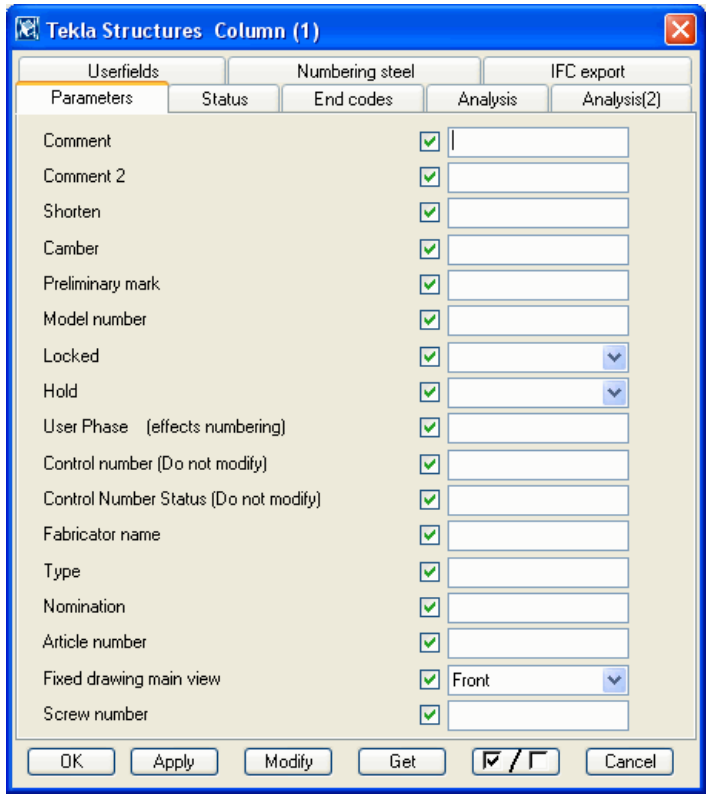
In variable **XS_ZOOM_STEP_RATIO** , you can set the zoom ratio (i.e. zoom speed) when using the mouse. Default value is 0.25. Increase this value if you want to a quicker zoom.

The zoom ratio for panning using the scrollwheel (not holding down) is set in variable **XS_ZOOM_STEP_RATIO_IN_MOUSEWHEEL_MODE**. Default value is 0.25.

The zoom ratio for panning using the scrollwheel (holding down) is set in variable **XS_ZOOM_STEP_RATIO_IN_SCROLL_MODE**. Default value is 0.02.

Changing file Objects.inp

Many dialogs contain User Defined Attributes for various objects, such as beams and columns:



These user defined fields can be modified in file `objects.inp`. It is also possible adding tabs and fields.

This make it easy to use contractor specific attributes.



Modifing the original file `objects.inp` in the folder `..version\environments\country-independent\inp` must be done for every new installation, so also for Tekla Structures 13.1.

Copying the original file `objects.inp` from one version to another is **not** allowed.

If the file `objects.inp` has been changed, all fields which affect numbering must be checked:

Tekla Structures 12.1

```
part(0,"Part")
{
 tab_page("Supplier")
  {
    unique_attribute("Supplier", "Supplier", option,"%s", no, none,"0.0",
    {
        value("j_default", 2)
        value("none (excisting)", 0)
        value("Steel supplier", 0)
        value("Concrete supplier", 0)
        value("Wood supplier", 0)
        value("Window frame supplier", 0)
        value("Material supplier", 0)
        value("Step supplier", 0)
        value("Glass supplier", 0)
    }
    unique_attribute("Free attribuut", "Free attribuut", string,"%s", yes, none,
    {
        value("0.0", 0)
    }
  }
 }
beam(0,"j_beam")
{
```

Tekla Structures 13.1

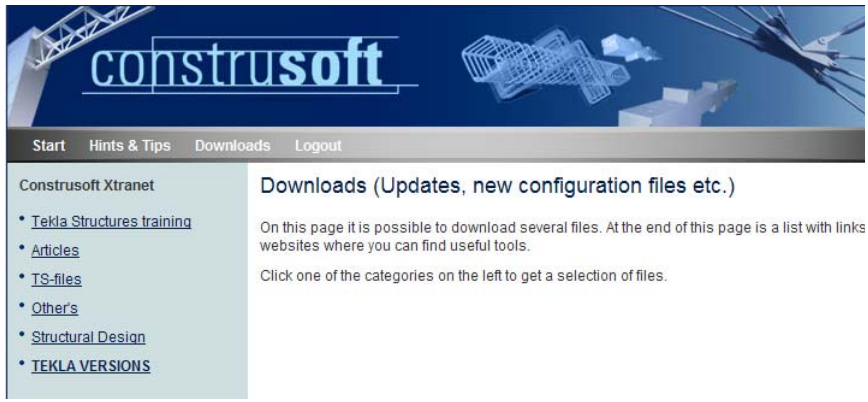
```
part(0,"Part")
{
 tab_page("Supplier")
  {
    unique_attribute("Supplier", "Supplier", option,"%s", no, none,"0.0",
    {
        value("j_default", 2)
        value("none (excisting)", 0)
        value("Steel supplier", 0)
        value("Concrete supplier", 0)
        value("Wood supplier", 0)
        value("Window frame supplier", 0)
        value("Material supplier", 0)
        value("Step supplier", 0)
        value("Glass supplier", 0)
    }
    unique_attribute("Free attribuut", "Free attribuut", string,"%s", yes, none,
    {
        value("0.0", 0)
    }
  }
 tab_page("Supplier","Supplier",1)
 modify(1)
}
beam(0,"j_beam")
{
```

This counts for both the original file `objects.inp` and the identical named files in the model folders.

Xtranet

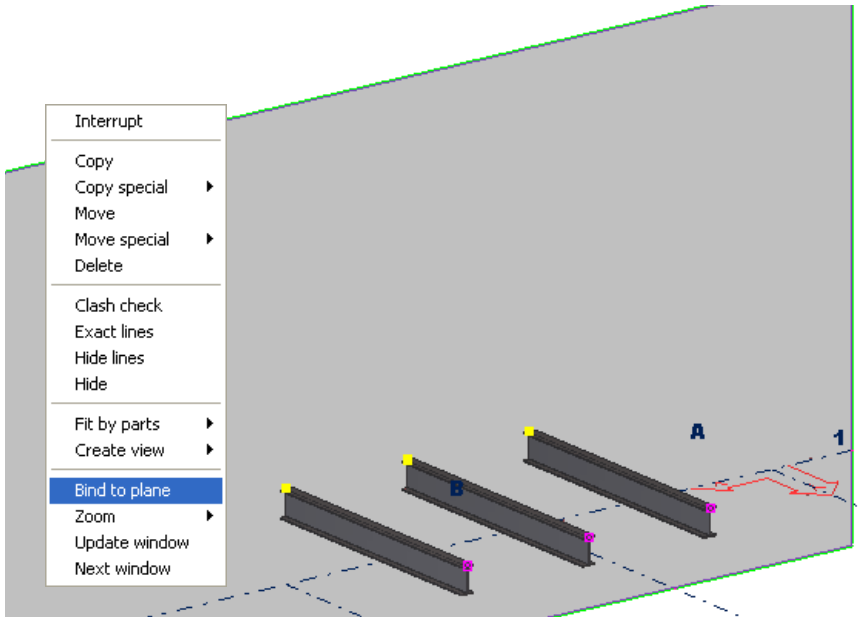
Downloads

More and more customers use our Xtranet. You can find the new versions, manuals, example models ect, in the download section.



Binding multiple handles to a plane

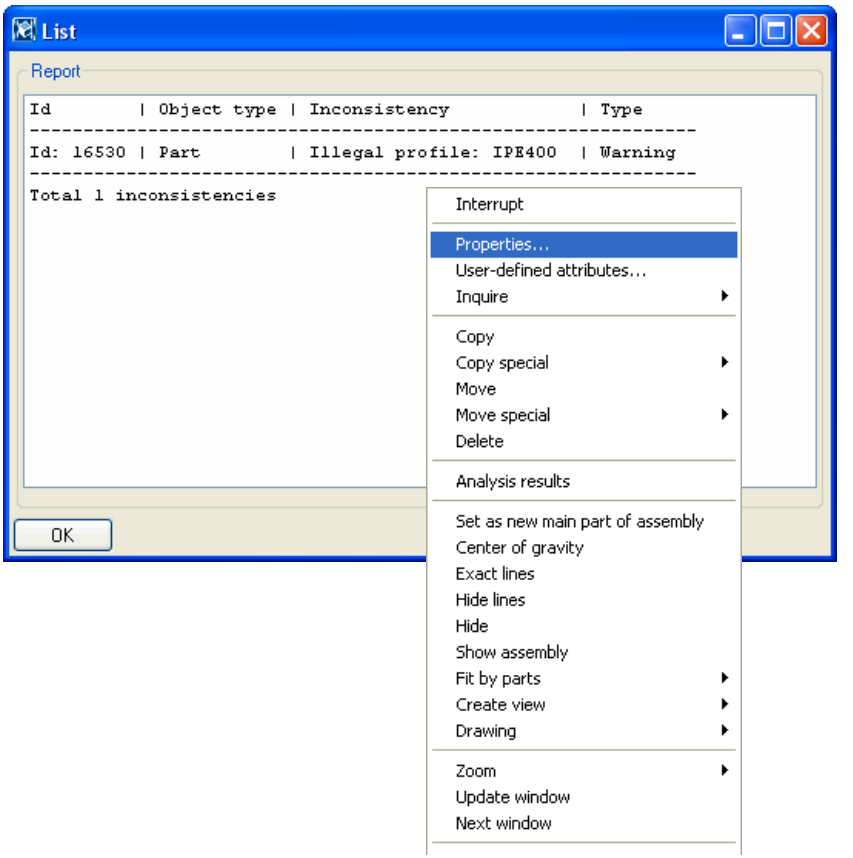
A major improvement in creating and editing custom components (and parametric modeling in general), is the possibility to bind multiple handles to a plane at once. This makes it more easy to define planes.



Check database and Full numbering

It is recommended to check the model for inconsequences regularly. This can be done using command **Tools > Check database**. This command checks for the presence of profiles and for possible "model inconsequences". Especially for models created by another, checking the database is strongly recommended.

The so called model inconsequences must always be removed. To do this, go to **Tools > Check database**. The errors are written in logfile *check_database.log* and stored in the model folder. By right-clicking a ""illegal profile"-line, you can get the profile properties:



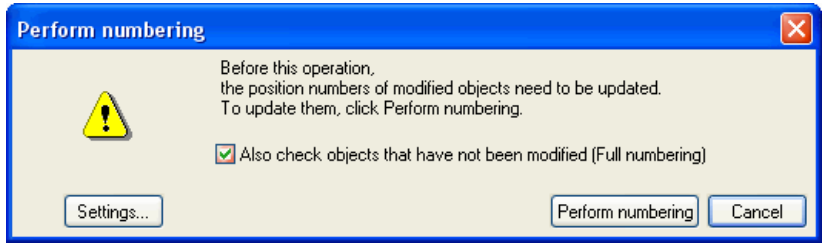
If the model is error-free after executing the Check Database command, this message shall appear in the Status bar:

Database checked and OK

See also For more information about the Check Database command, see **Tekla Structures F.A.Q.**

Full numbering

Just like the **Check Database** command, it is recommended to perform a Full Numbering command every day (insted of Modified Numbering). To do this, check the option **Also check objects that have not been modified (Full numbering)**.

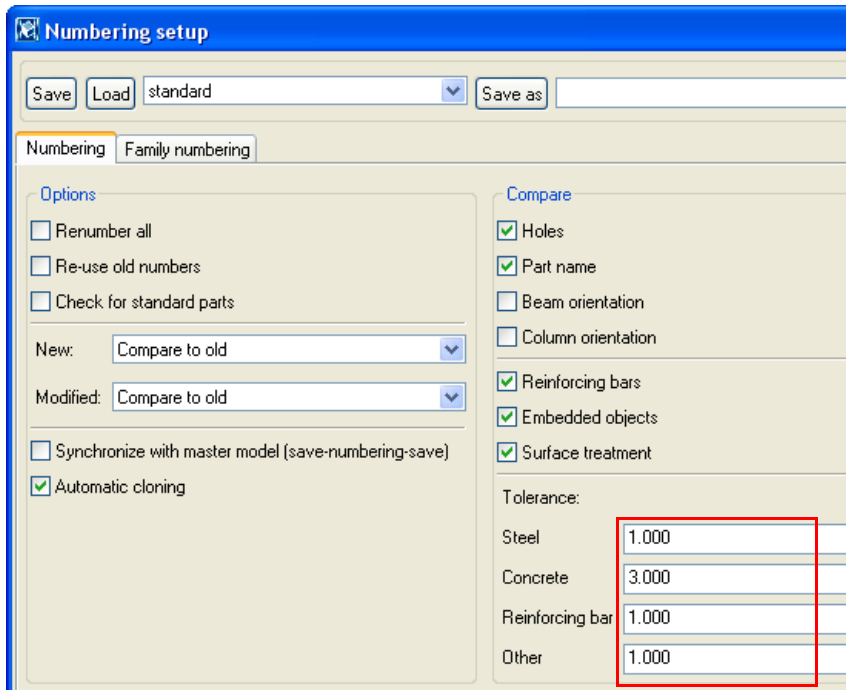


In some situations, it can happen that certain drawings receive an update-mark, even when nothing has been modified in the model.

Performing a Full Numbering will solve this issue.

Numbering setup

If a model has been numbered, the following tolerance has been used:



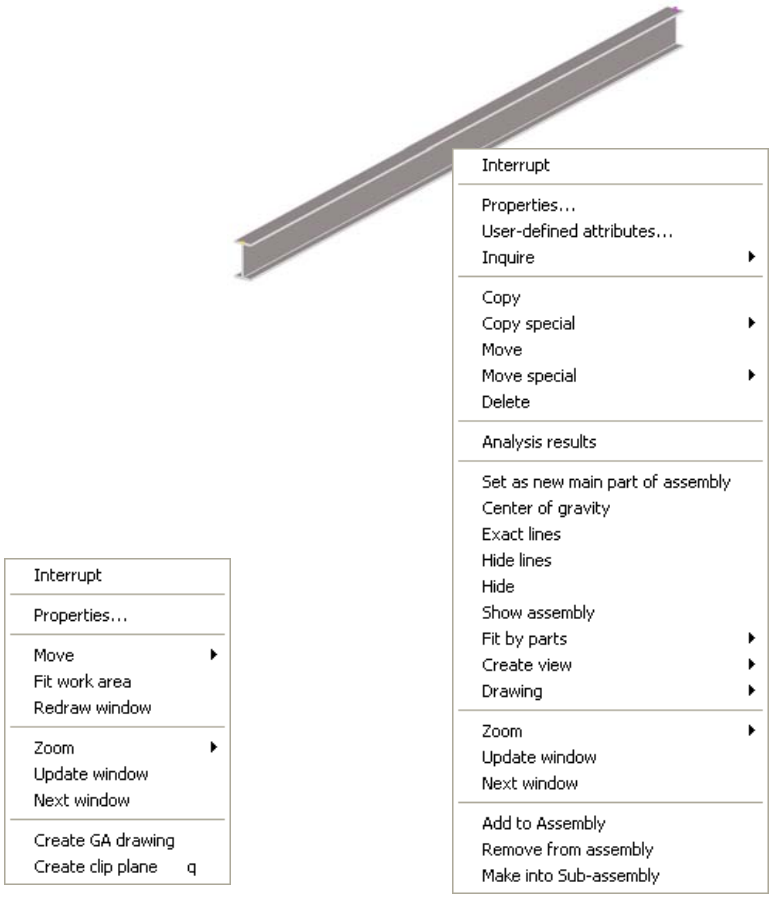
Look carefully to this setting; possibly you want to perform numbering with a higher (e.g. for concrete parts) or lower tolerance.

Select on right click

You can now also select objects by using the right mouse button. For this, go to **Setup** > **Select on right click**. This setting is switched off by default.

Right click

In case you switch this setting on, a pop-up menu appears automatically when you select an object by right-clicking:



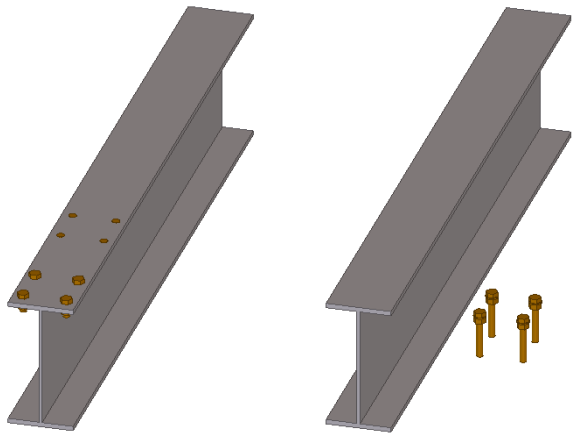
OFF **ON**

Left click

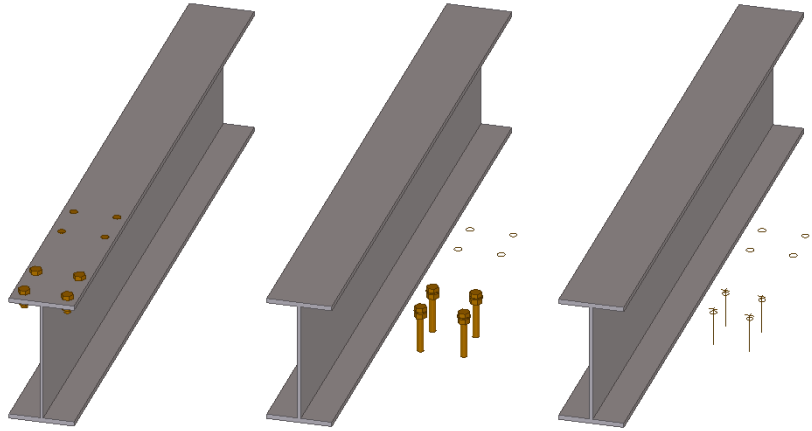
In case you select an object by left-clicking, you need to click the right mouse button after that to pop up the menu.

Representation of holes in rendered views

Holes outside profiles are now displayed in rendered views, in both **Fast** and **Exact** mode:



Tekla Structures 12.x

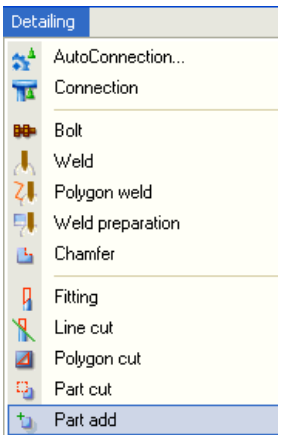


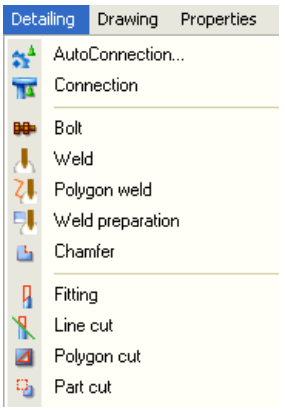
Exact **Fast**

Tekla Structures 13.1

Part Add command

The **Part Add** command has been removed from the pull down menu:





Tekla Structures 12.x Tekla Structures 13.1

The icon is still available (it's located in the **Detailing** toolbar) but only if the accompanying Registry key (located in the system-folder) has been added to the Windows Registry.



Report ID_assembly

In report **Id_assembly**, a column named **Volume (Vol)** has been added:

```
-------------------------------------------------------------------------------
TEKLA STRUCTURES ID ASSEMBLY LIST                          Page:    1
Model: test1                                               Project: 2007-123456
                                                           Date:    06.11.2007
                                                           Time:    13:47:08
-------------------------------------------------------------------------------
ID          Assembly  Profile          Length   Vol    Grid        Level
-------------------------------------------------------------------------------
Id:     15014 L12     IPE300          14592     1.4    A-D/2       +5.984
Id:     19058 L3      IPE300          14592     1.1    A-D/4       +5.984
Id:     78054 L8      UNP140           6992     1.0    D>/2-3      +2.700
Id:      8700 L2      IPE240          14588     0.8    A-D/5       +5.984
Id:     17177 L1      IPE300          14592     0.7    A-D/3       +5.984
-------------------------------------------------------------------------------
```

If there are abnormalities in the volumes, this may point to assemblies which are welded to each other unintentional! These large volumes are listed on top.

Report ID_original drawing

You can use report **Id_original_drawing** to find out to which part the drawing is linked:

To use the report: In the drawing list, select a drawing and create the report **Id_original_drawing:**

```
-------------------------------------------------------------------------------
TEKLA STRUCTURES ID_ORIGINAL_DRAWING_LIST                  Page:     1
Model:                                                     Project:  project number
                                                           Date:     13.11.2007
                                                           Time:     13:30:23

-------------------------------------------------------------------------------
     Original part for drawing
-------------------------------------------------------------------------------
     Id:858
-------------------------------------------------------------------------------
                                    END
-------------------------------------------------------------------------------
```

If a line is selected, the corresponding part will be highlighted in the model.

Report Boltlist complete

A new list **Boltlist complete** is available. This list shows bolts, nuts, washers, fillerplates and threaded ends:

```
-------------------------------------------------------------------------------
TEKLA STRUCTURES BOLTLIST COMPLETE                         Page:     1
Model: templates_ENU                                       Project:  project number
                                                           Date:     26.11.2007
                                                           Time:     09:26:54

===============================================================================
 Bolts, Nuts, Washers
-------------------------------------------------------------------------------
             Diam  Length(mm) Type                    Number   Type
      -------------------------------------------------------------------------
      Bolt    16       55     931/125A/934              4   931
      Nut     16              934                       4
      Washer  16              125A                      4
===============================================================================
 Fillerplates
-------------------------------------------------------------------------------
      Partno.      Profile       Material       Length   Number    Weight
      -------------------------------------------------------------------------
      H1           PL2*150       S235JR          290.0   2         0.7
      -------------------------------------------------------------------------
                                              Totaal:     2         1.4
===============================================================================
 Threaded ends
-------------------------------------------------------------------------------
      Partno.      Profile       Material       Length  Number
      -------------------------------------------------------------------------
      Pr15         R20           S235JR          200.0   4         0.5
      -------------------------------------------------------------------------
                                             Total:     4         1.9
```

Only profiles containing the name **FILLERPLATE** en **THREADED_END** will be selected and displayed on the list.

