Tekla Structures Basic Training

Tekla Structures 10.0

April 26, 2004

Contents

11 Assembly and Single Part Drawings

In this lesson

We will introduce the creation of assembly and single-part drawings in Tekla Structures.

We will first create single-part and assembly drawings by using predefined drawing wizards (drawing wizards are an automatic way of creating single, assembly and multi-drawings).

We will then edit the drawing properties and create a new drawing wizard. The Drawing Wizard will use the edited drawing properties and the select filters that we defined in an earlier lesson. Then we will show how the same drawings can be created manually. We will also demonstrate how updating effects the drawings.

Revision control of all drawing types is presented in Lesson 10 Principles of working with drawings.

11.1 Integration between Drawings and the Model

Assembly and Single-Part Drawings

Single-part drawings are workshop drawings of each of the individual steel parts in the model. Assembly drawings are workshop drawings, in which details of an assembly consisting of the steel parts are presented for fabrication.

All of the views in single-part or assembly drawings are current views of the members as they are in the model.

When the model contains any identical members, the drawing is a view of one of these members. The drawing, however, contains information about the quantity of all of the identical members. If the "host" member of the drawing is modified or deleted, it will get a new position mark at the next numbering. Tekla Structures will then automatically assign the original drawing to another member with the original position mark.

Tekla Structures integrates the drawings and reports with the model. This means that, for example, dimensions and marks in the drawings are always correct. Because the information in the drawings and reports comes directly from the model, you cannot delete any of the parts or bolts from the drawings. You are able to filter out parts and bolts in the drawings, or make them invisible.

You can create drawings and reports at any stage of the project. If you change the model, Tekla Structures updates the related drawings when you next carry out numbering.

For more information, see **Help: Drawing > Introduction to drawings > Basics**.

11.2 Create Drawings Using Drawing Wizards

Once you have numbered the model, you can create assembly and single-part drawings from the model. Drawing wizards are the most effective way to create drawings in Tekla Structures.

Wizards automatically produce different types of drawings of different parts, such as beams, columns, and braces. You can use wizards to create single-part, assembly, or multi-drawings using the settings defined in the wizard files.

You can use the predefined wizard files, edit them, or create your own drawing wizards. The **Wizard** dialog box lists the available wizard files.

For more information on drawing wizards, see:

Help: Drawing > Getting started with drawings > Using drawing wizards

Help: Drawing > Getting started with drawings > Drawings reference > File > Wizard…

Drawing wizards cannot be used to create cast unit (CU) drawings of concrete structures.

Functionality of Drawing Wizards

For each member type in the model, drawing wizards automatically perform the following steps:

1. Define the drawing type to be created (single, assembly or multi).
2. Define the drawing properties to be used.
3. With the given select filter, select the parts from which to create drawings.
4. Create drawings.

When you apply a wizard, you can choose whether the wizard creates drawings from all parts of the model, or just from the selected parts.

By creating wizards that match the select filters and drawing properties in the project you can automatically create all single and assembly drawings of the parts using the correct predefined properties.

Preconditions of using a drawing wizard:

- Numbering of the model must be up to date.
- The appropriate wizard file must exist.
- The saved drawing properties listed in the wizard file must exist.
- The saved select filters listed in the wizard file must exist.

Model members which will be selected by the select filter must exist.

Create Single-Part Drawings of Selected Parts

We will now use a drawing wizard to create single-part drawings of selected steel beams and plates.

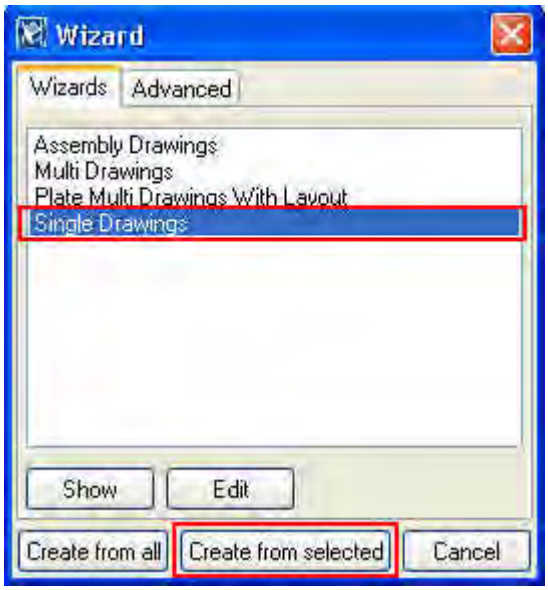
Create single-part drawings of steel beams

1. Open the BasicModelCombined.
2. Select the **BEAM_STEEL** from the drop down list of available **Select filters**.

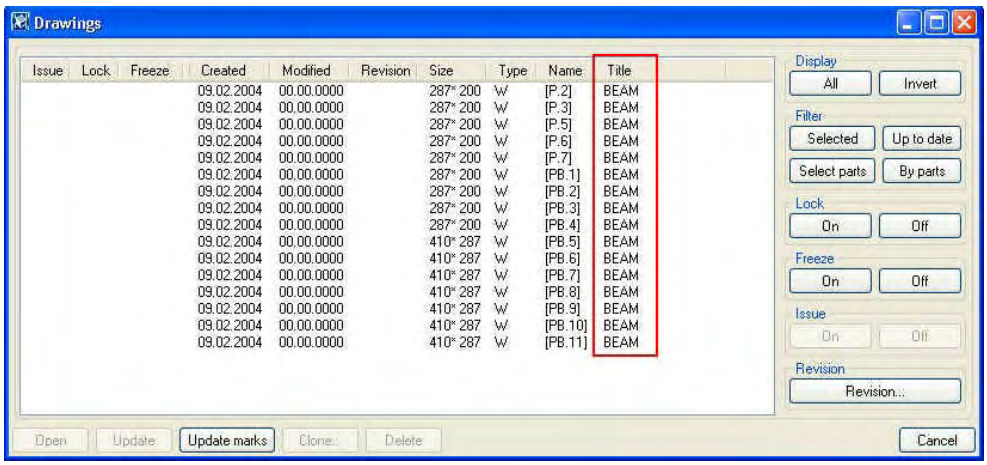
3. Select the whole model with an area selection.
4. To view the creation of the drawings, open the drawing list by clicking the **Open drawing list** icon.

5. Select **File > Wizard…** from the menu or click the **Wizard** icon on the **Standard** toolbar to open the **Wizard** dialog box.

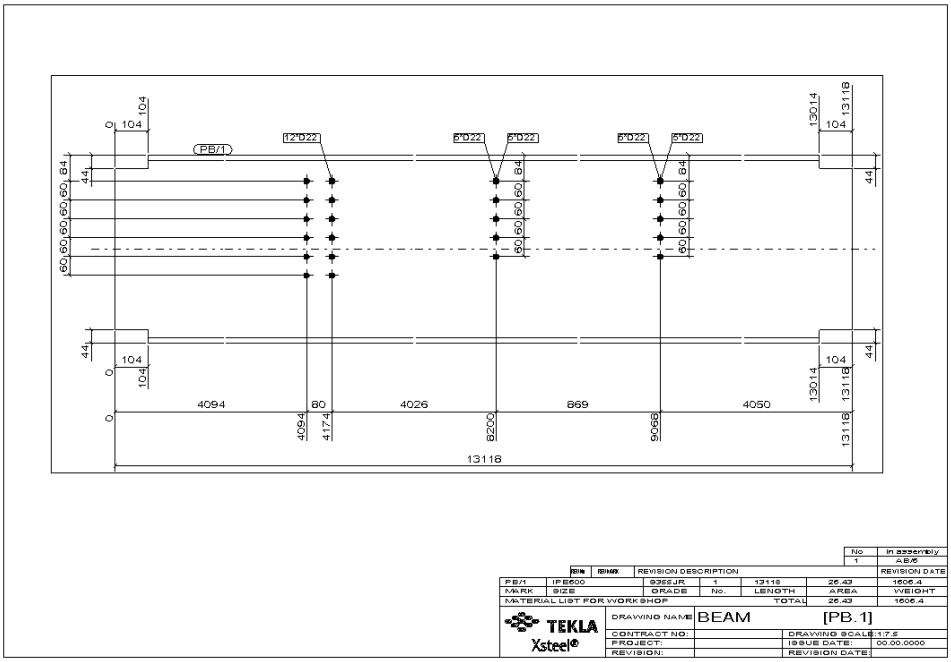
6. Select **Single Drawings** on the **Wizards** tab.



7. Click the **Create from selected** button.
8. In the drawing list check that the single-part drawings with the title **BEAM** were created.



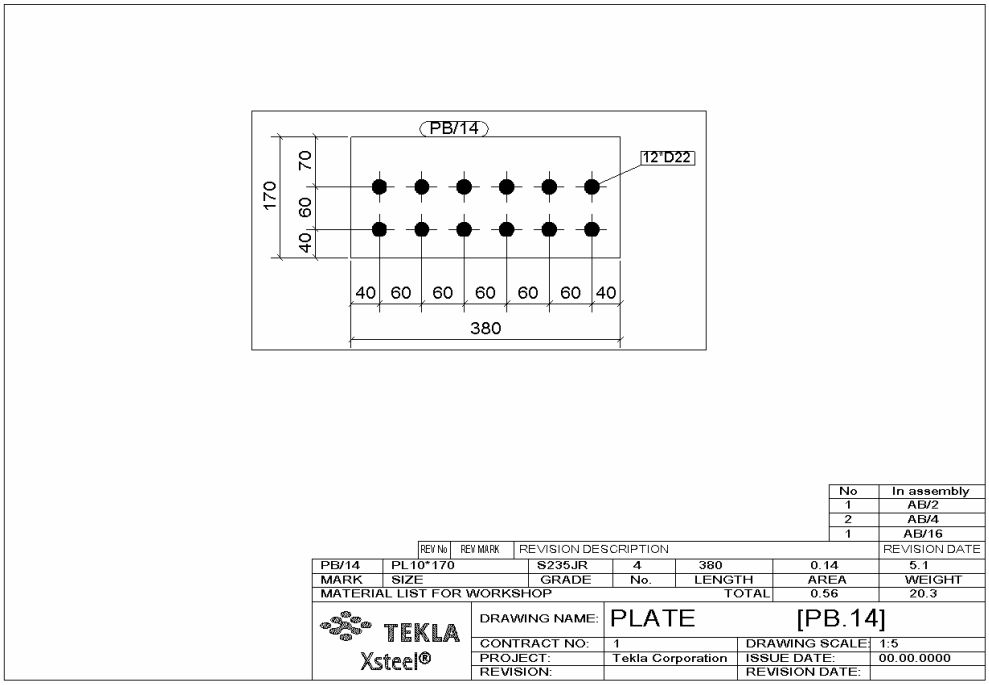
9. Open a few single-part drawings for viewing.



Create single-part drawings from plates

Following the procedure above, now create the single part drawings of the plates.

1. Select the **PLATE** select filter.
2. Select the whole model with an area selection.
3. Select **Single Drawings** on the **Wizards** tab in the wizard dialog box.
4. Click the **Create from selected** button.
5. In the drawings list check that the single-part drawings with the title **PLATE** were created.
6. Open a few single-part drawings for viewing.



Using the procedure outlined above, you could create single-part drawings from any other selected steel parts in the model (columns, braces, angles, etc.).

It is advisable to create all the single and assembly drawings with the wizard, even for a single part.

Tekla Structures displayes a **Cancel** dialog box during the creation drawings. Click **OK** in the dialog box to stop creating the drawings.

To create single-part drawings from all of the steel parts at once you can use the option **Create from all**.

Create Assembly Drawings of All Parts

Next we will create assembly drawings of all of the parts by using another drawing wizard.

