Assembly and Single Part Drawings

Tekla Structures 11.0 Basic Training

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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 can, however, 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 the next time you perform 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 > Creating drawings > Using drawing wizards

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

Drawing wizards cannot be used to create General Arrangement (GA) drawings or 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. Select the predefined 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 suitable 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.

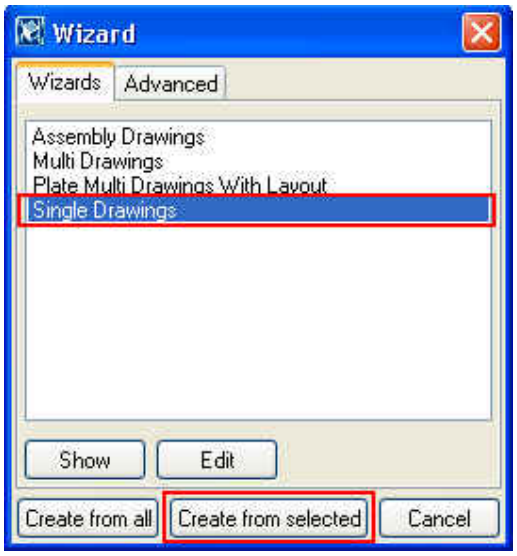
Create single-part drawings of steel beams

1. Open the BasicModelCombined model.
2. Select the BEAM_STEEL filter from the drop-down list of available Select filters

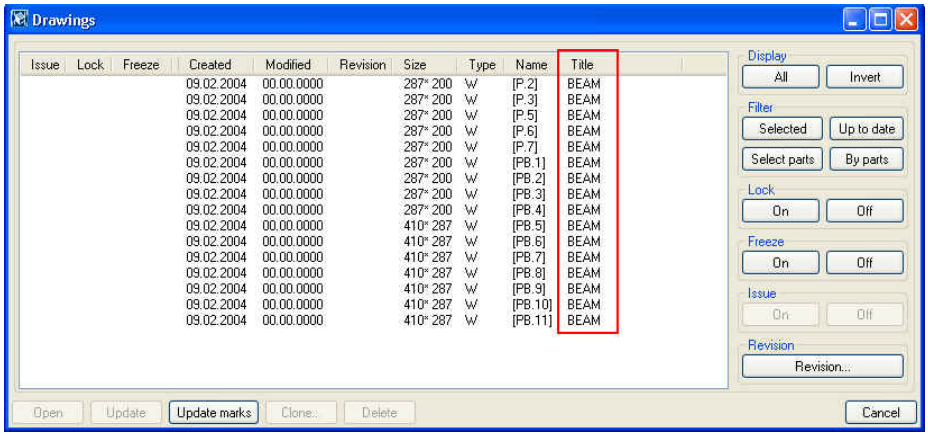
3. Select the whole model with an area selection
4. To see the creation of the drawings, open the drawing list by clicking the Open drawing list icon.

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

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

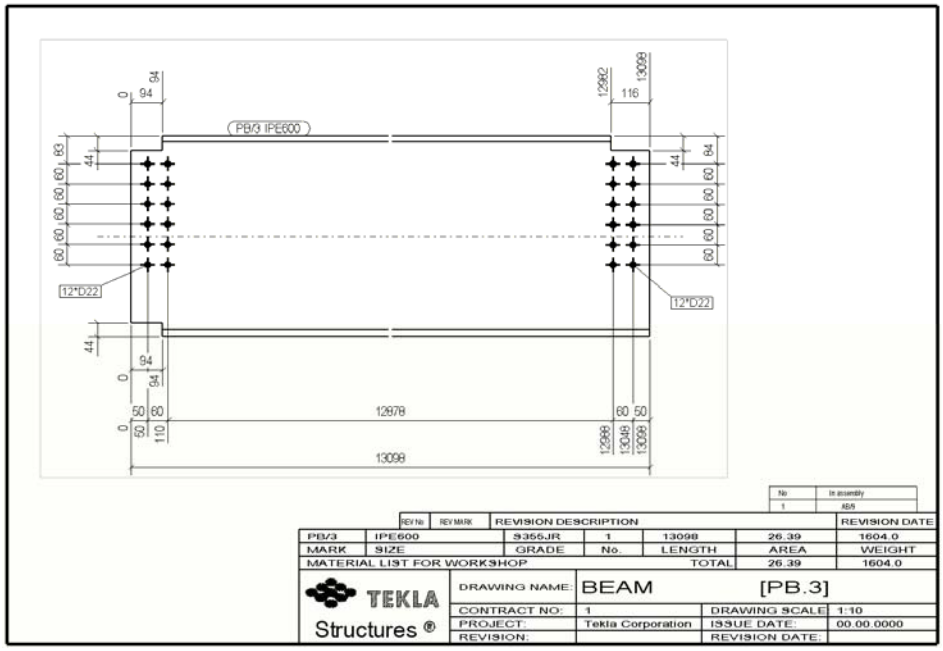


7. Click the Create from selected button.
8. In the drawing list check that single-part drawings with the title BEAM were created.



9. Open a few single-part drawings for viewing

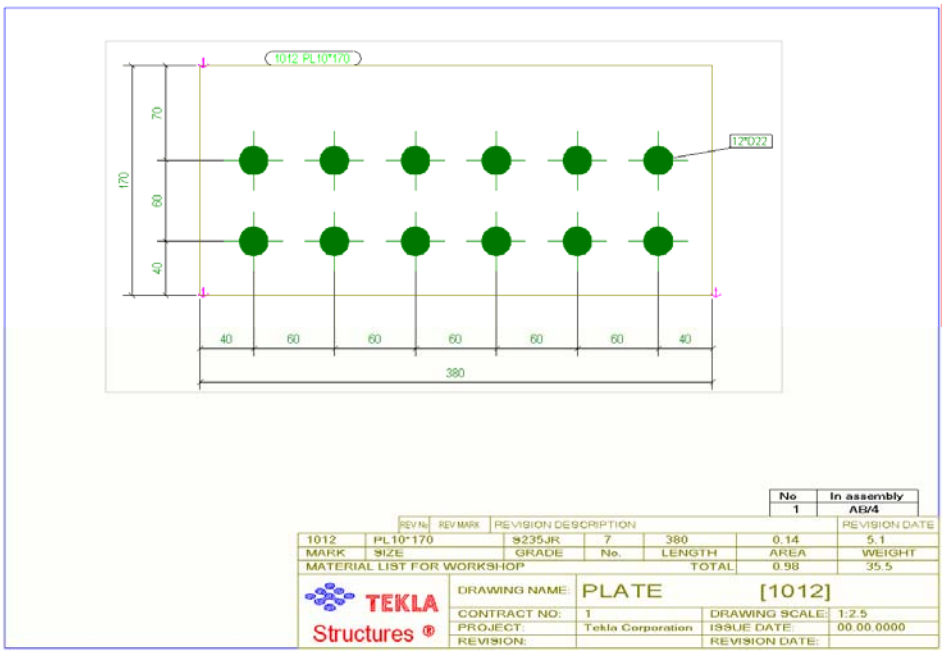
You can view the next drawing on the list by clicking **Next** or using the shortcut **Ctrl + Page Down**



Create single-part drawings from plates

Following the procedure above, now create single part drawings of all 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 on the **Create from selected** button.
5. In the drawings list check that 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 displays a **Cancel** dialog box during the creation of drawings. Click **OK** in the dialog box to stop creating the drawings.

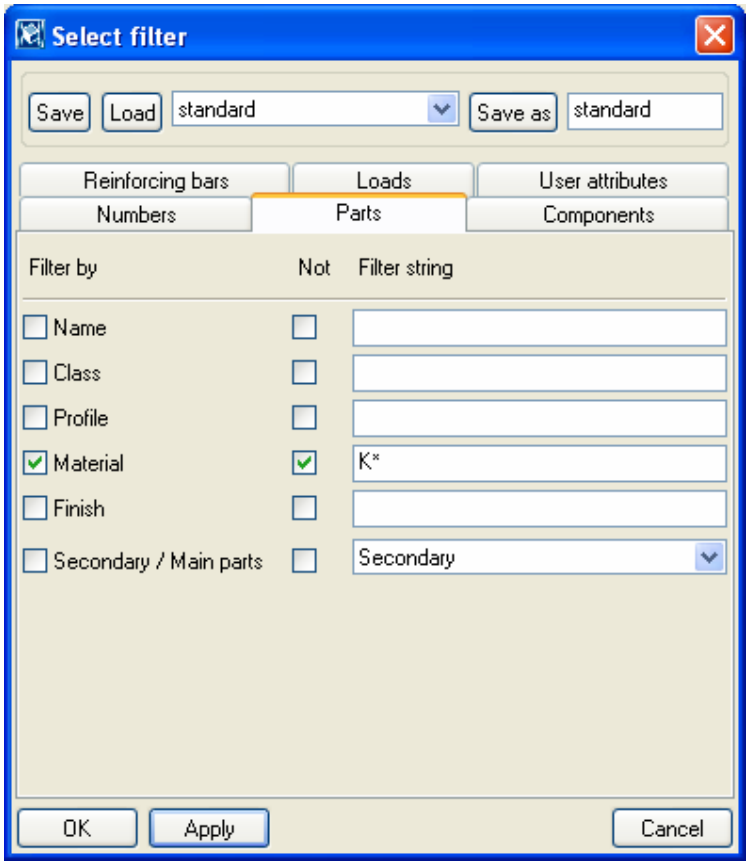
Create assembly drawings of all steel parts

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

To create all assembly drawings at once:

Create assembly drawings

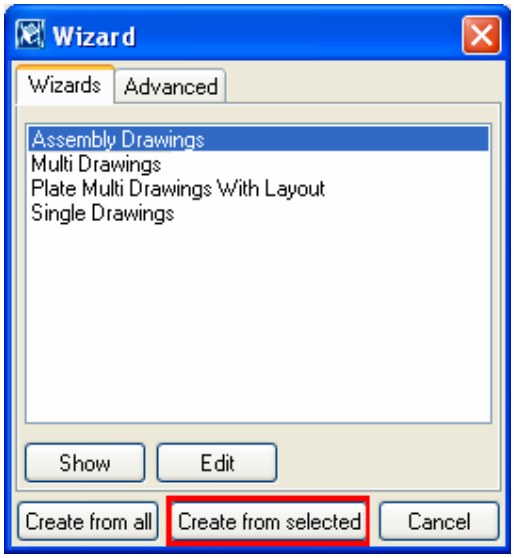
1. Define a select filter to filter away the concrete parts, click OK.