Report Multidrawing_content

A new report **Multidrawing_content** is added. This report displays the content of a multidrawing:

```
------------------------------------------------------------------------------
MULTIDRAWING LIST                                          Page:     1
Model:                                                     Project:  project numb
                                                           Date:     13.11.2007
                                                           Time:     13:59:01

-------------------------------------------------------------------------------
Name       Drawing size   Created     Revision          Multidrawing content
-------------------------------------------------------------------------------
PLATES      594 *420      13.11.2007     0     POS.      NAME          PROFILE
-------------------------------------------------------------------------------
                                                W P1      PLATE         PL10*3000
                                                W P3      PLATE         FLAT10*240
                                                W P6      ENDPLATE      FLAT10*150
-------------------------------------------------------------------------------
STANDAARD   1189*841      13.11.2007     0     POS.      NAME          PROFILE
-------------------------------------------------------------------------------
                                                A C1      COLUMN        HEA240
                                                A C3      COLUMN        HEA240
                                                A C4      COLUMN        HEA240
-------------------------------------------------------------------------------
```

Also the parts drawing type is listed (W = Workshop drawing, A = assembly drawing).

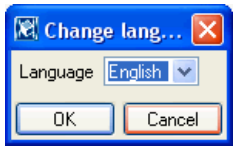
Unique numbers for assemblies

When using unique numbers for assemblies, it's more easy to check welds per assembly. The new variable XS_UNIQUE_ASSEMBLY_NUMBERS makes it possible to set unique partnumbers for each assembly, also for identical assemblies.

If this variable is switched on, all assemblies receive an unique assembly number. Parts are still numbered as before. To modify the variabele, go to **Tools** > **Advanced options** > **Numbering**. By default this variable is switched off.

Change language

Modifying the language in Tekla Structures has been simplified. Go to **Tools**> **Change language...**



A Tekla Structures retart is required to apply the change.

Large workarea in modelview

Occasionally, the workarea becomes very large. The reason is that one or more points are positioned very far away from the models origin. To re-adjust the workarea:

- Set the **Select Switches** toolbar in a way that only points can be selected and make sure the **2nd** button is switched on:



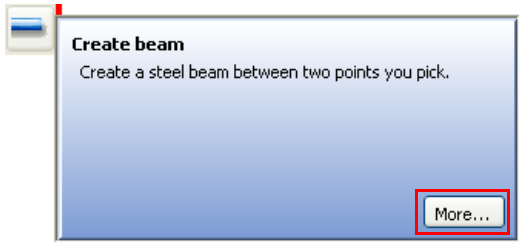
- Go to **Edit** > **Select all** (or <**Ctrl**> + A, all points will now be selected
- Check the status bar to see the amount of selected objects and press <**Delete**>
- Now fit the workarea.

Improved icon information

A new style of icon tips is introduced in Tekla Structures 13.1.

More command information is displayed now, sometimes including examples and tips.

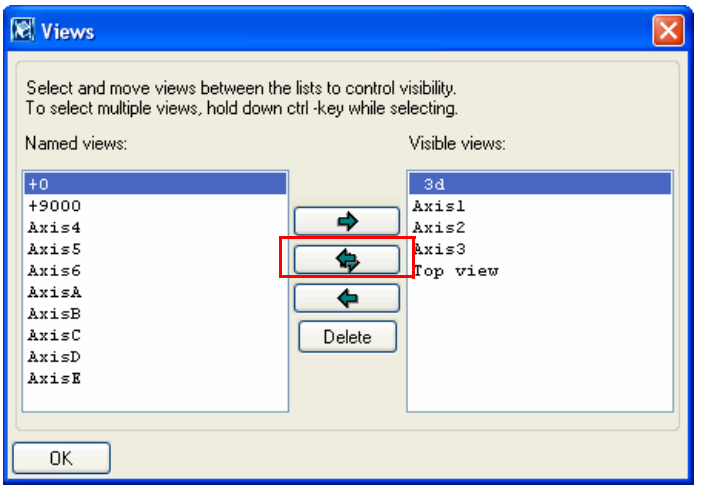
From the icon tip window, more information about the command can be gathered by clicking the **More** button:



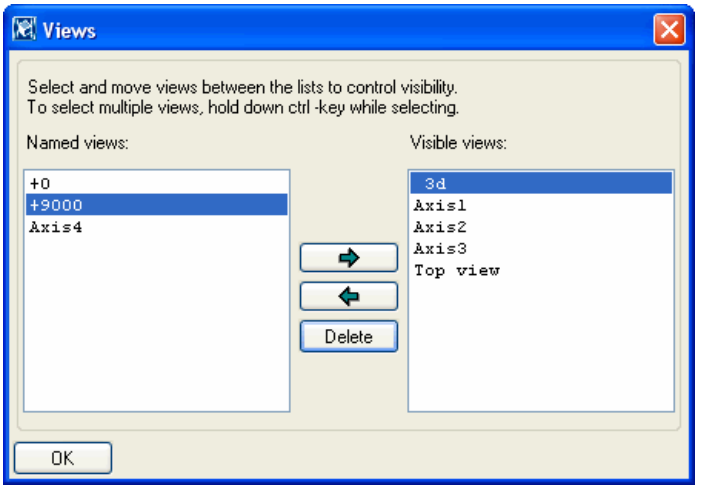
The improved icon tips is switched on by default. To switch off, go to **Help** and select **Enhanced Tooltips**.

View button

The ’double arrow’-button is removed from the View dialog since Tekla Structures 12.0. This button was able to show/hide multiple views at once:



Many users got confused from this button. The remaining buttons are better understandable:



Macros

What are macros

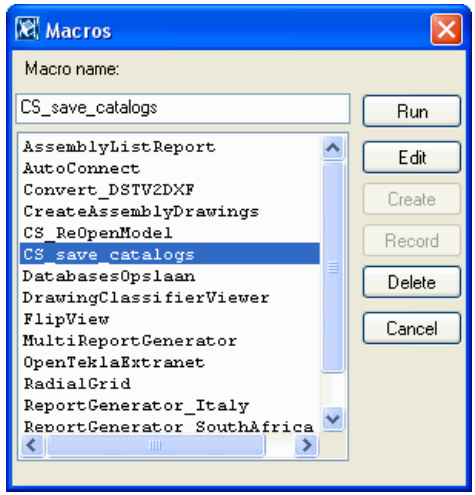
Macros are sets of operations which can be recorded from menus, dialogs short-keys ect. Usually it concerns often-used commands.

Record a macro

To record a macro:

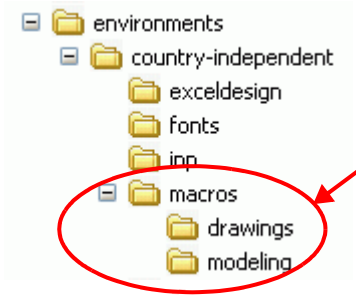


1. Click **Tools** > **Macro...** to open the **Macros** dialog.



2. Enter a name.
3. Click **Record**.
4. Do the operations to be recorded.
5. Click **Stop** to quit recording.

Routines are saved as `*.cs` files in the folder `../environments/country-independent/macros`:



The recorded routine is saved in the `drawings` or `modeling` folder , depending on the module you are working in..

To create a macro manually, click **Create** in the **Macros** dialog. An empty macro file is created, which can be edited manually, or you can copy/paste the commands to another macro..

Running a macro

To run a macro:

1. Click **Tools** > **Macros...** to open the **Macros** dialog.
2. Select the macro and click **Run**.

Editing a macro

To edit or view a macro:

1. First check if the correct editor is linked to the file extension (`.cs`).
2. Click **Tools** > **Macros...** to open the **Macros** dialog.
3. Select the macro and clik **Edit**.
4. The macro will be opened in the texteditor.

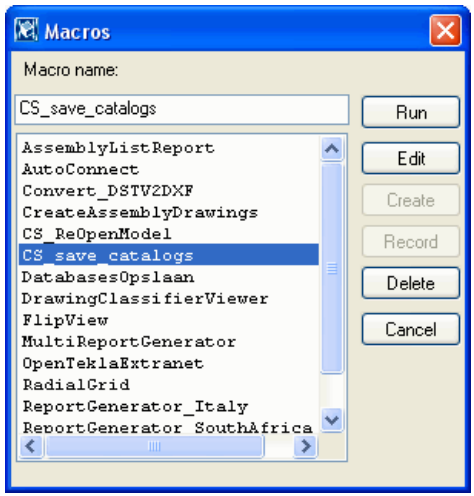
When you open the dialog **Macros**, a large nuber of macros are displayed. The ones starting with **CS_** are added by Construsoft.

CS_save_ catalogs

This macro saves all databases in the modelfolder. This can useful when you want to email a model to Construsoft Support and to be sure that all (modified) databases are present in de model:

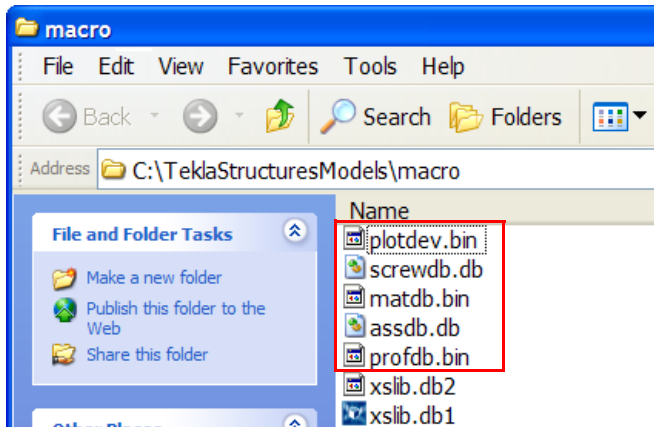
Step plan

- Go to **Tools** > **Macros**, the dialog appears:



- Select the routine **CS_save_databases** and click **<Run>**

The macro starts and saves the plotter- (plotdev.bin), the material- (matdb.bin), the bolts (screwbd.bin), the bolt assembly- (assdb.bin) and the profile database (profdb.bin) in the model folder.



When the model is being zipped, the just saved databases are also being zipped.

CS_ReOpen Model

This macro re-opens the current model.

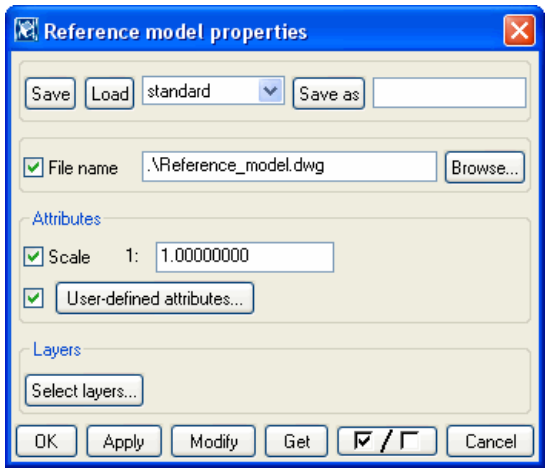
See also

For more information about macros, see the **Tekla Structures Online Help**.

Reference models

It is possible to add reference models as an instrument to position new profiles. In Tekla Structures 12.x, it was necessary to enter a scale in order to get a well scaled model.

If you inserted a file which did not have units determined, Tekla Structures assumed the model had units in meters. You had to (re-)scale the model manually:



This has been changed.

If you want to change this in Tekla Structures 12.x also:

Step plan

1. Go to folder *disc:\TeklaStructures\12.x\bat\user*
2. Select file *user.bat*
3. Right-mouse-click and select **Edit**
4. The file *user.bat* is being opened in a text editor, such as Notepad
5. Search for the lines:
   ```
   :ts_settings
   ```
6. Insert this variable
   ```
   set XS_DWG_IMPORT_IGNORE_UNITS=
   ```
7. Restart Tekla Structures
8. Now create a new model and add a reference model.

Note; this does not work for existing models!

or

1. Go to folder *disc:\TeklaStructures\12.x\bat\environment*
2. Select the file *europe_env.bat*
3. Right-mouse-click and select **Edit**

4. The file *europe_env.bat* is being opened in any text editor, e.g. Notepad
5. Search for the line:
   `set XS_DWG_IMPORT_IGNORE_UNITS=TRUE`
6. Modify the file as follows:
   `rem set XS_DWG_IMPORT_IGNORE_UNITS=TRUE`
7. Restart Tekla Structures
8. Create a new model and insert a reference model.

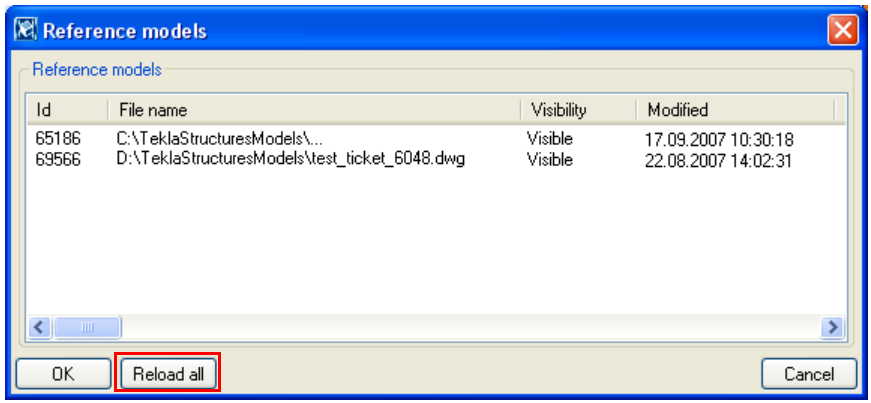
Note: this does not work for existing models!

Reload reference models

In Tekla Structures 13.1 you can reload the reference models if changes have been carried through in the original files.

To reload all reference models:

1. Click **File > Reference models...**
2. Click **Reload all**



Tekla Structures will regenerate the cache memory of all reference models.

Im- and export

Several import and export possibilities (such as Pro/Engineer, Autodesk Inventor, Archicad) are described in document **Tekla Structures.pdf**. This document is stored in folder

`disc:\TeklaStructures\version\environments\europe\pdf`



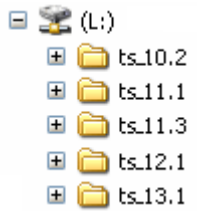
A shortcut is created and placed on the desktop automatically during installation:

TS folder on a server

You can create a "ts" folder on a server disc. In that folder you can store personal settings which can be accessed by all workstations. This way everyone uses the same settings.

We recommend to create seperate ts-folders per Tekla Structures version, because some settings are different per version, e.g. when new fields of User Defined Attributes are added.

Summary, don’t use 10.2 settings in the 13.1 version, but create new settings in 13.1 and put them in a new ts-folder:



To check which files have been changed, see chapter ’Administrator’s Guide’ in the Release Notes 13.1.

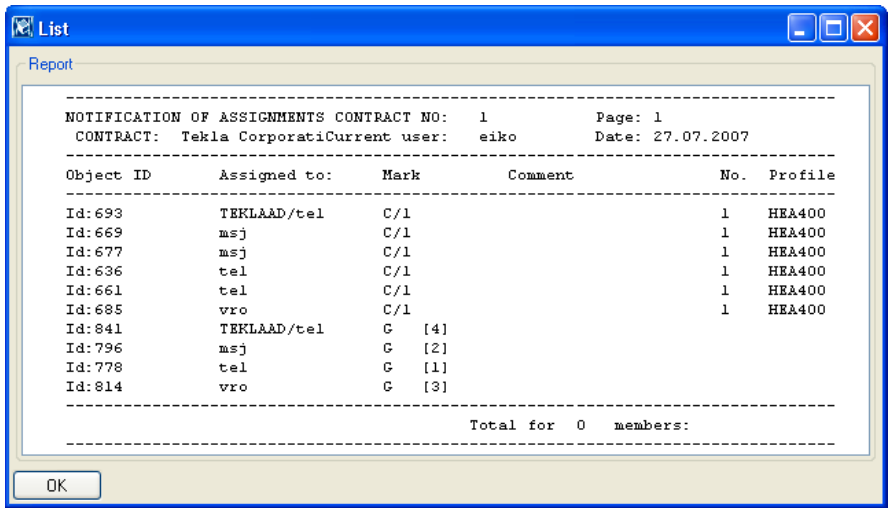
Modeling

Notification report and displaying project messages

For users who work in the same model, a notification report can be created and automatically be opened when the model is opened.

Displaying a notification report when opening a model

When opening a model, Tekla Structures shows a notification report containing messages about the objects in the model (drawings, parts and assemblies).



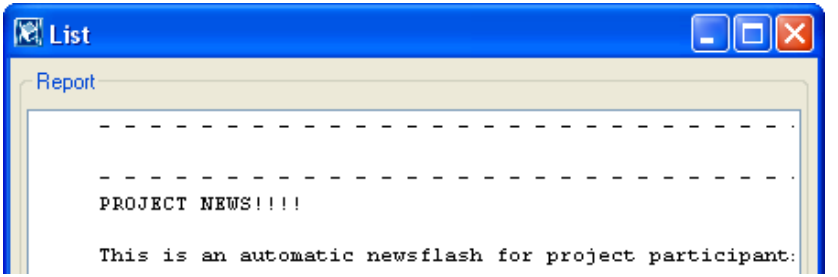
If you want the notification report to be displayed when opening a model, set variable `XS_SHOW_NOTIFICATION_REPORT` to `FALSE` (**Tools > Advanced options > Modeling properties)**.

Displaying project messages

The same functionality can be used to display messages to everyone who opens a model. To do this:

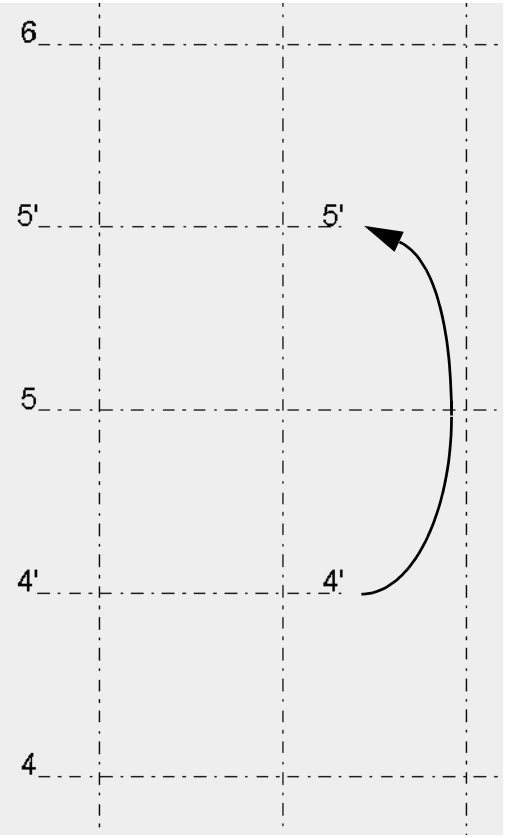
1. Go to **Tools > Template Editor...** to start the Template Editor.
2. Go to **File > Open...** and open report notification_report.rpt.
3. Modify the Header text.
4. Save the template in the model folder (do not change the report name).
5. Re-open the model.
6. Go to **Tools > Advanced options > Model properties**.
7. Check if variabele `XS_SHOW_NOTIFICATION_REPORT` is set to `TRUE`.

If the model is re-opened, a modified message appears:



Copying loose gridlines

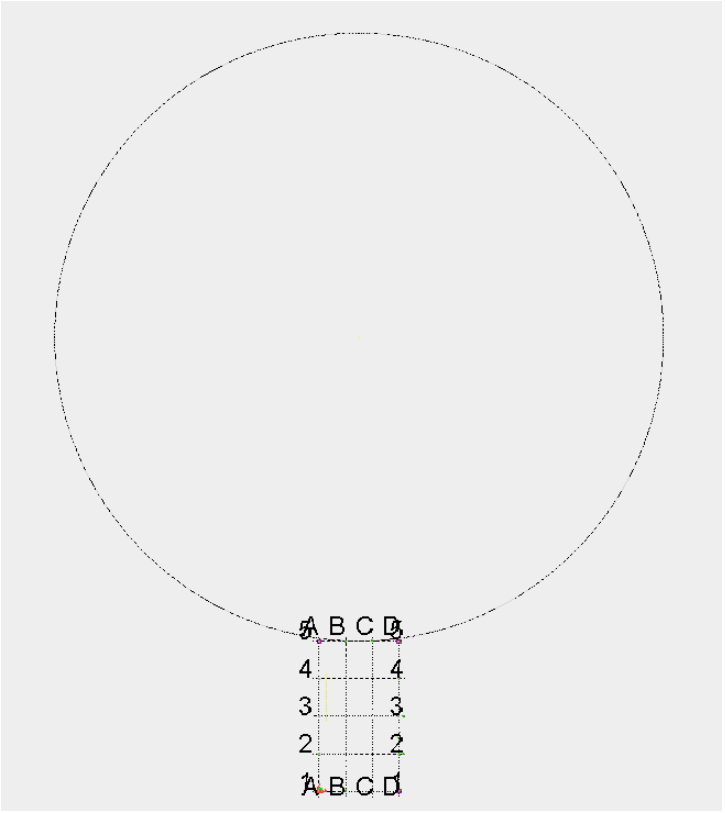
Loose gridlines, created using command **Grid Line**, can now be copied. Normal grids could already be copied for a longer time.



Speed improvements for large construction circles

Models containing large construction circles resulted often in a speed loss. E.g. selecting a 700 meter radius construction circle took about 45 seconds in Tekla Structures 12.0.

In Tekla Structures 13.1, this action takes about 7 times less time, so less than 10 seconds.



Speed improvement when opening a large component catalog

Opening a large component catalog won't result in speed loss. In previous versions, several Tekla Structures processes became slower.

Editing colors of dimensions, dimension texts and part marks in the Model editor

In Tekla Structures 13.1, you can change the color of part marks, dimensions en dimensions texts in rendered model views.