To create all of the assembly drawings:

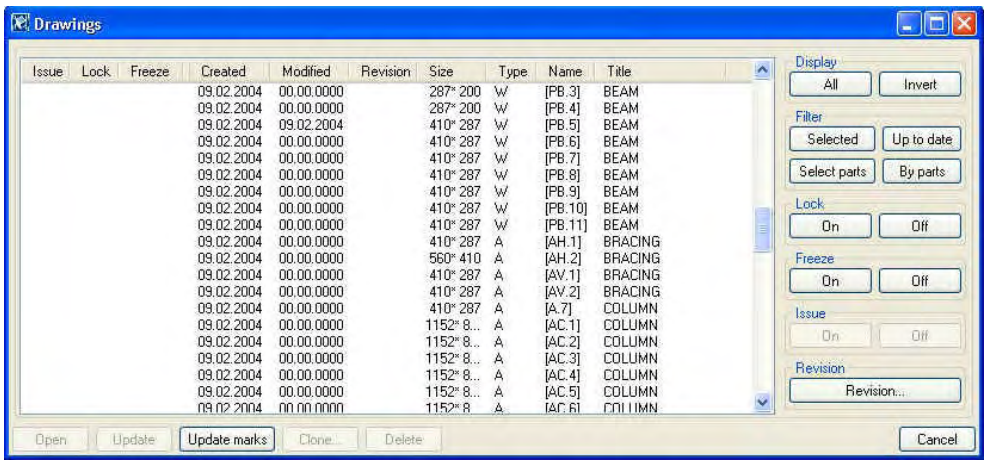
Create assembly drawings

1. Click the **Wizard** icon to open the **Wizard** dialog box.

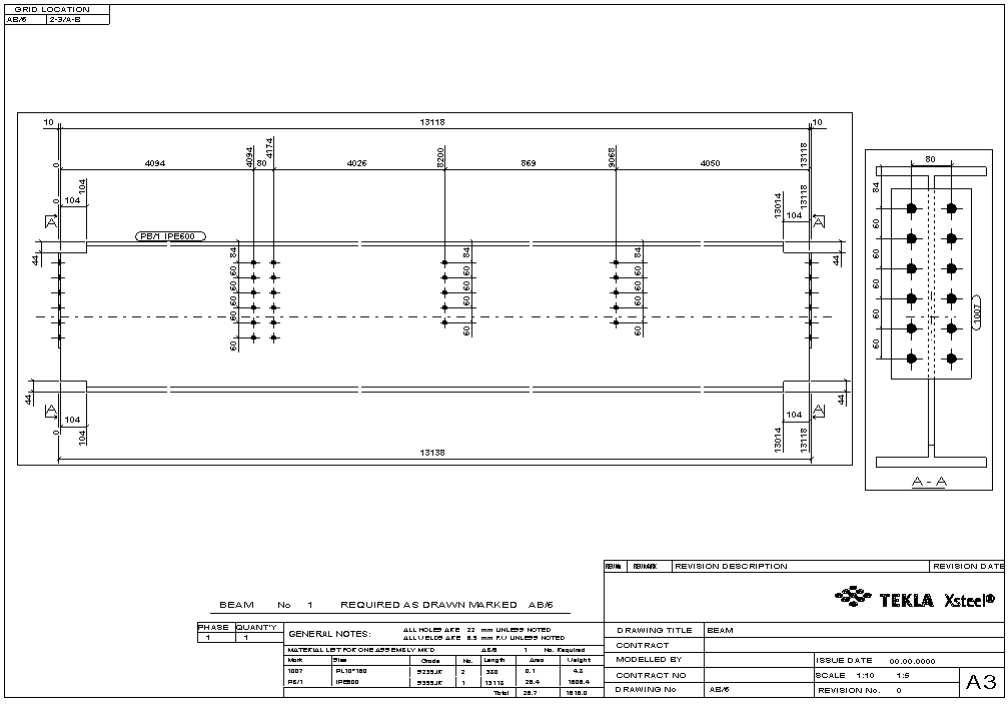
2. Select **Assembly Drawings** on the **Wizards** tab.
3. Click the **Create from all** button.

Use the **Create from all** button to create drawings from the whole model at once. Note that Tekla Structures will not create single-part drawings or assembly drawings from concrete parts.

4. In the drawings list check that the assembly drawings were created correctly (sort the drawing list by **Title**).



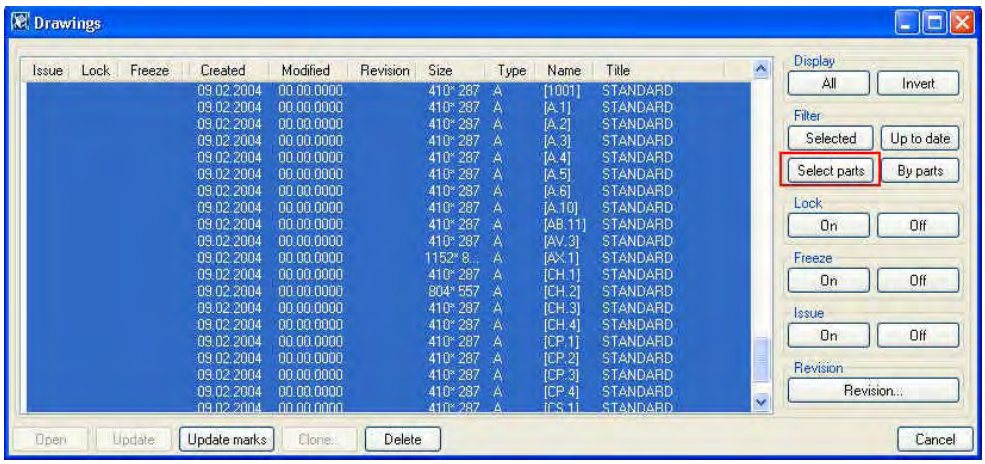
5. Open a few assembly drawings (which are not named **STANDARD**) for viewing.



For parts that don't match with the drawing properties or filters listed in the wizard file, the predefine wizards creates drawings using **STANDARD** properties.

Select parts with drawing title STANDARD

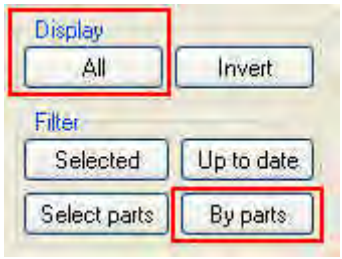
1. Select drawings with the title **STANDARD** in the drawing list.
2. Click the **Filter - Select parts** button.



The parts associated with the selected drawing are now highlighted in the model. You will find that e.g. that objects like braces are highlighted in the model (the default wizard does not have a request for bracing).

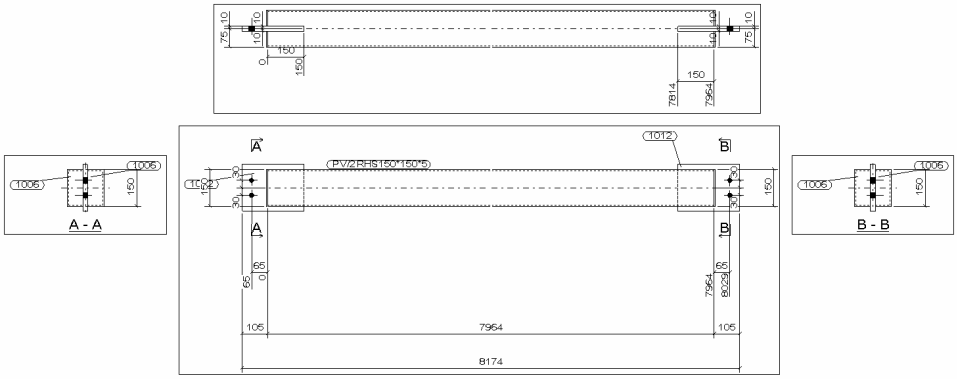
Open brace drawing

1. Select one vertical brace on gridline A.
2. Click the **Display - All** button to ensure that all drawings are shown in the list.
3. Click the **Filter - By parts** button.



The drawing list shows now only the assembly drawing created from the brace selected in the model.

4. Open the drawing



The brace drawing was created with standard properties and not properties defined for bracing.

A drawing wizard will not create a duplicate drawing for any member already having, e.g. a single-part or assembly drawing.

11.3 Edit Drawing Properties

We will now define specific drawing properties for both horizontal and vertical bracing and save the properties to be used later in the drawing wizard.

As an example we will open one vertical brace drawing for editing. Using this drawing we will save the properties for the horizontal bracing. We will then edit some more properties and save them for the vertical bracing.

We will edit dimensioning protection and part mark properties. To illustrate the effect of single fields, we will first remove the existing dimensions from the drawing.

The editing we will do are just examples of using the drawing properties. The final drawing you get depends on the environment you are using and may not be identical to this example.

Study the Online help for information on each of the separate fields available in the drawing properties.

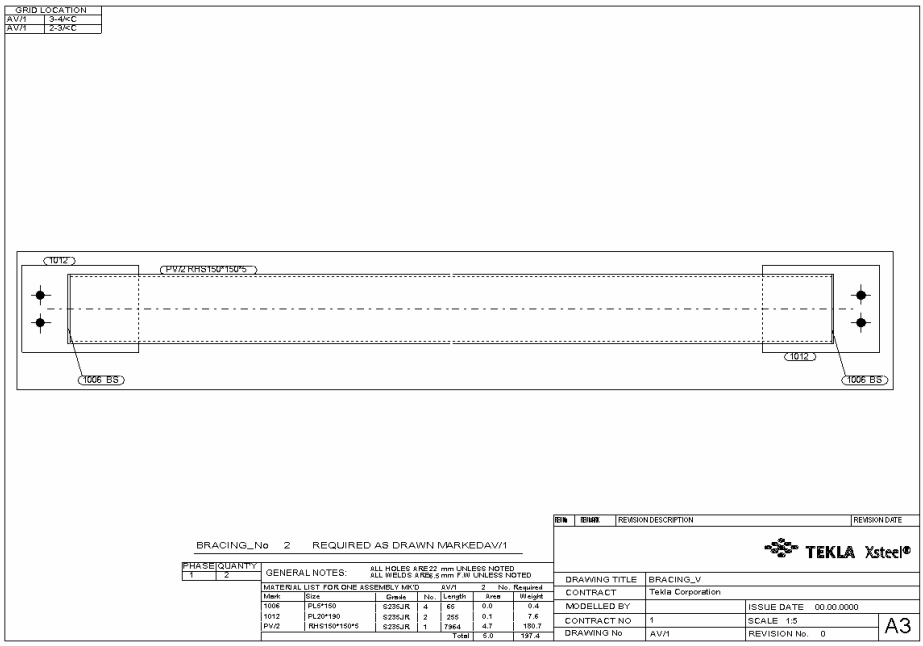
Help: Drawing > Dimensioning

Help: Drawing > Drawing Properties

Whenever possible, you should modify drawings by changing the drawing properties. These modifications remain when the drawing is recreated due to a model change. You can also use saved drawing properties in wizards.

Load properties no_dimension

1. Right-click on the drawing and select **Properties…** to open the **Assembly drawing properties**.
2. Load the predefined properties **no_dimensions**.
3. Click **Modify**.



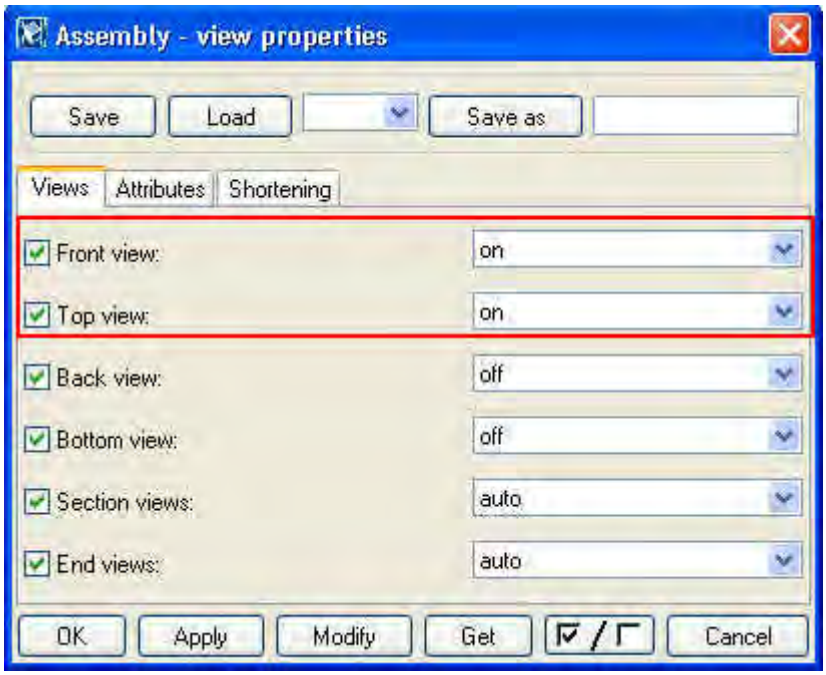
The drawing now appears without any dimensions and only the main view is visible.

By default, Tekla Structures creates the additional views only if it is necessary in order to show the dimensions in the drawing.

However, for our purposes, we want to have both the front and top views in the brace drawings regardless of the dimensions that may be needed.