2. Drag an area through the whole model to select all the steel parts
3. Click on the Wizard icon to open the Wizard dialog box

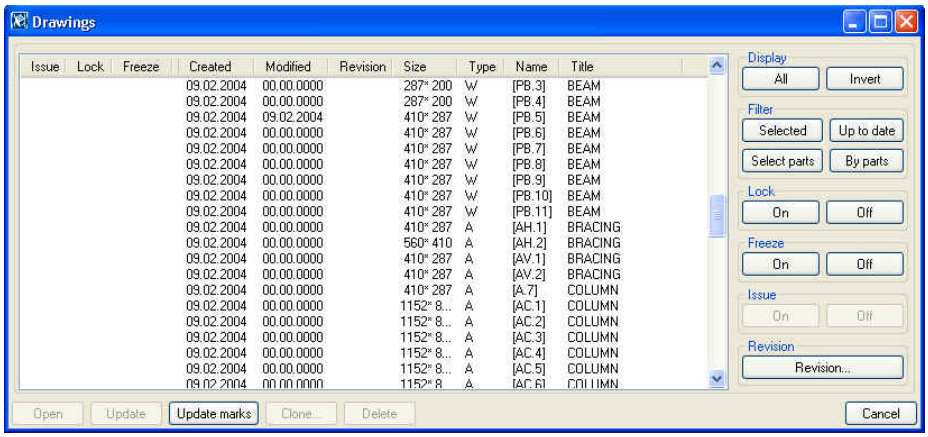
4. Select Assembly Drawings on the Wizards tab

5. Click Create from selected

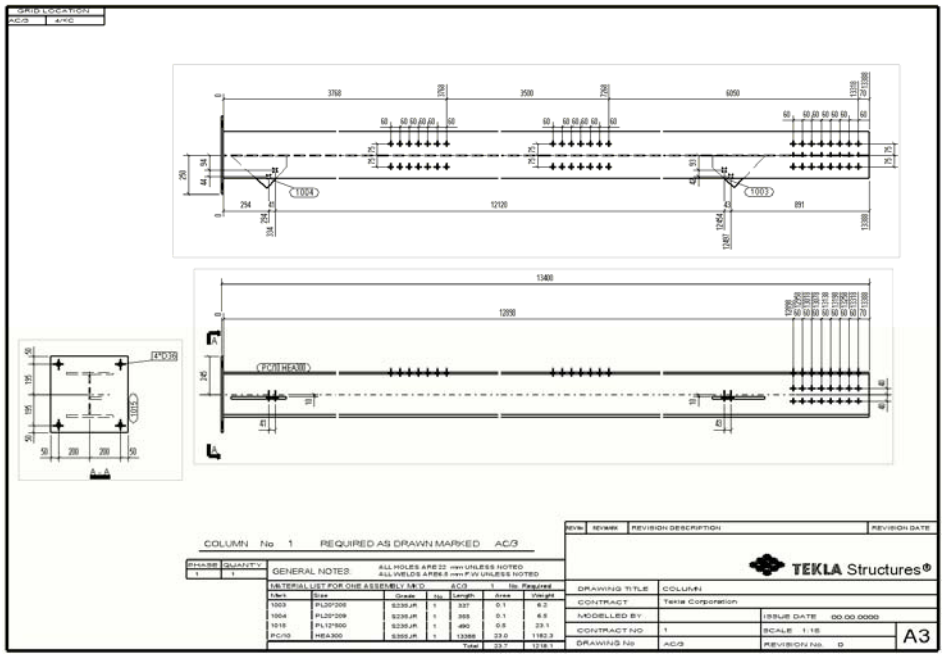


Use the **Create from all** button to create drawings from the whole model at once.

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



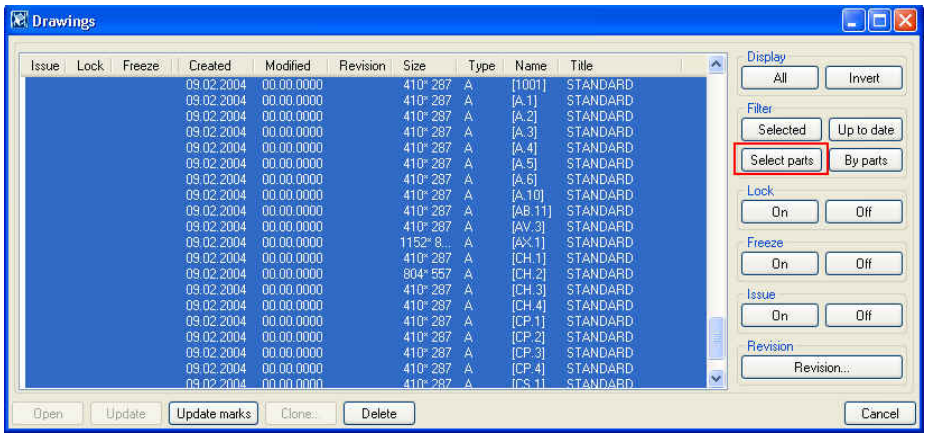
7. Open a few assembly drawings (which are not named STANDARD) for viewing



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

Select parts with drawing title STANDARD

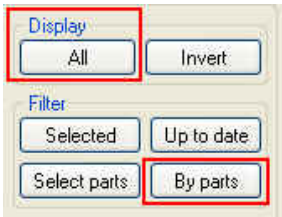
1. Select drawings with the title STANDARD in the drawing list.
2. Click the **Filter - Select parts** button



The parts associated with the selected drawings are now highlighted in the model. You will find, e.g., that objects that don't have a request in the default wizard are highlighted in the model.

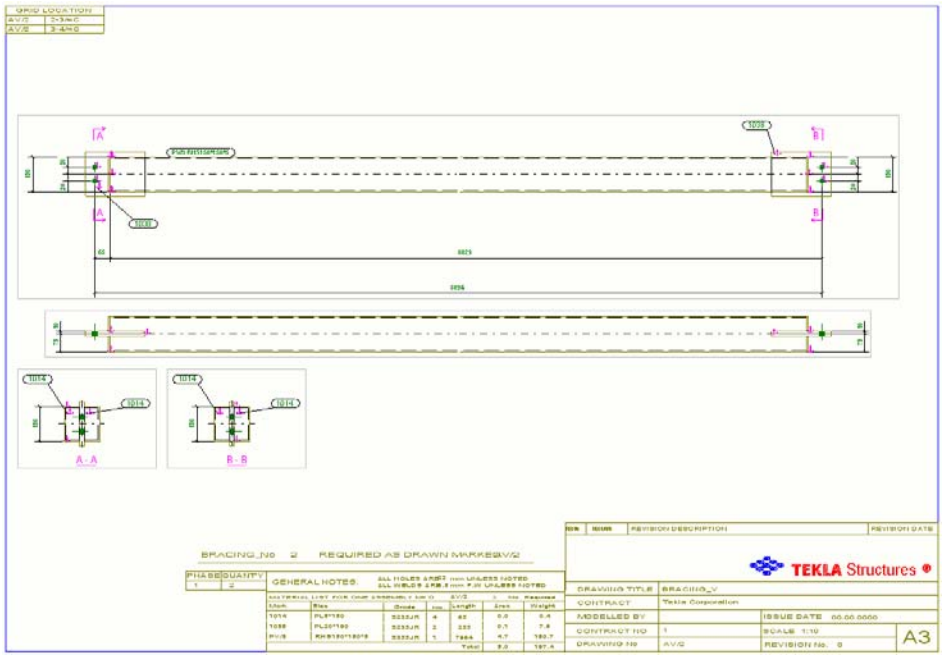
Open brace drawing

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. The brace drawing was created with **bracing** properties.

4. Open the drawing



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.

The editing actions 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

The drawings are created with the applied drawing properties (wizard applies the correct predefined properties automatically to parts).

The quality of the automatically created drawings depends on the drawing properties used

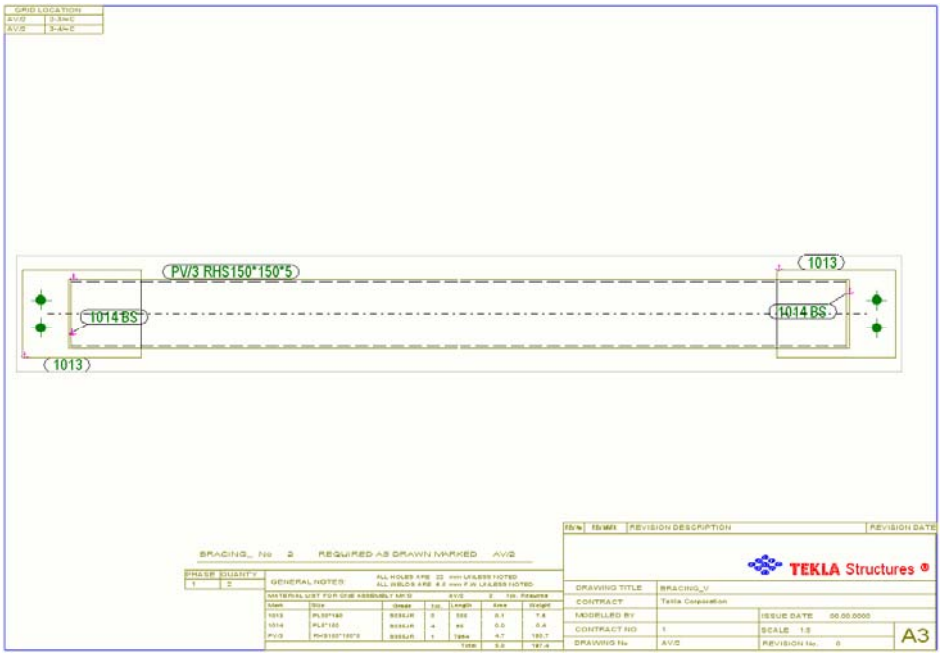
Whenever there is a need to edit the drawings, you should check if the result can be achieved by editing the drawing properties.