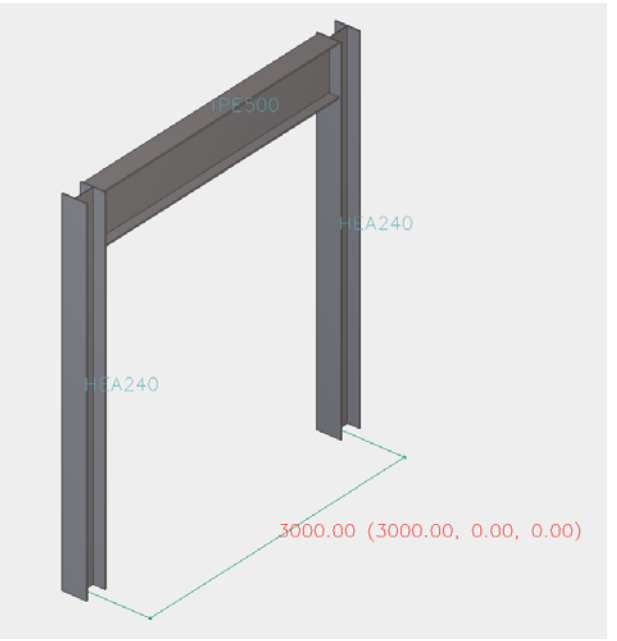
How to

To change a color:

1. Go to **Tools > Advanced options > Model view**.
2. Change the variable as below. Enter a RGB (Red Green Blue) value. The scale is from 0 to 1. For example, to change the color to red, enter "1 0 0".

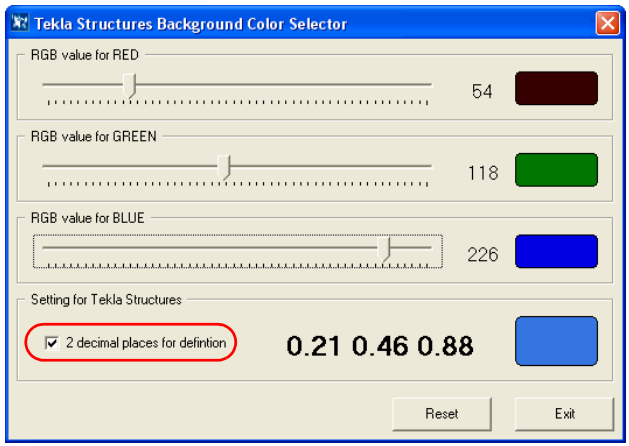
Variable	Use for
XS_VIEW_DIM_LINE_COLOR	Changing the dimension color
XS_VIEW_DIM_TEXT_COLOR	Changing the dimension text color
XS_VIEW_PART_LABEL_COLOR	Changing the part mark color

Example



Finding the right RGB values

Each color has an accompanying RGB value. Tekla has developed a tool to find the RGB values. The tool is named **TS_Background_Color.exe** and can be downloaded from the Construsoft Xtranet website.



RGB values can be entered with 2 decimals.

Background color in model views

In Tekla Structures 13.1, the default background color has been changed. In previous versions, the default was a gradient background color (blue>white), in Tekla Structures 13.1 there will be an **even blue** color.This background color is more calm for your eyes, and an even background color uses less memory.

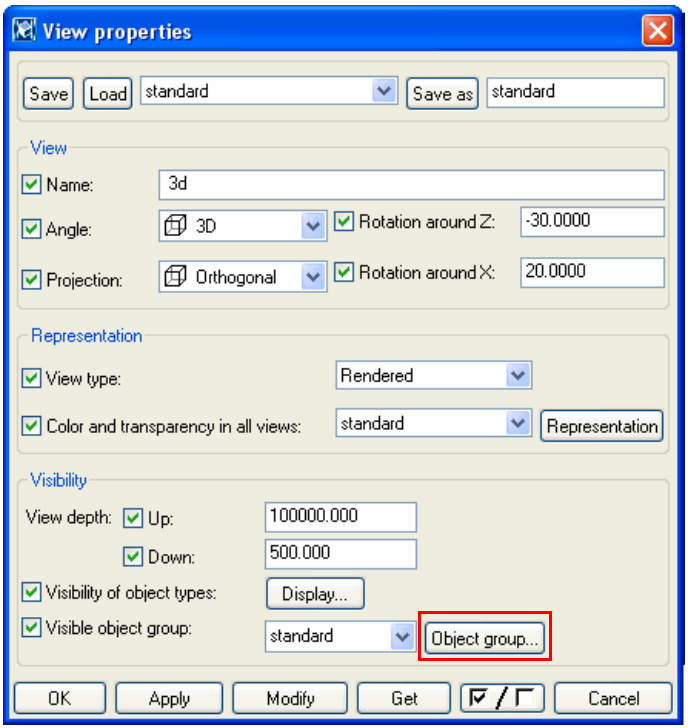
The background color can still be changed of course, if you want the 'old' gradient background, just remove the RGB values for variables XS_BACKGROUND_COLOR1 to XS_BACKGROUND_COLOR4. You can also use one of following colors:

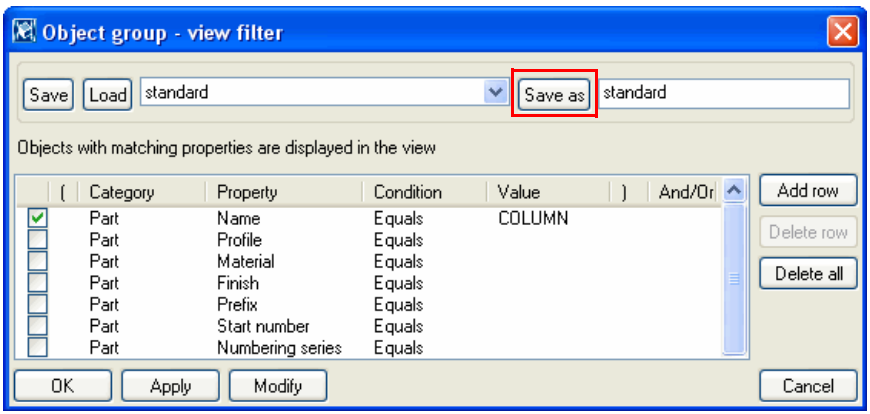
Color	RGB	Color	RGB
Black	0 0 0	Red	1 0 0
White	1 1 1	Yellow	1 1 0
Blue	0.21 0.46 0.88	Green	0 1 0
Dark blue	0 0 1	Grey	0.75 0.75 0.75
	0.9 0.9 0.9 0.9 0.9 0.9 0.2 0.3 0.5 0.2 0.3 0.5		

Re-open a view to activate the changes. The modified variables are saved in file **options_*username*.ini** (in folder disc:\TeklaStructures\version\nt\bin.

View filters keep there setting

In Tekla Structures 12.1, if you used the model filter, it was necessary to save the filters setting in order keep the filter active after re-opening the model.





This is not necessary anymore; the setting is now stored correctly. (before, if you re-opened the dialog, the the ’standard’-filter was displayed instead and the initial filter data was lost).

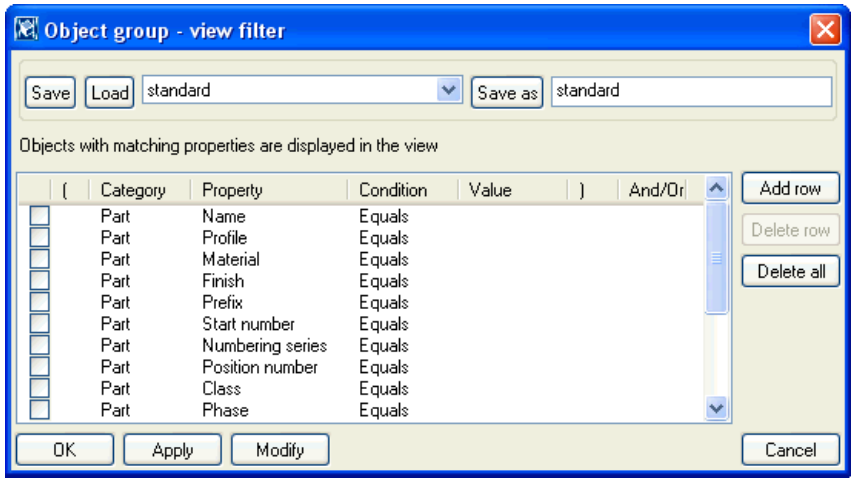
Filter remains active and displayed correctly, also after a restart.

Filtering for bolt diameter, bout length or hole diameter



The dialog **Object group- select filter** is used to define which objects can be selected. Defining Object groups is based on properties, filter strings and categories.

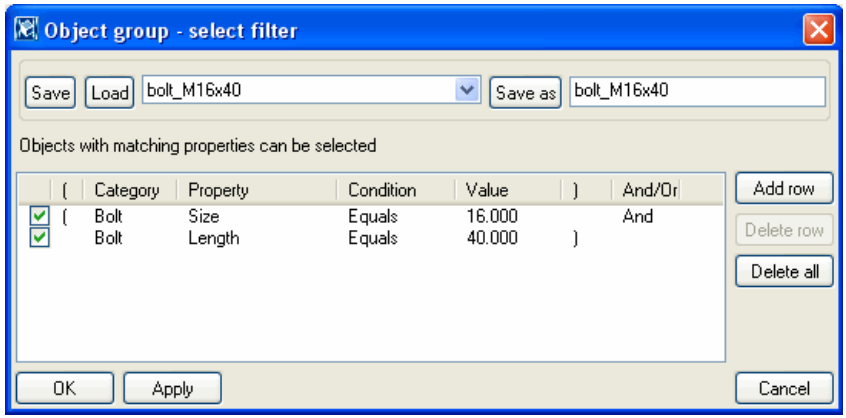
Go to **Setup> Select filter...** or click the icon; the following dialog shall appear:



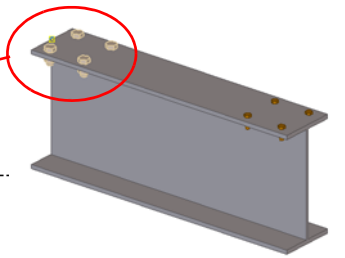
You can use a default setting (provided by Construsoft), or you can create your own settings. Some examples are listed below:

Bolt diameter + bolt length

To select bolts M16 x 40 only:



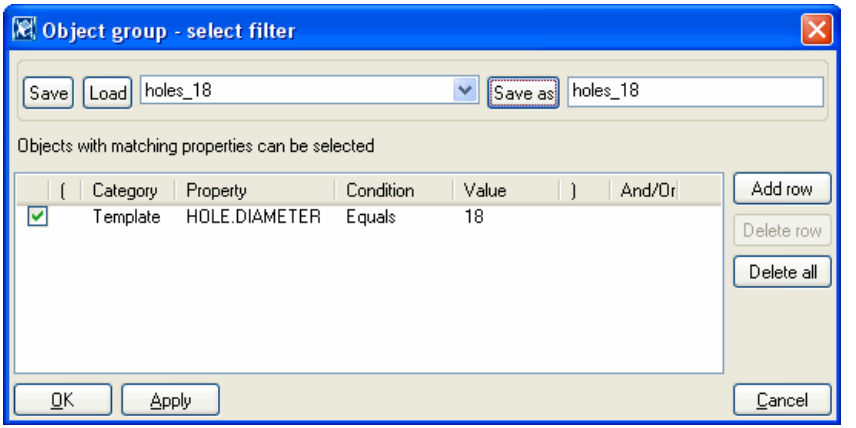
Now, if you create a bolt report, only bolts are listed which correspond to the filter settings.



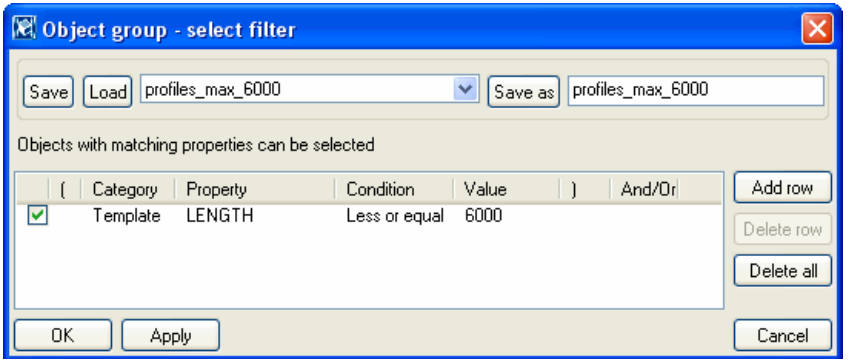

```
-------------------------------------------------
TEKLA STRUCTURES BOLT LIST
Model: testl

-------------------------------------------------------------------------------
   Name                    Type              Length (mm)              Number
-------------------------------------------------------------------------------
   BOUT-M16*40             931-8.8             40                       12
===============================================================================
                                                              Total:    12
===============================================================================
```

Holes Ø18

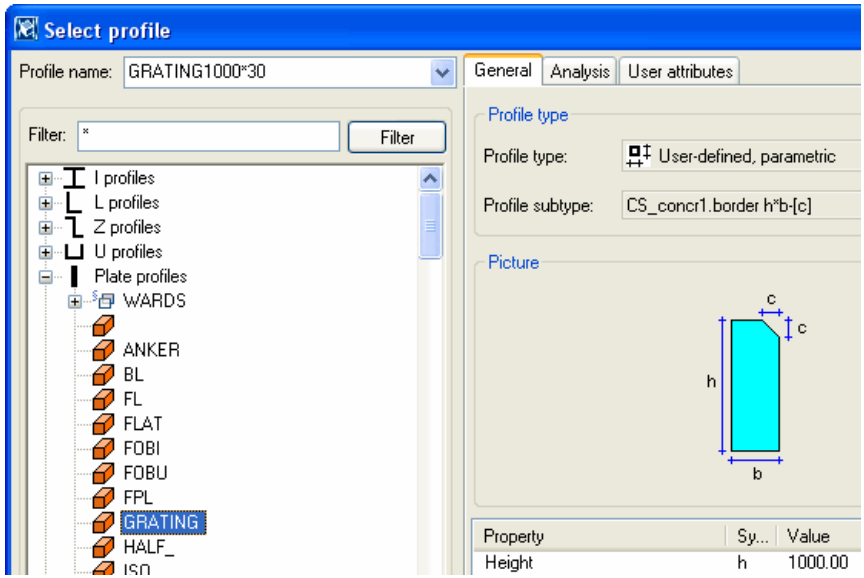


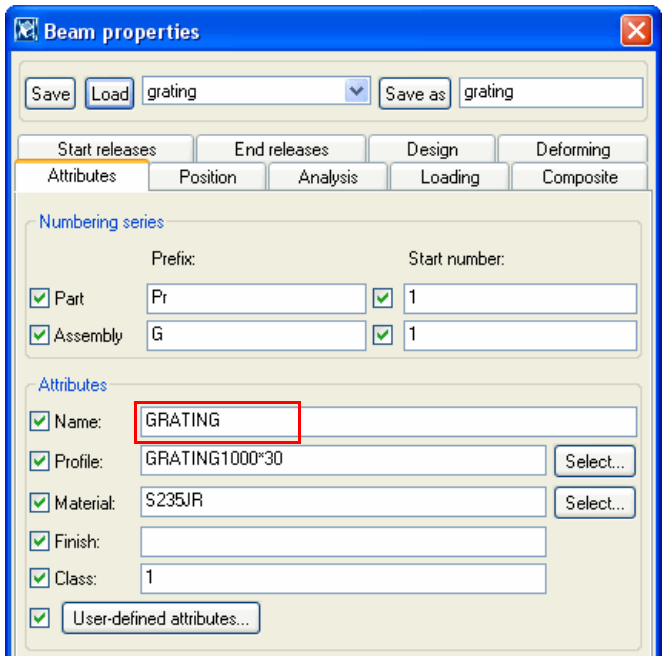
Profiles max. 6 meters



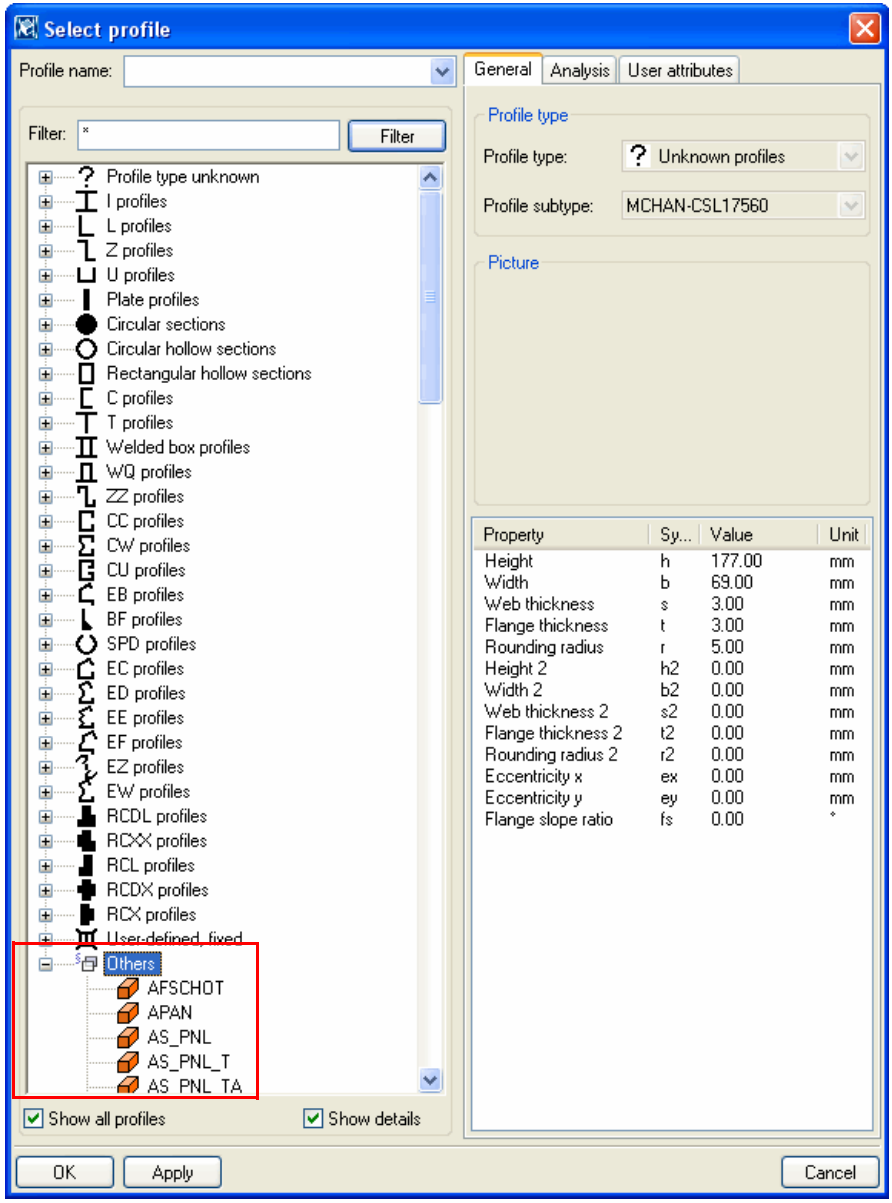
Modeling gratings

For modeling gratings as GRATING, the default value for the chamfer has been set to **2** mm, (**5 mm** in previous versions). If you omit this value, then also this default value of 2 mm will be used.





Note: The default value may be omitted for all parametric profiles in the **Others** category.

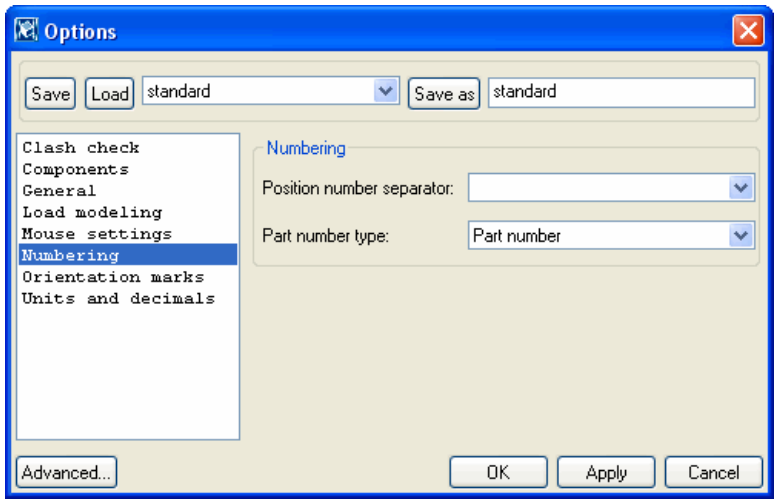


New dialog Options

Due to interface improvements, several settings from separate dialogs are moved and grouped in dialog **Options**. In previous versions, these settings often had their own dialogs in the **Tools** menu.

Collecting these settings in one dialog makes adjusting the settings a lot easier than before.

To open the Options dialog, go to **Tools** > **Options...**

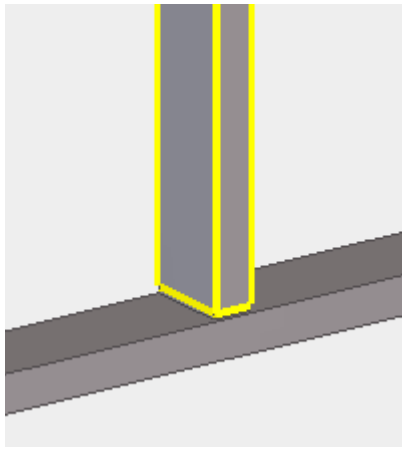


See also

For more information, see the **Tekla Structures Online Help** and **What's New** (**Help** > **Learning Center** > **What's New**).