Display both front and top views

1. In the **Assembly drawing properties** dialog box click the **View…** button
2. Choose the option **on** for the Front and Top views
3. Click **Apply** and then **Modify**

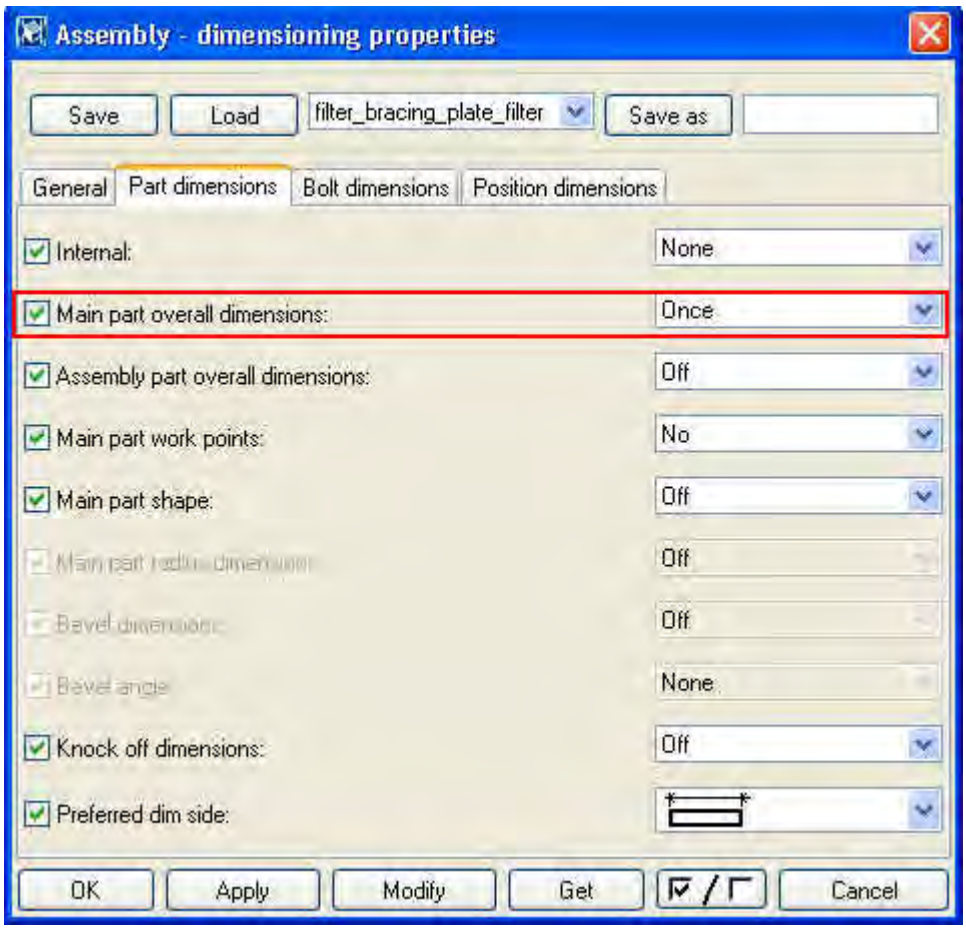


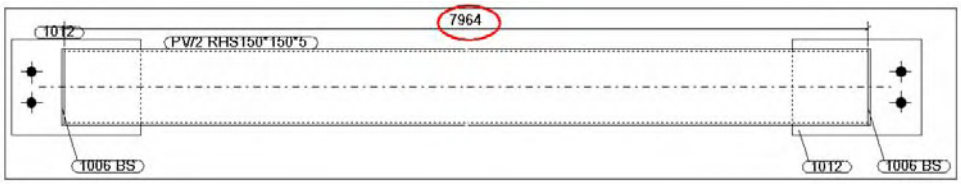
Now both the front and top views are displayed.

The first dimension we will add is the main part overall dimension.

Main part overall dimension

1. In the **Assembly drawing properties** dialog box click **Dimensioning… > Part dimensions**.
2. Select **Once** for **Main part overall dimensions**.
3. Click **Apply** and then **Modify**.



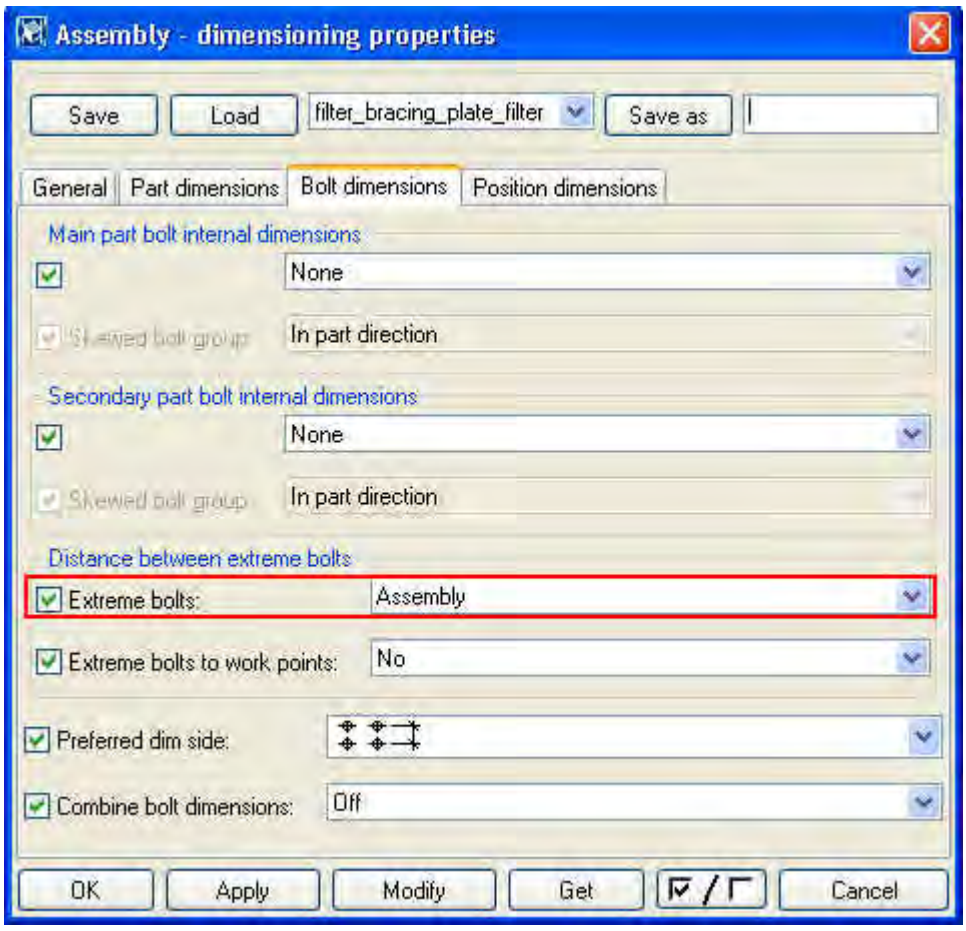


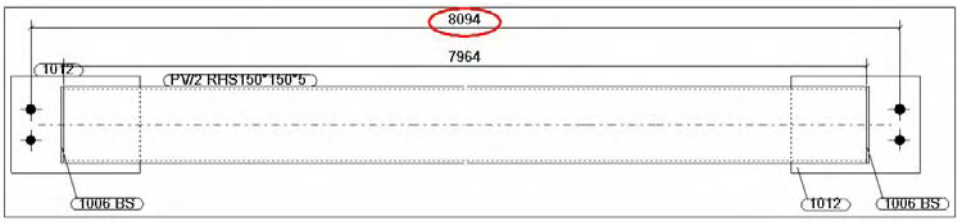
The main part overall dimension appears.

We will next add the dimension between the extreme bolts.

Extreme bolts

1. On the **Bolt dimensions** tab select **Assembly** for **Extreme bolts**.
2. Click **Apply** and **Modify**.





The distance between the extreme bolts appears.

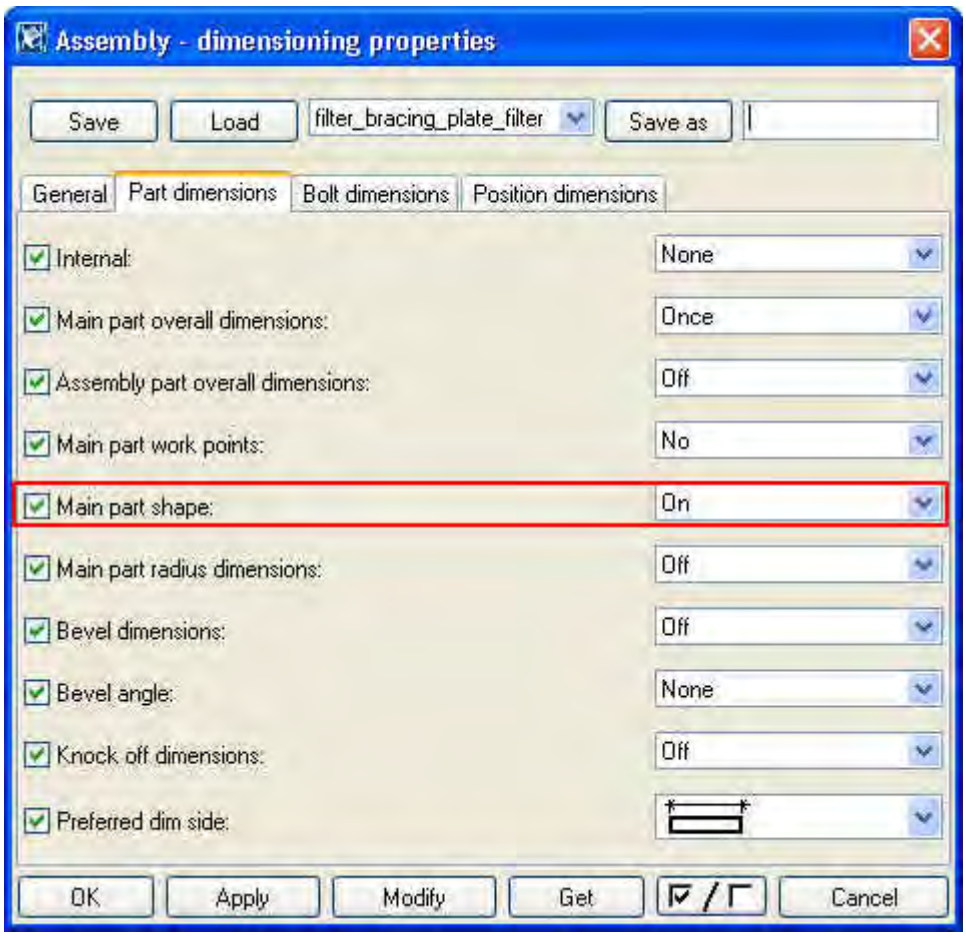
We then continue by dimensioning the main part cuts

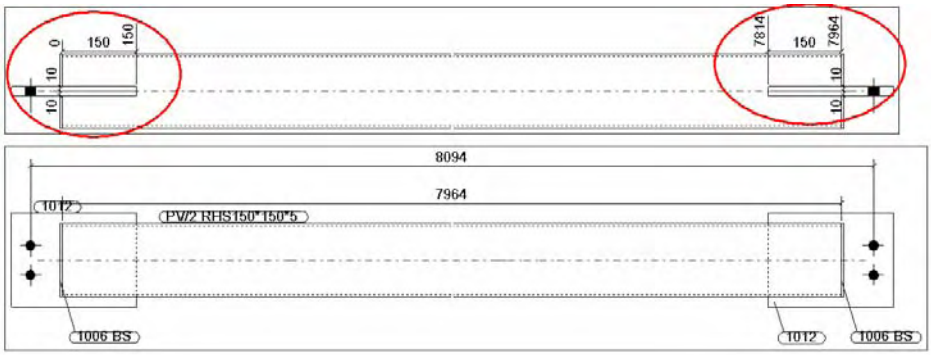
Main part shape

To dimension the cuts in the main part:

1. On the **Part dimensions** tab turn **Main part shape:** to **On**.
2. Click **Apply** and then **Modify**.

Help: Drawing > Dimensioning > Dimension planes

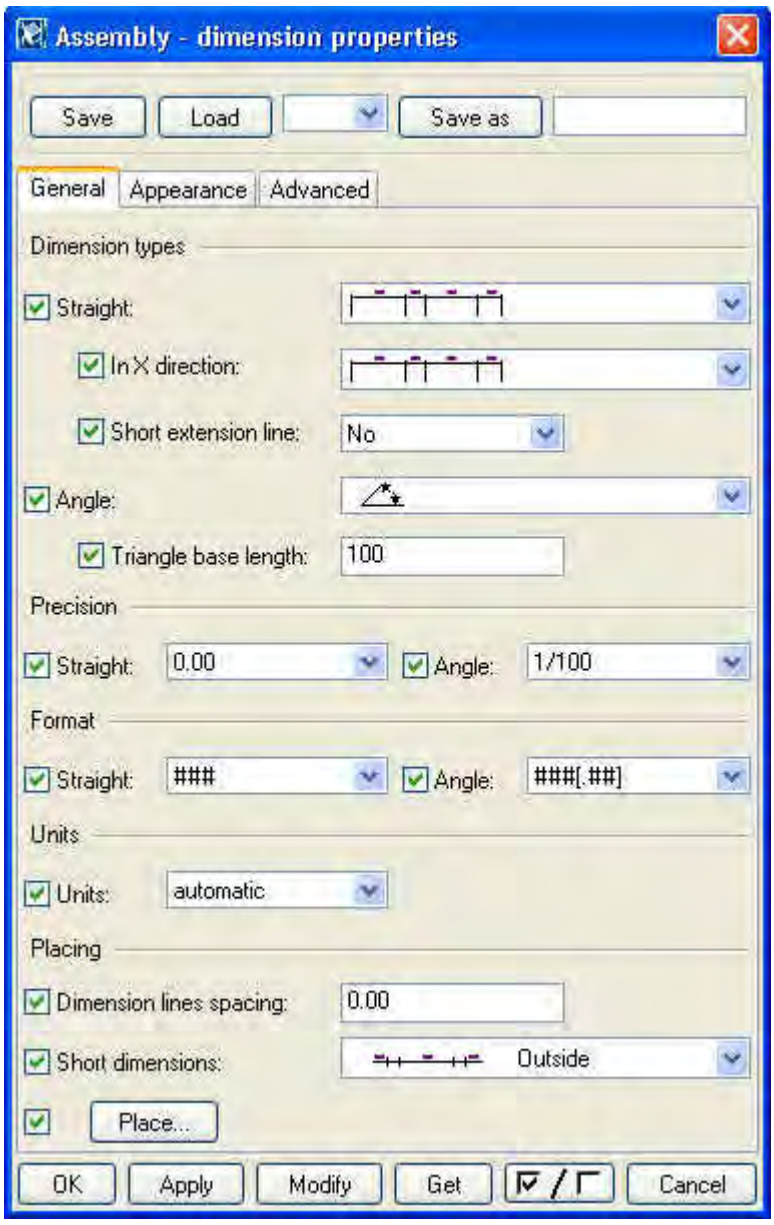


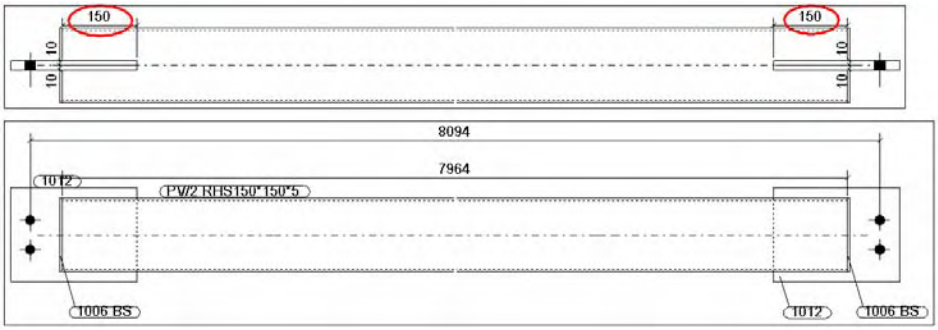


The dimensions of the cuts appear in absolute dimensions. This is due to the dimension type defined in the **Assembly – Dimension properties** tab. We will change the dimension type to relative.

Change dimension type from absolute to relative

1. In the **Assembly drawing properties** dialog box click **Dimension…**
2. Change the **Dimension types / In X direction** to relative (see fig below).
3. Click **Apply** and then **Modify**.





The dimensions of main part cuts now appear as relative dimensions.

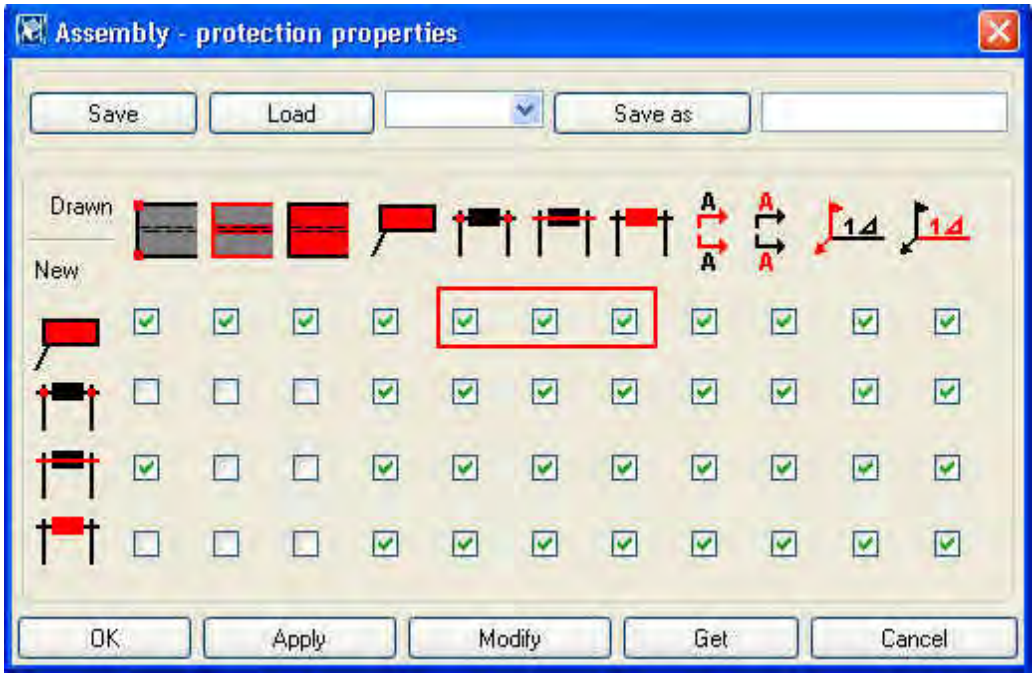
By commenting environment variable, XS_NO_RELATIVE_SHAPE_DIMENSIONS, out the shape dimensions would always be relative despite the option chosen in the **Dimension properties** dialog box.

Protection

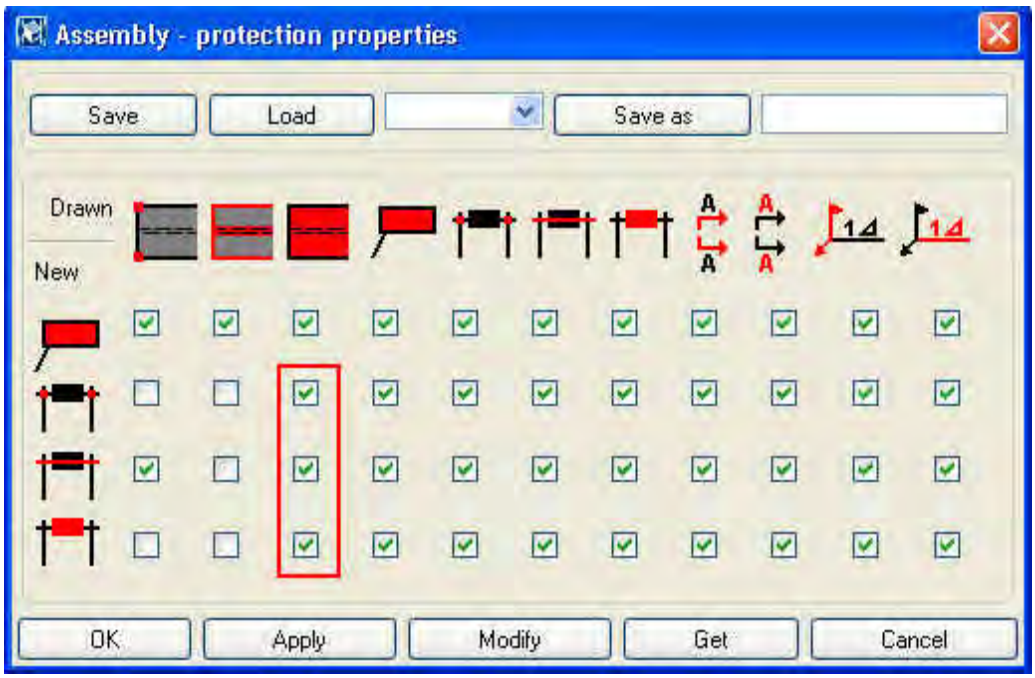
You can protect areas in drawings to prevent text or dimensions being placed there. This way you can e.g. prevent the part mark (1012 in the fig. above) overlapping with the dimension line.

In cases where Tekla Structures can not find a free place for an object the objects will overlap with each others despite the switches in the protection dialog box.

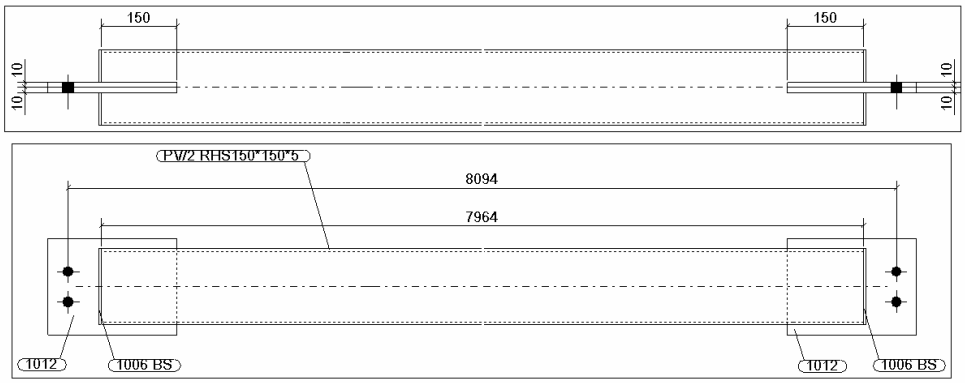
1. In the **Assembly drawing properties** dialog box click **Protection.**
2. Select the checkboxes shown below. These options define that Part marks may not overlap dimension lines.



3. Select the checkboxes shown below. These options define that dimension lines may not overlap parts.



4. Click **Apply** and then **Modify**

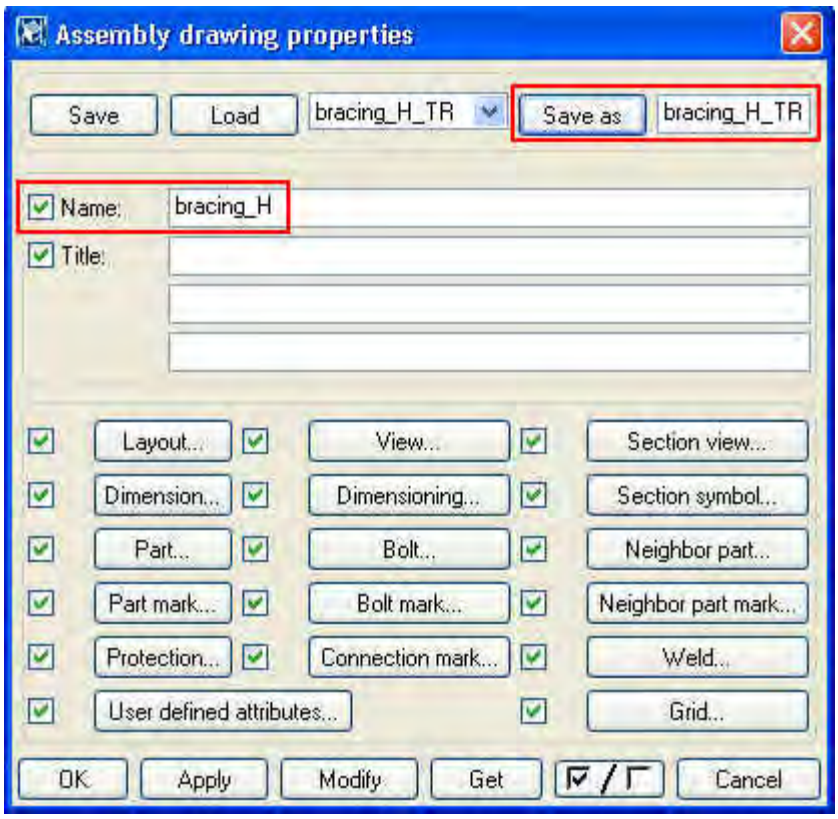


We will now save the properties that we have applied so far for horizontal bracing. We will then continue to edit the drawing little and save the properties for vertical bracing.

For vertical bracing we want to see the secondary part bolt internal dimensions and change the part mark frame little.

Save as properties for bracing_H

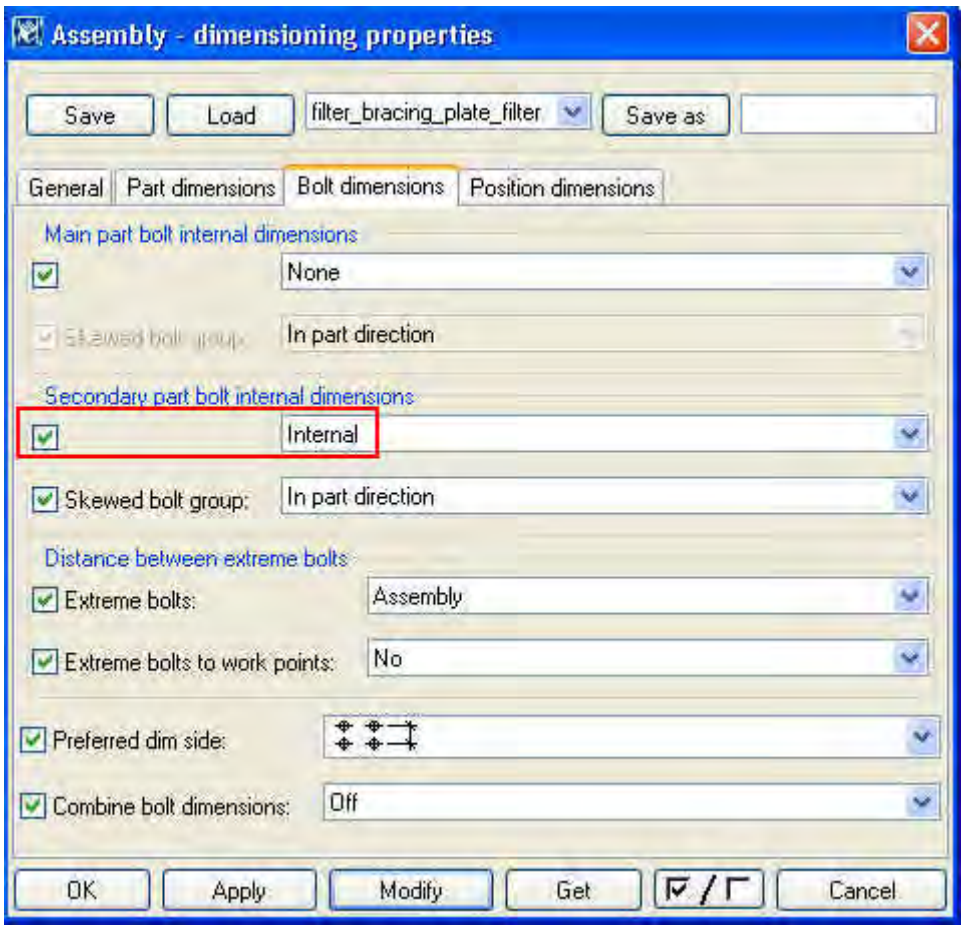
1. Type **bracing_H** in the **Name:** field of **Assembly drawing properties** dialog box.
2. Type **bracing_H_TR** in the **Save as** field, click the **Save as** button



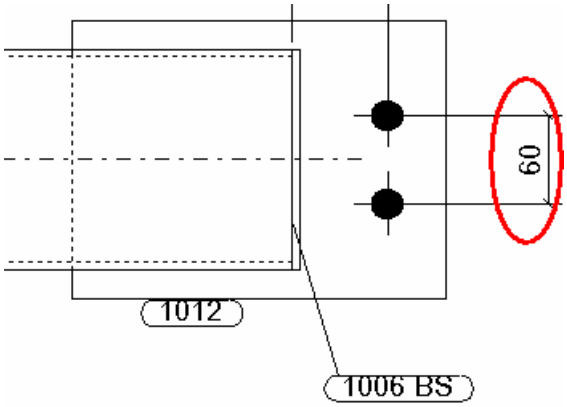
We will now add bolt dimensions to the vertical bracing.

Secondary part bolt internal

1. On the **Assembly - Dimensioning properties > Bolt dimensions** tab select **Internal** for **Secondary part bolt internal dimensions**.
2. Click **Apply** and then **Modify**.



The bolt distance dimensions of the gusset plates appear.

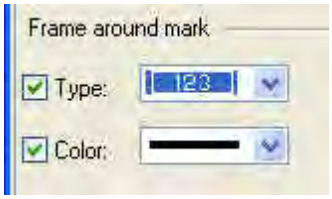


Change the frame of part marks

We will next change the part mark frame to be rectangular.

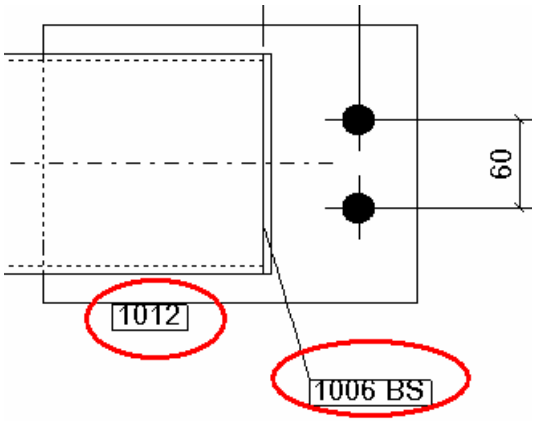
On the **Assembly - part mark properties / General tab**:

1. Change the **Frame around mark** to rectangular.



2. Click **Apply** and then **Modify**

The frames of part mark change to rectangular.



Save as properties for bracing_V

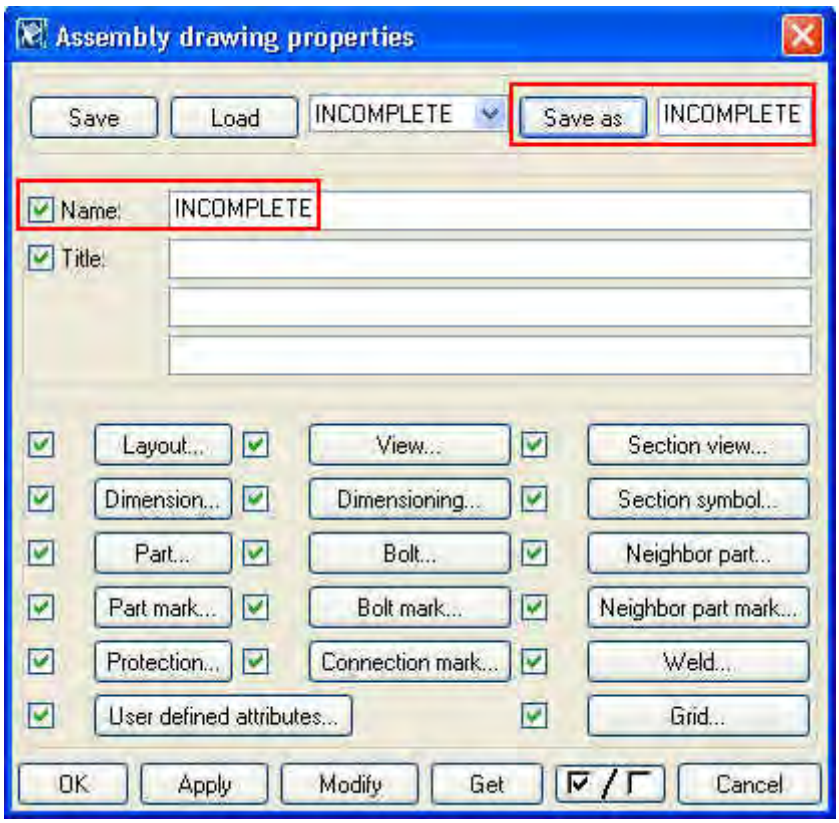
1. Type **bracing_V** in the **Name:** field of **Assembly drawing properties** dialog box.
2. Type **bracing_V_TR** in the **Save as** field, click the **Save as** button

11.4 Create New Drawing Wizard

We will next create a new drawing wizard that matches the select filters created earlier and the available drawing properties. We will add two new drawing requests to create horizontal and vertical braces and use the new properties defined for them. We will also save the properties for those parts that don't match our wizard to be shown as **INCOMPLETE** in the drawing list.

Create properties INCOMPLETE

1. Load **standard** drawing properties.
2. Edit the **Name:** field to **INCOMPLETE**.
3. Save as the properties with name **INCOMPLETE**.



We will now use an existing wizard as the basis for constructing our own wizard.

To create a new drawing wizard, we will perform the following steps:

1. Open an existing wizard file.
2. Save it with another name.
3. Modify the new file.
4. Test the functionality of the new wizard.

The following links present more information on customizing the wizard settings and the contents of a wizard file.

Help: System > Files and folders > Customizing Tekla Structures > Creating wizard files

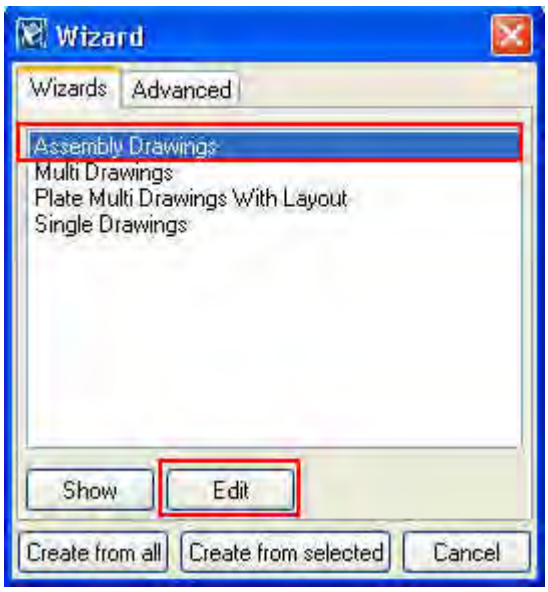
Help: System > Files and folders > Log files > Wizard log

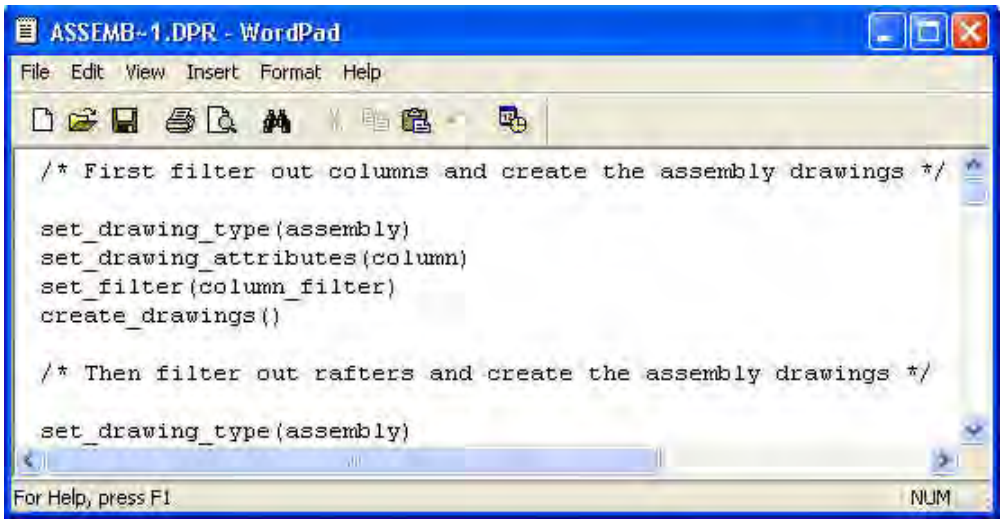
To create the new wizard file:

Open existing wizard file

1. Click the **Wizard** icon to open the **Wizard** dialog box.

2. Select **Assembly Drawings** on the **Wizards** tab.
3. Click **Edit** to open the wizard file in a text editor.





Save wizard file with another name

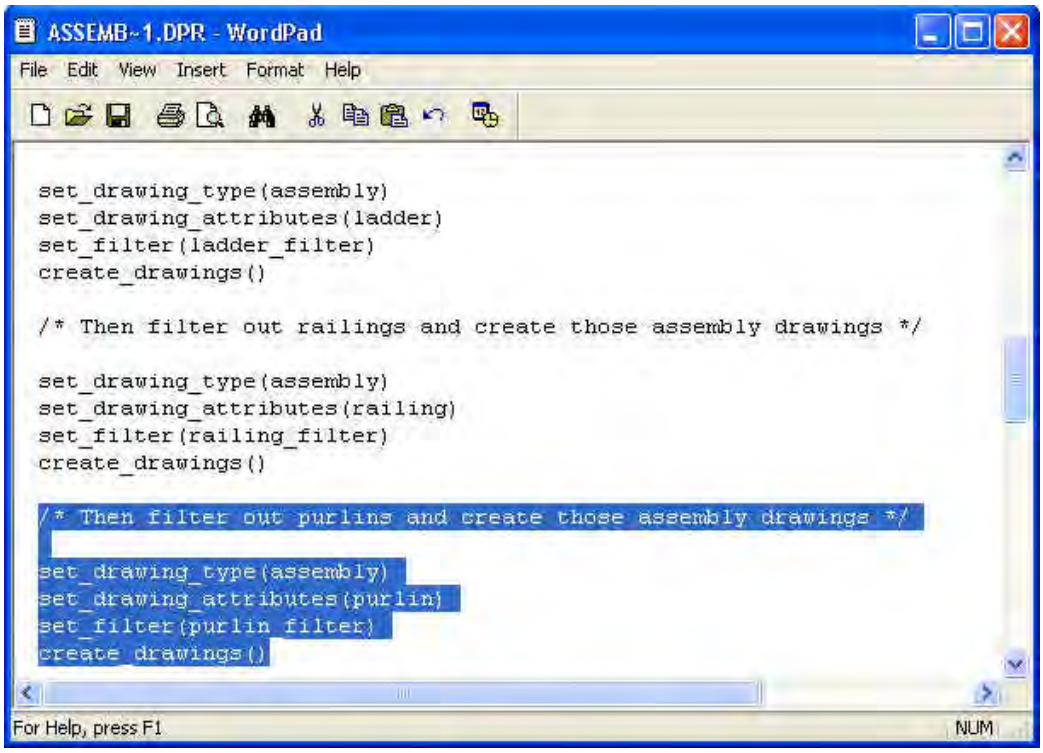
1. Select **Save as…** from the **File** menu of the text editor to save the wizard file with another name.
2. Browse to the attributes folder under your model folder, enter **ASSEMBLY_TR.dproc** as the file name and click **Save**. Note that the file extension has to be **.dproc**.
3. Check that the **ASSEMBLY_TR.dproc** file was created in the **attributes** subfolder by selecting **Open model folder** from the **Tools** menu.

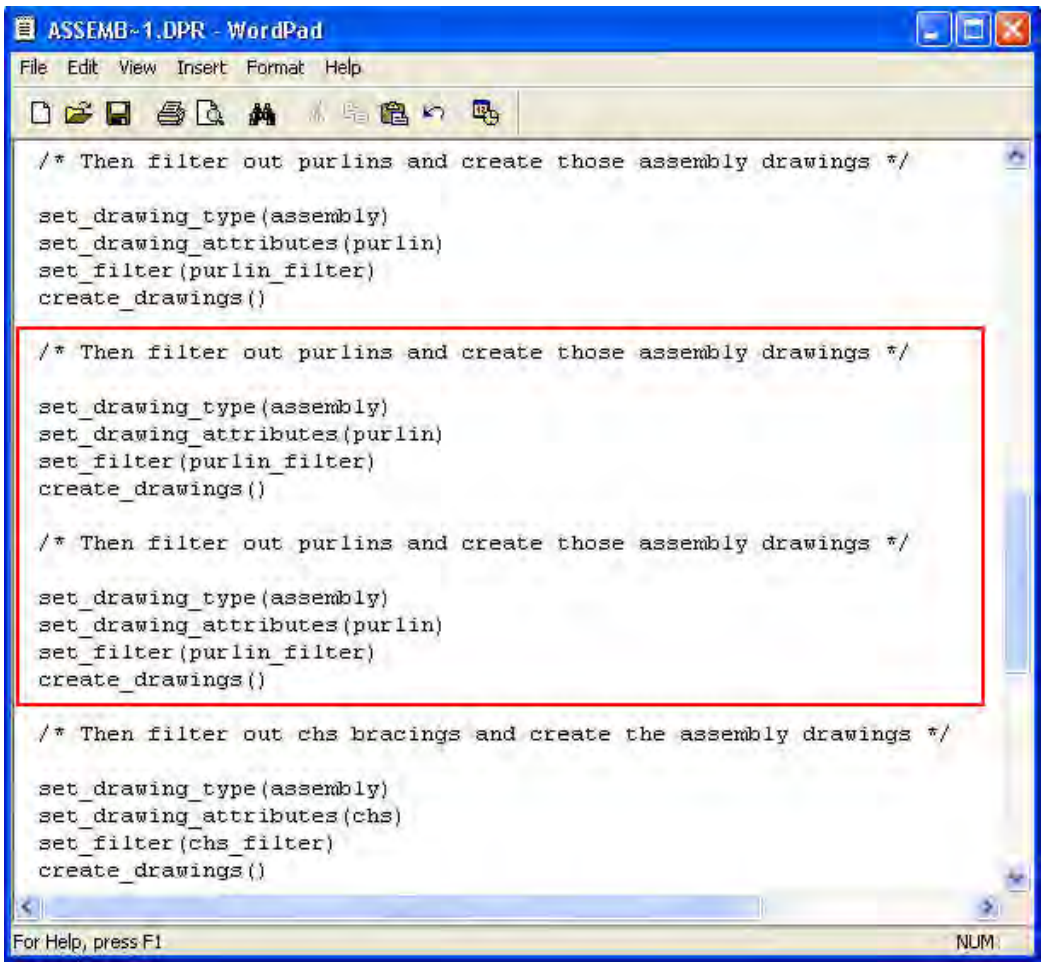
The predefined wizard files are located in the **..\countries*environment*\system** folder. In the system folder the wizard files are available for use in all models. Modified and saved wizard files are saved in the same folder, unless you define another destination folder (such as the model folder).

To have the new wizard file shown in the **Wizard** dialog box, you need to close and reopen the model.

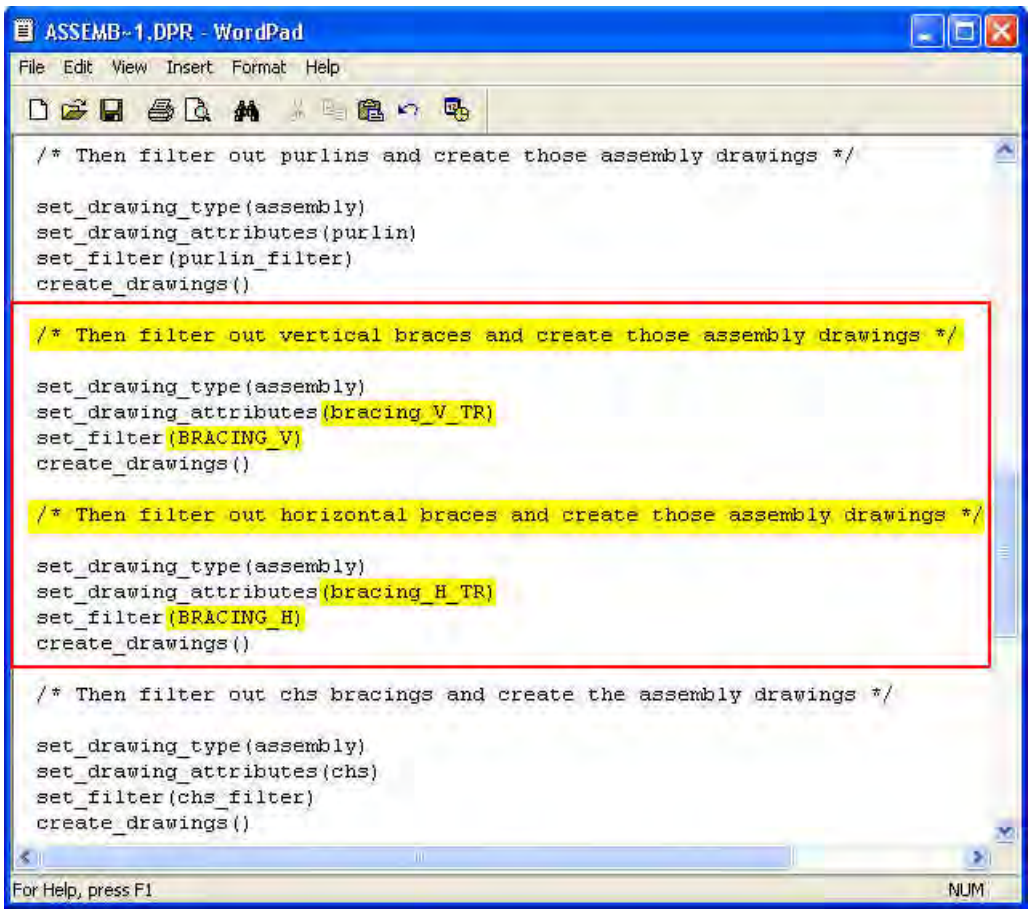
Modify wizard file

1. Open the **Wizard** dialog box.
2. Select **ASSEMBLY_TR** on the **Wizards** tab and click **Edit**.
3. Select an existing drawing request for purlins (shown highlighted below) and copy it two times under the original request.





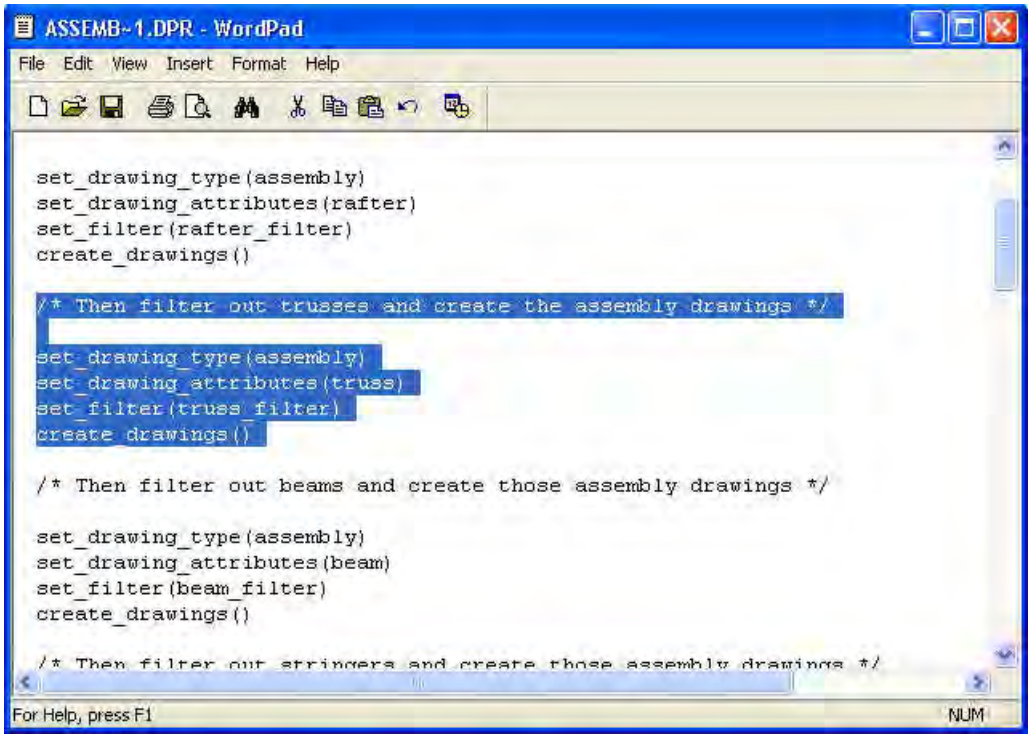
4. Edit the requests that you copied, to match the criteria for vertical and horizontal bracing shown highlighted in yellow below.



Delete requests

5. Delete the requests that don't match our select filters:

- trusses
- ladders

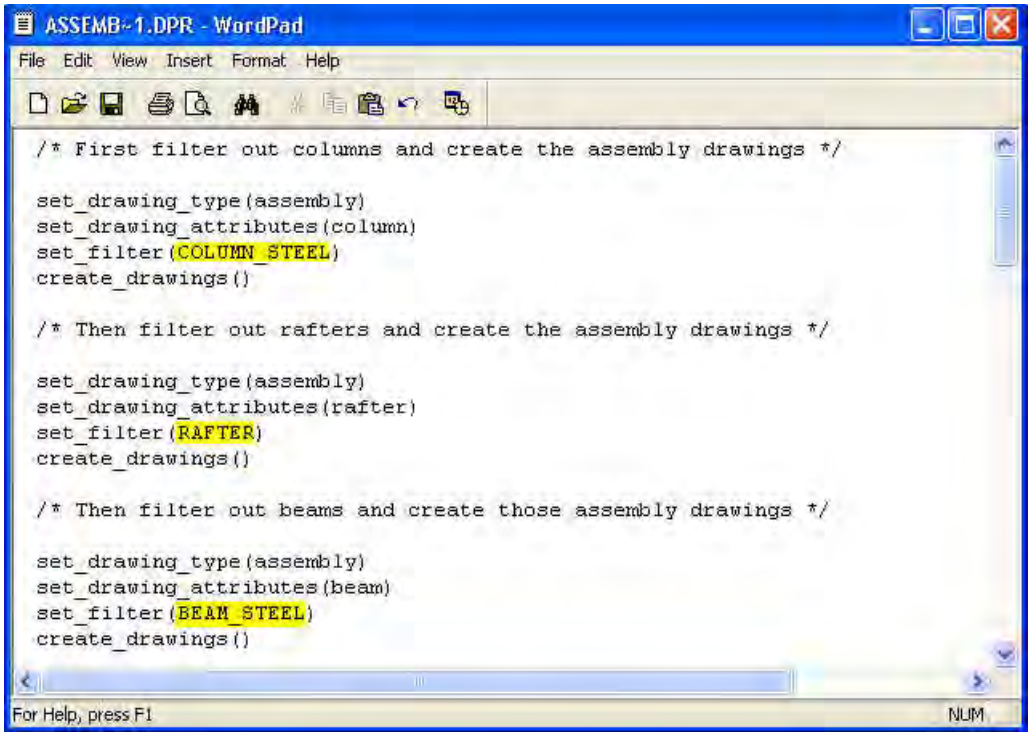


Next we will change all the requests to use the select filters that we defined earlier, for this project.

Change the select filters

6. Change the select filters defined for steel parts in this project:

- column_filter to COLUMN_STEEL
- beam_filter to BEAM_STEEL
- purlin_filter to PURLIN
- etc.



Request for the rest of the assembly drawings

7. Edit the last request to be as shown:

```
/* Create rest of assembly drawings */

set_drawing_type(assembly)
set_drawing_attributes(INCOMPLETE)
set_filter(standard)
create_drawings()
```

Save the wizard file

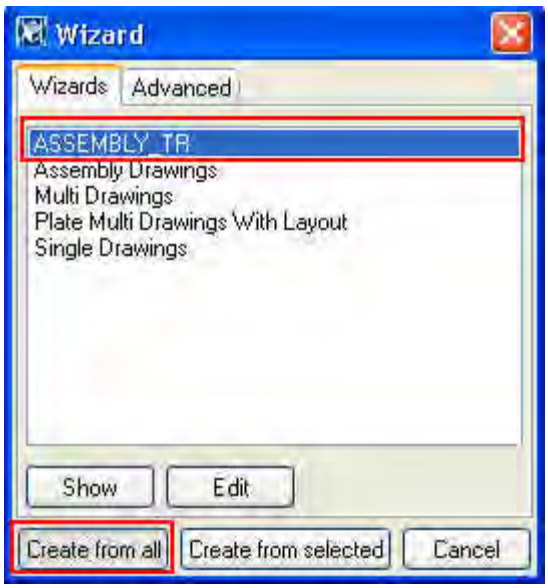
8. Click **Save** to save the changes and to close the text editor.

A wizard file is comprised of several sets of drawing requests. The order of these requests is important, since Tekla Structures creates only one drawing for each selected object. The first request that matches the criteria of select filters is applied.

Now we want to ensure that the new wizard file functions correctly.

Apply own wizard file

1. Delete all of the assembly drawings in the drawing list.
2. Open the **Wizard** dialog box, select the **ASSEMBLY_TR** wizard and click **Create from all.**



3. Check the drawing list to see that correct drawing properties were used:

- The titles of the created assembly drawings
- INCOMPLETE
- bracing_V, bracing_H

11.5 Create Drawings Manually

In Tekla Structures, you can create single-part and assembly drawings by loading and applying predefined drawing properties for layout, dimensions, marks, etc. for selected parts. This process is called the manual creation of drawings, in contrast to the drawing creation with wizards.

The steps for the manual creation of drawings are basically the same as the steps used by the drawing wizard:

1. Select the model objects.
2. Load the predefined drawing properties.
3. Click **Drawing > Assembly drawing** to create assembly drawings or **Drawing > Single-part drawing** to create single-part drawings.

Create Assembly Drawings

As example, we will create assembly drawings from the columns.

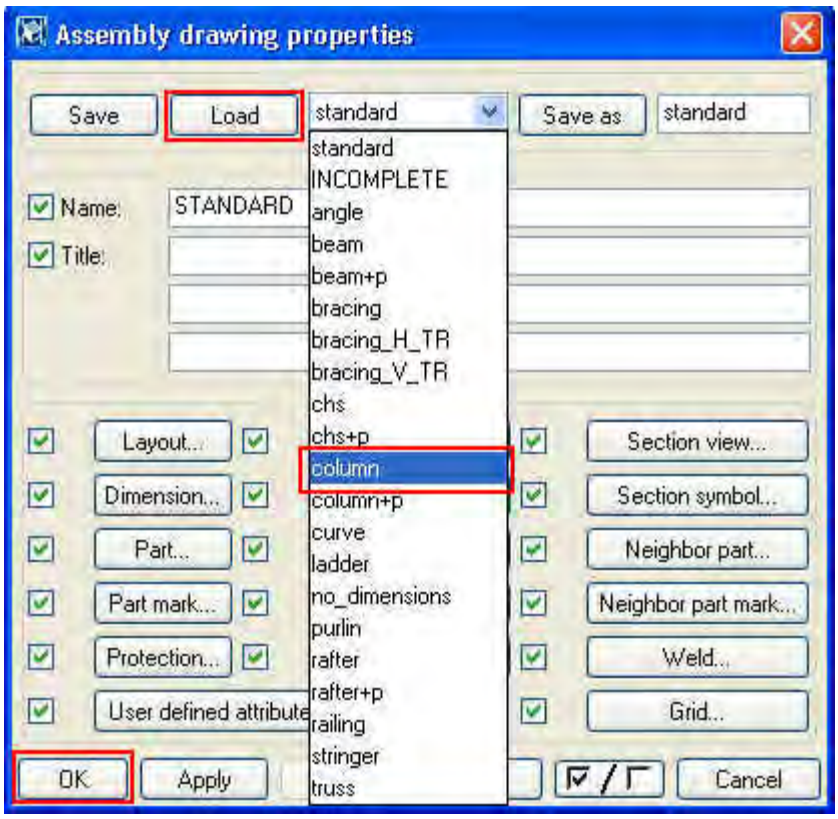
To manually create assembly drawings from all columns:

Delete column assembly drawings

Open the drawing list and delete all assembly drawings with the title **COLUMN**.

Create assembly drawings from columns

1. Select the **column_filter** on the **Select switches** toolbar and select the whole model.
2. Select **Properties > Assembly drawing…** in the menu to open the **Assembly drawing properties** dialog box.
3. Select **column** properties in the drop-down box next to the **Load** button.



4. Click **Load** and **OK**.
5. Select **Drawing > Assembly drawing** in the menu to create the assembly drawings.
6. Open the drawing list and check that the correct drawings were created.

You can also use the view filters in the **View filter** dialog box (opened from the **View properties** dialog box) to help selecting members in the model for drawing creation.

11.6 Edit Drawings Manually

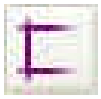
It is possible to manually add and edit additional drawing objects (i.e. dimensions, lines, text, symbols, graphics and marks) in Tekla Structures drawings.

Whenever possible, you should modify drawings by editing only the drawing properties. For example, if you use the drawing properties to modify the drawing, the modifications remain if the drawing must be recreated due to a model change.

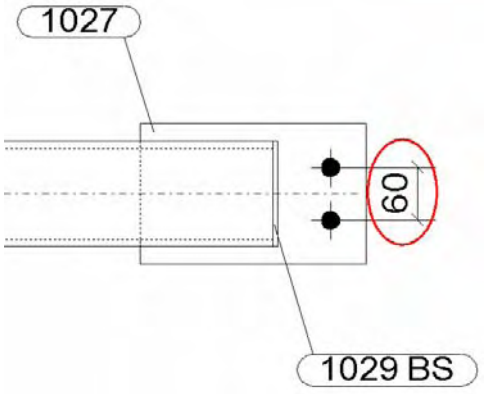
We will now edit a brace_H drawing manually to include same editing that we defined for vertical brace_V drawing properties (the bolt distance dimensions and the rectangular part mark frame).

Add bolt distance dimensions

1. Open the first bracing_H drawing in the drawing list.
2. Click **Create Y dimension** icon.



3. Pick the center points of bolts to be dimensioned.
4. Move the cursor to where you want the dimension to appear and click the middle button.

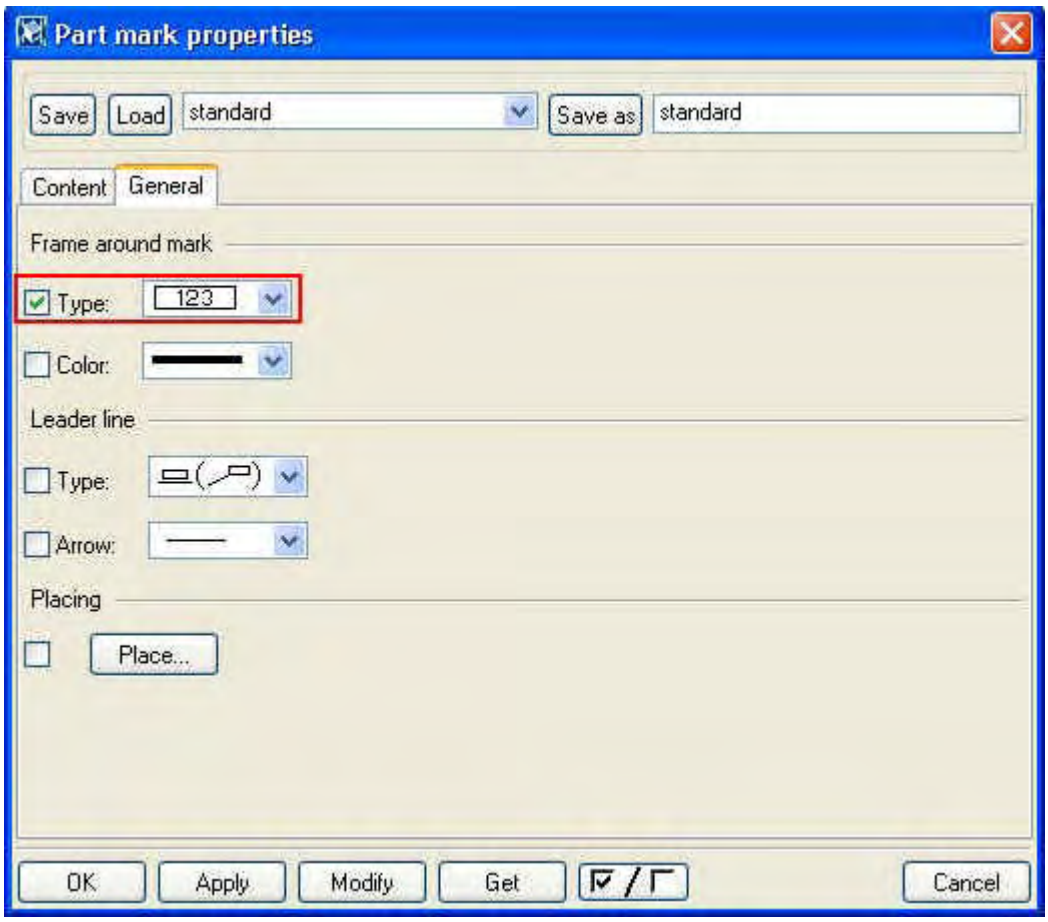


To move the dimension: Select it and drag it to the desired position.

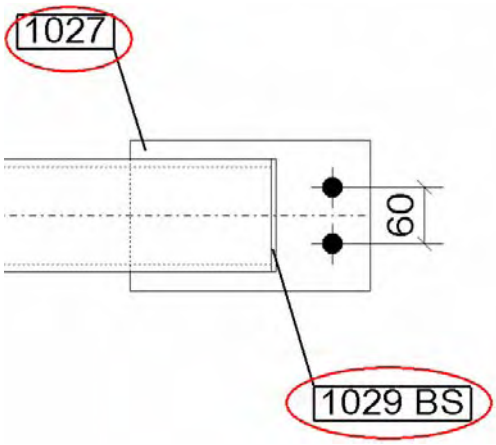
Add the bolt dimension to the other end the same way.

Change the frame of part marks

1. Double-click one of the part marks to open **Part mark properties** dialog box.
2. On **General** tab change the **Frame around mark** to rectangular and tick only this check box.



3. Select all the part marks in the drawing.
4. **Modify**.

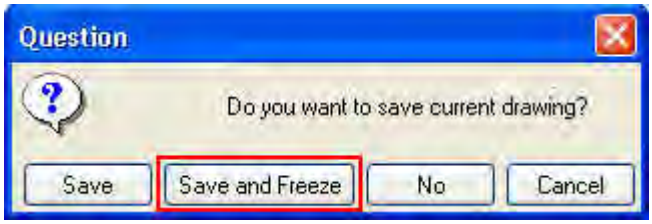


Now all the part marks in the drawing have rectangular frames.

5. Close the drawing, click **Save** in the confirmation dialog box.

Edit the other bracing_H drawing

1. Open the second bracing_H drawing.
2. Add the bolt distance dimensions.
3. Change the frame of part marks to rectangular.
4. Close the drawing, click **Save and Freeze** in the confirmation dialog box



An **F** appears in the drawing list to show this drawing is frozen.

Now the drawings of the horizontal bracing have the same editing as the vertical bracing. The difference is that part of the horizontal brace editing was done manually.

11.7 Updating Assembly and Single-part Drawings

We will now modify our model by changing the bolt spacing of all of the gusset plate connections. Changes in the model will result in some of the drawings will no longer be up-to-date. To be able to open the drawings you will need to run numbering and update them.

We will study how updating effects the edited drawings.

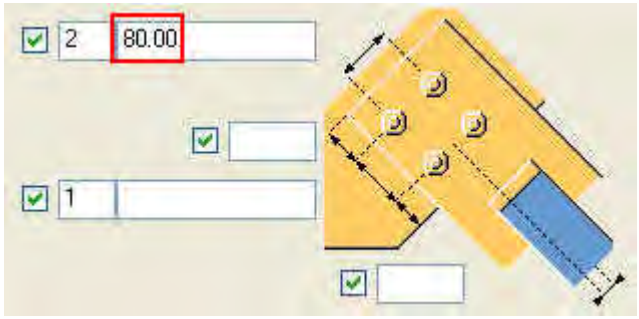
Updating will:

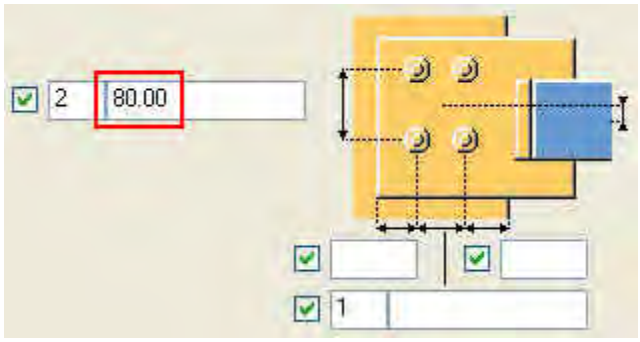
- Regenerate any drawings with a **P** flag that are unfrozen, the updating deletes all the manual editing (added dimensions, texts etc.).
- Update the quantities on a drawing with an **N** flag.
- Switch the **P** flag on a frozen drawing to an *****. This indicates that the picture (the parts and bolts) and the marks of the member have been updated but not the dimensions.

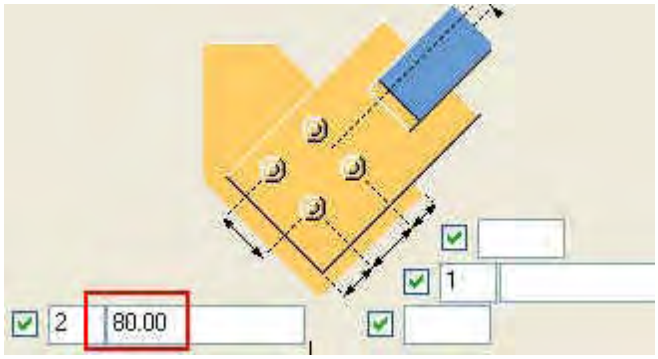
Changes in the Model

Change gusset plate bolt spacing

1. Select all the connections in the model.
2. From the page 4 connection toolbar, double-click the **Tube gusset (20)** icon.
3. Check that the **Ignore other types** is selected in the connection dialog box.
4. In the **Tubebolts1**, **Tubebolts2** and **Tubebolts3** tab pages edit the vertical bolt spacing to **80** as shown below.







5. Click **Modify**.

Now the model has changed and some of the drawings are no longer up to date. To be able to open the drawings you need to run numbering and update them.

Run numbering

Select **Tools > Numbering > Modified**.

Once the numbering is carried out the flags in the **Drawings list** shows all of the affected drawings that need to be updated.

Help: Drawing > Getting started with drawings > Drawing status flags

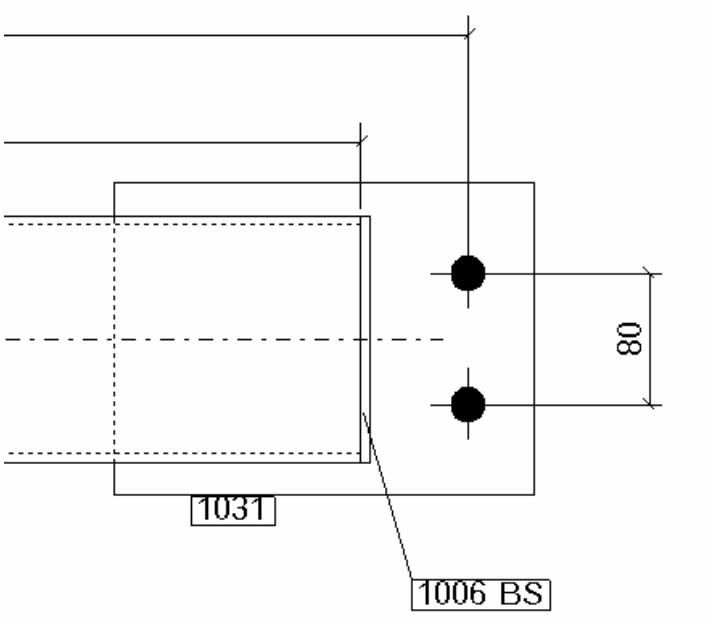
	P	10.02.2004	10.02.2004	410* 287	A	[AH.1]	bracing_H
F	P	10.02.2004	10.02.2004	410* 287	A	[AH.2]	bracing_H

Update vertical brace drawings

The vertical brace drawings were created with only the predefined drawing attributes. They will be complete right after updating, since they are simply recreated using their own attributes.

To update drawings:

1. Select the bracing_V drawings from the list.
2. Click **Update**.
3. Open the drawings to see that they are ok.

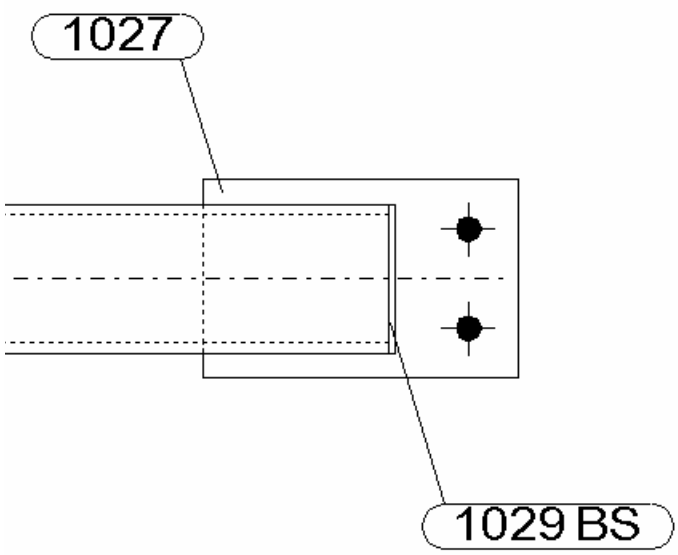


Update not frozen horizontal brace drawing

Since the drawing is not frozen the updating regenerates the drawing but deletes all the manual editing (in this case the added bolt dimensions and part mark frame change).

There is no **Undo** for the updating the drawings command.

1. Select the bracing_H drawing that is not frozen.
2. Click **Update.**
3. Open the drawing



You can see the manually added dimension has disappeared and the part mark frame is back to rounded.

Updating a frozen drawing updates the picture of the member and the marks, but not the dimensions or manual editing. We will need to fix up any dimensions that need updating.

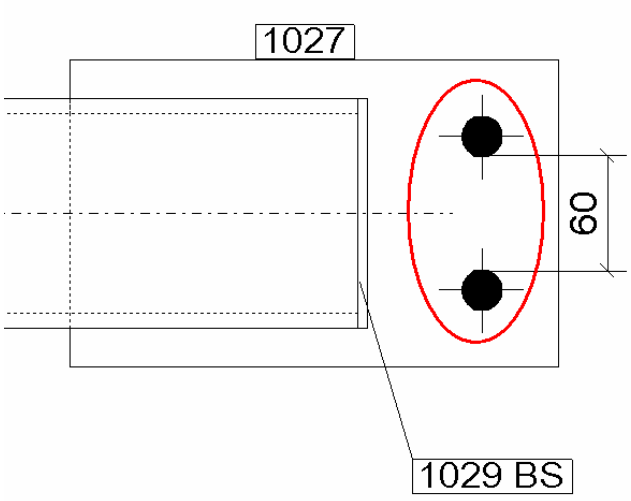
The fact that a lot of time can be spent editing drawings, it is advisable to always freeze manually edited drawings. If major modifications in the model create problems with a drawing you can always recreate the drawing with the automatic settings.

Update frozen horizontal braze drawing

To update frozen drawings:

1. Select the frozen bracing_H drawing from the list.
2. Click **Update**.
3. The P gets replaced by an *.

4. Open drawing with an *.



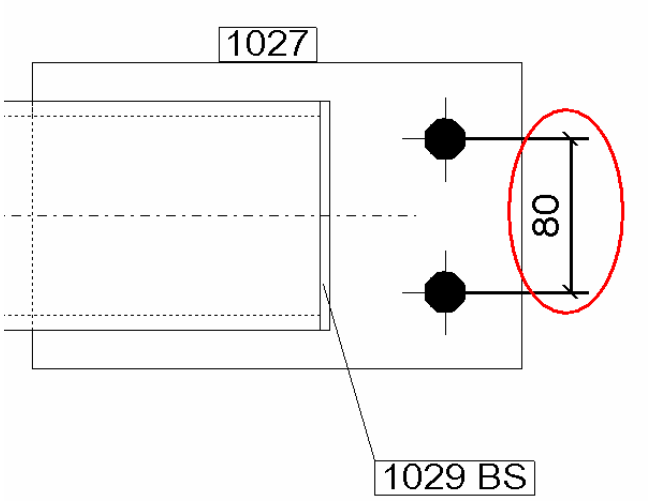
You can see that the bolt dimension still appears but is not correct.

Updating a frozen drawing can lead to a situation of incomplete or erroneous dimensioning. You must manually complete or edit the dimensioning.

5. Select the dimension line and click the **Add / remove dimension point** icon.

6. Pick the correct points (the center points of bolts) to dimension.
7. Click **Add / remove dimension point** icon.
8. Use **Shift** and pick the wrong dimension points to remove them.

The dimension is now correct.



9. Save the drawing and the ***** flag will be removed.

You can filter up-to-date drawings with **Filter > Up to date** button. To get non up to date drawings listed press **Display > Invert** button after **Filter > Up to date**.

It is not possible to open earlier revisions of the drawings. Due to the integration between drawings and the model a drawing that is not up-to-date cannot be opened.

11.8 Cloning Drawings

For more information, see **Help: Drawing > Getting started with drawings > Working with drawings > Cloning drawings**.