As long as you can manage to create complete drawings by using predefined drawing properties the creation / updating of drawings will be automatic

Properties for horizontal bracing

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



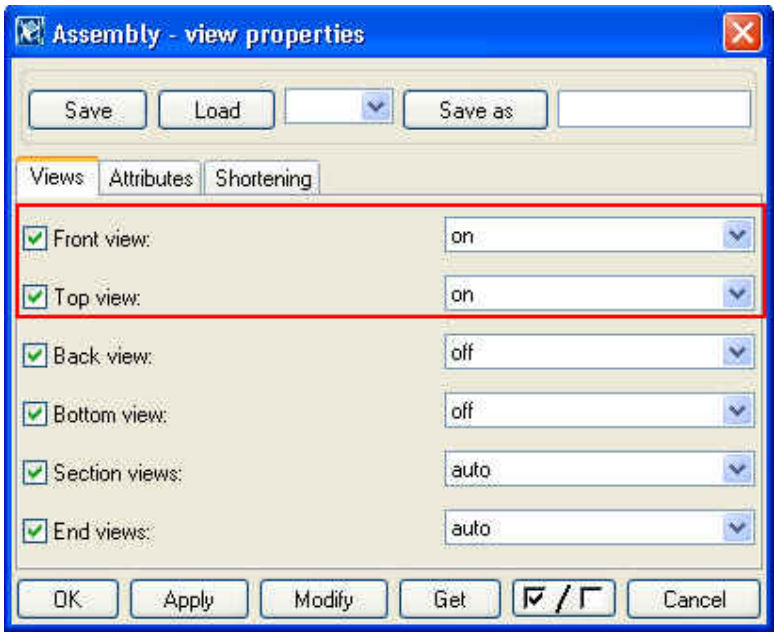
The drawing was regenerated with no_dimension properties. It appears now without any dimensions and only the main view is visible.

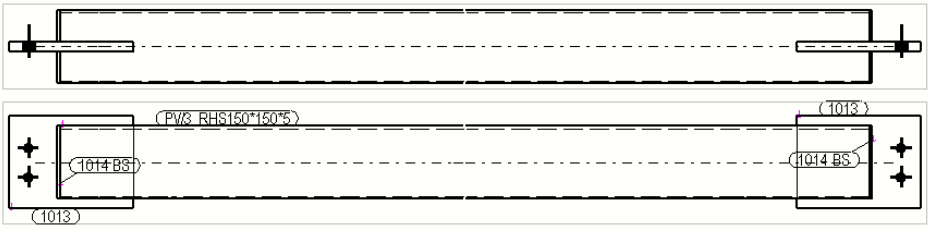
> By default, Tekla Structures creates the additional views only if it is necessary in order to show the dimensions in the drawing.

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 **Modify** and then **OK**

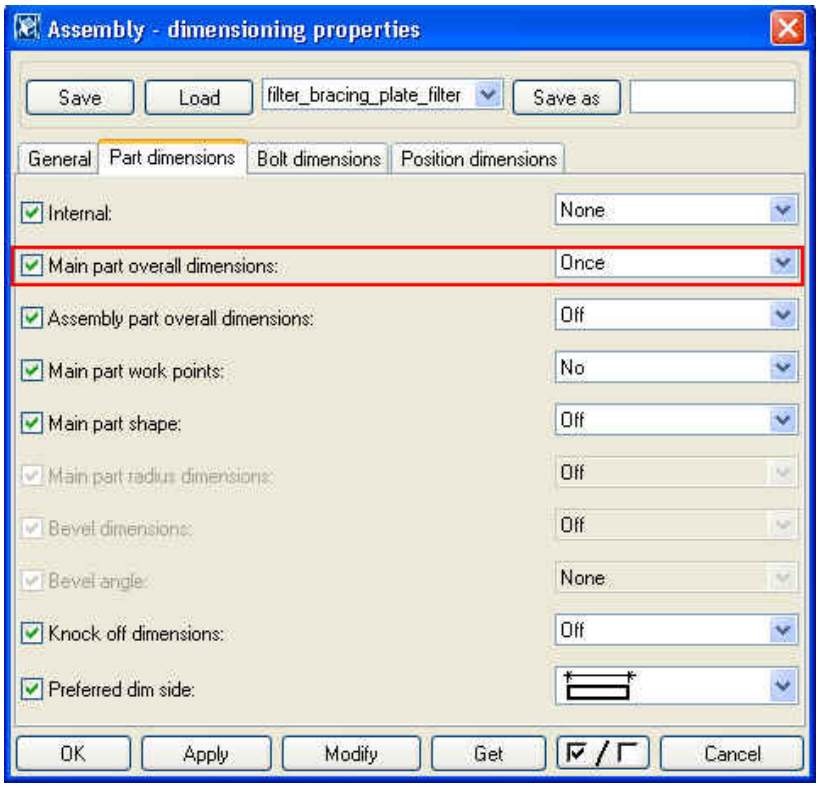


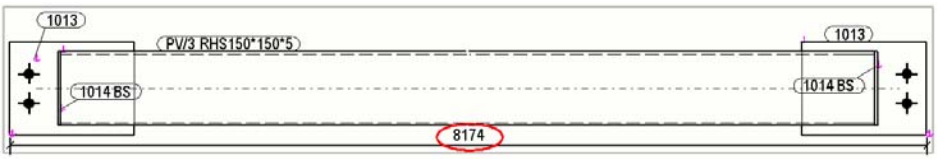


Main part overall dimension

The first dimension we will add is the 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 **Modify** and then **Apply**



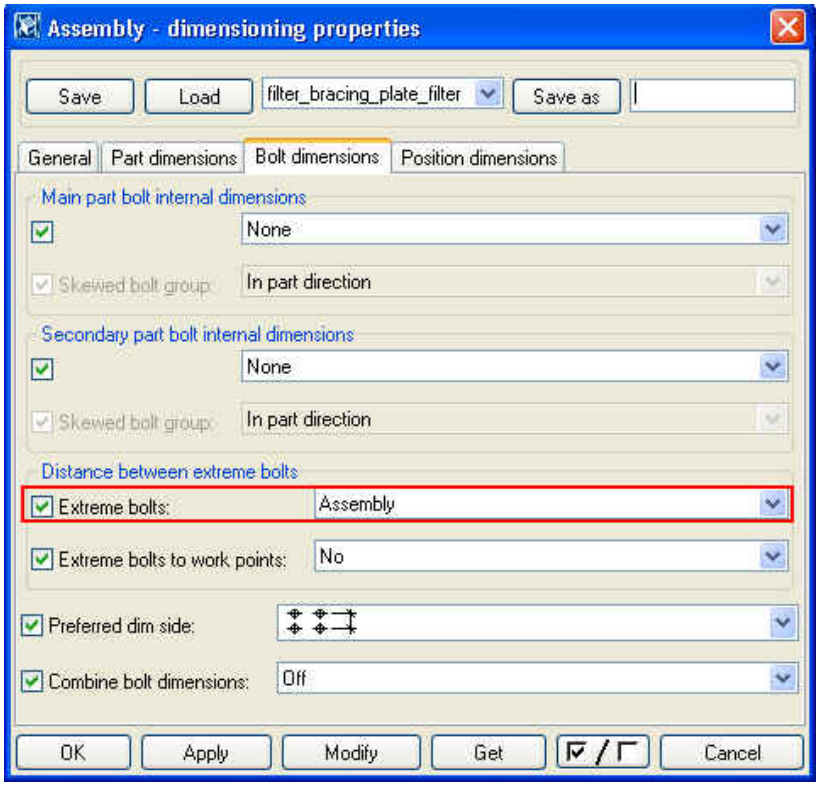


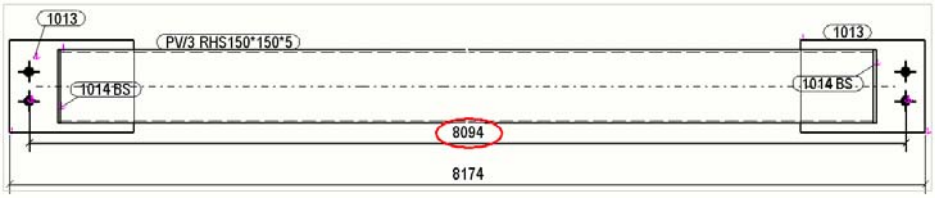
The main part overall dimension appears.

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 **Modify** and then **Apply**





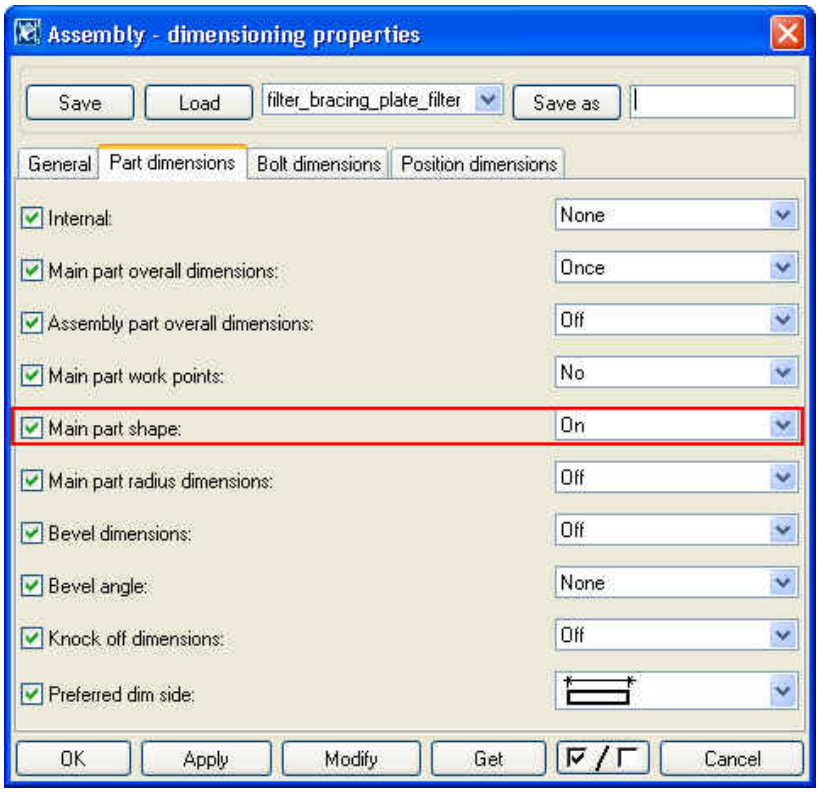
The distance between the extreme bolts appears.