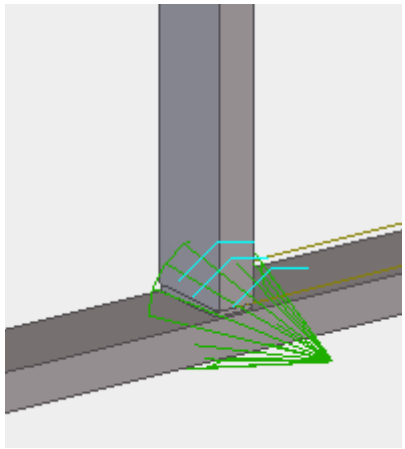
Displaying all parts objects

In some cases it may be useful to see all the parts objects, for example to examine if a part is welded in a wrong way:



Step plan

- Select the part
- Click <**Delete**>
- Press <**Ctrl**> + **Z** or click the <**Undo**> icon

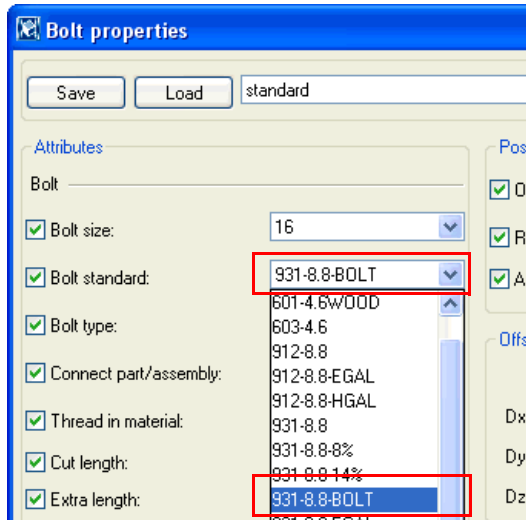


The part is now displayed, including all accompanying objects, such as components, welds, fittings ect.

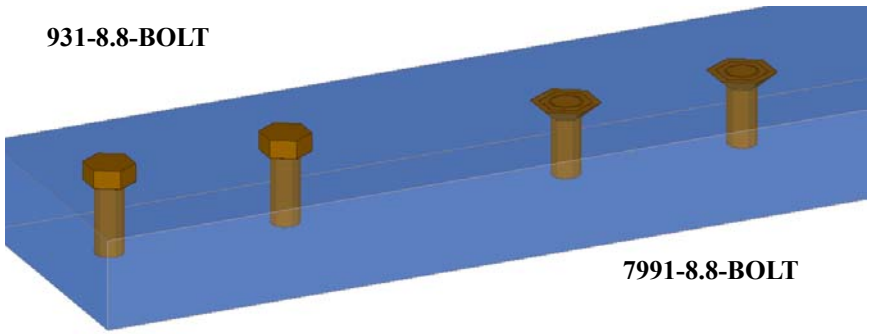
Modeling bolts

In Tekla Structures 13.1, a new countersunk bolt standard is added: 7991-8.8BOLT. This standard creates a bolt without a nut (the nut has a height of 0.1 mm and will therfore not be listed on a nutlist).

As for other bolts, you can determine the bolt length by entering a negative value for **Cut length** (e.g. **-60**).



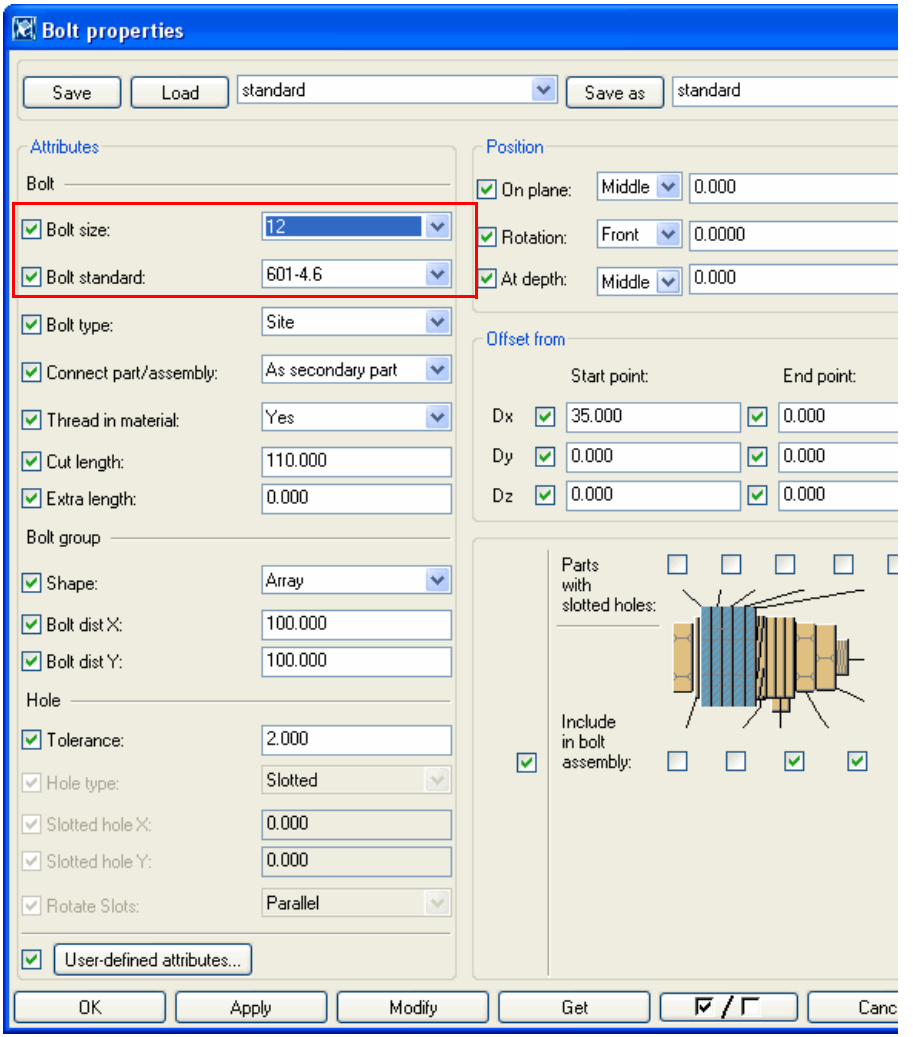
931-8.8-BOLT



7991-8.8-BOLT

Changing Bolt Standard: diameter remains

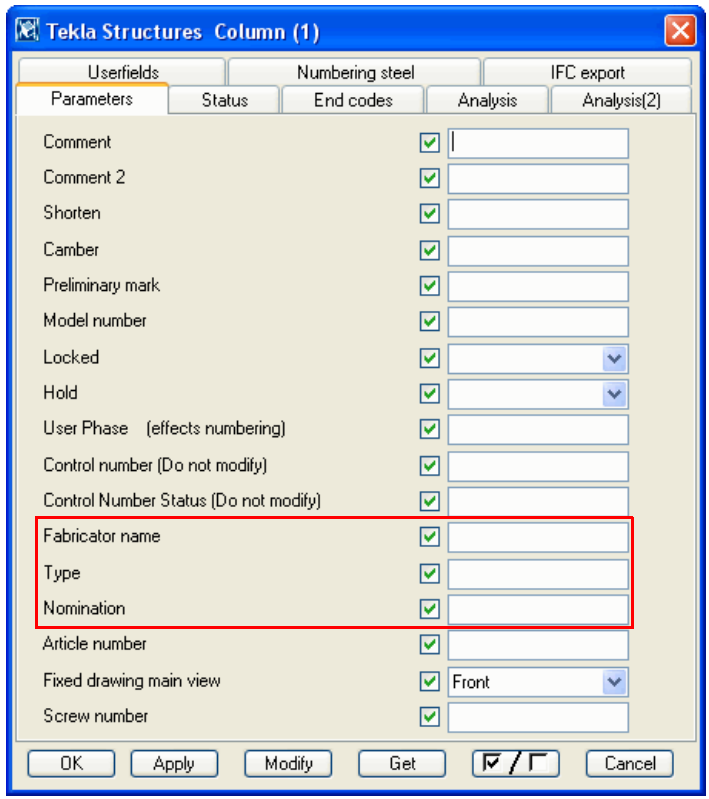
In Tekla Structures 12.1, when changing the bolt standard, the diameter listbox jumped to the smallest diameter available. This has been fixed. When changing bolt standard, the diameter stays the same (if diameter is available of course).



If the same diameter is unavailable, then Tekla picks the first smaller diameter from the listbox.

User-defined attributes: Fabricator name, Type and Nomination

The user-defined attributes **Fabricator name, Type** and **Nomination** do not affect numbering anymore:



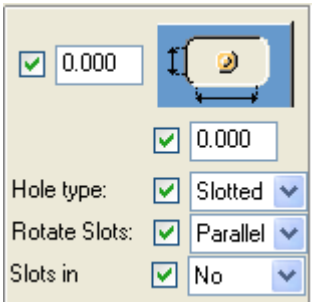
Variable XS_ASSOCIATIVE_REBARS

The variable XS_ASSOCIATIVE_REBARS is now set to FALSE, instead of TRUE in previous versions. This should give better results.

Components

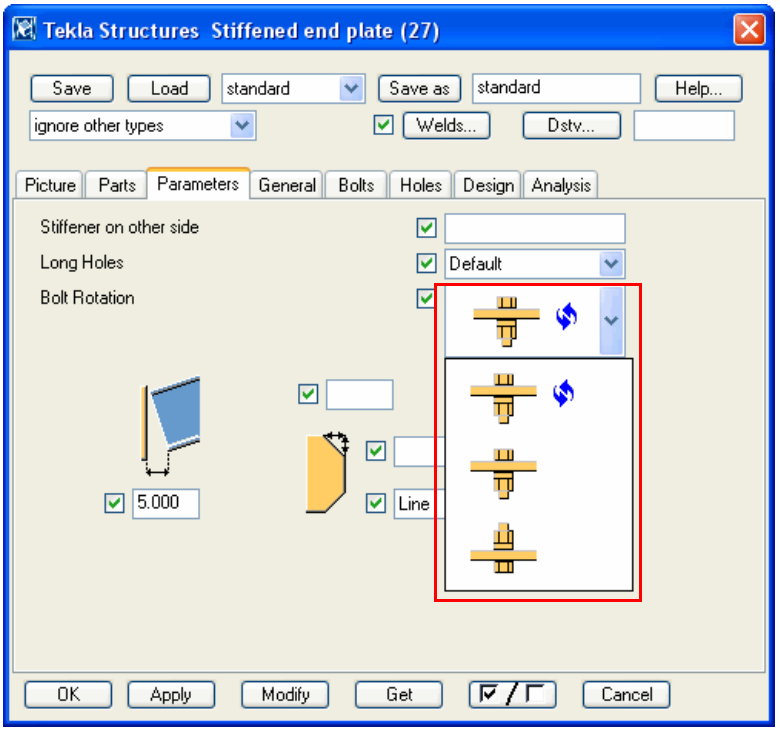
Bolted gusset (11) and Tube gusset (20)

Bolt group properties can now be set for 2nd and 3rd beam individually using tabs **Brace Bolts 2** and **Brace Bolts 3**.



Stiffened end plate (27)

An option **Bolt Rotation** is added to component **Stiffened end plate (27)**:

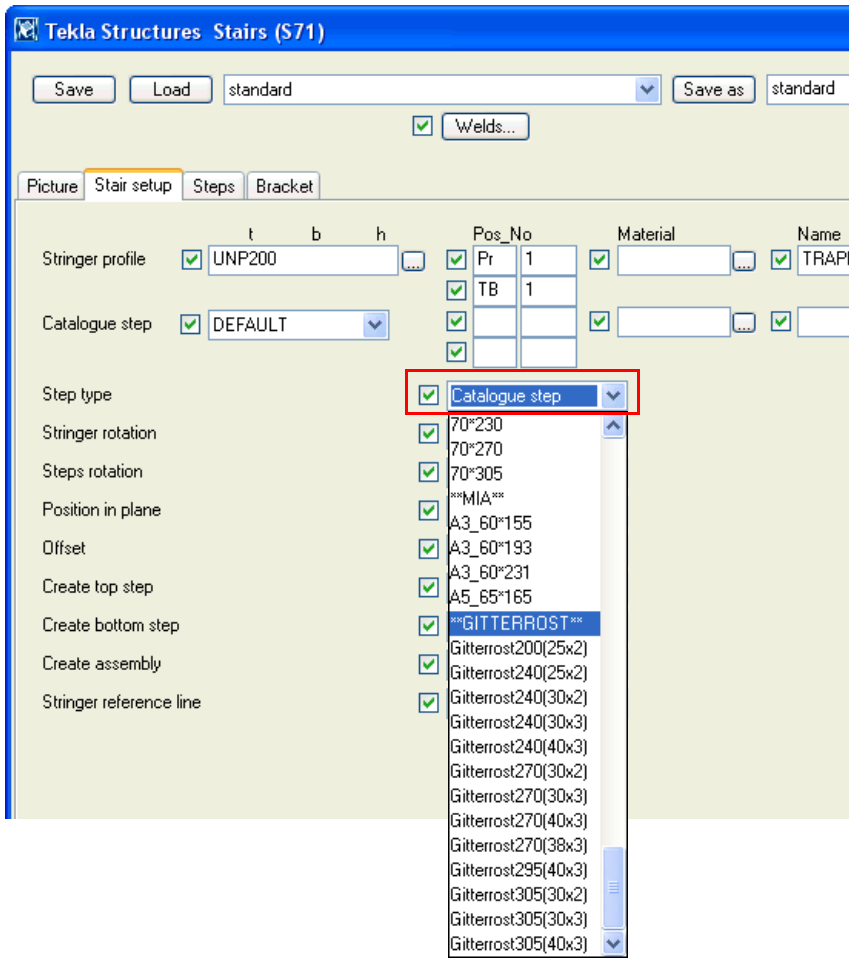


End plate (29)

The default settings have been removed from component **End plate (29)**. We recommend to use component **End plate (144)**. Component **End plate (144)** includes a lot more options.

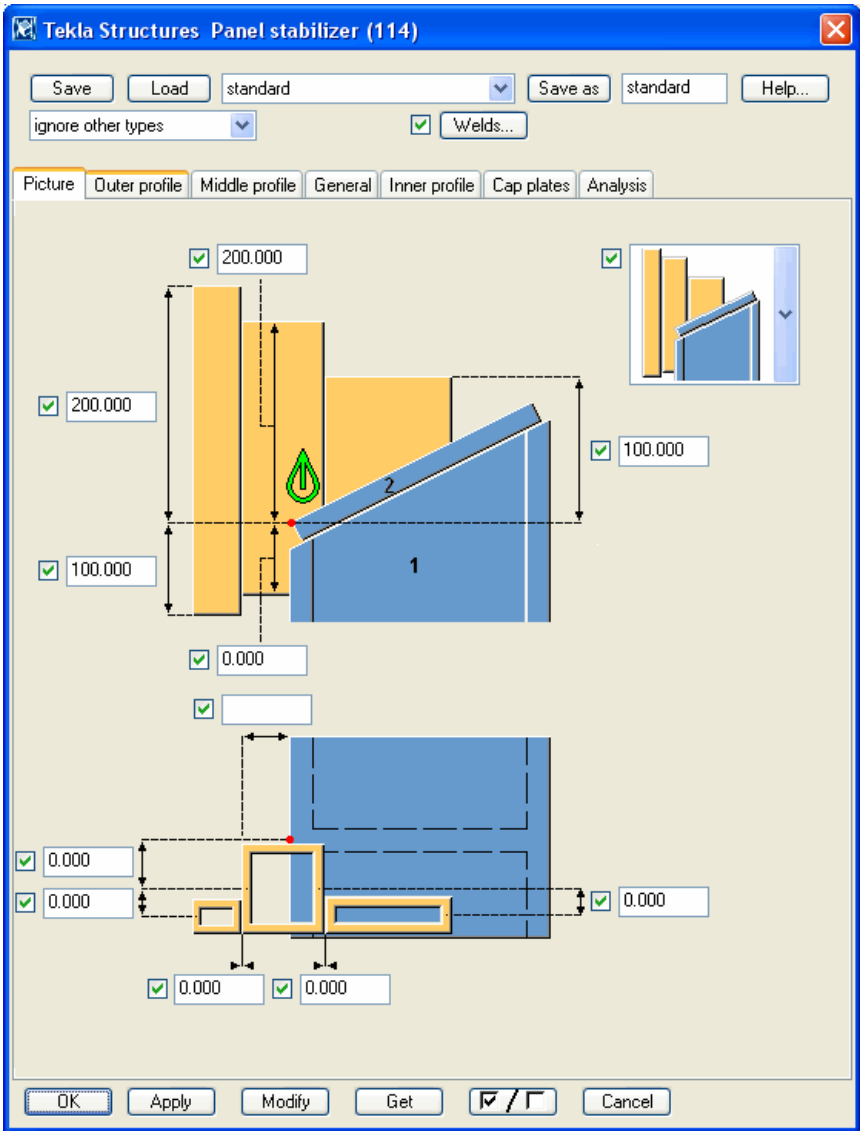
Stairs (S71) and Stairs (S82)

The Gitterrost step profiles are added to components **Stairs (S71)** and **Stairs (S82)**. For component **Stairs (S71)**, you have to set the **Step setup picklist** to **Catalogue step.** Then the step list will be available:



Panel stabilizer (114)

Component **Panel stabilizer (114)** has been modified:



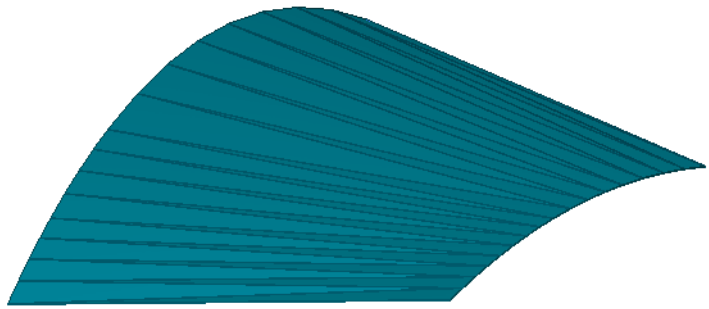
The red referencepoint in the picture is now used to create the component. Some settings are added. The default setting has been changed and a setting named **angle-steel** is added.

Unfold rules surface (21)

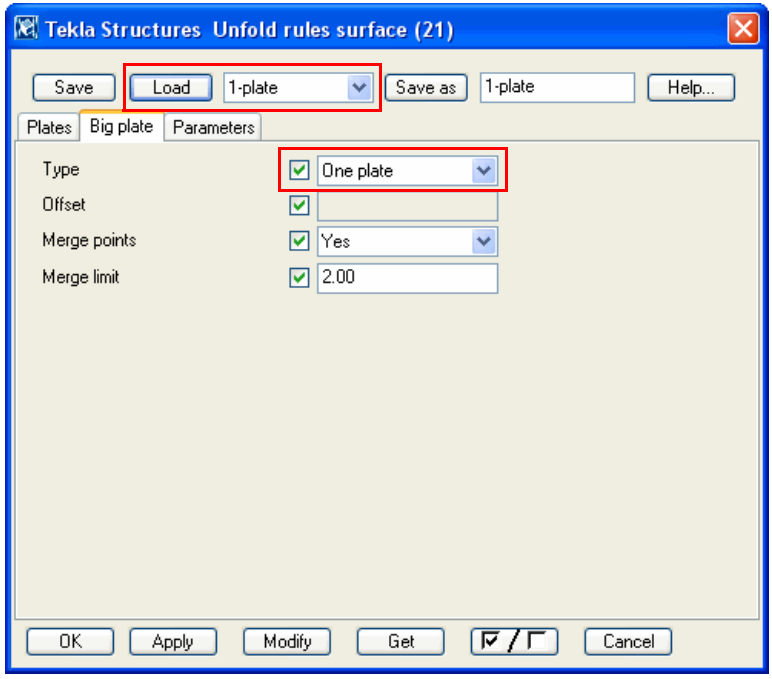
Component **Unfold rules surface (21)** is now able to create one single plate of the unfolded plates (instead of several loose plates).

Step plan

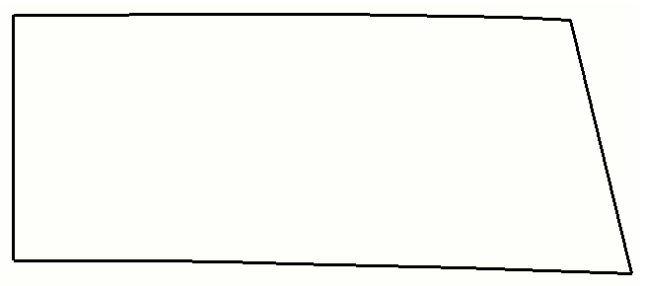
Using component **Triangles generation (19)** you can describe a contour by picking contour points:



Using component **Unfold rules surface (21)**, an unfolded pattern can be created. All small plate pieces can be joined together by chosing the "1-plate" option:

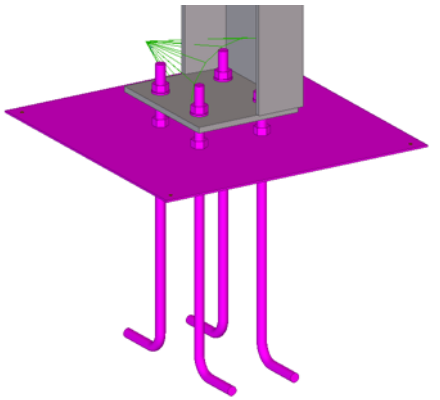


If you create a single part drawing of the flat pattern, use the **no-dims** setting to avoid unnecessary dimensions on the drawing:



Baseplate (1029)

Component **Baseplate (1029)** is upgraded. Castplate holes now follow the anchors correctly.



Fitting plates in components

Fitting plates are now available in following components:

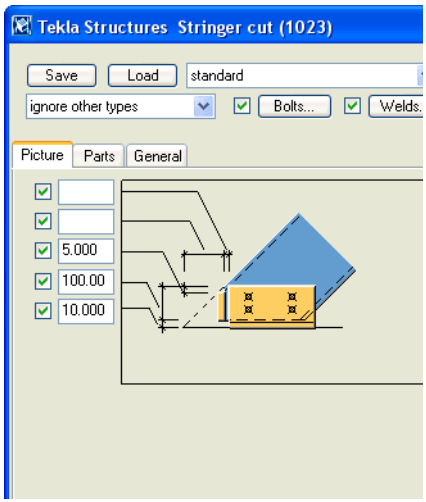


- **Double plate (27)**
- **Endplate partial stiffener (65)**
- **Baseplate (1004)**
- **Stiffened baseplate (1014)**
- **Baseplate (1047)**

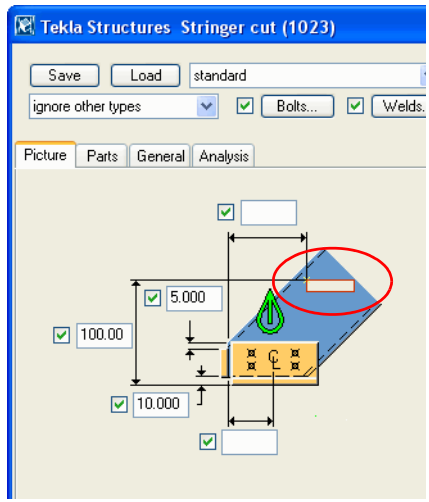
Stringer details

The Stringer Cut details have been updated. In Tekla Structures 13.1, the dialogs are better understandable. Use the point, shown in the picture, for positioning the components:

Stringer cut (1023)

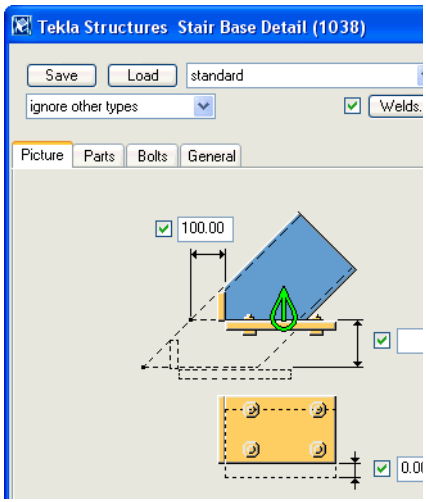


Old dialog

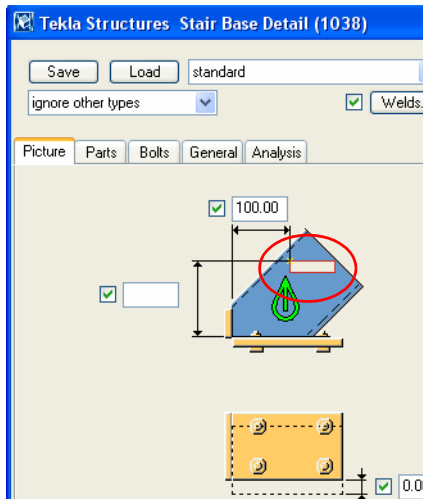


New dialog

Stairs detail (1038)

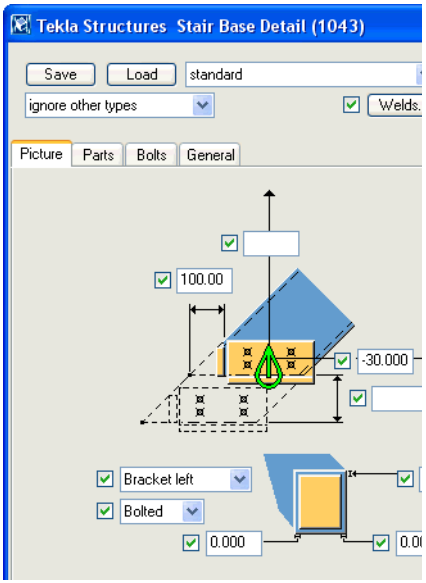


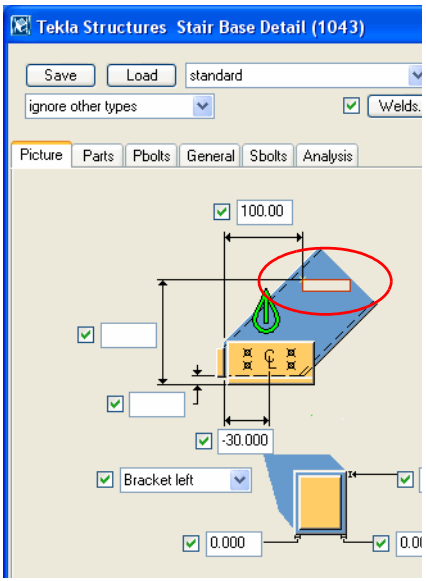
Old dialog



New dialog

Stair Base detail (1043)





Old dialog

New dialog

Improvements in functionality

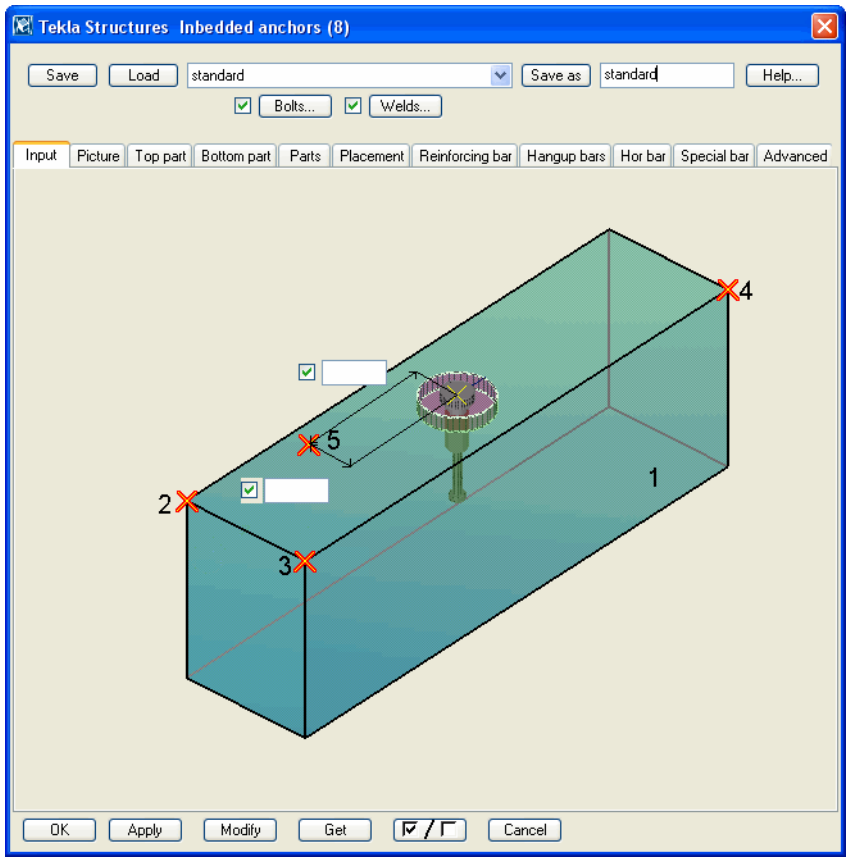
The improvements in the dialog results in better stair details.

The selected point determines the side where the detail is being placed. If a point on the right side is picked, the detail is created on the main parts leftside.



Components Imbeds (8) and (1008)

The available settings of both Imbed components (8) and (1008) are indentical now.



All these settings are stored in the *system* folder, and are available in all modules, e.g. `STD` and `SDE`. The reason is that anchors have to be modelled more and more in steel detailing.

Drawings

Improvements in dialog Drawinglist

In Tekla Structures 13.1, the Drawinglist has better functionality and has better performance (e.g. faster filtering).

Example

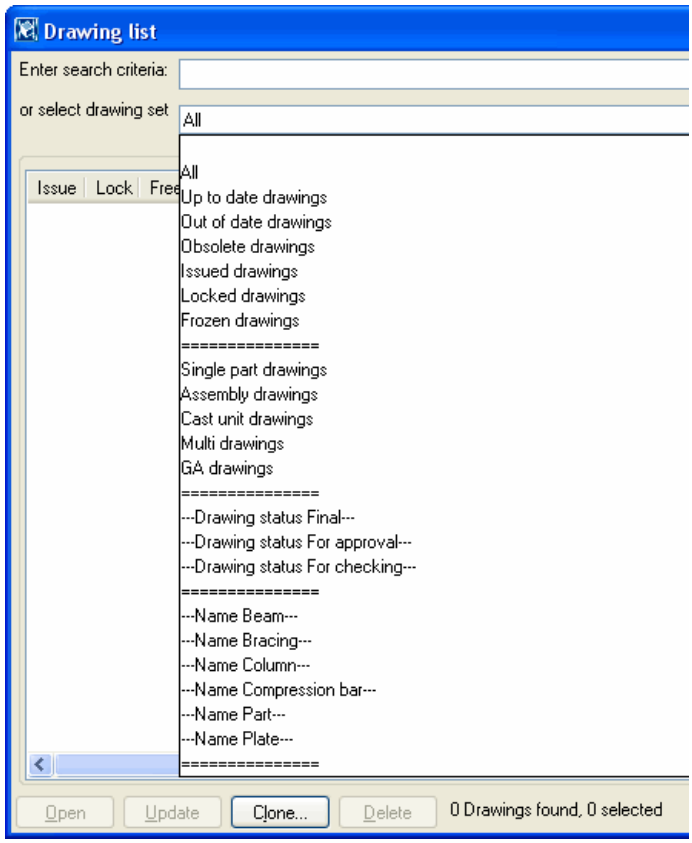
A model consists of 150000 steel parts, 15000 drawings and a 50 Mb database:

Action	Version	Time:
Opening Drawinglist	TS 11.2	2 min. en 30 sec.
	TS 13.1	9 sec.
Opening largest G-drawing	TS 11.2	10 min.
	TS 13.1	1 min. en 30 sec.

Drawinglist filtering is about twice as fast as before. Also some other actions , such as selecting drawings by parts is much faster than before.

Selecting drawings using filter settings

Pre-defined filter settings can be used now for displaying and selecting drawings in the drawing list. To do this, just select a picklist option.



For example, for filtering GA drawings only, select the "GA drawings" from the drawing set.

Drawing status

You can use the new inputfield **Drawing status** in the user defined attributes in the drawing.

For this, variabele **DRAWING.USERDEFINED.DR_STATUS** is used.

You can use this variable to idicate which drawings, for example, have been send to an architect: you can enter any random text.

The status information is displayed in the column **Drawing status** in the drawing list, you can select on this.

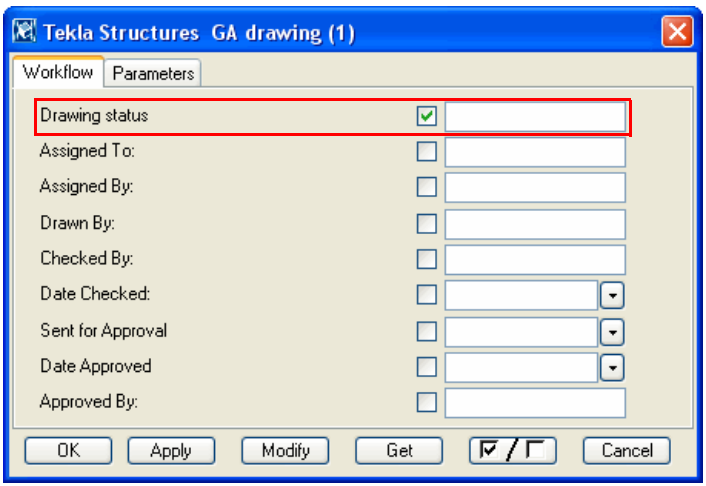
You need to add selectfilters to the drawing list by yourself, by default the selectfilters `Final`, `For checking` en `For approval` are added by Construsoft to the drawing list to display and select drawings.

See also

For more information about adding selectfilters, see **Searching and re-using search results** (pag. 64).

To assign a status to one or more drawings, select one or more drawing(s), i.e. several GA drawings and open the **User defined attributes**.

For this, click on the right mouse or use the shortcut <**Alt**> + **U**, the following dialog box appears:



Make use of the button  to switch off all checkboxes in one go, now check the field **Drawing status**.

Enter a status in the field **Drawing status** in the **User defined attribute** for the selected drawing(s) , for example `For checking`.

The entered status information is **not** case sensitive but per selectfilter a criterium is used:

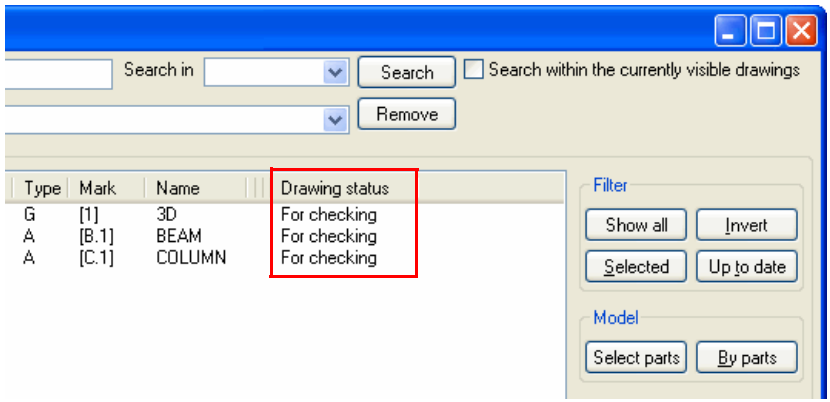
For checking So the selectfilter `For checking` must at least contain the text **check**. This means that you can also enter **forchecking** but not **For Ch.**.

Final The selectfilter `Final` must at least contain the text **final**.

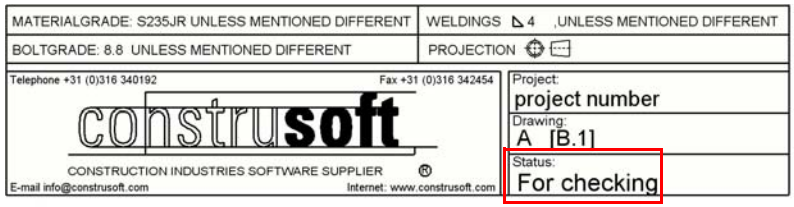
For approval The selectfilter `For approval` must at least contain the text **approval**.

Click <**Modify**> and <**OK**>.

The status information will be displayed in the column **Drawing status** in the drawing list, you can select on this by using the selectfilter `For checking`, for example:

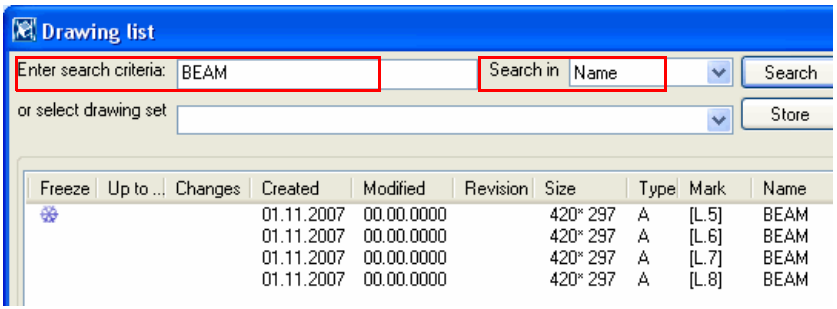


Template_A In the drawing, the drawing status will be displayed in **template_A** automatically:



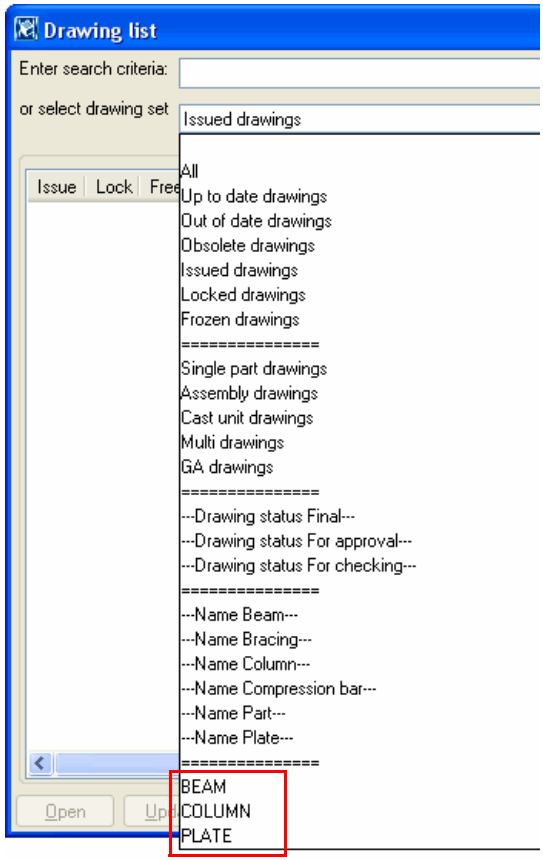
Searching and re-using search results

For faster finding of drawings, a Search option is added to the drawing list. You can enter search criteria in field **Enter search criteria**. You can restrict searching by using a column filter in field **Search in**. To search in a current list of drawings, select option **Search within the currently visible drawings**.



It is possible to save search results. To do this, press <**Store**> and enter a name in the pop up dialog "Store Search result". The search results are saved in folder DrawingListSearches, in the model folder. The search resulst can be moved to make them available for all models.

New stored search results are listed at the bottom of the picklist.





Only search results are stored.

Therefore, the <**Store**> button only appears after a search task

Selecting all drawings

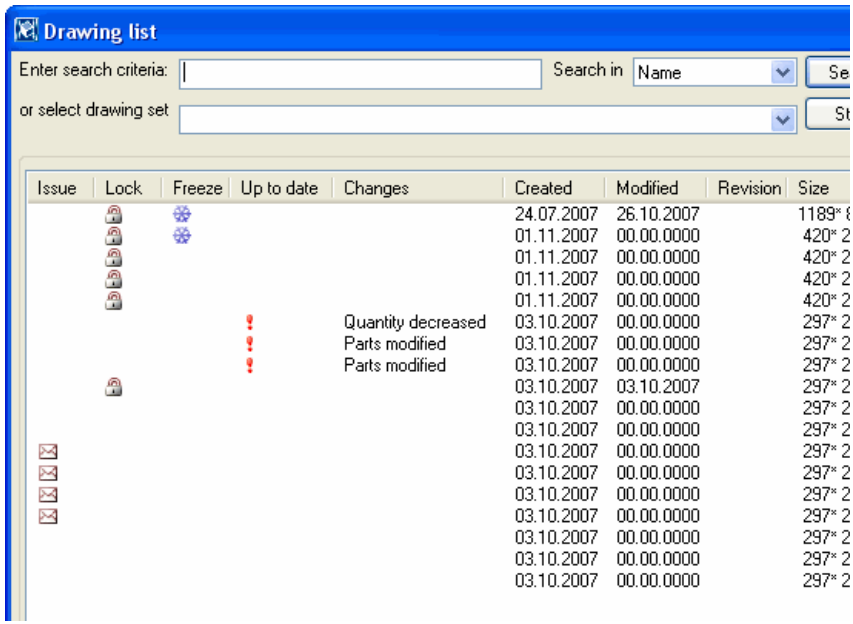
In Tekla Structures 13.1, you can select all displayed drawings at once. This can be done by pressing <**Ctrl**> + **A**> (dialog drawing list needs to be selected), or right-mouse-click > **Select all**.

New grafical symbols and status text

Tekla Structures 13.1 has new grafical symbols for status information. Added text in column **Changes** describes the drawings status. See table below for more information.

New symbol	Statusmark in old versions	Column	Status text
	**P**	**Up to date**	Parts are modified
	**N**	**Up to date**	Number increased Number decreased
	**n**	**Up to date**	Original parts are deleted
	**X**	**Up to date**	All relevant parts are removed from themodel
	**D**	**Up to date**	Linked drawing modified
	*	**Up to date**	Original parts are deleted, other identical parts do exist in the model. Drawing up to date
	*	**Up to date**	Cloned
	**I**	**Issue**	Drawing is issued. e.g. for drawings which are handed out to the workshop
	**M**	**Issue**	Issued drawings are modified
	**L**	**Lock**	The drawing is locked
	**F**	**Freeze**	The drawing is frozen

Example of the new symbols and texts:



Quicker UDA access

Drawings User Defined Attributes can be accessed quicker. To do this, right-mouse-click a drawing and select **User Defined Attributes**. Default shortkey is <**Alt**> + **U**.

Printing change symbols

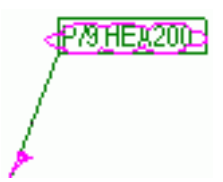
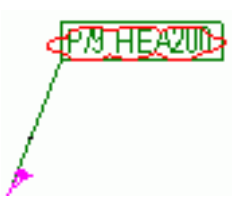
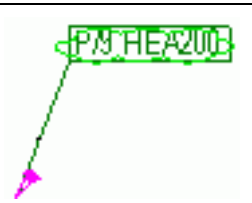
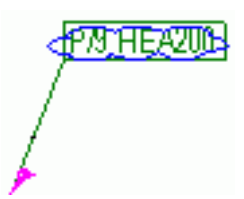
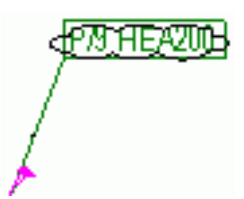
The change symbols on drawings are now displayed in red color. Consequence is that the symbols are printed too.

To change the change symbol color, another color needs to be entered in variable `XS_DRAWING_CHANGE_HIGHLIGHT_COLOR` (**Tools** > **Advanced options** > **Drawing Properties**. The options are `RED, BLACK, GREEN, BLUE and GREY`).



If you use the default color (magenta), the change symbols are displayed on screen, but they will **not** be printed.

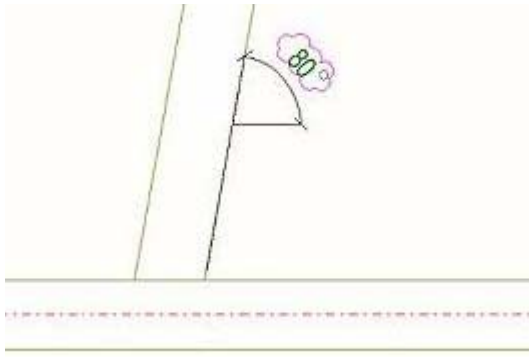
See the examples below:

Set variabele to	Color in screen	Printed color
190 (default)		not printed
RED		red
GREEN		green
BLUE		blue
BLACK		black
GREY		grey

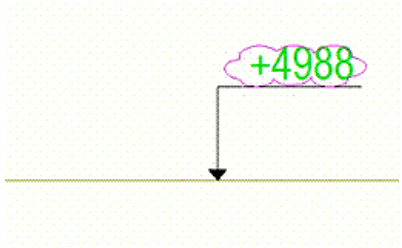
Angle dimensioning, level marks and associative notes

In Tekla Structures 13.1, modified angle dimensions, level marks and associative notes get change symbols after a model change.