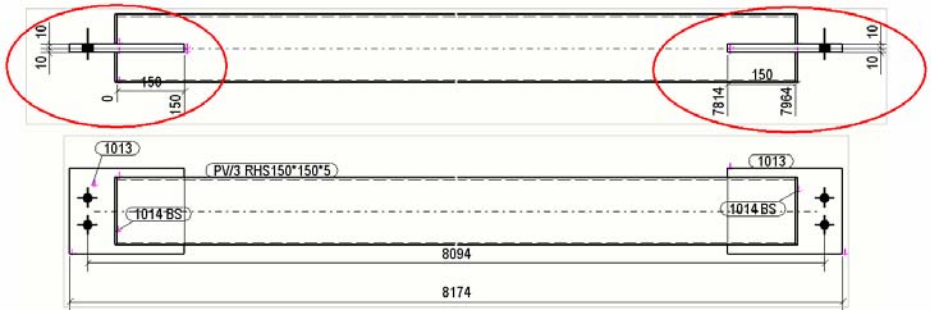
We then continue by dimensioning the main part cuts.

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 **Modify** and then **Apply**



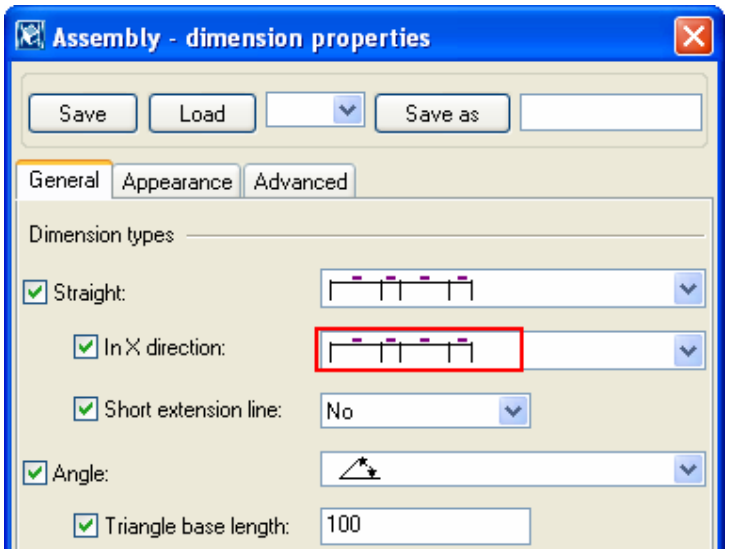


The dimensions of the cuts appear in absolute dimensions. This is due to the dimension type defined in the **Assembly – Dimension properties** tab.

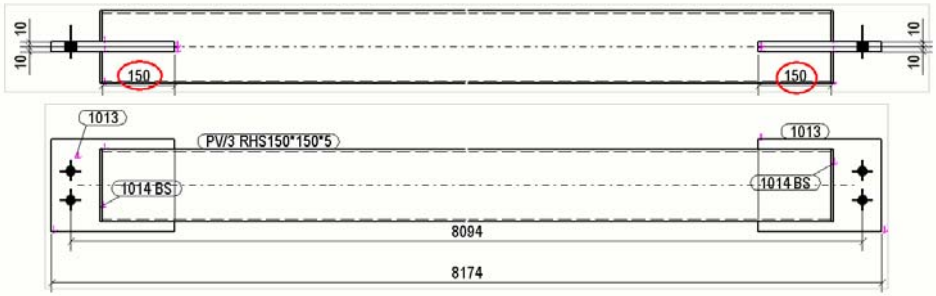
We will next change the dimension type to relative.

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 **Modify** and then **OK**



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

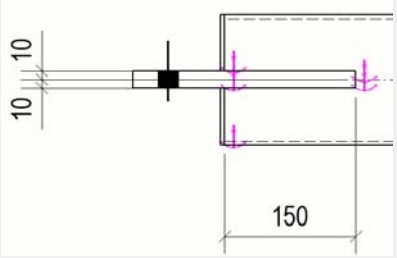
By commenting environment variable, XS_NO_RELATIVE_SHAPE_DIMENSIONS, out the shape dimensions would always be relative despite the option chosen in the **Dimension properties** dialog box.

By editing **dimension planes table** You can define how Tekla Structures dimensions different profiles in drawings. For example, you can have Tekla Structures always dimension rectangular hollow sections to the middle of the profile or to the top.

See Help: Drawing > Dimensioning > Dimension planes

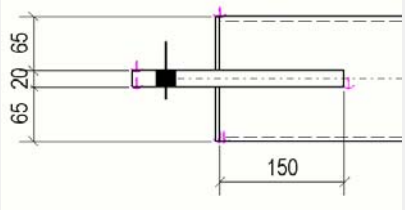
An example of dimensioning according the prof. type 8 (=rectangular hollow sections) is set in dimension plane table by default so that Tekla Structures dimension sections to the middle of the profile

8, -1.0, TRUE*, TRUE, TRUE, TRUE*, TRUE, TRUE



To change Tekla Structures to dimension rectangular hollow sections to the left / top of the profile, set the dimensioning of prof. type 8 as shown below:

8, -1.0, FALSE, TRUE*, TRUE, FALSE, TRUE, TRUE*

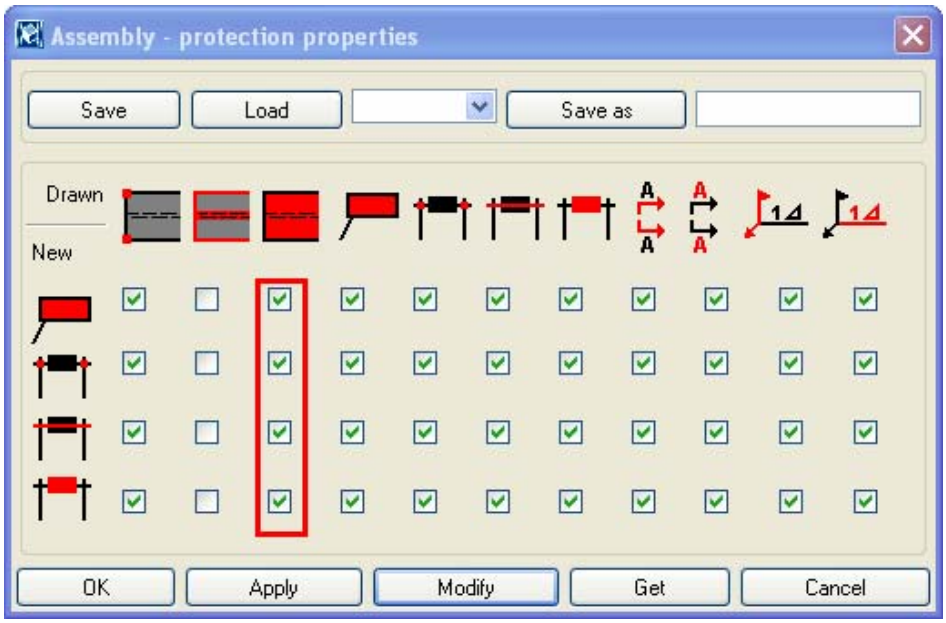


Protection

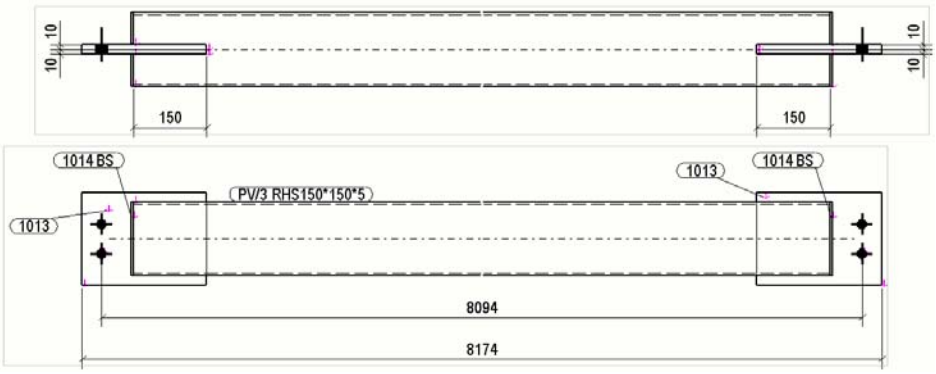
You can protect areas in drawings to prevent text or dimensions being placed there. This way you can e.g. prevent the part mark (1014 in the fig. above) overlapping with the part.

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 dimension lines may not overlap parts.



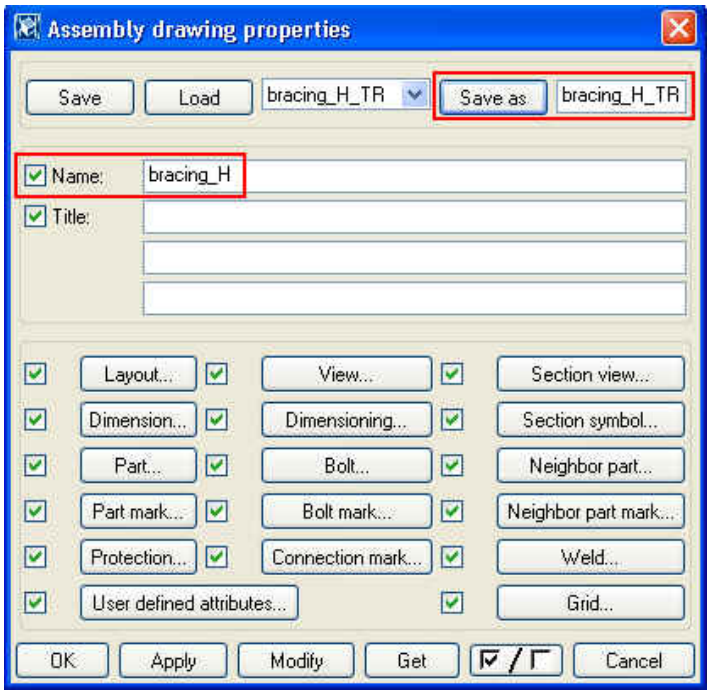
3. Click **Modify** and then **Apply**



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

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



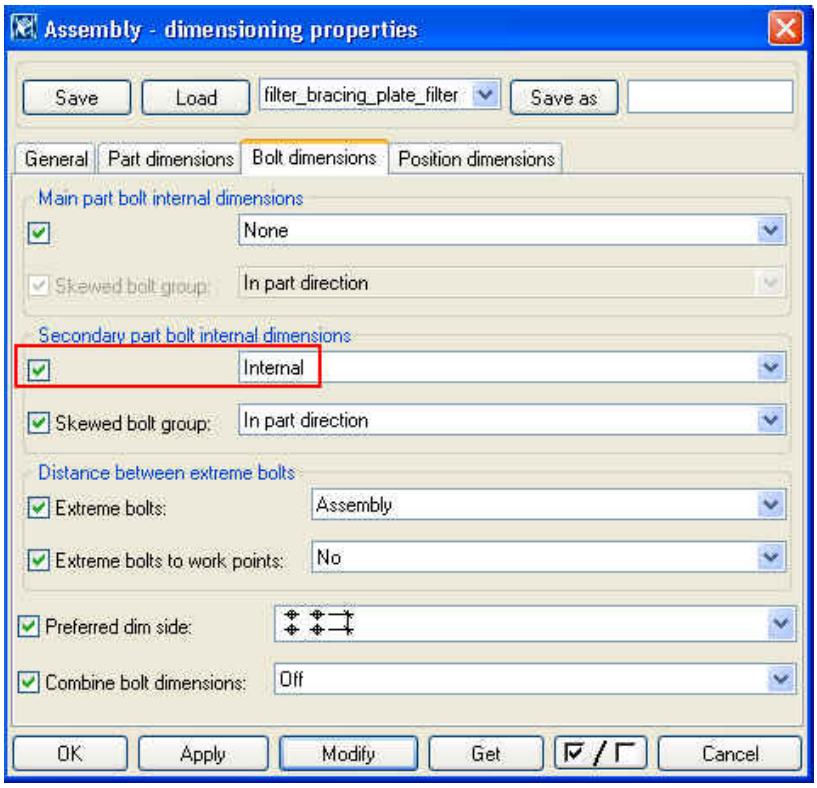
Properties for vertical bracing

For vertical bracing we want to see the secondary part bolt internal dimensions and change the part mark frame a little as well as include the single part views of the plates.

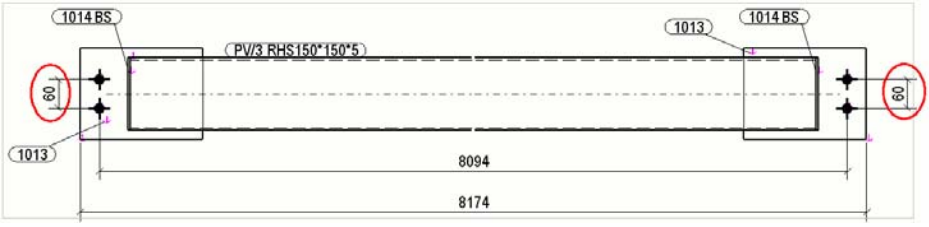
We will first add bolt dimensions to the vertical bracing.

Secondary part bolt internal

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



The bolt distance dimensions of the gusset plates appear.

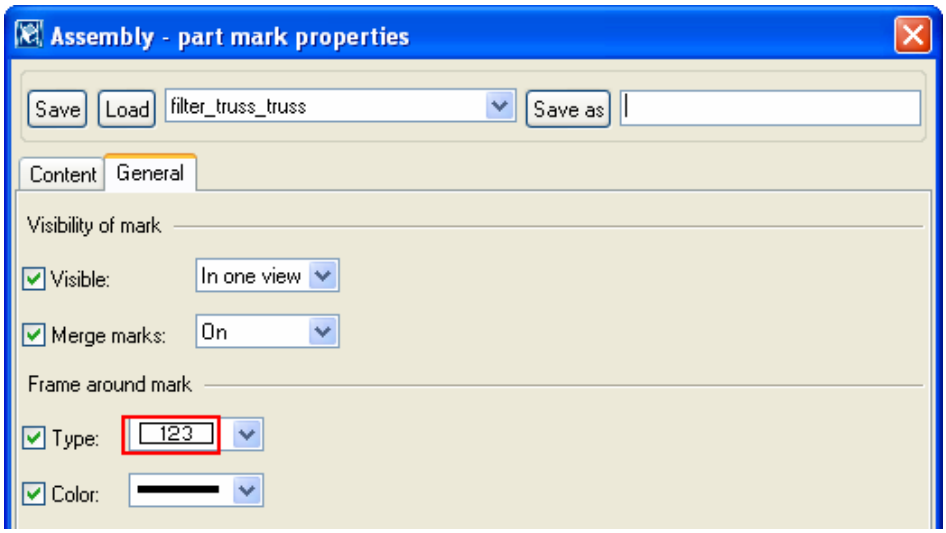


Change the frame of part marks

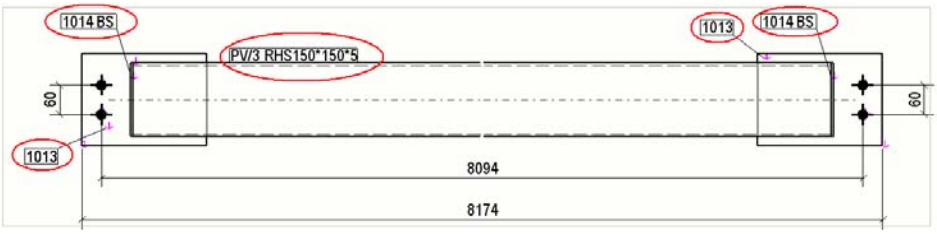
We will next change the part mark frame to be rectangular.

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

1. Change the Frame around mark to rectangular.



2. Click Modify and then Apply

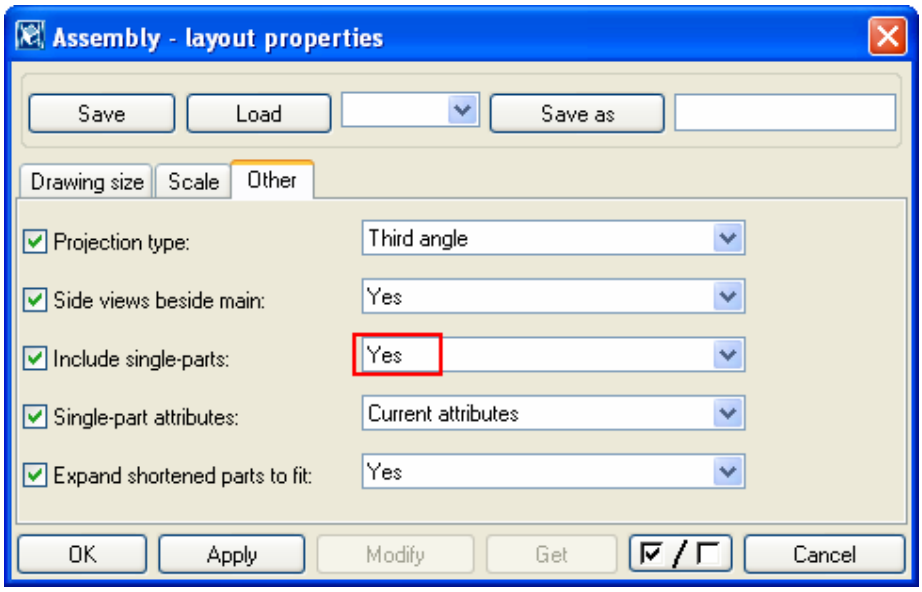


The frames of part marks change to rectangular.

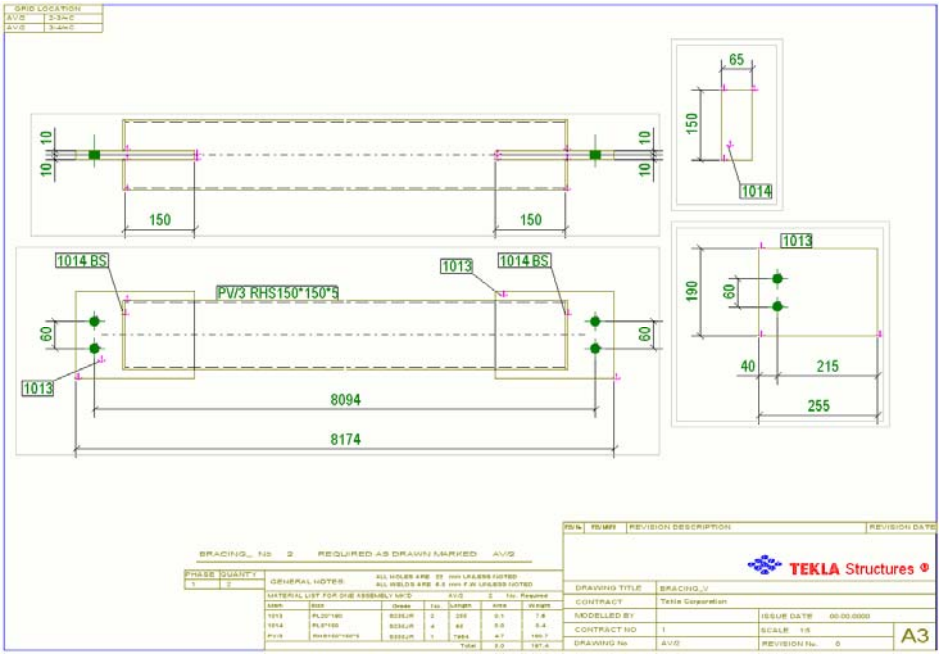
Include single parts to the brace_V drawings

We will now include single-part views of the individual part components that form the assembly.

1. In the Assembly drawing properties dialog box, click **Layout > Other**.
2. Set **Include single parts** to **Yes**. This activates also the Single part attributes field allowing you to choose any predefined single part attributes.



3. Click **Modify** and then **Apply**



The single part views for plates appear.

Tekla Structures has several environment variables to customize the single part presentation in assembly drawings. For example, you can choose to exclude some single-part views from assembly drawings.

Help: System > Appendix A: Variables > List of variables > S > XS_SINGLE_EXCLUDE

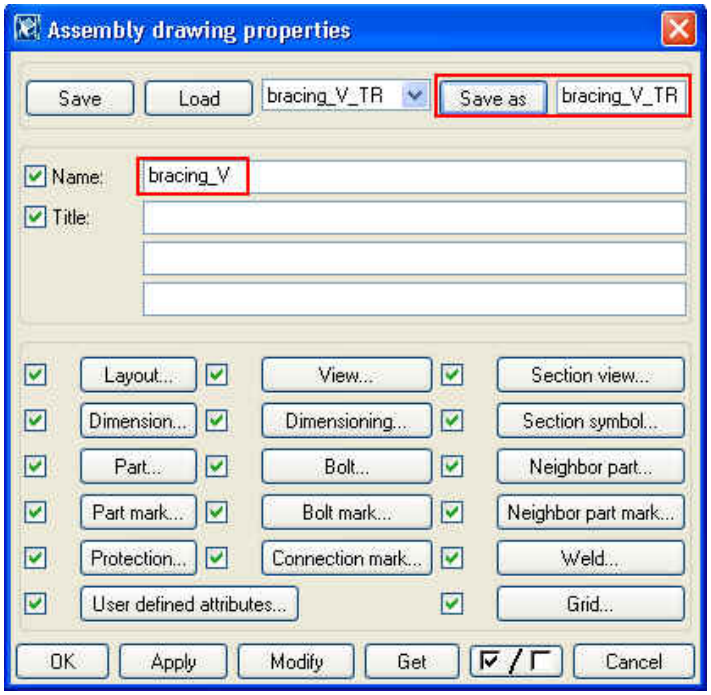
Or you can, for example, set the scale of single-part views included in assembly drawings.

Help: System > Appendix A: Variables > List of variables > S > XS_SINGLE_SCALE

Save as properties for bracing_V

We will now save the properties for vertical bracing

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



11.4 Create a New Drawing Wizard

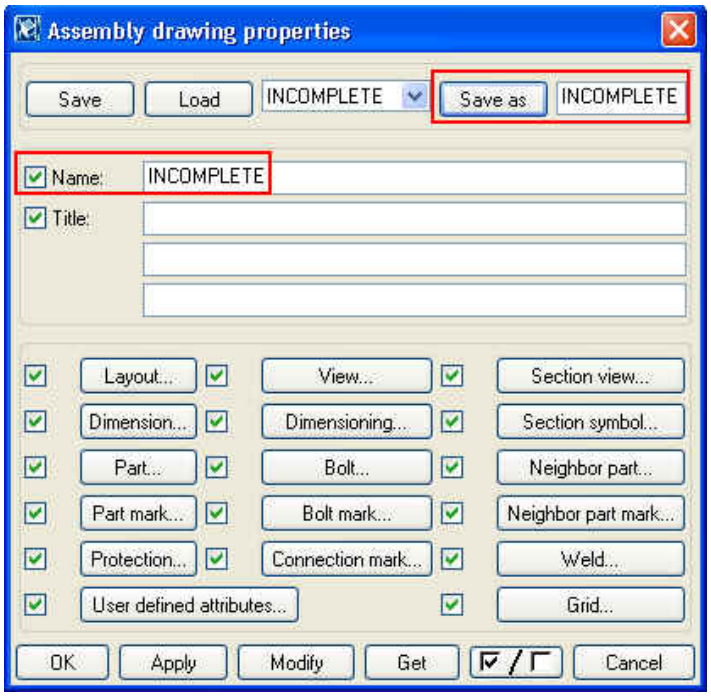
We will next create a new assembly drawing wizard that matches the part properties and select filters defined in lesson 5. We will use the available drawing properties added with the **bracing_H_TR** and **bracing_V_TR** properties defined in this lesson.

We start by creating properties for the parts not matching with the wizard requests

We will then edit existing drawing requests to suit horizontal and vertical braces and use the new properties defined for them. We will delete all those requests that are not needed and finally edit a request 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 the properties with the 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

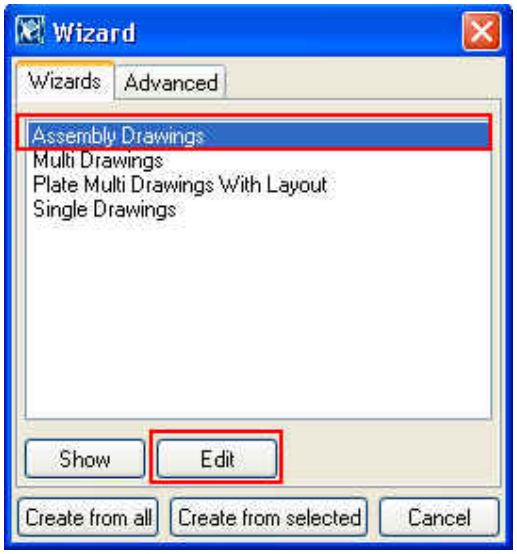
Help: System > Files and folders > Log files > Wizard log

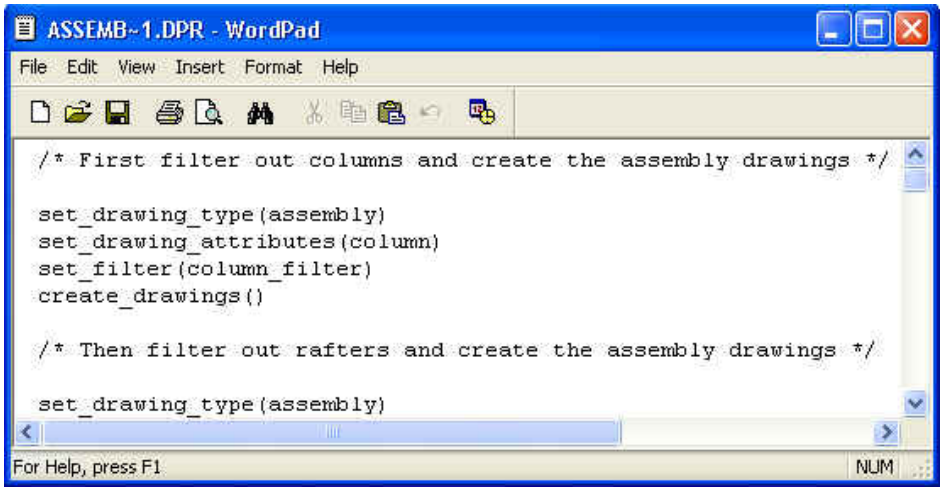
To create the new wizard file:

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

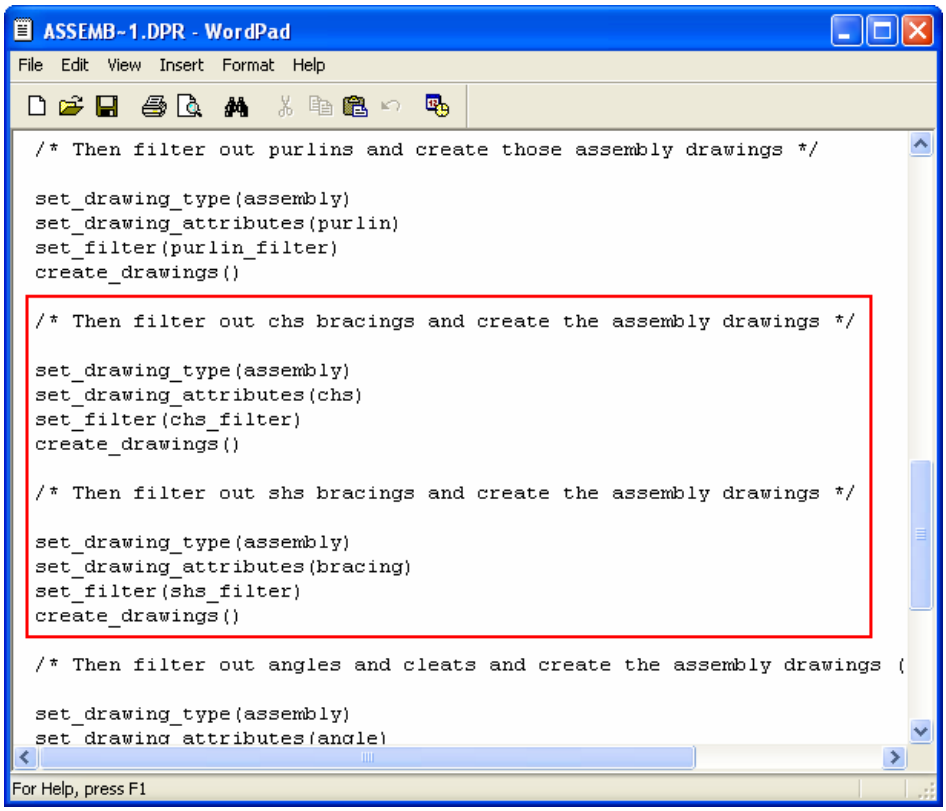
> The predefined wizard files are located in the folder **..\countries*environment*\system**. In the system folder, the wizard files are available for use in all models. Modified and saved wizard files are saved in the same folder, unless you define another destination folder (such as the model folder).

Edit the wizard file

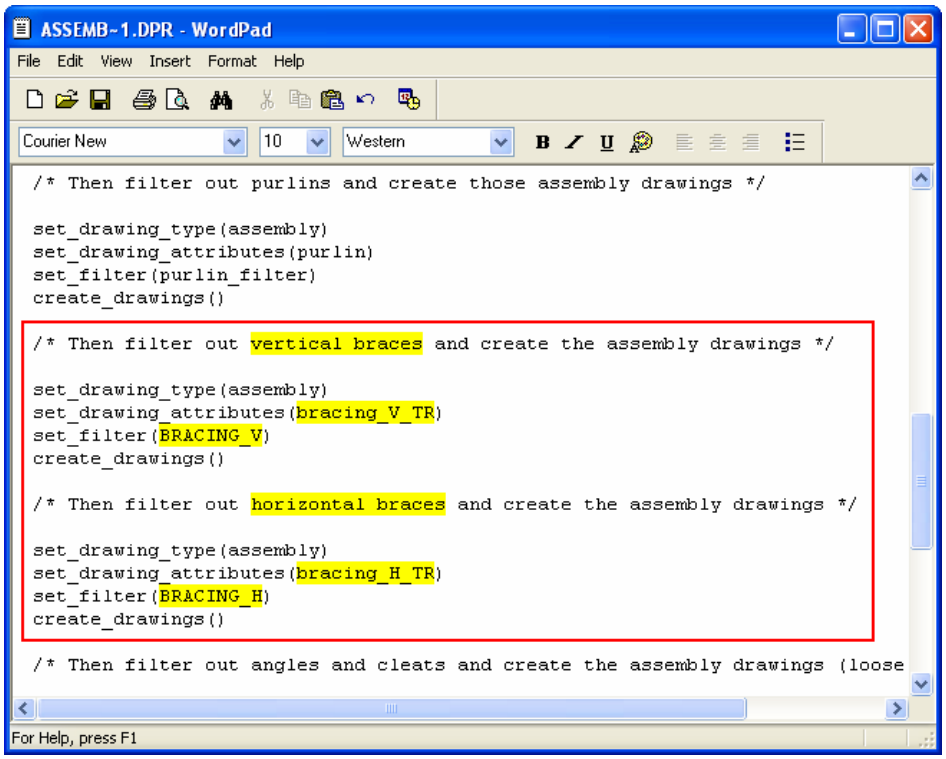
1. Save and Reopen the model.

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

2. Open the Wizard dialog box.
3. Select ASSEMBLY_TR on the **Wizards** tab and click **Edit**.
4. Find the requests for chs bracings and shs bracings.

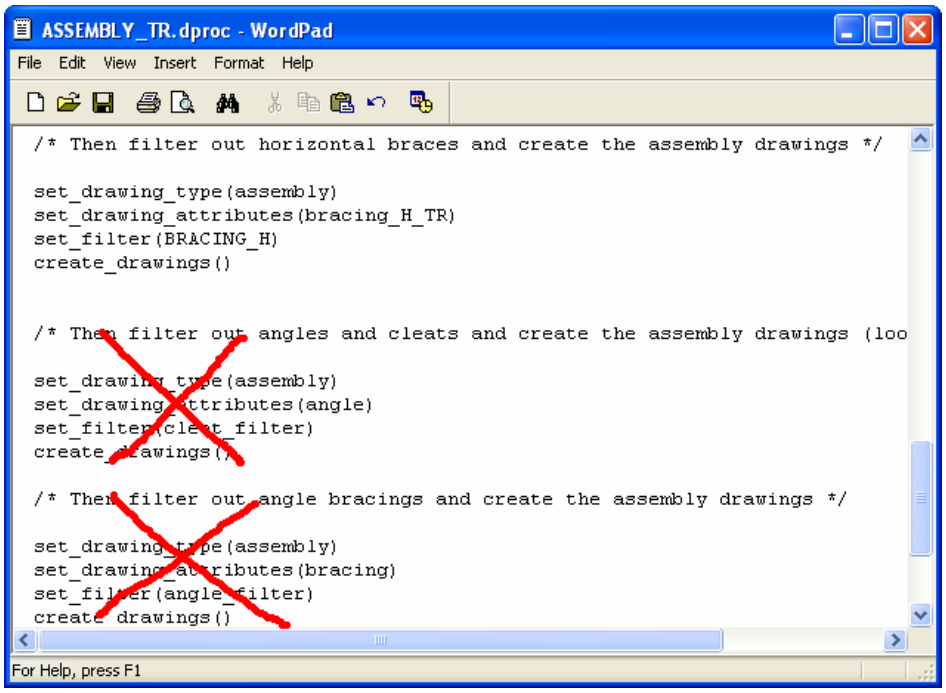


5. Edit the requests to match the criteria for vertical and horizontal bracing shown highlighted in yellow below.



Delete the extra requests

6. Delete all extra requests. Only keep the following:
   - Beams
   - Vertical braces
   - Horizontal braces
   - Rafters
   - Purlins
   - Columns

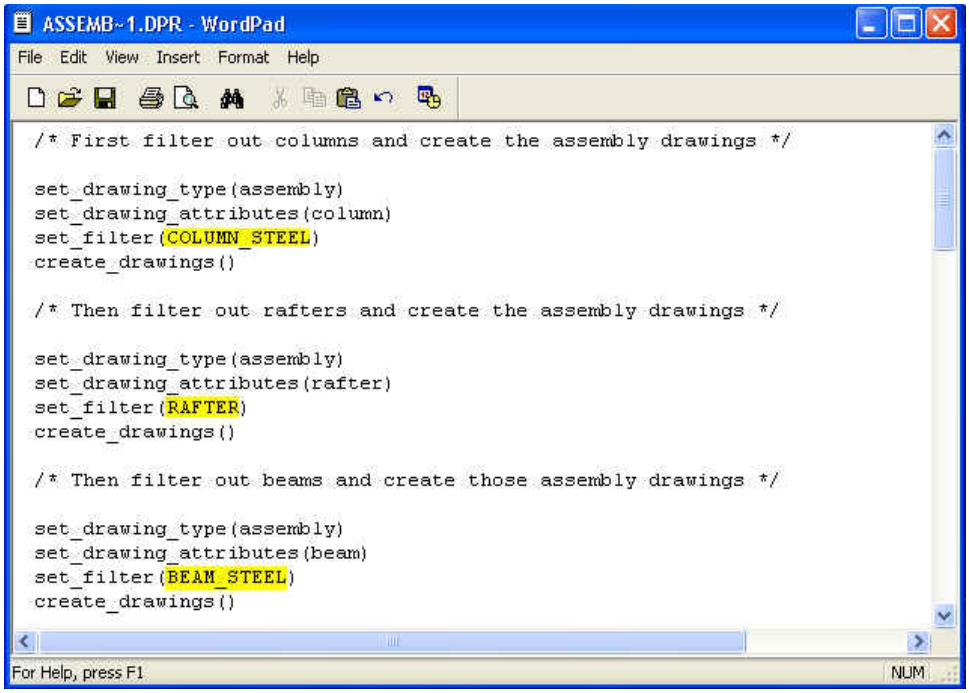


Change the remaining requests to use the select filters that we defined earlier, for this project.

Change the select filters

7. 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

8. 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

9. 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.

Apply own wizard file

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

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 Edit Drawings Manually

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

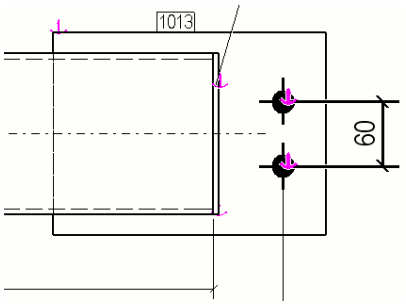
> Most drawing objects (both automatically and manually created) are associative and automatically update if the model changes. However some manually created drawing objects such as lines, polygons, and circles are yet not associative.

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

Add bolt distance dimensions

1. Open the first bracing_H drawing in the drawing list
2. Click on the 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 on the middle mouse button



> The associative symbols indicates which drawing objects are associated to the model and will be updated if the model is changed
>
> 
>
> Click SHIFT+A in the keyboard to display/hide the associative symbols

To move the dimension: select it and drag it to the desired position

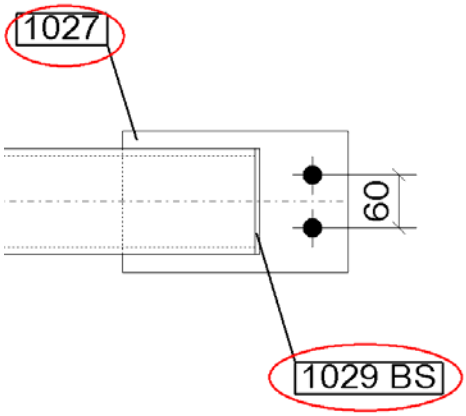
Add the bolt dimension to the other end in the same way.

Change the frame of part marks

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

   (You can use the Select part mark select switch.)

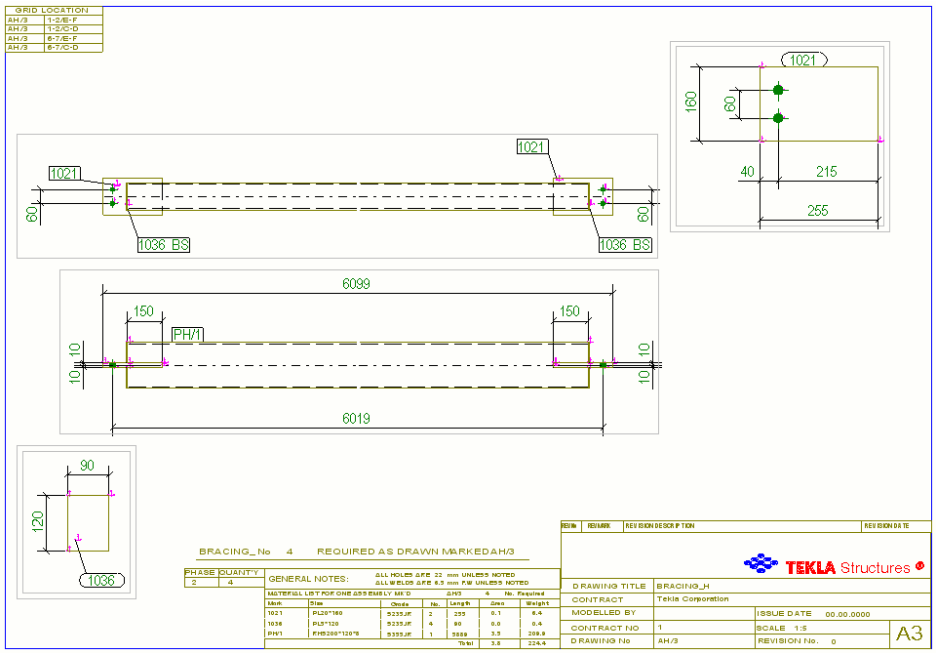
4. **Modify**



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

Include single part views

1. Using the **CTRL** key select each plate part in the drawing.
2. Right-click and select **Create single part views** from the popup menu.



3. Close the drawing, click **Save and Freeze** in the confirmation dialog box

When you close a drawing that has been changed, Tekla Structures prompts you to save the drawing.

Whenever you have manually edited the drawing it is recommended to select **Save and Freeze**. This way you will systematically freeze edited drawings and only them.

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

Now this one horizontal brace drawing has roughly the same editing as all the vertical brace drawings. The difference is that part of this horizontal brace drawing editing was done manually.

If we wanted the rest of the horizontal brace drawings to have the same editing, we could repeat the manual editing to them one by one. However, changing the bracing_H_TR drawing properties is a better solution.

STOP

To get the creation of drawings as automatic as possible, you should find good enough predefined drawing properties to create the drawings with, instead of editing them manually.

Whenever there is a need to edit the drawings, you should

first check if the result can be achieved by changing the drawing properties.

As long as you can manage to create complete drawings by using predefined drawing properties the creation and updating off the drawings will be more automatic.

11.6 Update 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 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.

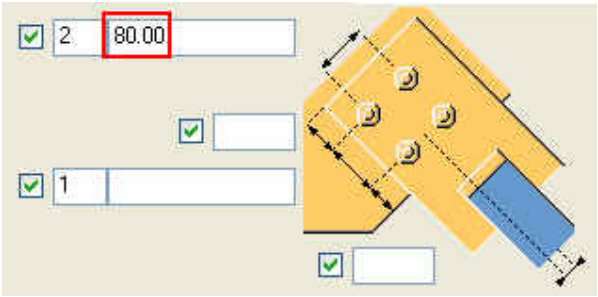
Updating will:

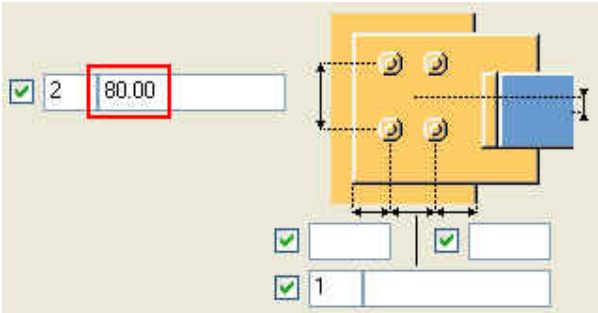
- Switch the **P** flag on a frozen drawing to an *. This indicates that the drawing has been updated (also manual editing, such as extra marks or dimensions).
- Regenerate any drawings with a **P** flag that are unfrozen, with the originally used drawing properties. The updating deletes all the manual editing (added dimensions, texts etc.).
- Update the quantities on a drawing with an **N** flag.

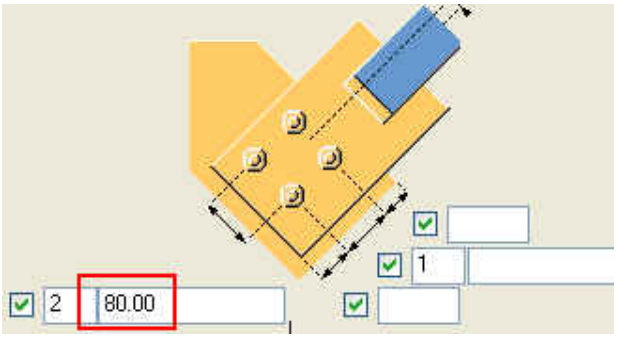
Change the model

Change gusset plate bolt spacing

1. Select all the connections in the model.
2. Find the Tube gusset (20) connection (Ctrl+F), and double-click on it to open it.
3. Check that **Ignore other types** is selected in the connection dialog box.
4. On the **Bracebolts1**, **Bracebolts2** and **Bracebolts3** 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.

STOP

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.

Run numbering

Select **Tools > Numbering > Modified**.

Once the numbering is carried out, the flags in the **Drawing list** show all those 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

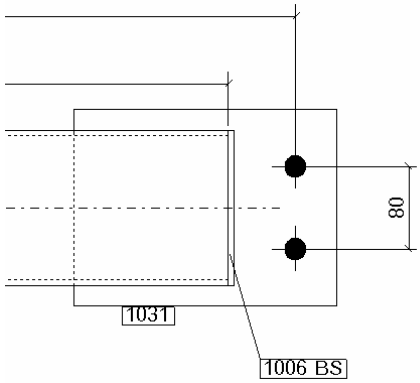
You can filter up-to-date drawings with **Filter > Up to date** button. To get non up to date drawings listed press **Display > Invert** button after **Filter > Up to date**.

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



Update horizontal brace drawing

We will next update all the horizontal brace drawings including the manually edited one.

Before updating drawings **Make sure that all the drawings having manual editing are Frozen.**

Updating will regenerate any drawings with a **P** flag that are unfrozen, the updating deletes all the manual editing (added dimensions, texts etc.).

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

1. Select all the bracing_H drawing from the list except the frozen one.
2. Click **Update**.

Because 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 regenerate the drawing with the predefined drawing properties.

To consciously override manual modifications in a drawing and regenerate the drawing with predefined drawing properties, you can:

3. Unfreeze the drawing.
4. Update the drawing

Or in case the drawing is already up to date:

1. Unfreeze the drawing.
2. Modify it first, using other drawing properties (e.g. No_dimensions)
3. Modify it with the original properties (e.g. bracing_H)

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).

Update frozen horizontal brace drawing

To update frozen drawings:

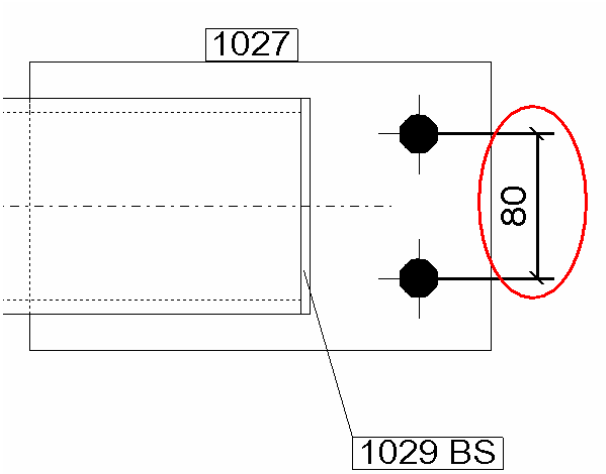
1. Select the frozen bracing_H drawing from the list.
2. Click Update.
3. The P gets replaced by an *

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

Check and save the drawing

1. Open the drawing with an *.

The dimensions should be correct.



2. Save the drawing, and the * flag is removed

11.7 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 automatically dine by the drawing wizard:

1. Select the model objects.
2. Load and Apply 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

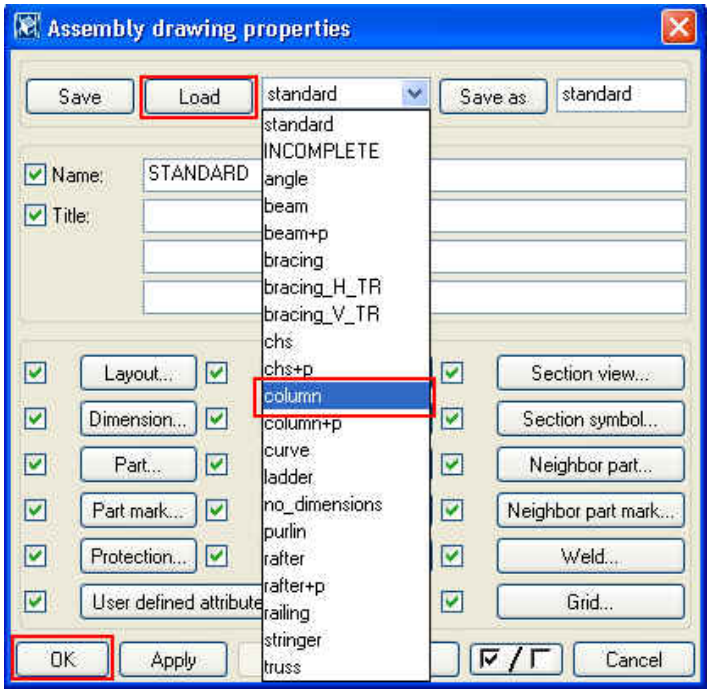
As an example, we will create assembly drawings of the columns.

To manually create assembly drawings of all columns:

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

Create assembly drawings from columns

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



5. Click **Load** and **OK**.
6. Select **Drawing > Assembly drawing** in the menu to create the assembly drawings.
7. 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.