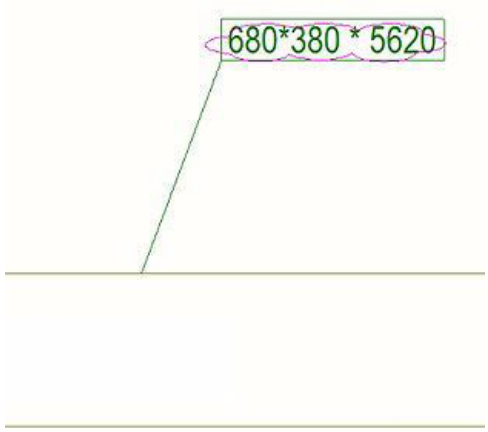
Angle dimensioning



Level marks



Associative notes

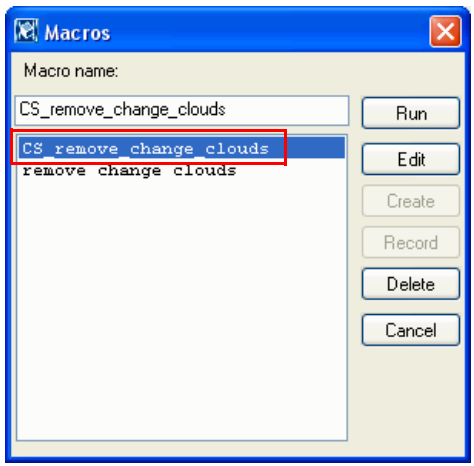


New macro Remove Change Clouds added

In the Drawing Editor, a new macro is added: **Remove Change Clouds**. This macro removes all change symbols (clouds) on the drawing at once, for both dimensions and marks.

Step plan

- Open a modified drawing and go to **Tools** > **Macros...**, the following dialog shall appear:



- Select the macro **Remove Change Clouds** and click **<Run>**

All change symbols will now be removed from the drawing.

Displaying Tekla Structures version on drawings

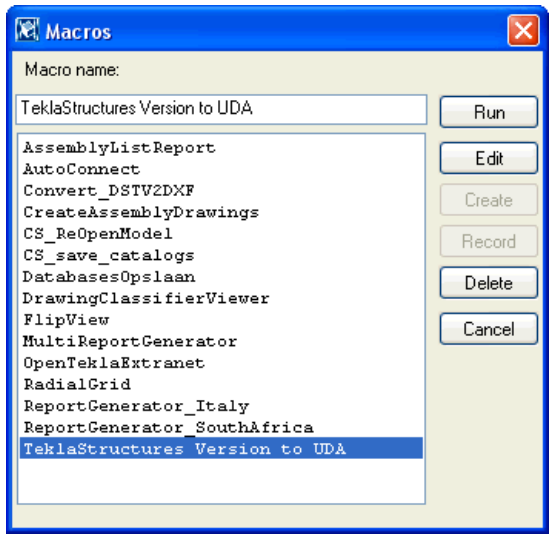
A new macro has been added: **TeklaStructures Version to UDA**. This macro stores the TS version in a User Defined Attributes field in the Project properties.

If a Value Field is added in a template which is linked to this UDA attribute, then the TS version will always be printed on drawings.

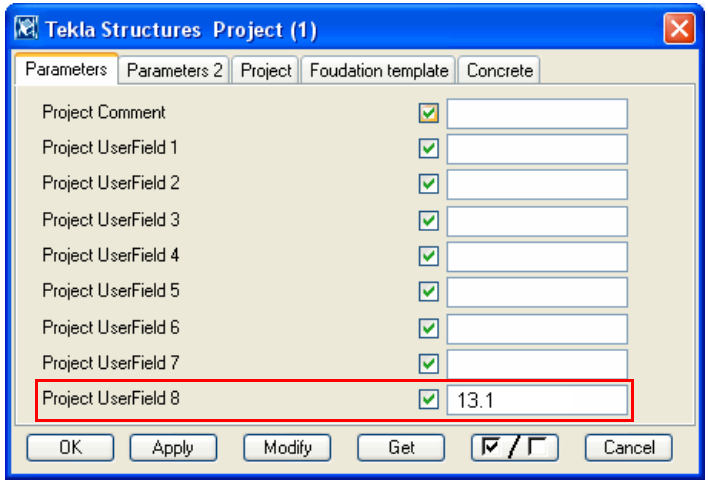
Step plan

- Download file **TeklaStructures Version to UDA.zip** from Construsoft Xtranet
- Unzip the file to folder disc:\TeklaStructures\version\environments\country-independent\macros\modeling.
- Restart Tekla Structures

- Go to **Tools** > **Macros**

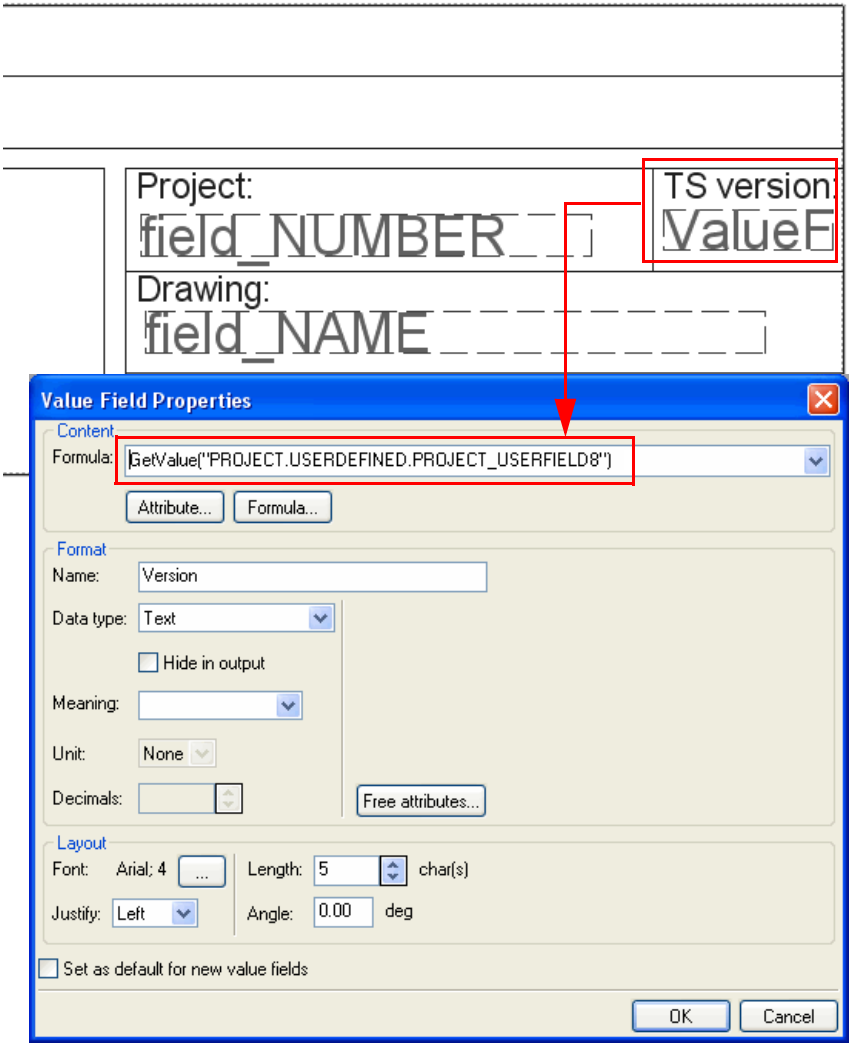


- Select the macro **TeklaStructures Version to UDA** and click **<Run>**
- Go to **Properties** > **Project properties** and click **User Defined Attributes**:



The TS version is entered in the field Project Userfield 8.

- Now, go to **Tools** > **Templates**, open a template and add the attribute as follows:



- Save the template and open a drawing; you will see the version number is displayed:



See also For more information about macros, see the **Tekla Structures Online Help**.

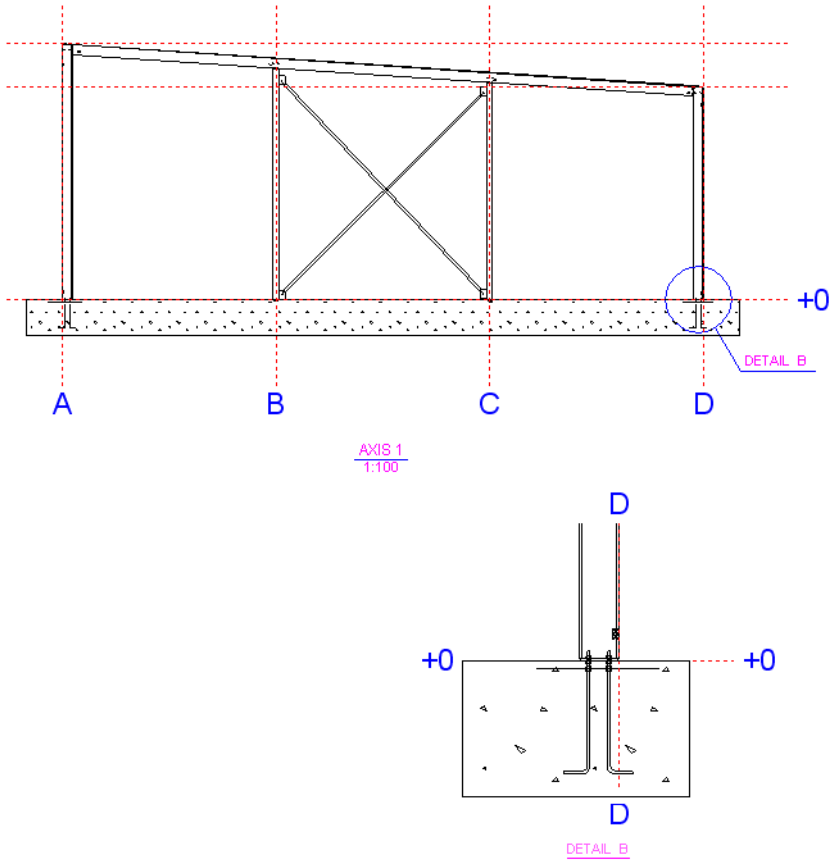
Scaling objects in drawing views

Variabele XS_SCALE_COPIED_OR_MOVED_OBJECTS_IN_DRAWINGS can be used to define if objects should be scaled or not if they are copied or moved to another view which has a different scale.

To scale objects according to the view scale, set variable to TRUE. To leave autoscaling switched off, enter FALSE.

Example

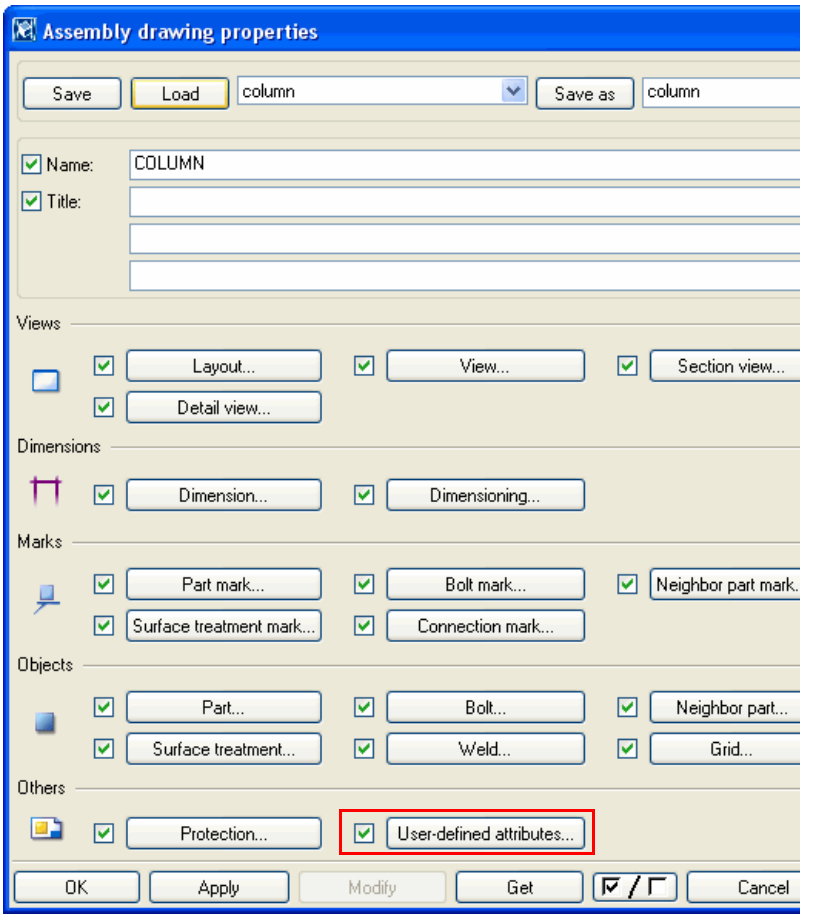
A grafical object containing a hatch is being copied from an axis view towards a detail view; the objects size will automatically be re-scaled:



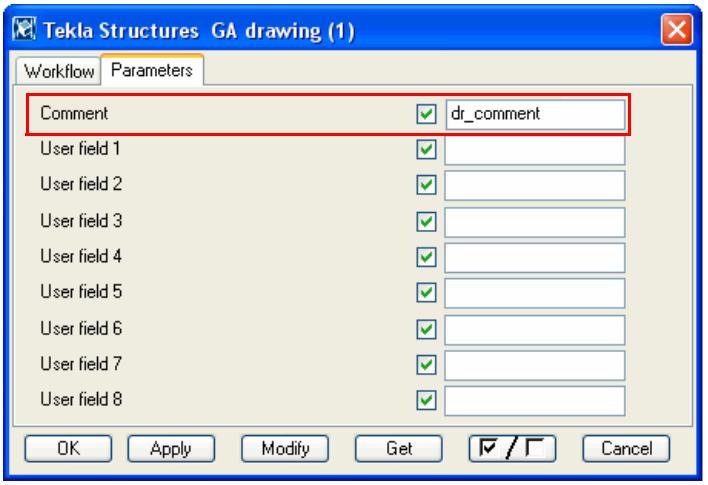
Comment-variable in drawings modified

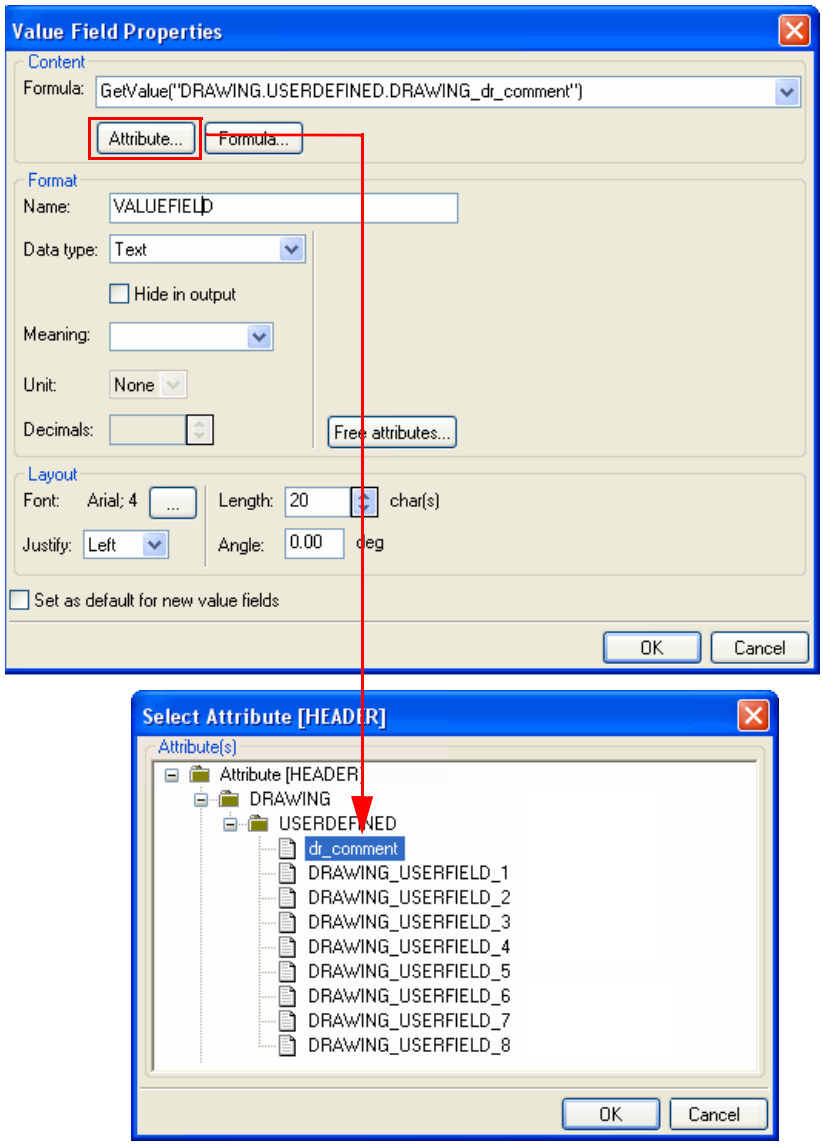
The **comment** variable, which can be used to add a comment to a drawing (which is displayed in templates) has been changed to **dr_comment**.

Assembly drawing properties



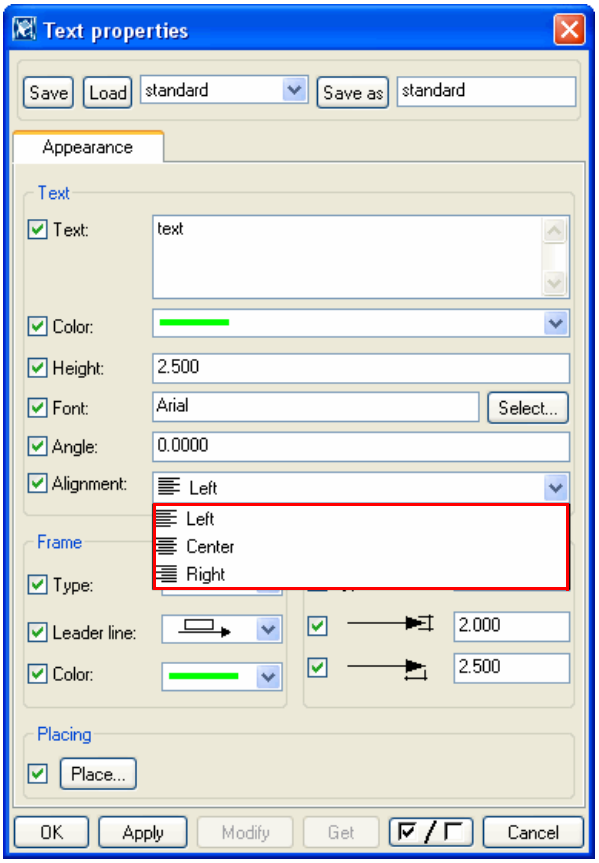
Used defined attibutes




Value Field Properties
Content
Formula:
GetValue("DRAWING.USERDEFINED.DRAWING_dr_comment")
Attribute...
Formula...
Format
Name:
VALUEFIELD
Data type:
Text
Hide in output
Meaning:
Unit:
None
Decimals:
Free attributes...
Layout
Font:
Arial; 4
...
Length:
20
char(s)
Justify:
Left
Angle:
0.00
deg
Set as default for new value fields
OK
Cancel
Select Attribute [HEADER]
Attribute(s)
Attribute [HEADER
DRAWING
USERDEFINED
dr_comment
DRAWING_USERFIELD_1
DRAWING_USERFIELD_2
DRAWING_USERFIELD_3
DRAWING_USERFIELD_4
DRAWING_USERFIELD_5
DRAWING_USERFIELD_6
DRAWING_USERFIELD_7
DRAWING_USERFIELD_8
OK
Cancel

Aligning Multi text

Tekla Structures 13.1 includes more text align options:



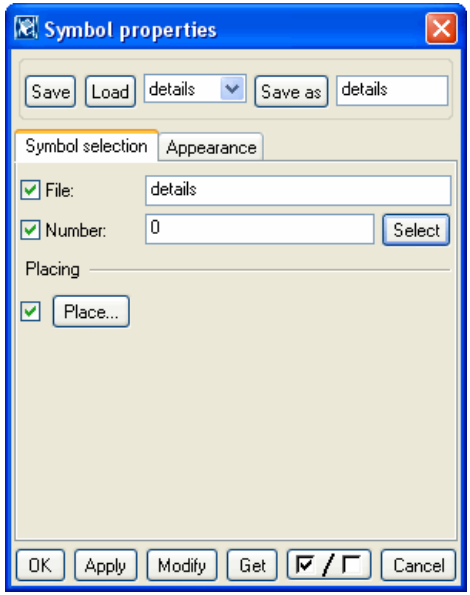
New Symbol file "Details.sym"

A new symbol file **Details.sym** is added.

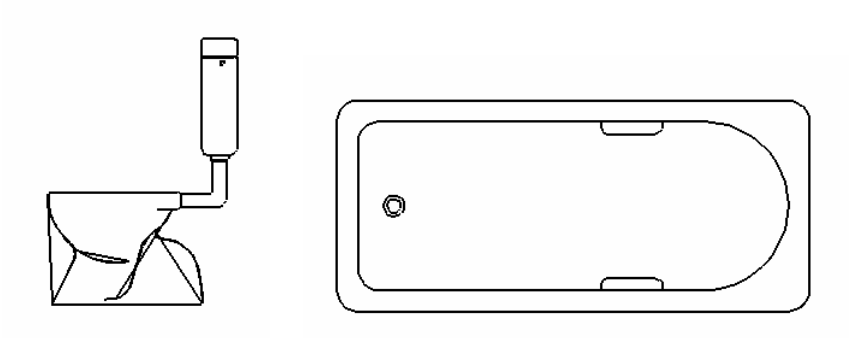
This file contains a number of pictures you can use on drawings. You can find this new file in the folder:

disk:\TeklaStructures\13.1\environments\country-independent\symbols.

In the Drawing Editor you can load the setting **details** which refers to the symbol folder.



Example

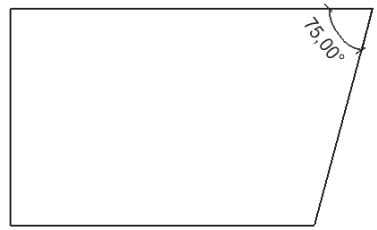


Angle dimensions on plate drawings

Angle dimensions are automatically placed on plate drawings (Single Part drawings):

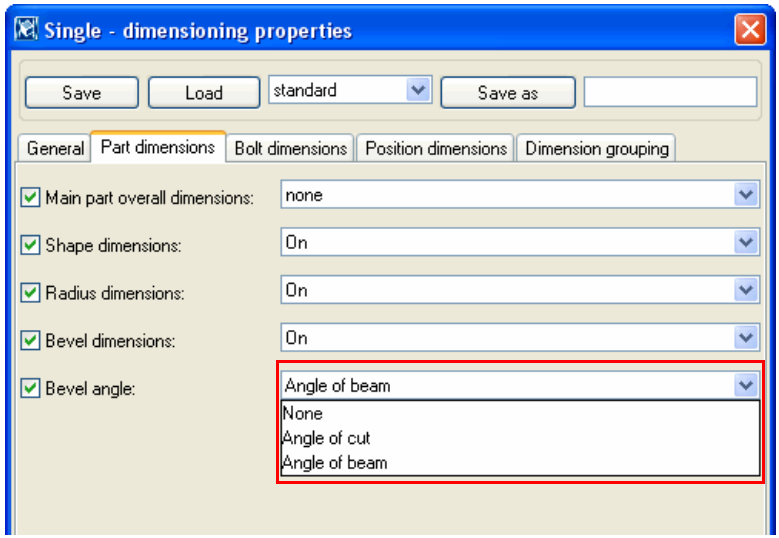


Cut angle



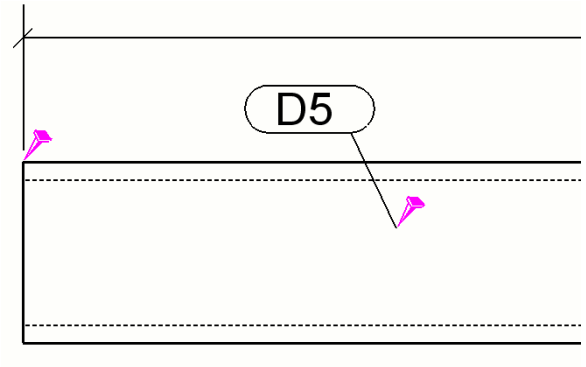
Material angle

It is also possible to hide the angle dimension. To set the appearence of angle dimensioning go to the properties of the single-part-drawing, click **Dimensioning...** go to the tab Part dimensions to set the bevel angle.



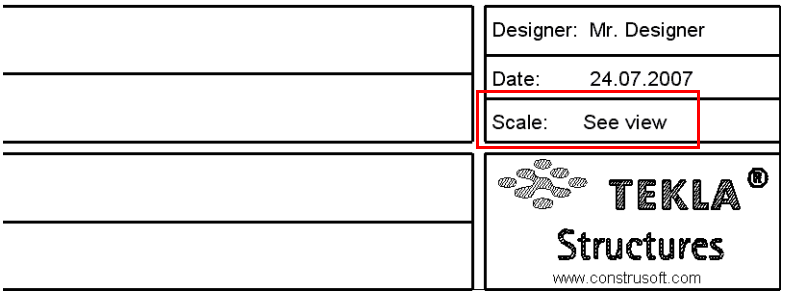
Moving the leaderlines start point

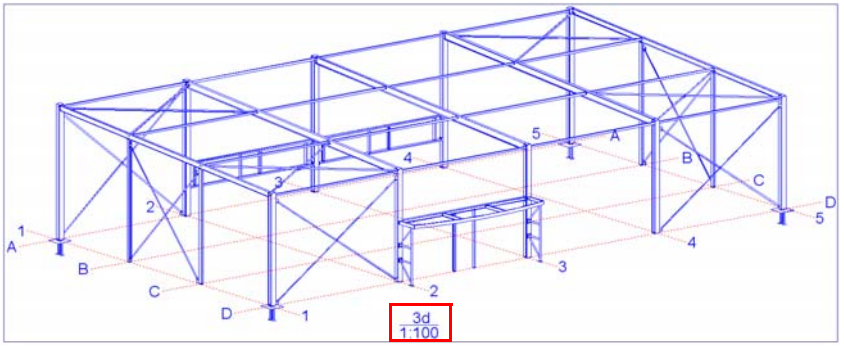
The leaderline startpoint can be moved now.



Template B

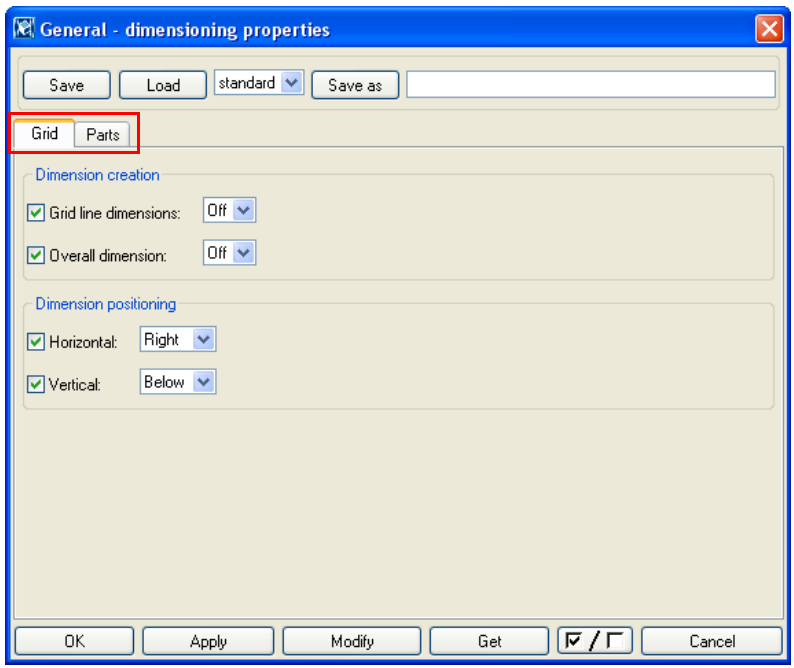
In the drawing layout, **Template_B** is no longer used, **Template_B2** is now used instead. **Template_B2** contains a reference for the view scales:





Automatic dimensioning of GA drawings

In Tekla Structures 13.1, grids and parts can be dimensioned automatically. To do this, click the new button **Dimensiong** in dialog **General arrangement drawing properties**:



See also For more information about dimensioning general arrangement drawings, go to **Help > Learning Center > What's new**.

Displaying connecting parts on an anchorbolt plan

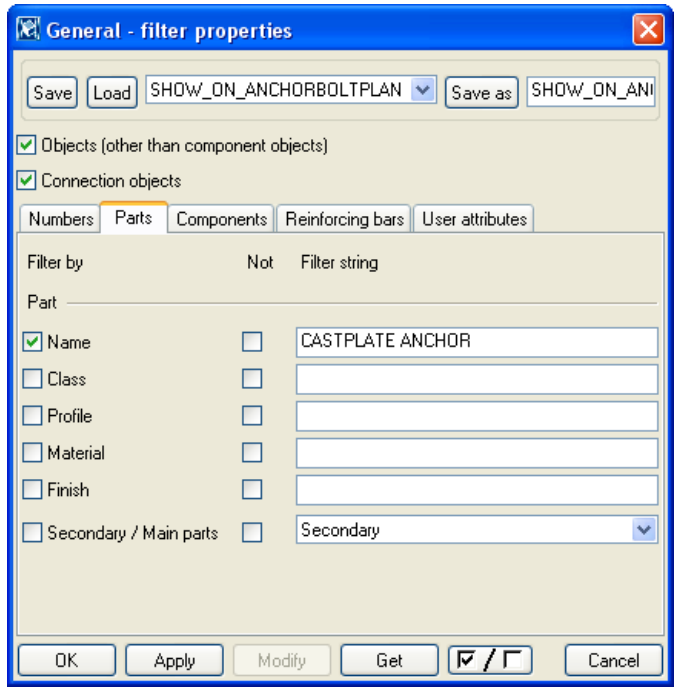
It is possible to display **connecting** parts, such as castplates, in anchorboltplans where 2 scales are used.

To do this, a GA-drawing filter must be created and the filter name must be entered in variable XS_ANCHOR_BOLT_PLAN_ADDITIONAL_PARTS_FILTER.

In next example you will display both anchors and castplates in an anchorbolt plan.

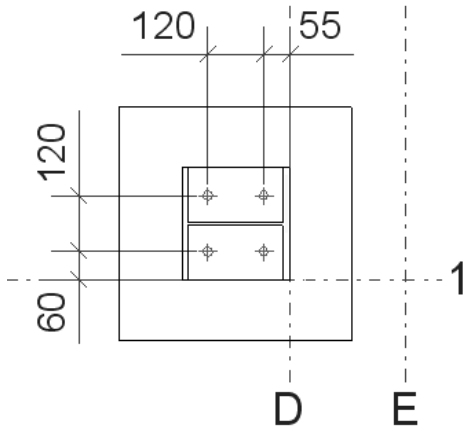
1. Specify a name for the General Arrangement Filter, e.g. SHOW_ON_ANCHORBOLTPLAN. Next, this filtername in entered in variable XS_ANCHOR_BOLT_PLAN_ADDITIONAL_PARTS_FILTER **(Tools > Advanced options > Drawing properties)**
2. Go to **Properties > Drawing...**

3. Press **Filter...** in dialog General Arrangement properties
4. In the **Parts** tab, enter CASTPLATE and ANCHOR in the Name-field



5. Save filter as SHOW_ON_ANCHORBOLTPLAN.

Now, if you create an anchorboltplan, both the anchors and castplates are displayed in the drawing.

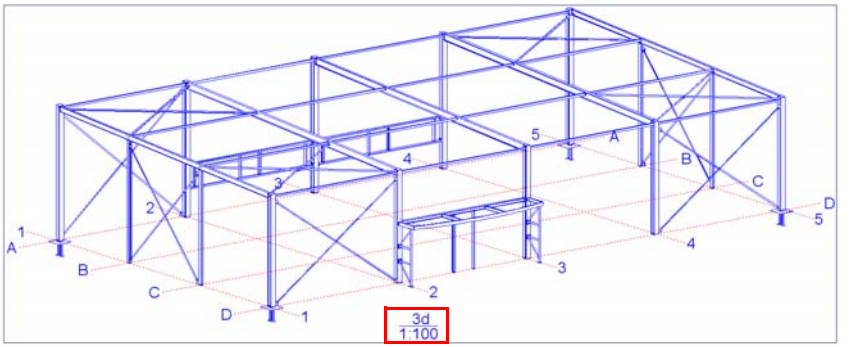


Labels in drawing views

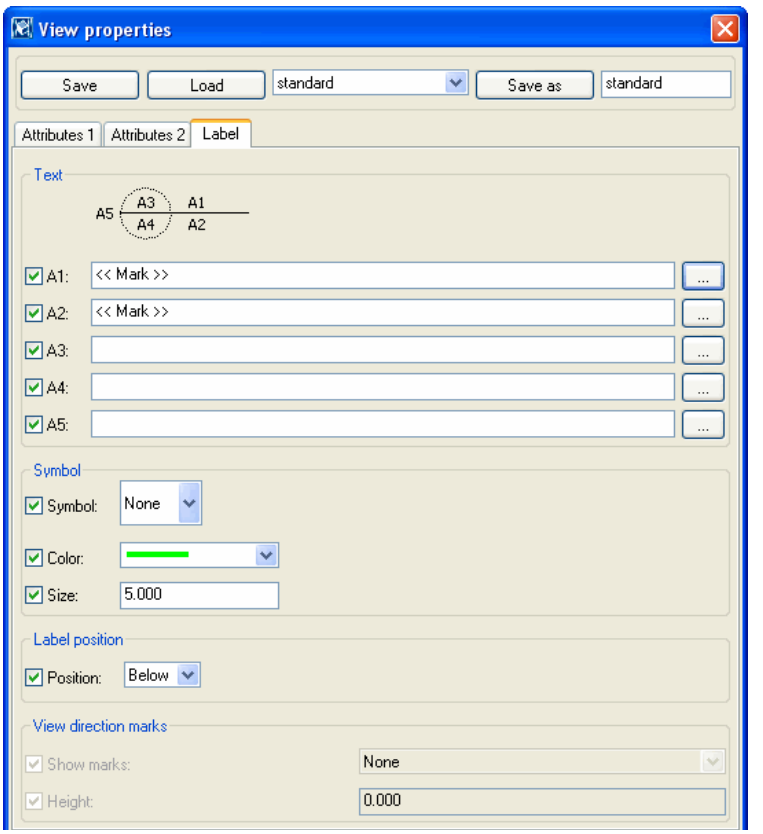
Drawing view labels can display more information than before. For example, it is possible to display both view name and view scale. Some examples.

General drawing view labels

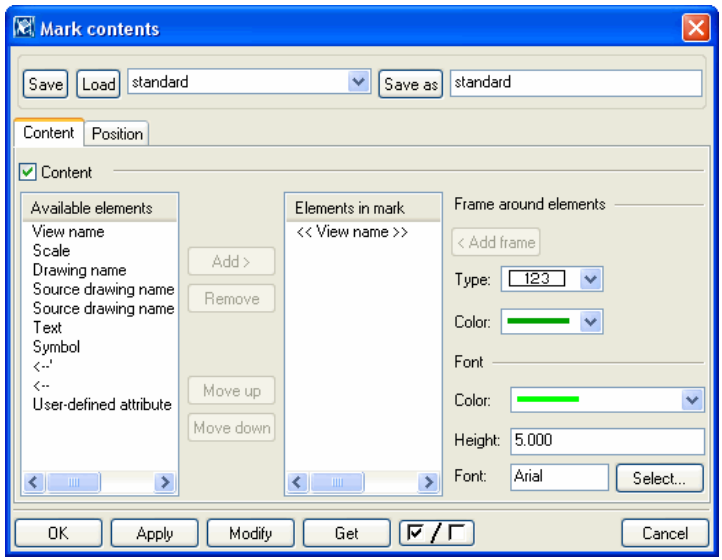
1. Doubleclick a random drawing view. The dialog **View properties** appears.



2. Click tab **Label** and define view label .

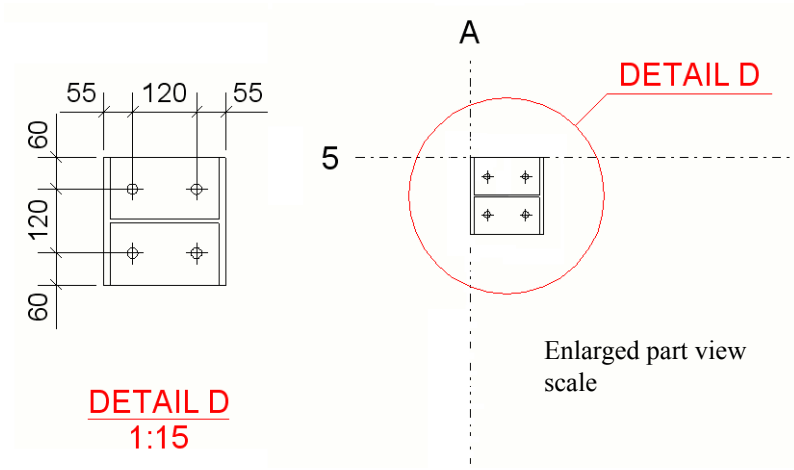


3. To define the view label content, click the **...** buttons beside the text fields to open the **Marks contents** dialog, e.g.**A1**:



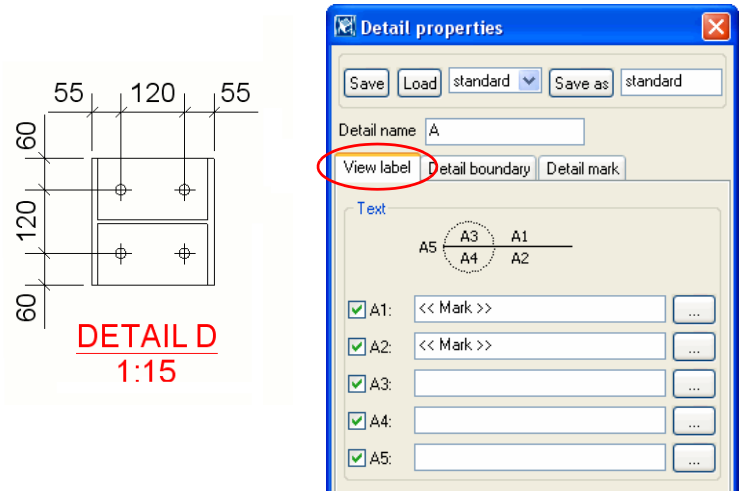
Labels in detail views

For example, a foundation view with views which includes the enlarged parts and detail views, you **need** to set this **before** you create dawings:

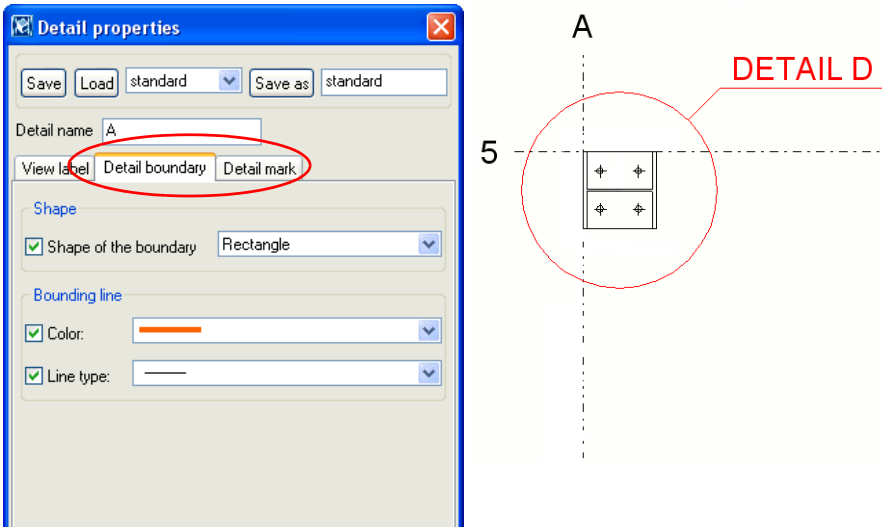


Detail view

1. In the Model Editor, go to **Properties > General arrangement drawing...** to open the dialog box **General arrangement drawing properties**
2. Click **Detail View...** to open dialog **Detail Properties**
3. In tab **View label** you can set the detail view label



4. In tabs **Detail boundary** and **Detail mark** you can set the views which includes the enlarged parts.

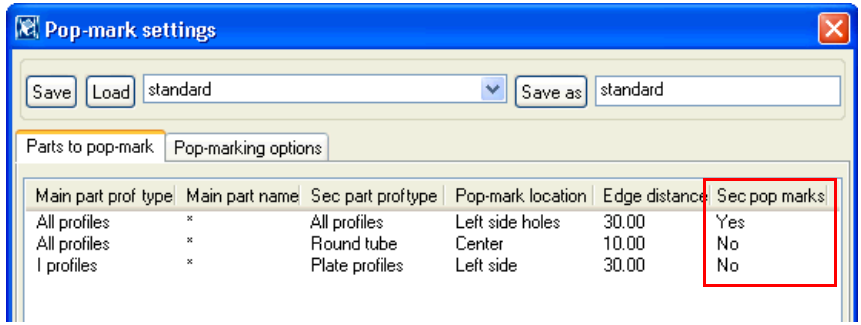


See also For more information, see the **Tekla Structures Online Help** and **What's New** (**Help** > **Learning Center** > **What's New**).

Pop marks

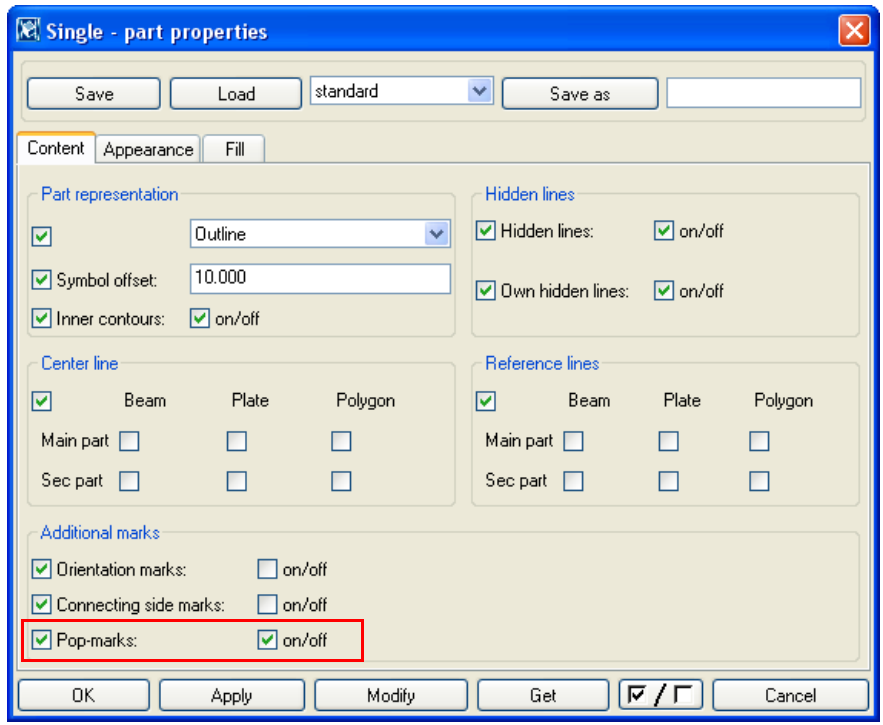
Pop marks in secondary parts

Pop marks can now also be created in secondary parts. To do this, switch option **Secondary pop marks** to "Yes".



Pop marks in drawings

Pop marks can now be displayed in single part and assembly drawings. To do this, go to the drawings Part Properties dialog:

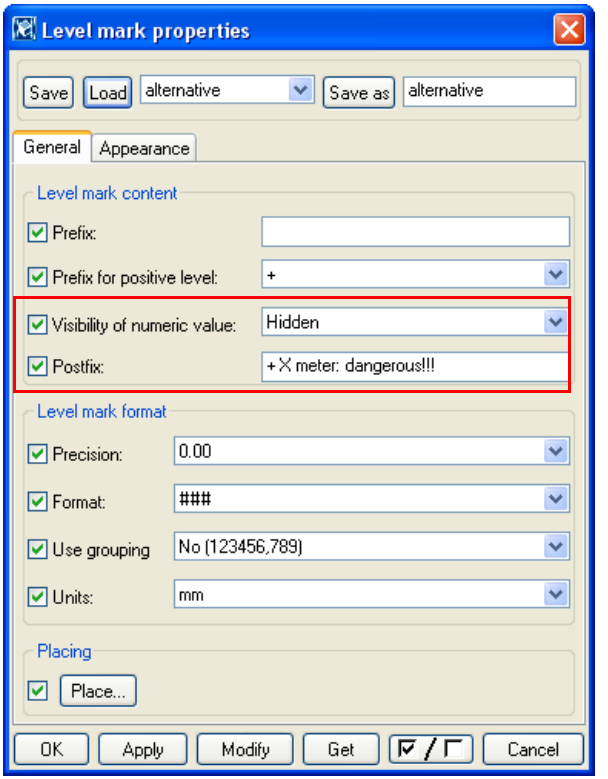


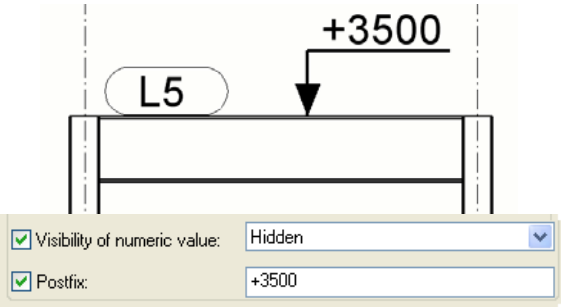
See also

For more information about pop marking, see the **Tekla Structures Online Help** and the **Release Notes 13.1**.

Alternative Level Mark

A new setting **alternative** is added to dialog Level mark properties. If this setting is used, you can enter a random value for the level mark, preceding by prefix "+". This prefix is obligatory.







These alternative level marks are never being updated automatically!

Custom components

Password protection for custom components

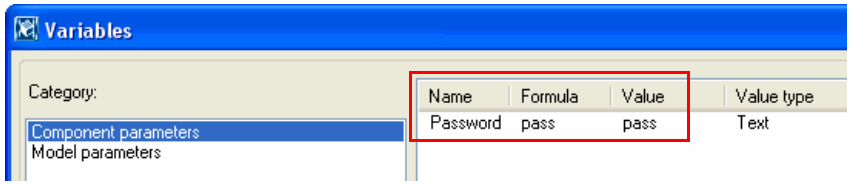
Custom components can now be protected by a password. The password protects the component from being edited.

How to use

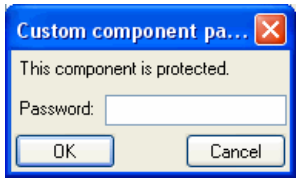
To set a password for a custom component:

1. Select a custom component you want to protect. Then right-mouse-click > **Edit custom component**.
2. In the Custom Component Editor, click the **Display variables**  to open the **Variables** dialog.
3. Add a new parameter by clicking the <**Add**> button.
4. Enter "**Password**" in the **Name** column.
5. Enter a password in field **Formula**, e.g. "pass".

In the picture below you can see how a password protection is created.



After that the parameter is added, everytime you edit the custom componentm you must enter the password:



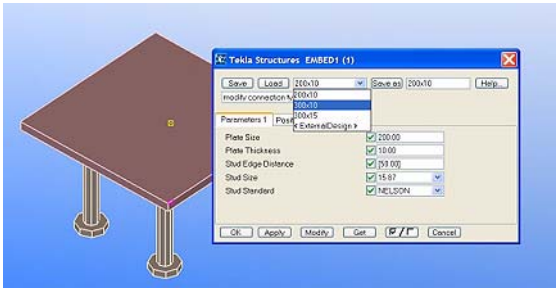
Component name and Component attribute file

It was already possible to specify the component name and component attribute file, but it was necessary to use textfields and exact filenames.

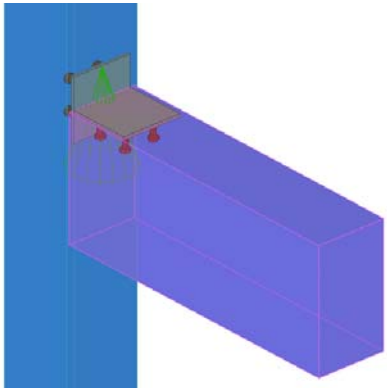
Tekla Structures 13.1 contains a new variable **Component attribute file**, to which the **Component name** is connected.

How to use

1. Create a parametric custom component.
2. Create some setting files.



3. Create a custom component which uses this part.



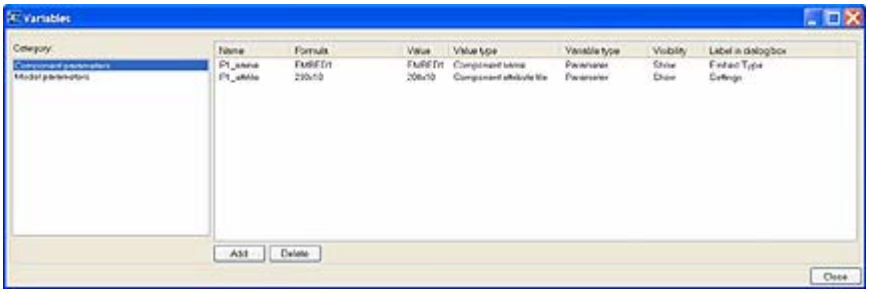
4. In the Custom Component editor, add the these two variables:

   Value type = Component name

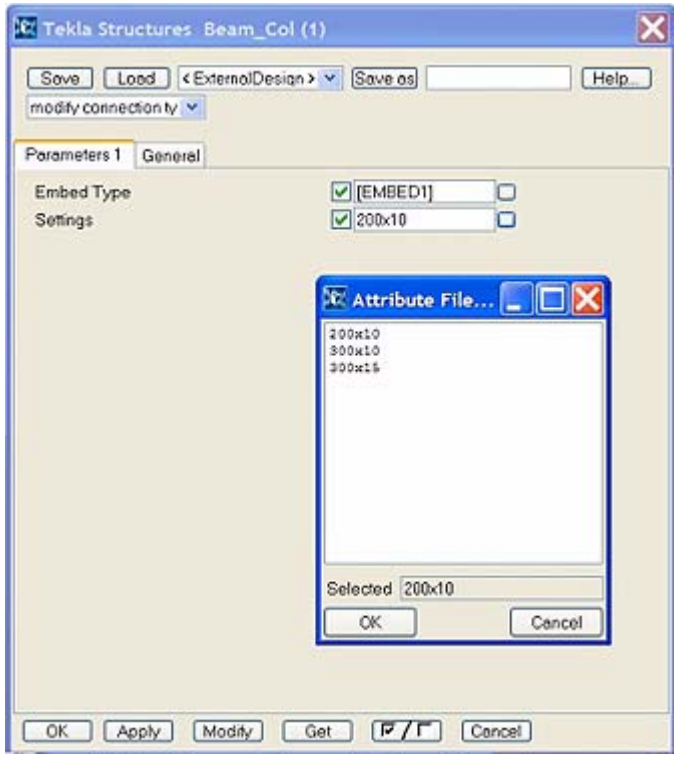
   Value type = Component attribut name



The prefix needs to be equal for both variables. (P1 in the example)



It is now possible to select the components settings.


Tekla Structures Beam_Col (1)
Save
Load
<ExternalDesign>
Save as
Help...
modify connection ty
Parameters 1
General
Embed Type
[EMBED1]
Settings
200x10
Attribute File...
200x10
300x10
300x15
Selected
200x10
OK
Cancel
OK
Apply
Modify
Get
Cancel

NC files

DSTV convert tool

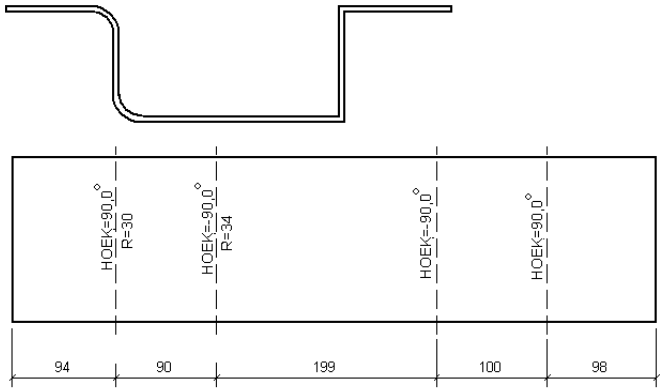
Several new options have been added to the new version of the DSTV convert tool, version 3.0.3:

1. In tab **DSTV - 1** , the option **Replace "s" in BO block into** has been added.

   The X-coordinate of a hole in the BO-block is often accompanied by a suffix; this can be the letters “s” (Symmetrical, “o” (Top) or a “u” (Bottom). Old CNC machines have problems reading the NC file if the suffix is a “s”. With this options, you can change the suffix into another character. Generally, if the “s” suffix is changed into an “o”, then the NC file read problems won’t occucur.

2. In tab **DSTV-files - 2** , the option **Remove KA block** has been added.

   Old machines cannot handle the KA block, this option removes the full KA block. The KA (Kantungen in German) block contains information about bendings for unfolded plates




```
      106.65   150.00     0.00     0.00
      101.42   150.00     0.00     0.00
       96.18   150.00     0.00     0.00
       90.94   150.00     0.00     0.00
       85.71   150.00     0.00     0.00
       80.47   150.00     0.00     0.00
       75.24   150.00     0.00     0.00
       69.99   150.00     0.00     0.00
        0.00   150.00     0.00     0.00
KA
     93.56     0.00    93.56   150.00    90.00
    183.83     0.00   183.83   150.00   -90.00
    382.53     0.00   382.53   150.00   -90.00
    482.53     0.00   482.53   150.00    90.00
EN
```

```
  106.65   150.00     0.00     0.00
  101.42   150.00     0.00     0.00
   96.18   150.00     0.00     0.00
   90.94   150.00     0.00     0.00
   85.71   150.00     0.00     0.00
   80.47   150.00     0.00     0.00
   75.24   150.00     0.00     0.00
   69.99   150.00     0.00     0.00
    0.00   150.00     0.00     0.00
EN
```

Tekla Structures NC file **NC bestand after conversion**

3. In tab **DSTV - 2** , the option **Remove countersunk data** has been added.

   If countersunk holes are placed in a profile, the hole is described by two lines in the NC file: the 1st line describes the round hole (coordinate and diameter); the 2nd line describes the countersunk process (coordinate, outer-diameter, countersunk depth and -angle, according to DSTV). Because most CNC drilling/sawing machines can not handle countersunk holes, it is sometimes better to have the line containing the countersunkhole data removed from the NC file. The line containing the round hole data remains in the NC file.

   Tekla Structures NC file:

```
BO
  o   35.00s     25.00    14.00      0.00
  o   35.00s     25.00s   24.00      6.50    0.00      0.00     90.00
  o   35.00s    125.00    14.00      0.00
  o   35.00s    125.00s   24.00      6.50    0.00      0.00     90.00
  o  135.00s     25.00    14.00      0.00
  o  135.00s     25.00s   24.00      6.50    0.00      0.00     90.00
  o  135.00s    125.00    14.00      0.00
  o  135.00s    125.00s   24.00      6.50    0.00      0.00     90.00
```

   NC file after conversion:

```
BO
  o    35.00s     25.00    14.00    0.00
  o    35.00s    125.00    14.00    0.00
  o   135.00s     25.00    14.00    0.00
  o   135.00s    125.00    14.00    0.00
```

See also DSTV convert tool v. 3.0.3 can be downloaded from **Construsoft Xtranet**.

Multi user

Multi user server

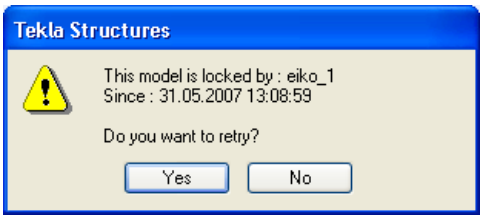
If you want to work in multiuser mode in Tekla Structures 13.1, you have to use the new Multi User Sever 2.1. You can download this program from our Xtranet.

You can only run one verion at the same time. So only version 1.1 or 2.1.

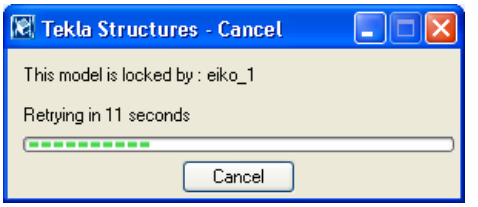
Version	Multi user server version
Tekla Structures 12.x	1.1
Tekla Structures 13.x	2.1

Automatic retry when a multi-user moidel is locked

If you work in multi-user mode, you sometimes use commands, like saving the model, that requires the model to be uinlocked. When the model is locked Tekla Structures can place the commend in a queue until the model is unlocked.



If you choose **Yes** , Tekla Structures shows the next dialog:



Tekla Structures will retry to execute the command every 15 seconds, or you can abort by clicking **Cancel**.

Opening a multi-user model

If you have created a multi-user model in Tekla Structures 12.1, and you want to open the model in Tekla Structures 13.1, you have to open the model in single-user mode first.

After that it is possible to open the model in multi-user mode.

Saving model revision notes

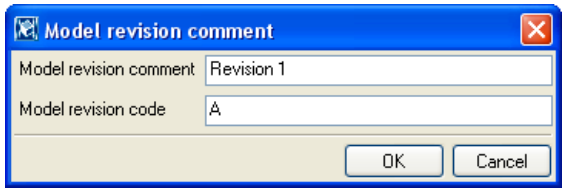
From Tekla Structures 13.1 on, it is possible to save model revision notes in multi-user models.

How to use

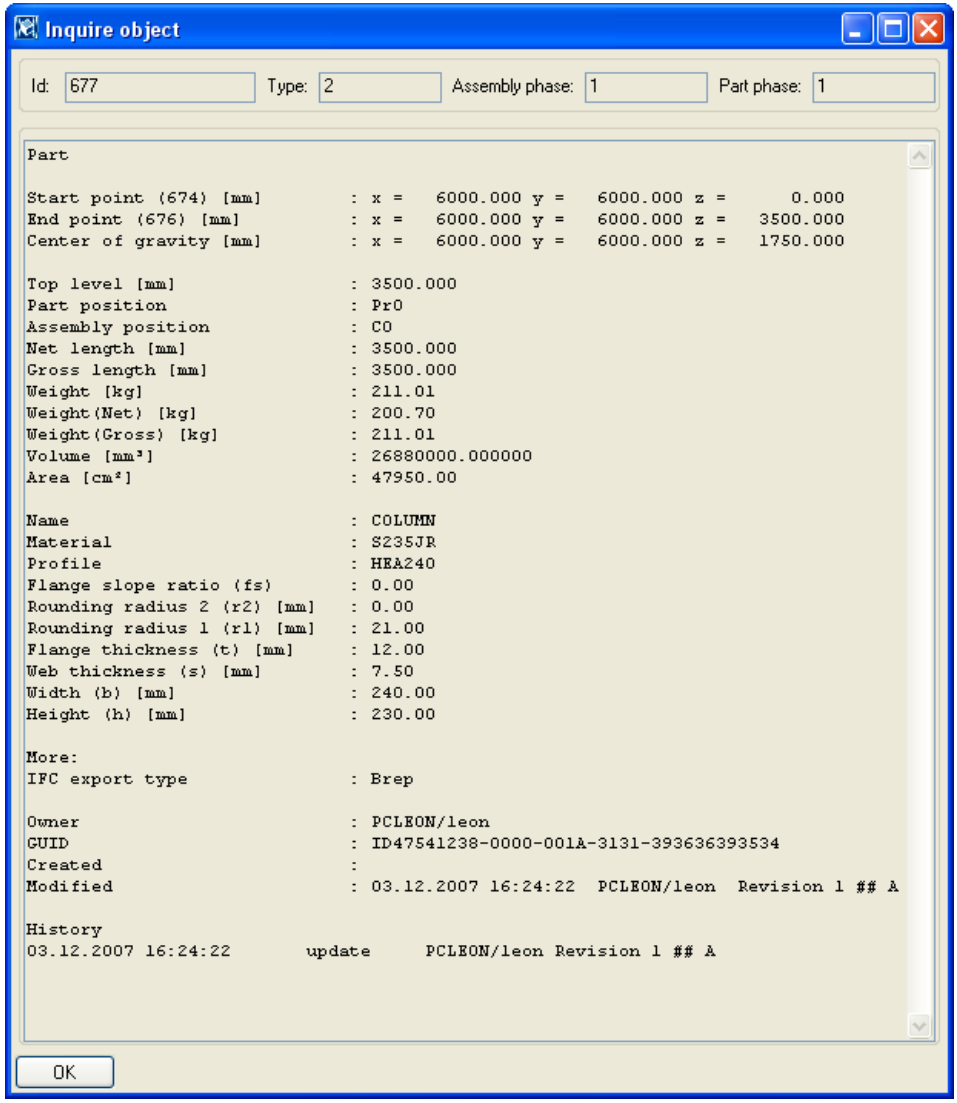
To save the notes, set variable XS_SAVE_WITH_COMMENT to TRUE (**Tools > Advanced options > Multi user...)** and set variable XS_COLLECT_MODEL_HISTORY to TRUE **(Tools > Advanced options > Speed and accuracy)**.

This is the default setting, if you want the dialog not to be displayed, do not enter any value for variable XS_SAVE_WITH_COMMENT.

If a multi-user model is saved, this dialog will appear:



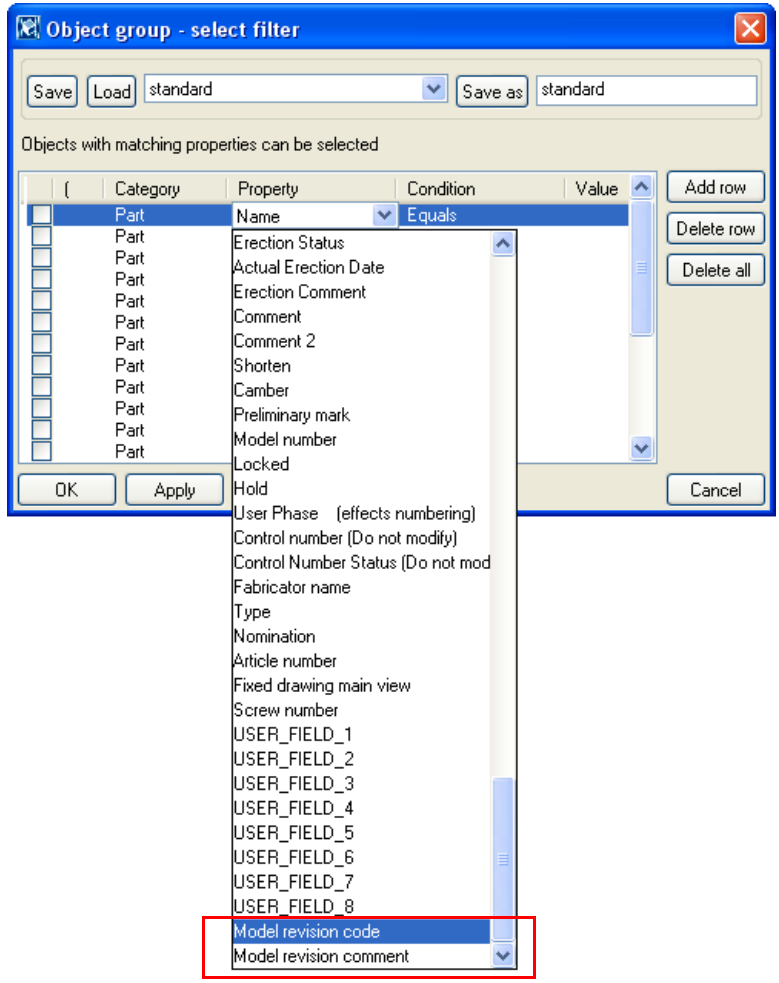
Tekla Structures will assign the entered values to only the parts that have been modified after the last time the model was saved. You will see the model revion information when you request the objects information by **Inquire Object**:





If the **Model revision comment** field is empty, no values will be allocated to the concerning parts.

You can also use this information to select and filter in Tekla Structures:


Object group - select filter
Save
Load
standard
Save as
standard
Objects with matching properties can be selected
(
Category
Property
Condition
Value
Add row
Delete row
Delete all
Part
Name
Equals
Erection Status
Actual Erection Date
Erection Comment
Comment
Comment 2
Shorten
Camber
Preliminary mark
Model number
Locked
Hold
User Phase (effects numbering)
Control number (Do not modify)
Control Number Status (Do not mod
Fabricator name
Type
Nomination
Article number
Fixed drawing main view
Screw number
USER_FIELD_1
USER_FIELD_2
USER_FIELD_3
USER_FIELD_4
USER_FIELD_5
USER_FIELD_6
USER_FIELD_7
USER_FIELD_8
Model revision code
Model revision comment
OK
Apply
Cancel

Notes: