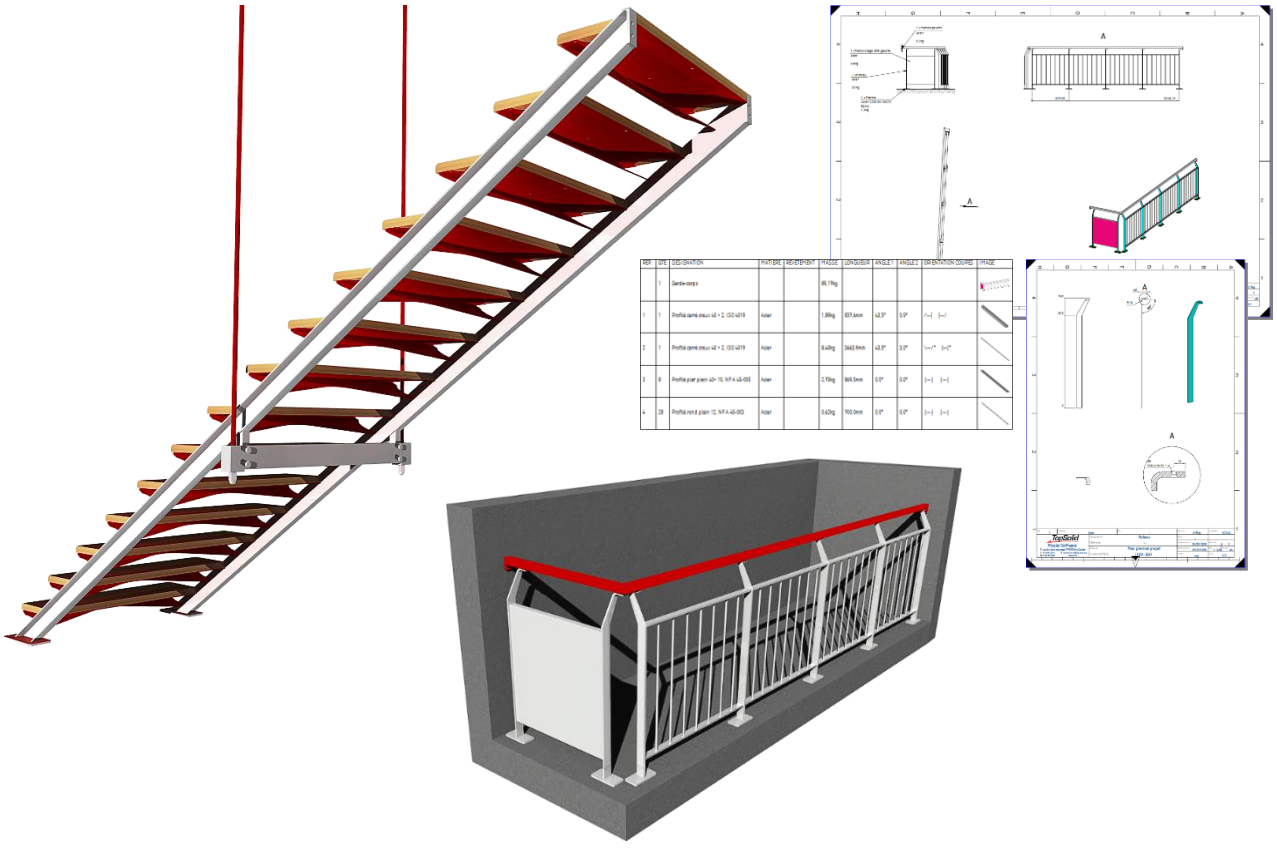
TopSolid Steel

Training Guide

TopSolid'Steel - Basics



MASTER YOUR MANUFACTURING PROCESS

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Note: If you are experiencing problems using this training guide, please feel free to send your feedback and comments at edition@topsolid.com.

Contents

Introduction to TopSolid 7

The purpose of this first section of the guide is to familiarize yourself with the **TopSolid 7** environment and some of the tools available.

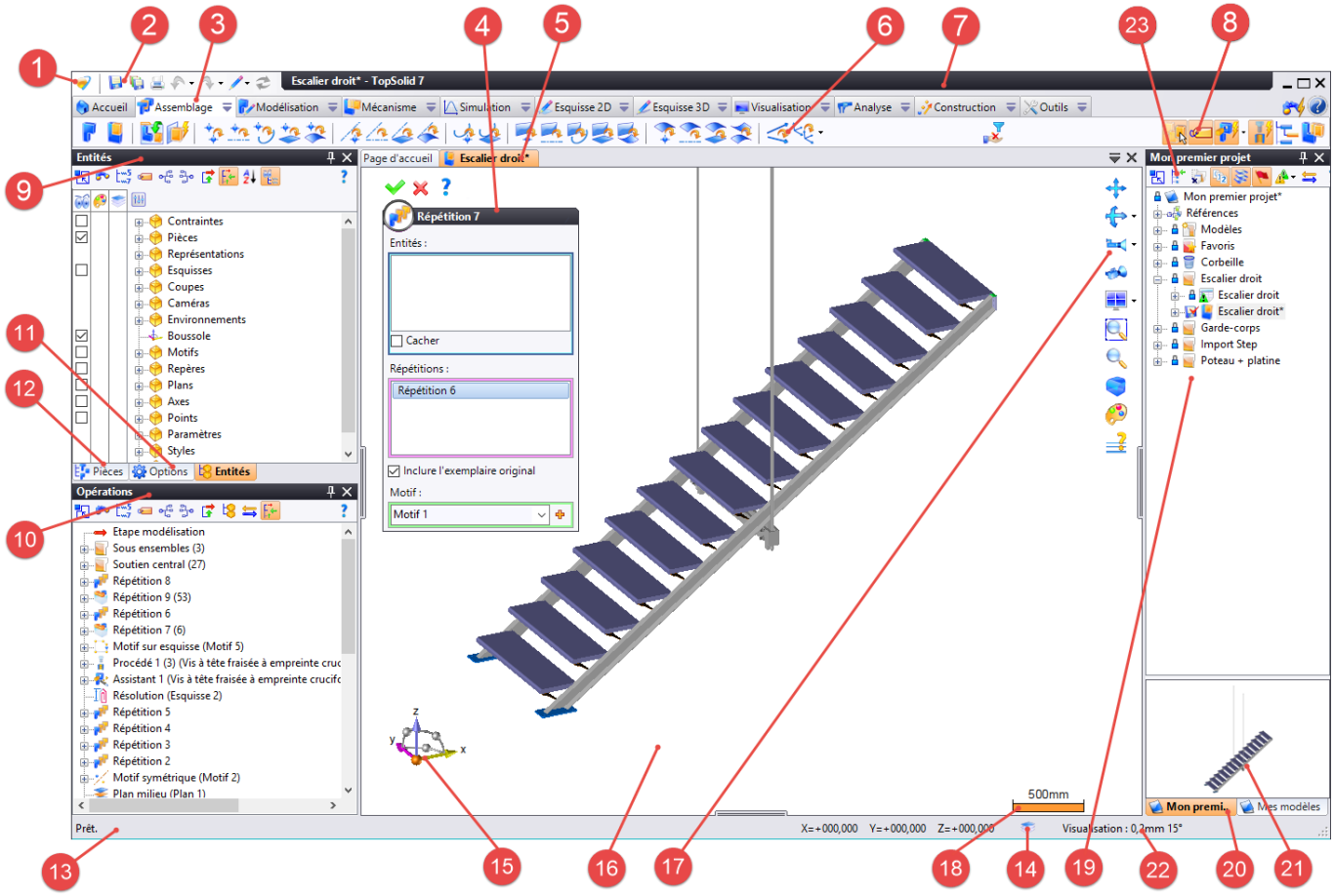
For your information, **TopSolid 7** is a certified Gold Level partner of Microsoft and is fully compatible with Windows 10.

TopSolid 7 environment

Start page

1	Creation of new documents
2	Recent/Pinned documents (favorites)
3	Recent/Pinned projects (favorites)
4	Shortcut access to the online help, **TopSolid'Design** tutorial, **TopSolid'Forum**, etc.

General user interface



1	TopSolid 7 button	13	Dialog area
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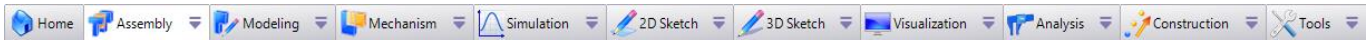
9	Entities tree	21	Preview area
10	Operations tree	22	Visualization tolerance
11	Parts tree	23	Tree toolbar
12	Options tree		

Context tabs

The tabs change depending on the type of document you are using and define the commands available in the control bar.

Example for a part document:

Example for an assembly document:



Example for a drafting document:

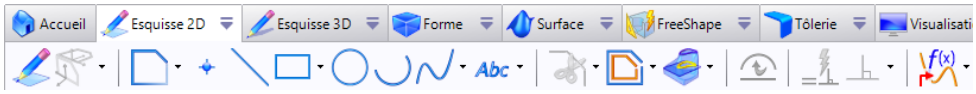
Control bar

The control bar includes the various **TopSolid** commands and changes depending on the active context tab.

Example for the sketch:

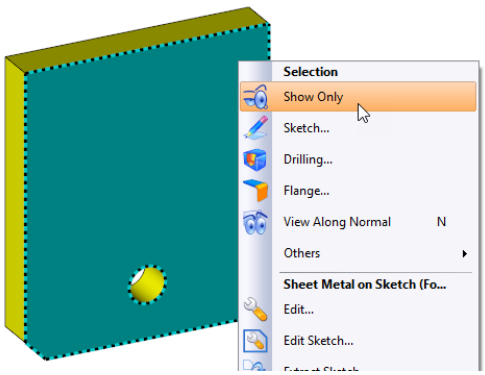
There are four ways to launch a command:

- via the control bar;



- via the drop-down menu of a context tab;

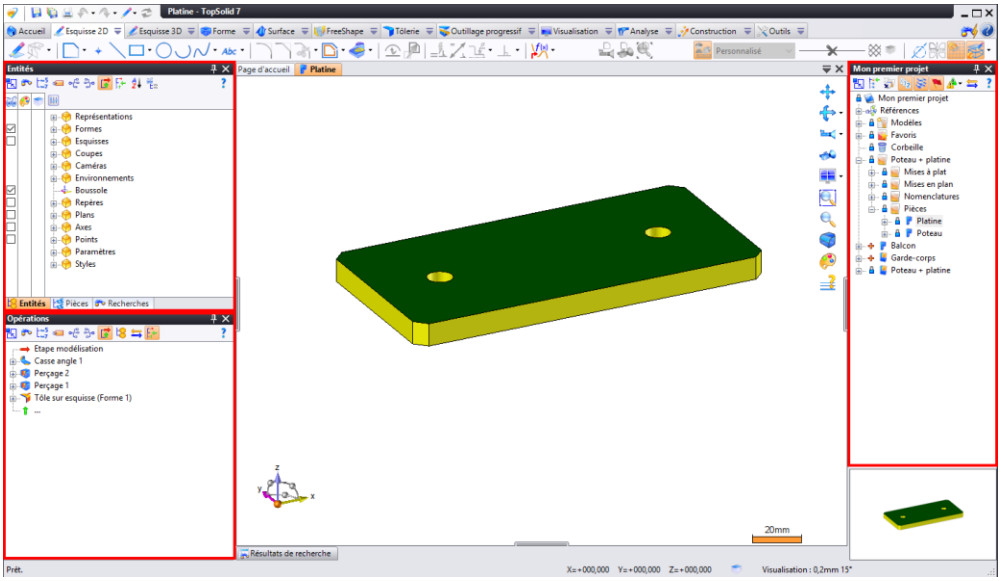
- via the contextual menu by right-clicking in the graphics area or on an entity;



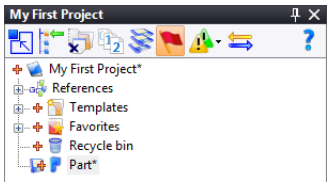
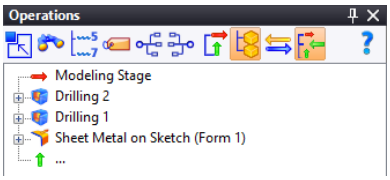
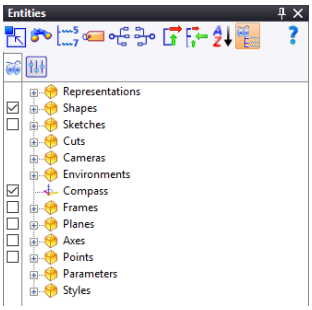
- via the keyboard shortcut that you have previously defined in the **Tools** > **Customize** > **Shortcut keys** tab.

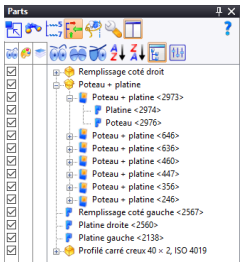
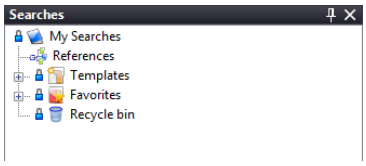
The main trees

The trees are scalable dialog boxes (size, position, on one or more screens, etc.) that contain all the information required to design and manage a project. They change depending on the type of document you are using.



The main trees are the following:

Project	Operations	Entities
Contains all the project's documents (parts, assemblies, drawings, etc.).	Contains all the operations created in the active document.	Contains all the entities created in the active document (sketches, points, planes, parameters, etc.).
		

Parts	Searches
Contains all the parts and sub-assemblies of an assembly document.	Contains all the defined searches. The searches allow you to quickly access documents according to a variety of criteria (material, name, property, type, etc.).
	

The main types of documents

Part	Assembly	Drafting
Contains the part definition (properties, materials, geometries, operations).	Allows you to assemble several parts, create extruded bars, etc.	Allows you to project, dimension, annotate, etc. parts and assemblies. This step usually ends a study.
		
Bill of Materials	**Unfolding**	**Family**
Lists all the parts/sub-assemblies of an assembly. This list is fully customizable (filters, columns, level of detail, etc.)	Contains the unfolding of a part. This document can be directly exported or nested.	Allows you to make a part parametric, an assembly or a 2D symbol in order to be used in different declinations.
		
Work		
Allows you to automate the creation of documents.		
		

Use of the mouse


Selection of item, icon, command, etc.


Editing an operation


Contextual menu of an item


Validation of a command


Zoom in/out on mouse cursor


Global zoom on the displayed entities

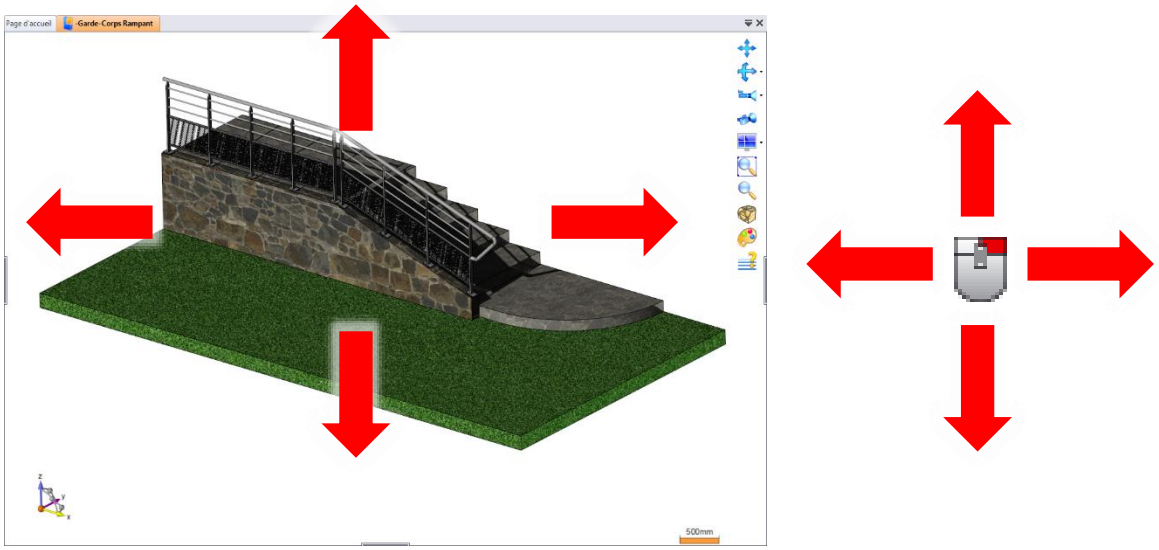

+ movement Dynamic rotation


+ movement Dynamic translation (planar movement of the graphics area)


In the graphics area: Rectangular selection of items or drag and drop items


On a document's tab: Close document


On the  icon of a context tab or  on a context tab: Drop-down menu of the context tab



Visualization


Spin


Pan


Rotation


Top camera and access to other cameras (bottom, front,


Perspective view


View management


Global zoom


Zoom


Render management


Default attributes


Default line styles

Keyboard shortcuts

TopSolid 7 uses a large number of keyboard shortcuts - most of them basic Windows shortcuts. You do not need to know them by heart necessarily, but they can help save you time in your everyday work. Here is a non-exhaustive list of the most frequently used keyboard shortcuts.

Keyboard shortcut	Action
Ctrl + **C**	Copy the selected item
Ctrl + **V**	Paste the selected item
Ctrl + **X**	Cut the selected item
Ctrl + **Z**	Undo an action
Ctrl + **Y**	Redo an action
Ctrl + **A**	Select all items in a document or dialog box
Ctrl + left mouse click	Select several items or remove items from a selection
Ctrl + **S**	Save the active document
Ctrl + **Shift** + **S**	Save all documents
Ctrl + N	Create a new document
Ctrl + **Shift** + **N**	Create a new project
Ctrl + **Shift** + **O**	Open the Project Manager
Ctrl + **I**	Check a document into the vault
Ctrl + **T**	Show the selected document in the Project tree
Ctrl + **R**	Regenerate the active document
Ctrl + **Shift** + **F**	Enable the front camera
N	View along normal
Ctrl + **P**	Print
Enter	Confirm an operation
Del	Delete the selected item
Esc	Cancel the active command or exit the dialog box
F1	Display the **TopSolid** online help
F2	Rename the selected item
F5	Refresh the active document
F11	Switch to full screen
Alt + specified values	Insert standard Windows symbols: - **Alt** + **157**: Insert a diameter symbol (Ø)

Introduction

Concepts addressed

In this chapter, the objective is to understand how the project manager, projects and libraries work and finally how to create a project.

Concept

The documents generated by **TopSolid** are stored in a database called PDM *(Product Data Management*). Similar to a mailbox, the PDM allows you to centralize all the information required for studies.

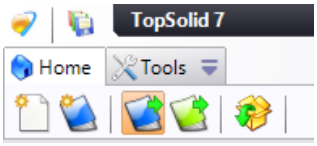
TopSolid has three levels of organization:

- **Projects**: They allow you to directly store your **TopSolid** documents and your external documents (Word, Excel, image, etc.). Depending on the company, a project can be a case, a production order, a company standard, a quote, etc.
- **Libraries**: They group elements that are common to several projects, such as railing ranges, extruded bars, materials, etc. **TopSolid** has hundreds of components (screws, nuts, bearings, extruded bars, etc.) in several standards (AFNOR, ANSI, CEN, ISO, etc.).

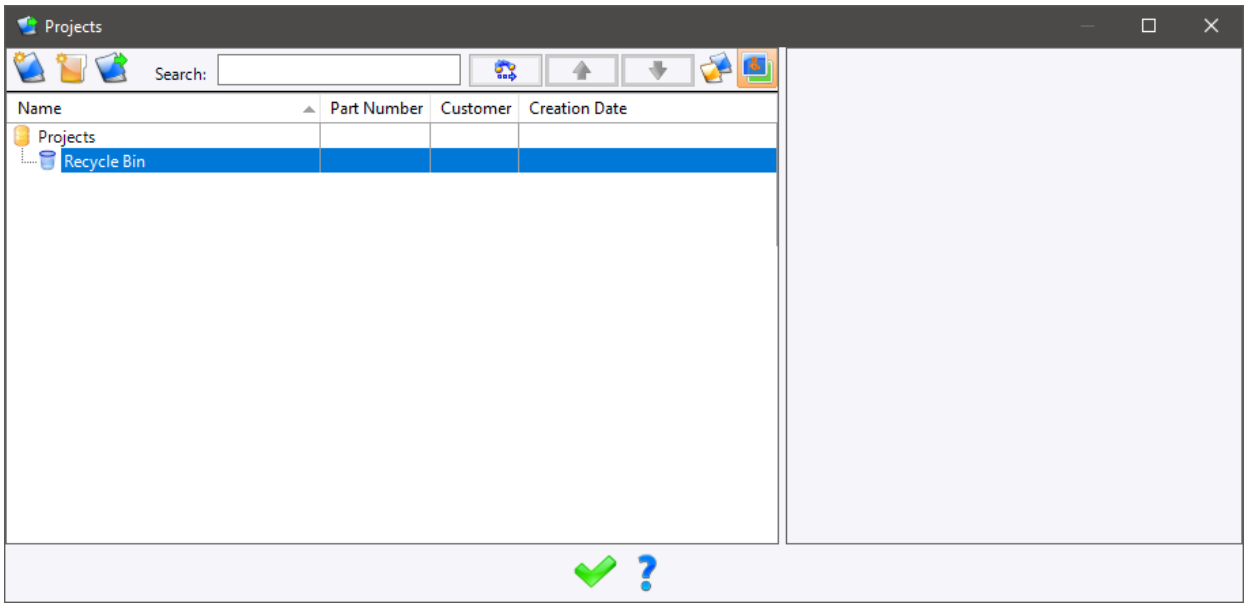
  The libraries can be freely enriched thanks to many websites such as **TopSolid'Store**, TraceParts, Cadenas, GrabCad, BIMObject, etc.
- **Project Manager**: It contains all the projects created. It allows you to group the projects by folder, sort the projects by property (customer name, delivery date, customer project identification number, etc.), preview the projects, and also search for projects. There is also a library manager.

Project Manager

- From the **TopSolid 7** start page, click on the **Projects** icon to display the project manager.



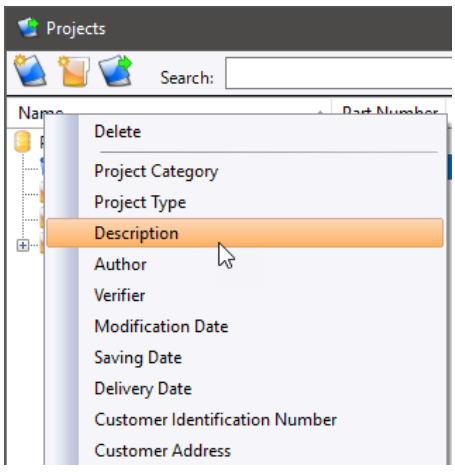
The project manager dialog box appears.



The project manager offers the following options (from left to right):

- Creating a new project
- Creating a new folder
- Opening the selected project
- Search bar
- Displaying the template projects
- Project previews

Note: You can add or remove properties in the Project Manager's columns. You simply have to right-click on a column and select or delete the desired property.



- From the Project Manager, create three **folders**.

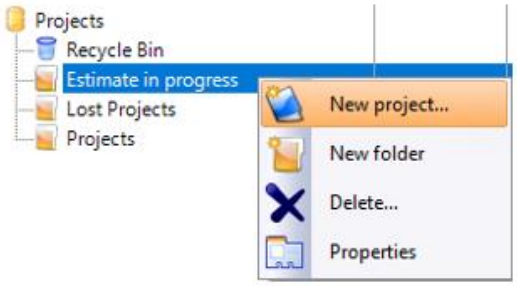
- Rename the created folders as follows:
  - *Projects*
  - *Lost Projects*
  - *Estimate in progress*

Note: You can create sub-folders.

To change the order in which projects are sorted, you simply have to click on a column heading and an arrow icon will appear on the right.

Creating a new project

- Right-click on the *Estimate in progress* folder and select the **New Project** command.

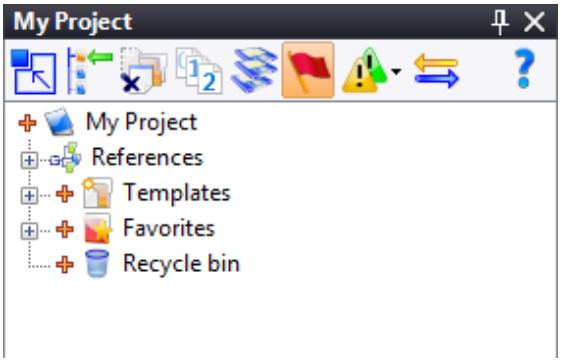


- From the **Steel Templates** folder, select **Design Template**, rename the project *My Project*, then click on to **confirm** the operation.

Warning: Never start with a blank project template because it does not contain any document template.
Templates are necessary to perform mass calculations to obtain precise cutting lists, etc.
If the wrong templates are used, you would have to restart the work or modify the parts one by one.

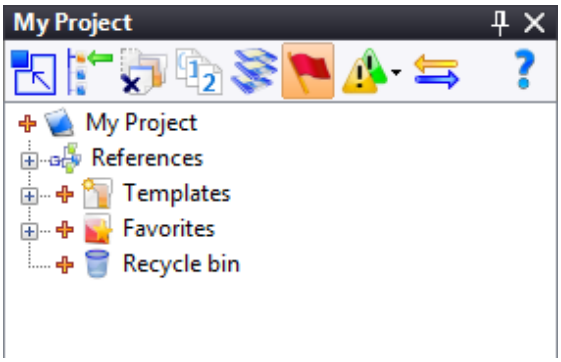
- Double-click on the project name that you have previously created to open it.

TopSolid then displays the Project tree.



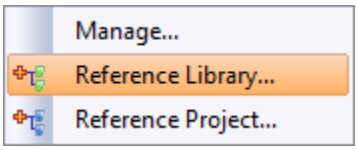
Introduction to the Project tree

Like all **TopSolid** trees, the Project tree includes an upper bar which contains options in order to display revisions, track documents, indicate which documents require updating, etc.



The lower section of the tree contains a header document, in this case, called *My project*, which contains all the information about the project (customer name, reference, project type, etc.). This information can be used in all **TopSolid** documents.

The **references** contain the referenced libraries for the active project and allow you to display only the elements useful in the current project. To add or delete the libraries, you simply have to right-click on the **References** node and select the **Reference Library** command.



The **templates** regroup document templates (unfolding, drafting, part, assembly, etc.) that can only be used in the current project.

Each project has its own **recycle bin**, which prevents any unexpected deletion or deletion of used documents.

Part 1 - Post Reassembly Design

The objective of this exercise is to draw simple sheet metal parts, assemble them and then use them in manufacturing documents (bill of materials and drafting).

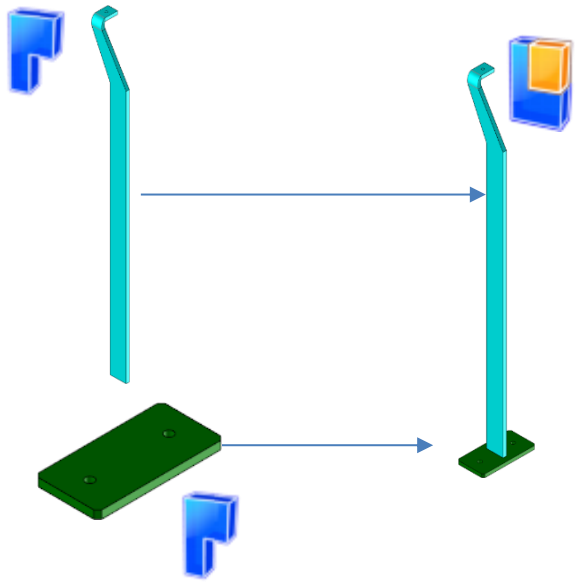
Concepts addressed:

- Creating the sheet metal parts
- Assembling the parts using constraints
- Creating a bill of material
- Creating an unfolding
- Creating parts and assembly draftings
- Understanding and using document and project properties
- Understanding the concept of vaulting and update cycles

Concept

The first way of designing in **TopSolid** consists of creating parts (or assemblies) independently of each other, then assembling them to produce a structure. This is called *reassembly design* or *bottom-up design*.

In this exercise, we will successively design a plate and a railing post that we will assemble in another step.



Creating a simple part: plate

- From the Project tree, right-click on the *My Project* name and select the **Part** command.

TopSolid uses the document template provided with the project.

Warning: Never start by using the **Blank template** document because it will not calculate the mass of the part nor its physical properties (surface, sheet metal thickness, etc.).

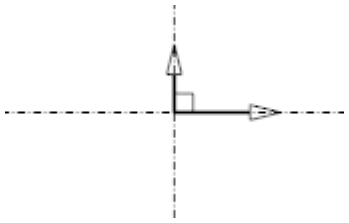
- Right-click on the part document you have just created and select the **Properties** command. Click on the **Edit** button. In the **Description** field, enter *Plate* and then click on to **confirm** the operation.

TopSolid automatically renames the part document with the same name as that which was entered in the description field.

Note: The icon displayed in front of the document name and the asterisk (*) shown after the document name mean that changes were made to the document, but they are not yet saved.

- Right-click in the graphics area and select the **Sketch** command.

TopSolid orients itself toward the selected plane. If no plane is selected, it uses the XY plane by default. In all cases, an XY frame is displayed in the center of the screen.

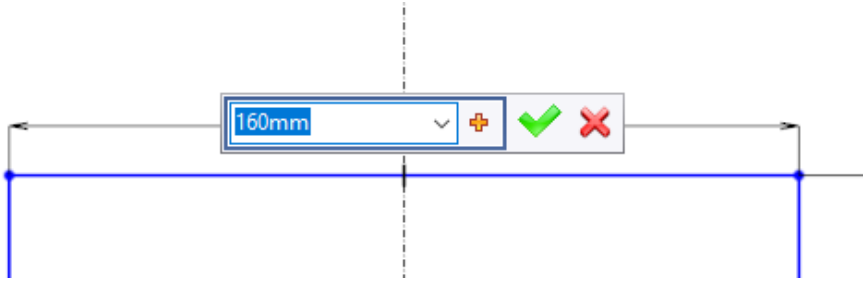


When a sketch is created, the **Contour** command is launched automatically and allows you to create a succession of segments.

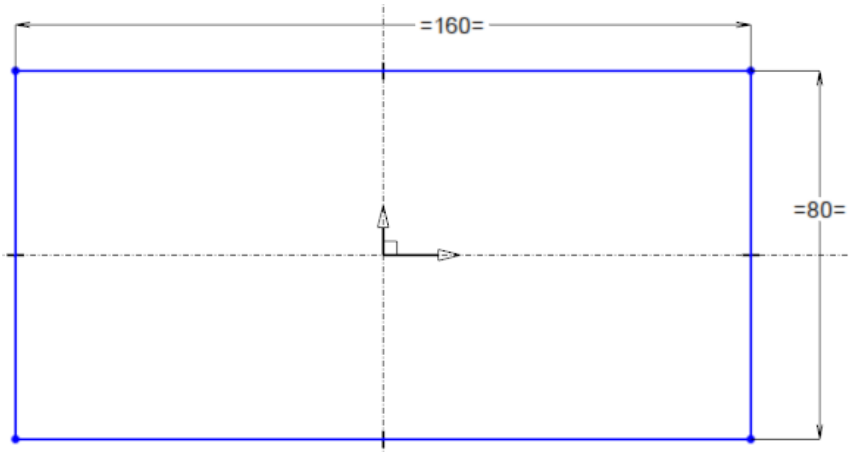
- From the **2D Sketch** tab, select the **Rectangle** command.
- Select the **Parallel to axes** option.
- Click on the first point at the bottom left of the graphics area.

When you move the mouse, **TopSolid** dynamically displays the dimensions of the rectangle. Centering constraints are directly applied if you try to create a rectangle centered on the frame.

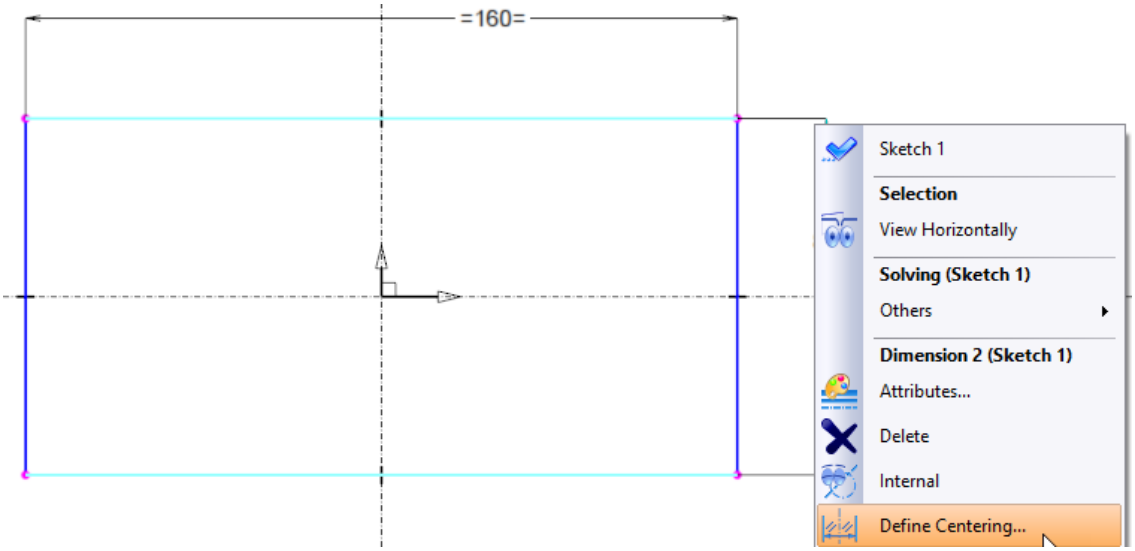
- Click on a second point at the top right of the graphics area to finish drawing the rectangle.
- Double-click on the upper dimension and enter the following value.



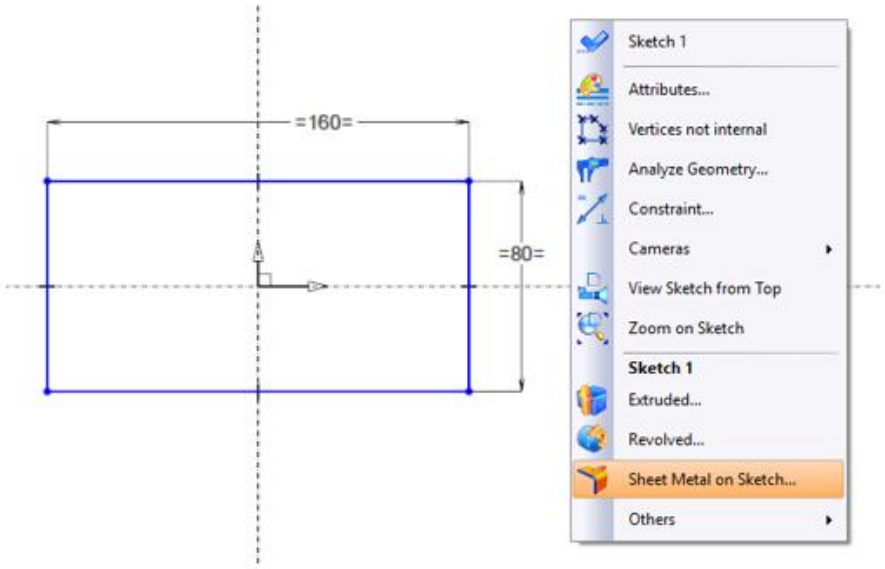
- Click on the icon or press the **Enter** key on the keyboard to confirm the entry.
- Finish constraining the rectangle by modifying the value of the second dimension as shown below.



Note: If the rectangle is not centered, some segments are displayed in pink. To center a dimension, you simply have to right-click on the desired dimension and select the **Define Centering** command. Select the centering axis.

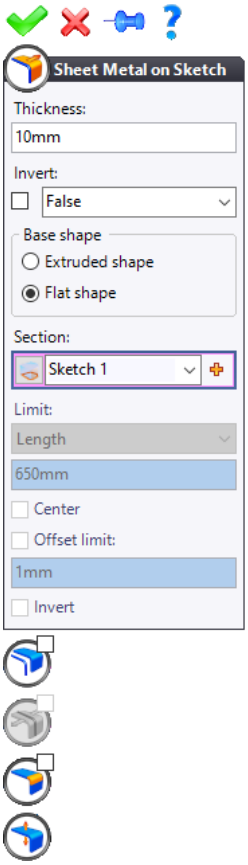


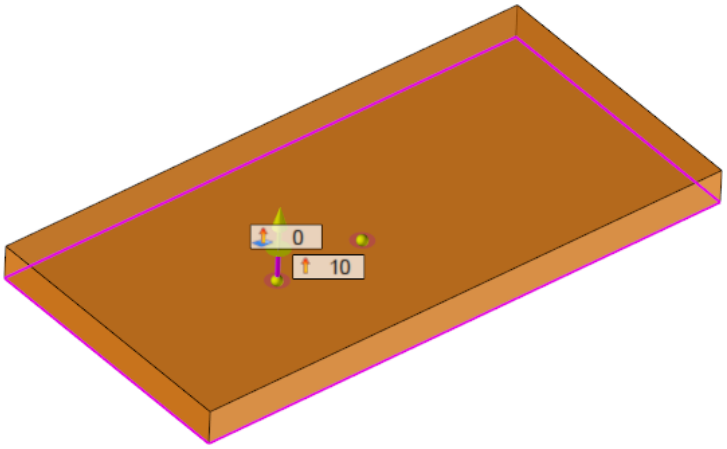
- Right-click in the graphics area and select the **Sheet Metal on Sketch** command.



- Adjust the **thickness** value to *10mm*.

Note: You can adjust the thickness value in different ways: either by entering the value in the corresponding field of the dialog box, or by double-clicking directly on the value label in the graphics area and adjusting the value, or by clicking and dragging the yellow arrow in the graphics area.



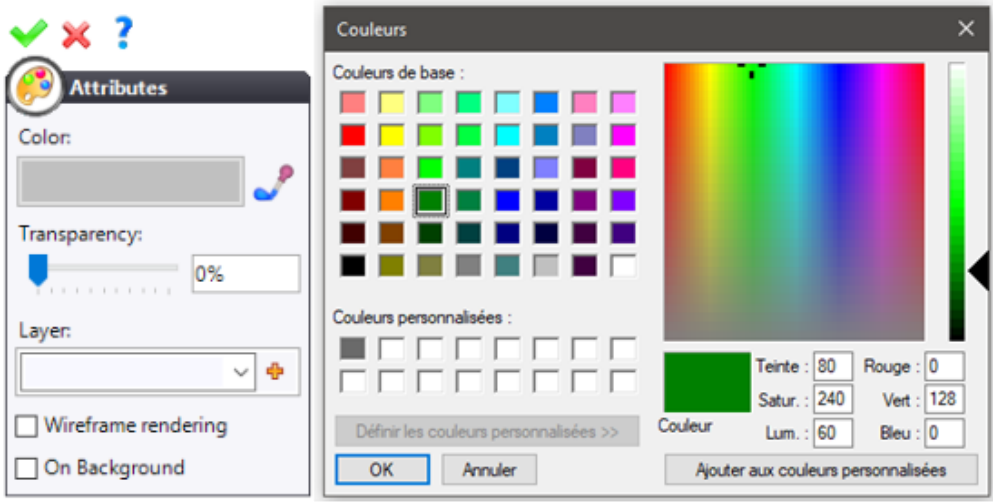


Also note that other options can be displayed directly from the graphics area by using the contextual menu.

- Click on to **confirm** the operation.

Note: The yellow border around the part means that the part is considered a sheet metal. **TopSolid** switches directly to the **Sheet Metal** tab and enables all of the commands in the tab.

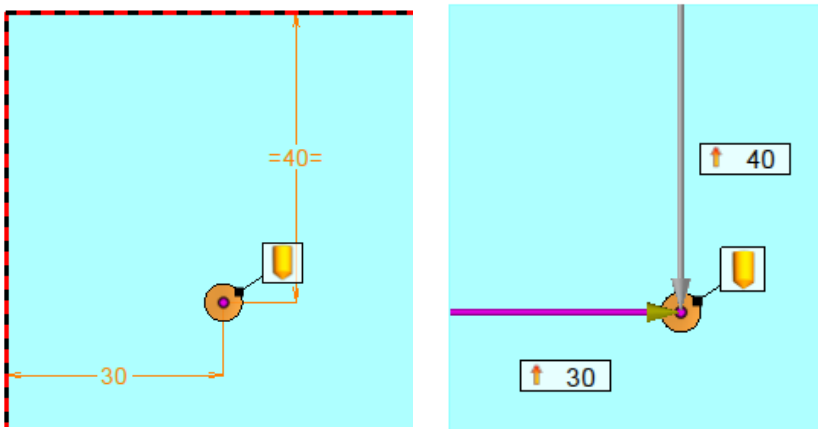
- Change the color of the part by right-clicking on it and selecting the **Attributes** command.
- Select a green color and click on to **confirm** the operation.



- Create a drilling on the plate. To do this, right-click on the top face of the part and select the **Drilling** command.

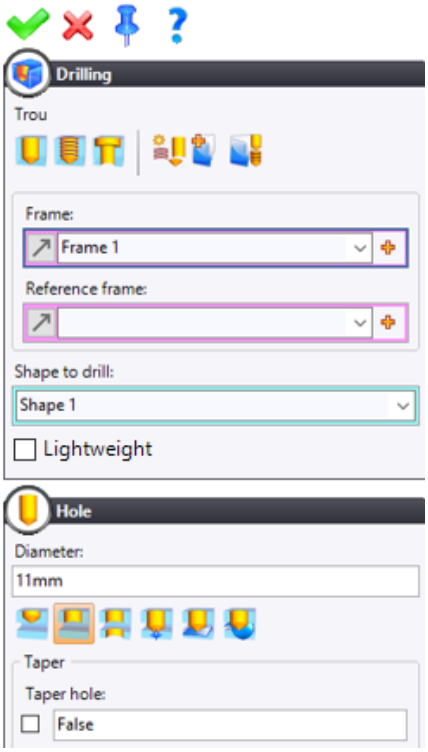
TopSolid dynamically positions the drilling based on the position of the mouse. The hooking operation is performed on the nearest edges. If the cursor is placed in the center, a centering constraint is automatically added.

- Click at the desired location to position the drilling by centering the hole across the width.



Note: You can adjust the position of the hole by modifying the dimensions. To do this, double-click on a dimension and enter the desired value.

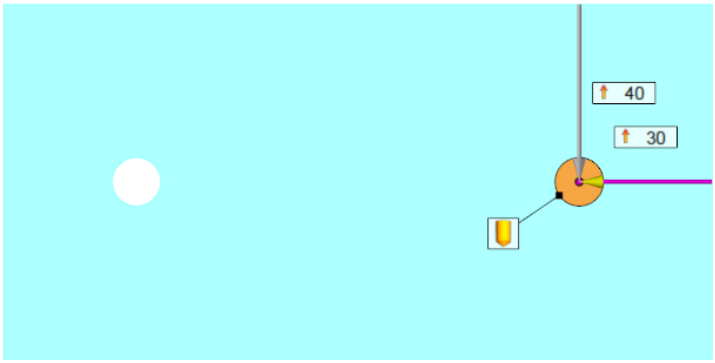
- Select **Hole** as the drilling type. Enter a drilling **diameter** of *11mm* and select the **Through** mode.



- **Pin** the command's dialog box and click on to **confirm** the operation.

The pin allows you to keep the command open after confirming the operation.

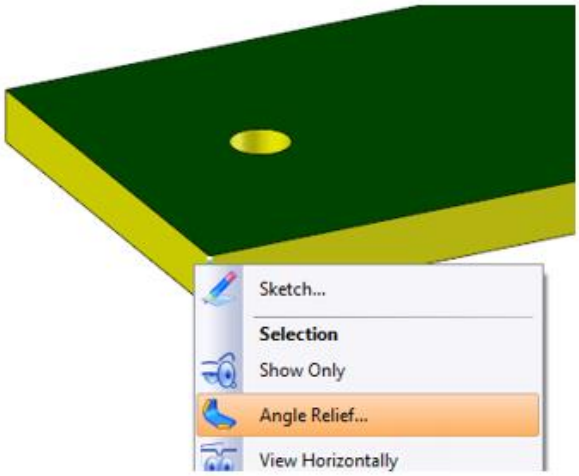
- Perform the same drilling operation on the other side of the plate and **close** the **Drilling** dialog box.



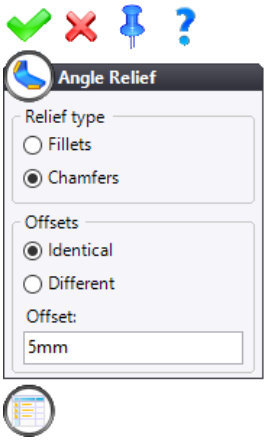
Note: The command can be confirmed by double-clicking with the right mouse button in the graphics area and then pressing the **Esc** key on the keyboard to close the dialog box.

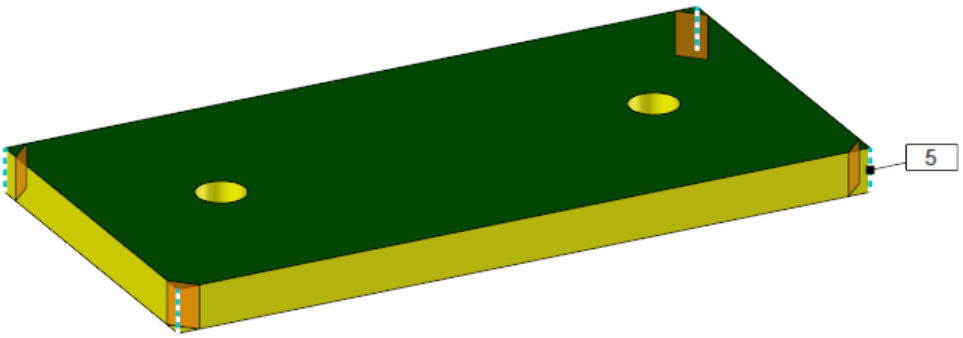
Later in the training, we will see how to position several holes at the same time, how to repeat them or how to create symmetries.

- Right-click on a vertical edge and select the **Angle Relief** command to relieve the angles of the plate.



- Select the other three angles using a bounding box, then select the **Chamfers** mode and enter an **offset** of *5mm*.





- Click on to **confirm** the operation.
- **Save** the plate by clicking on the icon or by using the **Ctrl + S** keyboard shortcut.

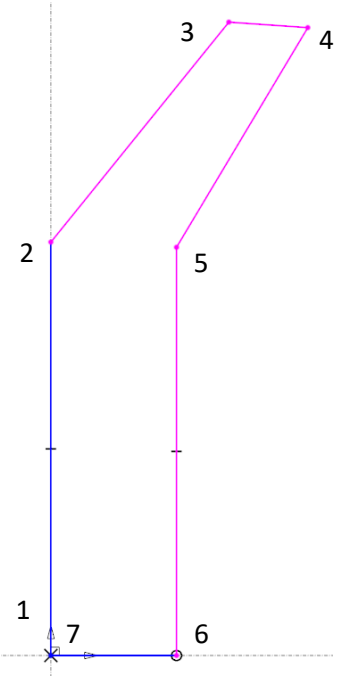
Creating a bent part: railing post

We will now create the post below.

- From the Project tree, right-click on the *My Project* name and select the **Part** command.
- Right-click on the part document you have just created and select the **Properties** command.
- Click on the **Edit** button. In the **Description** field, enter *Post* and then click on to **confirm** the operation.

Note: It is not mandatory to name the parts. Clear names simply make it easier to find files and understand the bill of materials. To identify/group the parts, **TopSolid** has mechanisms to manage part geometry and properties.

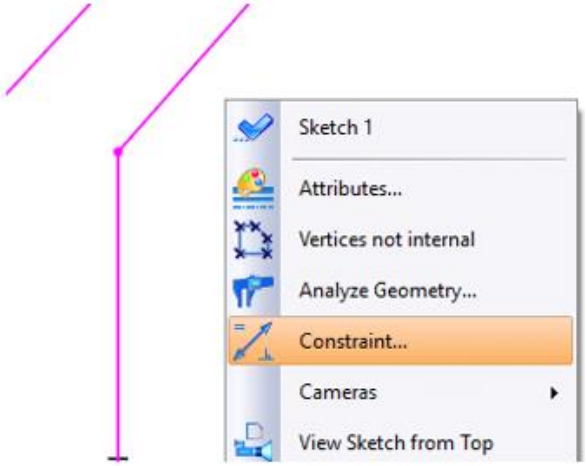
- Right-click in the graphics area and select the **Sketch** command.
- While keeping the **Contour** command enabled, draw the following contour using the order shown below.



The general idea is to approximately draw the desired shape without worrying about the dimensions. This step will come later.

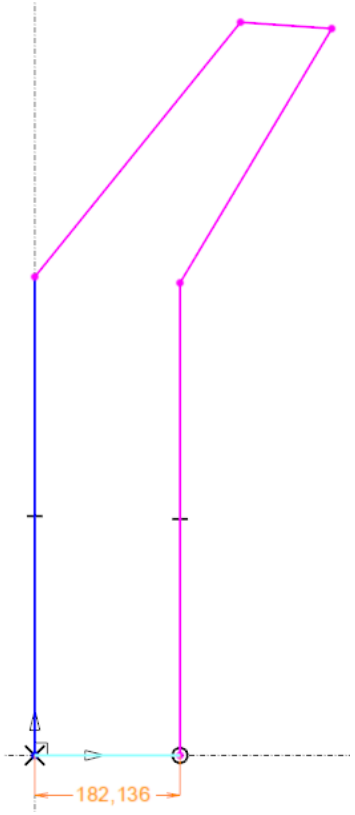
The pink segments indicate that the sketch is underconstrained, which means that there are missing dimensions and constraints to fix the shape in place.

- Right-click in the graphics area while making sure that no element is selected, then select the **Constraint** command.



The **Constraint** command allows you to add the dimensions but also, as its name indicates, to set the geometry constraints (perpendicularity, parallelism, coincidence, etc.).

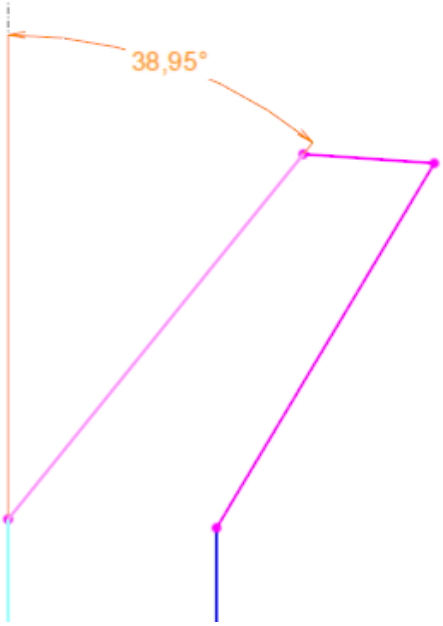
- Click on the segment as shown below and position the dimension.



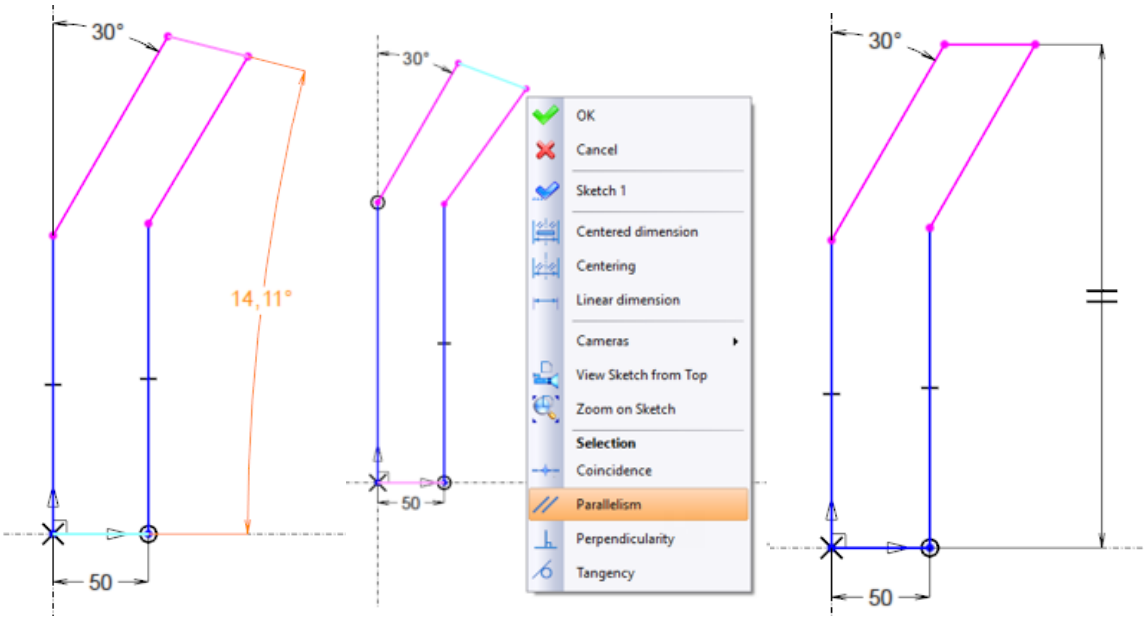
TopSolid offers to enter a value.

- Enter *50mm* and click on the icon or press the **Enter** key to **confirm** the entry.

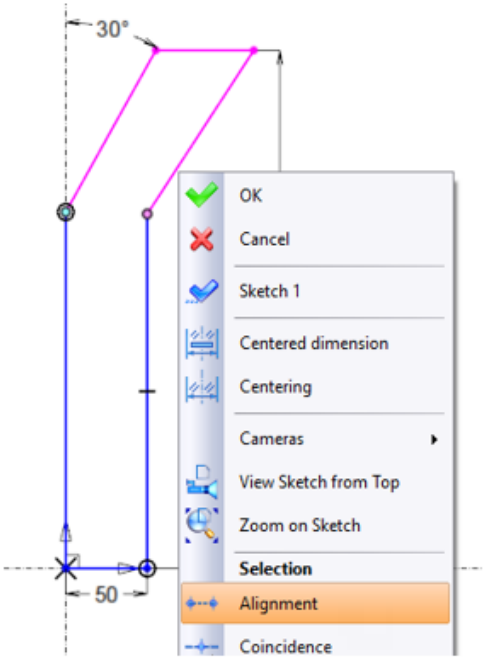
- Still using the **Constraint** command, select the following two segments. **TopSolid** automatically switches to angle dimension.



- Enter a value of *30°*.
- Using the **Ctrl** key, select the following two segments, right-click in the graphics area and select the **Parallelism** command.

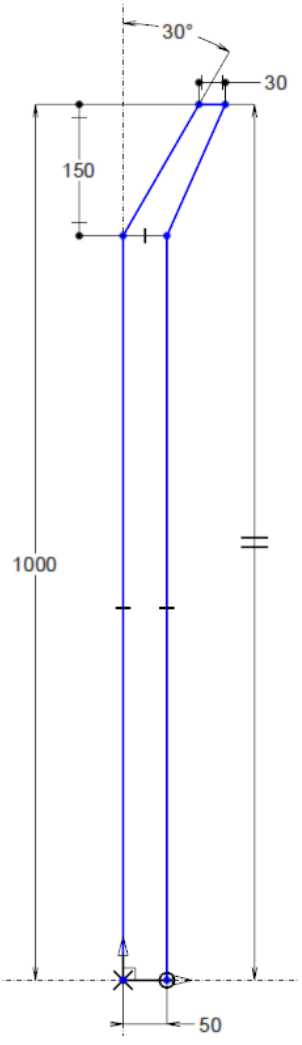


- Select the following two points, right-click in the graphics area and select the **Alignment** command.



Note: As in Windows, the **Ctrl** key allows you to select multiple entities.

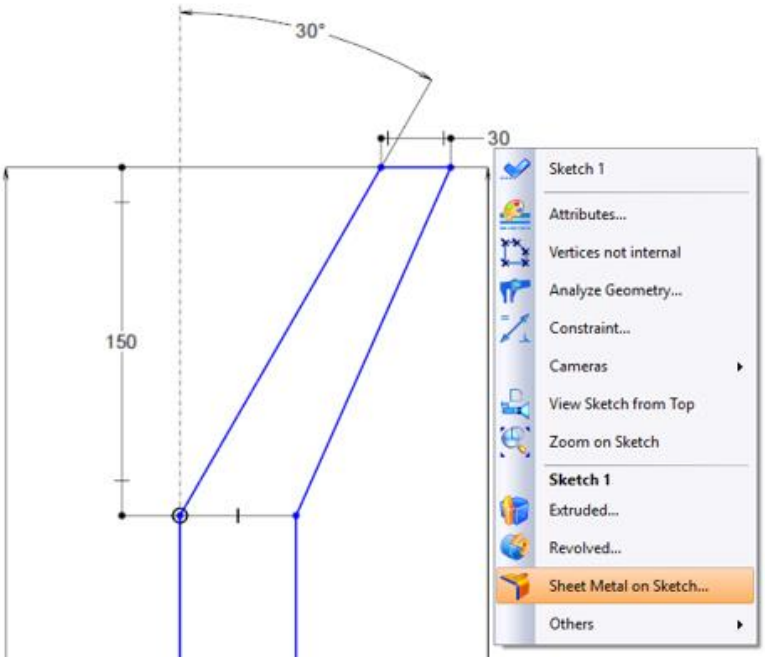
- Finish constraining the sketch as shown below.



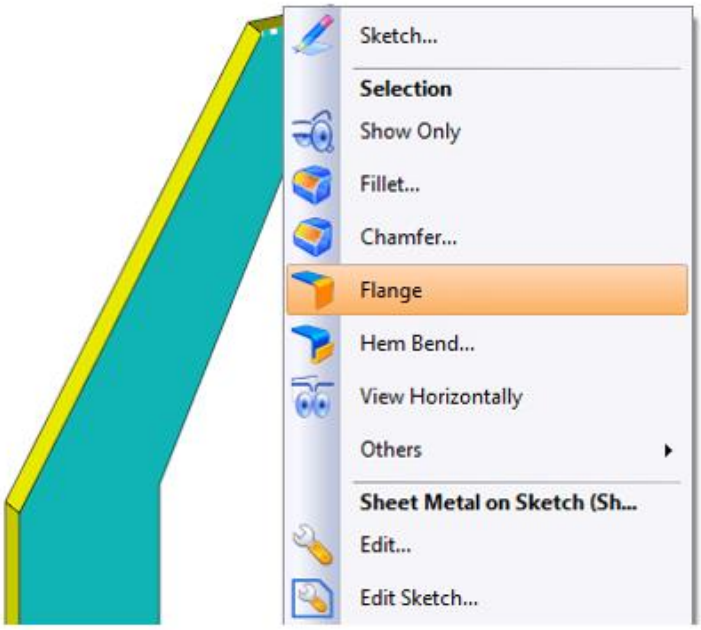
The sketch should appear in blue to indicate that it is fully constrained. If this is not the case, it means that constraints are missing.

The sketch is positioned on the XY plane by default. By convention, in the building industry, the Z axis often indicates verticality.

- From the **2D Sketch** tab, select the **Position Sketch** command and select **Absolute XZ Plane** as the **support plane** and click on to **confirm** the operation.
- Right-click in the graphics area and select the **Sheet Metal on Sketch** command.

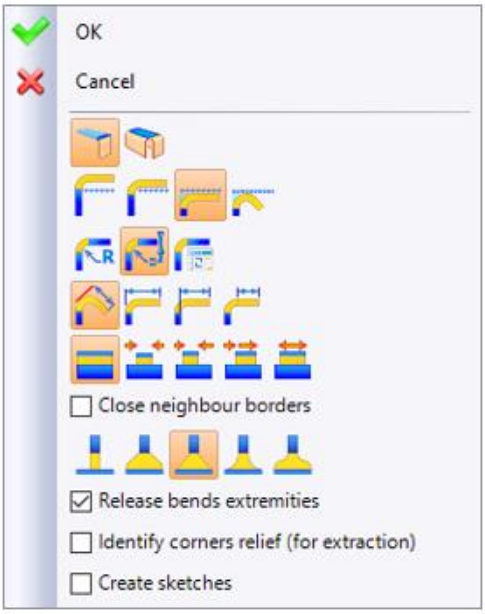


- Enter a sheet metal **thickness** of *10mm*. As a reminder, it is also possible to pull on the yellow arrowhead to modify the value graphically.
- Right-click on the following edge and select the **Flange** command.



- Enter an **angle** of *90°* and a **length** of *60mm*.

- Right-click in the graphics area and select the following options.



The  icon allows you to define the position of the bend in relation to the selected edge (in this case, below the edge).

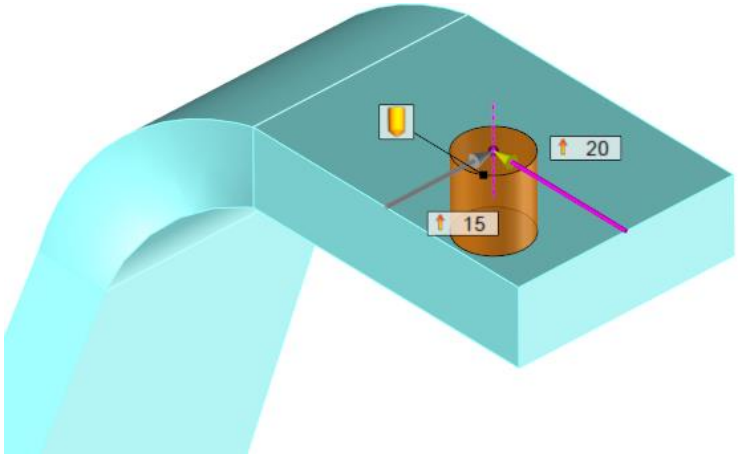
The  icon allows you to select the bending radius, which in this case, is equal to the thickness of the sheet metal part.

The  icon allows you to select the type of bend transition, which in this case, is the line type.

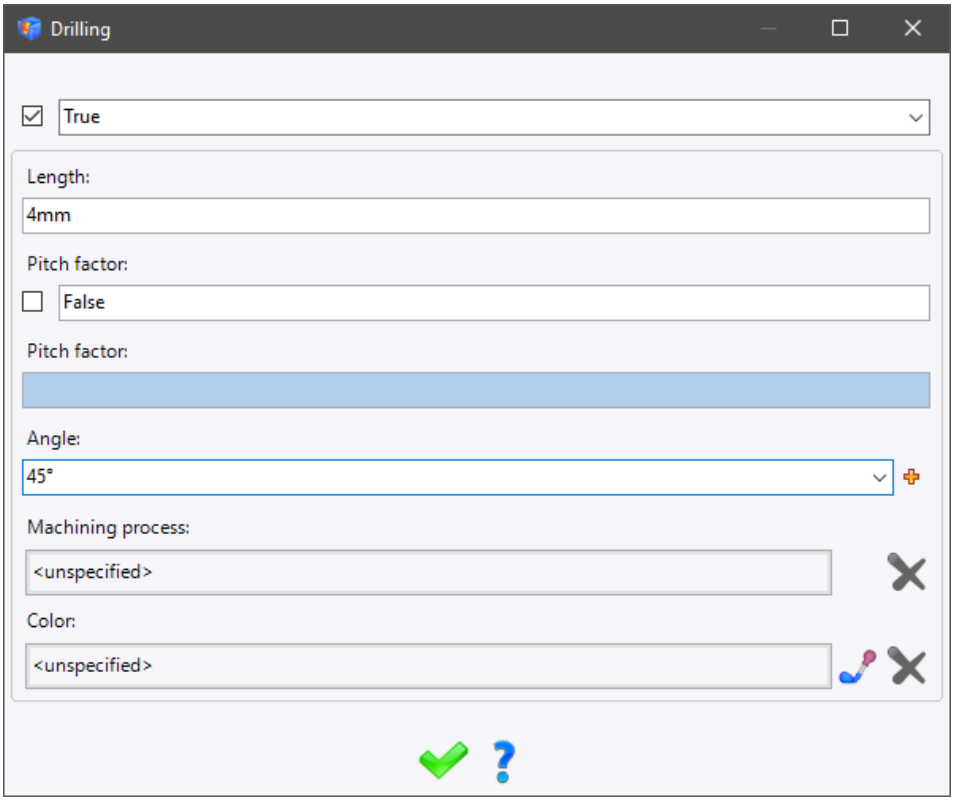
All of these options are also available in the **Flange** dialog box.

Note: To find out more about the different options available, you can display the **TopSolid** online help by clicking on the  icon or by using the **F1** keyboard shortcut once the command has been selected.

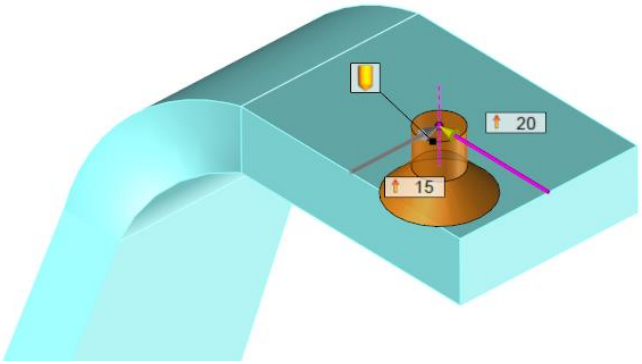
- Right-click in the graphics area and then click on  **OK**.
- On the top face of the post, position a *7mm* diameter drilling as shown below.



- In the **Drilling** dialog box, click on the **In/Out Counter Sinking** icon.
- Click on the **Unspecified** button in the **Out counter sinking** field, check the **Out counter sinking** box to change the value to **True**, then enter a **length** of *4mm*.



- Click on to **confirm** the operation.

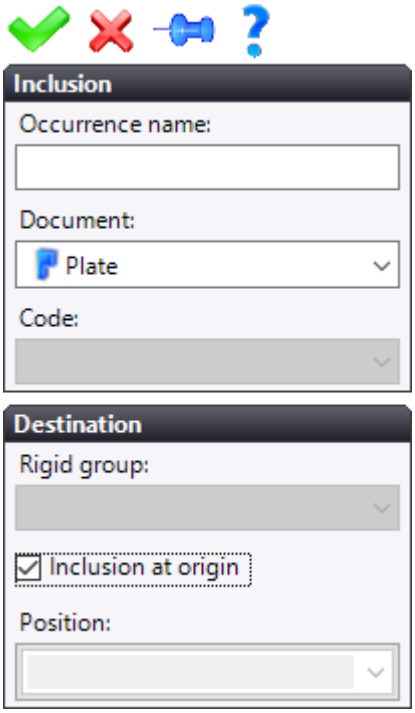


- Change the color of the part by right-clicking on it and selecting the **Attributes** command. Select the color light blue.
- **Save** the plate by clicking on the icon or by using the **Ctrl + S** keyboard shortcut.

Creating an assembly

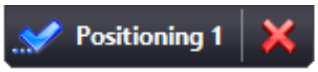
The goal now is to assemble the plate and the post in a single document that we will be able to use for the railings.

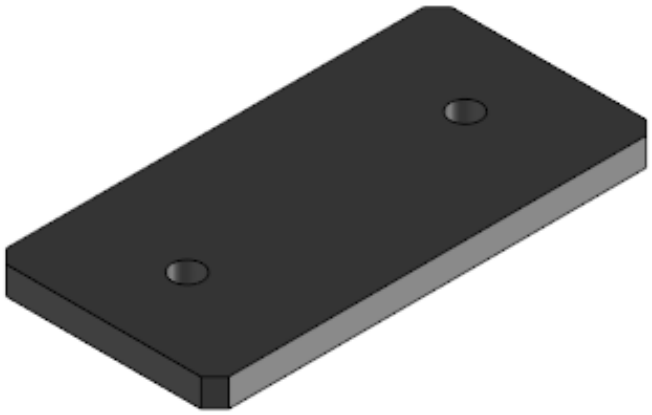
- From the Project tree, right-click on the *My Project* name and select the **Assembly** command.
- Right-click on the assembly document you have just created and select the **Properties** command.
- Click on the **Edit** button. In the **Description** field, enter *Post + Plate* and then click on to **confirm** the operation.
- From the **Assembly** tab, select the **Inclusion** command.
- Select the **plate** part and check the **Inclusion at origin** box.



- Click on to **confirm** the inclusion.

TopSolid switches to the positioning context in which you can add positioning constraints.



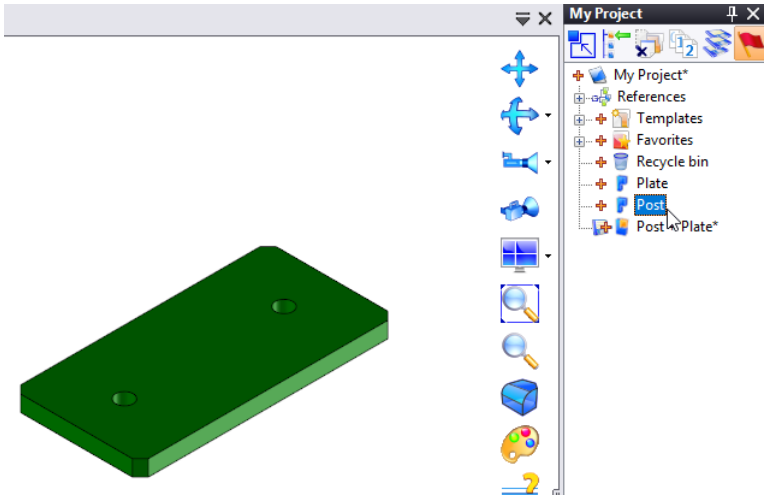


Note: In our exercise, the plate is the first part of the assembly. Therefore, **TopSolid** fixes this part at the origin. You can remove the fixity by right-clicking on the part and selecting the **Unfix** command.

- **Confirm** the positioning context either by clicking on the **Positioning 1** button at the top of the screen or by right-clicking in the graphics area and selecting **Positioning 1**.

Warning: The icon cancels the positioning and therefore the inclusion of the plate.

- Insert the post by dragging the *Post* part document into the graphics area of the assembly.

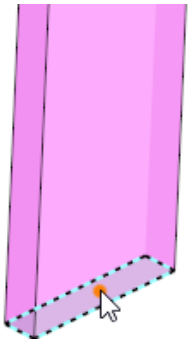


TopSolid switches once again to the positioning context. The color code is the same as for the sketches:

- the underconstrained elements are displayed in pink;
- the fully constrained elements are displayed in blue;

The **Constraint** command is automatically enabled. It allows you to add positioning constraints semi-automatically.

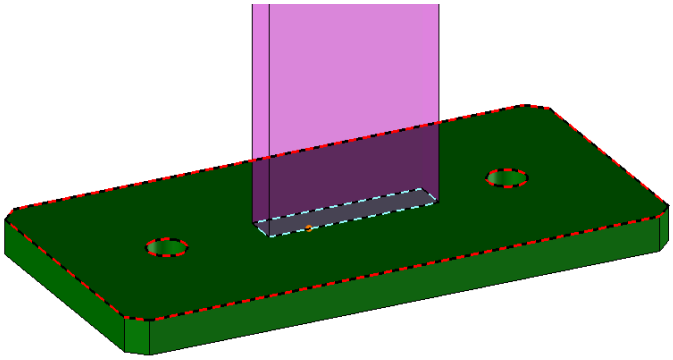
- Select the lower face of the post as the **source** by using the rotary picking technique. To do this, and to avoid rotating the view, position the cursor approximately on the face, hold down the left mouse button and click several times with the right mouse button. **TopSolid** then successively offers the geometries available under the cursor. Once the correct geometry has been selected, release the left mouse button.



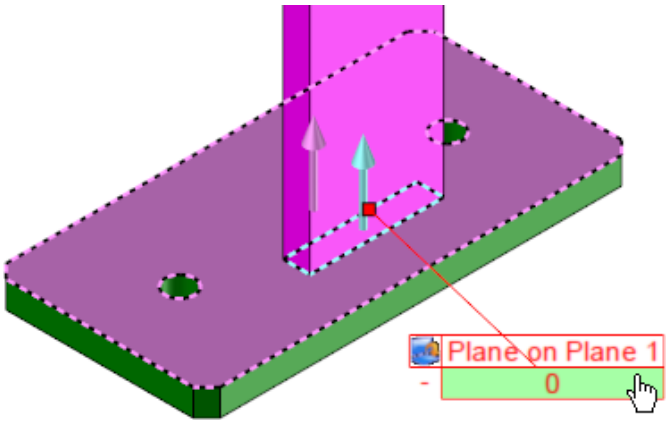
Summary of rotary picking:

- hold down: Select the first entity under the cursor.
- + : Scrolling through the available entities.
- release: Final selection of the displayed entity.

- Select the top face of the plate as the **destination**.



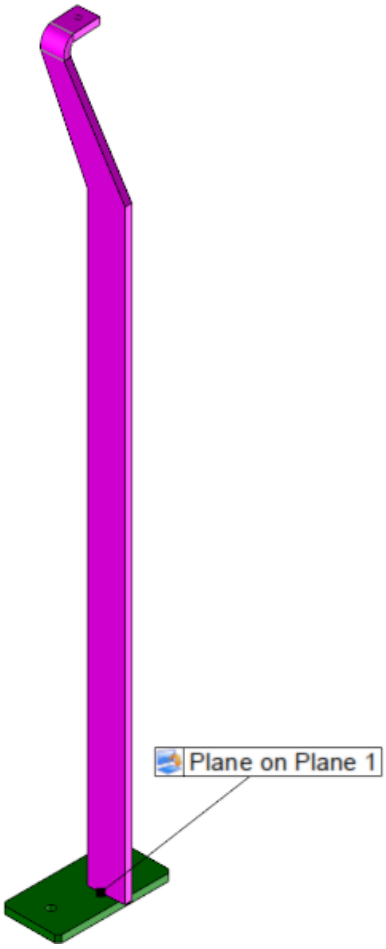
TopSolid dynamically positions the post on the plate and adds a **Plan on plan** constraint.



- Hover the mouse over the label. From the label, you can reverse the constraint (the post will be upside down), add an offset value or edit the label via the contextual menu by selecting the **Edit** command.

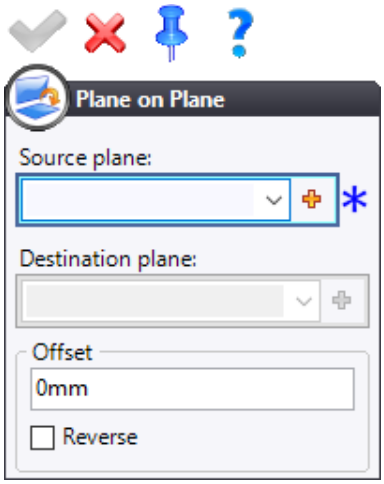
Note: To move the part, you simply have to click on it, hold down the left mouse button and move the mouse. To rotate the part, you simply have to hold down the **Alt** key on the keyboard, click on the part, hold down the left mouse button and move the mouse.

- Exit the **Constraint** command and orient the post as shown below using the **Alt** key.



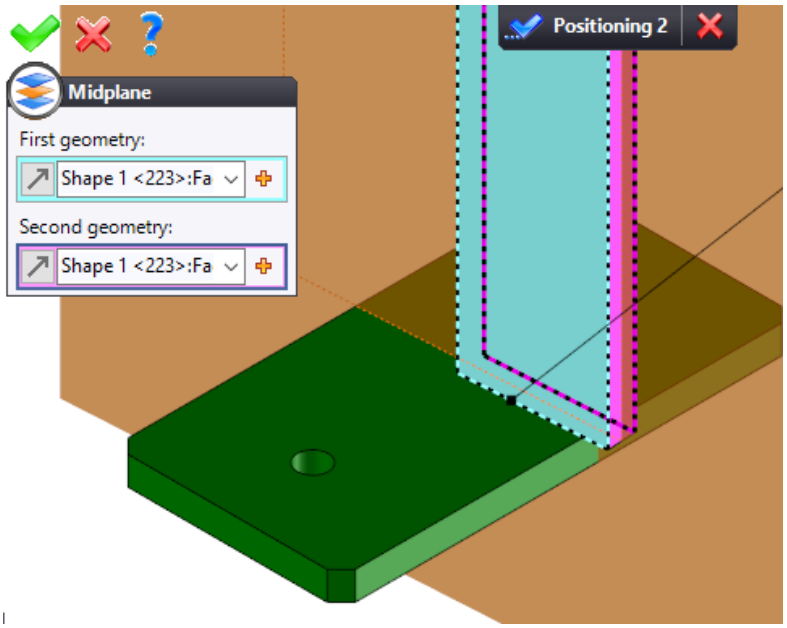
We will now position the post in the middle of the plate. To avoid doing calculations or taking measurements, you can create midplanes on-the-fly.

- From the **Assembly** tab, select the **Plane on plane** command.



TopSolid asks you to select a source plane. The icon allows you to create elements on-the-fly, including planes in this case.

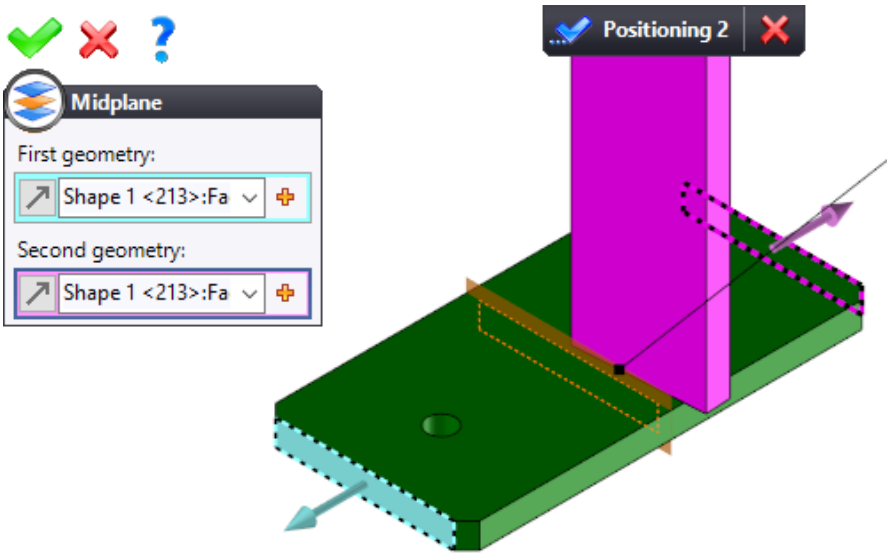
- Click on the icon and select **Midplane**. Select the two external planes of the post as shown below.



- Click on to **confirm** the midplane.

TopSolid then asks you to select the destination plane.

- Click on the icon and select **Midplane**. Select the external planes of the plate as shown below.

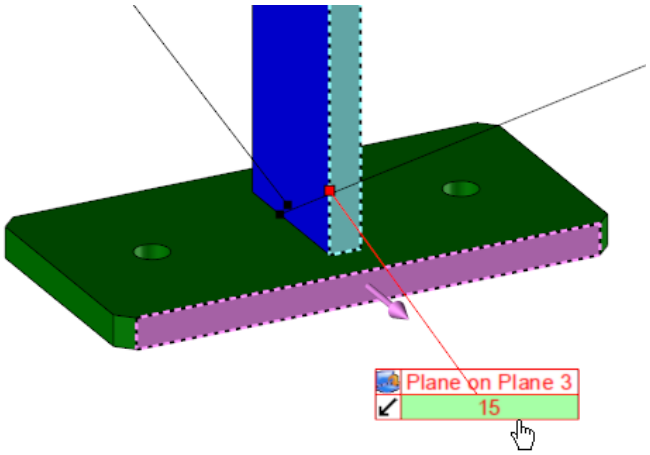


- Click on to **confirm** the midplane.

TopSolid adds a new constraint label and positions the post in the middle of the plate. If the plate is later modified, the post will still remain in the middle.

- Add a last **plane on plane** constraint on the front face of the post by entering an offset value of *15mm*. Click on the arrow to invert the gap distance if necessary.

Note: In this case, you can also create mid-planes.



The post should be displayed in blue to indicate that it is fully constrained.

- **Confirm** the positioning context by clicking on the Positioning 2 button or via the contextual menu.
- **Save** the assembly document (**Ctrl** + **S**).

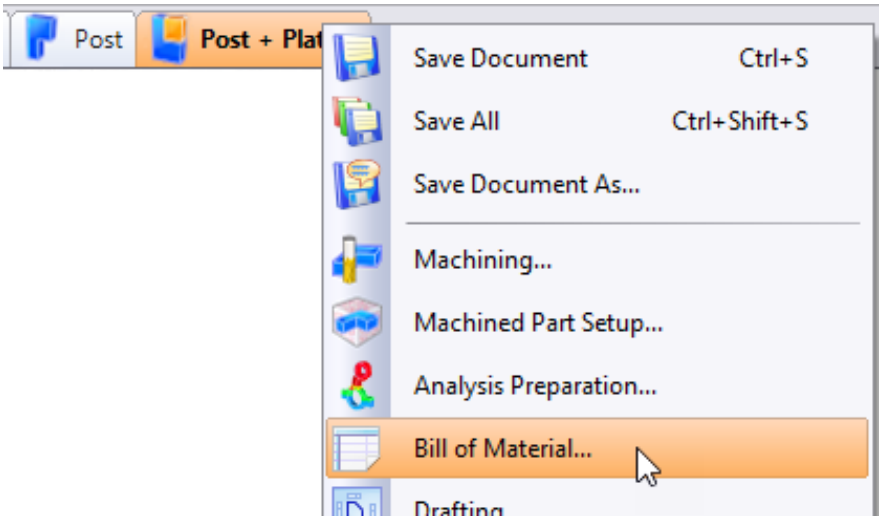
Congratulations, your first assembly is complete! We will now see how it can be used in required documents for its production.

Bill of Materials

As a reminder, the bill of materials list all the parts/sub-assemblies of an assembly. This list is fully customizable (filters, columns, level of detail, etc.) They also allow you to perform quick actions such as:

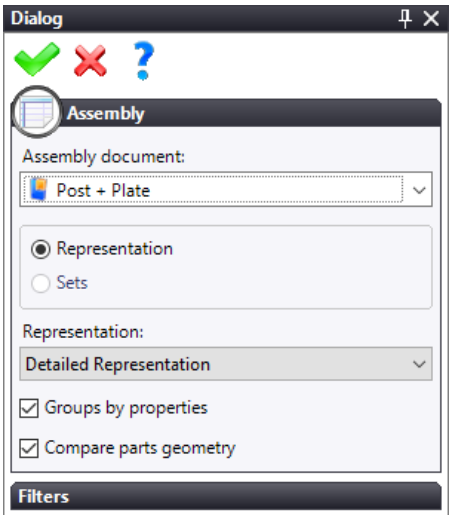
- modify the properties (name, reference, etc.);
- change material, coating and finishing;
- launch an automatic part numbering;
- create multi-drafts;
- create multiple unfoldings;
- generate printings.

- Right-click on the *Post + Plate* assembly document's tab and select the **Bill of Material** command.



- Select the **Multi-level** template from the **Steel Standard Templates - United States** folder and click on to **confirm** the operation.

TopSolid creates the bill of materials and displays the following dialog box.



- Click on to **confirm** the operation.

You should obtain the following result.

ID.	QTY	DESCRIPTION	MATERIAL	MASS	PICTURE
	1	Post + Plate		4.73kg	
1	1	Plate	Steel	0.99kg	
2	1	Post	Steel	3.75kg	

- Click on the first row of the bill of materials.

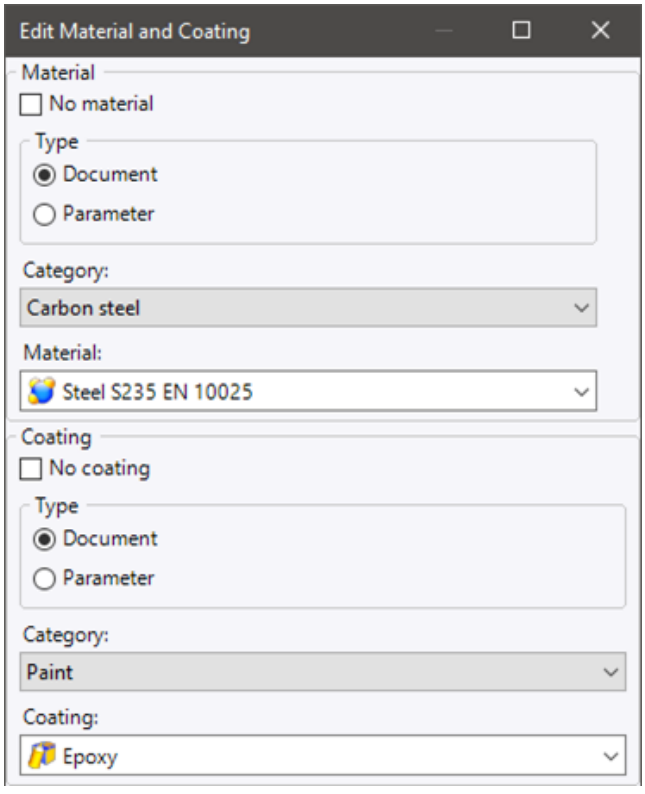
In the lower part of the screen, **TopSolid** displays the details of the assembly. The name given to the document is clearly visible. This name is the one that appears in the PDM and can be different from the description that rather appears in the bill of material, in the plans, etc.
Each of these fields can be parameterized, allowing you to automate their filling.

Here are a few examples:

- **Name** = **Part Number** - **Description** (1920 - 01 - 001 - Post)
- **Name** = **Description** - **Material** (Post - S235)
- **Description** = **Name** (Post)

In this case, the document template used automates the **Name** field. By default, it displays the description. If you add a part number, it will display **Part Number** - **Description**.

- On the **Plate** line, right-click in the **Steel** field of the **MATERIAL** column and select the **Edit Material and Coating** command.
- In the **Material** field, select **Steel S235 EN 10025**.
- Uncheck the **No Coating** box, select the **Paint** category and the **Epoxy** coating.



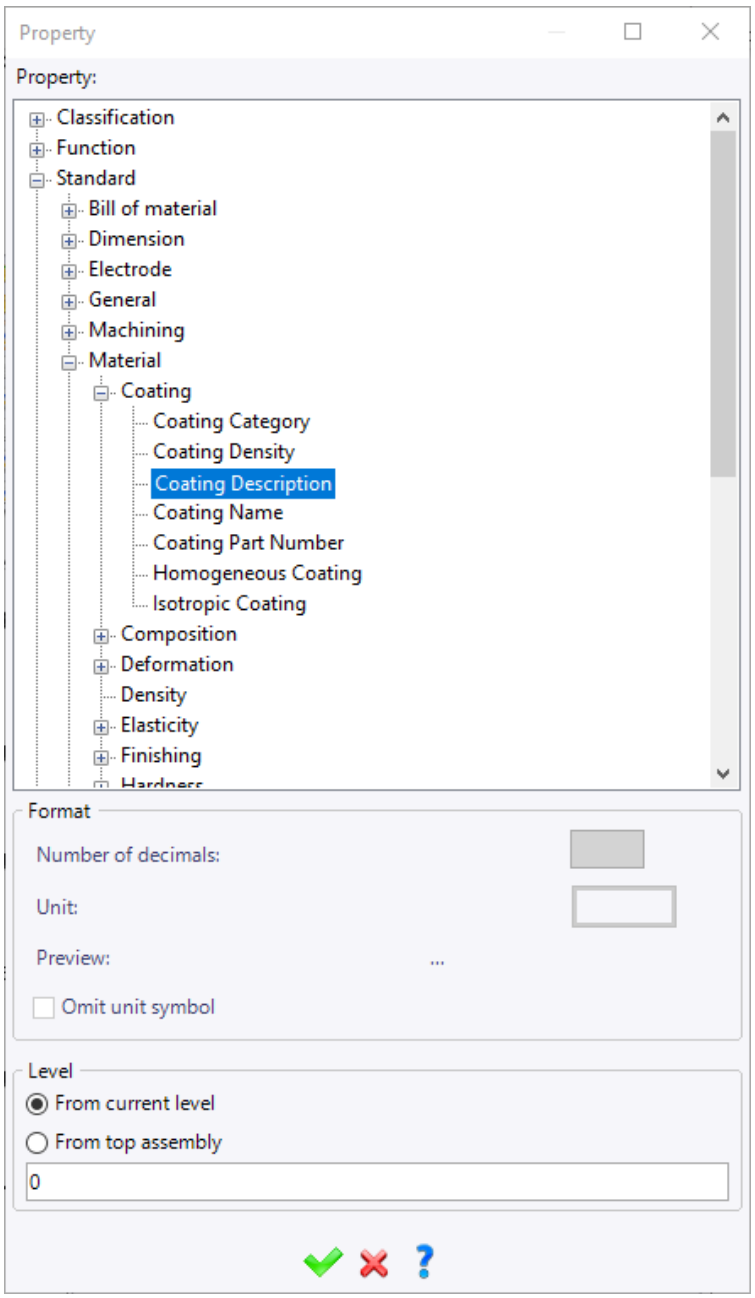
- Click on  to **confirm** the operation.

Note: You can define the materials from the part itself, but also from the assembly in which it is located. We will see that later.

- Repeat the manipulation on the post.

In this BOM template, the **Coating** column does not exist. You can add additional information at any time.

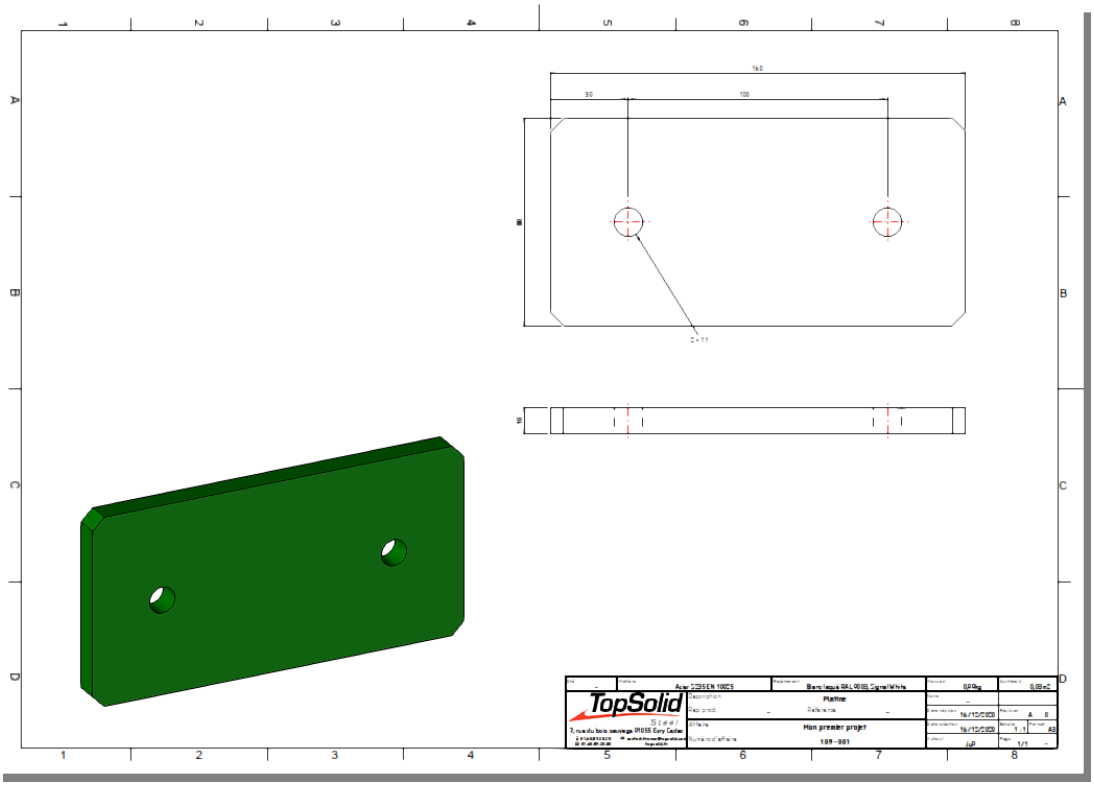
- To do this, right-click in the **MASS** column and select **Add column**.
- Select **Standard** > **Material** > **Coating** > **Coating Description** properties.



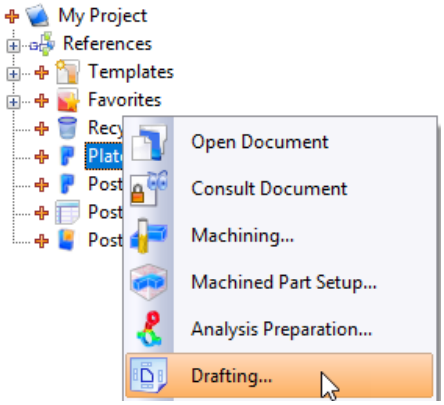
- Click on to **confirm** the operation.
- **Save** the bill of materials document (**Ctrl** + **S**).

Drafting a part

We will now draft the post as well as the plate.



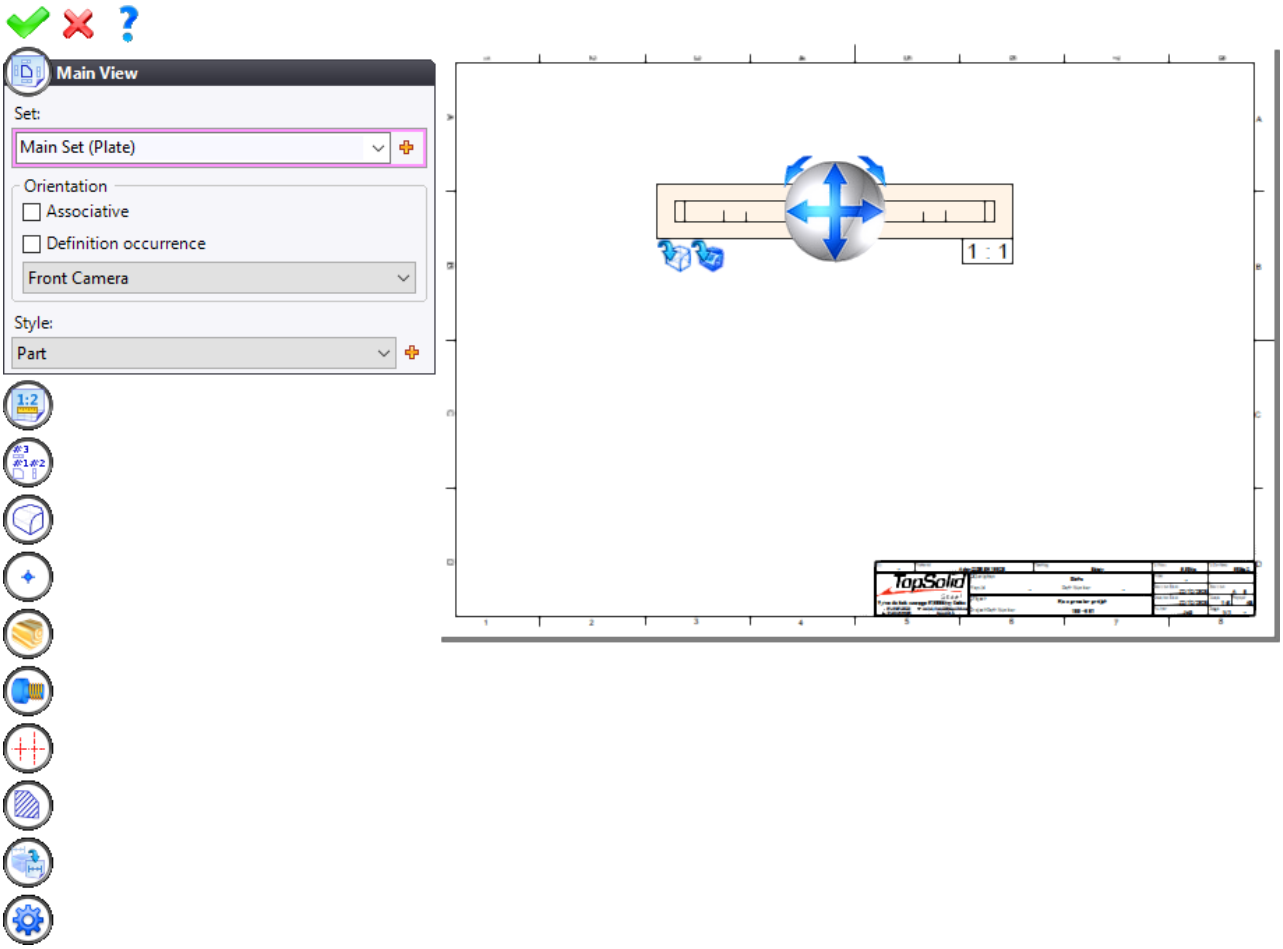
- Right-click on the *Plate* part document's tab and select the **Drafting** command. If the *Plate* document is not open, you can right-click directly on the *Plate* document in the Project tree to access the command.



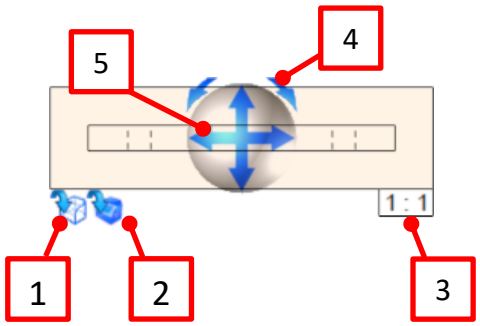
- Select the **Part A3 ISO Landscape** template from the **Steel Standard Templates - United States** folder.

TopSolid creates the drafting and offers to position the front view.

- Click approximately in the center of the plane. You can always reposition the view later.



The following view should appear.



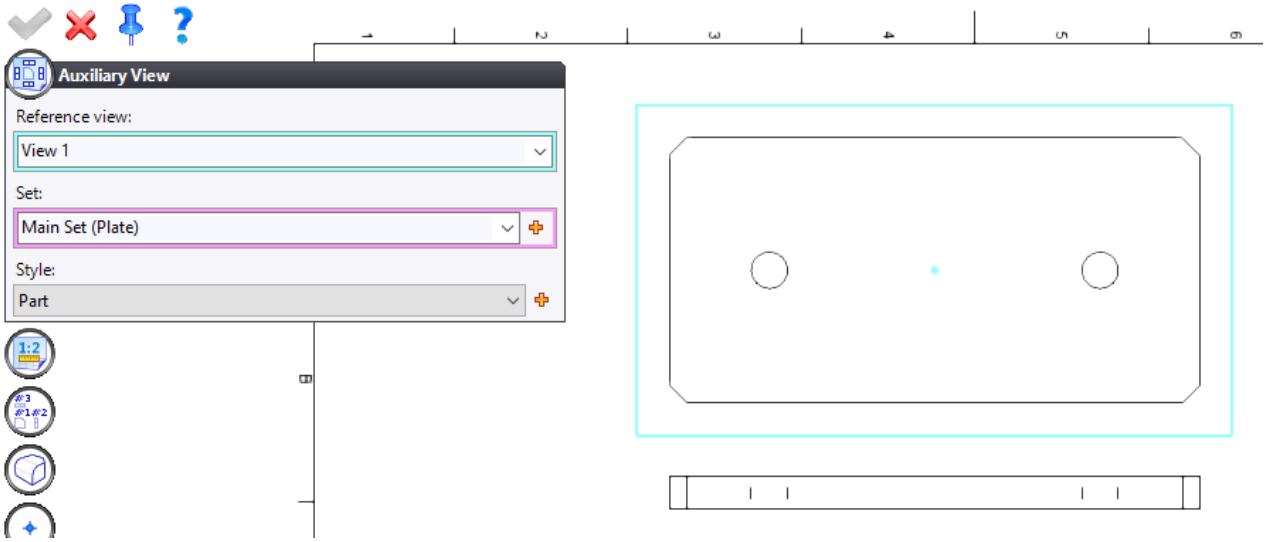
- 1: Allows you to show or hide hidden lines
- 2: Allows you to modify the type of rendering of the part (wireframe, shaded or realistic)
- 3: Allows you to modify the local scale of the view. Only this view will be modified. If the scale of the plane is 1:2 and that of the view is 1:2, then the part is at the scale of 1:4.
- 4: Allows you to rotate the part with respect to the axis at the center of the view
- 5: Allows you to rotate the part with respect to the center of the view

- Click on the lower part of the sphere to rotate the part as shown below.

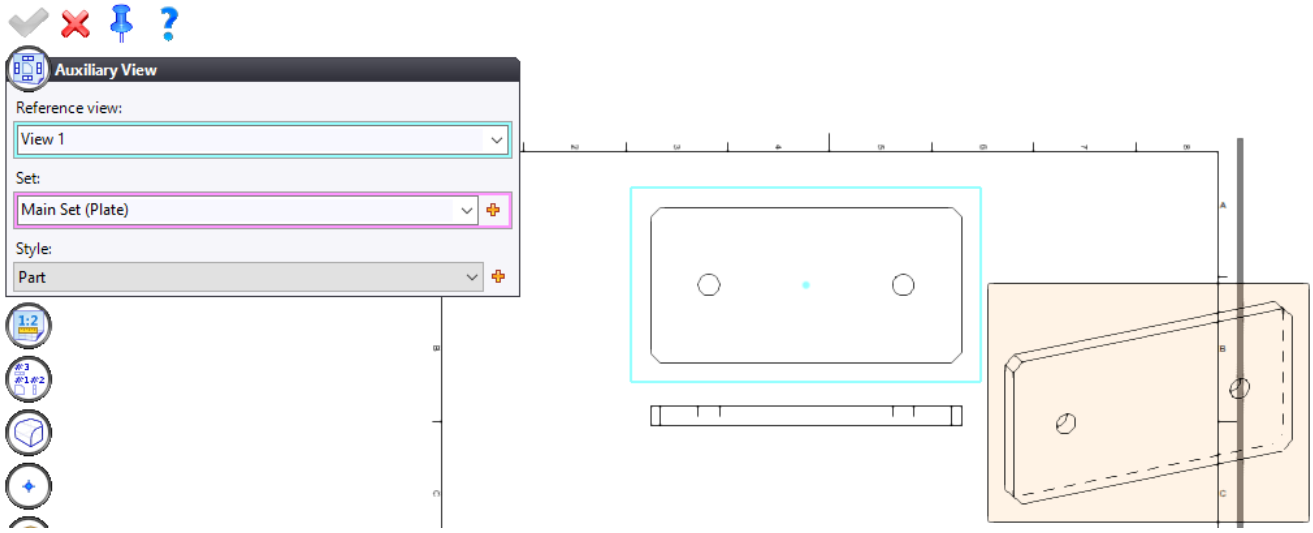
- Click on to **confirm** the main view.

TopSolid automatically launches the **Auxiliary view** command which allows you to create the projection of the first view placed according to the location of the mouse.

- Create the following top view and position it as shown below.



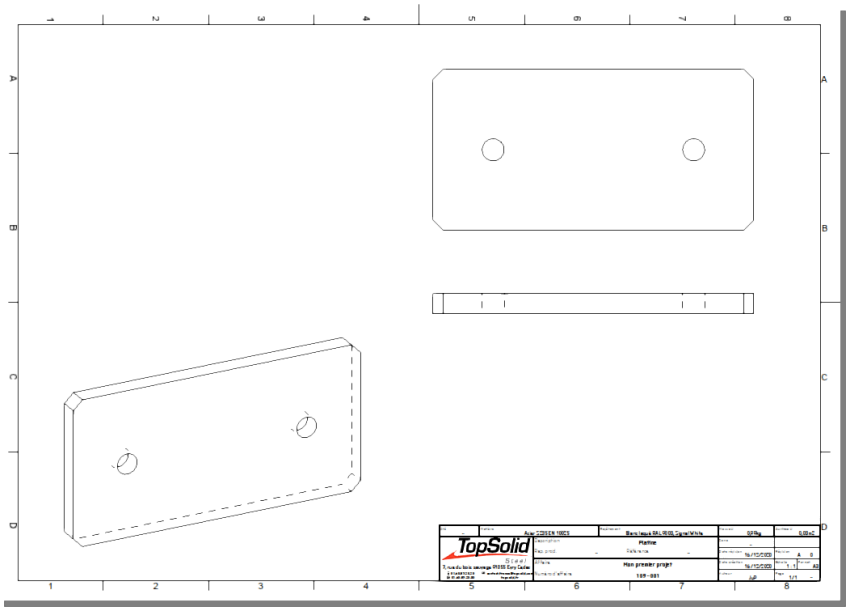
- Create the following isometric view and position it as shown below.



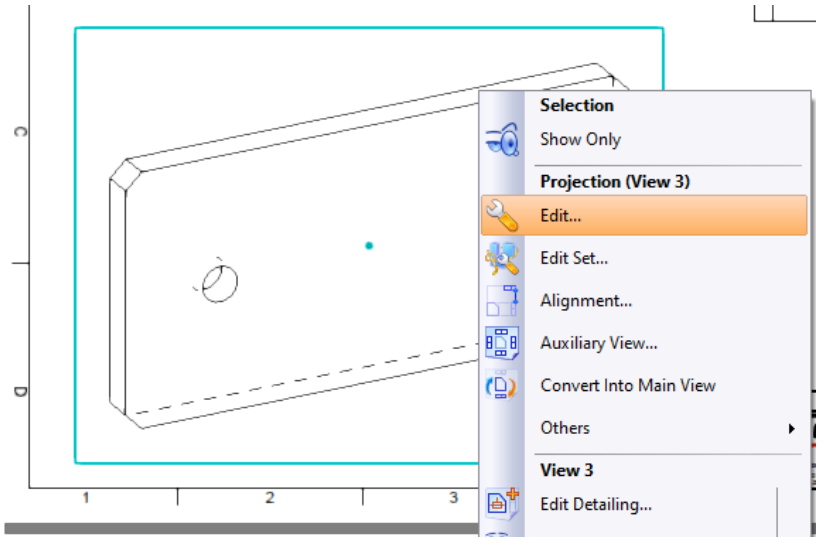
- Exit the command by pressing on the **Esc** key or by clicking on the icon.

Note: If you exit this command before you are finished, you can create these projections by right-clicking on a view and selecting the **Auxiliary View** command.

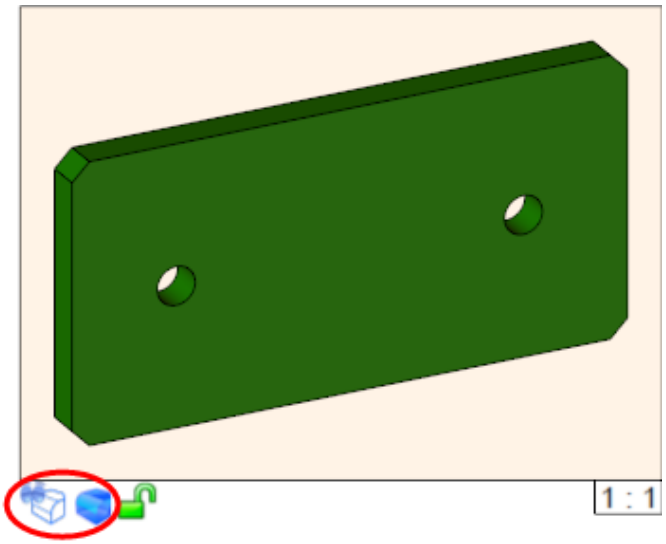
- Drag the isometric view under the other two views as shown below.



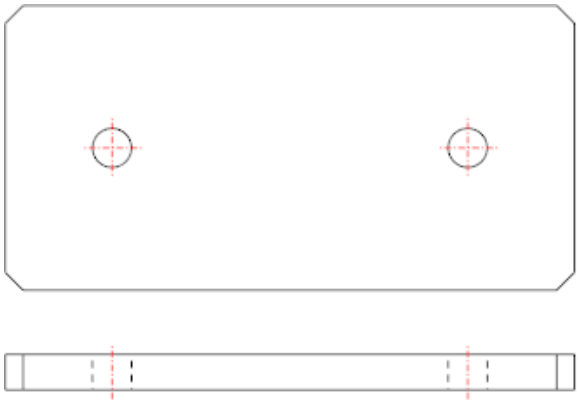
- Right-click on the isometric view and select the **Edit** command.



- Then click on the icons below to hide the hidden edge lines and to show the part in shading render mode.



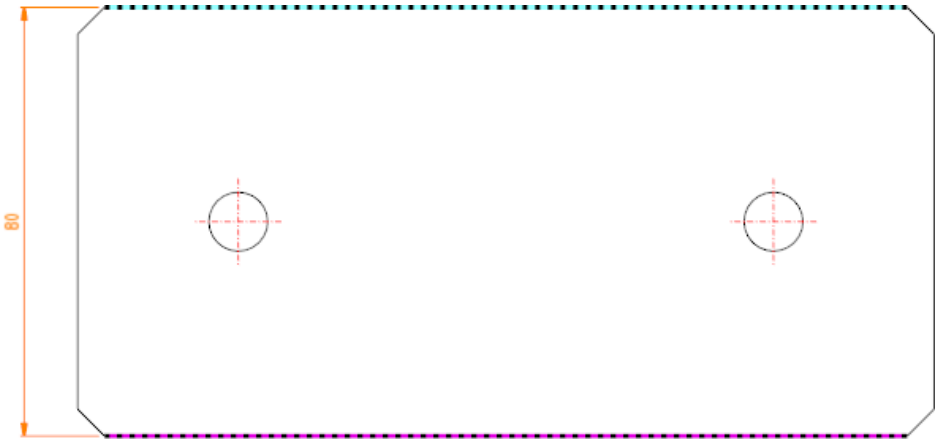
- Add the axes by right-clicking on the front view and selecting the **Automatic Axes** command. Repeat the operation on the top view.



- Right-click in the graphics area outside of the drawing and select the **Dimension** command. Make sure that the pin on the dialog box is enabled.

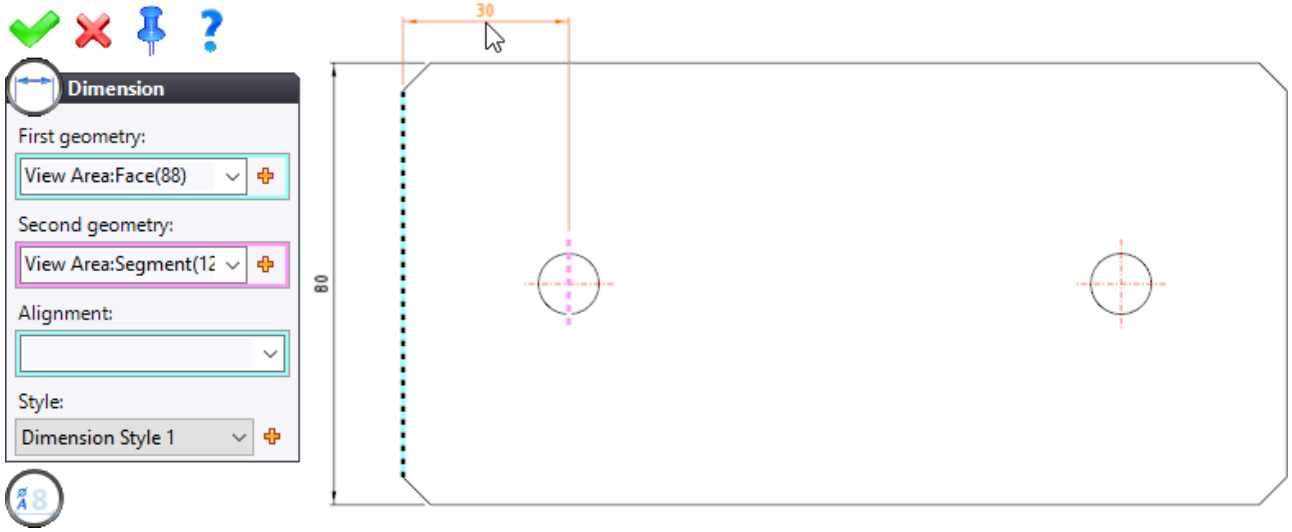
Reminder: The first left click of the mouse allows you to select the entity to be dimensioned. The next left click allows you either to position the dimension, or to set the dimension in relation to another entity.

- Select the upper and lower edges of the plate successively and then position the dimension to the left of the part.

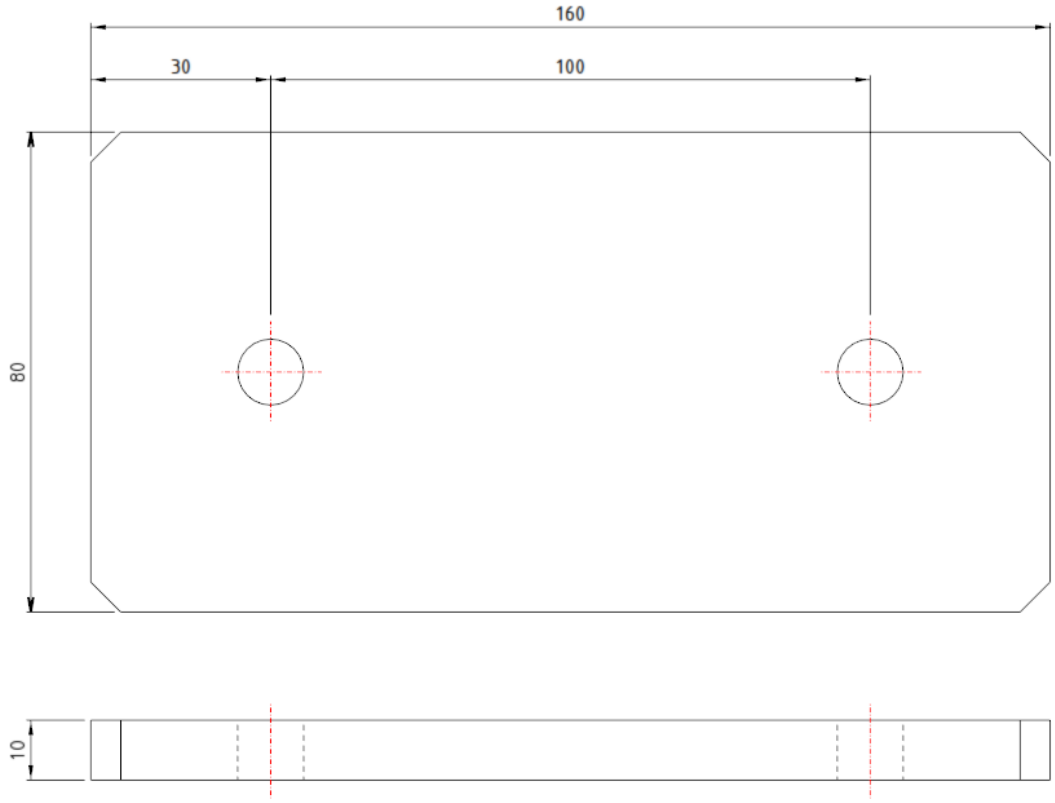


Once positioned, the dimension is displayed in black.

- Select the left side of the plate and then the first drilling.



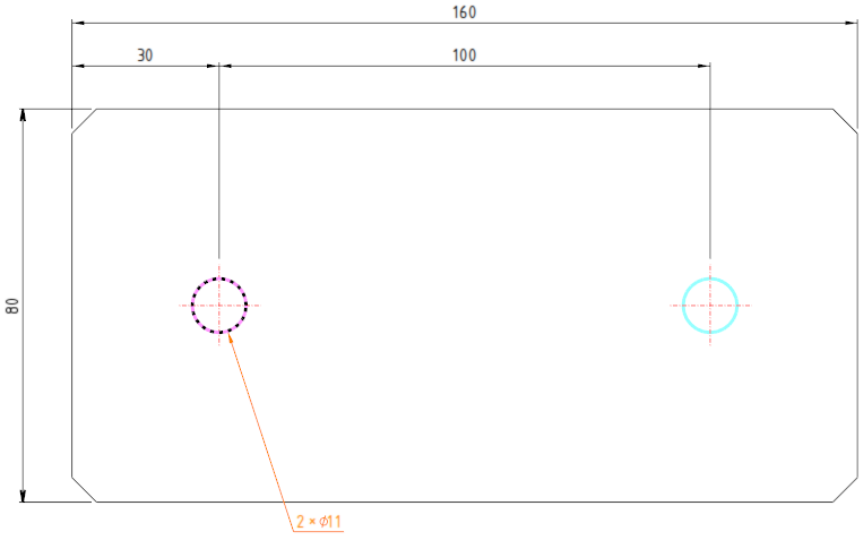
- Add the following dimensions.



- **Exit** the **Dimension** command.

All of these dimensions can be created automatically. We will see this later in the training course.

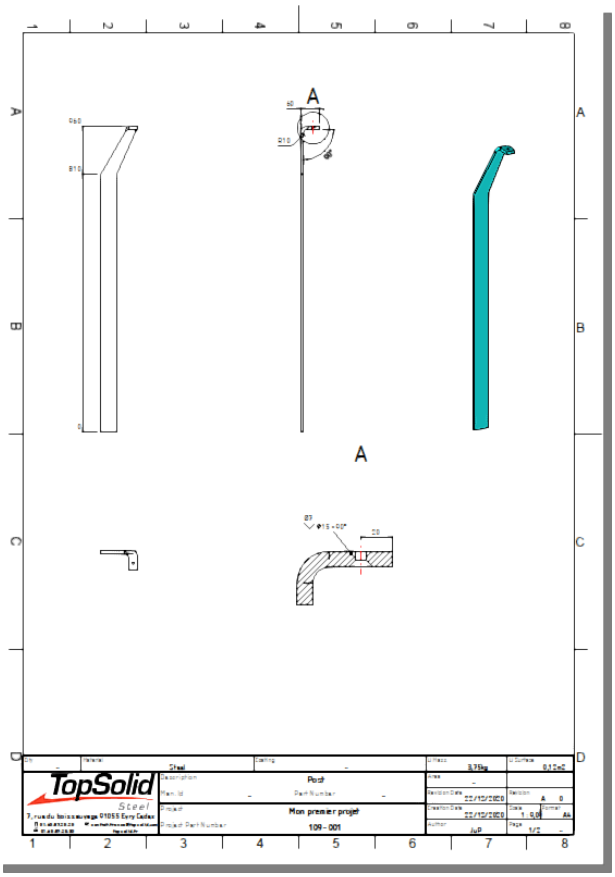
- From the **Detailing** tab, select the **Drilling note** command and then select one of the drillings.



TopSolid automatically groups similar drillings together. As for some dimensions, these annotations can be positioned automatically.

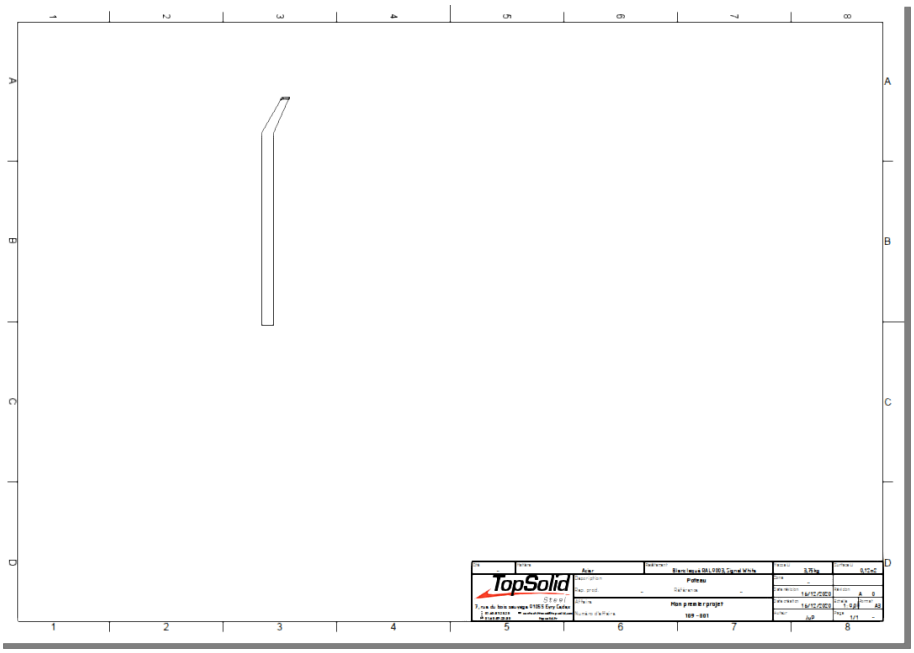
- **Save** the drafting document (**Ctrl** + **S**).

We will now repeat all the operations previously created on the post.



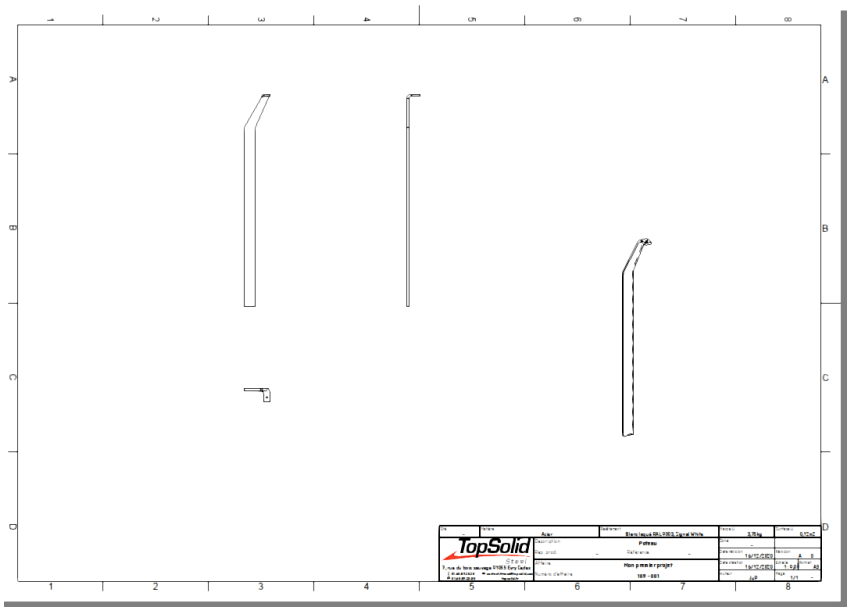
- Right-click on the *Post* part document's tab and select the **Drafting** command. If the *Post* document is not open, you can right-click directly on the *Post* part document in the Project tree to access the command.
- Select the **Part A3 ISO Landscape** template from the **Steel Standard Templates - United States** folder.

TopSolid creates the drafting and offers to position the front view.



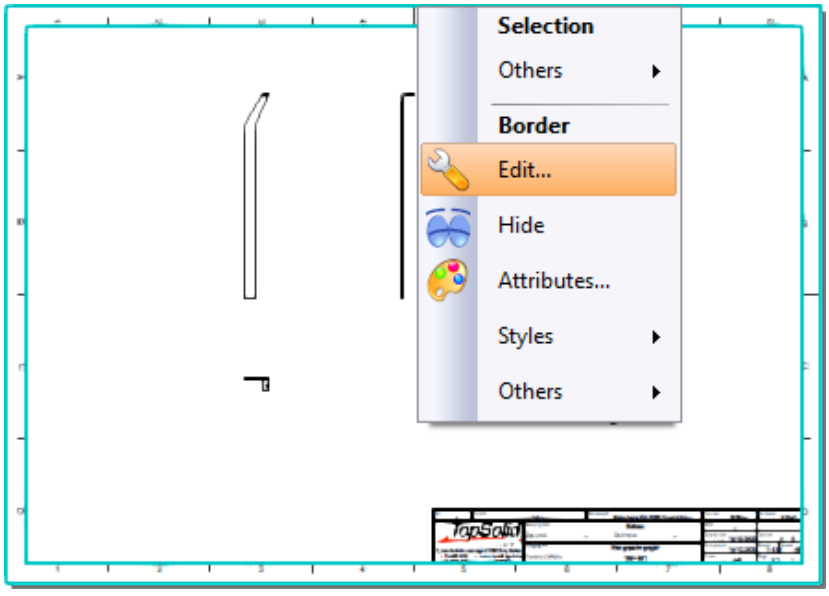
- Click on to **confirm** the operation.

- Add the rest of the projections as shown below.

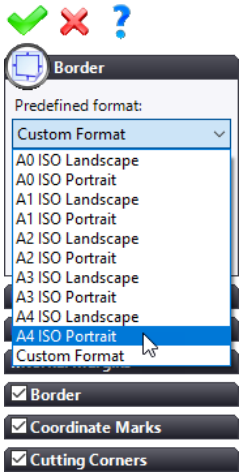


You can change the paper size at any time.

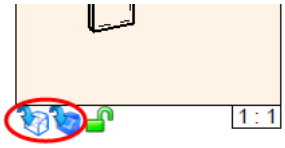
- Right-click on the drawing frame and select the **Edit** command.



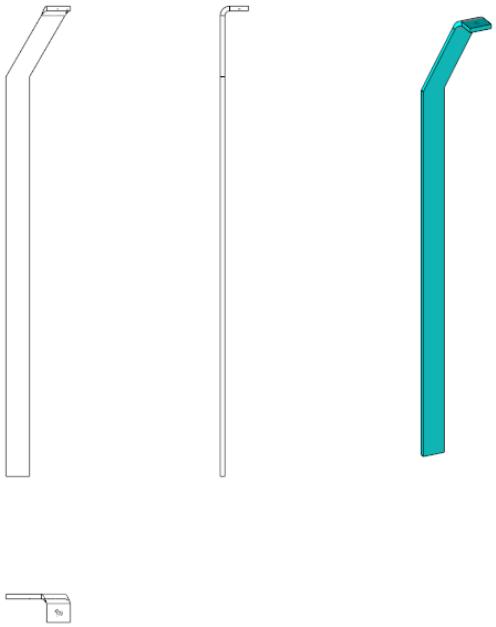
- In the **Predefined format** field, select **A4 ISO Portrait** and then click on to **confirm** the operation.



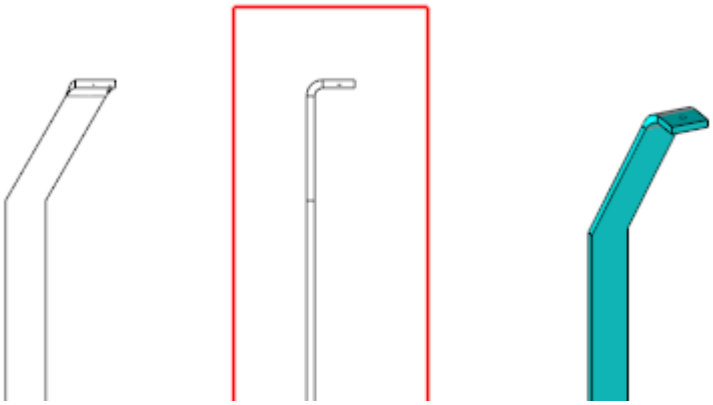
- Reposition the views in the frame.
- Edit the isometric view, then click on the two icons as shown below to hide the hidden edges and switch the view to shading render mode.



- Reposition the isometric view as shown below.



- From the **View** tab, select the **Local Cross Section** command and select the left view.

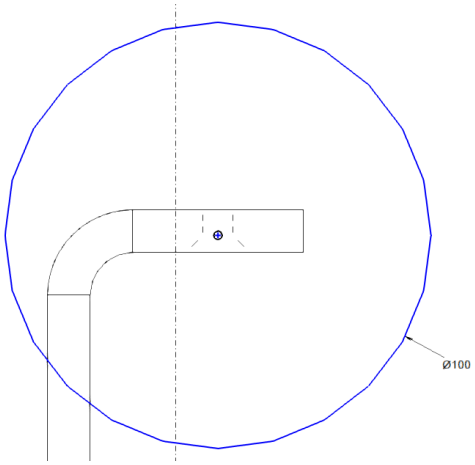


- Click on to **confirm** the operation.

TopSolid asks which area should be cut.

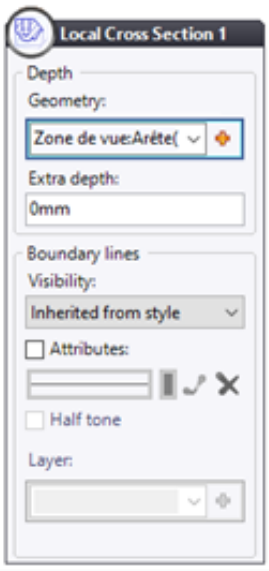
- Exit the **Spline** command using the **Esc** key and select the **Circle** command.

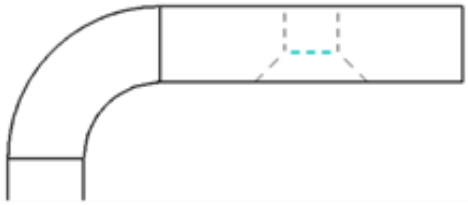
- Create a circle with a diameter of *100mm* and position it as shown below.



- **Confirm** the cutting sketch via the contextual menu or by clicking on the  button.

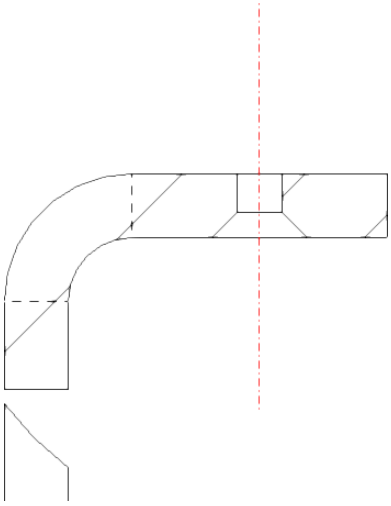
- In the  **Local Cross Section** dialog box, indicate the cutting depth by clicking on one of the edges of the drilling.



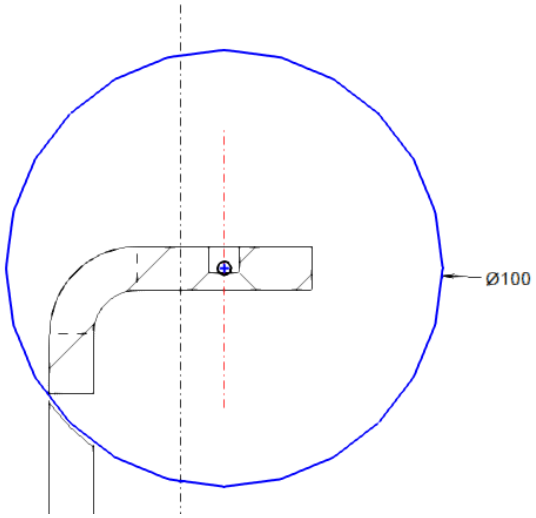


- Click on  to **confirm** the operation.

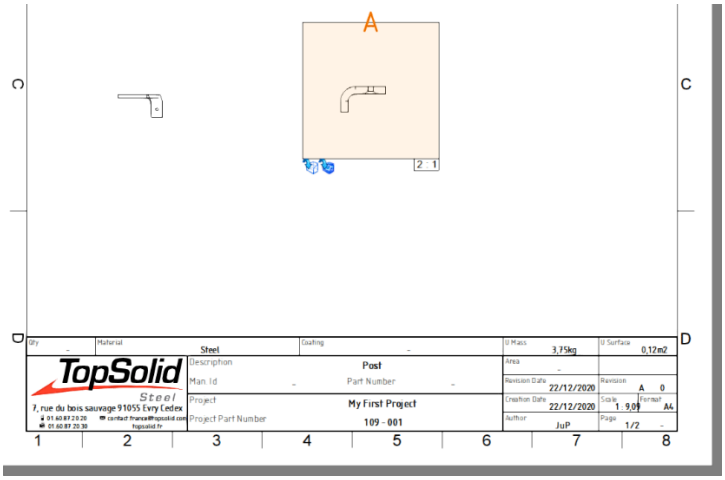
- Add the axes by right-clicking on the same view and selecting the  **Automatic axes** command.



- From the **View** tab, select the **Detail View** command. Select the left view, then draw a circle as shown below.



- **Confirm** the detail sketch.
- Position the view under the isometric view.

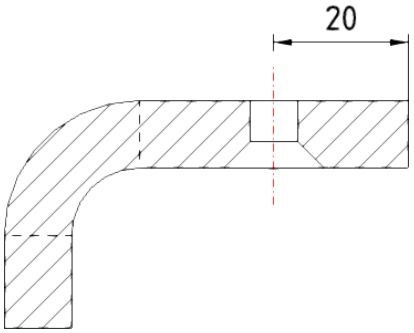


- Adjust the scale factor. To do this, double-click on the label of the view and enter *5*.

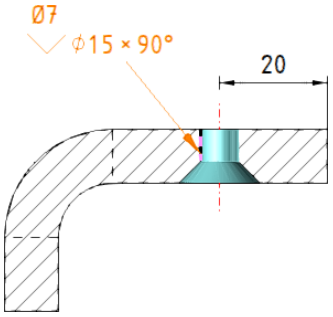
- Click on to **confirm** the operation.

Note: This scale factor will be added to that of the drawing.

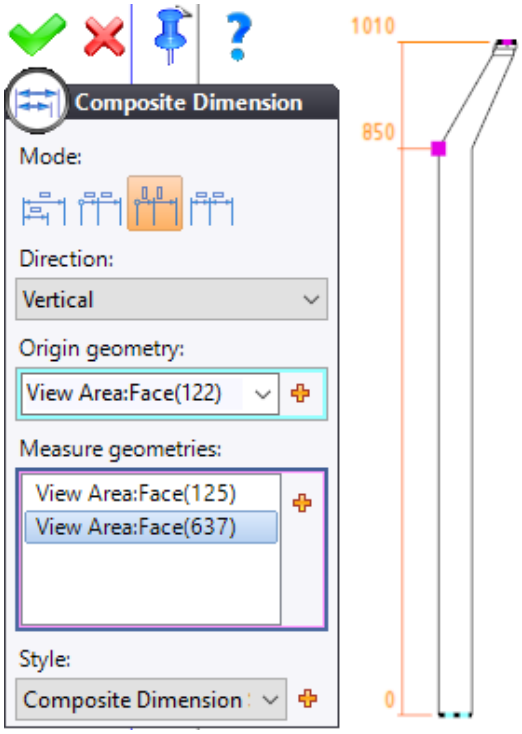
- Add the **automatic axes** to the detail view.
- Add the following **dimension**.



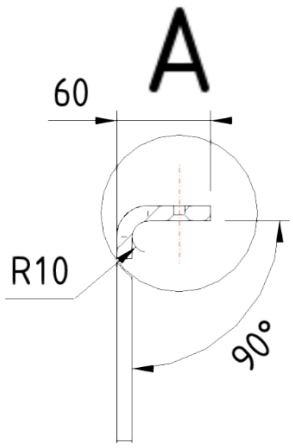
- From the **Detailing** tab, add a **drilling note** on the hole.



- From the **Detailing** tab, select the **Composite Dimension** command. This type of dimensioning allows you to set several dimensions from the same reference.
- Select the **Ordinate** mode and the **Vertical** direction, then on the face view, select the base of the post as the **origin geometry** and the two elements as shown below as the **measure geometries**.



- Add the following three **dimensions** on the left view.

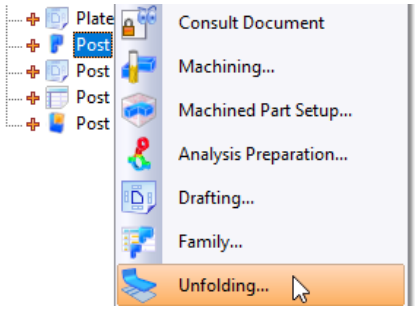


- **Save** the drafting document (**Ctrl** + **S**).

Unfolding

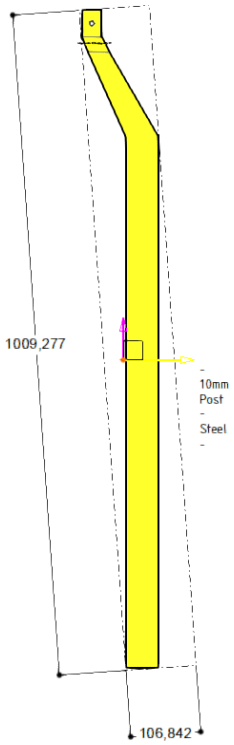
We will now unfold the post and integrate it into its detail drafting.

- Right-click on the *Post* part document's upper tab and select the **Unfolding** command. If the *Post* document is not open, you can right-click directly on the *Post* part document in the Project tree to access the command.



TopSolid creates a new unfolding document.

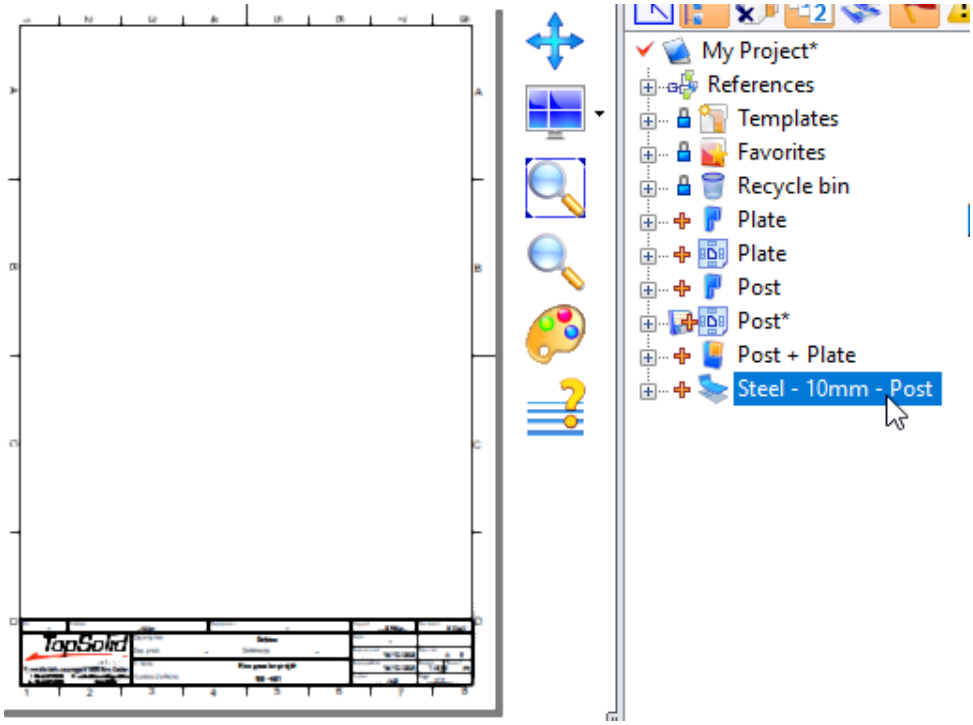
- Click on to **confirm** the unfolding operation.



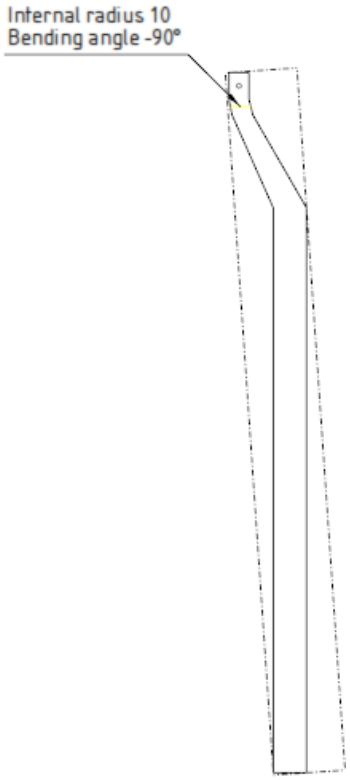
The unfoldings are in 3D by default. You can export them directly in DXF/DWF and DSTV (.nc1) formats or even integrate them in a drawing. You can also customize the unfolding rules used (see *Annexes*).

- **Save** the unfolding document (**Ctrl** + **S**).
- Return to the post drafting document.
- At the bottom left of the graphics area, click on the icon to add a new page.

- Drag and drop the unfolding document into the document.



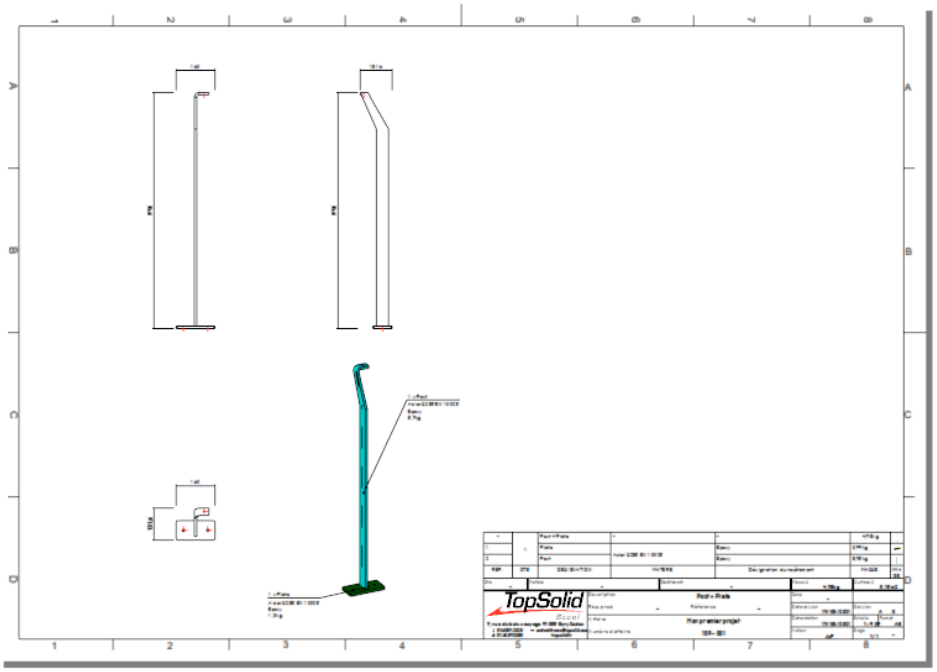
- Position the view in the drawing and click on to **confirm** the operation.
- From the **Detailing** tab, select the **Automatic Bend Notes** command and select the view.
- Reposition the note outside the view if necessary.



- **Save** the drafting document (**Ctrl** + **S**).

Drafting an assembly

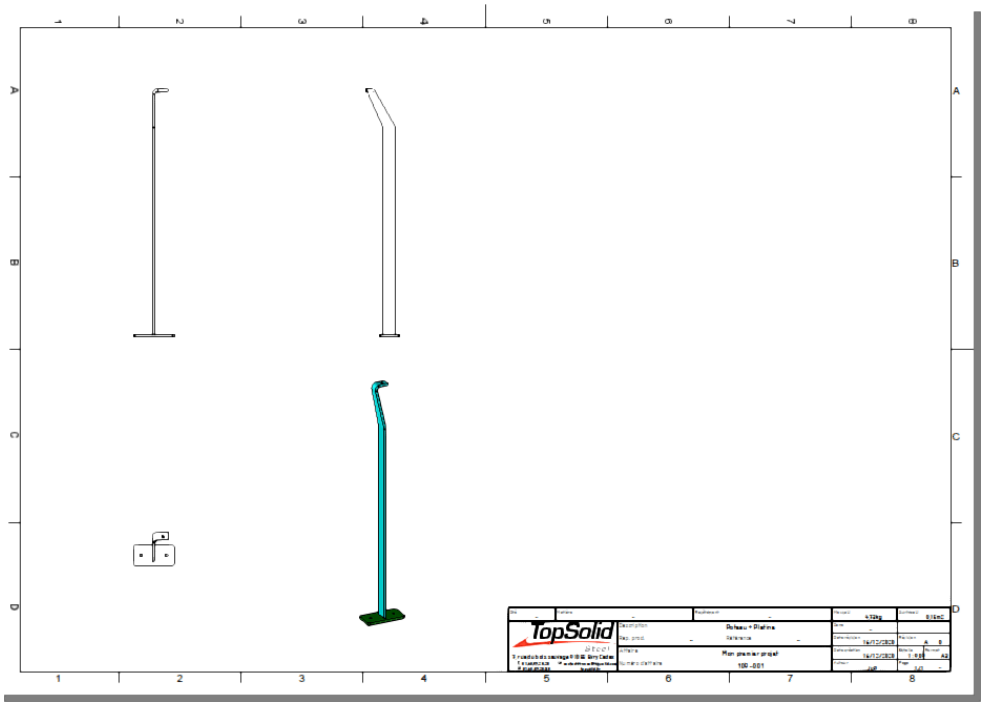
Now that the detail draftings are created and the drafting mechanism is understood, we will be able to draft the *Post + Plate* assembly.



- Right-click on the *Post + Plate* assembly document's tab and select the **Drafting** command.

Reminder: If the assembly document is not open, right-click directly on the *Post + Plate* document assembly in the Project tree.

- Select the **Assembly A3 ISO Landscape** template from the **Steel Standard Templates - United States** folder.
- Position the views as shown below.



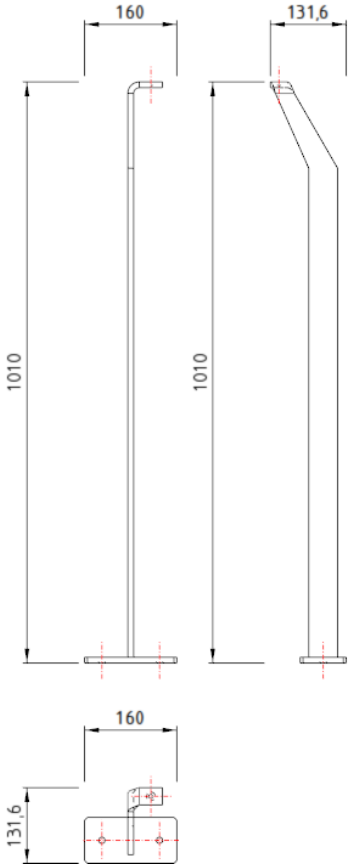
- Add the **automatic axes** on each of the views except the isometric view.

- From the **Detailing** tab, select the **Automatic Dimensions** command.

The following two modes are available:

- **Enclosing dimensions**: Allows you to add the enclosing dimensions along X and Y.
- **Length dimensions**: Allows you to dimension the length of the profiles.
- **Cutting dimensions**: Allows you to dimension the cutting angles of the profiles.

- Select each of the views except the isometric view. Make sure that the pin in the dialog box is enabled; this will avoid having to relaunch the command to switch from one view to another.

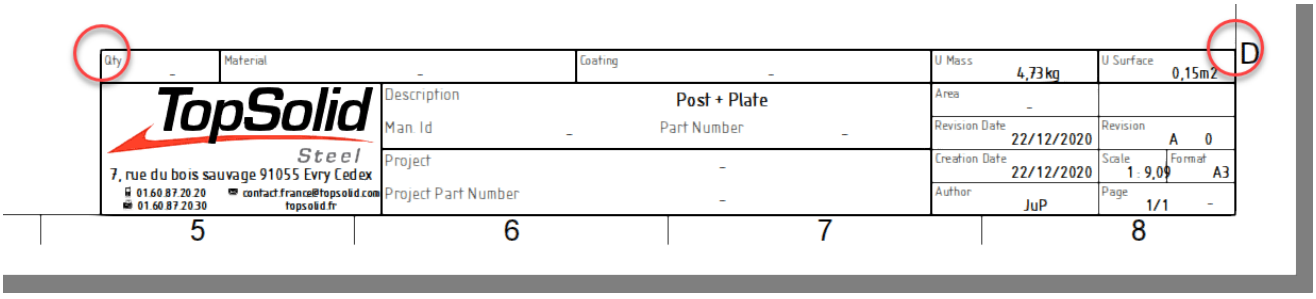


Note: You can hide redundant or uninteresting dimensions. To do this, right-click on the dimension to be hidden and then select the **Hide** command.

- From the **Detailing** tab, select the **Bill of Material Table** command.

Warning: The *Post + Plate* bill of materials document must be opened in order to create the bill of materials table.

- Position the BOM table by clicking either on the two vertices of the title block as shown below (from left to right), or directly on the title block's upper line.



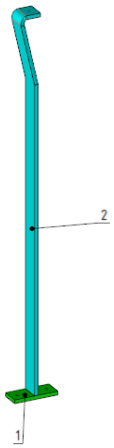
- In the dialog box, check the **Add line for set** box. **TopSolid** will then add a line for the assembly, which will allow you to recover any properties of the assembly (mass, etc.).
- Click on to **confirm** the BOM table.

You should obtain the following result.

ID	QTY	DESCRIPTION	MATERIAL	Coating Description	MASS	PICTURE
-	1	Post + Plate	-	-	4.89kg	
1		Plate	Steel S235 EN 10025	Epoxy	0.99kg	
2		Post		Epoxy	3.91kg	

Qty: -	Material: -	Coating: -	U Mass: 4,89kg	U Surface: 0,15m2	
TopSolid Steel 7, rue du bois sauvage 91055 Evry Cedex 01.60.87.20.20 / 01.60.87.20.30 contact.france@topsolid.com / topsolid.fr	Description: Post + Plate Man. Id: - Part Number: -		Area: -		
	Project: -		Revision Date: 22/12/2020	Revision: A 0	
	Project Part Number: -		Creation Date: 22/12/2020	Scale: 1 : 9,09	Format: A3
			Author: JuP	Page: 1/1	-

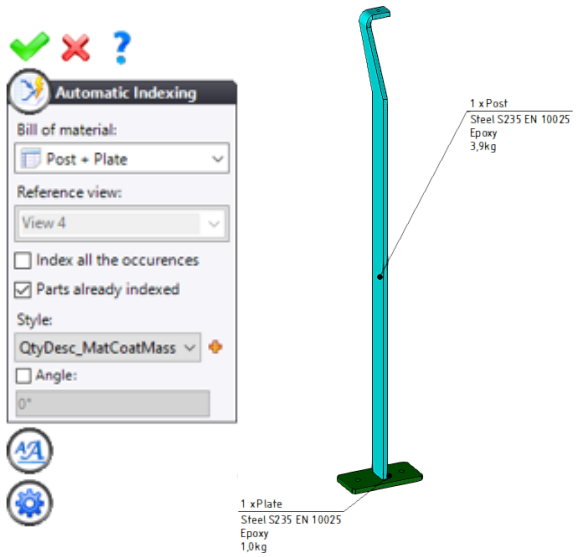
- From the **Detailing** tab, select the **Automatic BOM Index** command. Select the isometric view and click on to **confirm** the operation.



TopSolid has added a bill of materials index per part.

The indexes are fully customizable. For example, you can directly integrate information from the bill of materials or even from the part in the index.

- To do this, right-click on an index and select the **Edit** command in the **Automatic indexing** section. In the **Style** field, select **QtyDesc_MatCoatMass** from the drop-down list.

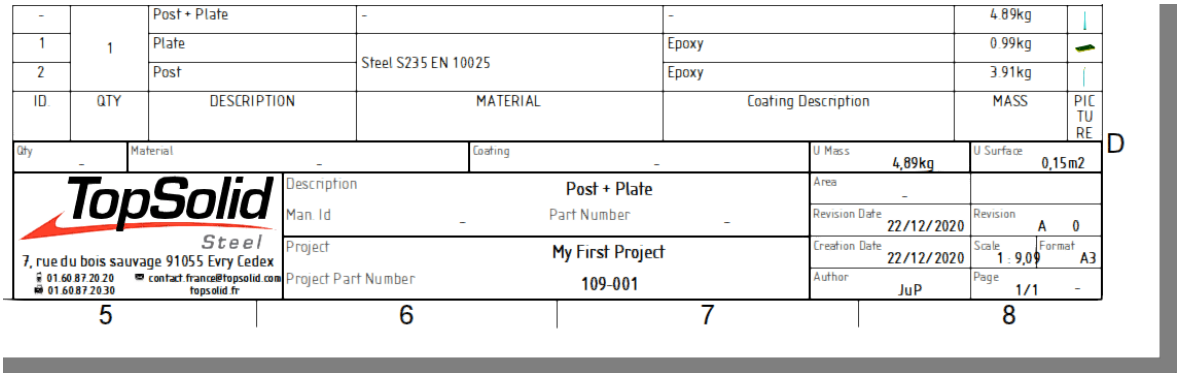


- **Save** the drafting document (**Ctrl** + **S**).

Updating the project information

In the default title block delivered with **TopSolid**, a section is allocated to the project. This information is automatically inherited from the project itself. In other words, if you want to reuse a set (parts, assembly, bill of materials, drafting, etc.), you simply have to copy and paste it into a new project. The draftings will be automatically updated with the new project information.

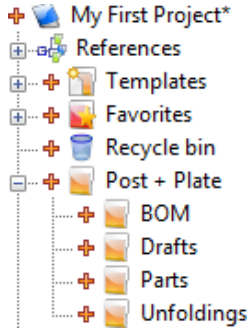
- From the Project tree, double-click on the *My Project* name.
- Modify the **Name** field by entering *My First Project*.
- Modify the **Description** field by entering *My First Project*.
- Modify the **Part Number** by entering *109 - 001*.
- **Save** the project (**Ctrl** + **S**).
- Return to one of the drafting documents. The **Project** area has indeed inherited the modifications in an associative way.



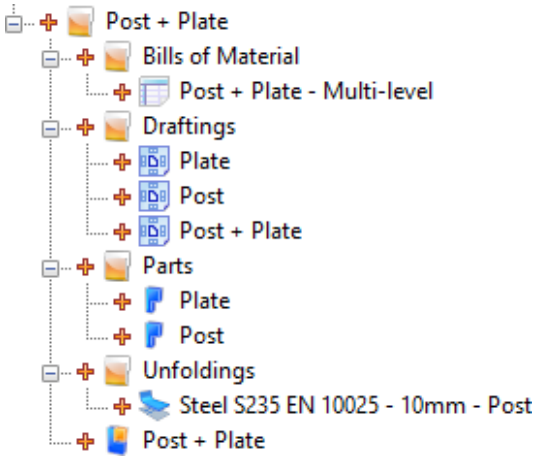
Organizing the project

As you will have noticed, a complete project generates a number of different documents. The flexibility of the PDM allows, without losing the link, you to make modifications such as renaming a document, moving it to another folder, etc. Therefore, you can focus on modeling at first and then rename the parts and organize the project at the end.

- From the Project tree, create a folder named *Post + Plate*. To do this, right-click on *My first project* name and select the **Folder** command.
- In the *Post + Plate* folder, create the following four folders in the same way.



- Drag the following documents into the appropriate folders.



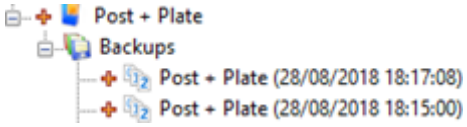
When creating a project, you can have a pre-established folder structure. This saves precious time, but above all guarantees a certain homogeneity.
Finally, non-TopSolid files (images, Excel, Word, etc.) can also be integrated into the project by dragging them from Windows and dropping them into the **TopSolid** project.

Check in

So far, we have been content with creating document backups. **TopSolid** manages its backups document by document.

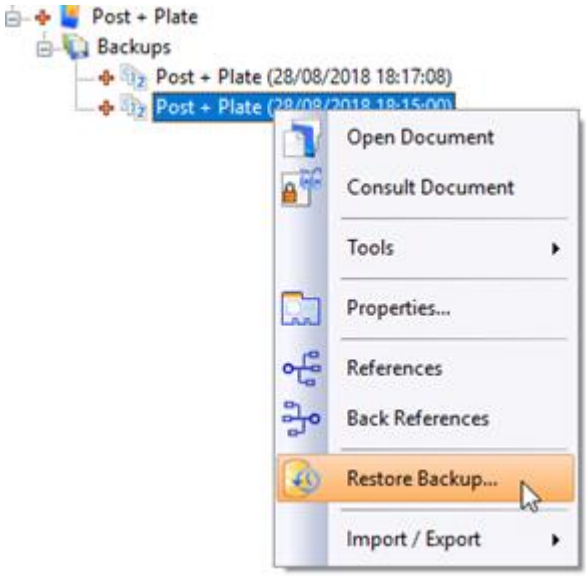
- In the Project tree's icon bar, click on the **Show backups** icon.

An ⊞ icon appears to the left of the documents allowing you to display the backups.



You can restore a backup by right-clicking on the desired backup and by selecting the **Restore Backup** command.

Warning: Do not do this as part of this exercise.



A backup is kept by default. To adjust the number of backups, you simply have to select the **Tools** > **Options** > **PDM** > **PDM** command and adjust the value in the **Number of backups without back references to keep** field. Increasing the number of backups decreases the performance of **TopSolid** at each backup.

In server mode, everything that has been created so far is visible only to us.
In order to benefit from the automatic index management system and to make these documents available to other users, you have to check them into the vault. You can check them into the vault document by document, or by selecting a folder or even an entire project.

Warning: When a document is checked in the vault, **TopSolid** deletes all the backups that were not used by other documents.

When to check elements into the vault?

There are no rules. Checking an element into the vault can be done at the end of a design stage, which is our case, at the end of the day, when exiting **TopSolid**, etc. It really depends on the person and the context.

- Right-click on *My first project* name and select the **Check In** command.

A dialog box appears that lists all the documents concerned. **TopSolid** may offer to update certain documents. If this is the case, you simply have to click on the **Update** button and then start again.

- Click on to **confirm** the check-in.

The icon then appears in front of the name of each document/folder in the vault.

Understanding how updates work

Most of the time, a study has to evolve. **TopSolid** facilitates update cycles thanks to the PDM.

- From the Project tree, open the *Post* part document.
- Double-click on the part. All the dimensions are displayed.
- Double-click on the *1000mm* height dimension and then subtract *40mm*.

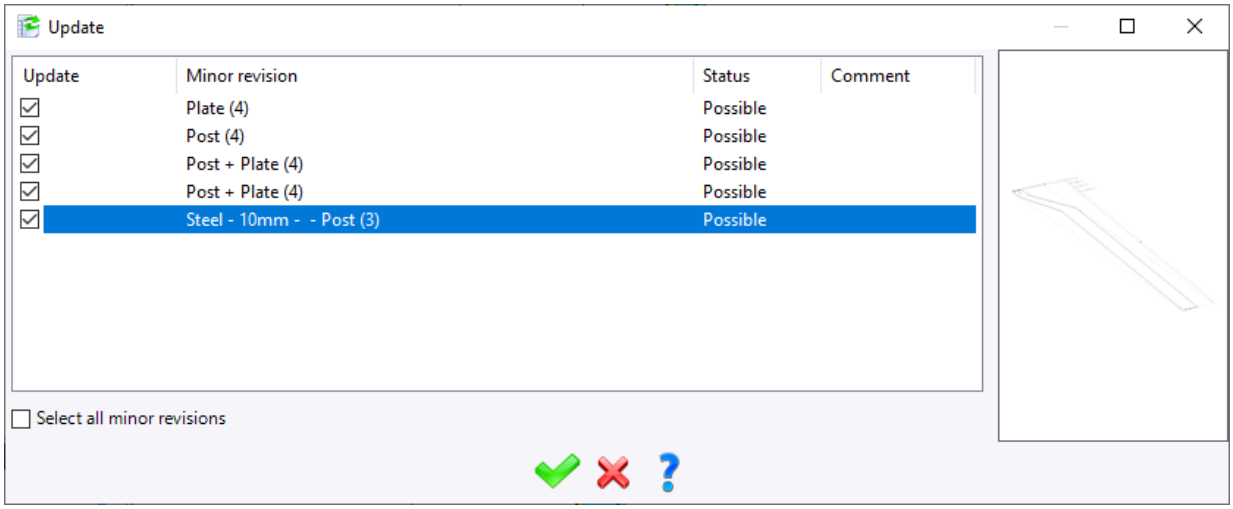
TopSolid displays the result while keeping the calculation history.

- **Save** the part document (**Ctrl** + **S**).
- In the Project tree, make sure that the following icons are selected.

TopSolid dynamically displays an exclamation mark in front of each document requiring an update. When opened, the document will update automatically.

- Right-click on the *Post + Plate* folder and select the **Update** command.

A dialog box appears that lists all the documents to be updated.



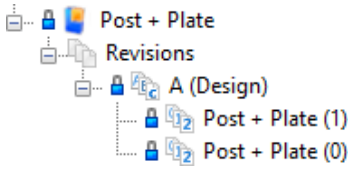
- Click on to **confirm** the update.

All documents are up to date with the new post height. The relevant documents have also been removed from the vault; the icon has disappeared and has been replaced by this icon.

In server mode, the documents will remain visible to other users, but will be locked indicating that the document is being modified.

- Right-click on the *Post + Plate* folder and select the **Check In** command.

Each time a document is modified and placed in the vault, **TopSolid** increments the minor revisions of the document.



As with backups, revisions are stored below the document. You can restore one of the revisions to return to a previous state.

Understanding how deletions work

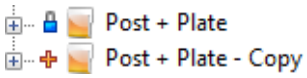
We have just seen that **TopSolid**'s PDM provides design flexibility. Its role is also to secure your files. Therefore, it will be completely impossible to delete a document that is used in another document (a part used in an assembly, for example).

- Right-click on the *Post + Plate* folder and select the **Copy** command or use the **Ctrl** + **C** keyboard shortcut.

Note: Copying an entire folder (draftings, parts, assemblies, etc.) allows you to keep the links between the documents.

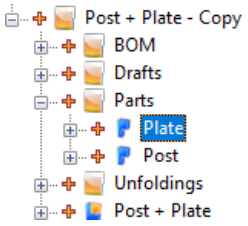
- Right-click on *My first project* name and select the **Paste** command or use the **Ctrl** + **V** keyboard shortcut.

TopSolid creates a copy of the complete folder.

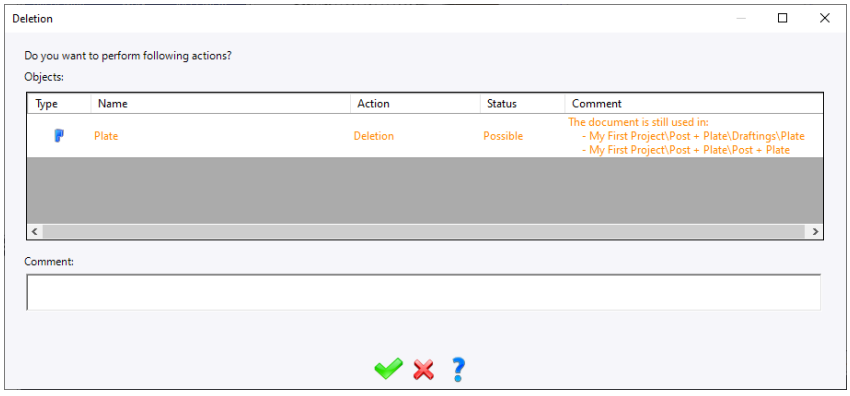


Note: No link is kept from the initial *Post + Plate* folder.

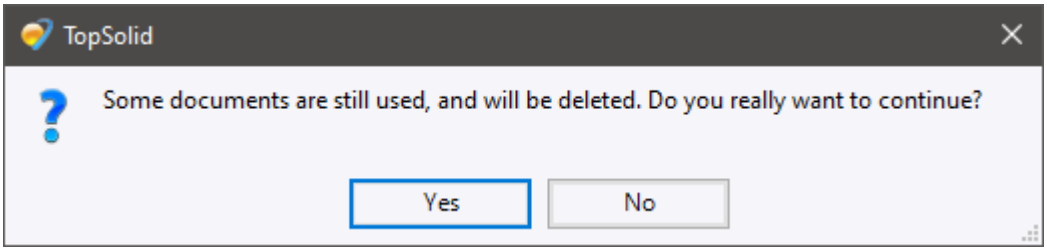
- In this new folder, delete the plate by pressing the **Delete** key or by right-clicking on the *Plate* document and selecting the **Deletion** > **Delete** command.



TopSolid warns you that the document is being used, but still allows it to be placed in the recycle bin.

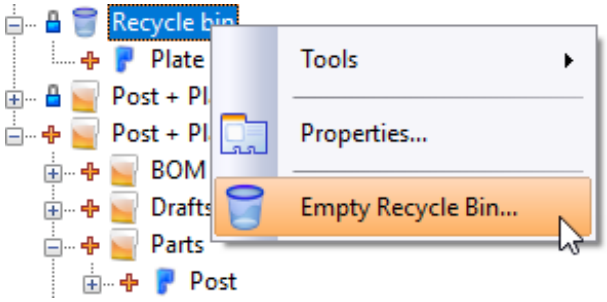


- Click on to **confirm** the deletion and click on **Yes** to **confirm** the following message.

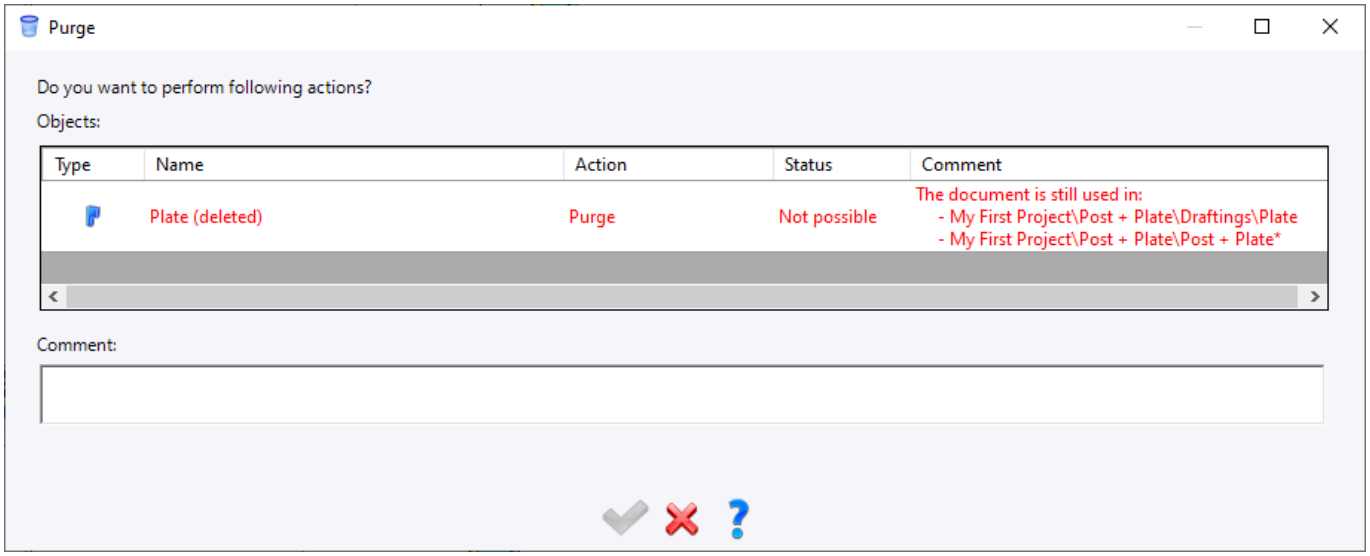


The document is now in the recycle bin.

- From the Project tree, open the **Recycle bin** node.
- Right-click on the **Recycle bin** node and select the **Empty Recycle Bin** command.



TopSolid prohibits the deletion of the document.



- **Close** the dialog box by clicking on the icon.
- In the Project tree, delete the *Post + Plate - Copy* folder with the **Delete** key or via the contextual menu.
- **Empty the recycle bin** again. Deletion is now possible.

Part 2 - In-Place Design of the Railing

The purpose of this exercise is to design a railing based on an environment and to create all the manufacturing draftings.

Concepts addressed:

- Understanding the concept of background document
- Understanding in-place design

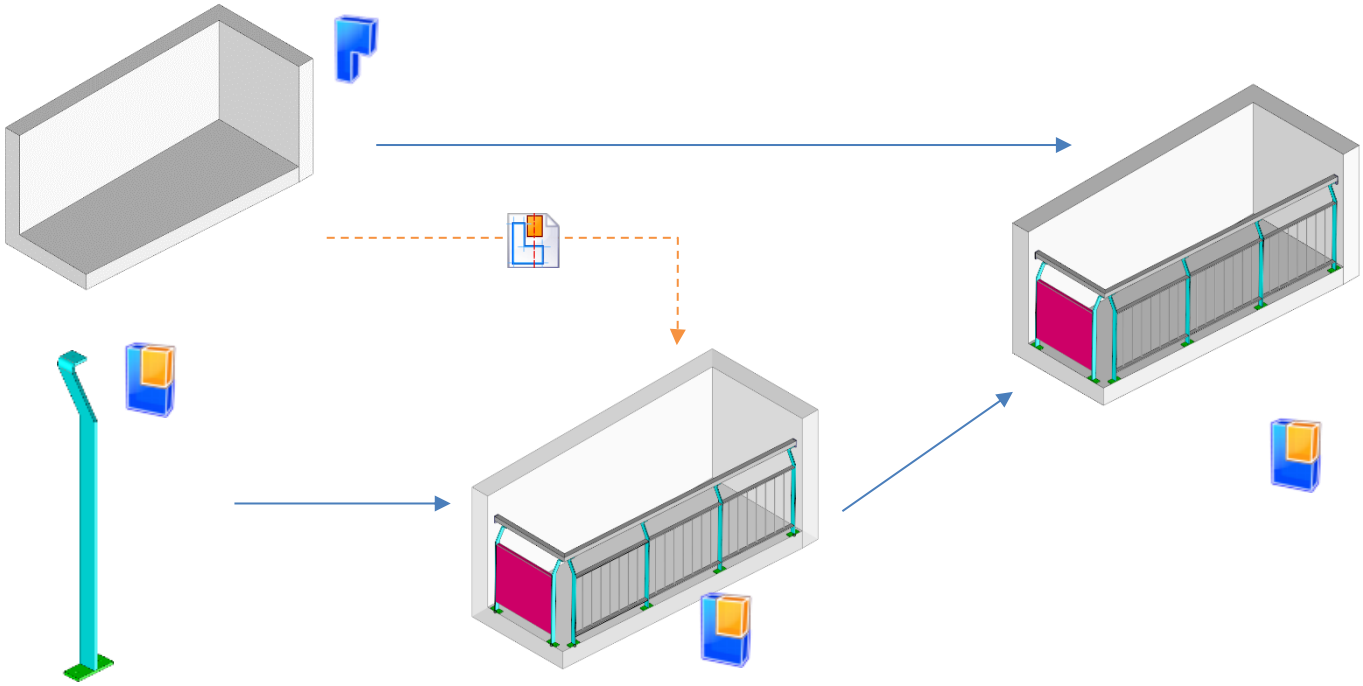
Concept

You now know how to produce assemblies from parts created separately and how to produce all of the associated production documents. This operation is ideal to include a fairly standard or commercial part.

TopSolid offers another mode of creation: in-place design.

This working method allows you to create parts and assemblies that are linked to each other. This design mode is useful when the parts are strongly constrained by their environment.
In this case, if the environment changes, all the parts and assemblies will be modified while respecting your constraints.
A stair will be recalculated according to its hopper, a railing according to the balcony, a door according to its opening.

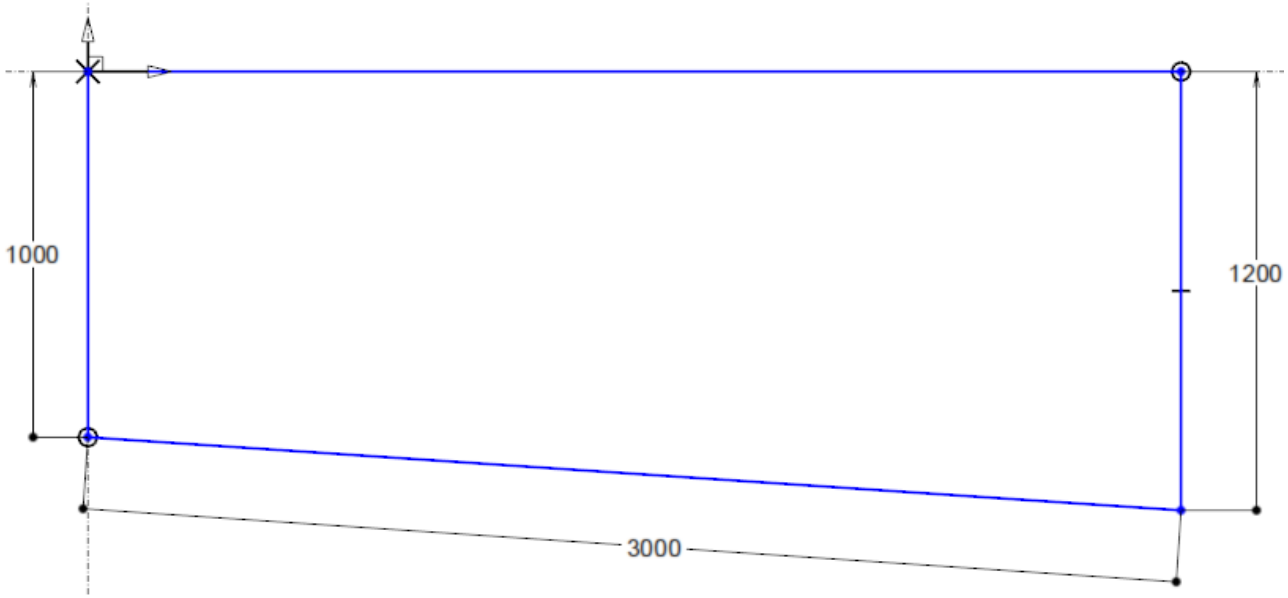
The following exercise deals with the layout of the following balcony.



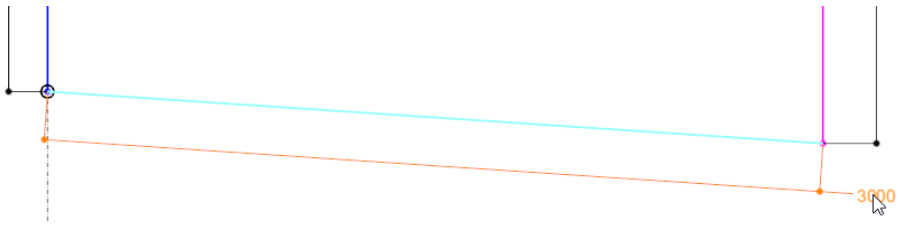
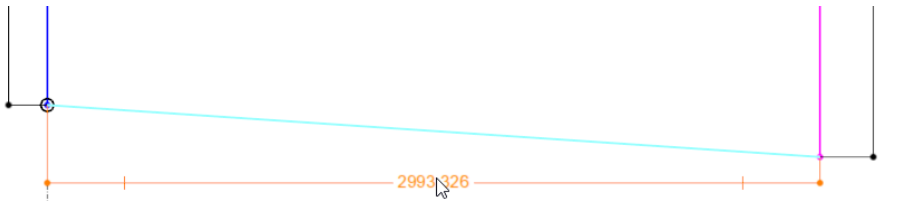
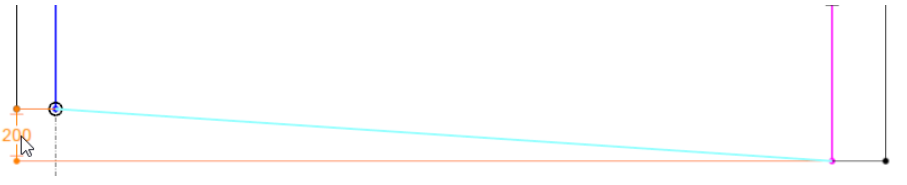
We will start by creating the environment and then use it as a background document to position the posts. We will then create the fillings, the handrails, the fixings, etc. Finally, we will assembly the railing in its environment to create a layout drafting.

Creating the environment

- Right-click on *My first project* name and select the **Part** command.
- Right-click on the part document you have just created and select the **Properties** command. Click on the **Edit** button. In the **Description** field, enter *Balcony* and then click on to **confirm** the operation.
- Right-click in the graphics area and select the **Sketch** command.
- Keeping the **Contour** command enabled and then draw the following contour.

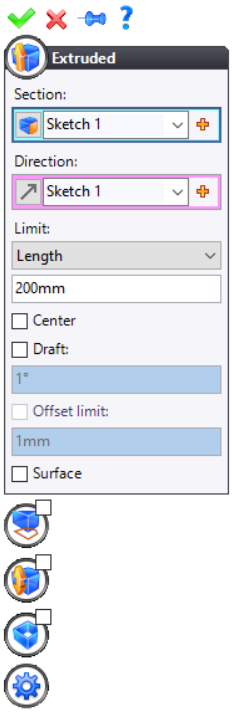


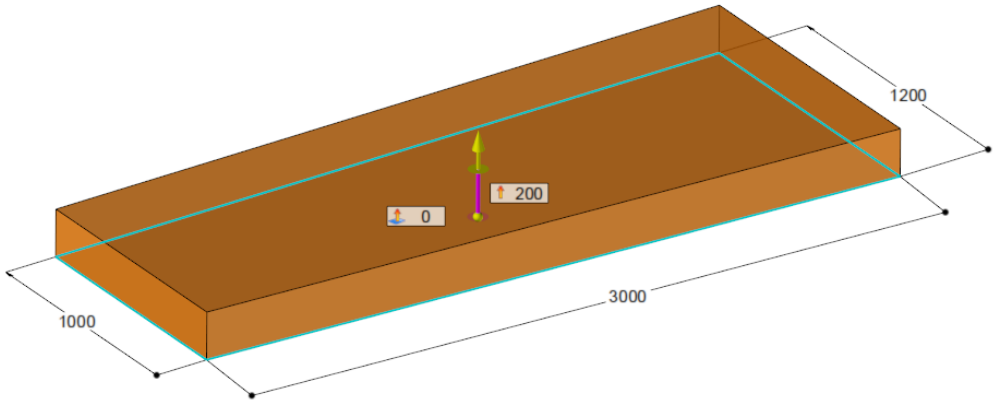
Warning: To create the 3000mm dimension, click on the line. Depending on where you position the mouse, **TopSolid** offers a dimension that is either parallel, projected on X or projected on Y.

Parallel	
Projected on X axis	
Projected on Y axis	

- Consider the case of a parallel dimension.
- Still in the sketch, right-click in the graphics area with no active selection and select the **Extruded** command.

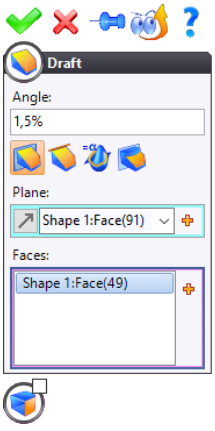
- Extrude the sketch to a height of *200mm*.

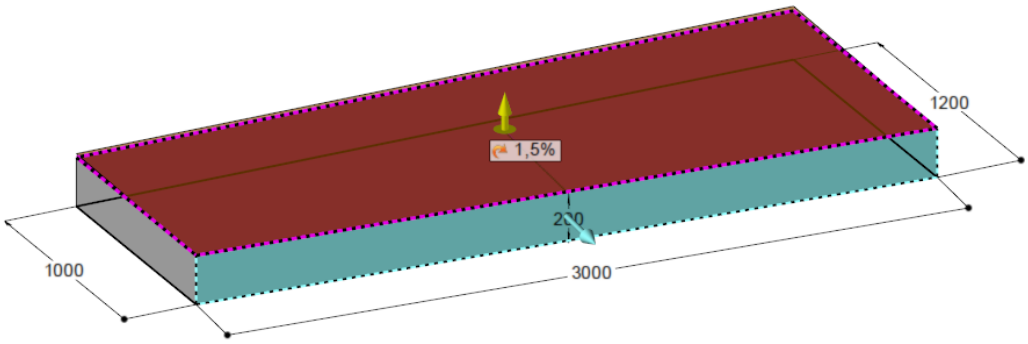




Generally, this type of structure has a slope. This can be done in several ways, including the following.

- From the **Shape** tab, select the **Draft** command.
- Enter an **angle** of *1.5%*. Make sure that you add the *%* sign, otherwise the value will be expressed in degrees (°).
- In the **Plane** field, select the front face of the balcony.
- In the **Faces** field, select the upper face. In this case, the 200mm dimension at the end of the balcony will be kept.



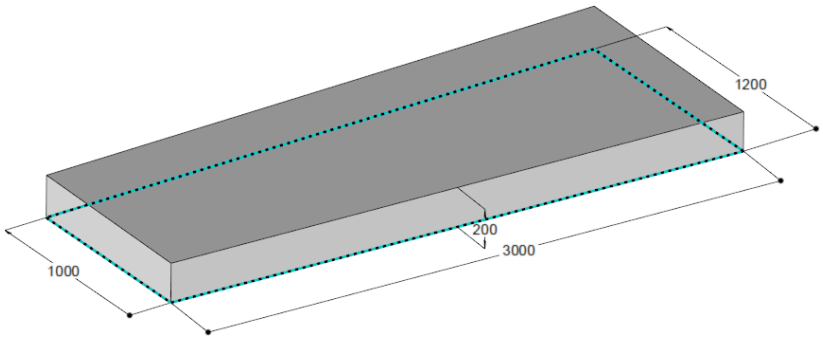


- Click on to **confirm** the operation.

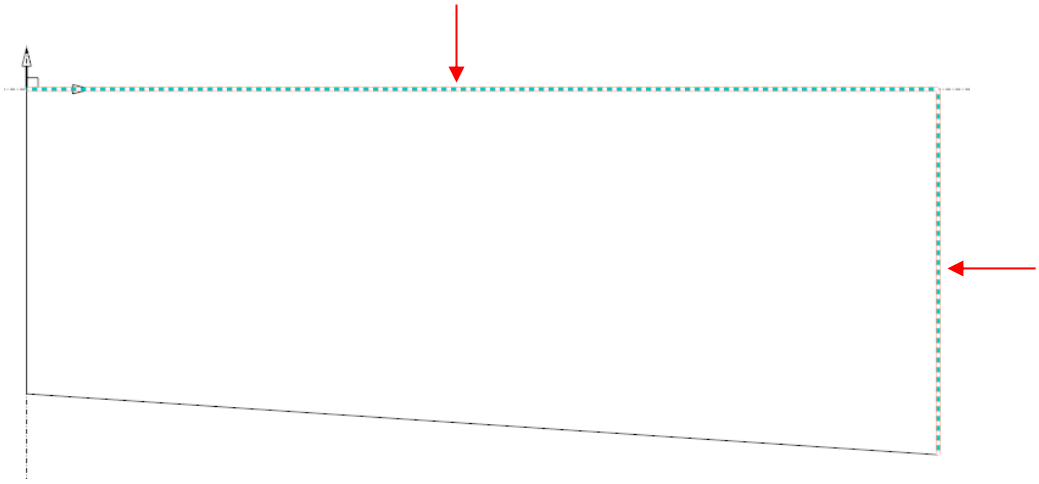
Now we will create the wall.

- Right-click on the lower face of the part and select the **Sketch** command. You can use the rotary picking technique to select the face without rotating the camera.

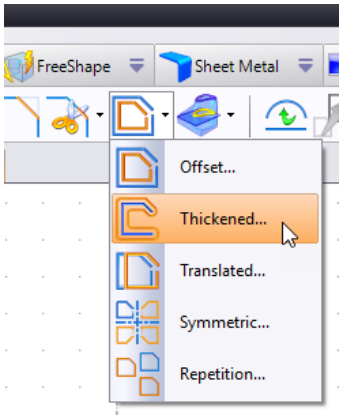
Reminder: hold down: Select the first entity under the cursor (the top face).
+ : Scrolling through the available entities.
release: Final selection of the displayed entity (the bottom face).



- Exit the **Contour** command.
- From the **2D Sketch** tab, select the **Project** command.
- Select the **Curves or edges** mode and click on the following two edges.

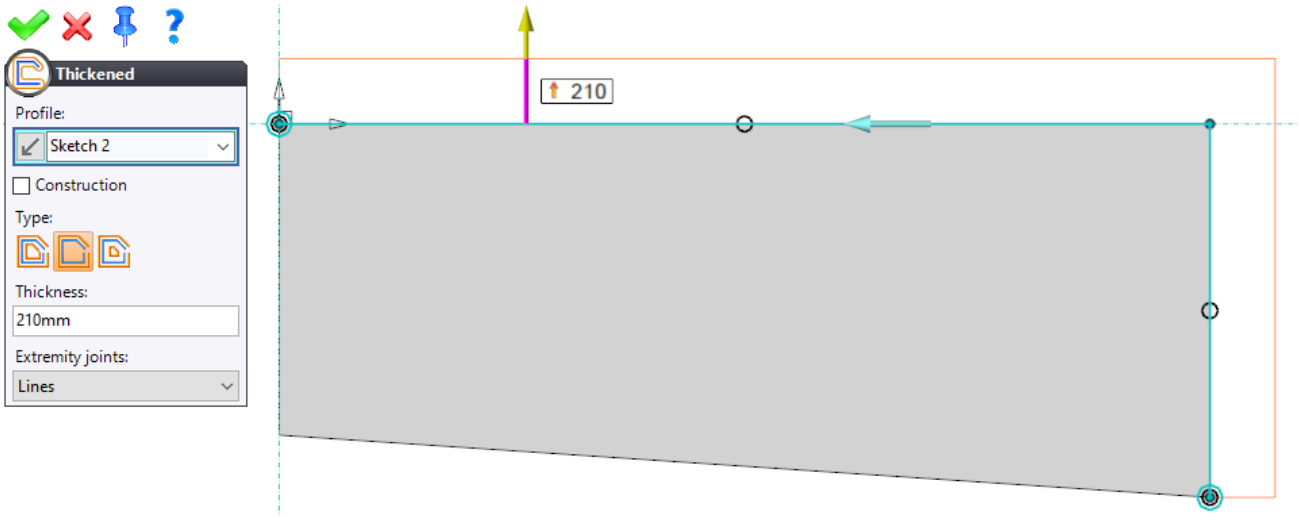


- Click on to **confirm** the projection. **TopSolid** draws a line for each selected edge.
- Still in the Sketch, select the **Thickened** command from the **2D Sketch** tab.

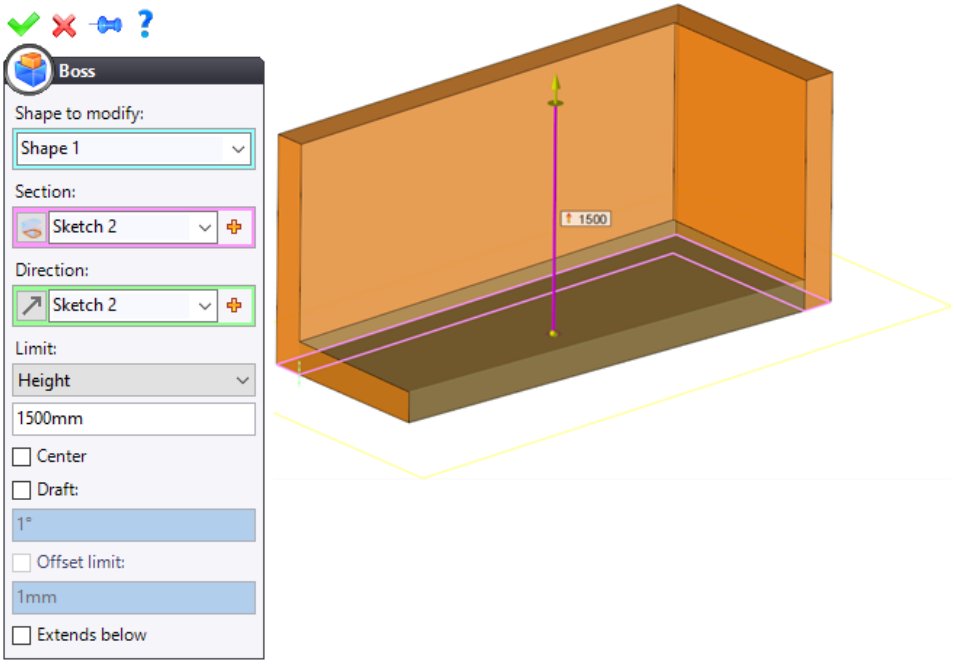


As its name suggests, this command allows you to add thickness to a set of lines, centered on one side or on both sides. It also allows you to manage the extremities. To find out more about this command, you can refer to the **TopSolid**'s online help by clicking on the icon or by using the **F1** key once the command has been selected.

- Select the previously created sketch. Select the **One side** type, enter a **thickness** value of *210mm*, then select **Lines** from the **Extremity joints** drop-down list. The other joints are used to make oblong holes, for example. Make sure that the **Construction** box is unchecked.



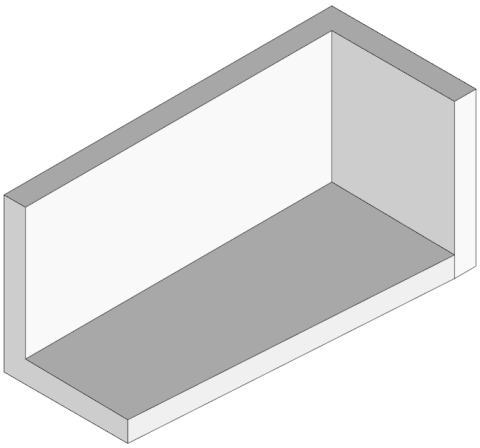
- Click on to **confirm** the operation.
- Right-click in the graphics area and select the **Boss** command.
- Enter a height of *1500mm*.



Note: If the boss does not work, select **Absolute Z axis** in the **Direction** field.

- Click on to **confirm** the boss.

You should obtain the following result.



- **Save** the part document.

Creating the background document

We will now use the environment as a background document; it will be used for the design but will not appear in the bill of materials nor drafting documents.

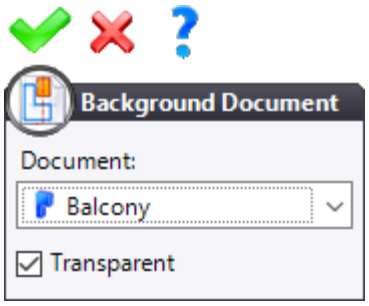
A background document can be of different types:

- Part
- Assembly
- 2D Drawing
- Etc.

In our case, the background document is a part, but it could have been an assembly modeled in **TopSolid** or even an assembly resulting from a 3D file import (IFC, Rvt, Step, etc.) or a 3D scan (on-site measurements or scan).

You can also change and even break the link with the background document to make the future railing independent.

- Right-click on *My first project* name and select the **Assembly** command.
- Right-click on the assembly document you have just created and select the **Properties** command. Click on the **Edit** button. In the **Description** field, enter *Railing* and then click on to **confirm** the operation.
- At the top left of the screen, click on the **TopSolid 7** icon and select the **File** > **Background Document** command. Select the *Balcony* document.



The **Background Document** command only displays the open documents in the drop-down list. Therefore, make sure that the *Balcony* part document is open before selecting the command.

- Check the **Transparent** option, then click on to **confirm** the operation.

We can now design the railing.

Creating the railing

Introduction to constrained linear pattern

In **TopSolid**, a repetition is divided into two parts:

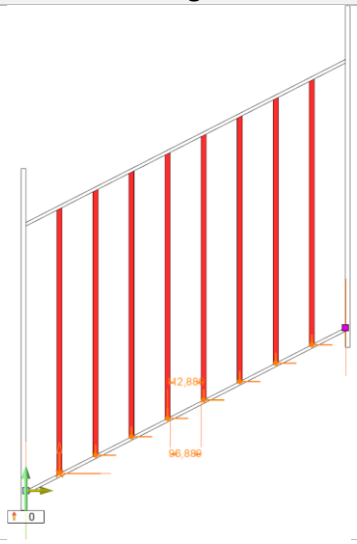
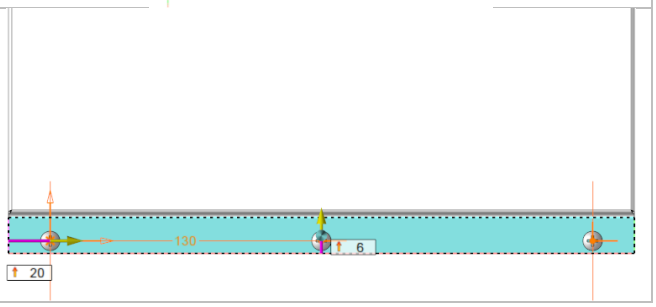
- what you can repeat (**Entities** and **Repetition** fields);
- how to repeat it (**Pattern** field).

A pattern can be common to several repetitions, so you do not have to redefine it each time.

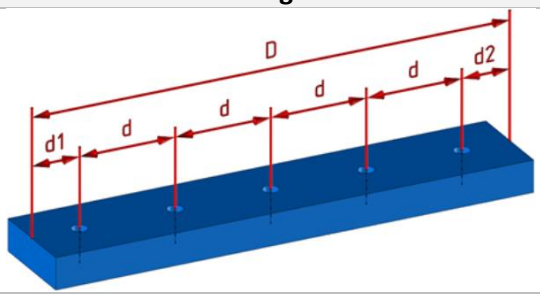
Usually, the object to be repeated is positioned and then the pattern is created. In metalwork, often times the inverse is true, because the position of the object is given by the pattern itself (posts, bars, etc.).

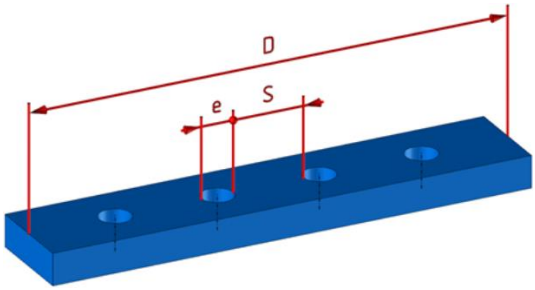
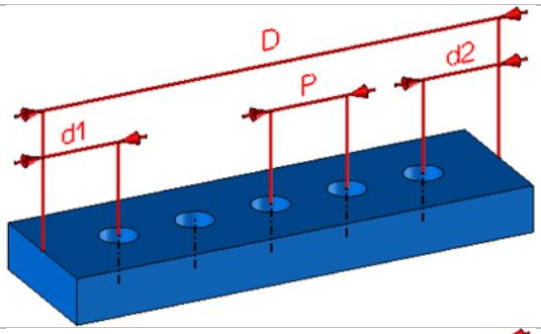
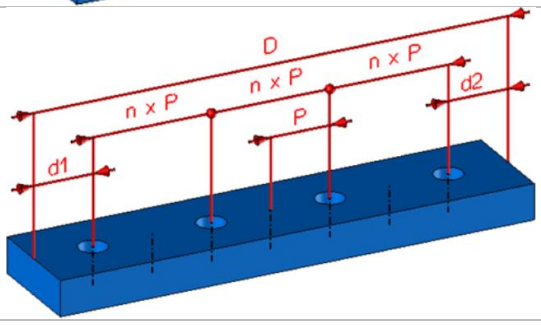
TopSolid includes a particularly popular pattern to calculate the distributions: the **constrained linear pattern**.

This has two installation methods.

Icon	Description	Use	Image
	Between points	Ideal for calculating a repair between elements (walls, posts, etc.) with a certain orientation.	
	On plane	Application centered on a plane or with an offset from the edge. Useful for positioning elements on a plane (post on concrete, screws on sheet metal, etc.).	

And different calculation methods:

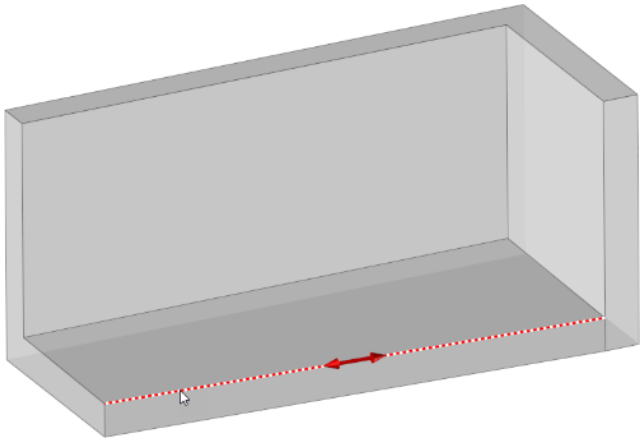
Icon	Description	Use	Image
	Fixed count	Divide the distance (**D**) by the number entered. Ideal for evenly distributing elements (screws, supports, drillings, etc.).	

	Maximum distance	Keeps a maximum distance between the instances (**S**). The **Thickness** (**e**) option allows you to take the thickness of the repeated element into account. Ideal when the distribution must comply with a standard or resistance constraints (railing, structural posts, railings, etc.).	
	Step	Divide the distance by a step (**P**) with or without a quantity. The distance is respected. The margins (**d1** and **d2**) allow you to precisely position the first instance or to give a minimum distance to the extremities.	
	Unit Step	Equivalent to **Step** mode but with a module (**n**).	

For more information about the **Constrained linear pattern** command, refer to the online help.

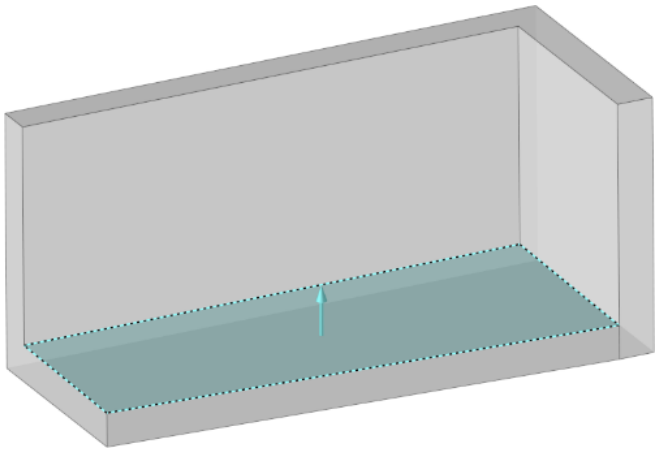
Post distributions

- From the **Construction** tab, select the **Constrained Linear Pattern** command.
- Select the **On Plane** mode.
- In the **Direction** field, select the edge as shown below.



This field defines the direction in which the distribution of the posts will be calculated.

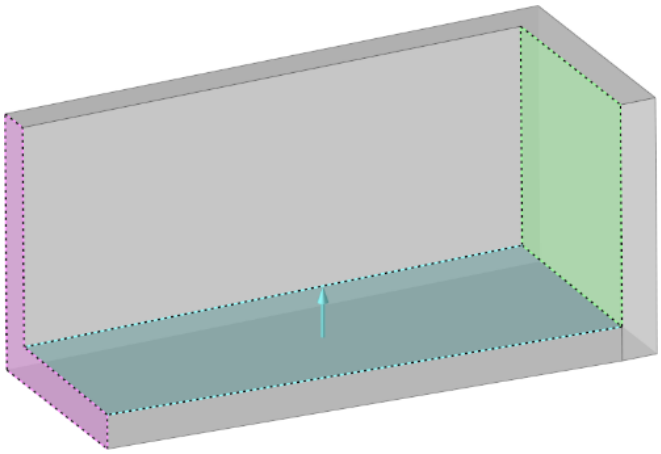
- In the **Support plane** field, select the face as shown below.



The support plane will define the orientation of the post. In this case, it will be perpendicular to the ground and therefore be slightly inclined.

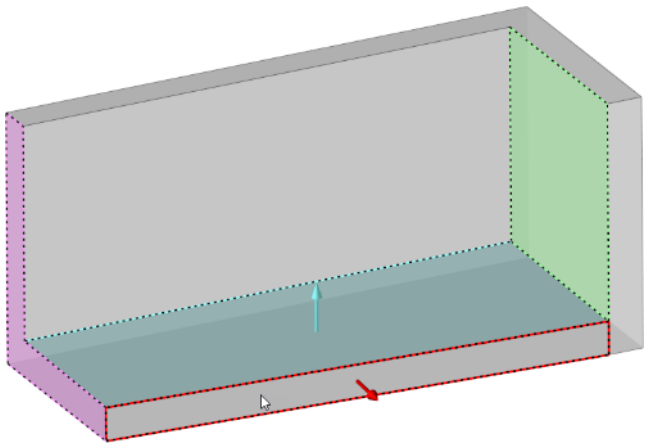
Note: If we had wanted to position the vertical post with a wedging, we would have had to recreate a correctly oriented plane using the icon.

- In the **Start geometry** and **End geometry** fields, select the following planes.



We will now position the posts in relation to the edge of the balcony.

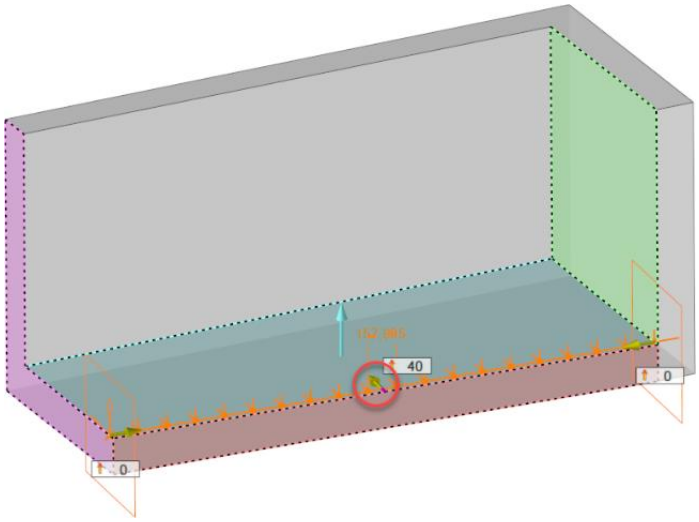
- Open the **Axis Position** dialog box and select the **Offset** mode.
- In the **Reference geometry** field, select the plane as shown below.



- Make sure the **Offset** box is checked and enter a value of *40mm*.

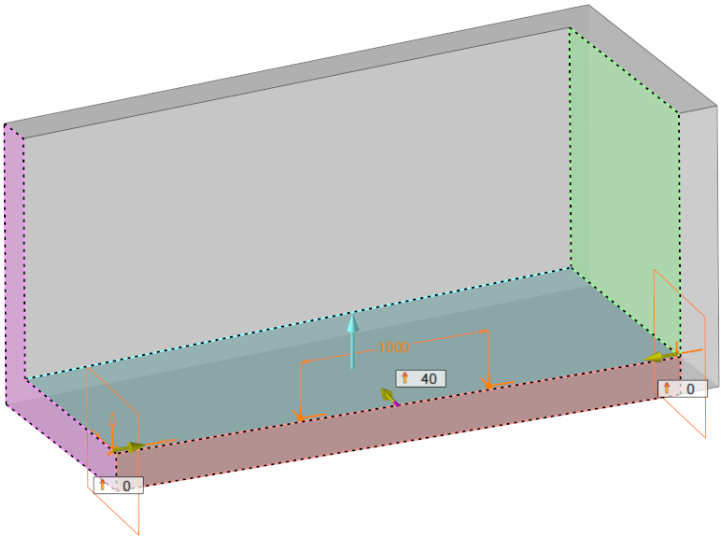
TopSolid displays a preview of the post distribution.

- Make sure that the yellow arrow in the center is pointing toward the inside of the balcony.



Note: If there had been no slope, **TopSolid** would have automatically selected all the planes necessary for the calculation and positioning of the pattern.

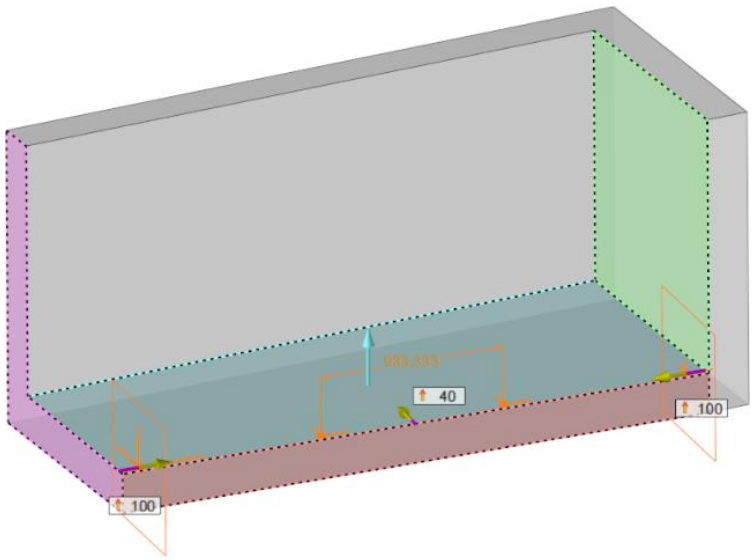
- Open the **Distribution** dialog box and select the **Maximum distance** mode.
- Enter a **maximum spacing** of *1000mm*, uncheck the **Thickness** box and check the **Edge to edge** box.



Note: In the image above, the orange frame on the left indicates the position of the first post and the orange frames on the right indicate the repeated instances (in this case, four).

You can now add offsets to correctly position the first and last post.

- Select **Distinct margins** from the drop-down list, then check the **Start margin** and **End margin** boxes. Enter a value of *100mm* for each of the margins.

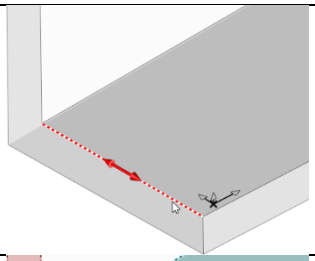
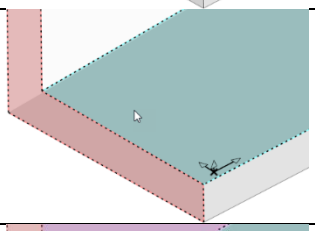
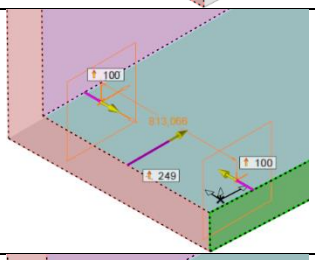
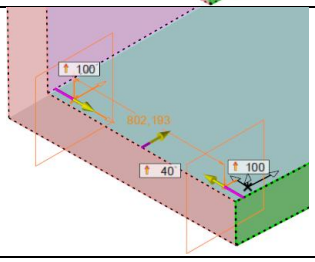


Note: You can also enter a value for the margins and offset directly in the graphics area.

- **Pin** the dialog box, then click on to **confirm** the operation.

We can now create the second pattern.

- Fill in the following fields for the constrained linear pattern.

Direction	
Support plane	
Start geometry and **End geometry**	
Offset value: *40mm*	

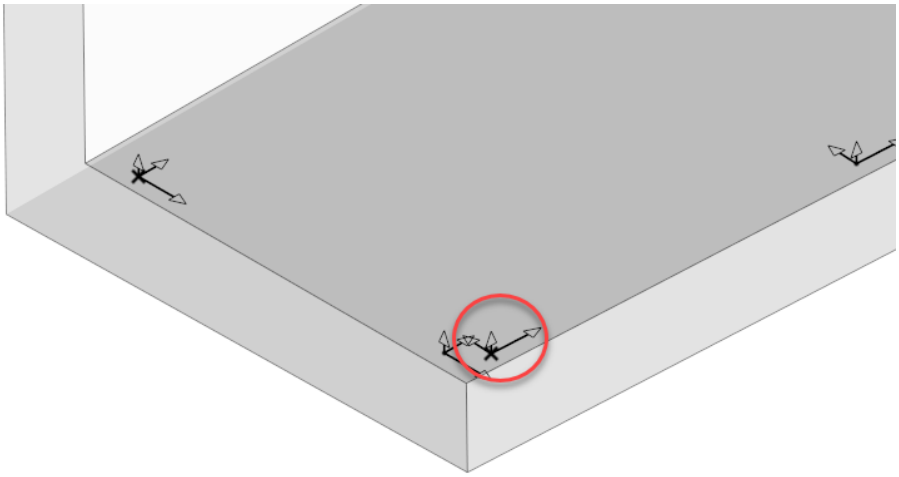
- Click on to **confirm** the operation.

Now that the patterns have been calculated, we can insert the posts and repeat them.

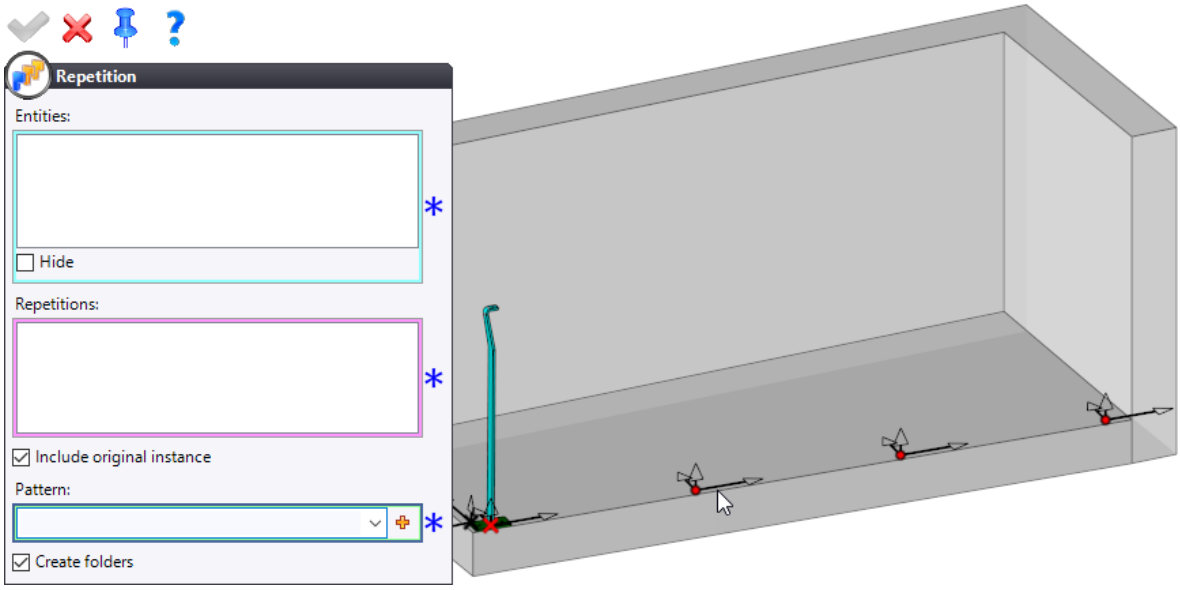
- From the Project tree, drag the previously created *Post + Plate* assembly into the graphics area of the *Railing* document.

TopSolid switches to the positioning context. You can then add constraints to the inserted element in order to position it in the assembly.

- From the **Assembly** tab, select the **Frame on Frame** command.
- In the **Source frame** field, select *Post + Plate* from the drop-down list.
- In the **Destination frame** field, select the frame as shown below.

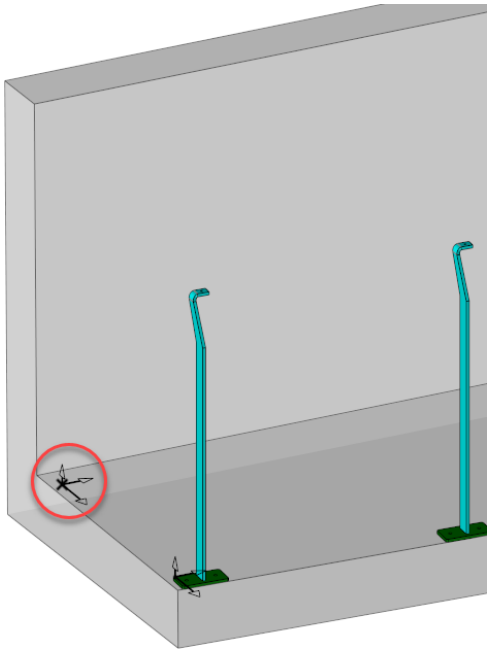


- Click on to **confirm** the positioning.
- From the **Construction** tab, select the **Repetition** command.
- Select the previously included post as the **entity** to be repeated. Click in the **Pattern** field to enable it. Select a point of the constrained linear repetition as shown below.

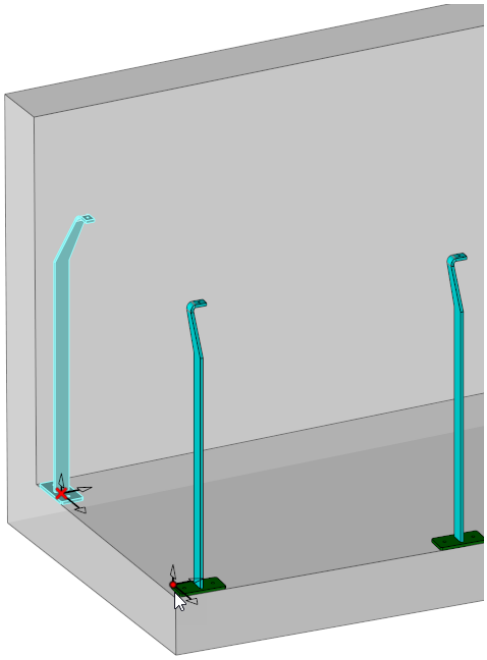


- Click on to **confirm** the repetition.

- From the Project tree, drag the previously created *Post + Plate* assembly into the graphics area of the *Railing* document.
- From the **Assembly** tab, select the **Frame on Frame** command.
- In the **Source frame** field, select *Post + Plate* from the drop-down list.
- In the **Destination frame** field, select the frame as shown below (the one with a cross).

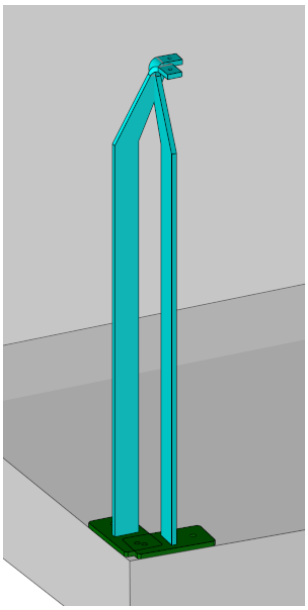


- From the **Construction** tab, select the **Repetition** command.
- Select the previously included post as the **entity** to be repeated. Click in the **Pattern** field to enable it. Select a point of the constrained linear repetition as shown below.



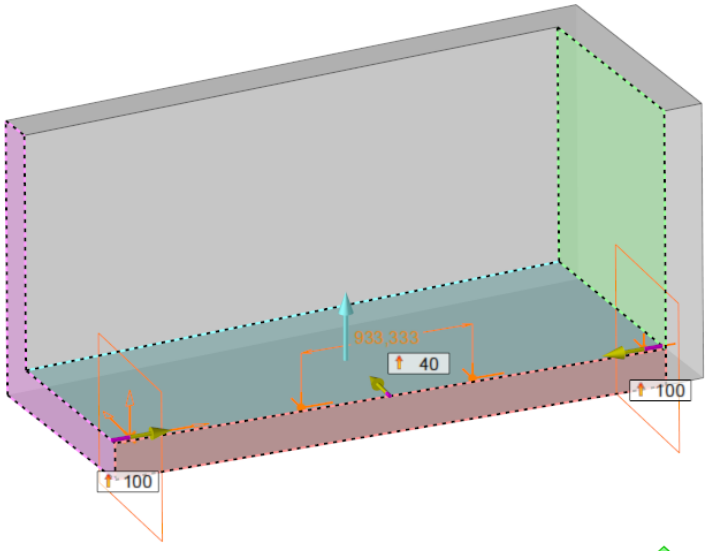
- Click on to **confirm** the repetition.

You can see that the two posts are colliding.



You can modify the patterns to adjust your work at any time.

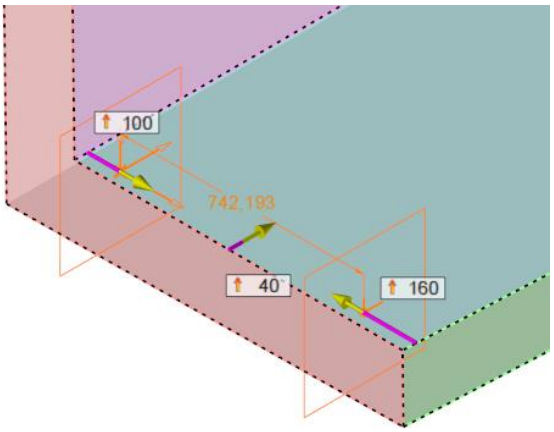
- Right-click on one of the repeated posts along the longest length of the balcony and select the **Edit Pattern** command.



- In the **Start margin** field, adjust the value from 100 to *160mm*, then click on to **confirm** the operation.

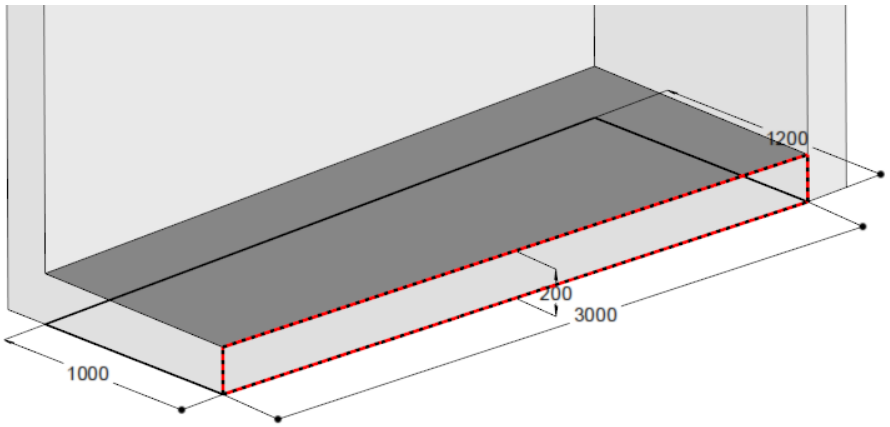
TopSolid automatically adjusts the positioning of the first post and calculates the repetition.

- Proceed in the same way for the second repetition, but this time by modifying the **end margin**.



We will now test the repetition by modifying the environment.

- **Save** the *Railing* assembly document.
- Return to the *Balcony* part document.
- Double-click on one of the side faces of the slab to display the dimensions.

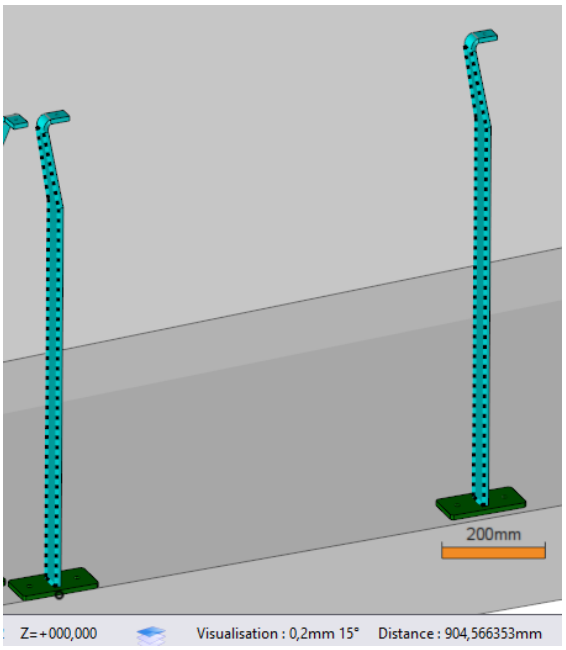


- Double-click on the 3000mm dimension and adjust the value to *3850mm*.

The balcony document needs to be updated.

- Double-click on one of the side faces of the slab to hide the dimensions. This action is not mandatory, but you should be aware that the background document keeps the display of the document as is. In other words, if you leave the dimensions displayed, when you return to the railing assembly, the dimensions will also be visible.
- **Save** the *Balcony* part document.
- Return to the *Railing* assembly document. You will notice that a post has been added.

If you wish to check the center distance of two posts, **TopSolid** has a fast measuring system. You simply have to press and hold down the **Ctrl** key and select the faces as shown below. The distance is then displayed in the status bar at the bottom right of the screen.

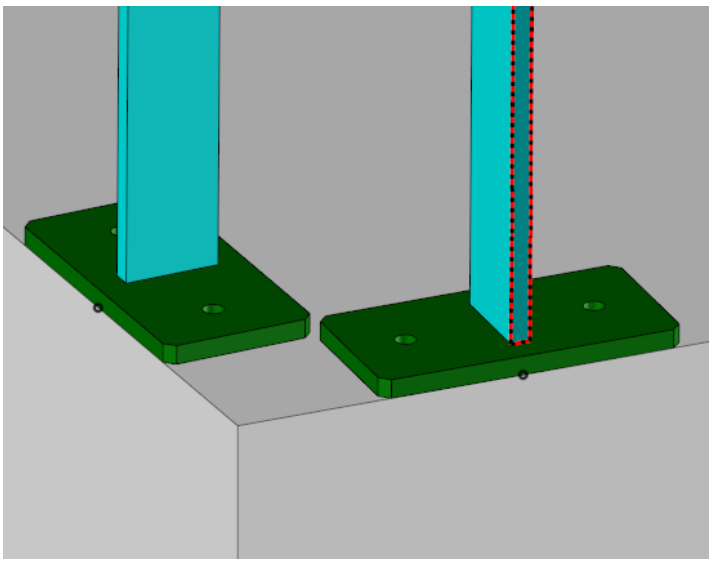


TopSolid also has an advanced measurement system. You simply have to right-click in the graphics area with no active selection and the select the **Analyze Geometry** command.

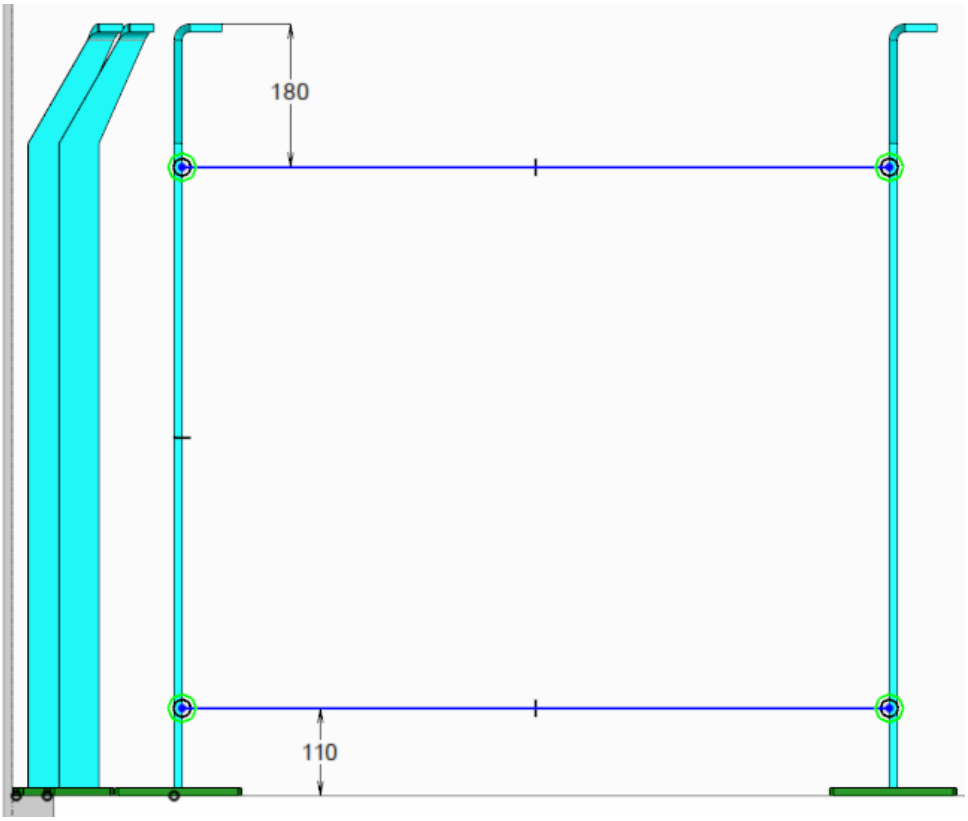
For more information, refer to the online help.

Creating the right filling

- Right-click on the face as shown below and select the **Sketch** command.



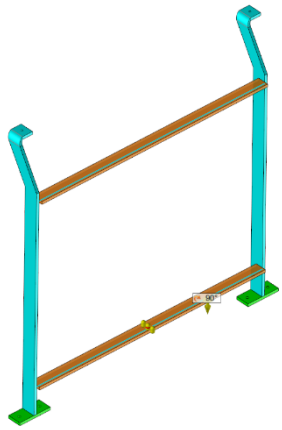
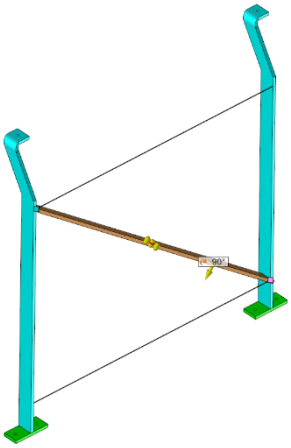
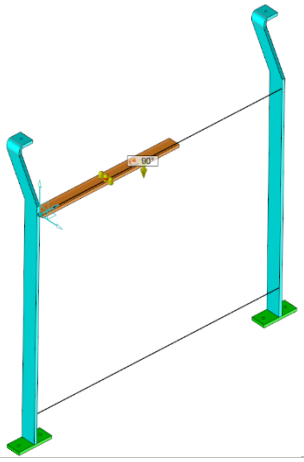
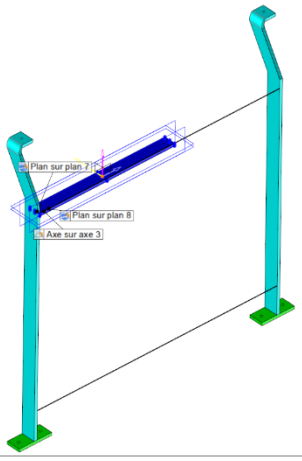
- Draw the following sketch.



- **Confirm** the sketch.
- Right-click on one of the segments and select the **Extruded bar** command.

- In the **Family** field, select **Full flat section NF A 45-005**.
- In the **Code** field, select **40 x 10**.
- Select the **Sketch/Segment/Profile/Edge/Face** mode. This mode allows you to select a segment, a profile or a complete sketch.

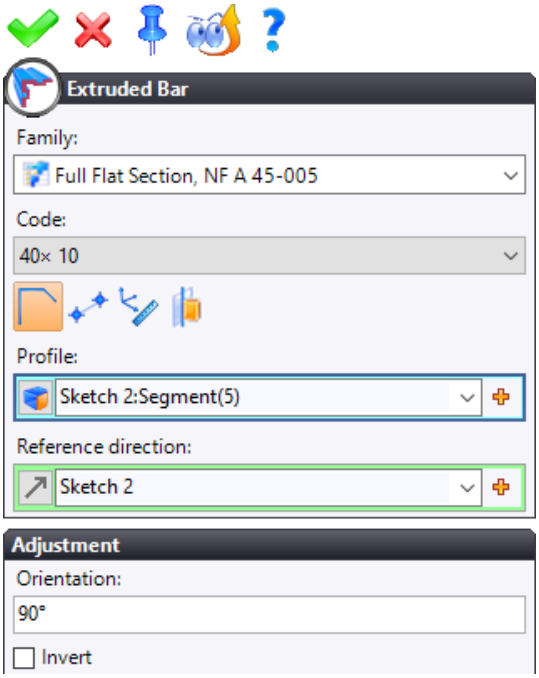
The four modes below deal with positioning.

Sketch/Segment/Profile/ Edge/Face	**Two Points**	**Frame and Length**	**Constrained positioning**
			

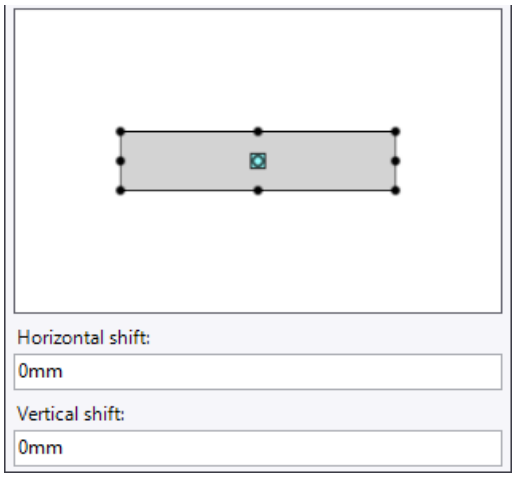
- In the **Profile** field, select the previously created sketch.

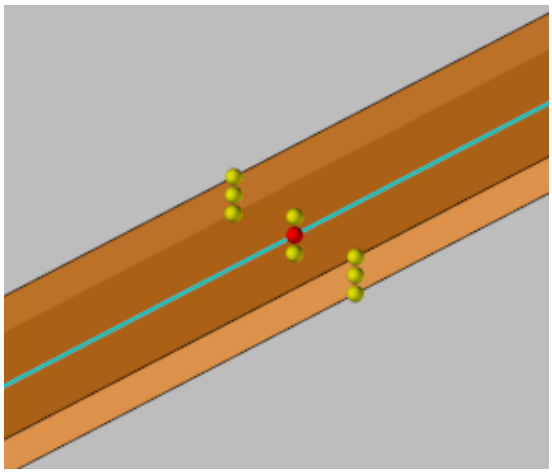
Each extruded bar has a reference direction and an orientation in the form of an angle.

- Adjust the **orientation** to *90°*.

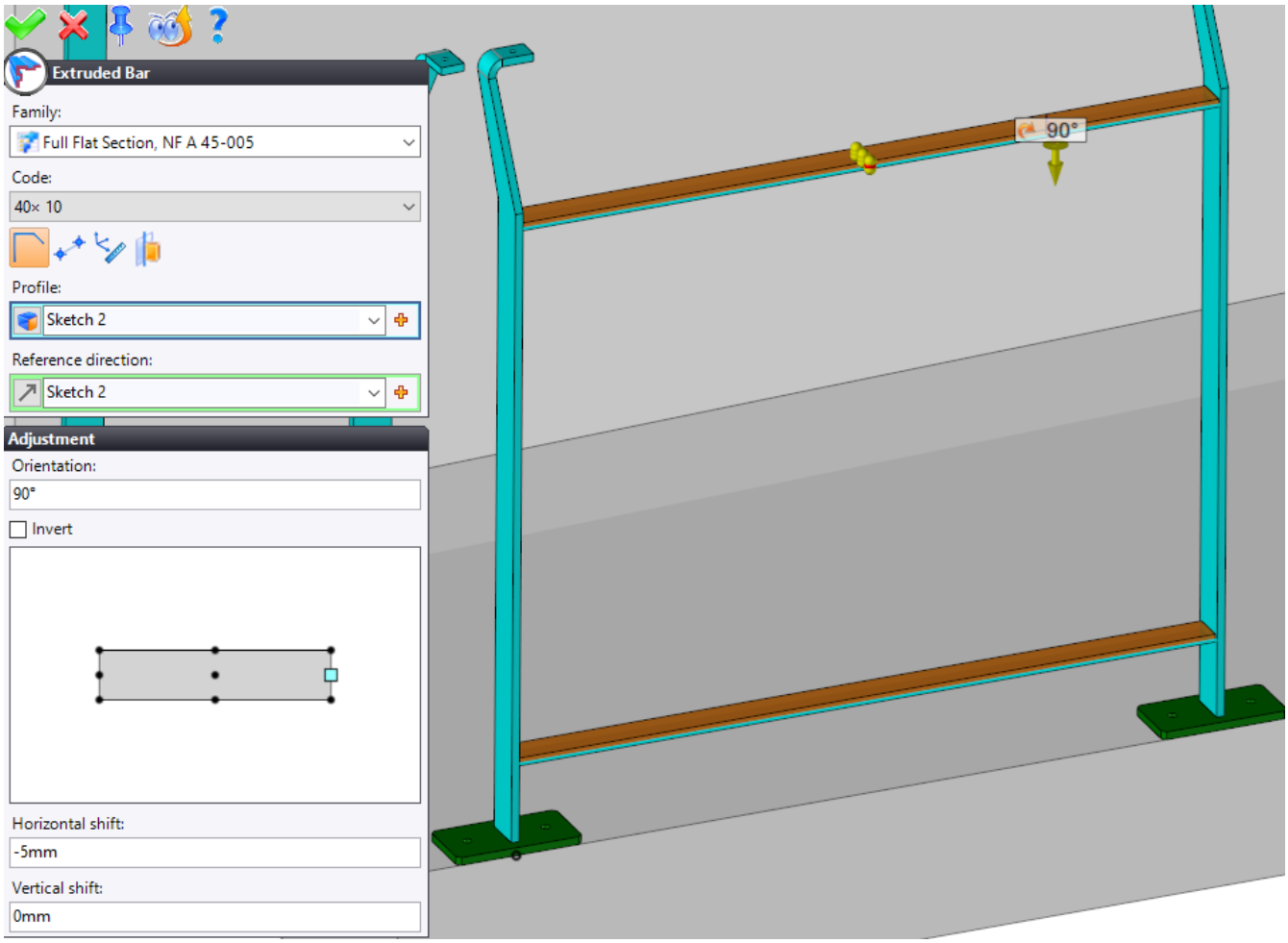


The lower part of the dialog box deals with the positioning of the extruded bar. Each extruded bar has nine standard positioning points.
You can select the point either in the dialog box or directly in the graphics area.





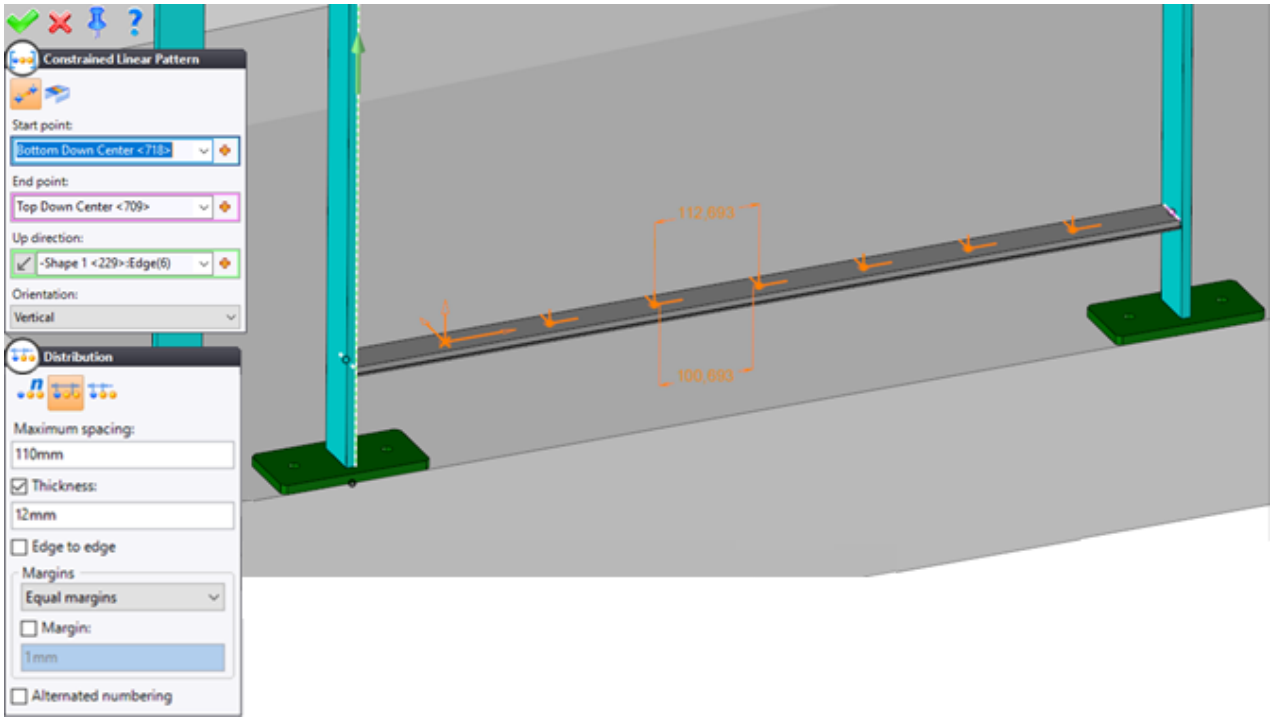
- Position the extruded bars as shown below so that they are centered on the post.
- Enter a **vertical shift** of *-5mm*, otherwise the extruded bars will be positioned directly above the posts.



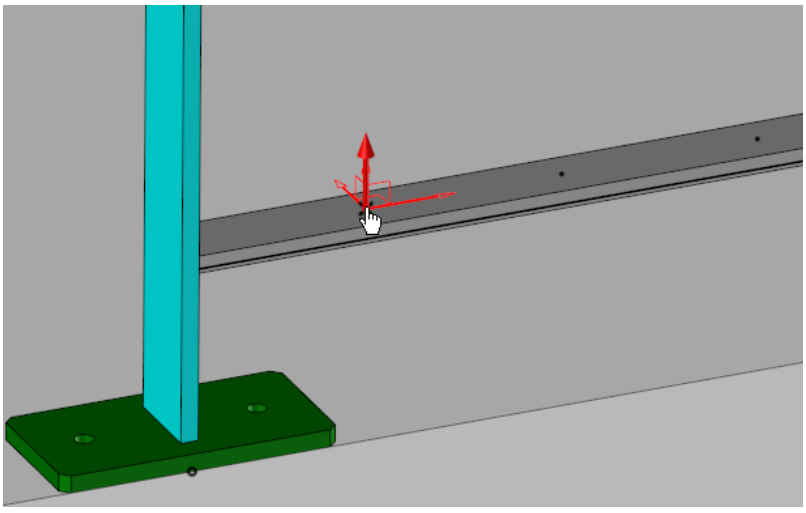
- Click on ✔ to **confirm** the operation.

We will now create a bar with a full circular section of Ø12mm with a maximum distance of 110mm.

- From the **Construction** tab, select the **Constrained Linear Pattern** command.
- Select the **Between points** mode.
- Select a point in the center of the left edge of the previously created low extruded bar as the **starting point**.
- Select a point in the center of the right edge of the previously created low extruded bar as the **end point**.
- Select a vertical edge of the left post as the **vertical direction** so that the bars are parallel to the posts.
- Select the **Maximum distance** distribution mode.
- Adjust the **maximum spacing** to *110mm*.
- Check the **Thickness** box and enter a value of *12mm*. This value corresponds to the width of the bar.
- Uncheck the **Edge to edge** box and select **Equal margins**.

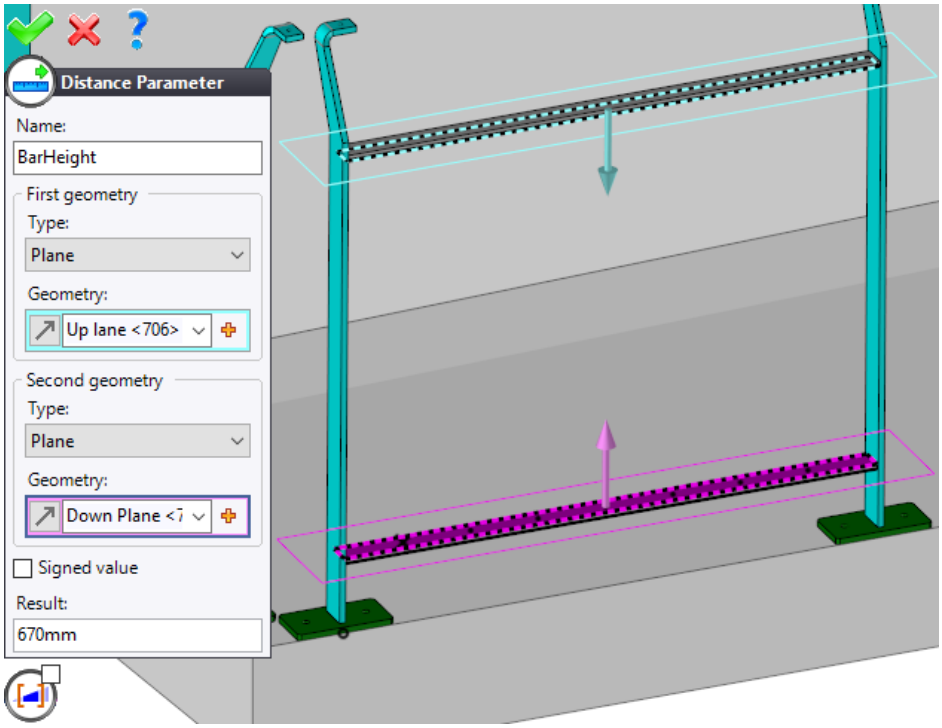


- Click on to **confirm** the pattern.
- From the **Modeling** tab, select the **Extruded Bar** command.
- Select the **Full Circular Section, NF A 45-003** family and the **12** code.
- Select the **Frame and Length** mode.
- In the **Destination frame** field, select the frame as shown below.



We could measure the height of the bar with the measuring tool. Unfortunately, if the gap between the two rails changes, you will have to think about adjusting the height of the bar.
TopSolid is able to keep this measurement by creating a parameter.

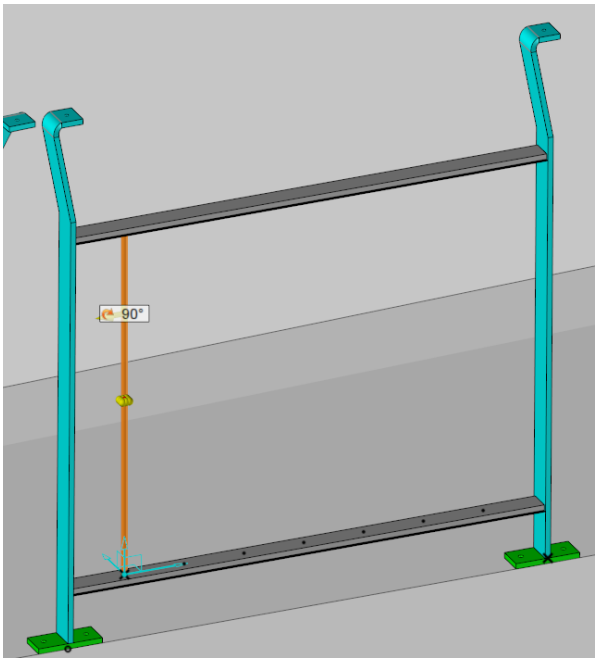
- In the **Length** field, click on the icon and then select **Distance Parameter**.
- In the **Name** field, enter *BarHeight*. This will make it easier to identify the parameter.
- Select the bottom face of the top rail as the **first geometry** and the top face of the bottom rail as the **second geometry** as shown below.



- Click on to **confirm** the parameter.

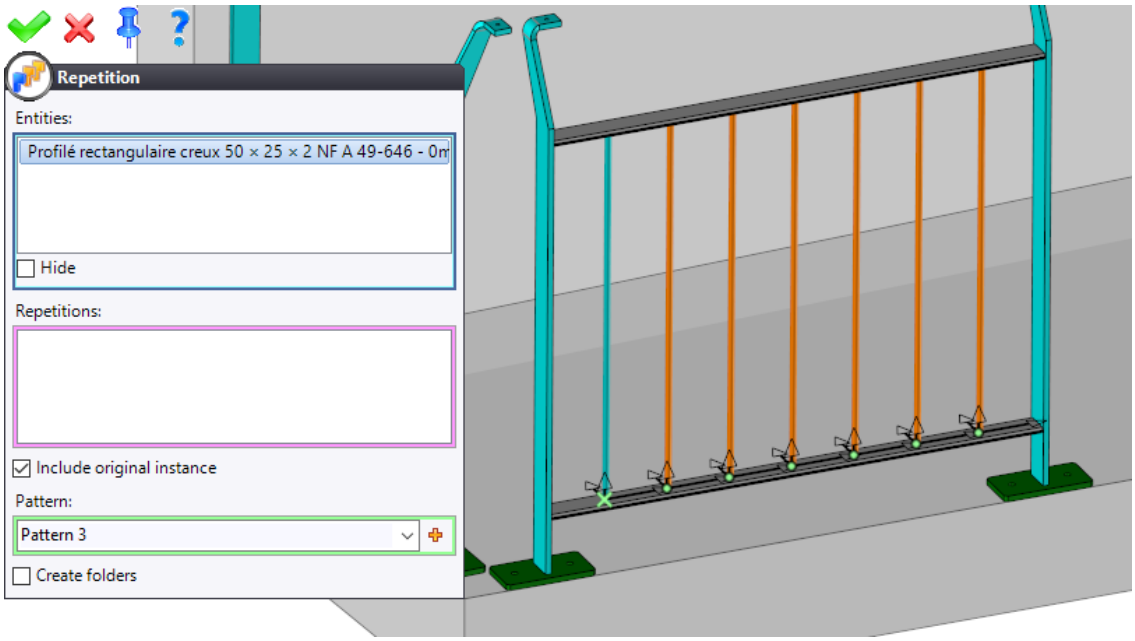
- In the **Extruded bar** dialog box, make sure that the shift values are set to *0mm*.

The bar is normally correctly dimensioned.



- Click on to **confirm** the inclusion of the extruded bar.

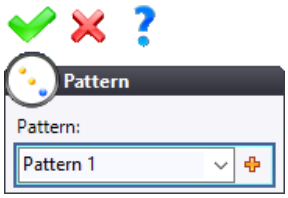
- From the **Construction** tab, select the **Repetition** command.
- Select the bar as the **entity** to be repeated and select the previously created constrained linear pattern.



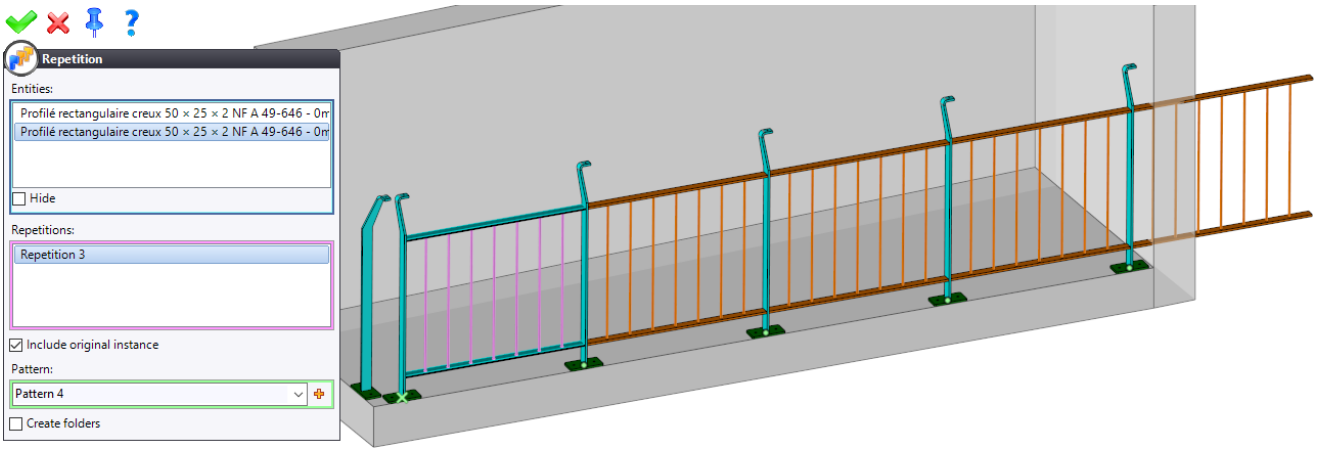
- Click on to **confirm** the repetition.

We will now repeat the rails and the bar. In order to avoid redefining everything, we will build on what has already been created with the posts.

- From the **Construction** tab, select the **Repetition** command. Select the two rails as the **entities** to be repeated. In the **Repetitions** field, select the bar.
- In the **Pattern** field, click on the icon and select **Pattern**.
- Click on one of the posts, then click on to **confirm** the operation.



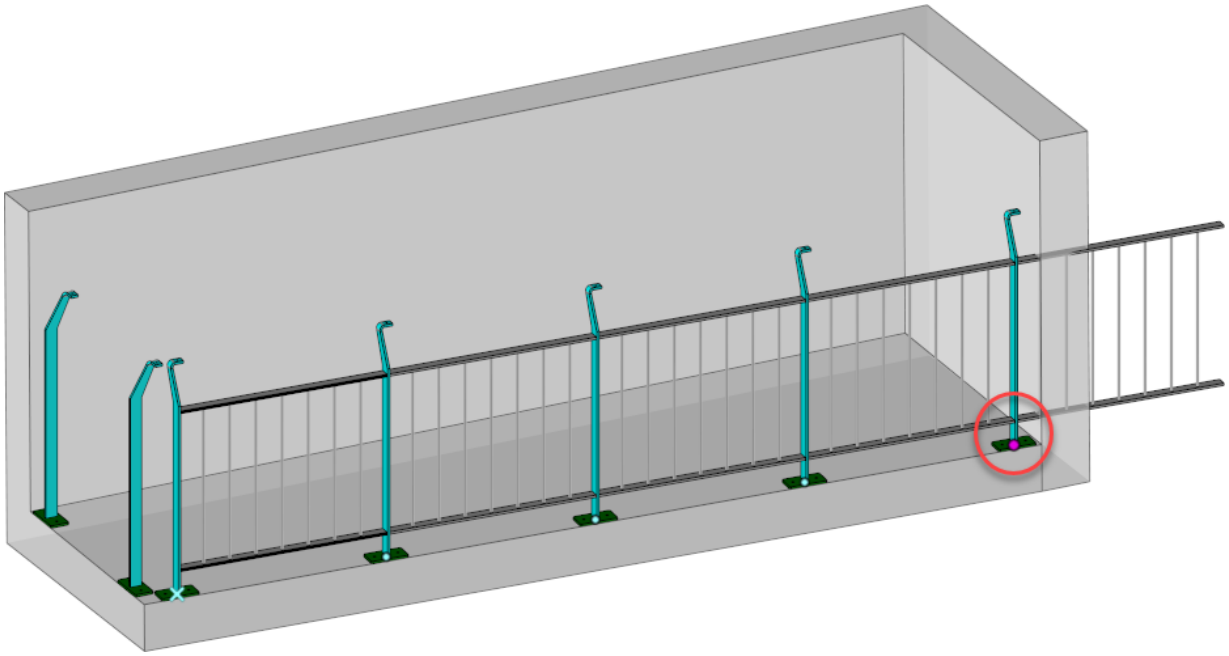
This step allows you to derive the pattern of the posts. This is mandatory if the filling pattern is to be modified without affecting the pattern of the posts.



- Click on to **confirm** the repetition.

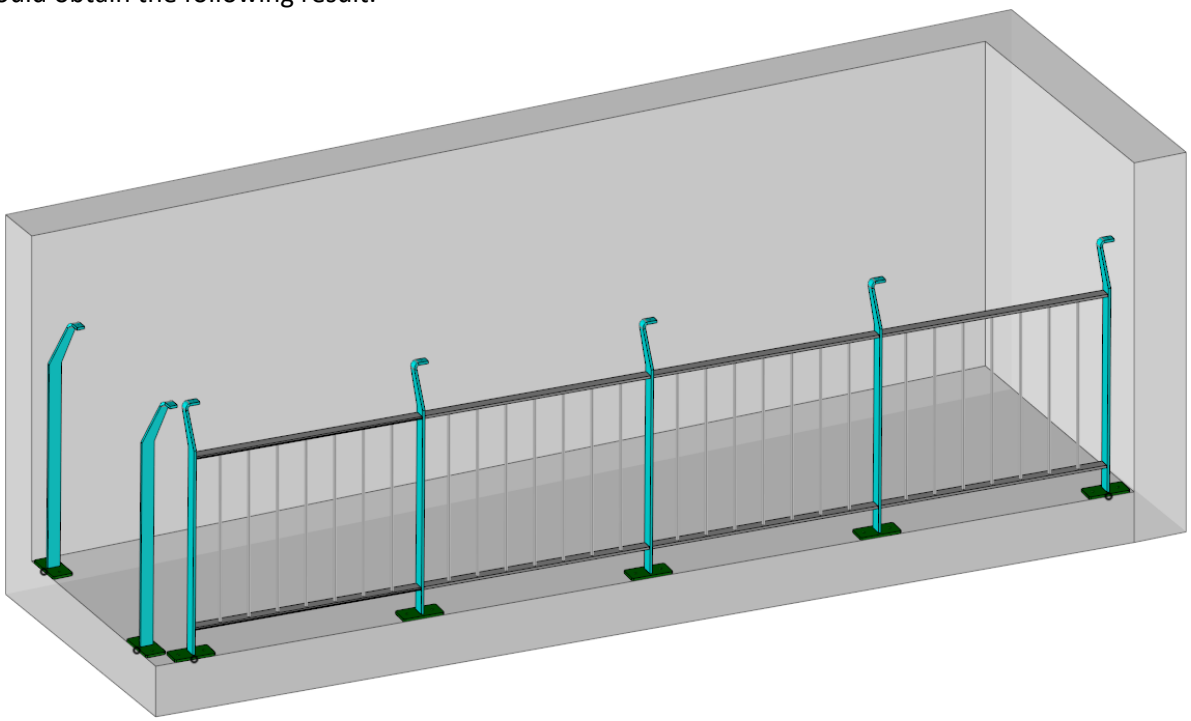
There is an excess filling. Indeed, since we used the pattern of the posts, the quantity is the same.

- Right-click on one of the repeated fillings and select the **Exclusion** command. Select the last occurrence as shown below.



- Click on to **confirm** the exclusion.

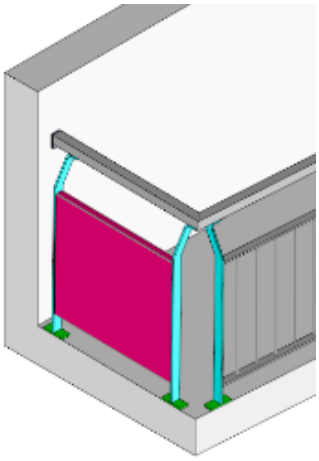
You should obtain the following result.



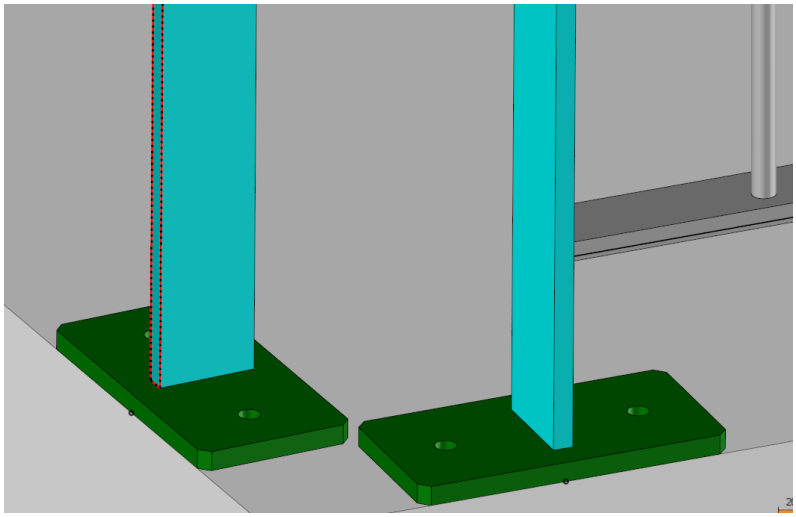
Note: You can return to the *Balcony* part document at any time to vary its dimensions and therefore confirm your parameters.

Creating the left filling

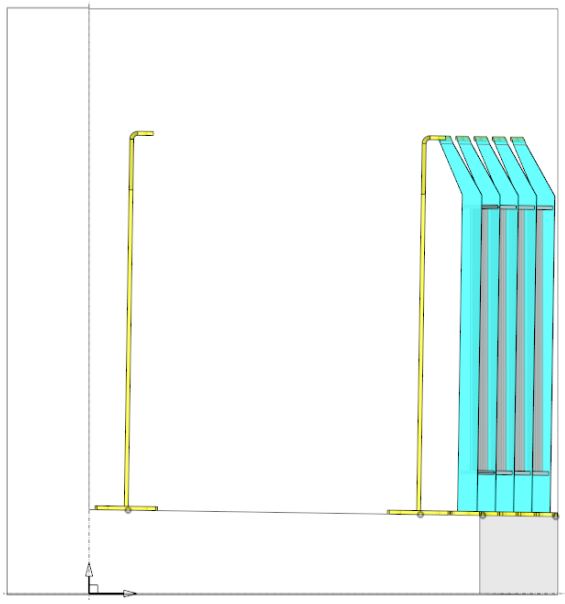
On the left side, we will create a sheet metal that will make the connection between the posts.



- Right-click on the front face of the life side of a post as shown below and select the **In Place Part** command.

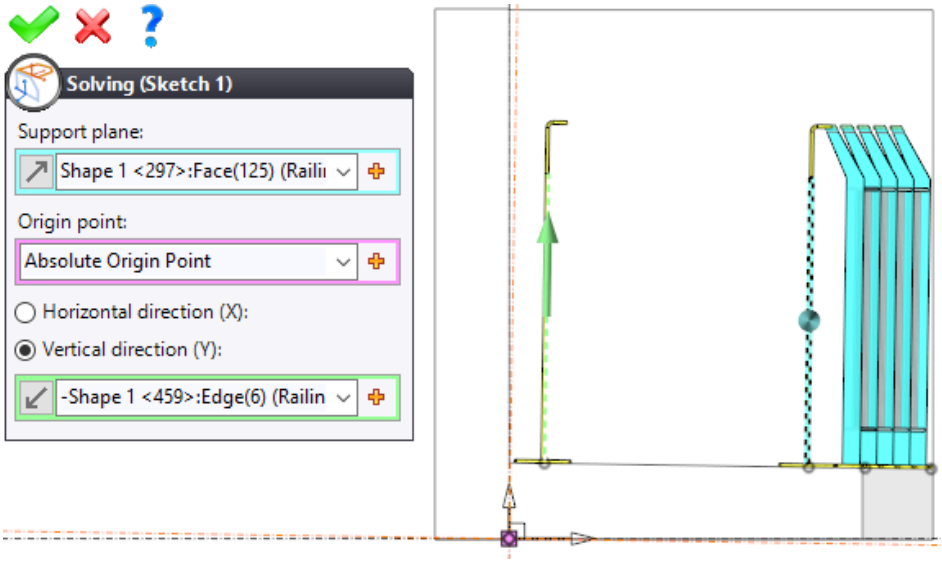


TopSolid creates a new part document and switches to the sketch context.

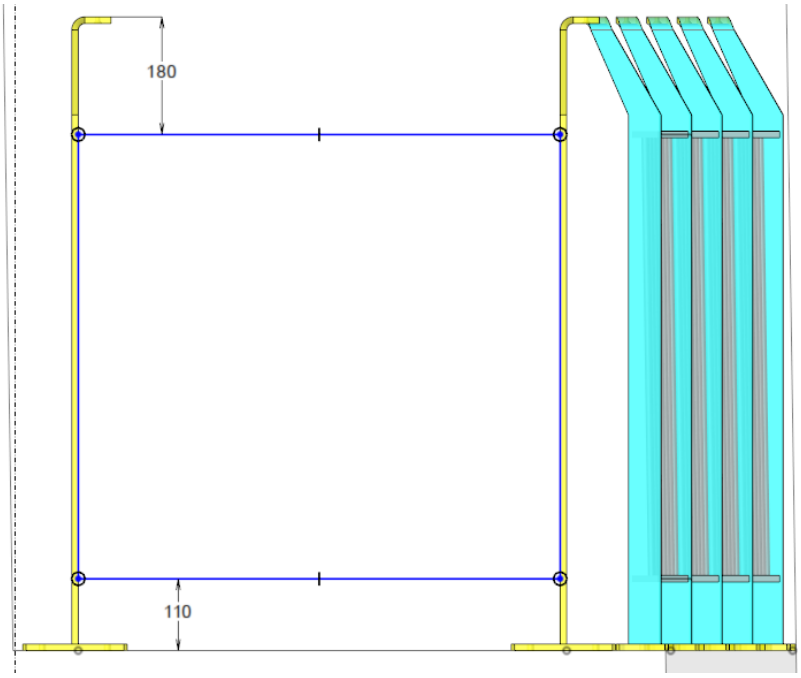


You can reorient the sketch so that the vertical is that of the posts and not that of the stage.

- To do this, from the **2D Sketch** tab, select the **Position Sketch** command.
- Select the **Vertical Direction** option and select one of the edges of the post as shown below. The arrow must be pointing upward; to reverse it, click on the icon to the left of the **Vertical Direction** field.

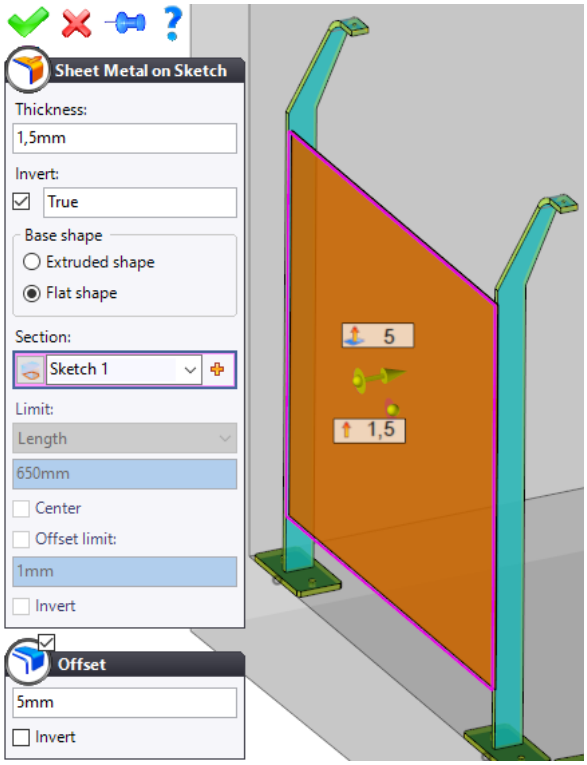


- Click on to **confirm** the operation.
- Draw the sketch shown below using the **Rectangle** command.

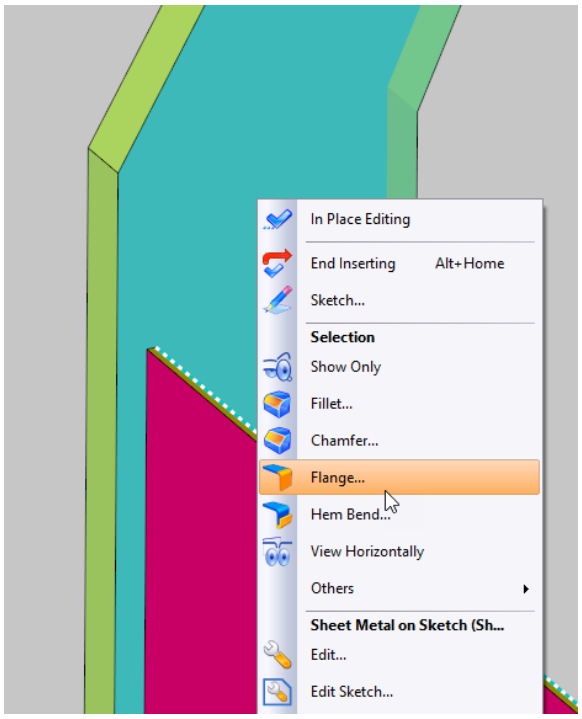


Note: If automatic dimensioning is enabled, you simply have to delete the height dimension of the rectangle in order to constrain it with respect to the post and the ground. If the post is modified later, the rectangle will follow.

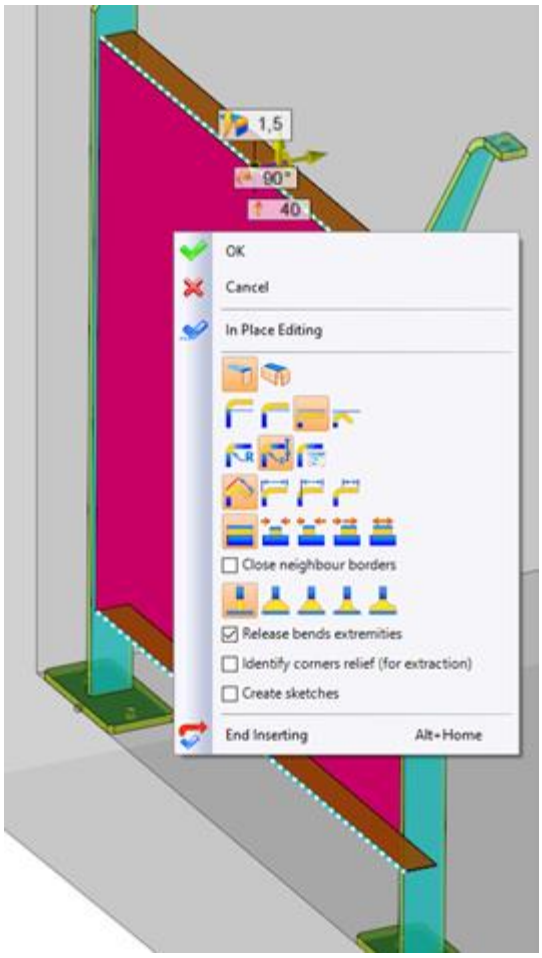
- Right-click in the graphics area, making sure that there is no active selection and then select the **Sheet Metal on Sketch** command. Enter a **thickness** of *1.5mm* and an **offset** of *5mm*.



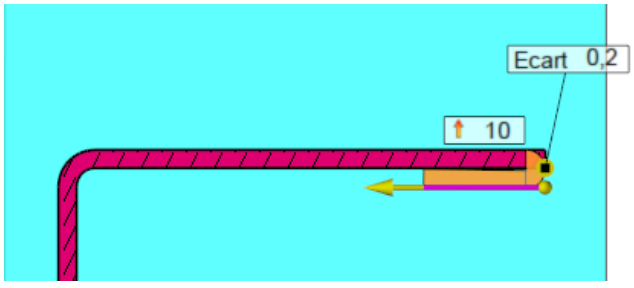
- Click on to **confirm** the operation.
- Right-click on the sheet metal and select the **Attributes** command to change its color. Select the color pink.
- Right-click on the inner edge of the sheet metal part and select the **Flange** command.

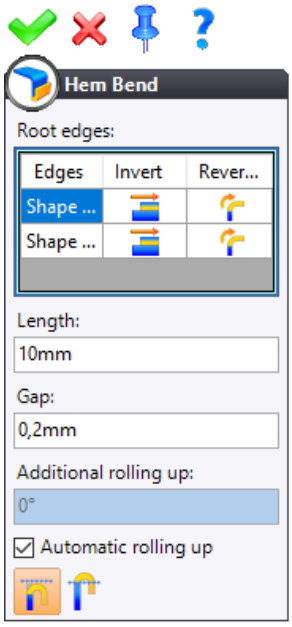


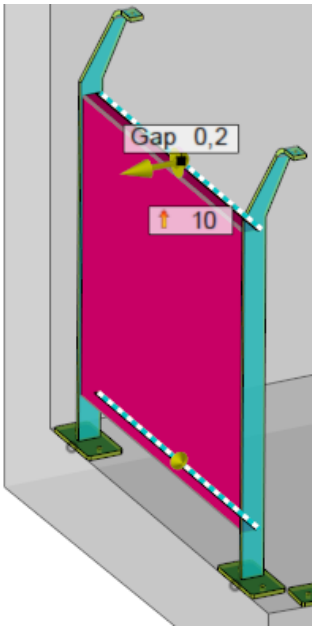
- Select the lower inner edge as well.



- Click on to **confirm** the operation.
- Via the contextual menu, add two **hem bends** as shown below.

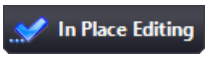




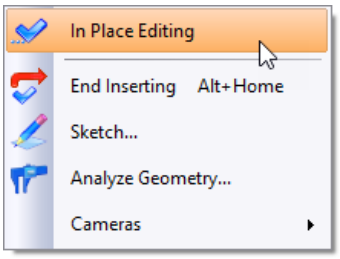


- Click on to **confirm** the operation.

The part is now complete.

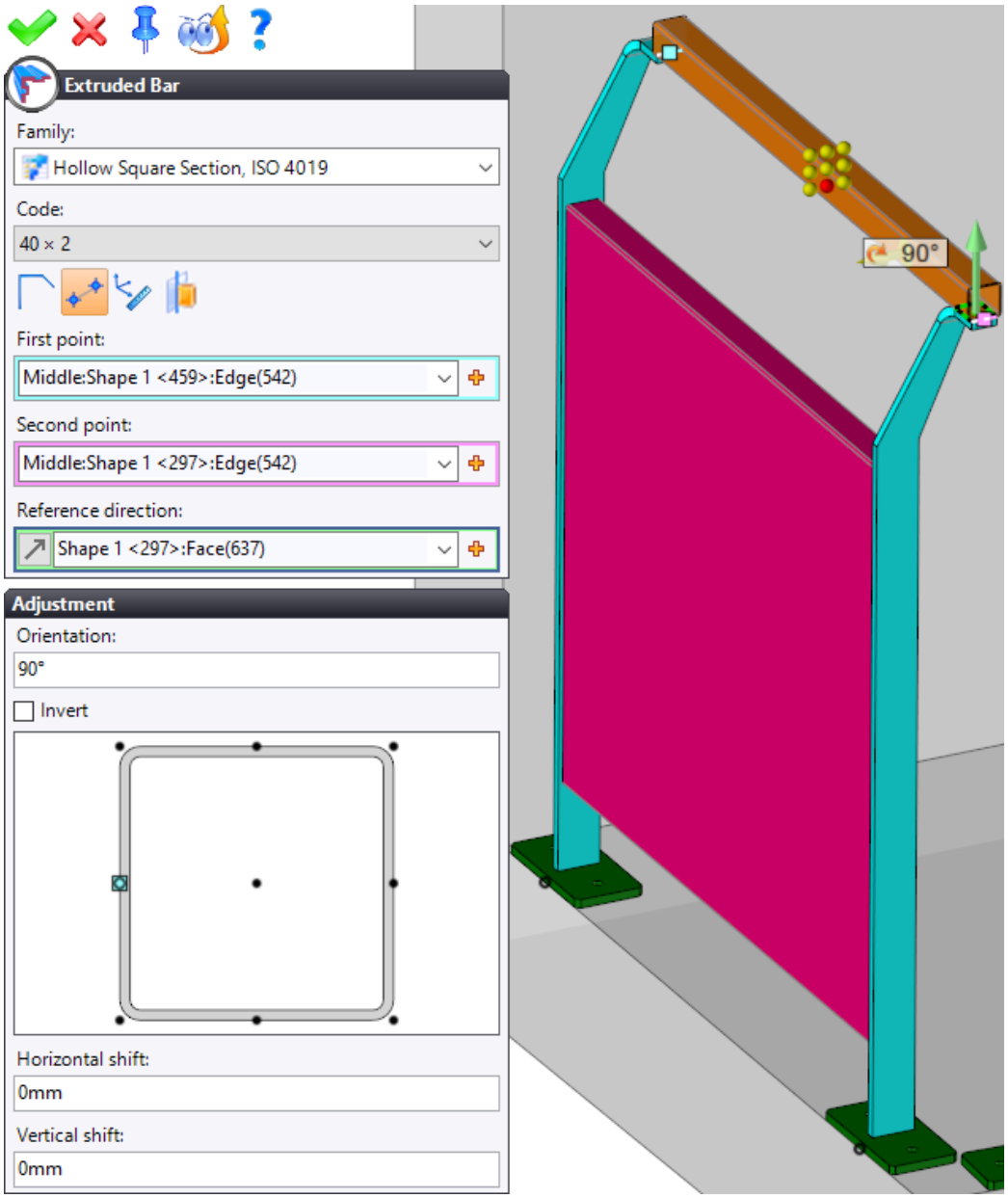
- Return to the assembly editing by clicking on the 

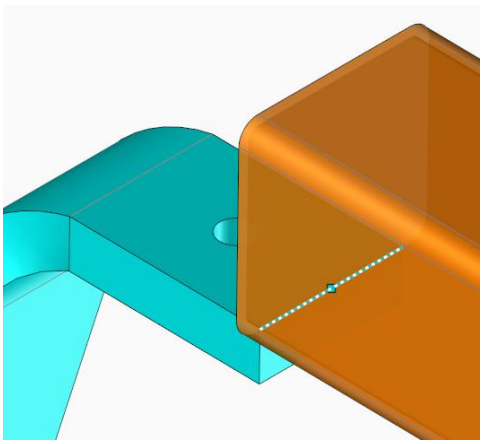
button or via the contextual menu without active selection.

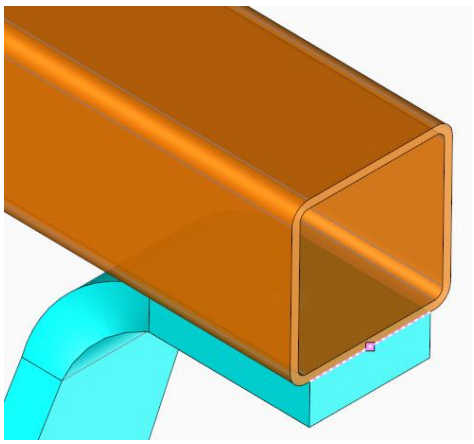


Creating the handrail

- From the **Modeling** tab, select the  **Extruded Bar** command.
- Select the **Hollow Square Section, ISO 4019** family and the **40 x 2** code.
- Select the  **Two points** mode and select the two points as shown below.

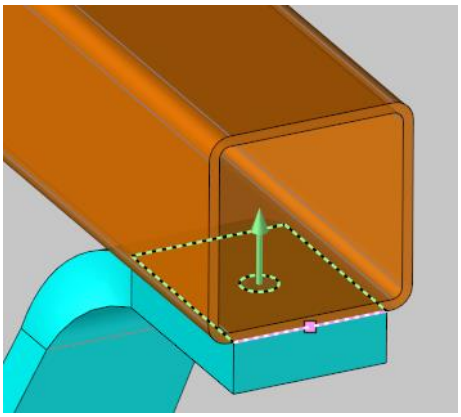






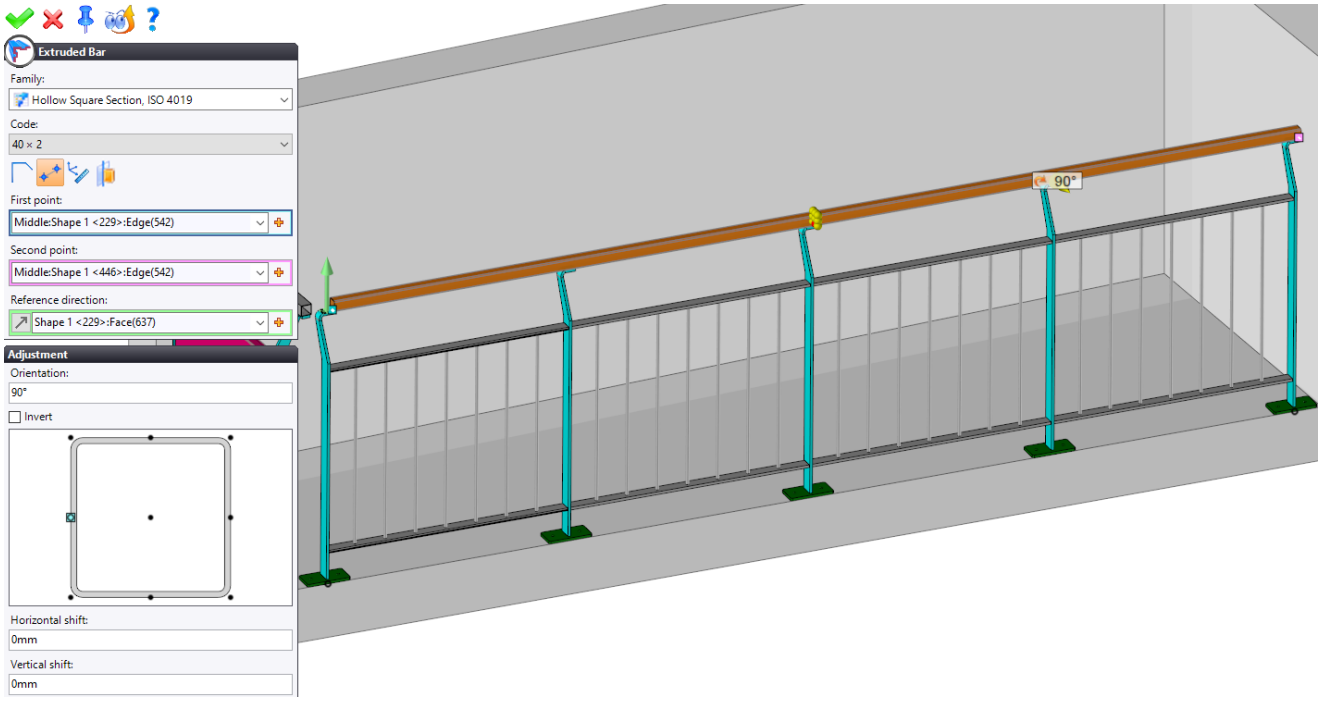
By default, **TopSolid** selects the Z axis as the **reference direction**. Since the balcony has a slope, you will have to redefine this orientation.

- In the **Reference Direction** field, select the top face of the railing.

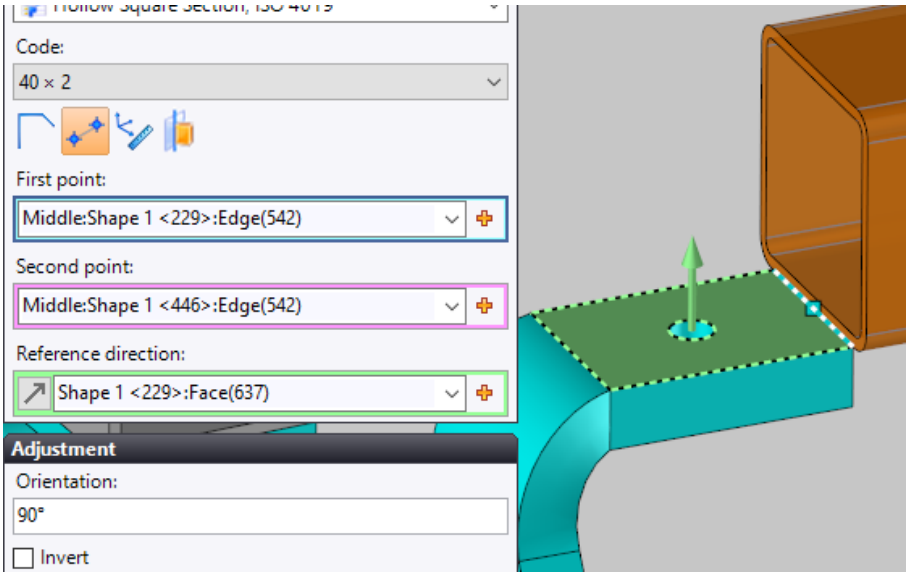


Warning: **TopSolid** keeps the values which were last used by the command. Make sure that the shifts are set to 0mm.

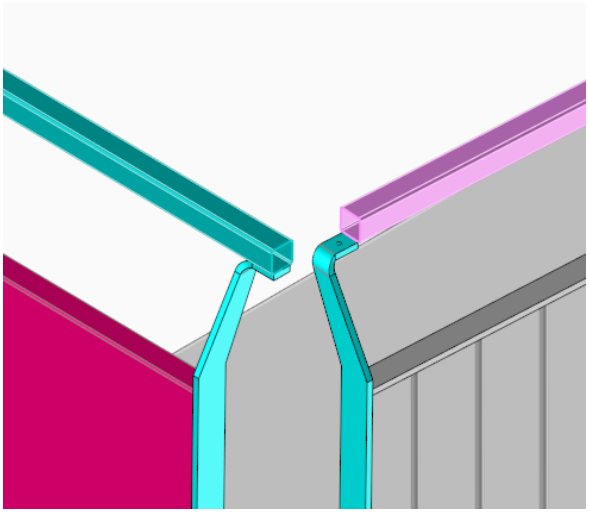
- Click on  to **confirm** the inclusion of the extruded bar.
- Repeat the operation on the second handrail.



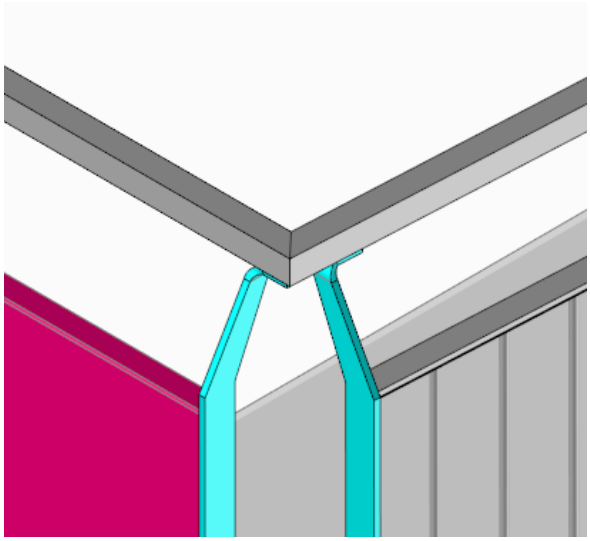
- In the **Reference direction** field, select the top face of the railing.



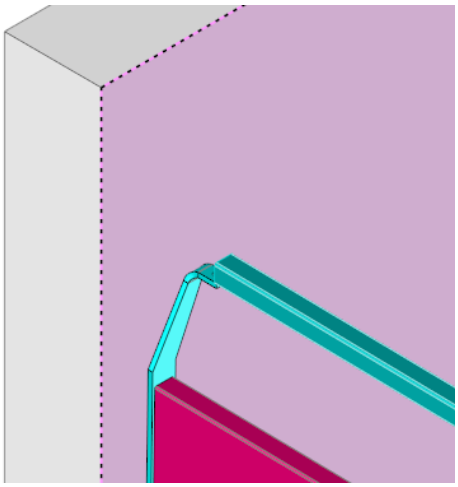
- Click on to **confirm** the inclusion of the extruded bar.
- Right-click on one of the extruded bars and select the **Miter Trim** command.
- Select the second extruded bar.



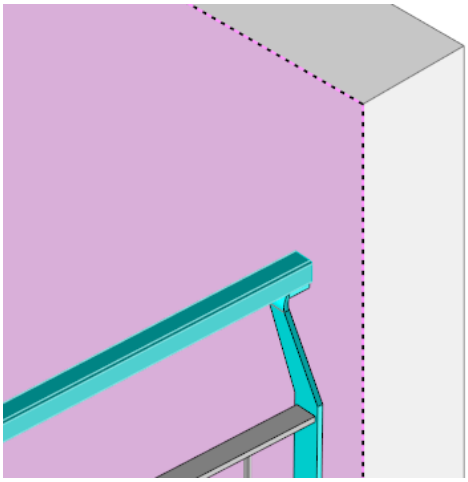
- Click on to **confirm** the operation.



- Right-click on the left handrail and select the **Planar Trim** command.
- Select the left wall as the **trimming plane**.



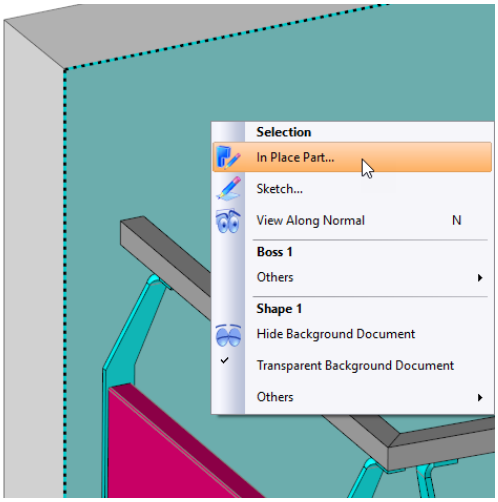
- Click on to **confirm** the operation.
- Repeat the operation on the right wall.



- Click on to **confirm** the operation.

We will now create the wall fixing plates.

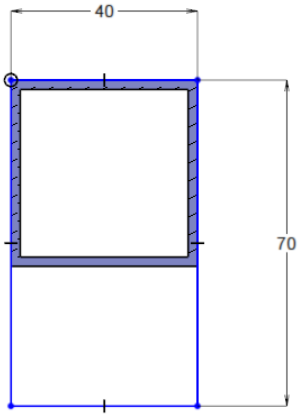
- Right-click on the face of the left wall and select the **In Place Part** command.



TopSolid creates a new part document and switches to the sketch context.

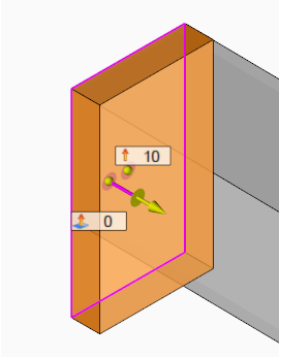
- At the top right of the screen, click on the arrow as shown below.

- Select the **Cut** mode. **TopSolid** cuts everything between the camera and the sketch plane.
- Draw the sketch as shown below.

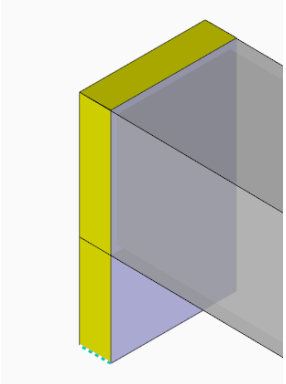


Note: The extruded bar has a slight angle due to the slope of the balcony. This being weak, we will not need to reorient the sketch in relation to the extruded bar.

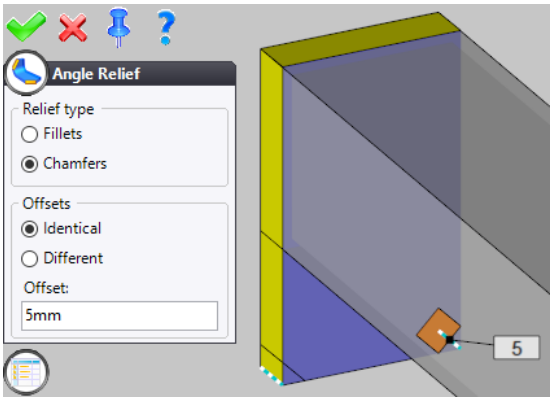
- Right-click in the graphics area, making sure that there is no active selection and then select the **Sheet Metal on Sketch** command. Enter a **thickness** of *10mm.*



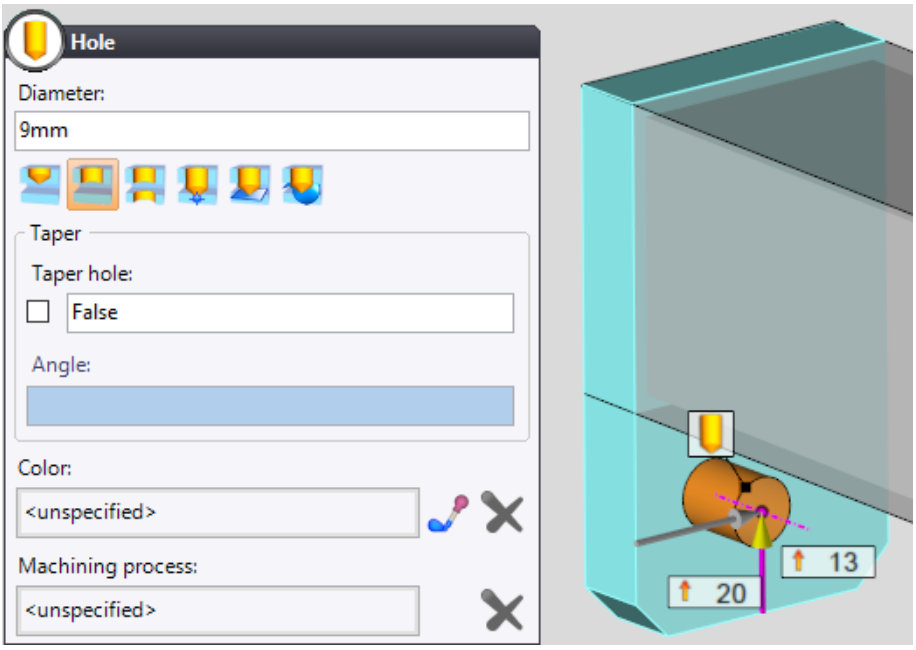
- Click on to **confirm** the operation.
- Right-click on the sheet metal and select the **Attributes** command to change its color. Select the color light purple.
- Right-click on the following edge and select the **Angle Relief** command.



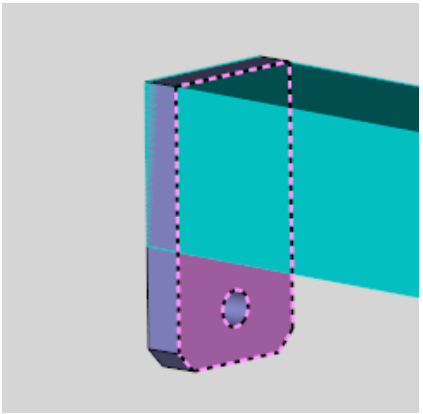
- Select the second edge as shown below.
- Select the **Chamfers** and **Identical Offsets** options and then enter an **offset** of *5mm*.



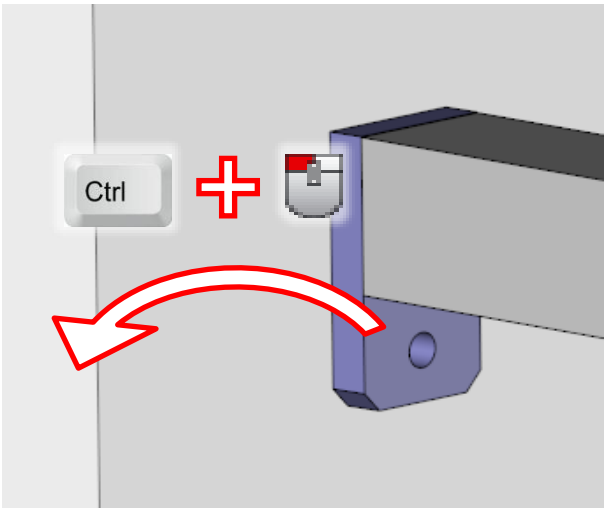
- Click on  to **confirm** the operation.
- Right-click on the top face of the part and select the  **Drilling** command.
- Create a *Ø9mm* drilling positioned as shown below.



- Click on  to **confirm** the operation.
- **Confirm** the in-place editing context.
- Right-click on the left handrail and select the  **Planar Trim** command.
- Select the top face of the plate as the **trimming plane**.

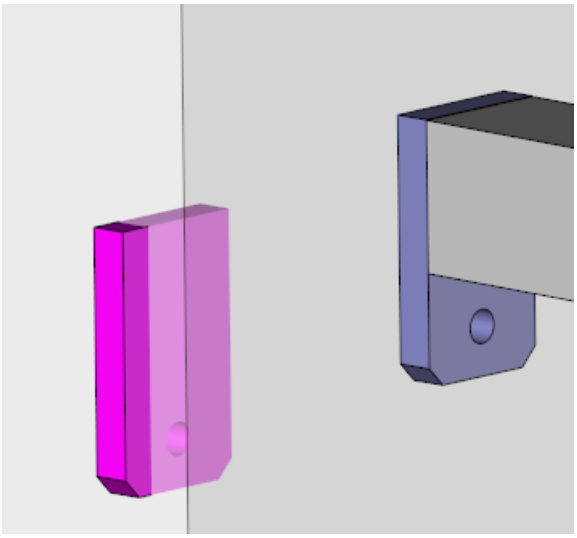


- Duplicate the plate. To do this, press and hold down the **Ctrl** key and then drag the part.

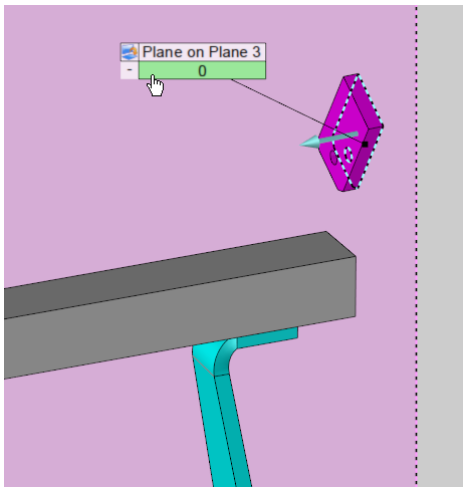


Note: You can also use this tip to copy operations in a part document.

TopSolid switches to the positioning context.

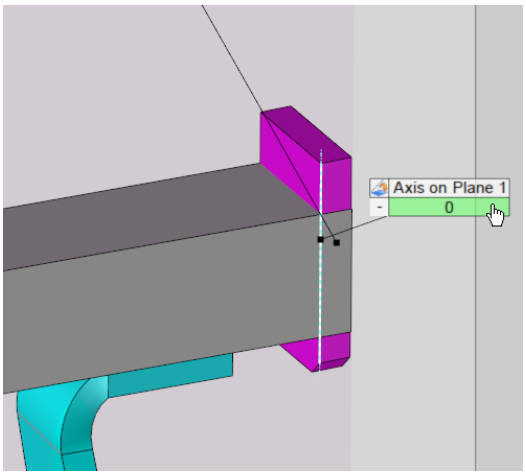


- Position the plate using the following constraints.
- Select the back face of the plate, then the right wall.

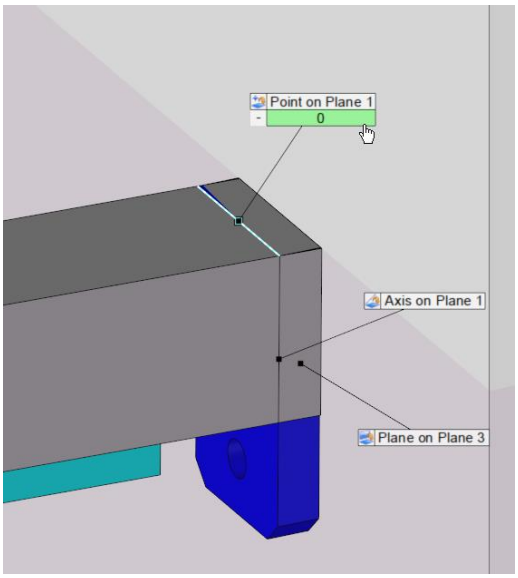


Warning: Remember that the handrail is not perpendicular to the wall.

- Select the edge as shown below, then the front face of the handrail.

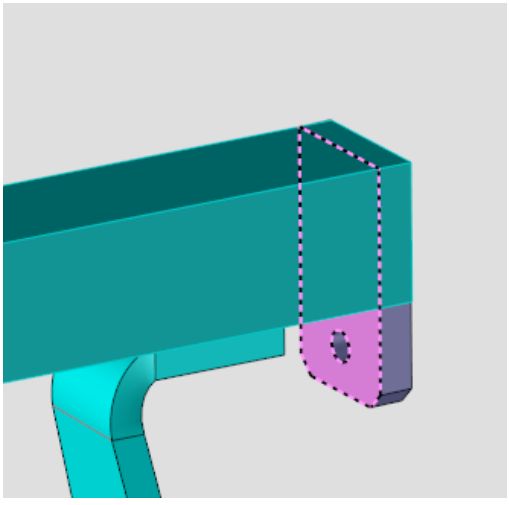


- Select the upper edge of the plate, then the upper face of the handrail.



The plate is displayed in blue, which means that it is fully constrained.

- **Confirm** the positioning context.
- Right-click on the right handrail and select the **Planar Trim** command.
- Select the top face of the plate as the **trimming plane**.



- Click on to **confirm** the operation.

During assembly, the post may interfere with the positioning of the screw on the plate side. The plate on the left handrail is the same as that on the right handrail, but you can change the right plate. This concept is called "derivation for modification". The plate can be modified while keeping a link with the original part.

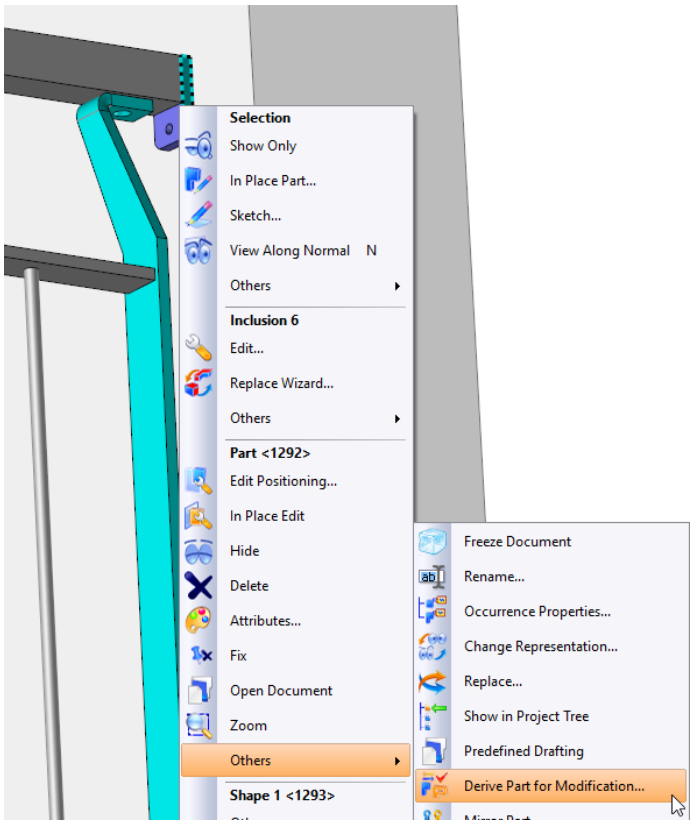
A second possibility is the "copy for modification." In this case, the link is broken and the plate is completely independent. Please note, this action is irreversible and only works on parts that were not created in place, which is not our case.

To fully understand how this works, here are a few examples of modifications made to the different cases.

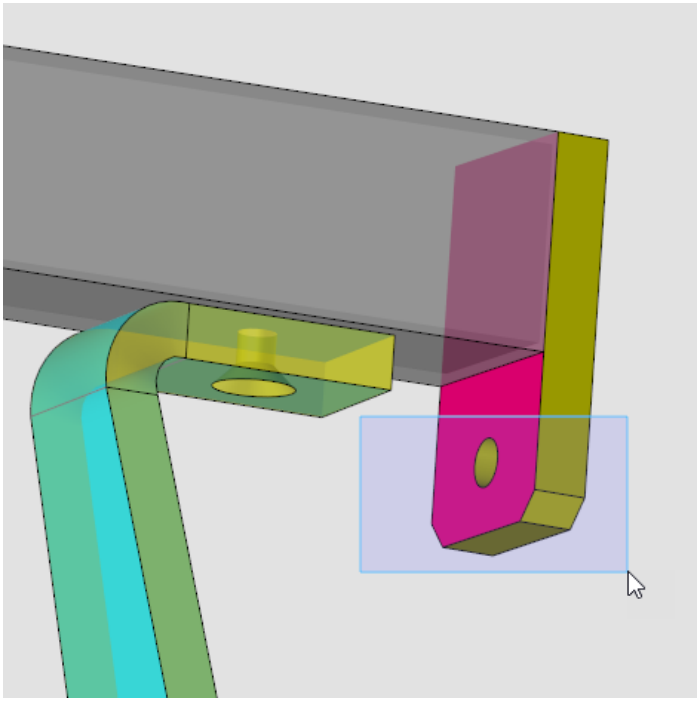
	Action	Original Plate	Repositioned	Derived for modification	Copied for modification
1	Adjusting the height of the original plate				
2	Offsetting the drilling of the plate derived for modification				
3	Adjusting the diameter of the original plate				
4	Adjusting the width of the plate copied for modification				

1: Only the plate copied for modification is not affected by the change in height.
2: No modified plate apart from the one derived for modification.
3: Only the plate copied for modification is not affected by the change in diameter.
4: No modified plate apart from the one copied for modification.

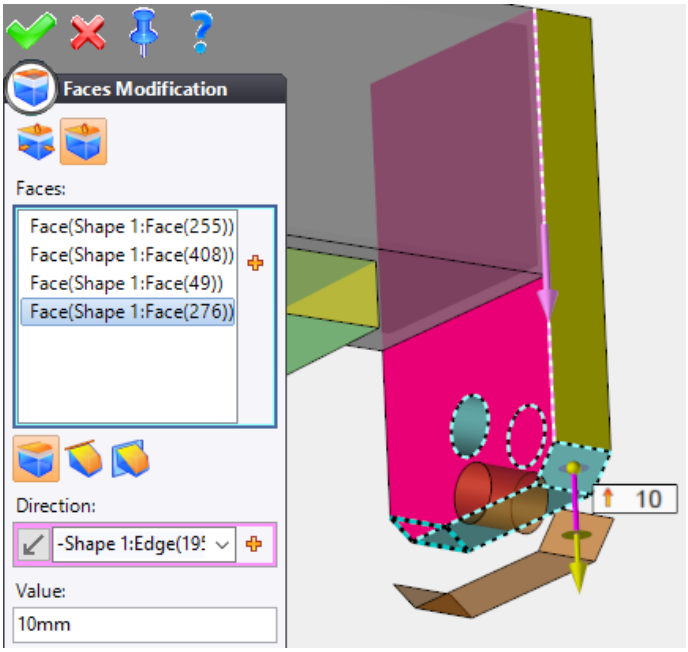
- Right-click on the right plate and select the **Others** > **Derive Part for Modification** command.



- Select the **Derived Part (New designation)** template. This will allow you to freely change the designation of the plate.
- Double-click on the plate to edit it in place.
- Right-click on the plate and select the **Attributes** command to change its color. Select the color pink.
- From the **Shape** tab, select the **Faces Modification** command.
- Select the **Move** mode. Click in the **Faces** field to enable it and select all of the faces as shown below.



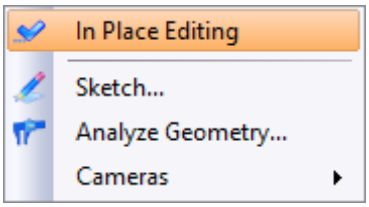
- Select an edge of the plate as the **direction** and enter a **value** of *10mm*.



- Click on to **confirm** the operation.

Note: This command also allows you to resize the parts (adjusting thickness, drilling diameter, dimensions, etc.) by disregarding the previously performed operations on the part. It is therefore very useful to modify the parts which have been imported that were not designed in **TopSolid**.

- Return to the assembly editing by clicking on the In Place Editing button or via the contextual menu without active selection.

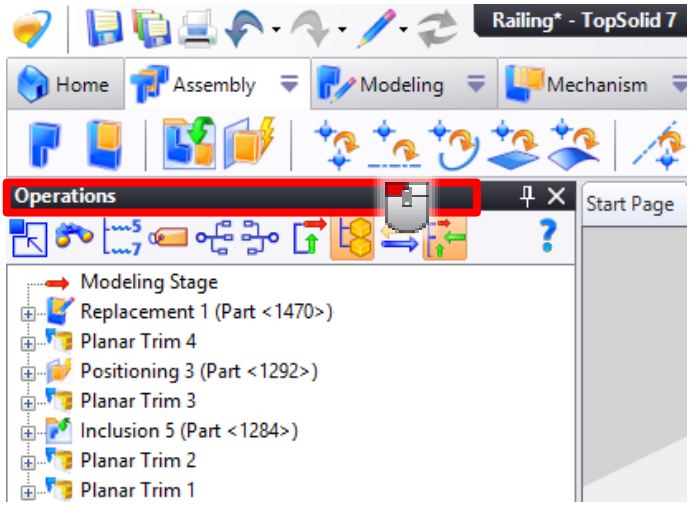


Modifying a design step from the Operations tree

Our design is now complete. We saw previously that making a change to the background document (balcony) will automatically change the railing.

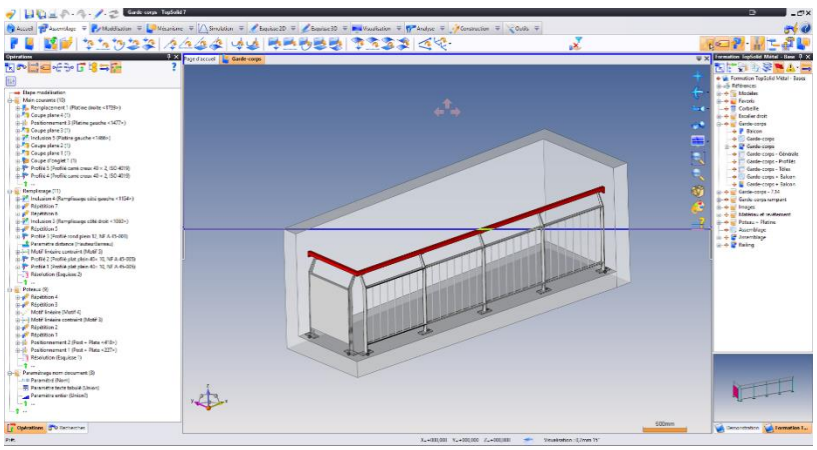
You can also to go back to a previous design operation using the Operations tree which contains the chronology of everything that has been done on the document.

- Return to the *Railing* document or open it if it has been closed.
- Display the Operations tree. If the tree is not visible on the screen, click on the **TopSolid 7** icon and select the **Display** > **Operations** command.
- You can reposition the Operations tree. To do this, click on the upper bar of the tree, then while holding down the left mouse button, drag the tree to the desired location.

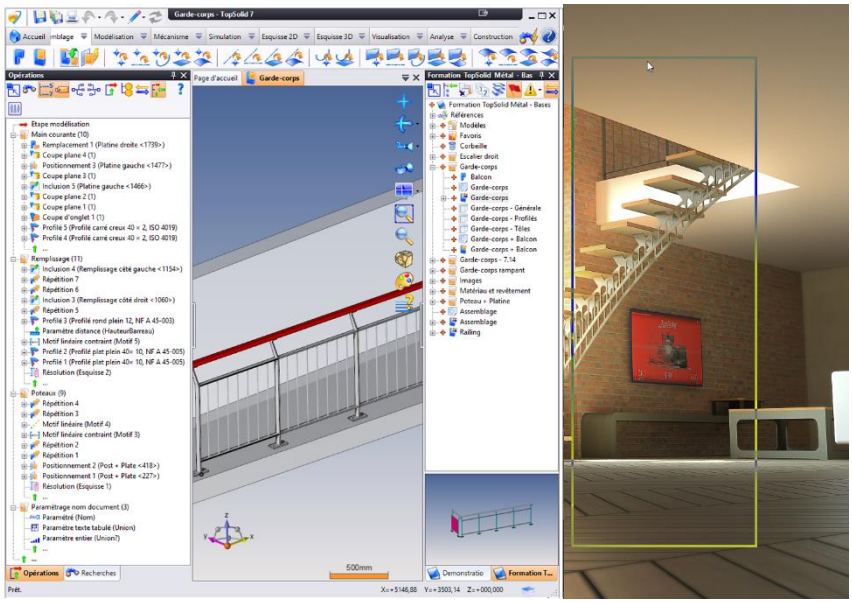


TopSolid offers four types of positioning depending on where you drag the tree.

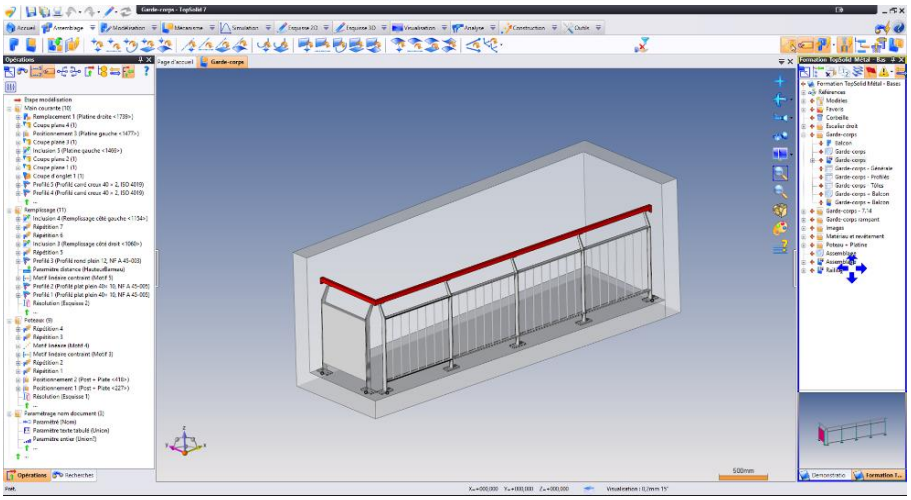
- Hooking on one side (left, top, right or bottom). If you get close to an edge, **TopSolid** then offers to hook the tree on it by displaying a blue frame with arrows.



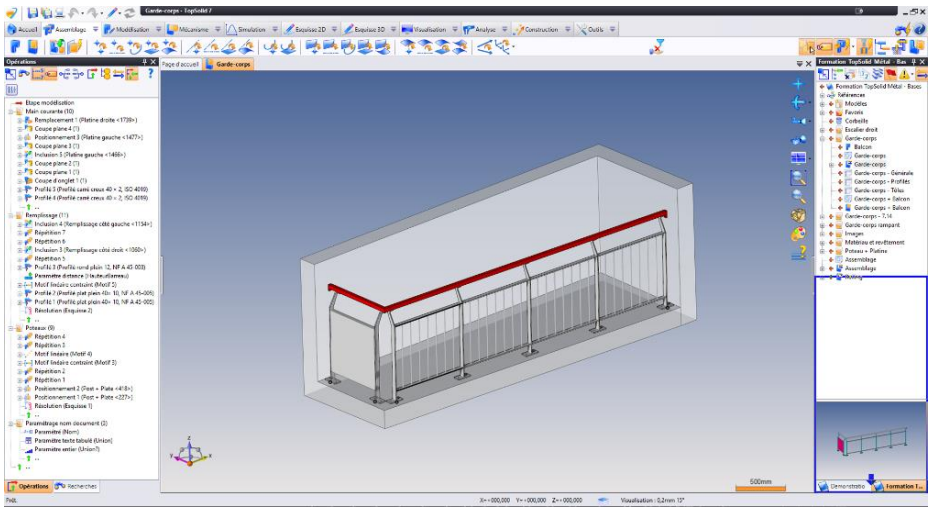
- Freely position outside of the **TopSolid** window (ideal for use on multiple screens). You simply have to drag the tree outside of **TopSolid**.



- Grouping with an existing tree (here, the Project tree). You simply have to drag the tree onto the central part of an existing tree. **TopSolid** then adds a tab at the bottom of the tree.



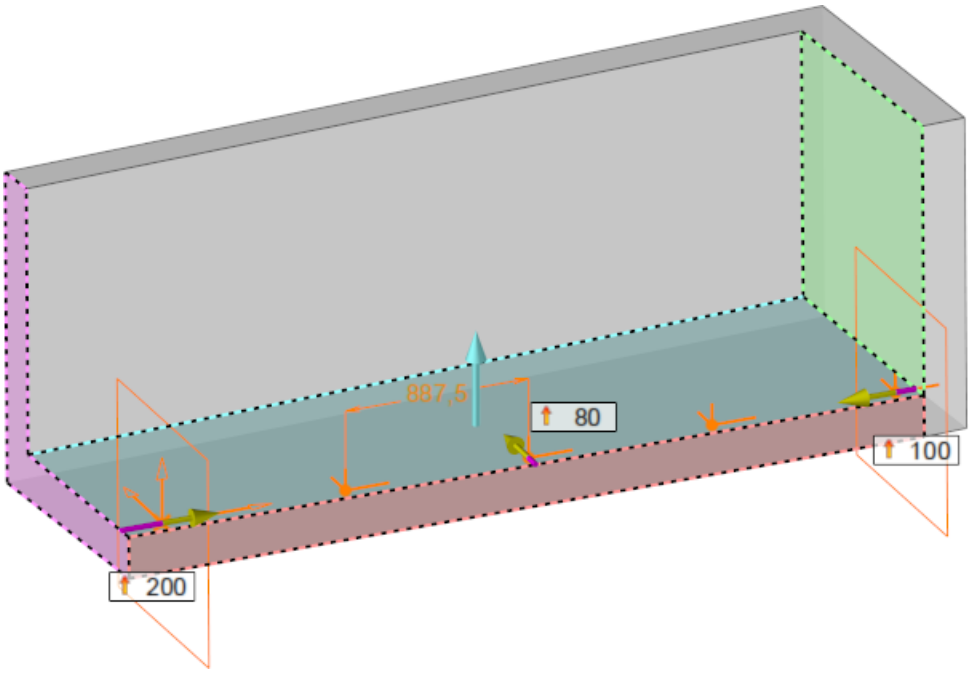
- Mosaic on an existing tree. You simply have to drag the tree onto the upper or lower section of an existing tree.



- Place the Operations tree on the left.

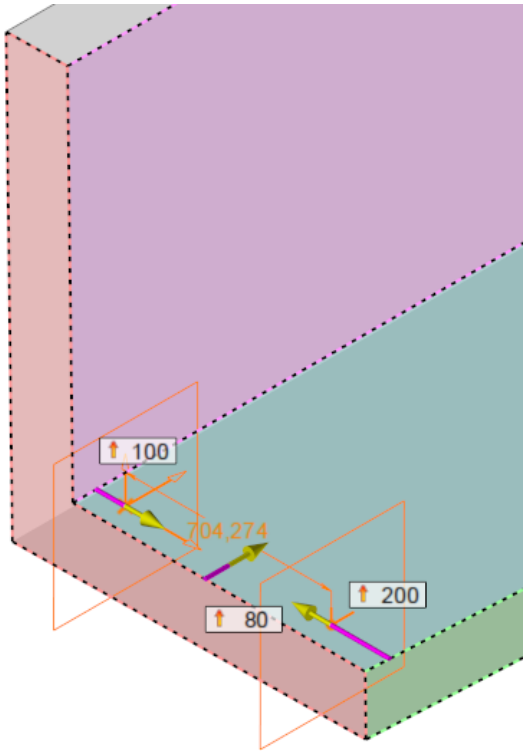
Here, we will therefore find all the operations performed in the active document, from the most recent to the oldest starting from the top.

- From the Operations tree, double-click on the **Constrained Linear Pattern (Pattern 1)** operation to edit it.
- Adjust the **offset** from 40mm to *80mm* and the **starting margin** from 160mm to *200mm*.

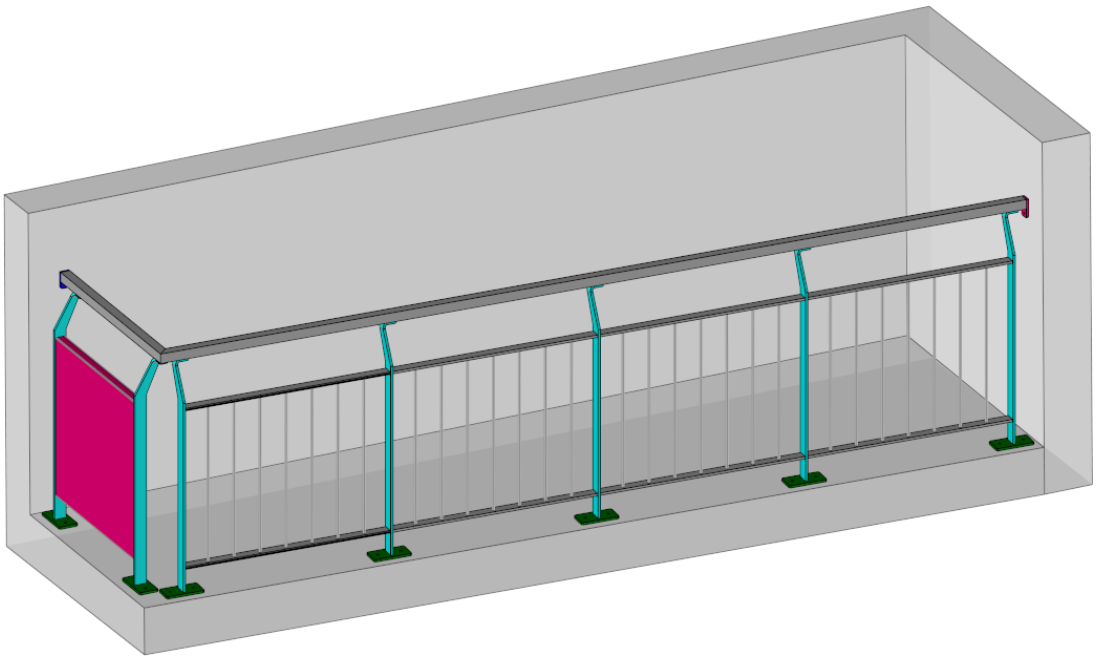


- Click on to **confirm** the pattern.

- From the Operations tree, double-click on the **Constrained Linear Pattern (Pattern 2)** operation to edit it.
- Adjust the **offset** from 40mm to *80mm* and the **end margin** from 160mm to *200mm*.

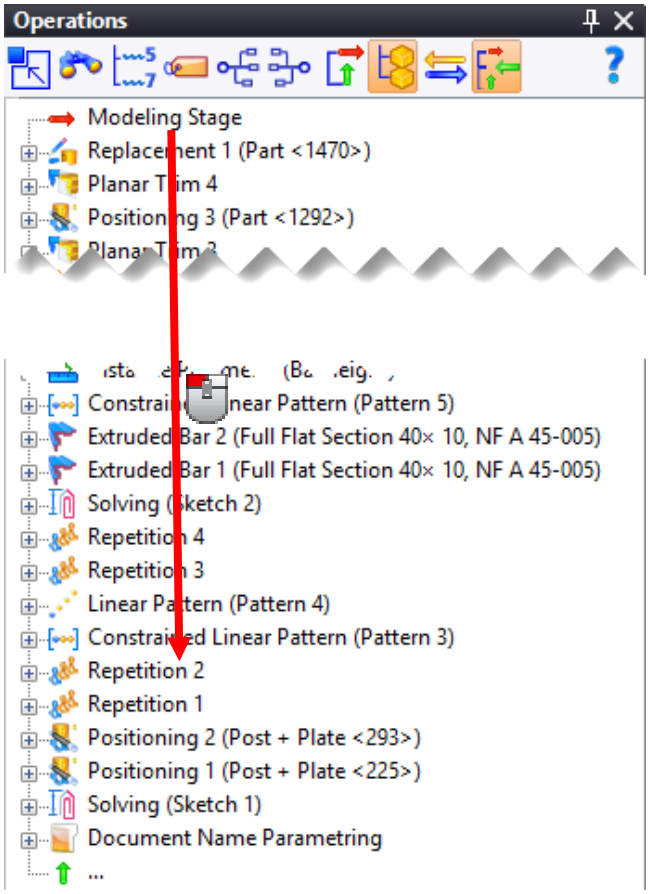


- Click on to **confirm** the pattern.

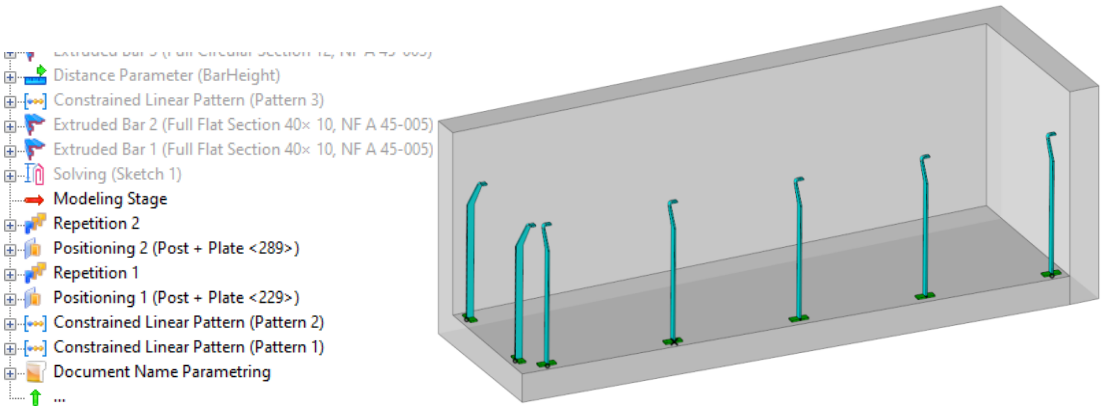


You can also display the assembly in a previous state.

- From the Operations tree, drag the **Modeling Stage** cursor between the **Solving (Sketch 2)** operation and the **Repetition 2** operation.



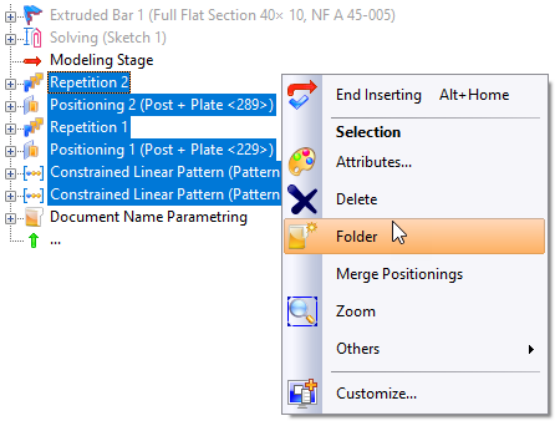
In the tree, all the operations located after the cursor appear in gray and no longer appear in the graphics area.



In our case, we have just finished positioning the posts.

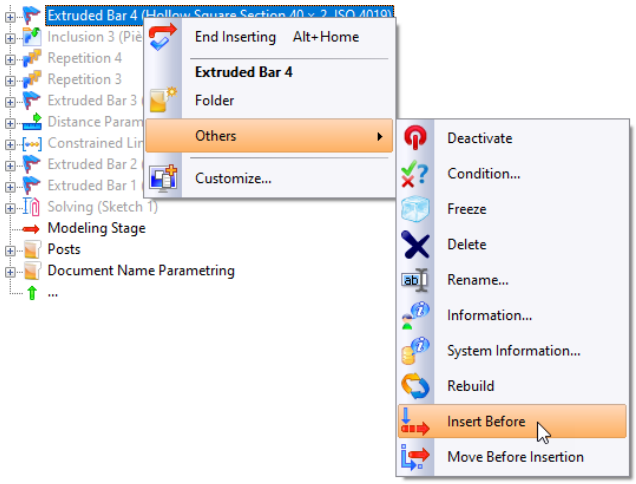
As you can see, a design can involve a large number of steps. In order to facilitate the modification of a project or its subsequent reworking, you can organize the Operations tree.

- From the Operation tree, select the **Repetition 2** operation, press the **Shift** key and while holding down the key, select the **Constrained linear pattern (Pattern 1)** operation. **TopSolid** selects all the elements present between these two operations.
- Right-click in the graphics area and select the **Folder** command.
- Rename the folder *Posts*.

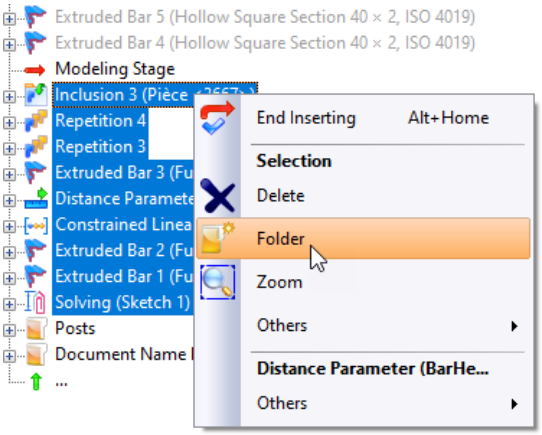


We have seen previously that to go back to a previous operation you simply have to move the **Modeling Stage** cursor. Another method exists.

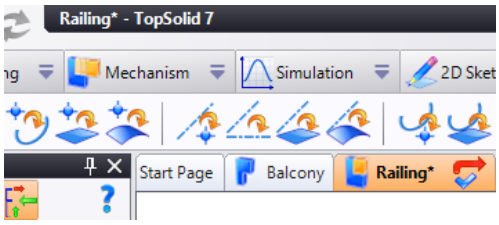
- Right-click on the **Extruded bar 4** operation (this operation must correspond to the left handrail) and select the **Others** > **Insert Before** command.



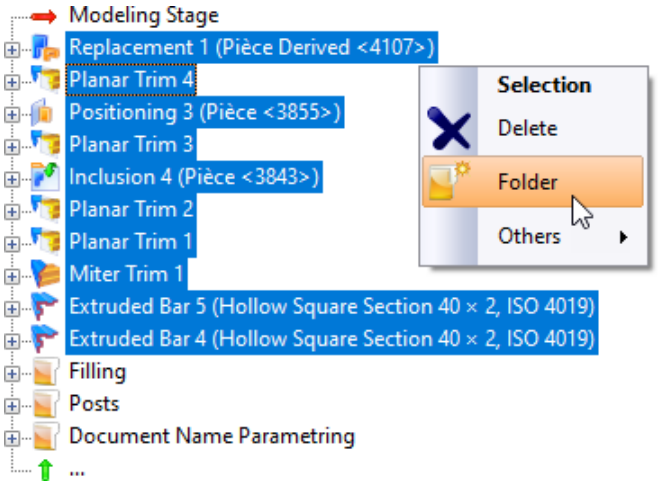
- Using the **Shift** key, select all the operations between the *Posts* folder and the **Modeling Stage** cursor, then right-click and select the **Folder** command.
- Rename the folder *Filling*.

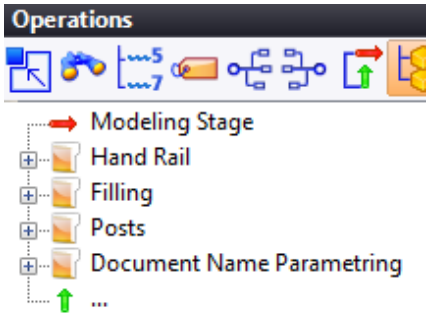


When the **Modeling Stage** cursor is not positioned at the end, **TopSolid** displays the icon on the document tab to warn the user.



- Click on the **End inserting** icon to move the cursor back to the top of the tree.
- Create a last folder called *Handrail* with the remaining operations.





Note: A final tip to move the **Modeling Stage** cursor is to click on it and, while holding down the **Alt** key, press the up or down arrow on the keyboard.

There are a number of options that can be enabled/disabled at the top of the Operations tree.

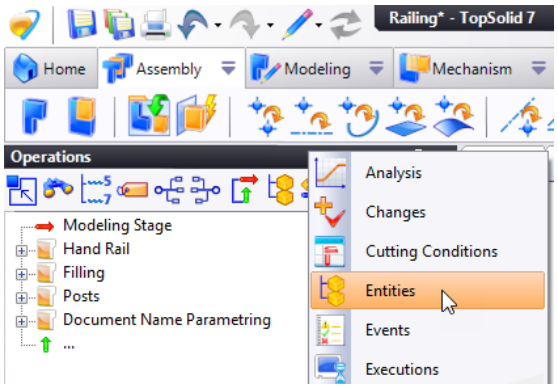
You will find below the description of each option (taken from the **TopSolid** online help).

Icon	Command
	Collapse all: Allows to collapse all opened sub-folders.
	Show search: A field appears to fill the beginning of a word to find it in the tree. The tree is automatically unrolled and the first matched word found is highlighted. The ✕ icon allows to empty the field.
	Show folders counts: Display the information counts found in the tree. For example, in the entities tree, Sketches (5) means that there are 5 sketches in this folder.
	Show definitions: If this mode is disabled, a fillet in the operations tree does not display any information. By enabling this option, an information node appears under the fillet and displays its parameters.
	Allows to display a subnode named **References** under the elements which depend on other elements. These latter elements appear in a list when you expand the subnode.
	Allows to display a subnode named **Back References** under the elements that other elements depend on (in the same document). These latter elements appear in a list when you expand the subnode.
	Show parent and children operations: Display the parent and children operations. For example, if this mode is activated in the entities tree of an assembly, in the Parts folder, you are able to unroll (⊞) a sub-assembly and display its positioning and inclusion. If the mode is disabled, you are not able to unroll this sub-assembly.
	Show entities: This mode, only available for the operations tree, allows to display the operations entities. By unrolling the operation, an entities sub-folder will be displayed (⊞). For example, by unrolling a extruded, the shape and the height dimension will be displayed.
	Show synchronizations: This mode, only available for the operations tree, allows to display synchronizations (for example, parts modified according to an assembly geometry).
	Track entities/operations: When selecting an entity or an operation in the graphic area, it is also highlighted in the tree. By default, this mode is disabled in the projects tree.
	Sort entities in alphabetical order: This mode, only available for the entities tree, allows to sort entities in alphabetical order to find them easier.
	Show entities that are not available. This mode, only available for the entities tree, allows to show entities even if they are not available, like for example a created plane when the inserting cursor is under it.

Entities tree

TopSolid has another essential tree: the Entities tree.

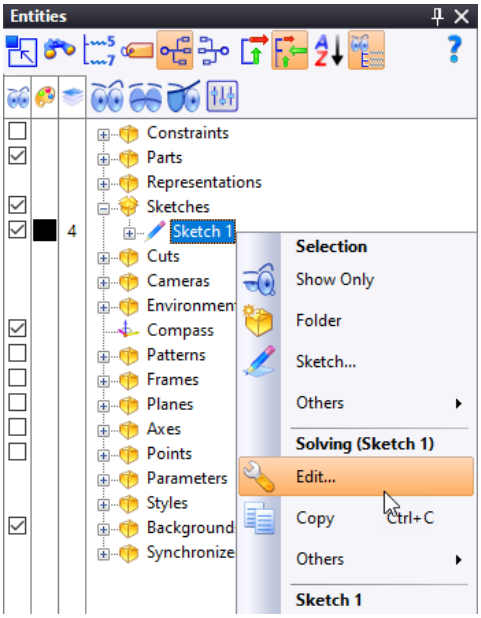
- To display the Entities tree, click on the **TopSolid 7** icon at the top left of the screen and select the **View** > **Entities** command, or right-click on the upper bar of an existing tree and select **Entities**.



- Position the Entities tree on the Operations tree.

The Entities tree automatically regroups all the created entities, which can be planes, sketches, parameters, etc.

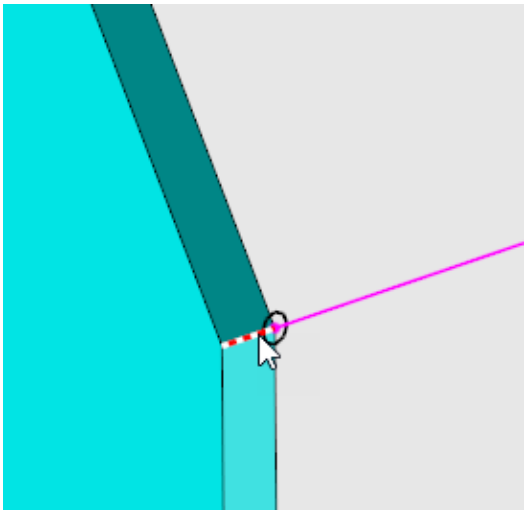
- From the Entities tree, open the **Parameters** folder. We find the *BarHeight* parameter.
- Open the **Sketches** folder, right-click on **Sketch 2** and select the **Edit** command.

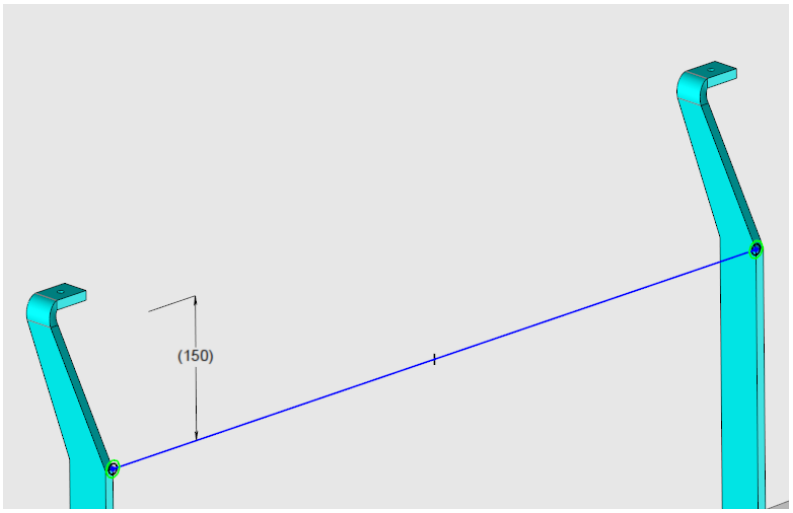


Note: If the symbol (-) appears in front of a sketch in the Operations/Entities tree, it means that the sketch is underconstrained.

- Right-click on the 18mm dimension and select the **Deactivate** command. The dimension is in brackets; it is no longer driving but just informative.

- Add a coincidence constraint to position the segment as shown below. To do this, drag the segment onto the edge of the post.





- **Confirm** the sketch by double-clicking with the right mouse button in the graphics area without active selection.

You will notice that the *BarHeight* parameter has been automatically adjusted.

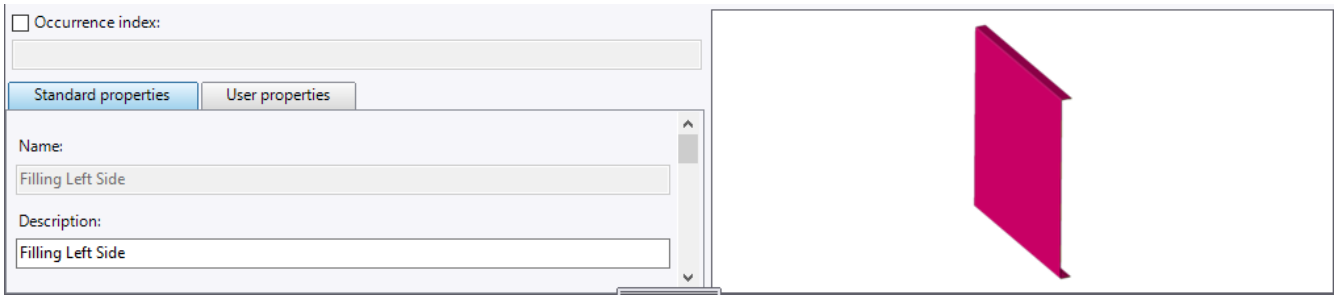
- **Save** the *Railing* assembly document.

Bill of materials of the railing and identification

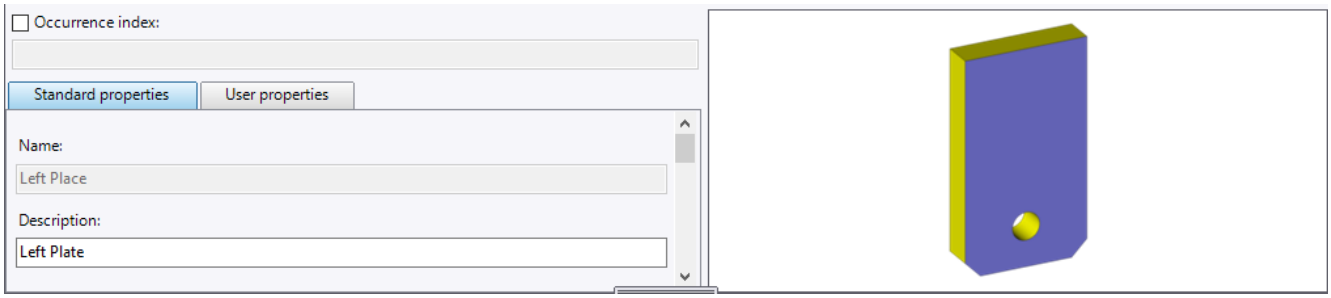
General bill of materials

This bill of materials allows you to have a global and tree view of the assembly. It is useful for viewing the different sub-assemblies and their constituents.

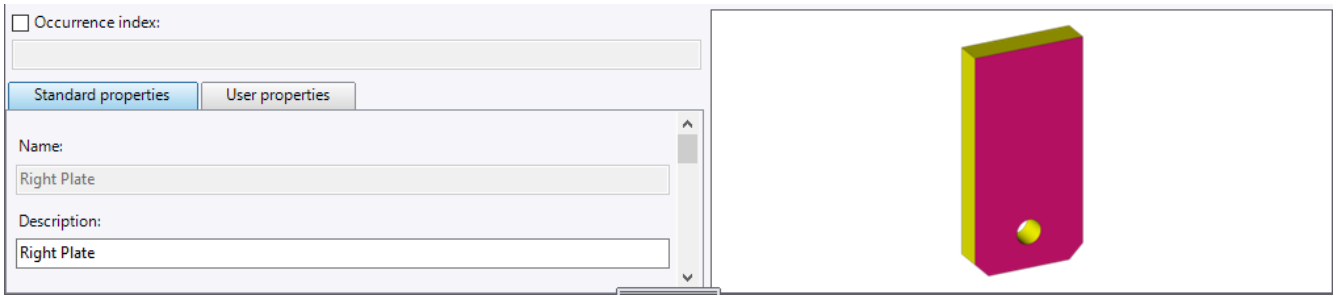
- Right-click on the *Railing* assembly document's tab and select the **Bill of Material** command.
- Select the **Multi-level** template from the **Steel Standard Templates - United States** folder and click on to **confirm** the operation.
- Click on to **confirm** the **Assembly** dialog box that appears.
- Rename the following parts:
  - *Filling left side*



  - *Left plate*



  - *Right plate*



When renaming, **TopSolid** reorganized the bill of materials by applying a sorting to the designation (default sorting). As a result, the bill of materials frames are no longer in order.

ID.	QTY	DESCRIPTION
	1	Railing
1	1	Filling Left Side
7	28	Full Circular Section 12, NF A 45-003
12	4	Full Flat 40× 10, NF A 45-005
13	4	Full Flat 40× 10, NF A 45-005
2	1	Hollow Square Section 40 × 2, ISO 4019
3	1	Hollow Square Section 40 × 2, ISO 4019
4	1	Left Plate
5	7	Post + Plate
5.1	1	Plate
5.2	1	Post
8	1	Right Plate

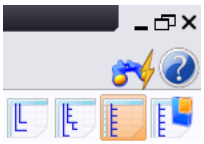
- To reinitialize the indexes, select the **Reinitialize Bill of Material Indexes** command from the **Bill of Material** tab.
- **Save** the bill of materials.

Bill of materials for identification by manufacturing index

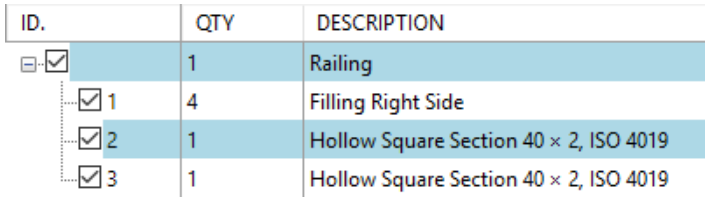
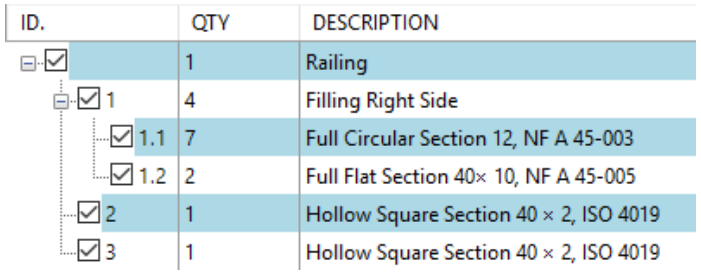
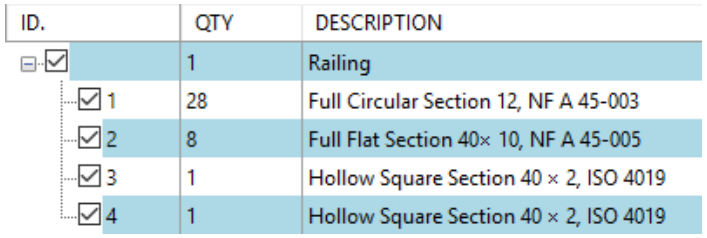
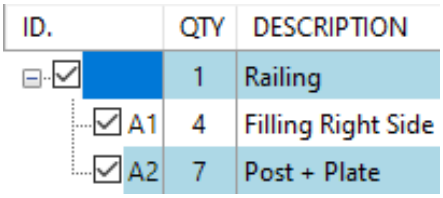
In order to facilitate the production of a work, you have to identify the parts and assemblies to be produced. In the interest of optimization, it is necessary to group identical parts and assemblies as much as possible in order to avoid creating similar production documents.

- Right-click on the *Railing* assembly document's tab and select the **Bill of Material** command.
- Select the **Part-Flat** template from the **Steel Standard Templates - United States** folder, then click on to **confirm**.

The assemblies are ignored and all the parts are regrouped. This is possible thanks to the four display modes available at the top right of the screen.



Here is a comparison of the four display modes:

Top level			Corresponds to the highest level. The detail of the sub-assemblies is not displayed.
Multi-level			Provides a hierarchy view of the assembly.
Flat			This is the lowest level. The intermediate assemblies are hidden. This is the ideal mode to create cutting rates.
Assembly Only			Only the assemblies are kept without hierarchical distinction.

To calculate its groupings, **TopSolid** uses two elements by default:

- the properties used as the columns in the bill of materials;
- the geometry of the part.

Note: You can delete these criteria by editing the **BOM** operation and unchecking the **Compare parts geometry** and **Groups by properties** boxes.

Properties have a secondary role; they make it possible to order the list of parts to be indexed.

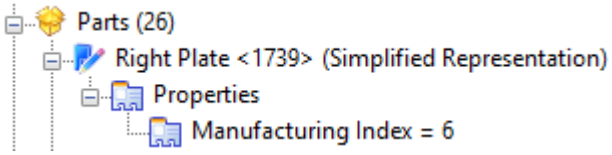
The default sorting is as follows:

- Part number
- Manufacturer
- Manufacturer part number
- Material
- Thickness
- Length

As for the multi-levels bill of materials, the indexes are calculated. We will use these indexes to identify our parts by returning them to the assembly.

- From the **Bill of Material** tab, select the **Manufacturing Index** command.

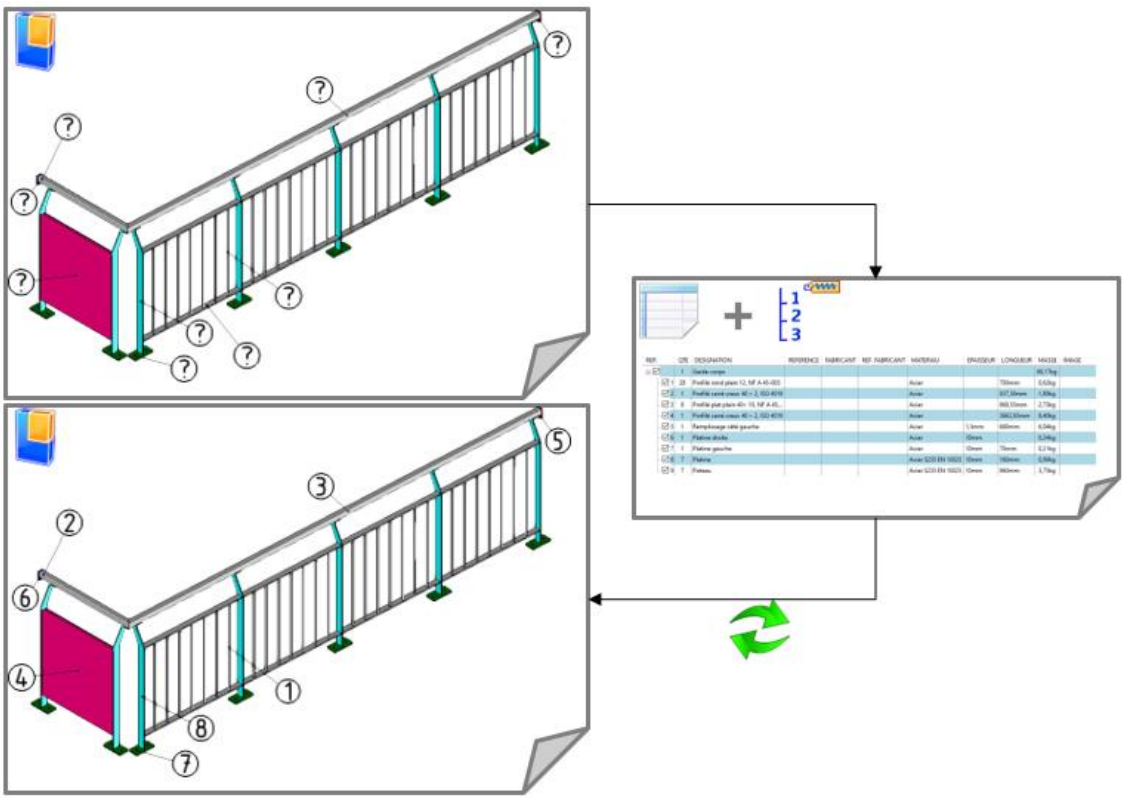
At this stage, **TopSolid** copies the indexes on the different occurrences of the assembly, by adding a property named **Manufacturing Index** to each of them. This property is visible on each occurrence in the **Parts** folder of the assembly document's Entities tree.



The manufacturing index are updated manually by relaunching the command. You can delete them by using the **Clear Manufacturing Index** command in the **Bill of Material** tab.

To summarize:

- Once the assembly is completed, create a *Parts – Flat* bill of materials.
- Select the **Manufacturing Index** command.
- The assembly is automatically updated with all the manufacturing indexes.



Now let us move on to identifying the assemblies.

- Right-click on the *Railing* assembly document's tab and select the **Bill of Material** command.
- Select the **Assemblies-Flat** template from the **Steel Standard Templates - United States** folder, then click on to **confirm**.

Warning: The geometric comparator only works for the parts. For the assemblies, **TopSolid** is based on the priorities used in the BOM.

This BOM has a default index rule which can be edited using the **Index Rules** command.

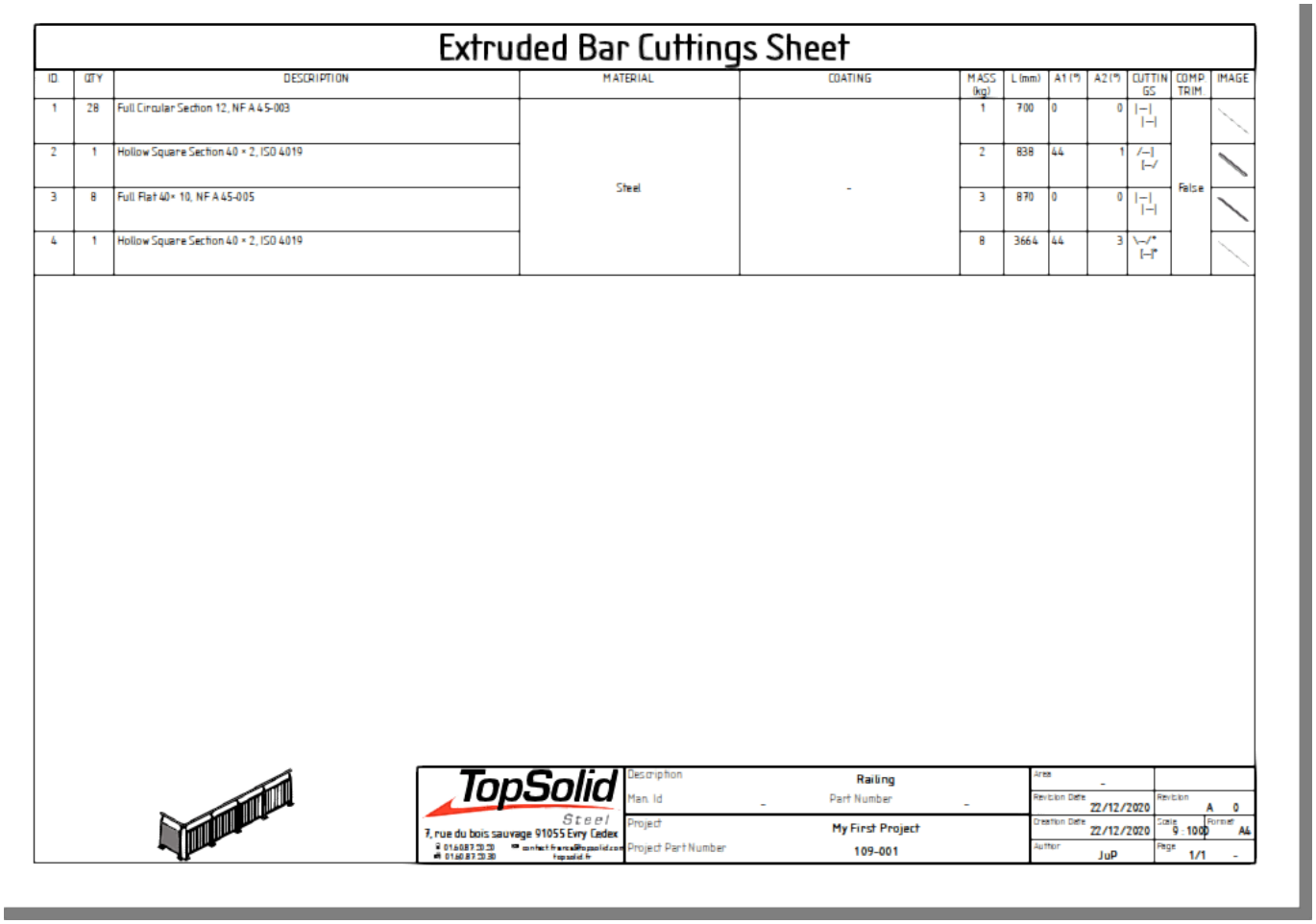
- From the **Bill of Material** tab, select the **Manufacturing Index** command.

Cutting sheets

The cards are automatic drawings made up of a title block and a schedule. These are documents ready to be printed.

Extruded bar cutting sheets

- Right-click on the *Railing* assembly document's tab and select the **Drafting** command.
- In the **Steel Standard Templates - United States** folder, open the **Sheets** sub-folder, select the **A4 extruded bar cutting sheet** template, then click on to **confirm** the operation.



Extruded Bar Cuttings Sheet

ID	QTY	DESCRIPTION	MATERIAL	COATING	MASS (kg)	L (mm)	A1 (°)	A2 (°)	CUTTINGS	COMP TRIM	IMAGE
1	28	Full Circular Section 12, NF A 45-003	Steel	-	1	700	0	0	|—| |—|	False	
2	1	Hollow Square Section 40 × 2, ISO 4019			2	838	44	1	/—| |—/		
3	8	Full Flat 40× 10, NF A 45-005			3	870	0	0	|—| |—|		
4	1	Hollow Square Section 40 × 2, ISO 4019			8	3664	44	3	\—/* |—|		

TopSolid Steel
7, rue du bois sauvage 91055 Evry Cedex

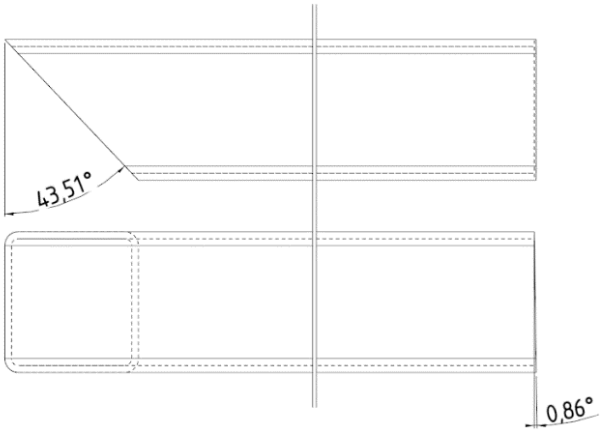
Description: Railing
Man. Id: - Part Number: -
Project: My First Project
Project Part Number: 109-001
Revision Date: 22/12/2020
Creation Date: 22/12/2020
Author: JuP
Revision: A 0
Scale: 0 : 1000 Format: A4
Page: 1/1

Note: The document template to create the extruded bars includes the symbolization of the cuts. The upper symbol (on the left in the bill of materials) represents the front view of the extruded bar. This is the great height for rectangular tubes, IPE, etc.
The second symbol represents the top view.

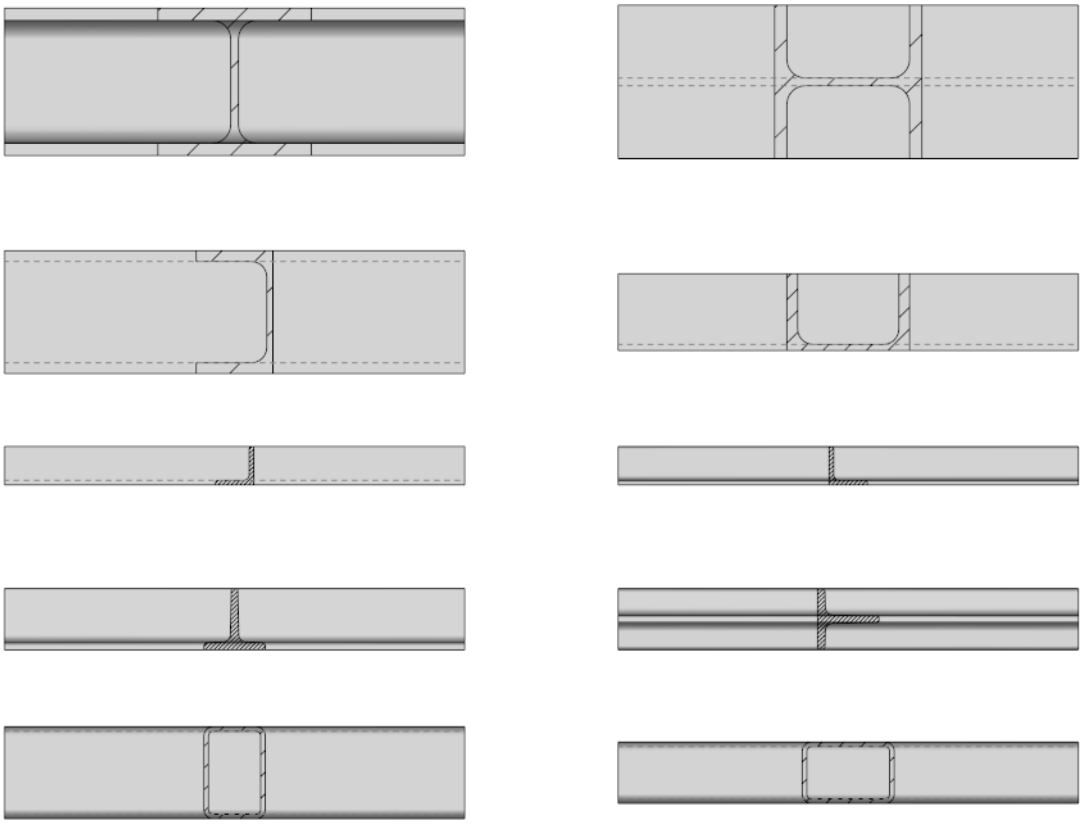
|: no cutting
[: cutting on another plane
/: positive cutting
****: negative cutting
*: disoriented cutting
/ ! \: symbol calculation problem

Example:

Angle 1: 43.5°
Angle 2: 0.9°
Symbol: /--] [--/
Extruded bar:



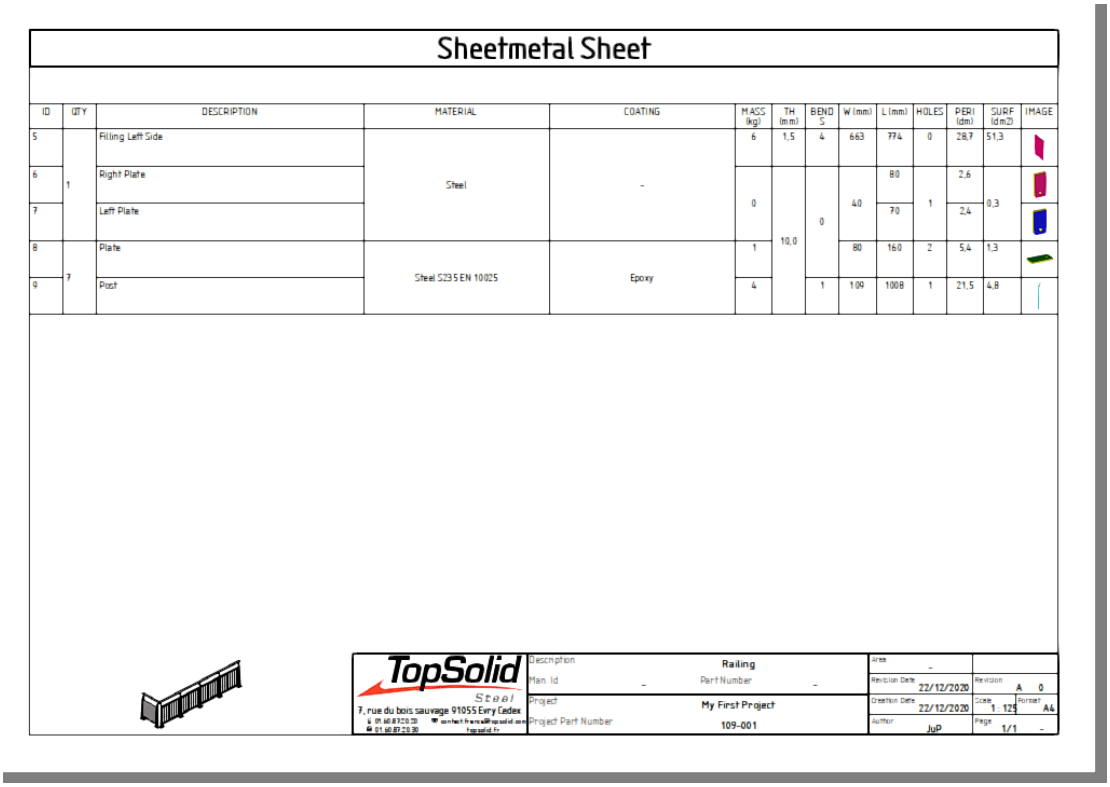
Here is the orientation of the main extruded bars in relation to the two symbols.



- **Save** the drafting document.

Sheetmetal sheet

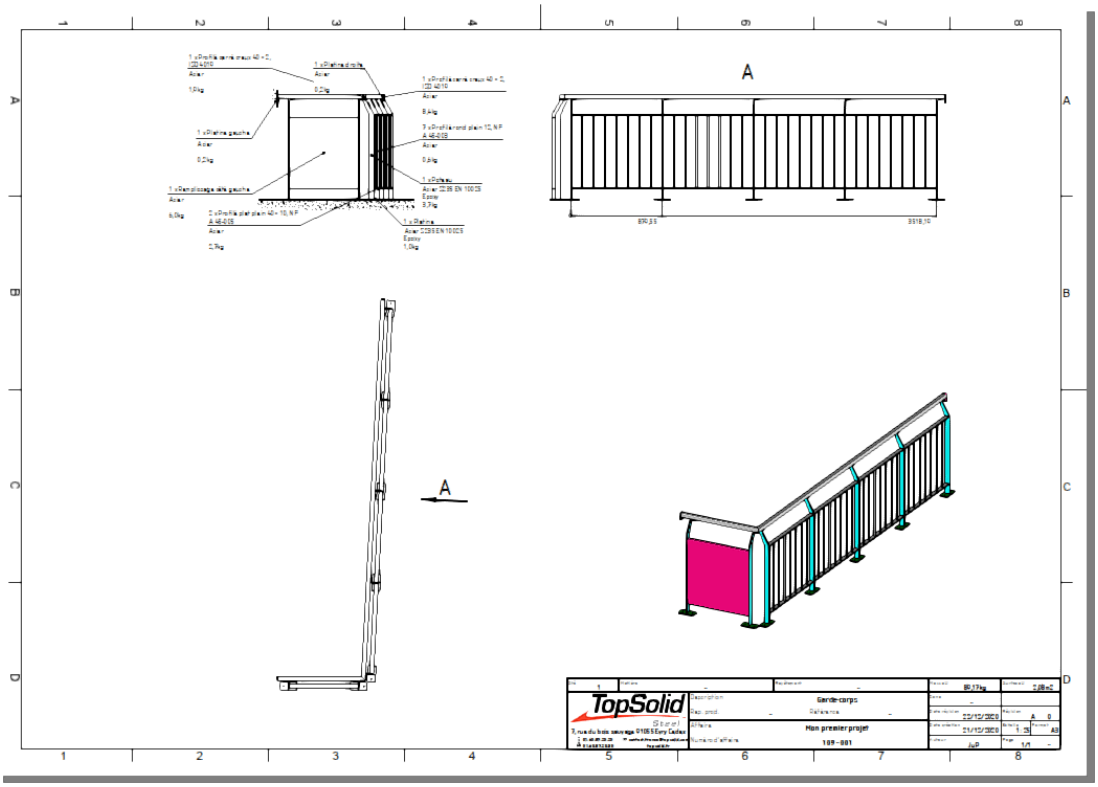
- Right-click on the *Railing* assembly document's tab and select the **Drafting** command.
- In the **Steel Standard Templates - United States** folder, open the **Sheets** sub-folder, select the **A4 extruded bar cutting sheet** template, then click on to **confirm** the operation.



Note: This sheet can be printed directly. To do this, you simply have to select the **Print** command in the system toolbar at the top left of the screen.

- **Save** the drafting document.

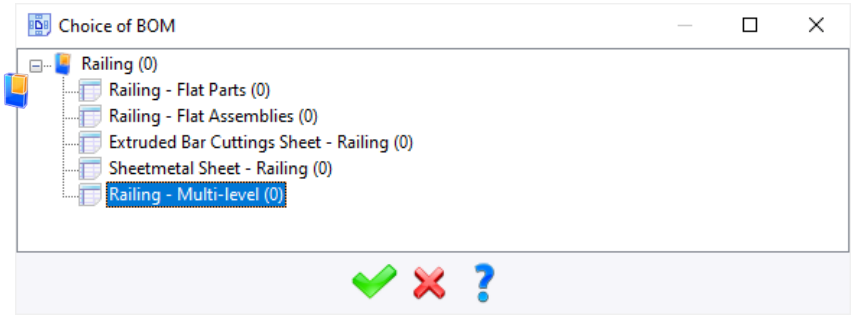
Railing drafting



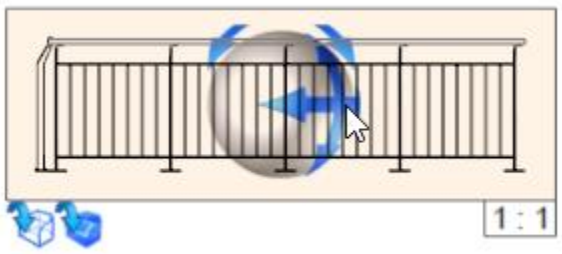
- Right-click on the *Railing* assembly document's tab and select the **Drafting** command.
- Select the **Assembly A3 ISO Landscape** template from the **Steel Standard Templates - United States** folder.

TopSolid prompts you to select a reference BOM.

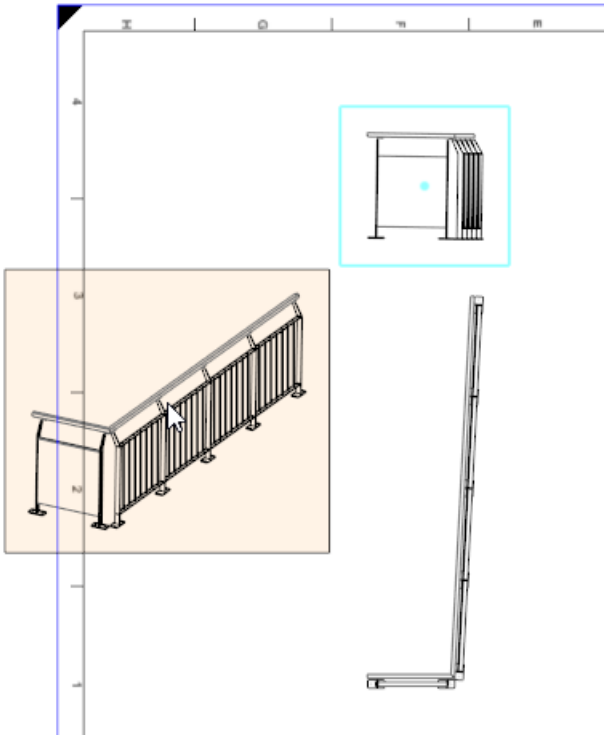
- Select the *Railing – Multi-level* bill of materials.



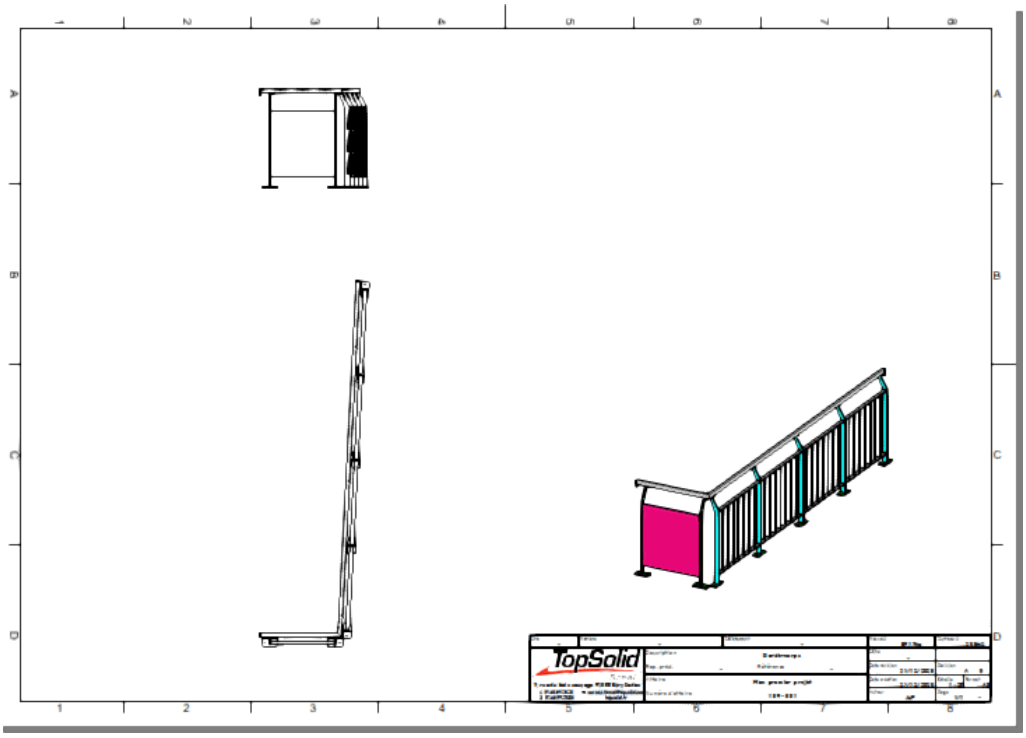
- Rotate the view so that it faces the left side. To do this, click on the sphere as shown below.



- Add the top view, then the isometric view.

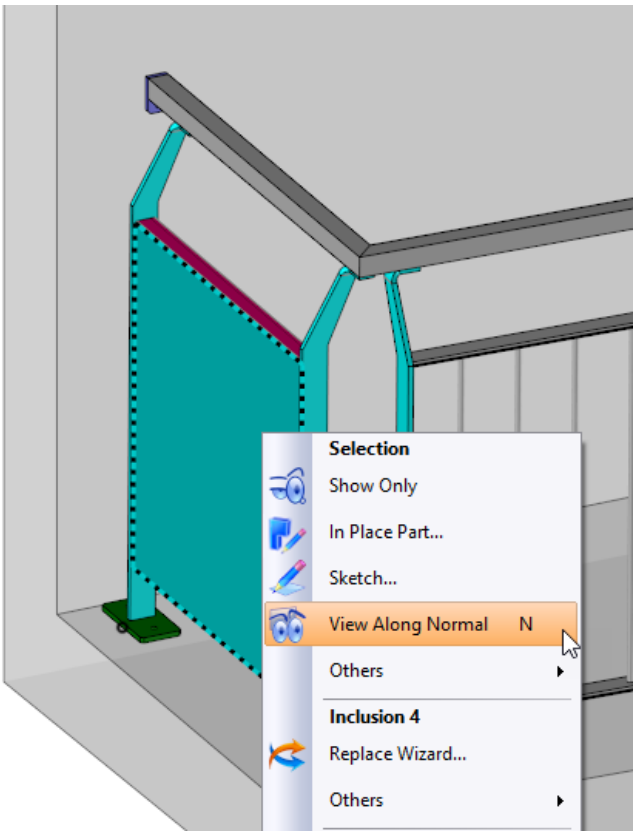


- Reposition the isometric view at the bottom right and switch to shading render mode.

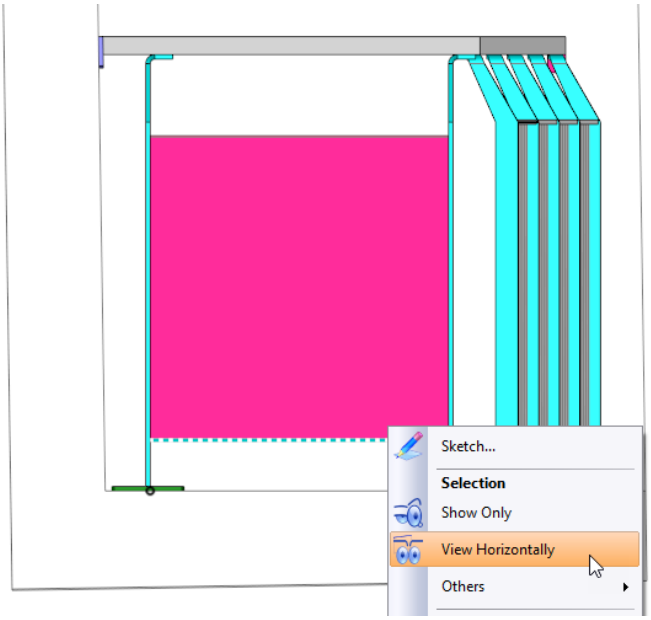


As you can see, the railing has kept its inclination due to the slope. We are going to reorient it correctly in 3D.

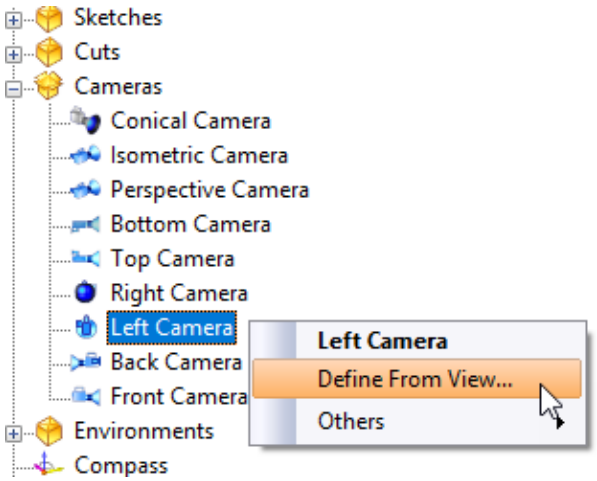
- Return to the *Railing* assembly document.
- Right-click on the sheet metal as shown below and select the **View Along Normal** command.



- Right-click on one of the rails and select the **View Horizontally** command.

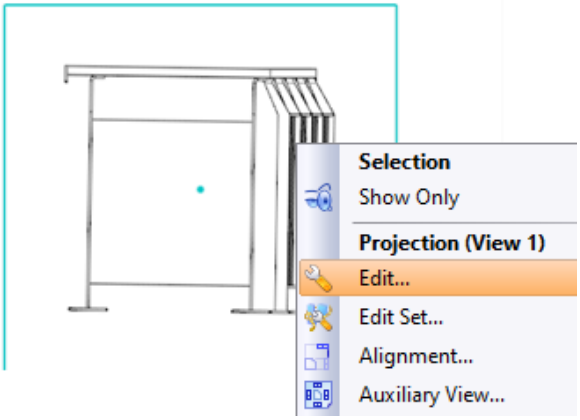


- From the Entities tree, open the **Cameras** folder.
- Right-click on **Left Camera** and select the **Define From View** command.

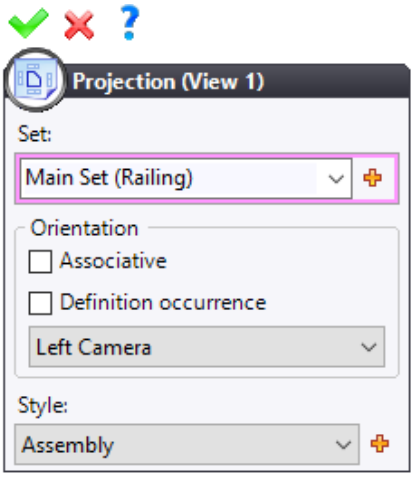


- Click on to **confirm** the dialog box.
- **Save** the assembly document.

- Return to the drafting document and **edit** the first view.



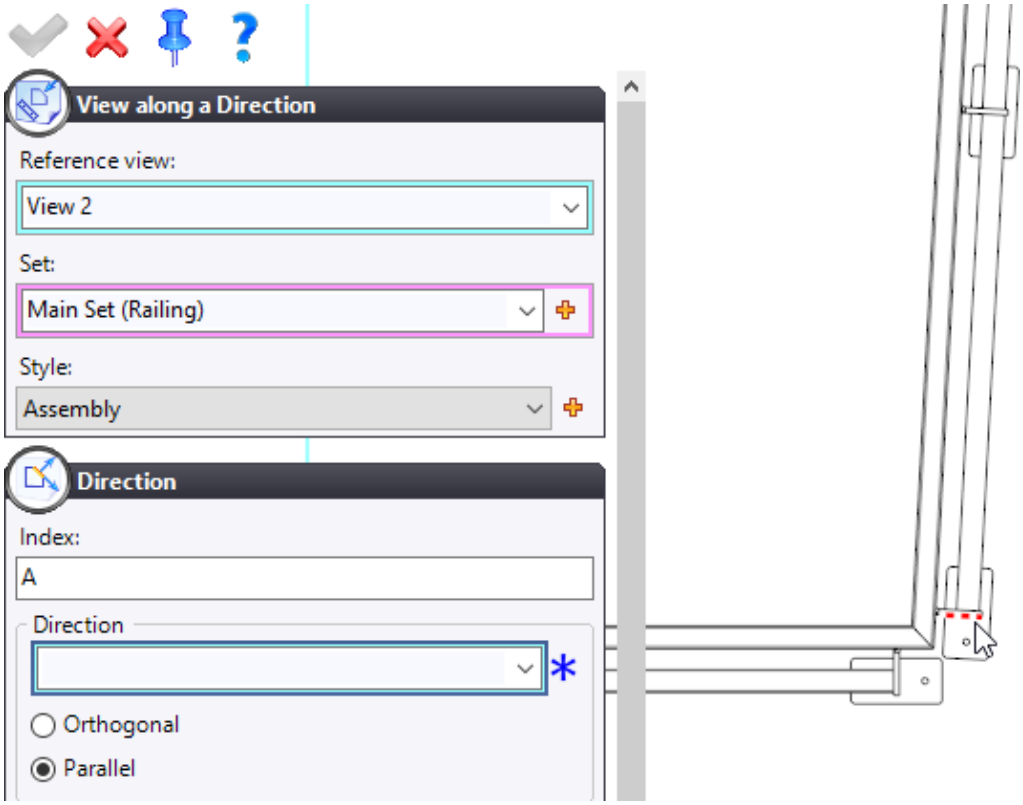
- Select **Left camera** from the drop-down list.



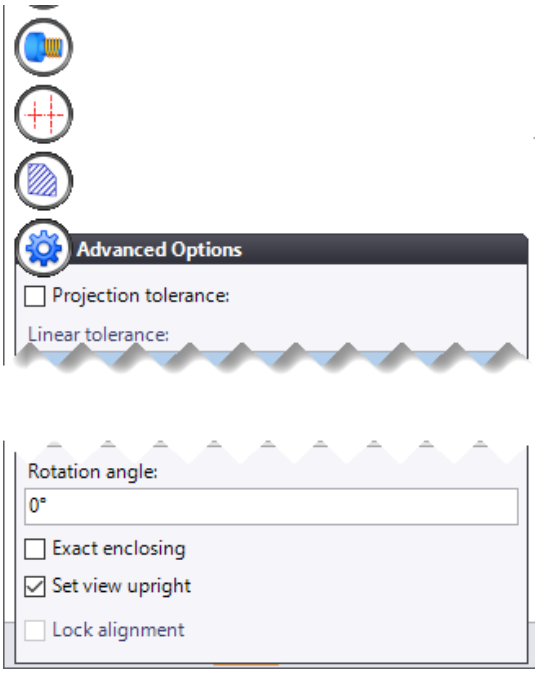
- Click on to **confirm** the operation. All of the views reposition themselves.

Warning: If the angle of the balcony changes, you will have to repeat this action because the angle of the railing will also change.

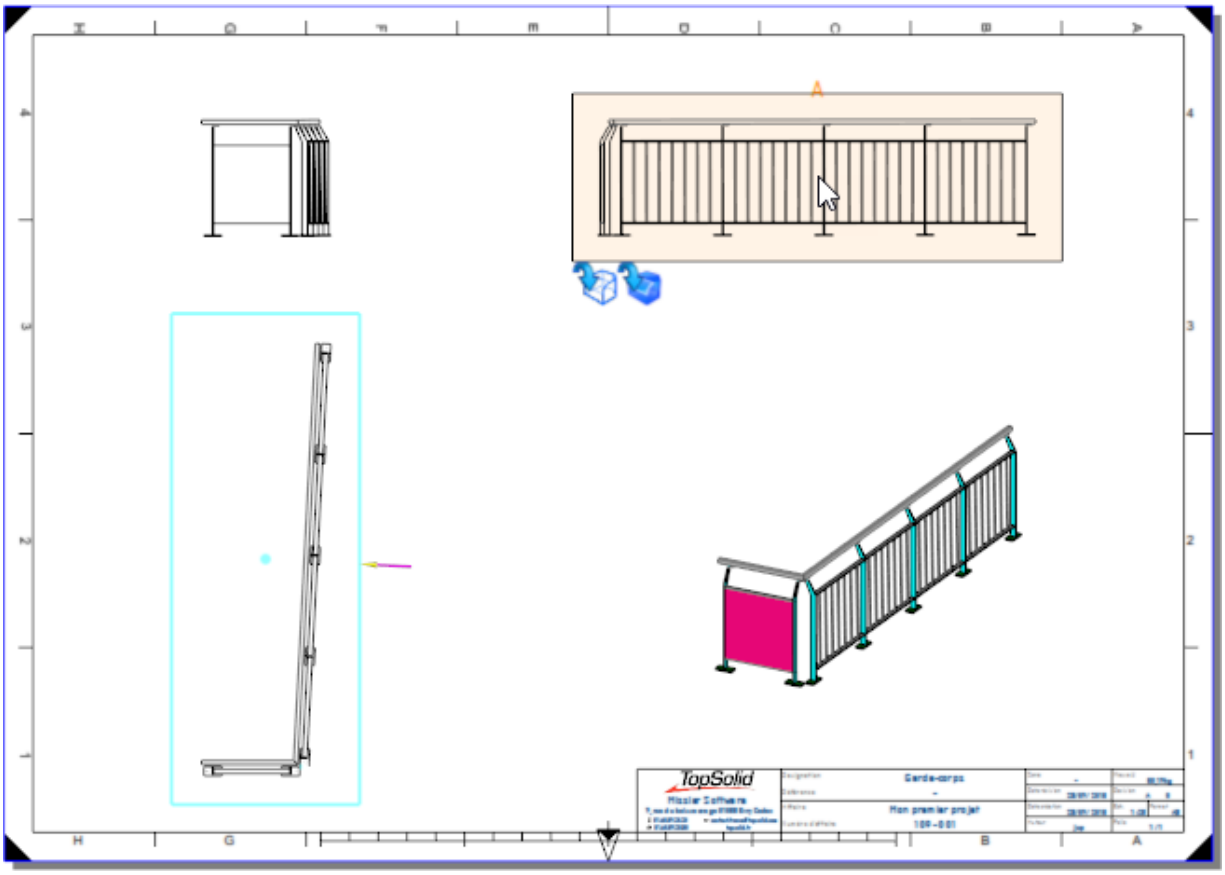
- From the **View** tab, select the **View along a Direction** command.
- Select the top view as the **reference view** and select an edge of the plate or post as the **direction**.
- Select the **Parallel** option.
- Click on the icon to move to the next step.



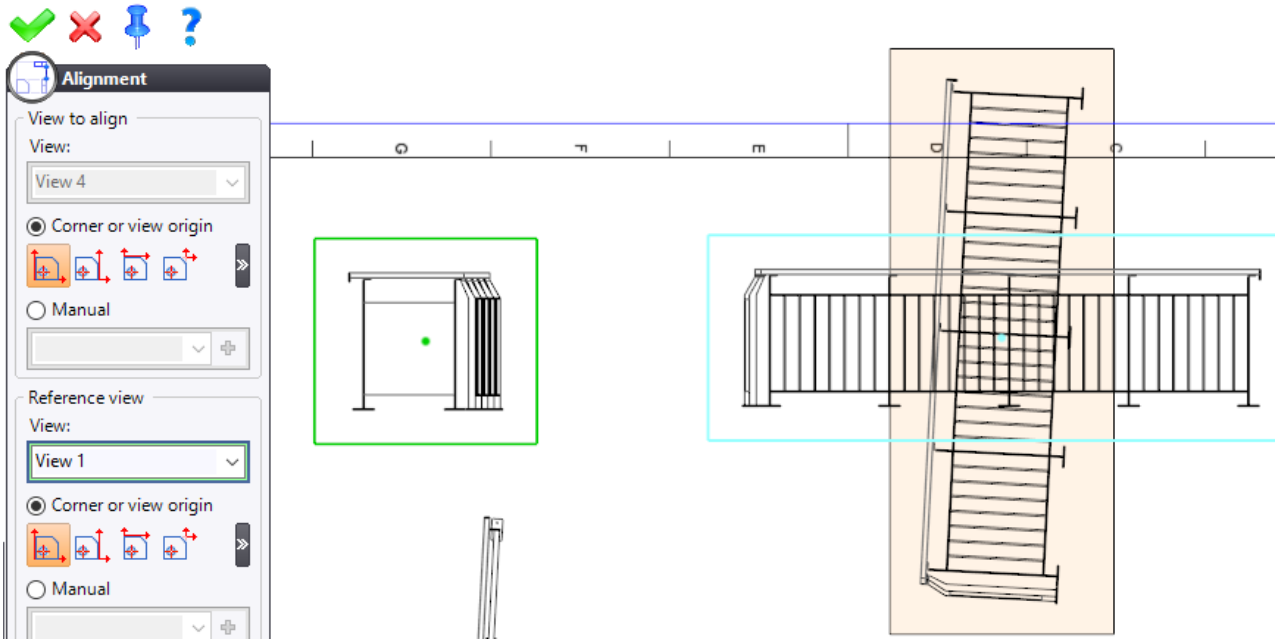
- In the **Advanced Options** dialog box, check the **Set view upright** box.



- Position the view as shown below. Click on the red padlock under the view if necessary. If the view is not straight while positioning, it will be set upright when the command is confirmed.

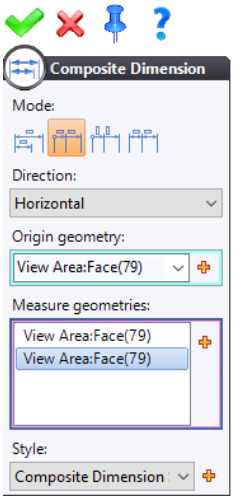


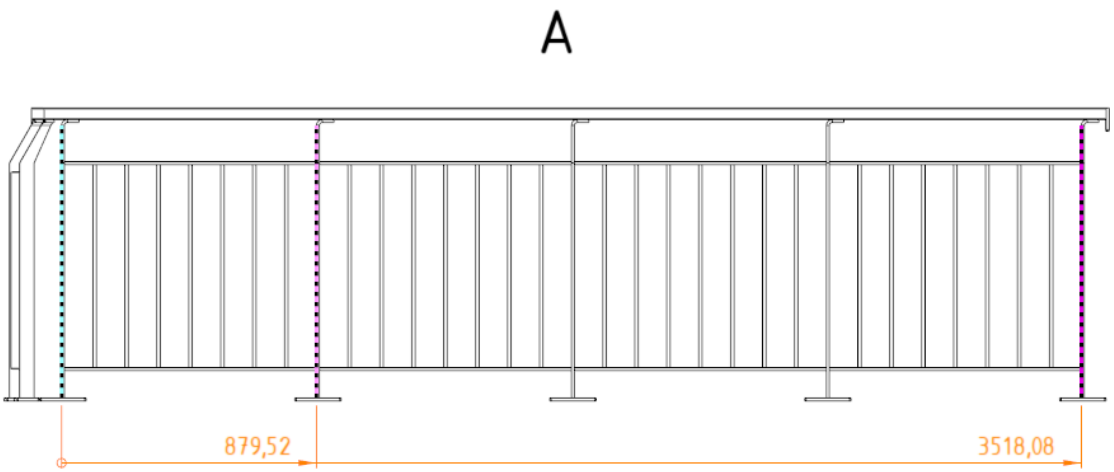
- Click on to **confirm** the operation.
- Right-click on the previously created isometric view and select the **Alignment** command.
- In the **Reference View** section, select the first positioned view (left camera).



- Click on to **confirm** the operation.

- From the **Detailing** tab, select the **Composite Dimension** command. Select the **Cumulative** mode and the **Horizontal** direction. Select the first post on the left as the **origin geometry** and select the second and last posts as the **measure geometries**.

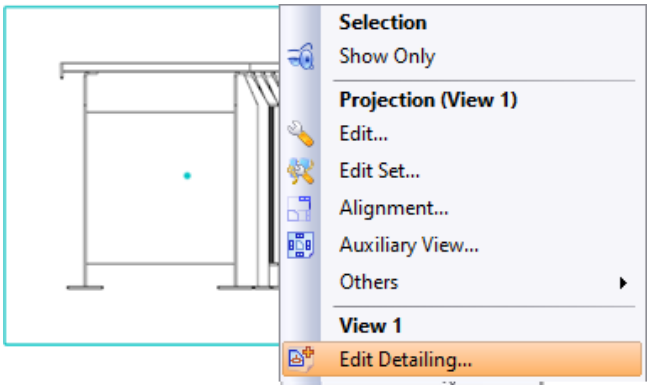




- Click on to **confirm** the operation.

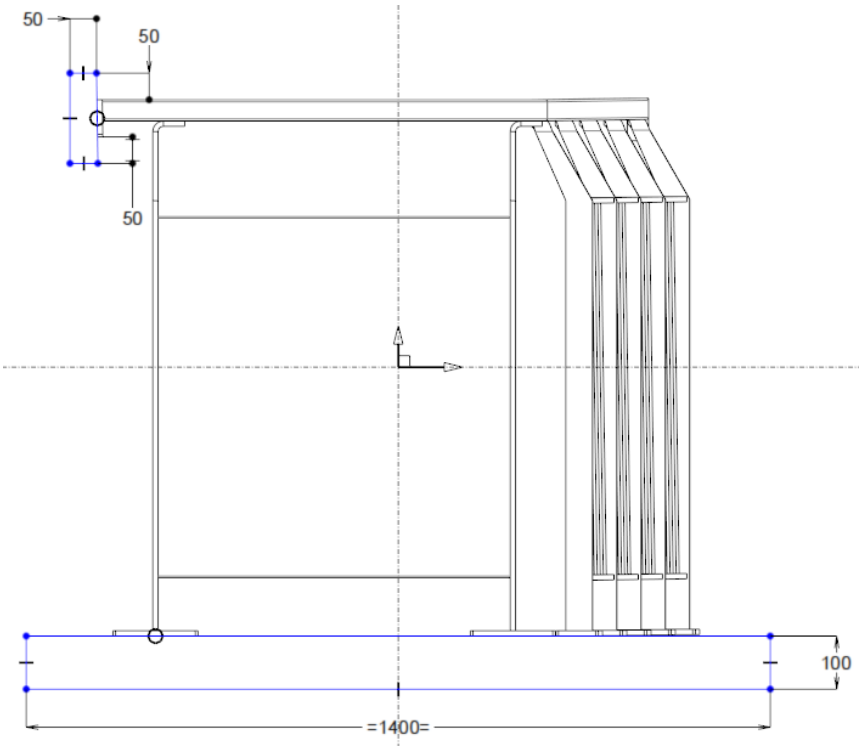
You will notice that the railing is alone. This allows you to set the dimensions of the work without loading the drawing with the environment. However, it is possible to create 2D outlines to facilitate the understanding of the set.

- Right-click on the view as shown below and select the **Edit Detailing** command.

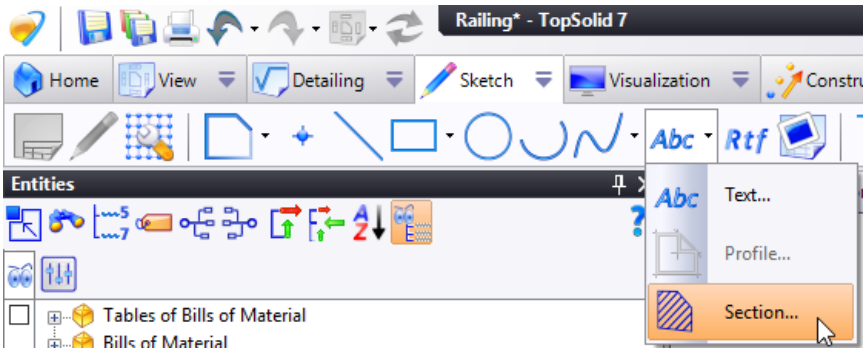


TopSolid switches to the sketch context directly at the view's scale.

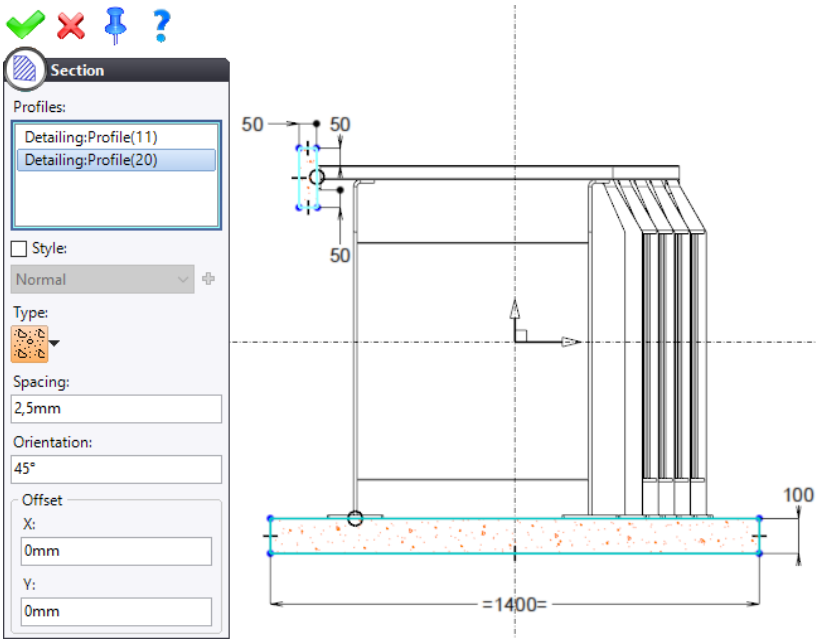
- Draw the following sketch.



- From the **Sketch** tab, select the **Section** command.



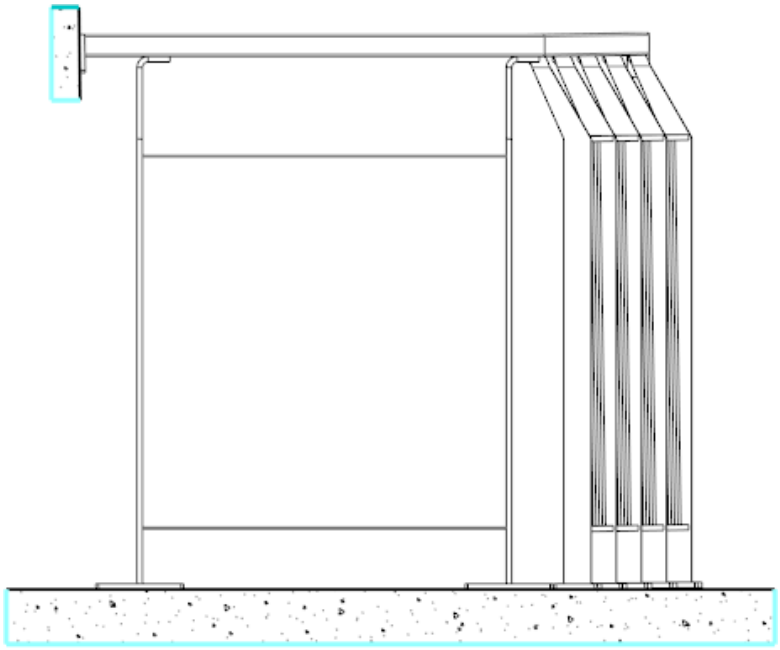
- Select the two previously created rectangles and select the **Concrete** type.



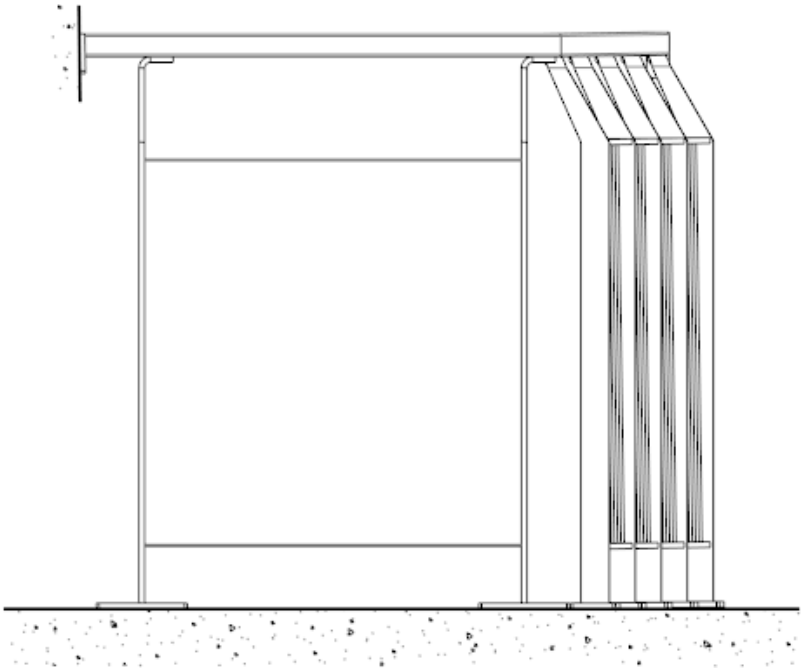
- **Confirm** the sketch by double-clicking with the right mouse button in the graphics area without active selection.

To make the result more aesthetic, you can hide excess lines.

- Right-click on the view and select the **Geometries Attributes** command.
- Select the lines as shown below and select the **Hidden line** category.

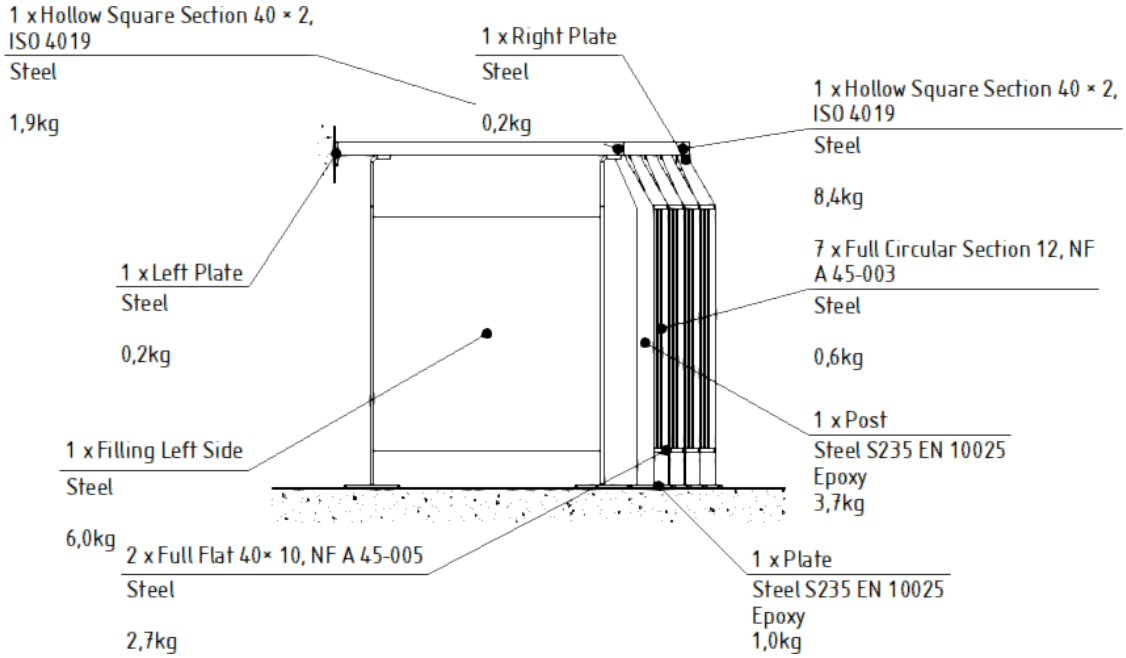


- Click on to **confirm** the operation again.



We saw previously how to add an automatic indexing. You can manually add interesting indexes on different views.

- From the **Detailing** tab, select the  **Bill of materials Index** command.
- Select the **QtyDesc_MatCoatMass** style and add some indexes.



-  **Save** the drafting document.

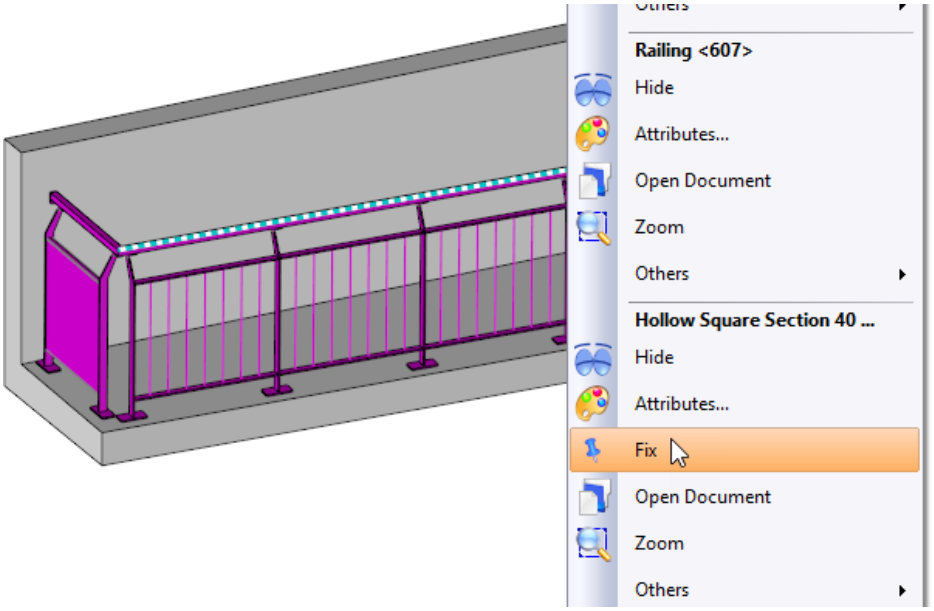
Reassembly of the environment and the railing

We have just created the railing drafting document without its environment. In order to be able to create a layout drafting of both, you have to assemble the railing with its balcony in a new assembly document.

- From the Project tree, right-click on the project name and select the **Assembly** command.
- Right-click on the assembly document you have just created and select the **Properties** command.
- Enter *Railing + Balcony* in the description field.
- Make sure that the *Balcony* and *Railing* documents are open.
- From the **Assembly** tab, select the **Inclusion** command.
- Select the *Balcony* document. Make sure that the **Inclusion at origin** box is checked and then click on to **confirm** the operation.
- Select the **Inclusion** command again.
- Select the *Railing* document. Make sure that the **Inclusion at origin** box is checked and then click on to **confirm** the operation.

The railing is positioned in the right place, but it is still necessary to add positioning constraints.

- Exit the **Constraint** dialog box.
- Still in the positioning context, right-click on the railing and select the **Fix** command.

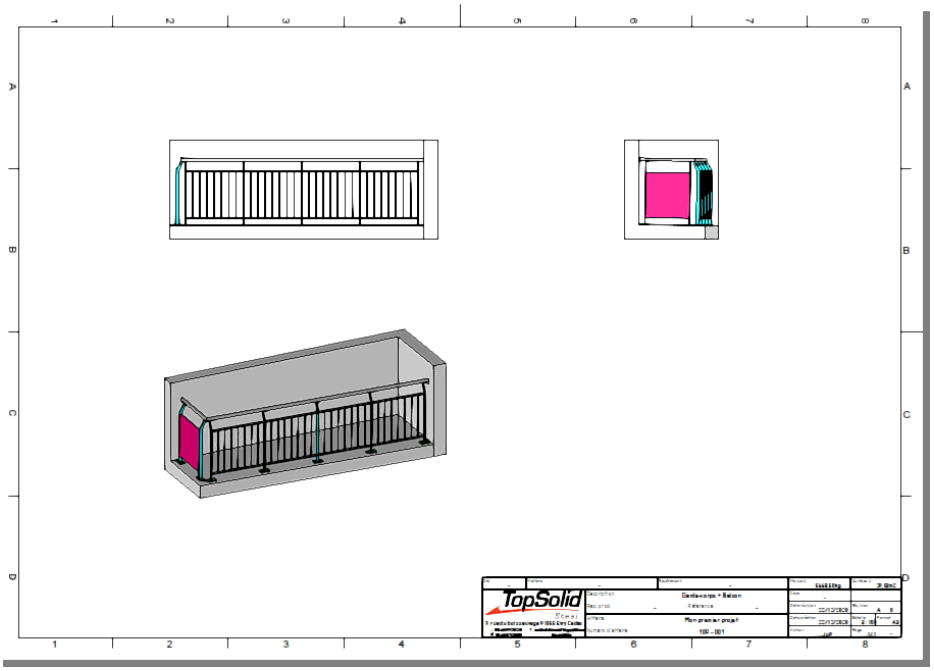


The railing is displayed in gray. The positioning operation is complete.

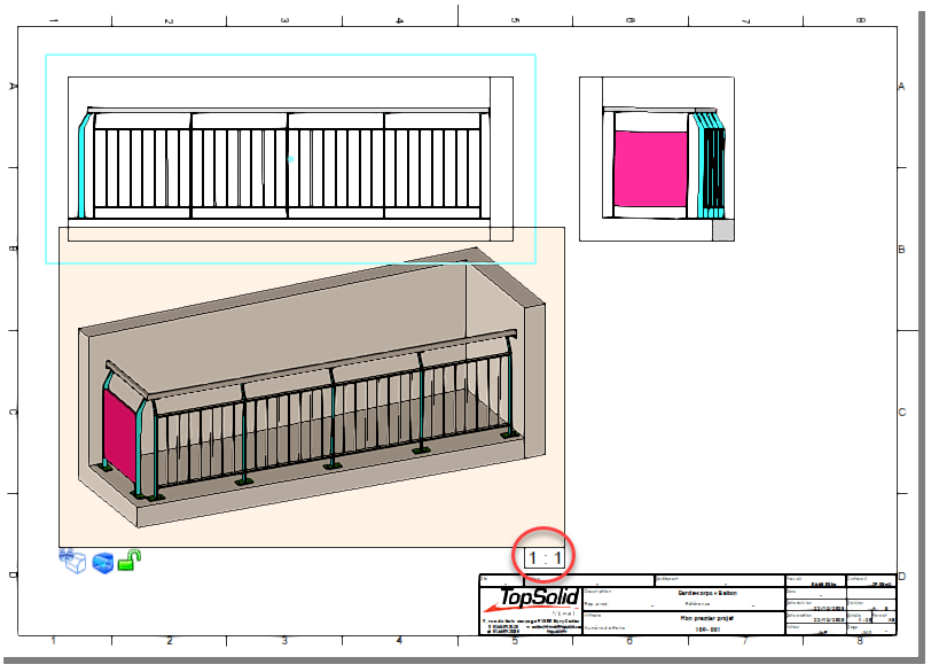
- **Confirm** the positioning context.
- **Save** the *Railing + Balcony* assembly document.

Complete stage drafting

- Right-click on the *Railing + Balcony* assembly document's tab and select the **Drafting** command.
- Select the **Assembly A3 ISO Landscape** template from the **Steel Standard Templates - United States** folder.
- Position the views as shown below.



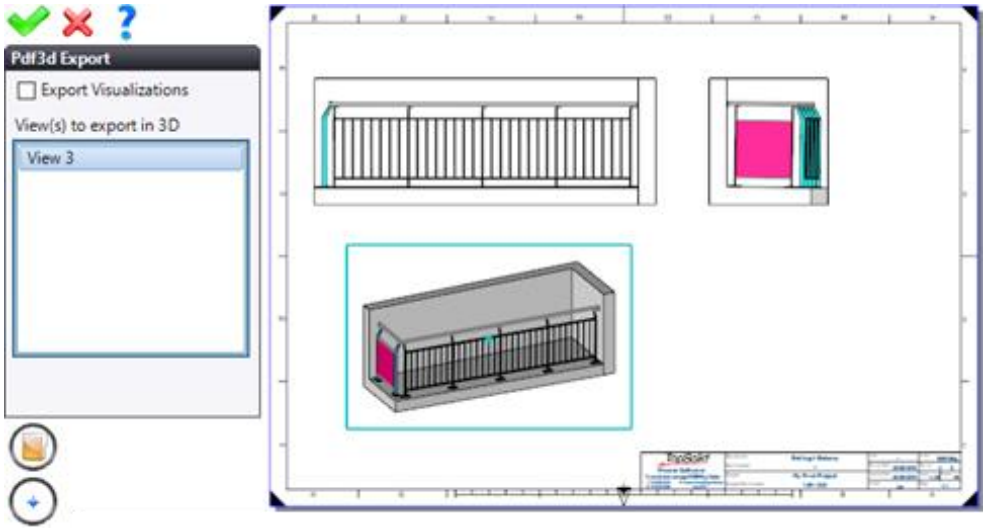
- Right-click in the graphics area and select the **Scale** command.
- Enter *1/20* and click on to **confirm** the operation.
- Right-click on the isometric view and select the **Edit** command. Modify the relative scale of the view by entering a **scale factor of** *0.7* and click on to **confirm** the operation.



Note: This scale factor will be added to that of the drawing.

- **Save** the drafting document.

- From the Project tree, right-click on the *Railing + Balcony* drafting document and select the **Import/Export** > **Export Document with Conversion** command.
- Select the desktop as the destination and select the **Acrobat PDF3D (*.pdf)** format.
- Click on **OK** if **TopSolid** asks you to calculate the projection.
- Select the view as shown below.



- Click on to **confirm** the operation.

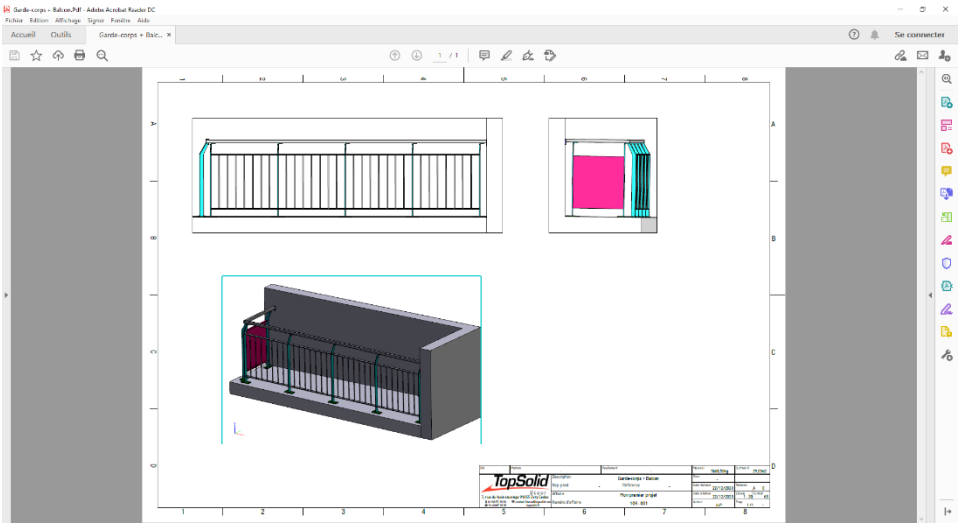
Note: When exporting PDF, you can adjust the line thicknesses directly via the **Tools** > **Options** > **Translators** > **Pdf 3D** command.

- From the desktop, open the generated PDF document.

Adobe normally displays a message indicating that the 3D content is disabled.

- Select the **Edit** > **Preferences** > **3D and Multimedia** command.
- Check the **Enable playing of 3D content** box, then click on **OK** to confirm.

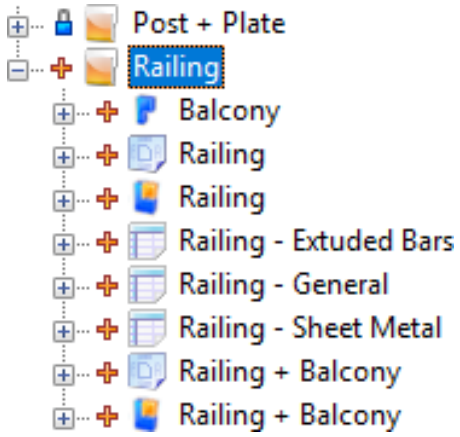
You can now modify the isometric view.



or hold down	Zoom
hold down	Rotation
hold down + hold down	Dragging

Note: You can export an assembly or a part directly as a 3D PDF without using a drafting document. To do this, you simply have to right-click on the part or assembly and select the **Import/Export** > **Export Document with Conversion** command.

- In the Project tree, create a new folder *Railing* and drag all of the created documents into it.



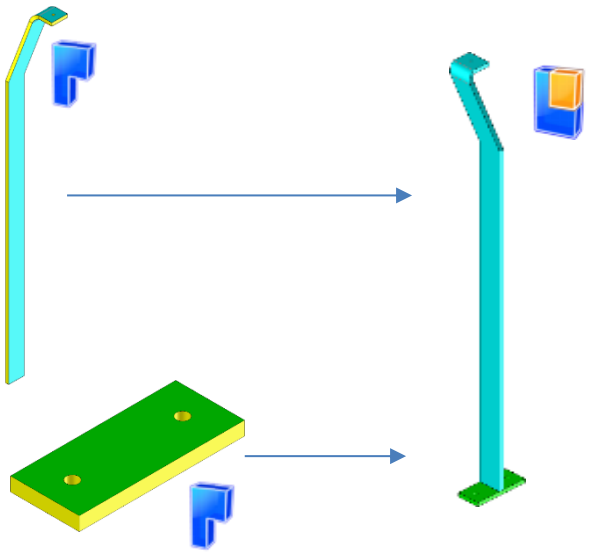
- Right-click on the *Railing* folder and select the **Check In** command.
- Click on to **confirm** the check-in operation.

Conclusion

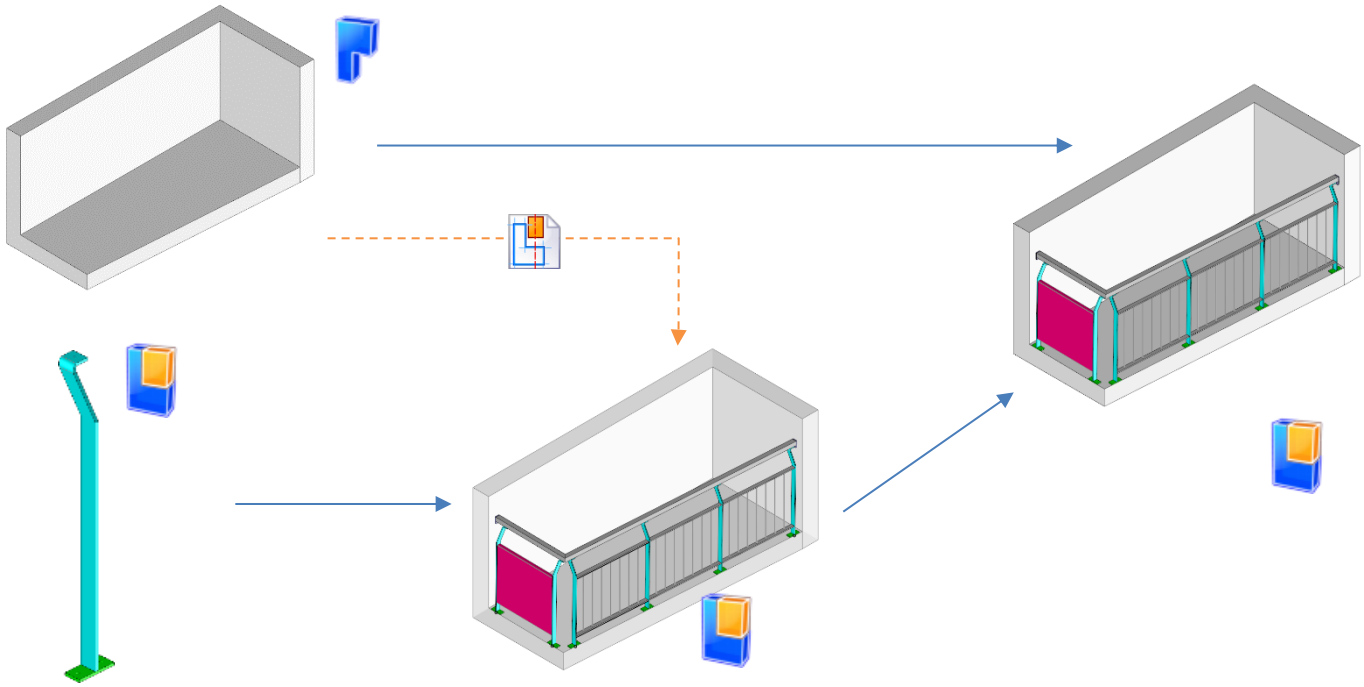
We have just used two very different design methods, each with its advantages and disadvantages.

Reminder:

- Reassembly design (bottom-up):



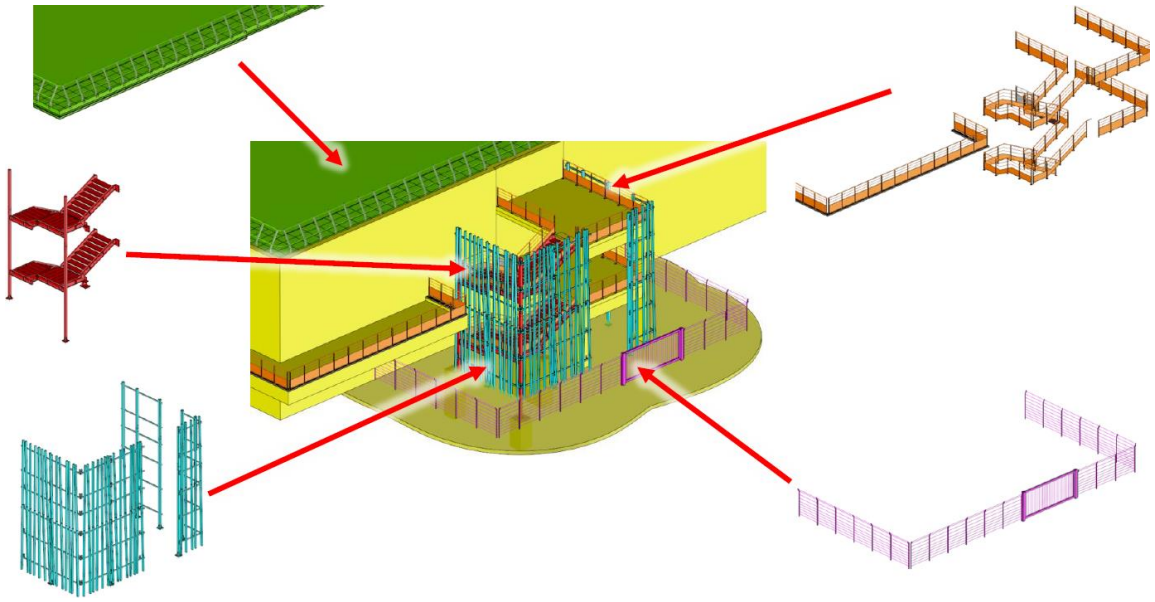
- In-place design (top-down)



Overall, the reassembly design is useful to create the company's standards or to split a design into several independent or weakly linked assemblies.

This cutting can be geographical or functional.

Here is a sample splitting:

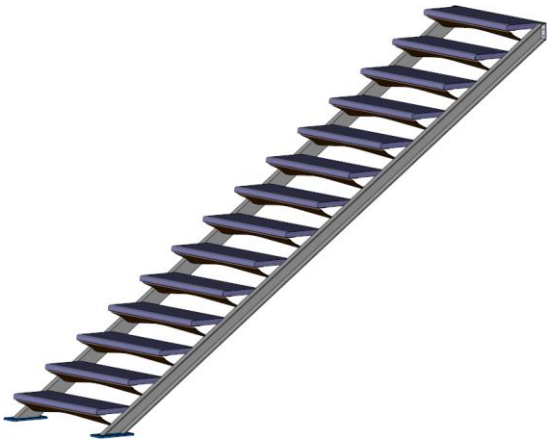


Each assembly represents an existing in place design that was then integrated into an overall assembly. All areas can be worked on independently while keeping a link with civil engineering: their background document. A modification of the latter will result in the updating of each of the sub-assemblies.

Part 3 - Introduction to the Parameterization with the Stair

We have seen that **TopSolid** integrates a certain number of automatisms with regard to repetition patterns (railing posts, bars, etc.). In addition, the parameterization can be used to manage more complex cases according to your design rules.

The following exercise serves as an introduction to the parameterization which can be supplemented with other training courses.



Concepts addressed:

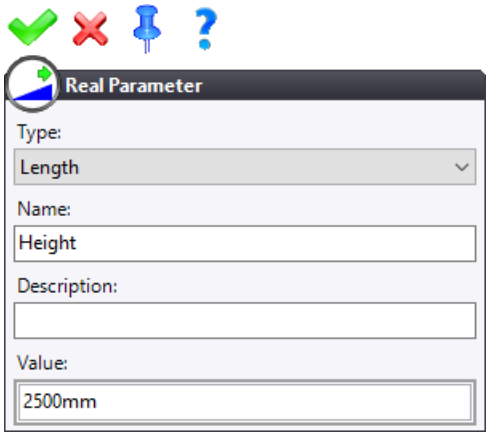
- Creating the parameters
- Creating relationships between parameters
- Using parameters to design
- Integrating standard components (screws)
- Understanding the concepts of process and assistant

Creating the basic parameters

- From the Project tree, right-click on the project name and select the **Assembly** command.
- Rename the assembly document *Right stair*.
- From the Entities tree, right-click on the **Parameters** folder and select the **Real Parameter** command.

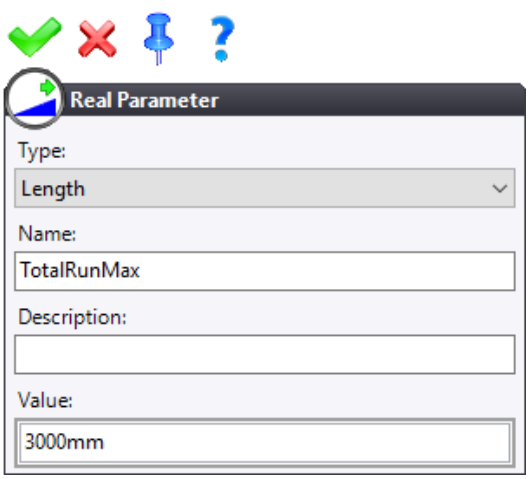
The real parameters are the most common parameters. They can be of any type (length, angle, without unit, surface, etc.) and can integrate formulas themselves. For more experienced users, it is possible to integrate VB.net or C# scripts.

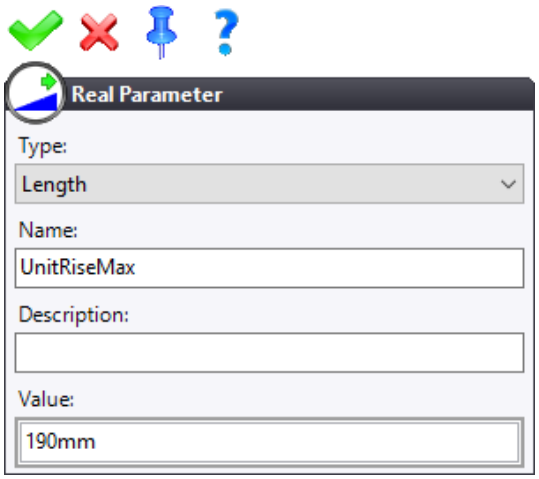
- Select **Length** as the type, enter *Height* as the **name** and a **value** of *2500mm*.



- **Pin** the command's dialog box and click on to **confirm** the operation.

- Create the *TotalRunMax* and *UnitRiseMax* parameters as shown below.





We will now move on to the calculation section.

- Create a new **real parameter**.
- Select the **Factor** type. This is a type without unit, ideal for entering quantities.

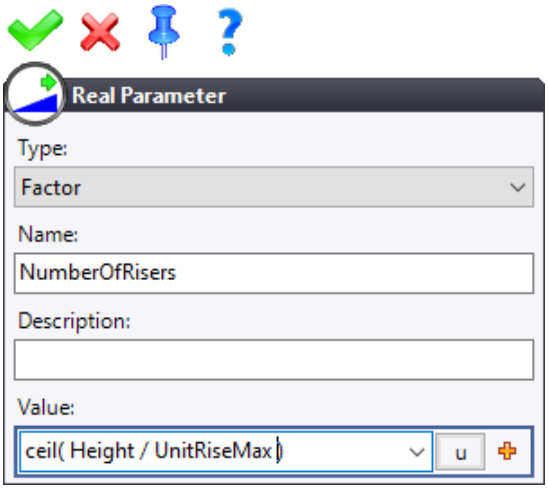
Note: Entering the *Height/UnitRiseMax* value is one solution but not the best because the result will contain decimals. **TopSolid** will round off to the nearest number when we do the repetitions.

TopSolid is capable of managing three types of rounding:

Name	Operation	Formula	Result
	Without rounding	Height/UnitRiseMax	13.16
Floor	Less than	Floor (Height/UnitRiseMax)	13
Round	Nearest	Round (Height/UnitRiseMax)	13
Ceil	More than	Ceil (Height/UnitRiseMax)	14

In this case, we will use the **Ceil** formula since the rise height is a maximum value.

- Enter the following values.

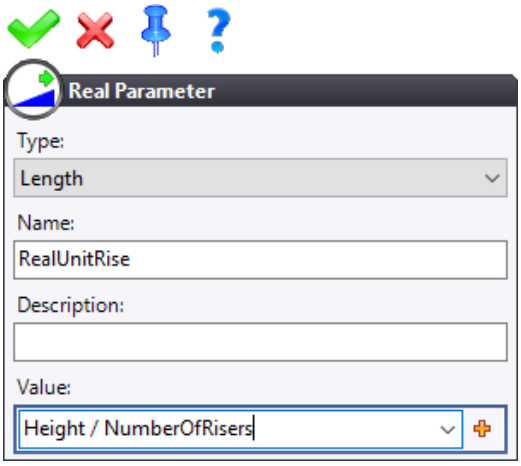


When you enter the name of a parameter, **TopSolid** offers the list of parameters already created. You can navigate through this list using the mouse or the up and down keys on the keyboard. Once the parameter is selected, you simply have to press the **Tab** key to enter the name of the parameter.

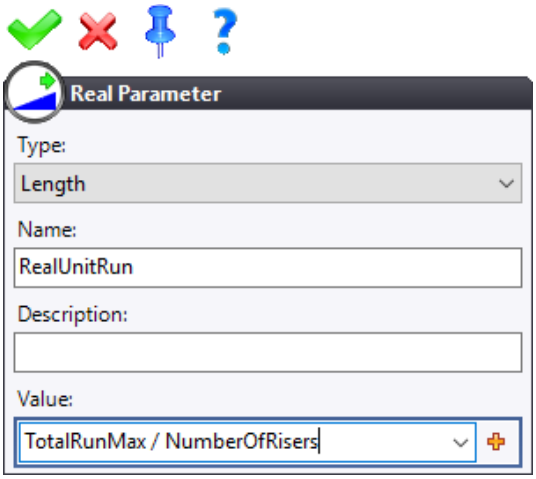
- Click on to **confirm** the operation.

Note: You can resize the dialog box by dragging the right edge.

- Create a new *RealUnitRise* parameter using the **Length** type.



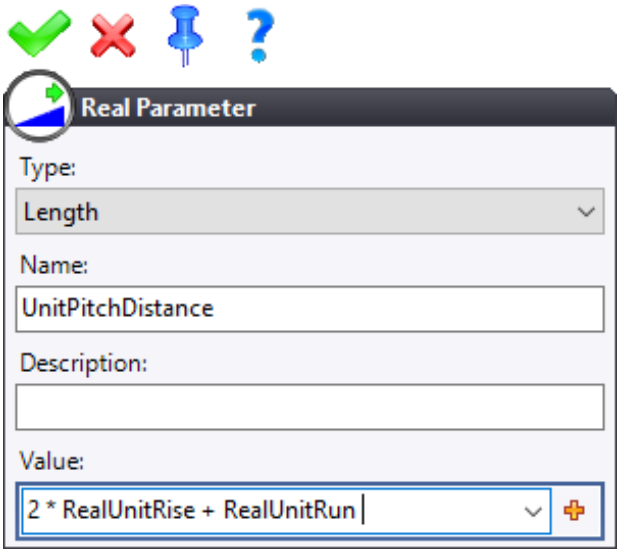
- Create a new *RealUnitRun* parameter using the **Length** type.



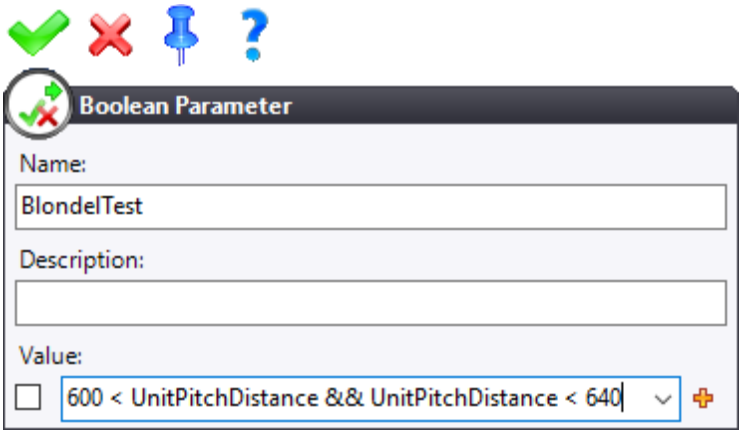
The basic parameterization is complete by now. This is not the ultimate rule for calculating a right stair, it is just a rule. The advantage is that you are able to configure according to your know-how, as well as your work habits and work constraints.

We will add verification parameters, but this step is not mandatory.

- Create the *UnitPitchDistance* parameter using the **Length** type.



- From the Entities tree, right-click on the **Parameters** folder and select the **Boolean Parameter** command.
- Enter the name parameter *BlondelTest* and the following formula.



Note: The following symbols can be used:

- &&: AND
- ||: OR

- Click on to **confirm** the operation.

The value is wrong, which means that the stair does not comply with Blondel's formula.

- Edit the *UnitRiseMax* parameter, enter a value of *200mm* and then click on to **confirm** the operation.

All the parameters are recalculated and the unit pitch distance is now correct.

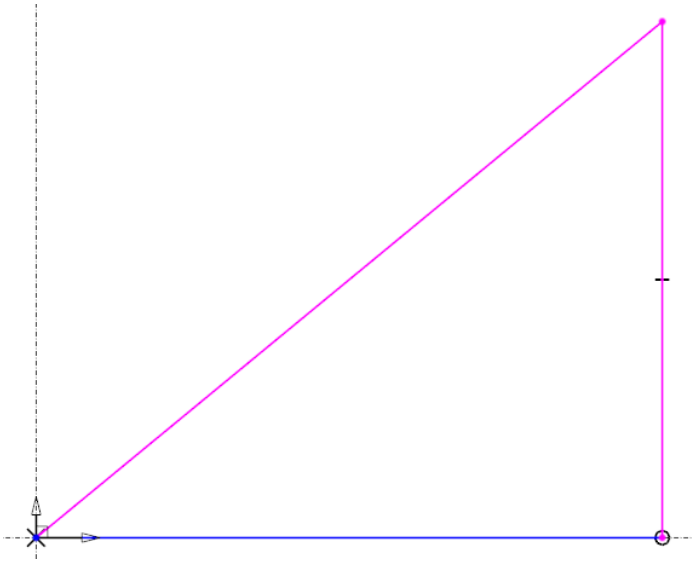
Summary of common parameters in TopSolid

Icon	Description	Default unit	Use
	Real Parameter	M², °, mm, u, etc.	The most common parameter that allows you to define and calculate with a large number of value types (area, angle, length, factor, etc.).
	Angle Parameter	°	Measures the angle between two directions. This angle can be used in a command or a formula.
	Distance Parameter	mm	Measures the distance between two entities. This distance can be used in a command or a formula.
	Length Parameter	mm	Measures the length of a profile. This profile can be a sketch or directly an edge or a set of edges on a shape.
	Boolean Parameter	None	Parameter whose value is **True** or **False**. Ideal for asking a question with a "Yes/No" answer, for managing options and variants ("Last rise?", "Bars?"). Can be used directly to enable/disable operations via conditions.
	Integer Parameter	None	Parameter without decimal point or unit. This parameter can be used to define quantities, for example.
Abc	**Text Parameter**	None	As the name suggests, this parameter contains text. This text can be used in properties or formulas (text concatenations, etc.).

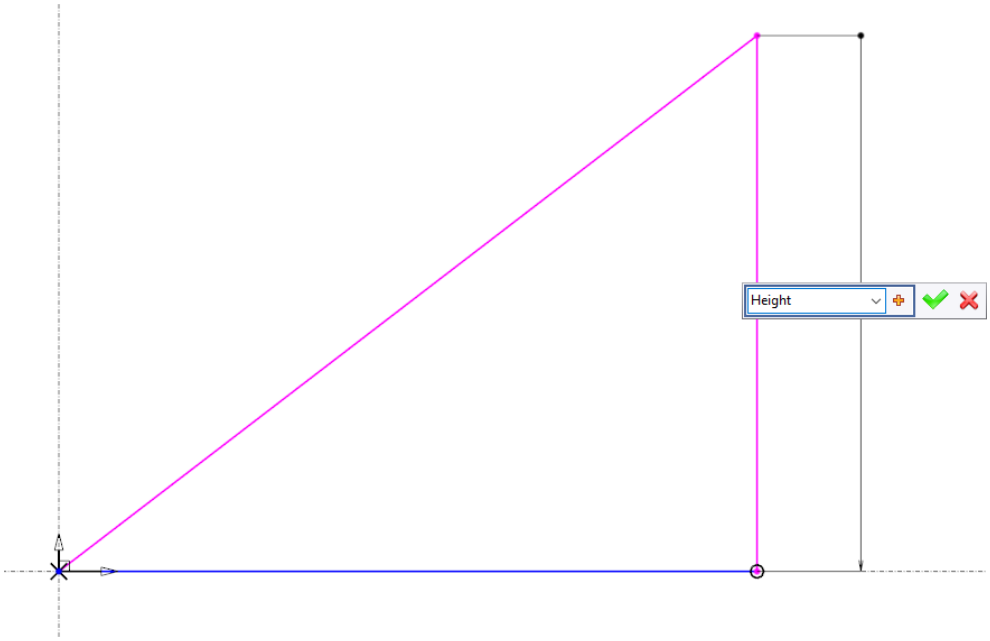
Creating the layout

We will now draw the skeleton of the stair.

- Right-click in the graphics area and select the **Sketch** command.
- Draw the following rectangle.

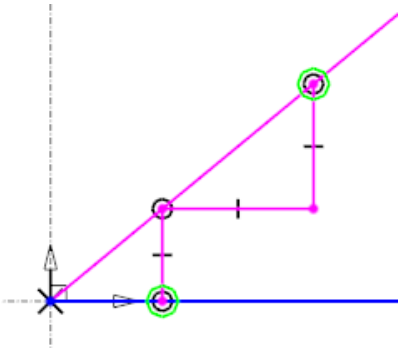


- From the **2D Sketch** tab, select the **Position Sketch** command and select the **absolute XZ plane** as the **support plane**. This will allow the Z axis to be vertical.
- Add the following dimension using the **Height** parameter as the value.



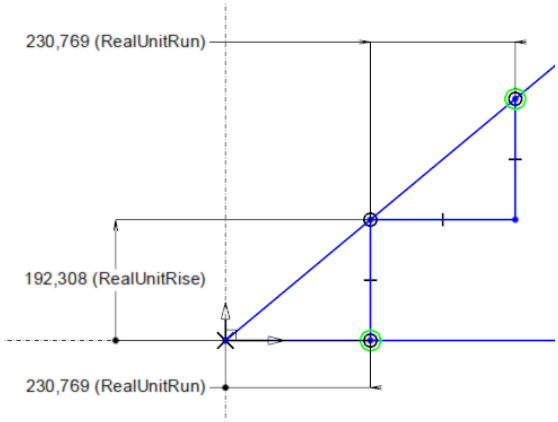
TopSolid automatically retrieves the value of the parameter.

- Select the **Contour** command and draw the segments as shown below.



These segments represent the first step as well as the nose of the second step.

- Add the following dimensions using the previously created parameters.



The sketch should appear in blue to indicate that it is fully constrained.

- **Confirm** the sketch.

Creating the step

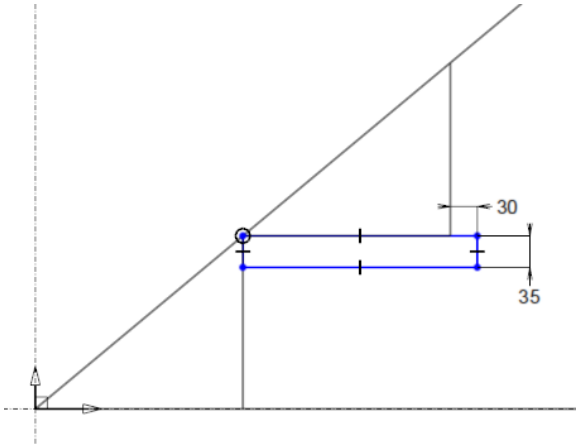
The step will consist of two parts:

- a wooden step;
- a bent sheet metal step support.

- Right-click on the layout sketch and select the **In Place Part** command.

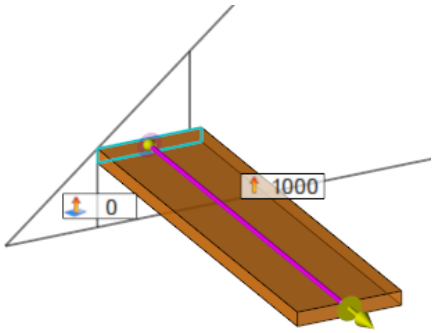
Note: Using the sketch allows **TopSolid** to orient itself in the same way, thus avoiding the reorientation step.

- Draw the sketch shown below using the **Rectangle** command.

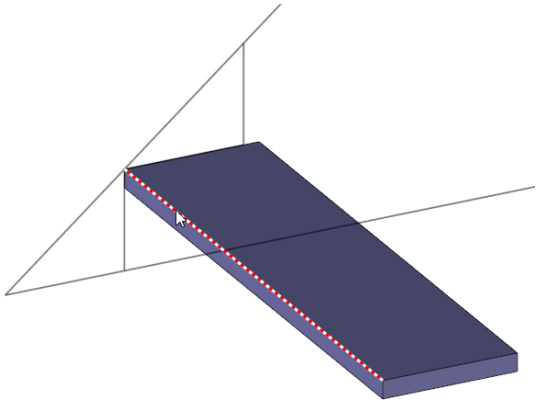


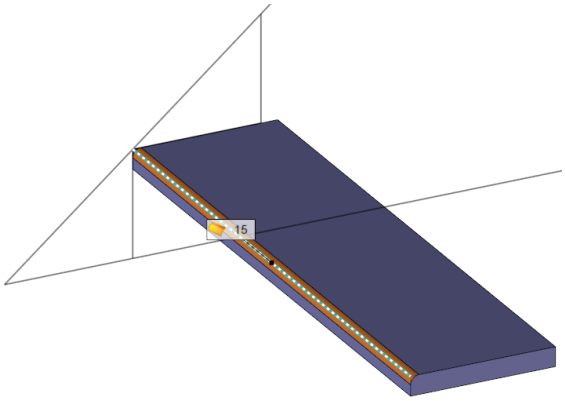
It is important to set the dimensions against the second step. Indeed, if the unit run changes, the step will be recalculated automatically.

- Right-click in the graphics area with no active selection and select the **Extruded** command. Enter a **length** of *1000mm* and click on to **confirm** the operation.



- Right-click on the edge as shown below and select the **Fillet** command. Enter a **value** of *15mm* and click on to **confirm** the operation.

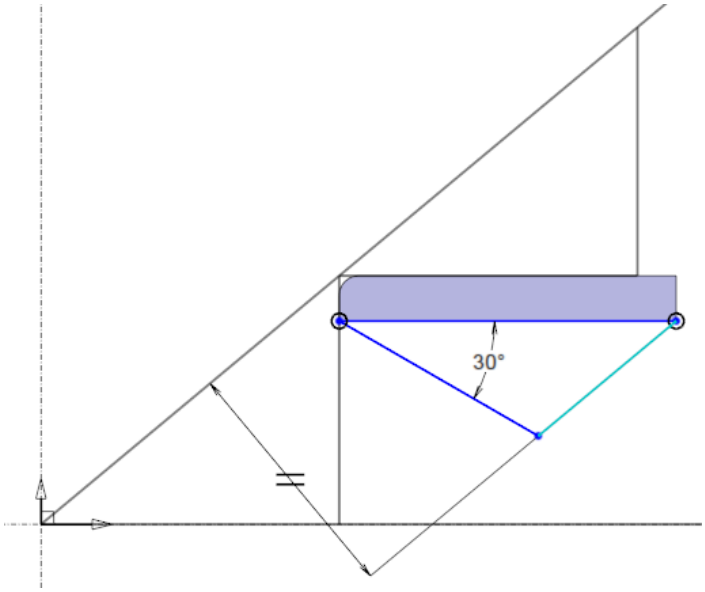




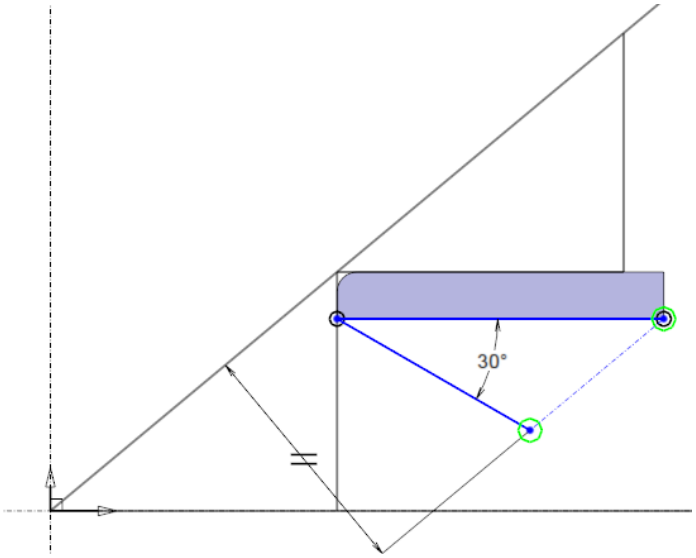
- **Confirm** the in-place editing context.

We will now create the support.

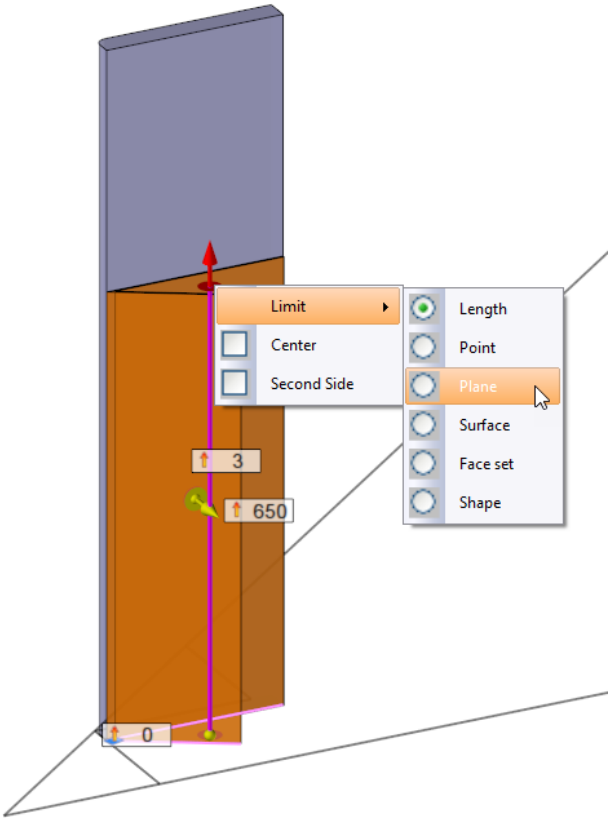
- Right-click on the layout sketch and select the **In Place Part** command.
- Draw the following sketch.
- Add a **parallelism** constraint between the treading and the highlighted segment as shown below.



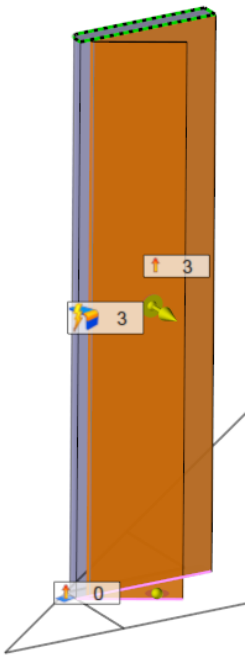
- Right-click on the same segment and select the **Construction** command. This command is used to indicate to **TopSolid** that this element is simply used to facilitate the construction. It will therefore not be taken into account when switching to 3D.



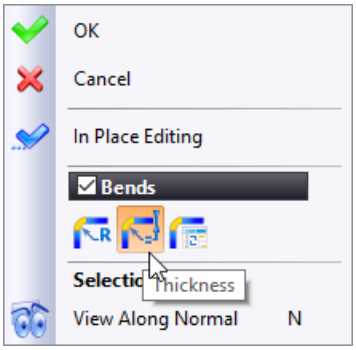
- Right-click in the graphics area with no active selection and select the **Sheet Metal on Sketch** command.
- Enter a **thickness** of *3mm.*
- Right-click on the arrow in the graphics area and select **Limit** > **Plane**.



- Select the side face of the step as the **plane**.



- Right-click in the graphics area with no active selection. Make sure the **Bends** box is checked and select the **Thickness** mode so that the bend is equal to the thickness of the sheet metal.



- Click on to **confirm** the operation.
- **Confirm** the in-place editing context.
- **Save** the assembly document.

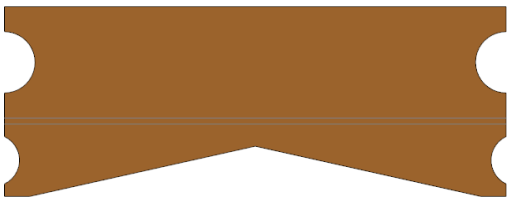
Although the step support has been created in place, it has its own document, allowing you to work on it in isolation.

- Right-click on the step support and select the **Open Document** command.

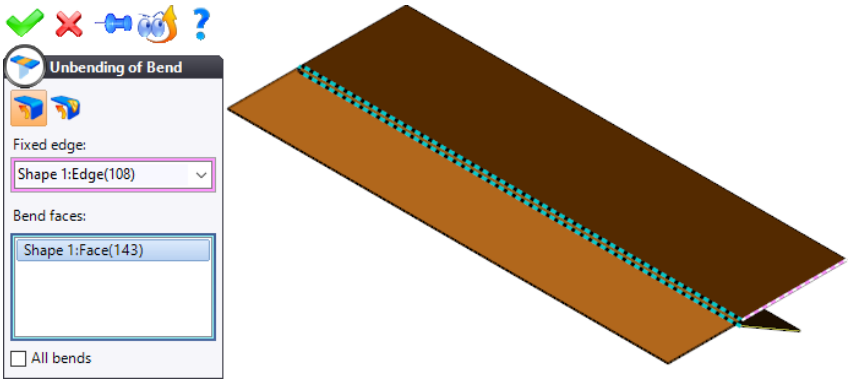
TopSolid opens the part in a new tab.

- Refocus the camera on the step by double-clicking on the mouse wheel.

It is often easier to create the bent parts directly. It is sometimes necessary to unfold them to add the operations that occur before bending, such as the cutting operations. This is what we are going to create here by producing the cutting as shown below before bending.



- From the **Sheet Metal** tab, select the **Unbending of Bend** command.
- Select an edge of the horizontal section as the **fixed edge** and select the bend as the **bend face**.

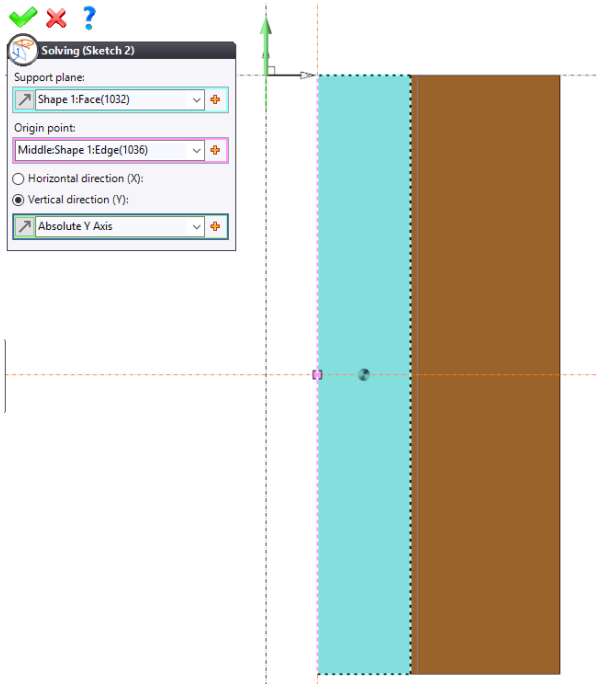


Note: Checking the **All bends** box would have been possible.

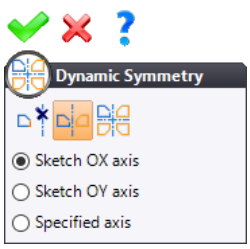
- Click on to **confirm** the operation.

We will now draw the sketch that will be used for the cutting operation. To save time, we will work symmetrically.

- Right-click on the top face as shown below and select the **Sketch** command.
- From the **2D Sketch** tab, select the **Position Sketch** command.
- Select the edge as shown below as the **origin point**.



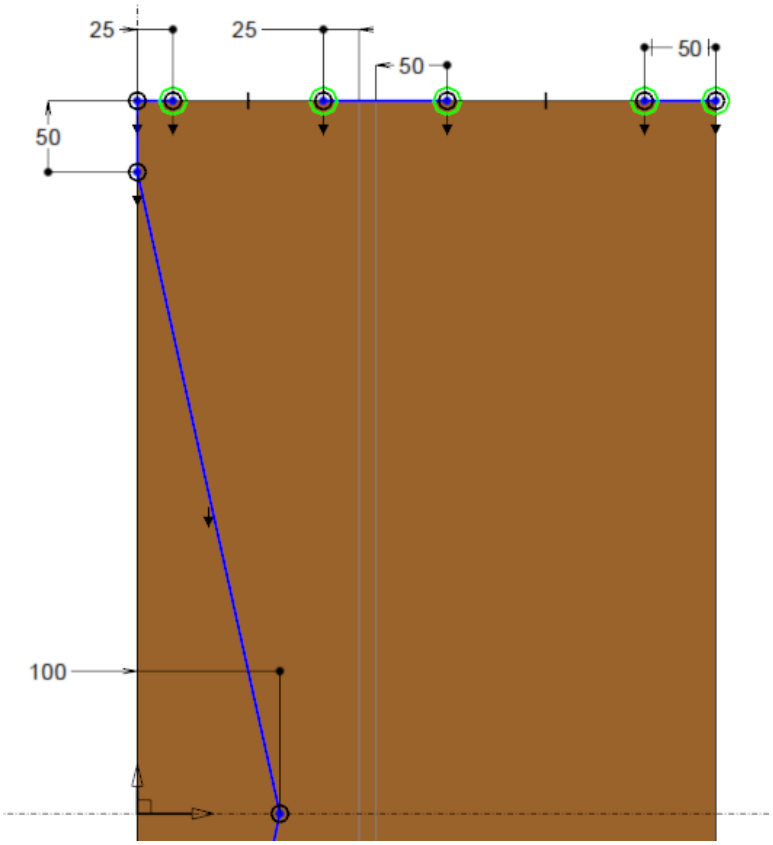
- At the top right of the screen, click on the **Dynamic Symmetry** icon.
- Select the **Simple symmetry** mode and select the **Sketch OX Axis** option.



- Click on to **confirm** the operation.

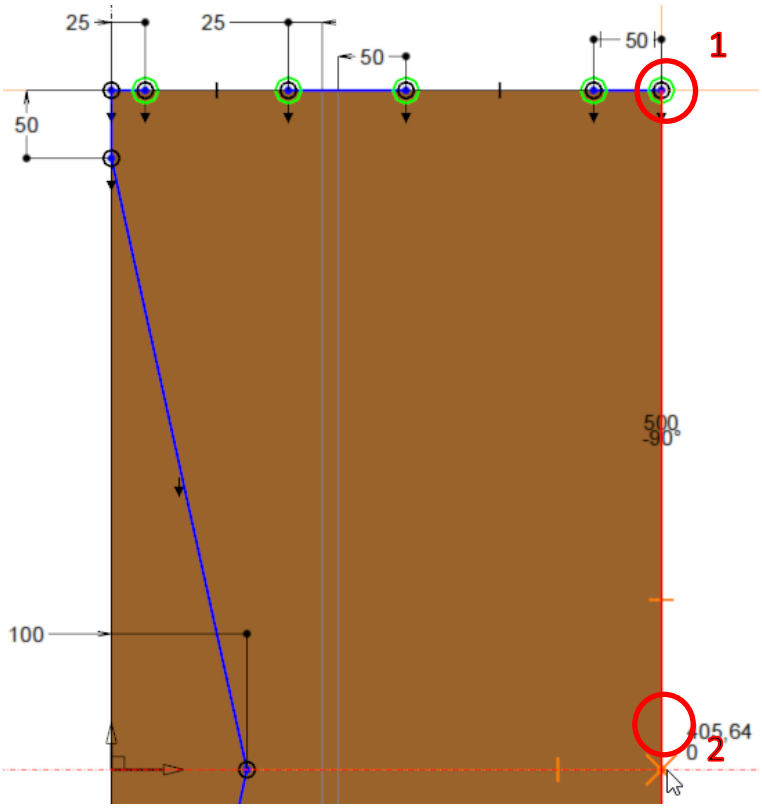
TopSolid will dynamically draw the reverse side.

- Draw the sketch shown below using the **Line** command.

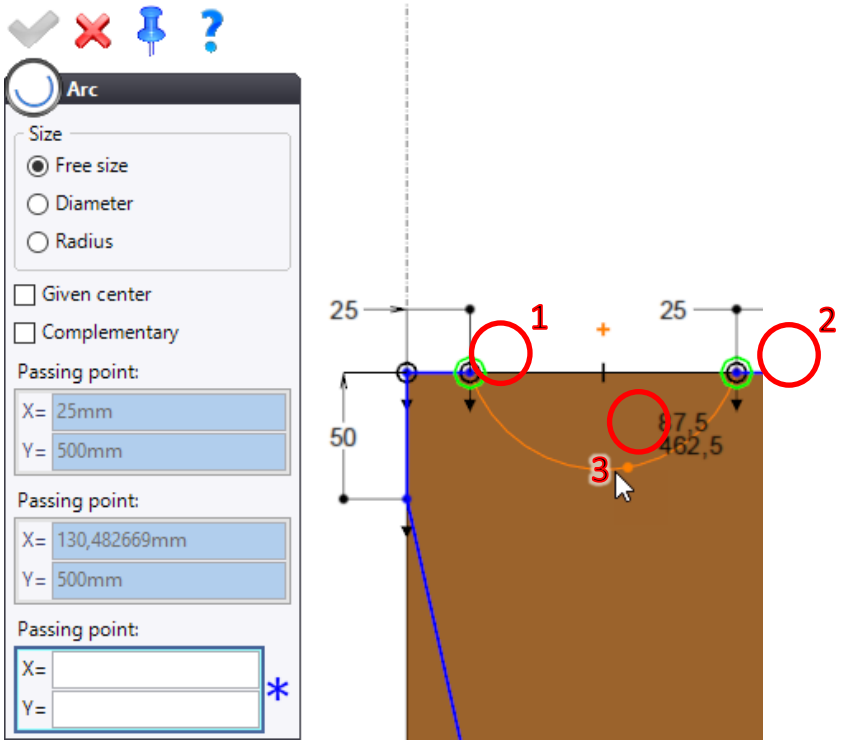


Note: If a line has to pass through the symmetry, you simply have to make it touch the axis of symmetry.

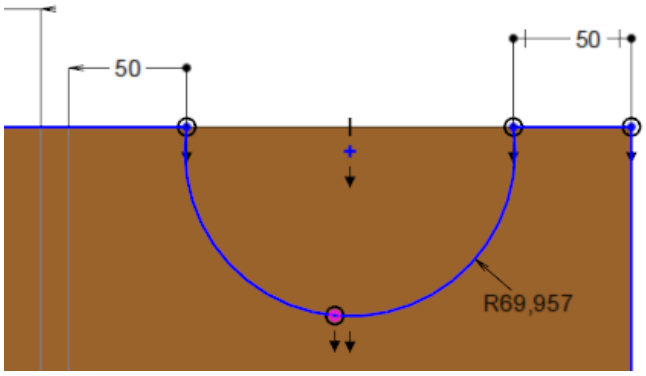
- Draw the line as shown below.



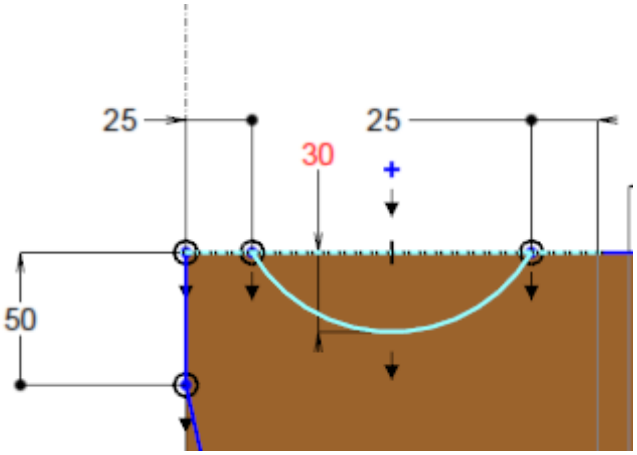
- Select the **Arc** command. Make sure that the **Free size** mode is selected and that the **Given center** box is unchecked.
- Draw the first arc as shown below.



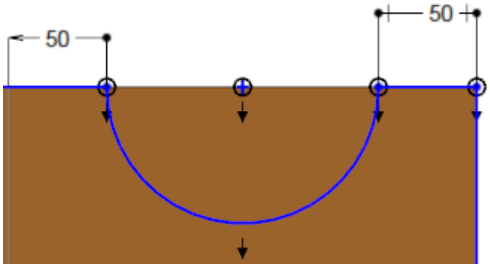
- Draw the second arc as shown below.



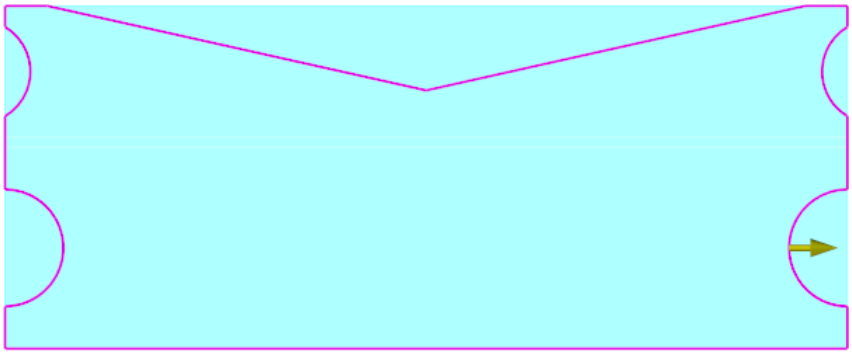
- Delete the radius dimensions on both arcs.
- Add the dimension as shown below on the left arc.



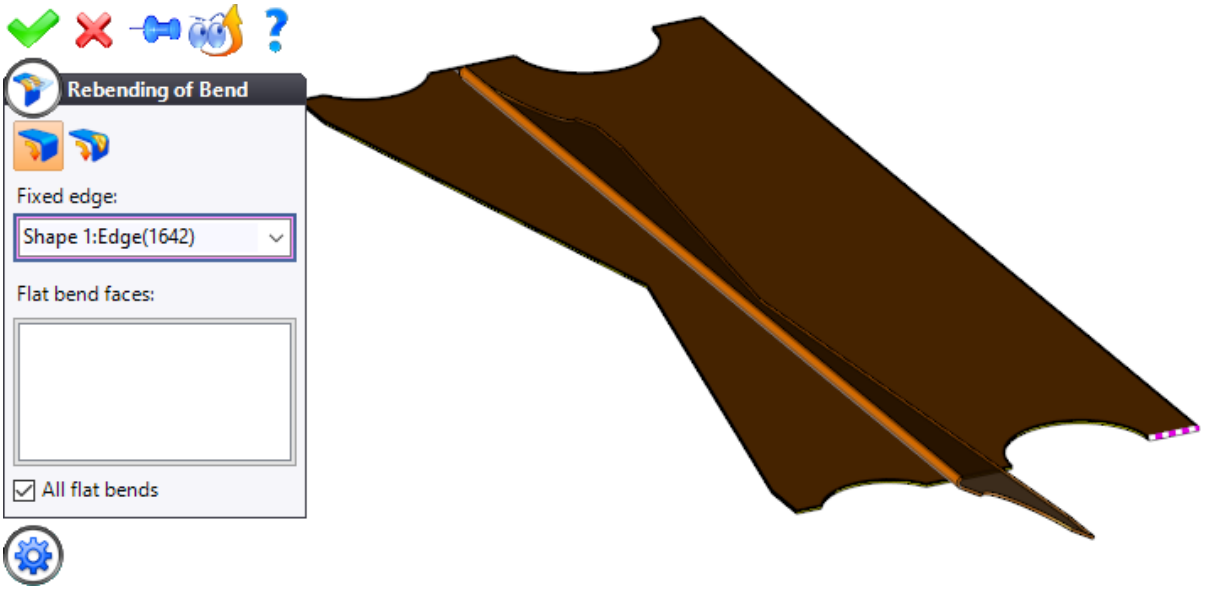
- Add a **coincidence** constraint on the center of the right arc.



- Right-click in the graphics area with no active selection and select the **Trim by Profile** command. The yellow arrow indicates the position of the waste part; in our case, it should point outward. If this is not the case, double-click on the yellow arrow.

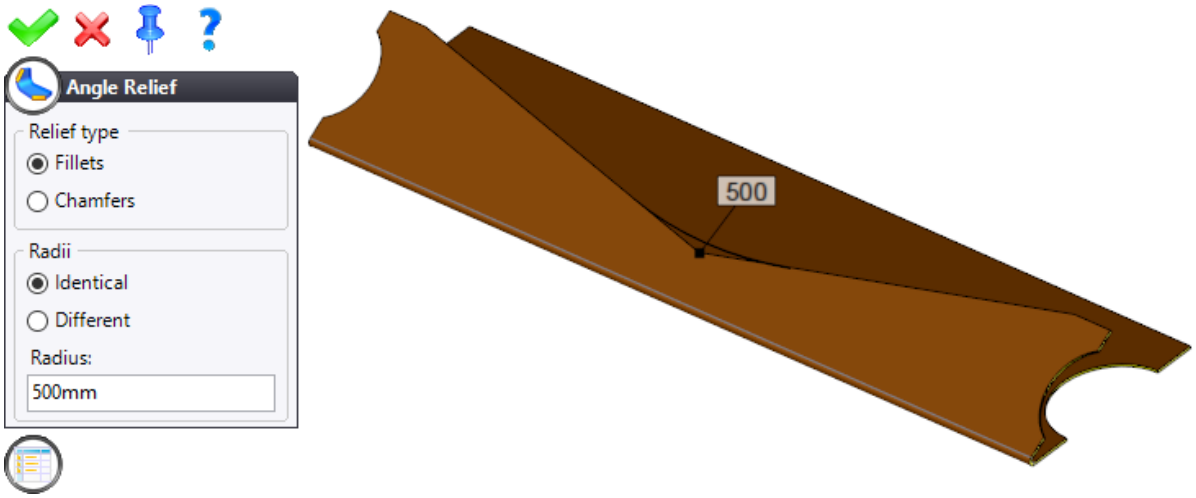


- Click on to **confirm** the operation.
- From the **Sheet Metal** tab, select the **Rebending of Bend** command.
- Select an edge of the horizontal section as the **fixed edge** and check the **All flat bends** box.



- Click on to **confirm** the operation.

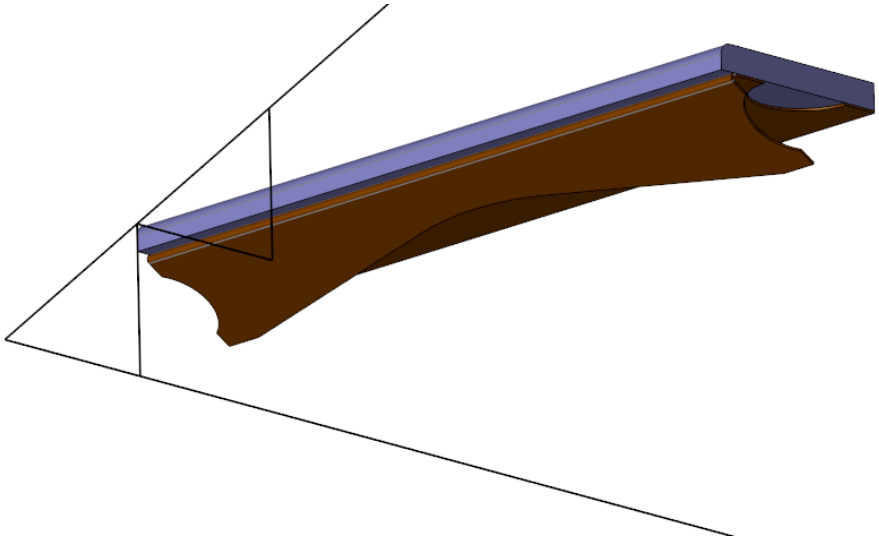
- Right-click on the edge as shown below and select the **Angle Relief** command.
- Select the **Fillets** mode and enter a **radius** of *500mm*.



- Click on to **confirm** the operation.
- **Save** the part document.

- Return to the *Right stair* assembly document.

You will notice that the step has been updated.

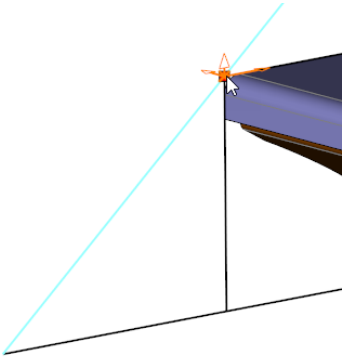


The entire step could have been created completely independently, as was the case with the railing post. The advantage is that it could have been reused on another stair.

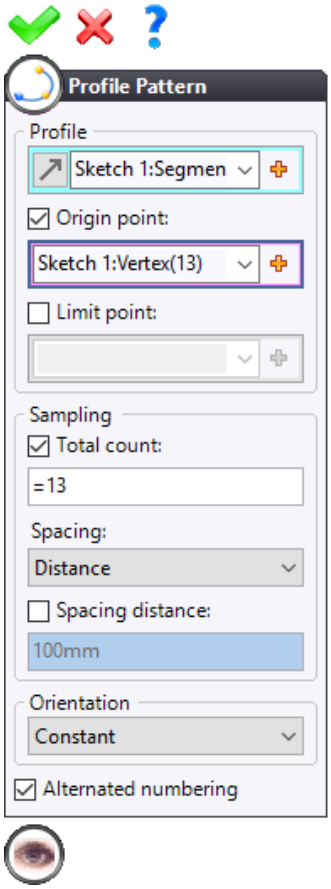
Creating the repetition

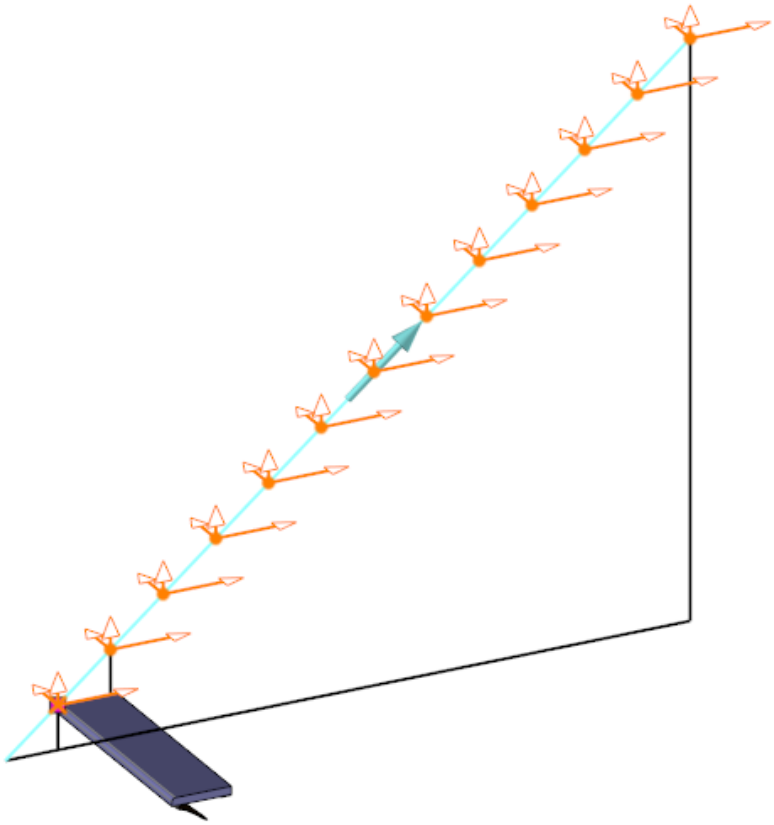
We will now create the step and its support.

- From the **Construction** tab, select the **Repetition** command.
- Select the step and its support as the **entities** to be repeated.
- In the **Pattern** field, click on the icon and select **Profile Pattern**.
- Select the treading as the **profile**. To do this, use the rotary picking technique to avoid selecting the entire triangle.
- Check the **Origin point** box and select the nose of the first step.



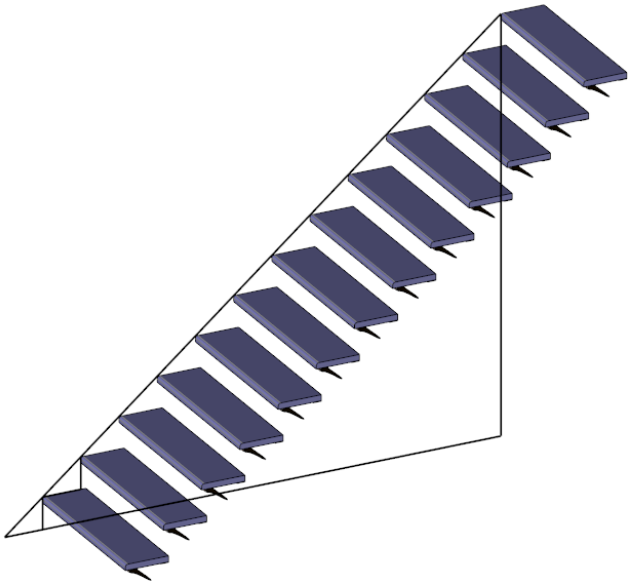
- Check the **Total count** box and select the **StepNumber** parameter. Select **Distance** as the **spacing** type, uncheck the **Spacing distance** box, then select a **Constant** orientation type.





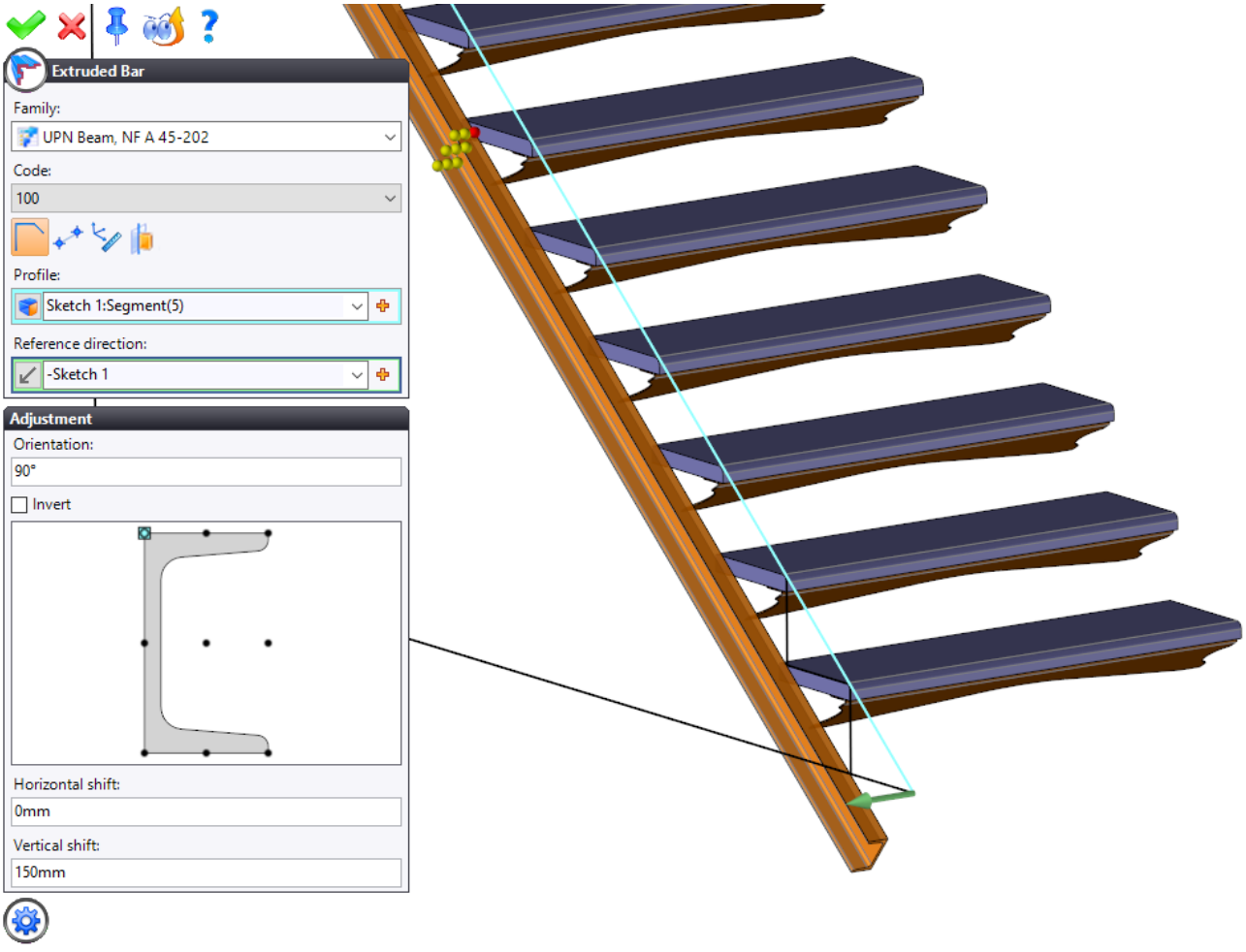
Note: This pattern is very practical because, in addition to being able to work with a quantity, we could have simply used the step height or the tread.

- Click on to **confirm** the pattern, and then the repetition.

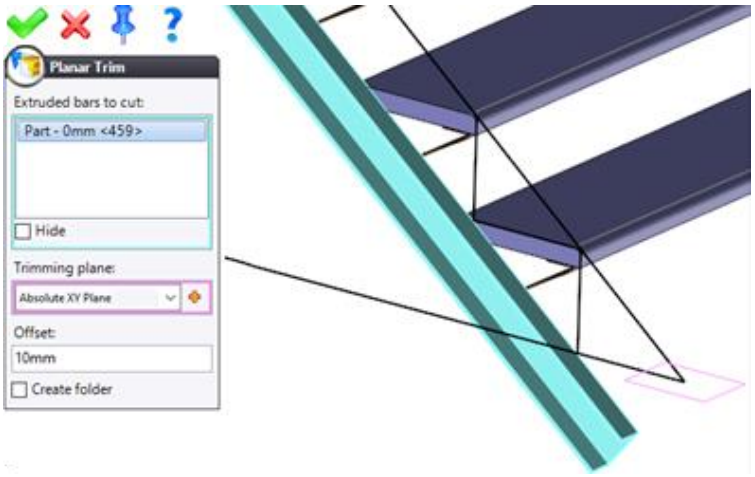


Creating the stringers

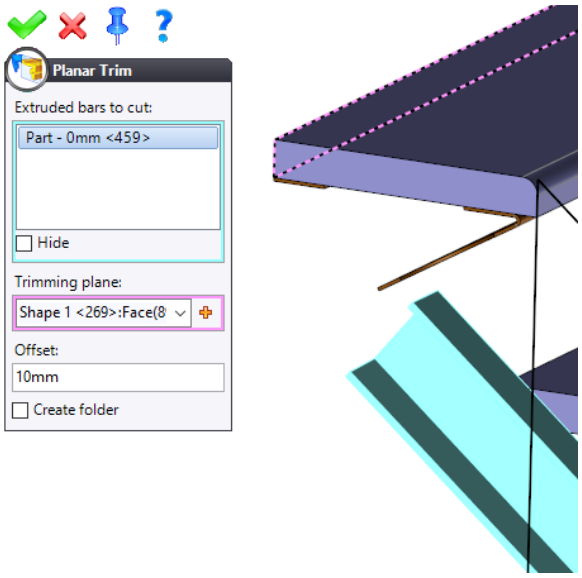
- Right-click on the treading segment and select the **Extruded bar** command.
- Select the **UPN Beam 100, NF A 45-202** family and the **100** code.
- Invert the **reference direction**.
- Enter an **orientation** of *90°* to position the U outward, then select the top left positioning point.
- Enter a **vertical shift** of *150mm*.



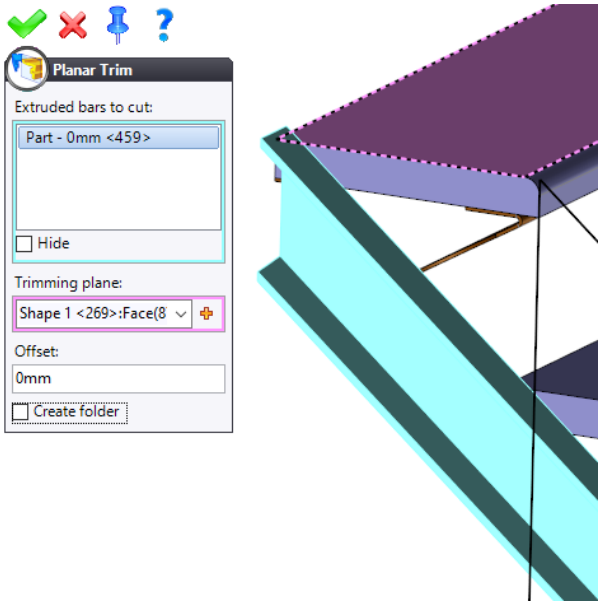
- Right-click on the extruded bar and select the **Planar Trim** command.
- Select **Absolute XY plane** from the drop-down list and enter an **offset** of *10mm*.



- Click on to **confirm** the operation.
- Repeat the operation on the top part of the stringer by selecting the back face of the landing step.

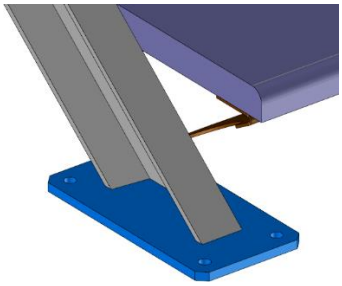


- Perform a final planar trim with the top face of the landing step. Make sure to adjust the offset value to *0mm*.

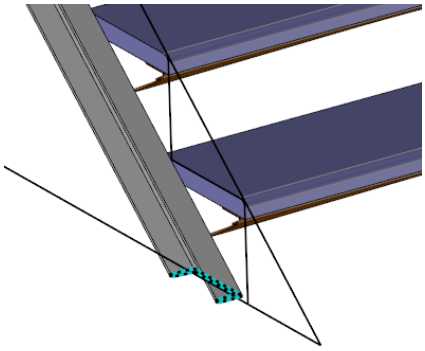


Creating the plates

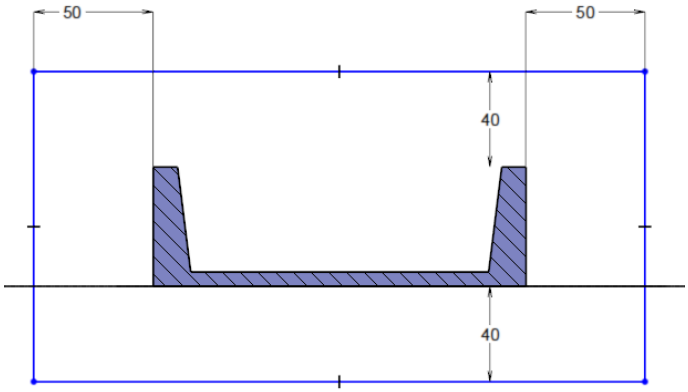
Down plate



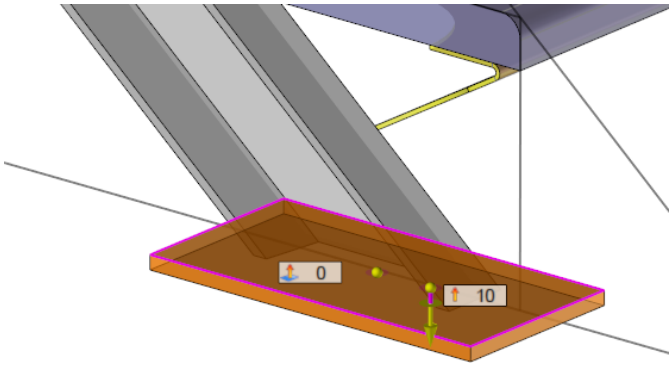
- Right-click on the following face and select the **In Place Part** command.



- Enable the **Cut** mode at the top right of the screen.
- Draw the sketch as shown below.

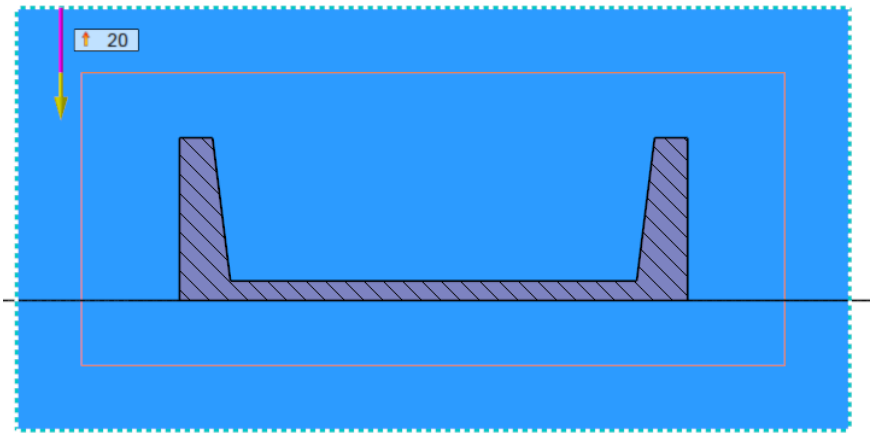


- Right-click in the graphics area with no active selection and select the **Sheet Metal on Sketch** command.
- Orient the part downward and enter a **thickness** of *10mm*.

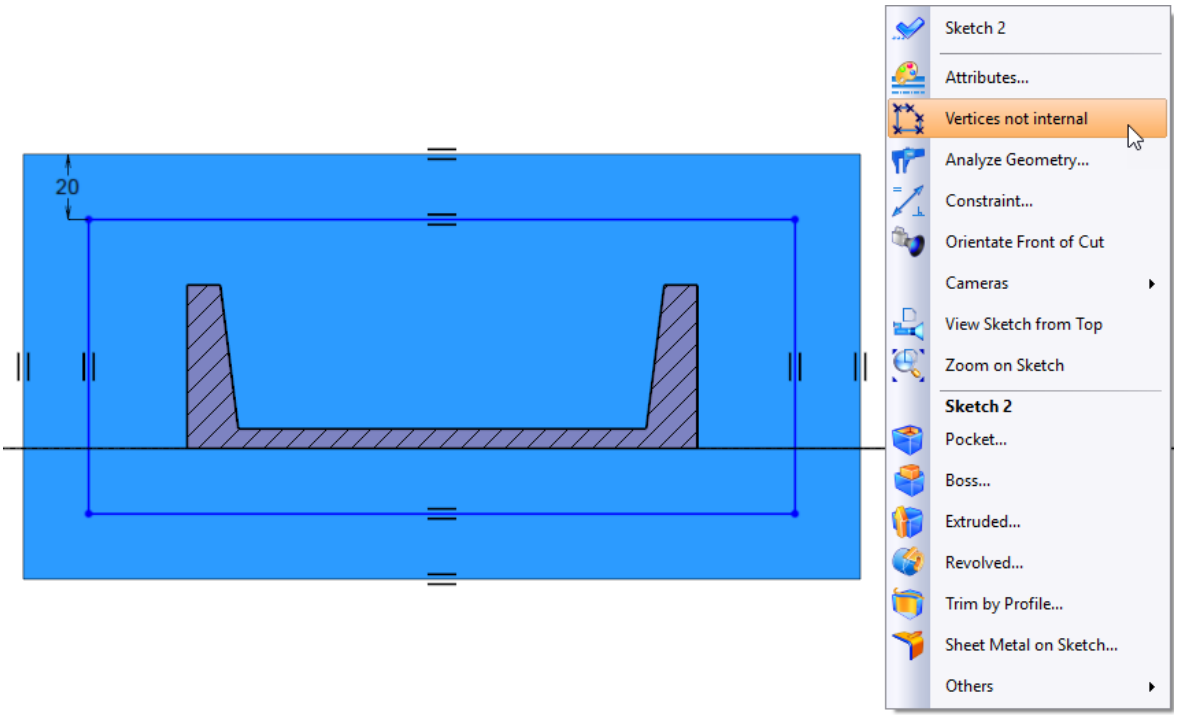


- Click on to **confirm** the operation.

- Create a new sketch on the top face of the plate.
- Select the **Offset** command.
- Select the contour of the plate, enter a **distance** of *20mm* and orient it inward.

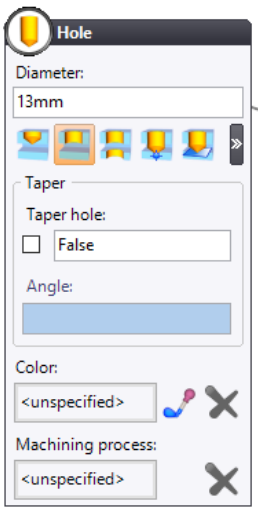


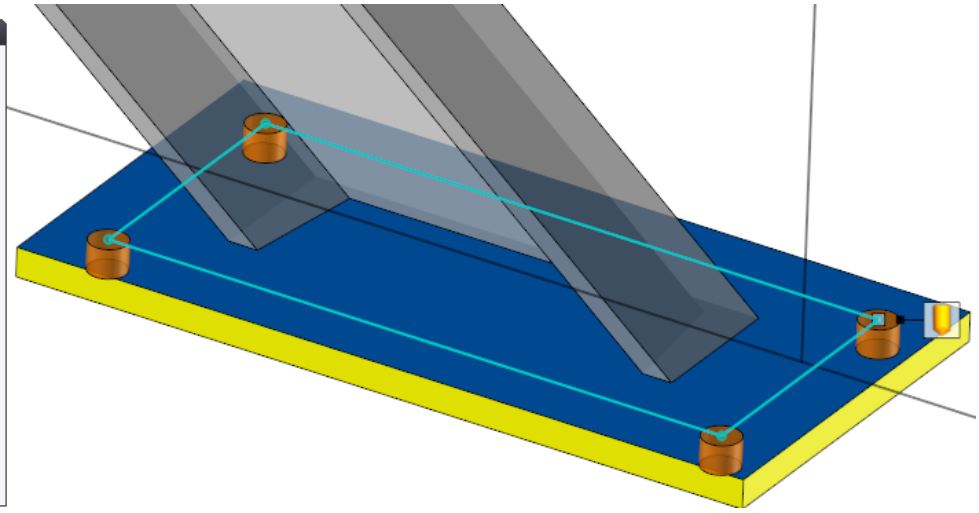
- Click on to **confirm** the operation.
- Right-click in the graphics area with no active selection and select the **Vertices not internal** command.



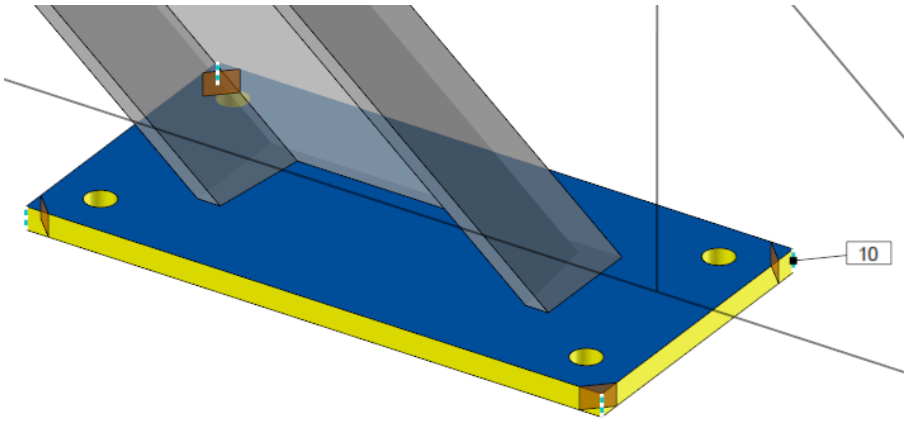
This command makes all the vertices of the sketch available outside of the sketch. You can modify this state locally on one or more vertices by selecting the **Internal** or **Not Internal** command via the contextual menu.

- Right-click in the graphics area with no active selection and select the **Drilling Group** command. **TopSolid** positions a drilling on each vertex declared as not internal.
- Enter a **diameter** of *13mm.*





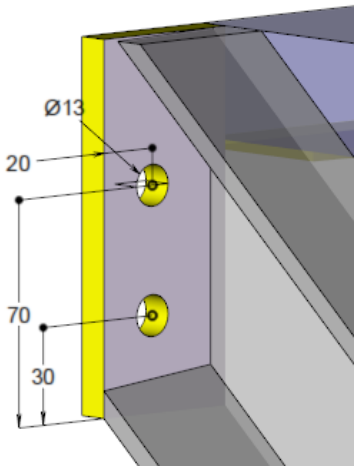
- Click on to **confirm** the operation.
- Right-click on one of the angles and select the **Angle Relief** command.
- Select the **Chamfers** mode, enter an **offset** of *10mm* and select the other three angles.



- Click on to **confirm** the operation.
- **Confirm** the in-place editing context.

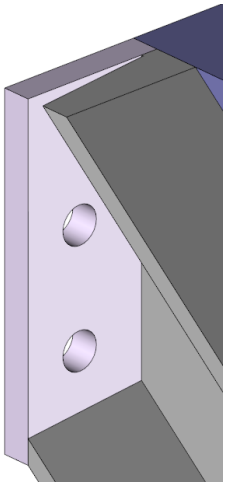
Upper plate

On the upper part of the stringer, we will now add the plate as shown below with its drillings.

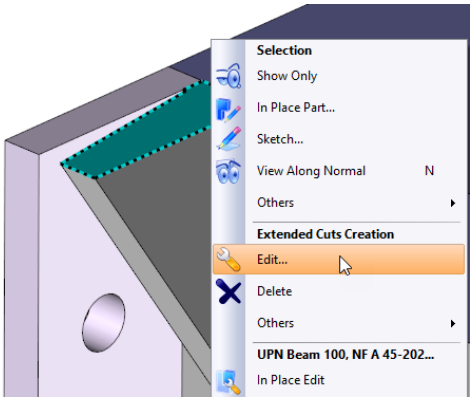


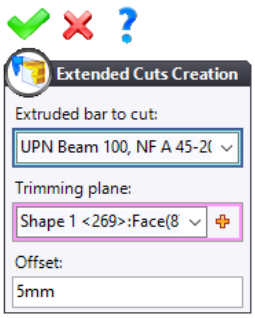
- To do this, create a new **in-place part**.
- Create a **sheet metal on sketch** with a thickness of *10mm*.
- Create the above sketch with non-internal vertices.
- Create a **drilling group**.
- **Confirm** the in-place editing context.

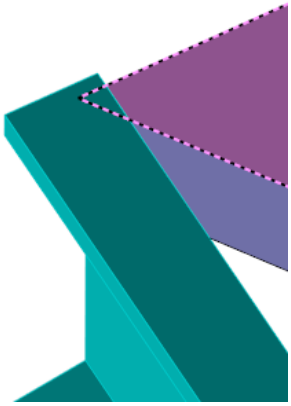
You will then notice the disparaging detail as shown below.



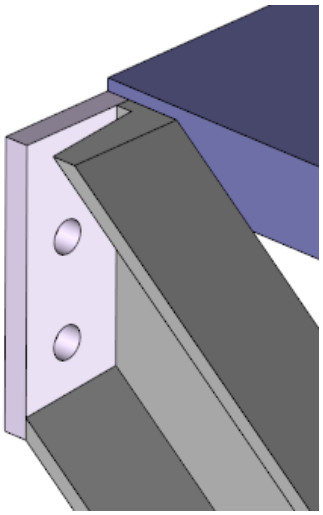
- Right-click on the section of the extruded bar and select the **Edit** command.
- Add an **offset** of *5mm*.



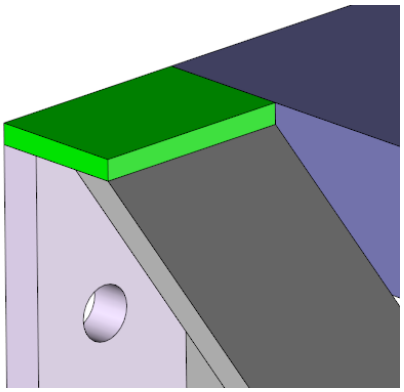




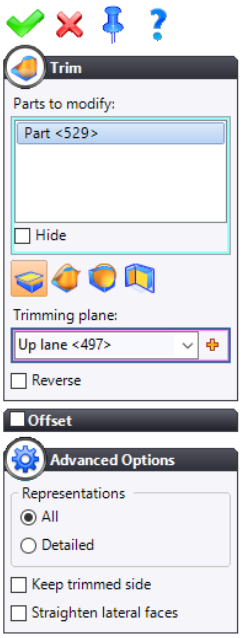
The extruded bar and the plate should normally have an offset of 5mm. If this is not the case for the plate, you have to modify the sketch of the plate.

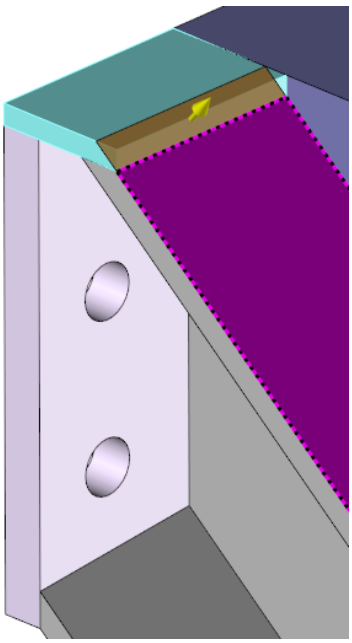


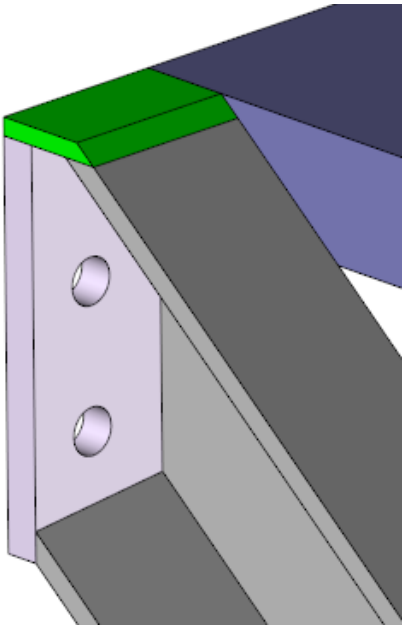
- Create a new **in-place part** with a **sheet metal sketch** with a thickness of *5mm*.



- **Confirm** the in-place editing context.
- From the **Modeling** tab, select the **Trim** command in the assembly document.
- Select the previously created part. Select the **Plane** mode and select the top face of the UPN beam. Uncheck the **Straighten lateral faces** box in the advanced options.
- Make sure to point the yellow arrow toward the waste part.

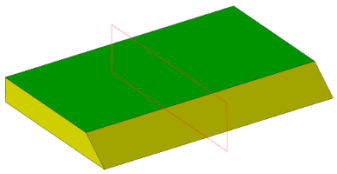
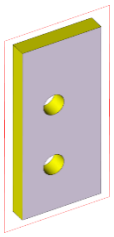
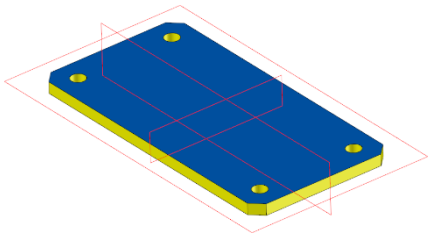






Symmetry management

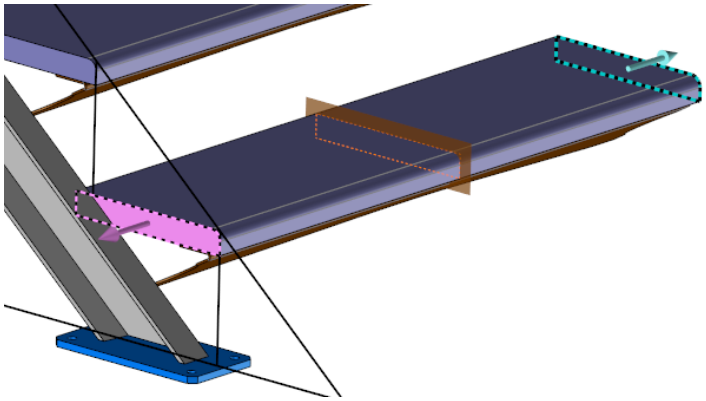
We will now look at symmetry management. Firstly, it is important to distinguish between the mirrored part and the symmetrical part. Here are a few examples:

	1 symmetry plane Symmetrical part
	1 symmetry plane Symmetrical part
	3 symmetry plane Symmetrical part
	0 symmetry plane Mirrored part

All the symmetrical parts can therefore be positioned on the right side of the stair by simply turning them over. The stringer will have to have a mirrored part. We will start with the latter.

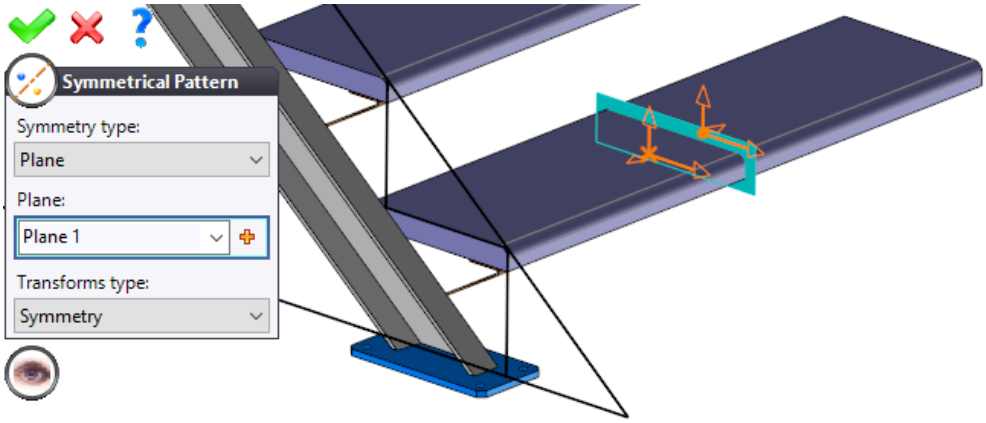
Mirrored part

- From the **Construction** tab, select the **Repetition** command. Select the stringer as the **entities** to be repeated. In the **Pattern** field, select **Symmetrical Pattern**.
- Select **Plane** as the **type of symmetry**. In the **Plane** field, click on the icon and select **Midplane**. Select the two side planes of the step.



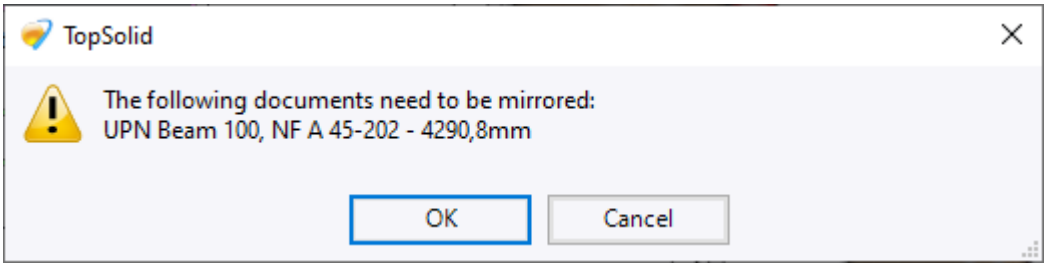
- Click on to **confirm** the operation.

- In the **Transforms type** field, select **Symmetry** from the drop-down list.



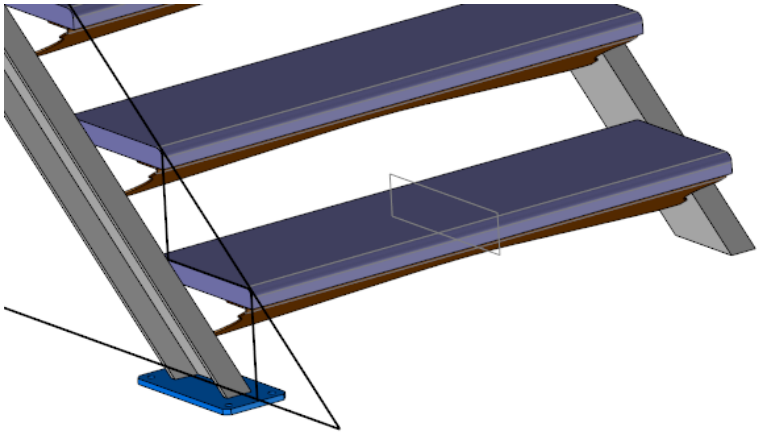
- Click on to **confirm** the pattern.
- Click on to **confirm** the repetition.

The following message appears.



- Click on **OK**.

TopSolid prompts you to select a document template for the mirror document. In this case, it is an extruded bar. It is therefore necessary to use the **Part - 0mm (Extruded bar, Derivation, Mirror, Partial part)** template. In case of a wrong choice, **TopSolid** will be in error when saving the document.

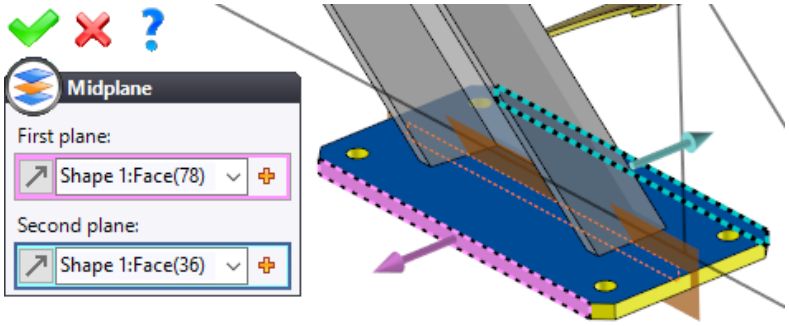


- Click on to **confirm** the operation.

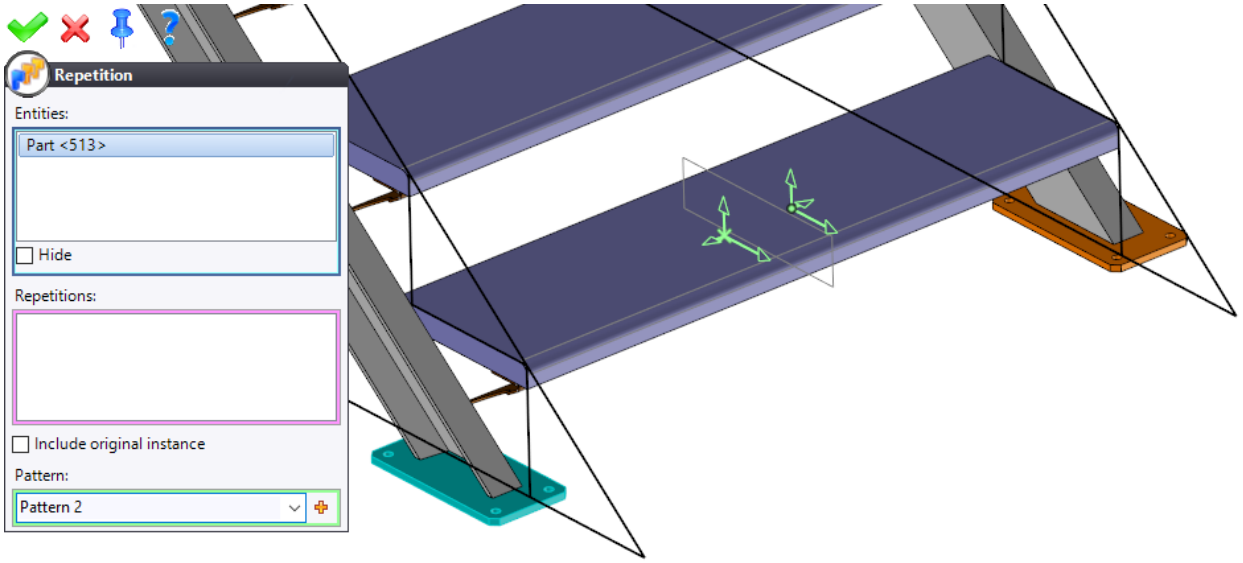
For the plates, we will see three alternative solutions.

Part with declared symmetry

- Double-click on the down plate to edit it.
- From the **Tools** tab, select the **Symmetries** > **Plane Symmetry** command.
- Click on the icon and select **Midplane**.
- Select the planes as shown below.



- Click on to **confirm** the midplane.
- Click on to **confirm** the symmetry.
- **Confirm** the in-place editing context.
- From the **Construction** tab, select the **Repetition** command.
- Select the down plate as the **entities** to be repeated.
- In the **Pattern** field, click on the mirrored stringer.

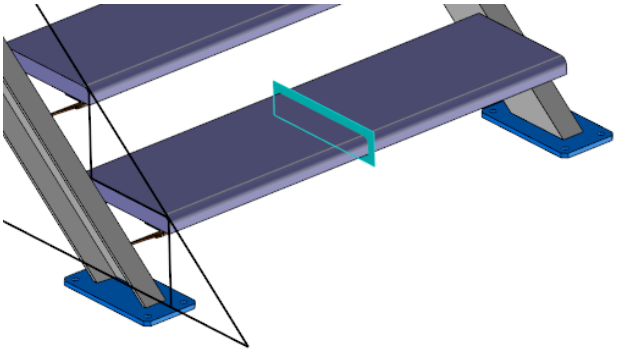


- Click on to **confirm** the repetition.

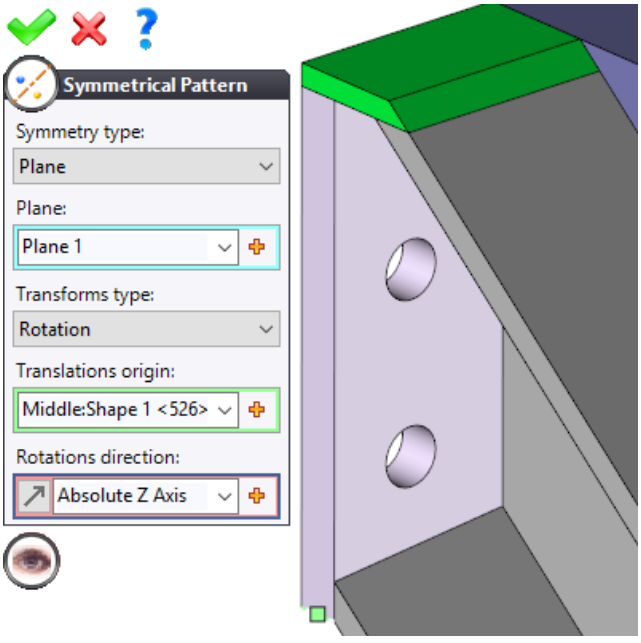
This solution takes the longest to set up but the quickest to use. It is therefore ideal for standard parts or those parts that are most likely to be reused.
Declaring symmetry also works on assemblies.

Symmetry by rotation

- From the **Construction** tab, select the **Repetition** command.
- Select the up plate as the **entities** to be repeated.
- In the **Pattern** field, select **Symmetrical Pattern**.
- Select the previously created plane.

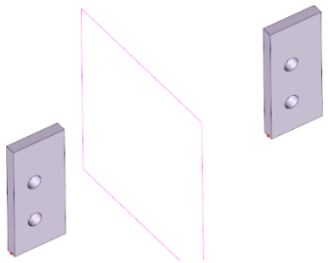
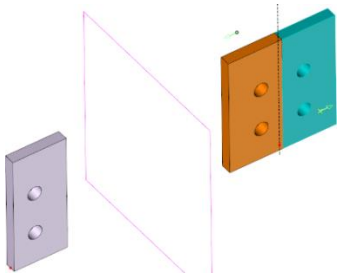
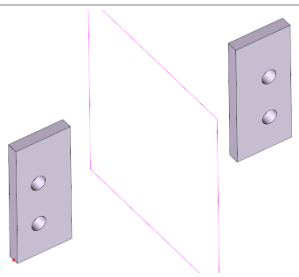


- Select **Rotation** as the **transforms type**.
- Select a point in the middle of the thickness as the **translations origin** and select the **Z axis** as the **rotations direction**.



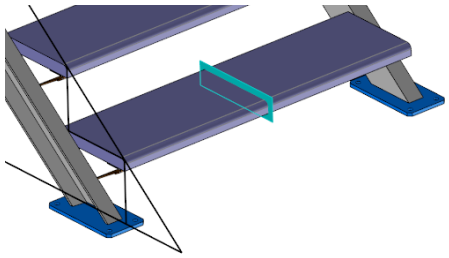
- Click on to **confirm** the pattern.
- Click on to **confirm** the repetition.

Here is what **TopSolid** does:

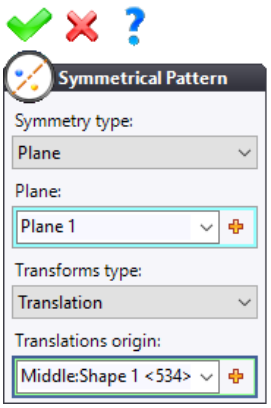
	Symmetry of the point with respect to the plane
Rotation of the plate with respect to the point (translations origin) and in the direction of the selected rotations	
	Result

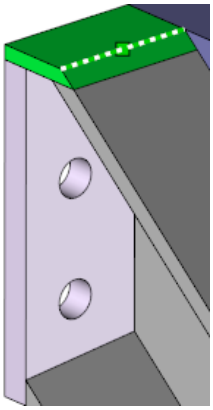
Symmetry by translation

- From the **Construction** tab, select the **Repetition** command.
- Select the last upper plate as the **entities** to be repeated.
- In the **Pattern** field, select **Symmetrical Pattern**.
- Select the previously created plane.



- Select **Translation** as the **transforms type**. Select a point in the middle of the plate as the **translations origin**.





As with rotation, **TopSolid** symmetrizes this point. It will then measure the distance between the points and apply this value to the plate.

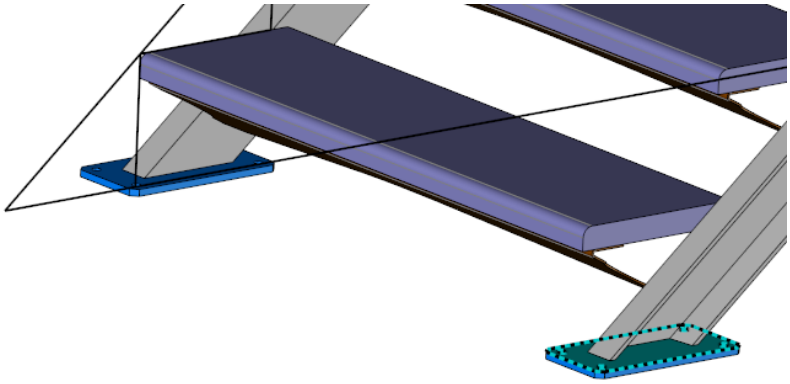
- Click on to **confirm** the pattern.
- Click on to **confirm** the repetition.

Using screw components

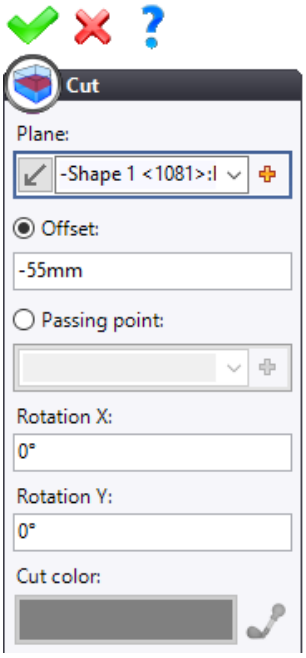
TopSolid is delivered with numerous fixing elements (screws, nuts, washers, etc.) in several different standards (ISO, AFNOR, DIN, etc.). Here we will see how to use them.

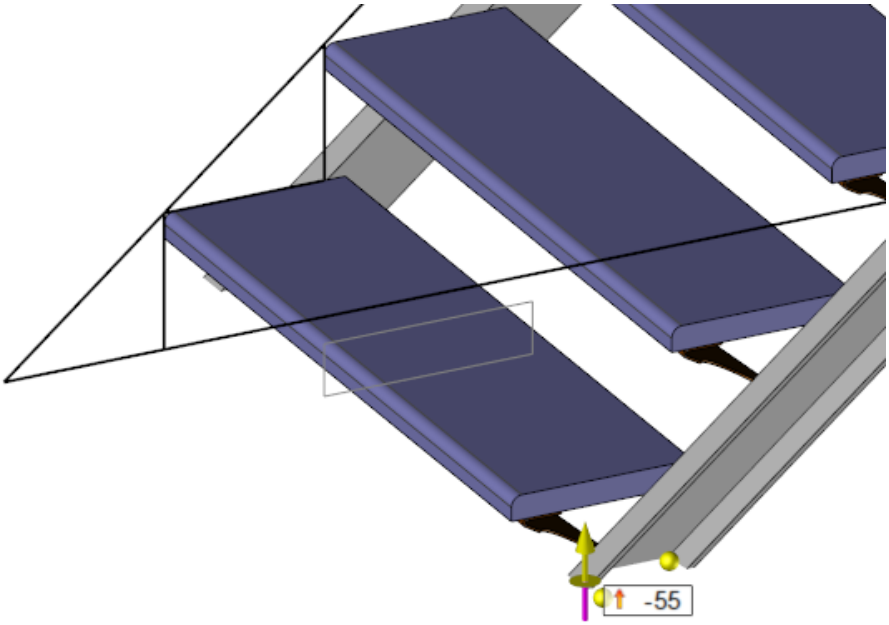
The idea is to create a sketch that will symbolize the position of the different screws, place a screw and finally repeat it on the sketch. The advantage is that if the step dimensions change, the screws will reposition themselves correctly.

- From the **Visualization** tab, select the **Cut by planes** command.
- Select the top face of the plate as shown below.



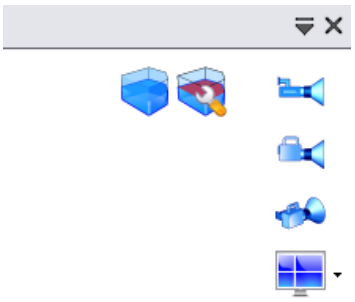
- Double-click on the yellow arrow in the graphics area to invert the direction, then drag the arrow upward to add an **offset** of *-55mm*.



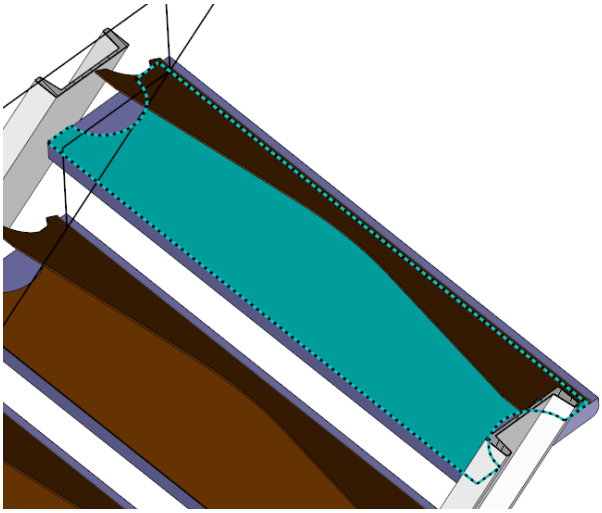


- Click on to **confirm** the operation.

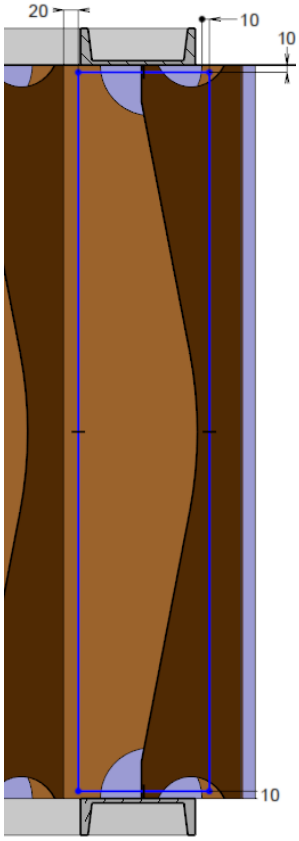
Note: The cuts are visible in the Entities tree. You can edit them from the tree, using the icons at the top right or directly via the contextual menu (without active selection).

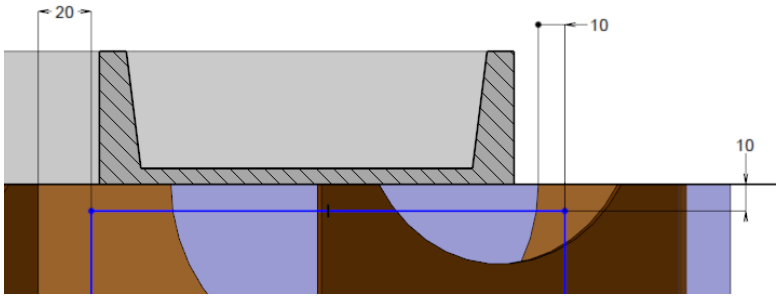


- Right-click on the following face and select the **Sketch** command.

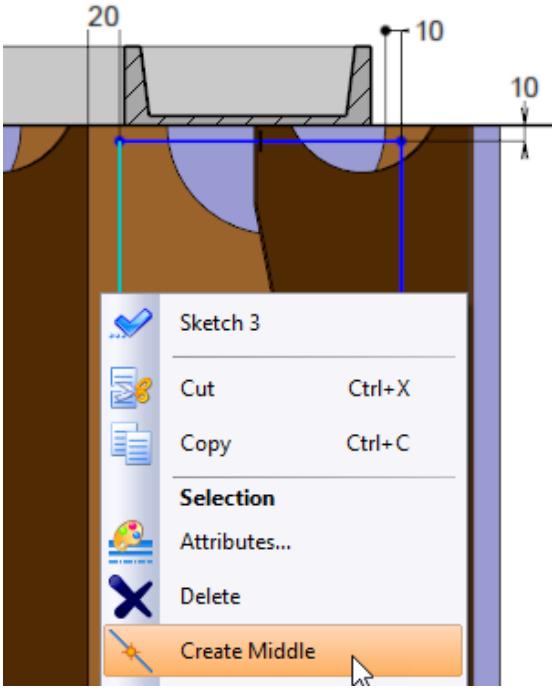


- Draw the following rectangle.





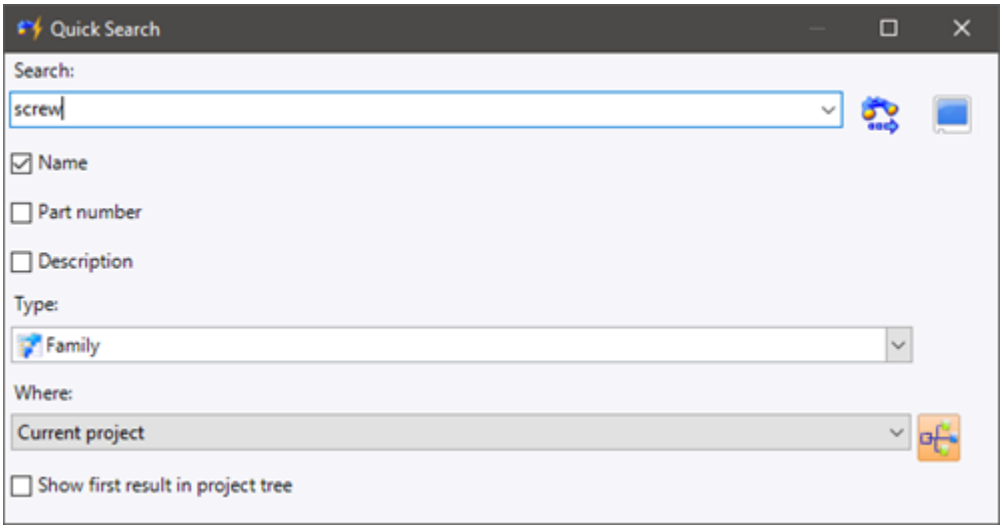
- Right-click on the left vertical segment and select the **Create Middle** command.



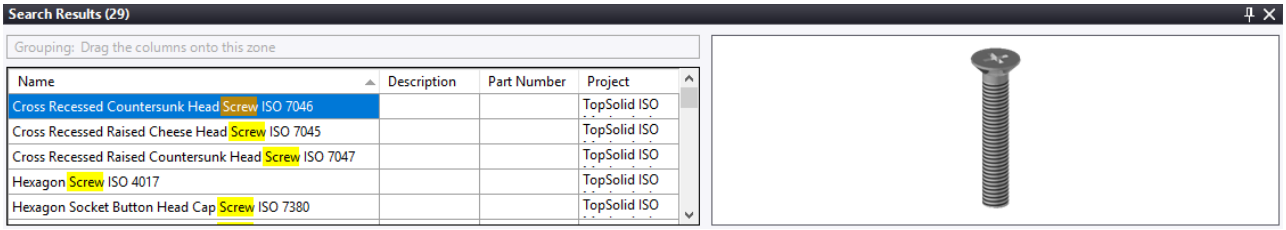
- Right-click in the graphics area with no active selection and select the **Vertices not internal** command.
- **Confirm** the sketch.

We will now search for a screw.

- At the top right of the screen, select the **Quick Search** command.
- Adjust the settings as shown below.



- Run the search by clicking on the icon. **TopSolid** displays the following search results.



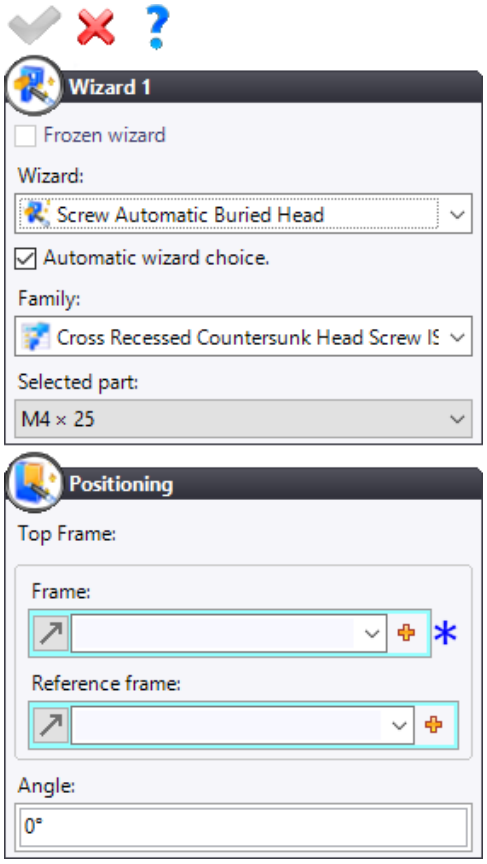
- **Close** the quick search dialog box.

Note: You can lock the search results dialog box in the open state by clicking on the pin at the top right of the screen.

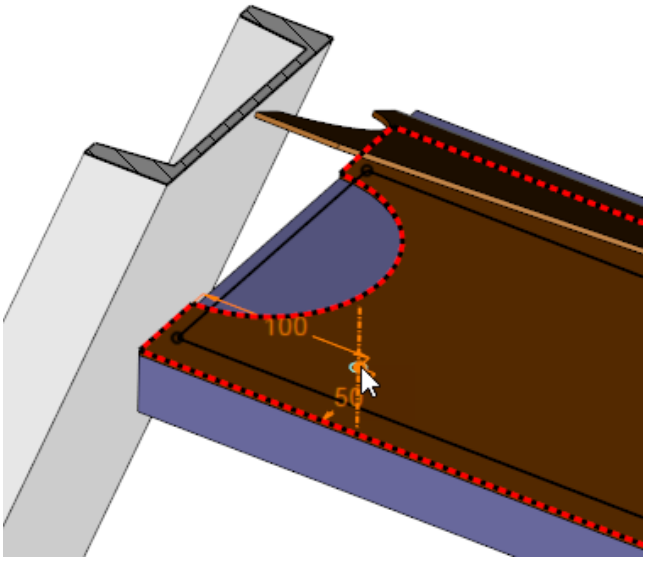
- Include the **Cross Recessed Countersunk Head Screw ISO 7046**. To do this, click on the screw line and drag it into the graphics area.

The positioning wizard is displayed.

- Select the **Screw Automatic Buried Head** wizard and the **M4 x 25** size.

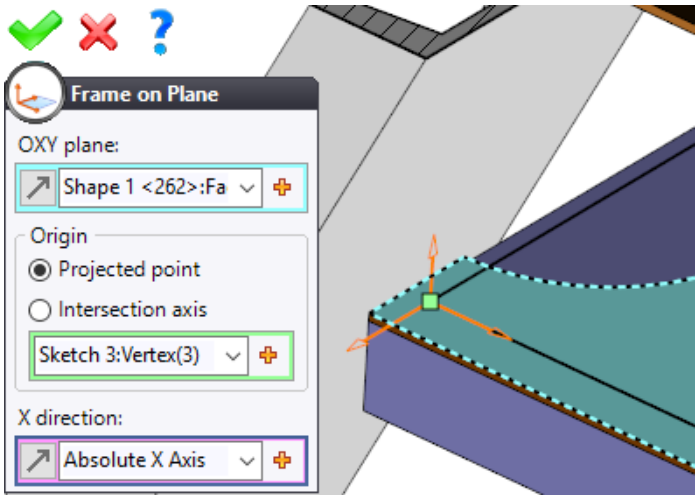


As for drillings, **TopSolid** dynamically positions the screw based on the nearest edges.



It could have worked, but that is not what we want here. We would like to hook on to the sketch.

- In the **Frame** field, click on the icon and select the **Frame on Plane**.
- Select the face of the sketch as the **OXY plane**, select the **Projected point** mode, and then select the point of the sketch as shown below.

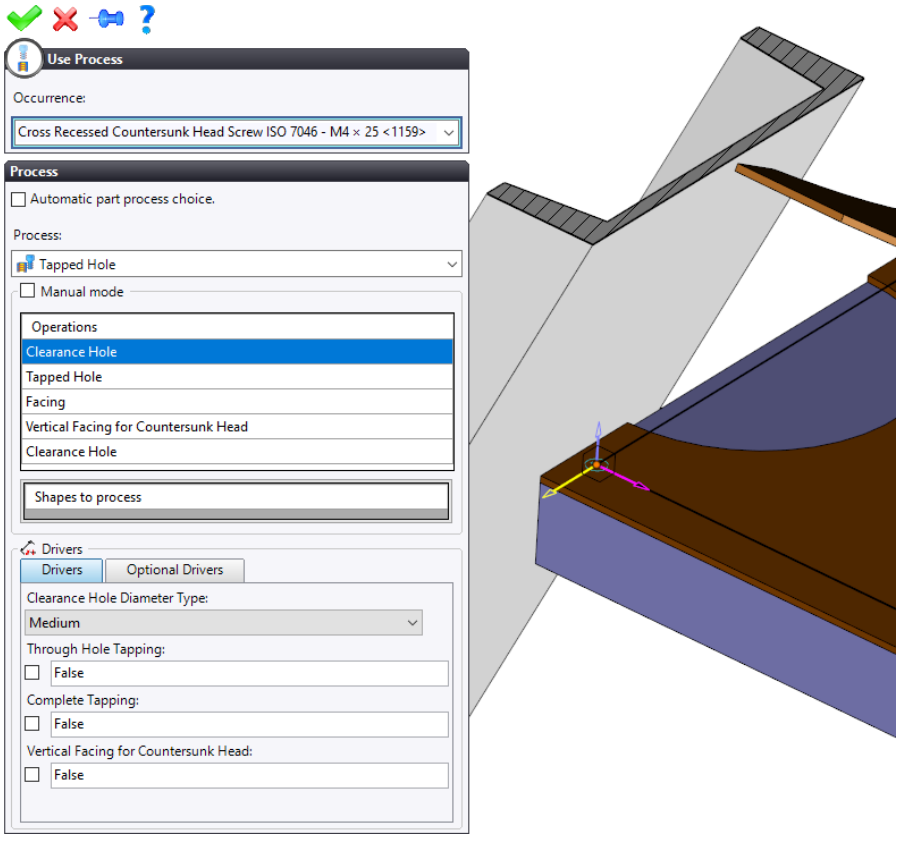


- Click on to **confirm** the frame.
- Click on to **confirm** the wizard.

TopSolid then proposes to create the process associated with the screw. A process is a set of operations related to the component, in this case the screw. It can be a drilling, countersinking, tapping, cutting, etc.

- Select the **Tapped Hole** process.

TopSolid uses the ISO standard to dimension the various operations. The values can nevertheless be forced in the lower part of the dialog box.

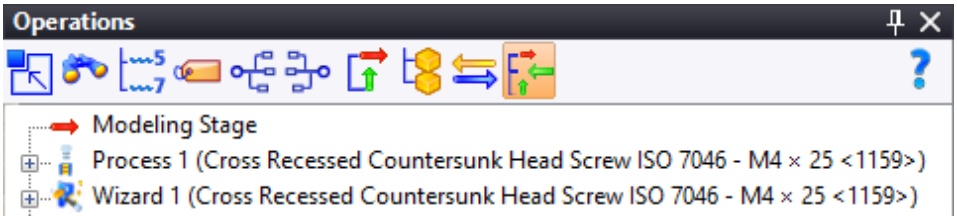


- Click on to **confirm** the process.

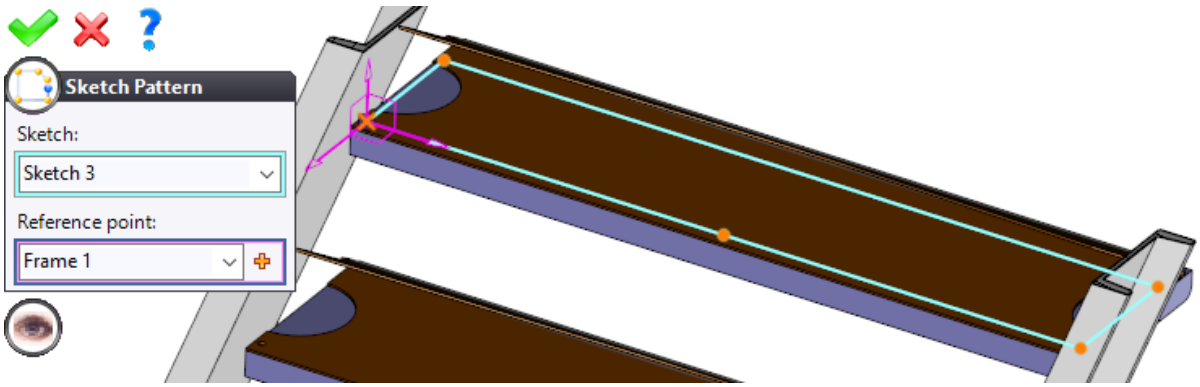
The drilling, countersinking and tapping are created.

The process is linked to the screw. If we move, delete, repeat or modify (diameter change) the screw, the process will be impacted.

The Operations tree displays two very distinct operations. The first **Wizard** operation corresponds to positioning and the second **Process** operation corresponds to the associated machining.

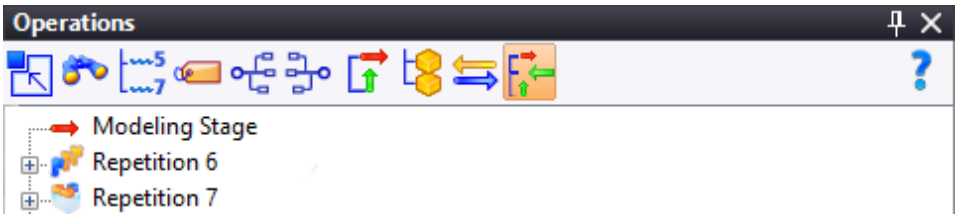


- From the **Construction** tab, select the **Repetition** command.
- Select the screw as the **entities** to be repeated.
- In the **Pattern** field, select **Sketch Pattern**.
- Select the sketch as shown below and select the frame that was used to position the screw as the **reference point**.

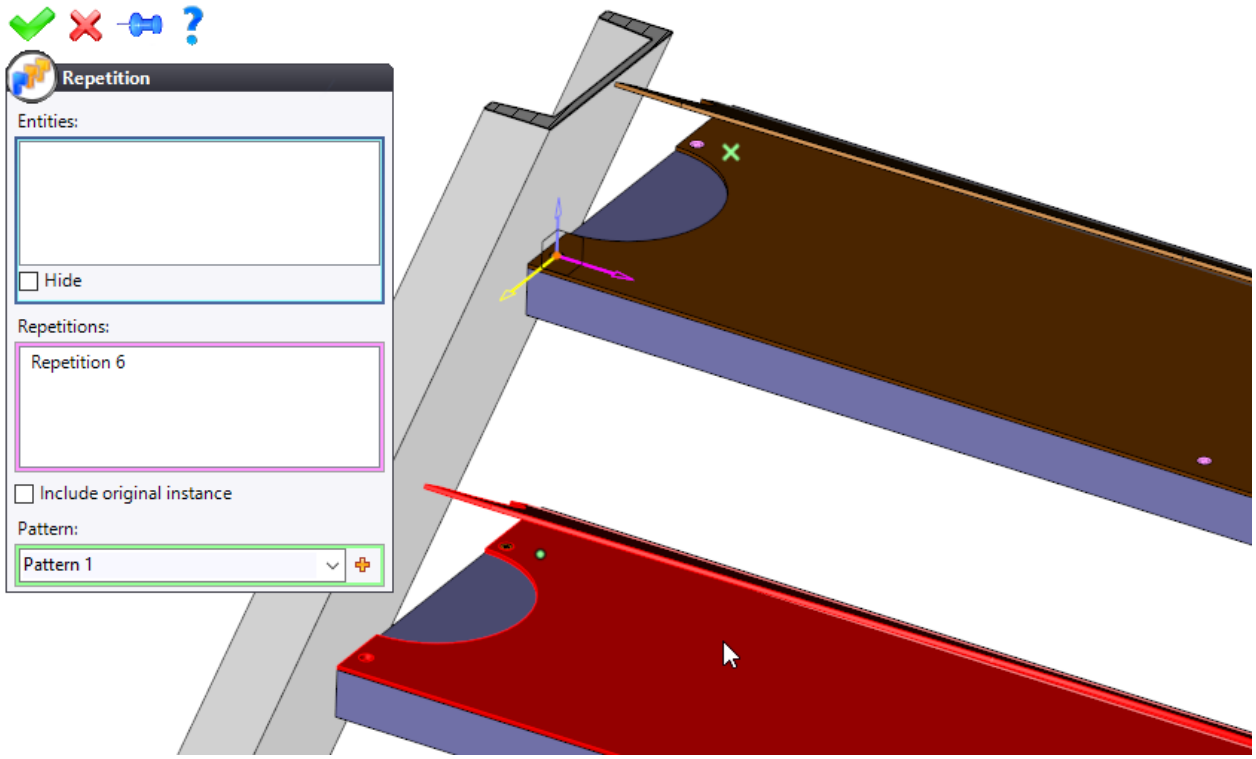


- Click on to **confirm** the pattern.
- Click on to **confirm** the repetition.

As for the inclusion of the screw, the repetition was done in two steps, one for the screw and one for the process.



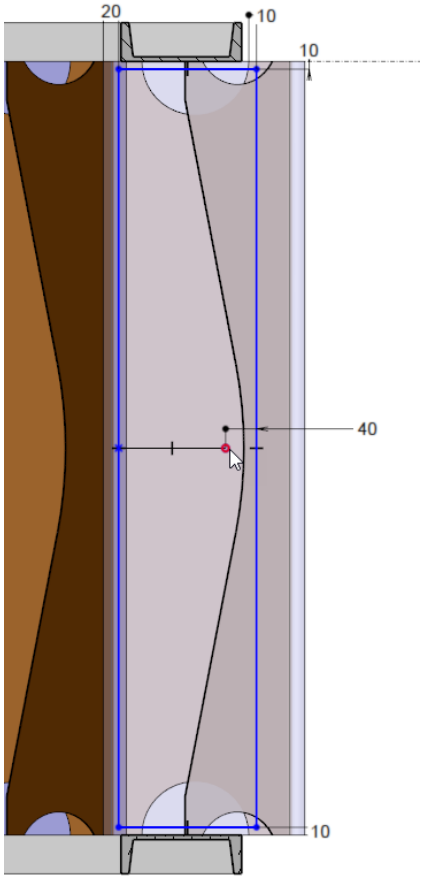
- From the **Construction** tab, select the **Repetition** command.
- In the **Repetitions** field, select the repetition of the screw.
- Check the **Include original instance** box.
- In the **Pattern** field, select one of the repeated steps.



Note: Using the **Repetitions** field instead of the **Entities** field allows you to manage the case where the number of screws changes.

- Click on to **confirm** the repetition.
- From the Operations tree, right-click on the sketch that was used for the repetition of the screws (normally sketch 2) and select the **Edit** command.

- Using the **Point** command, add the point as shown below, then constrain it.



Note: By default, the points created with the **Point** command are not internal.

- **Confirm** the sketch.

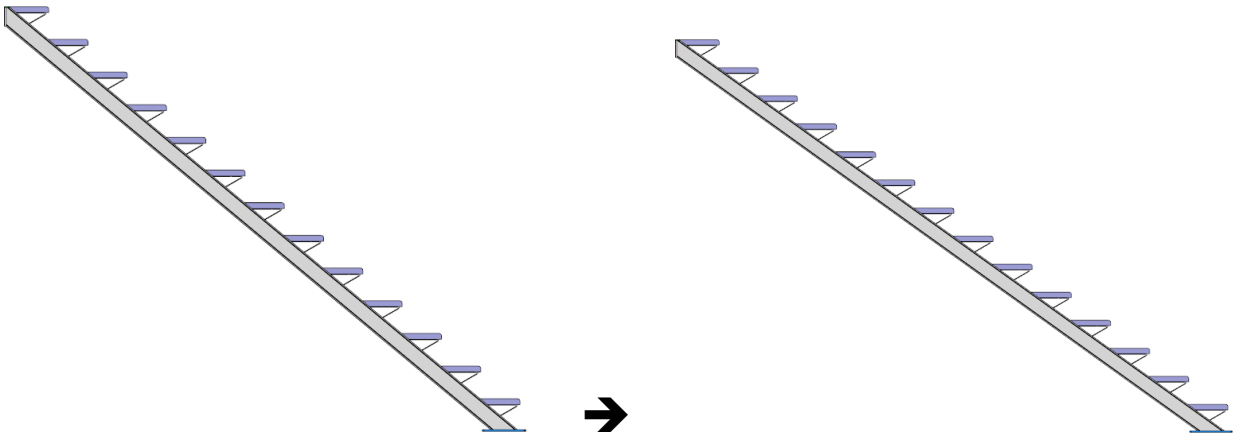
The set is updated with the addition of the screw, the process, the repetition of the screws and the repetition on all the steps.

- Right-click in the graphics area with no active selection and select the **Deactivate cut** command.

We will now make sure that the entire construction is well done.

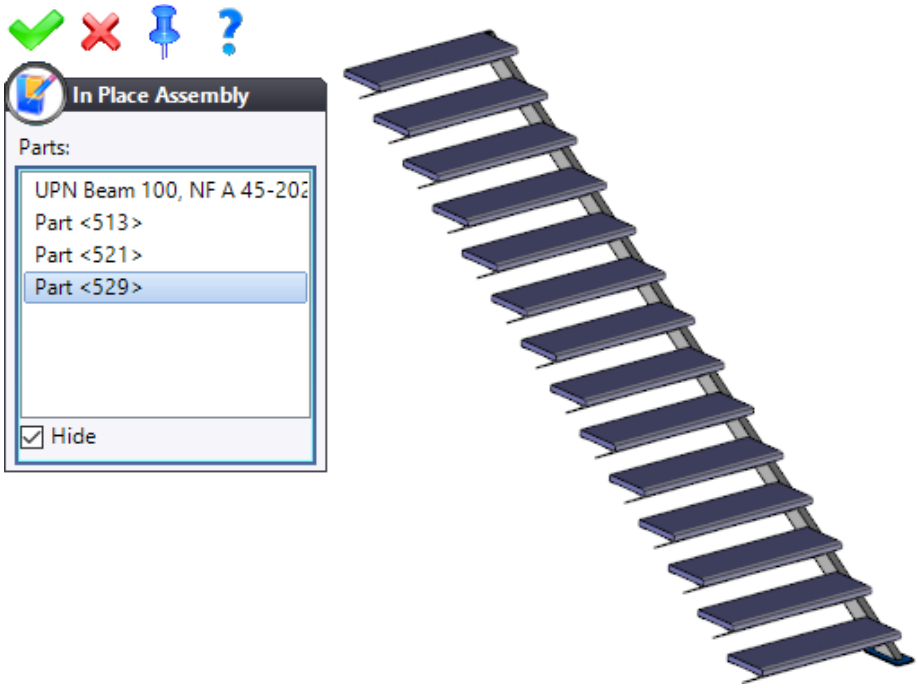
- From the Entities tree, adjust the value of the **TotalRunMax** parameter to *3500mm* and that of the **UnitRiseMax** parameter to *180mm*.

The entire stair must be recalculated without any problems.



Creating the stringer sub-assemblies (in-place assemblies)

- From the **Modeling** tab, select the **In Place Assembly** command.
- Select all the parts constituting the left stringer. To make the selection easier and make sure you do not forget anything, check the **Hide** box.



- Click on to **confirm** the operation.
- Repeat the operation on the right side.
- **Save** the assembly document.

Whenever possible, it is important to create the sub-assemblies at the end of the study. Indeed, adding an intermediate assembly hides part of the context. For example, on an extruded bar, it will no longer be possible to edit a section. In the Operations tree, you will need to move the cursor before creating the assembly in place.

Bill of Materials

- Right-click on the *Right stair* assembly document's tab and select the **Bill of Material** command.
- Select the **Multi-level** template from the **Steel Standard Templates - United States** folder and click on to **confirm** the operation.
- Make sure that the **Groups by properties** box is checked and click on to **confirm** the operation.

ID.	QTY	DESCRIPTION
	1	Right Stair
1	2	Assembly
1.1	1	Part
1.2	1	Part
1.3	1	Part
1.4	1	UPN Beam 100, NF A 45-202
2	71	Cross Recessed Countersunk Head Screw ISO 7046 - M4 × 25
6	14	Part
7	14	Part

Here is an example of one of the disadvantages of regrouping. The two stringers have been merged although they are two separate assemblies. As they have strictly the same properties, **TopSolid** therefore considers that they can be regrouped.

- Right-click on the **Assembly** line and select the **Show in project tree** command. **TopSolid** selects the assembly in the Project tree.
- Right-click on the assembly document and select the **Properties** command. Enter *Left Stringer* in the **Designation** field and click on to **confirm** the operation.

ID.	QTY	DESCRIPTION
	1	Right Stair
9	1	Assembly
9.1	1	Part
9.2	1	Part
9.3	1	Part
9.4	1	UPN Beam 100, NF A 45-202
2	71	Cross Recessed Countersunk Head Screw ISO 7046 - M4 × 25
10	1	Left Stringer
10.1	1	Part
10.2	1	Part
10.3	1	Part
10.4	1	UPN Beam 100, NF A 45-202
6	14	Part
7	14	Part

The left and right stringers are separated.

- Rename the second sub-assembly *Right Stringer*, then rename the remaining parts as you wish.
- **Reinitialize the Bill of Material Indexes**.

ID.	QTY	DESCRIPTION
	1	Right Stair
1	71	Cross Recessed Countersunk Head Screw ISO 7046 - M4 × 25
2	1	Left Stringer
2.1	1	Down Plate
2.2	1	Finishing Plate
2.3	1	Up Plate
2.4	1	UPN Beam 100, NF A 45-202
3	1	Right Stringer
3.1	1	Down Plate
3.2	1	Finishing Plate
3.3	1	Up Plate
3.4	1	UPN Beam 100, NF A 45-202
4	14	Step
5	14	Step Bracket

- Right-click in the **Material** column of the **Step** line and select **Edit material and Coating**.
- Select the **Wood** category and the **Oak** material.
- **Save** the bill of materials.
- From the Project tree, create a folder named *Right Stair* and then place the bill of materials and the assembly documents in it.
- **Check** the entire folder into the vault.

Part 4 - PDM

Some reminders

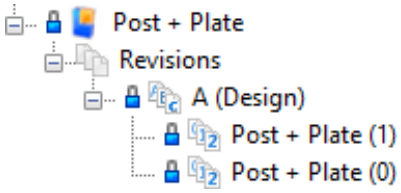
Backups

We saw previously that **TopSolid** automatically manages the backups and that it is possible to restore an earlier version at any time.

Two backups are kept by default. To adjust the number of backups, you simply have to select the **Tools** > **Options** > **PDM** > **PDM** command and adjust the value in the **Number of backups without back references to keep** field.

Check in

We then saw that in order to benefit from the revision system and make the documents available to users, you have to check them into the vault. To do this, you simply have to right-click on a document, folder or project and select the  **Check In** command.
This time the backups are deleted and the  icon is displayed in front of the document names.



With each modification, **TopSolid** automatically updates the appropriate documents, exits the vault (the  icon has disappeared and has been replaced by this  icon) and increments the minor revision (A1, A2, A3, etc.).

It is important to check documents regularly into the vault.

Validation (life cycle)

The validation option allows you to use the concept of life cycle.

It is often used when the document is going to be distributed in another department, a subcontractor or when the project completes an important phase (study milestone, launch in manufacturing, end of project, etc.) or creating company's standards.

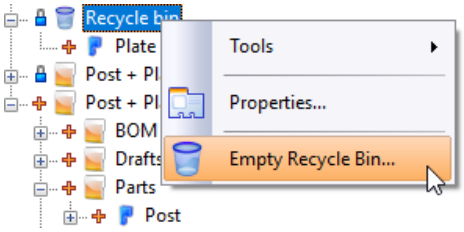
In general, changes are much less frequent than for checking documents into the vault.
After validation, updates are not automatic. It is up to the user to choose whether or not to update his document through reference redirection. If it works, **TopSolid** will increment the major revision (B, C, D, etc.).

Deletion

Deletion cycles were also discussed. If the documents are not in the vault, you simply have to delete them using the **Delete** key or via the contextual menu > **Delete**).

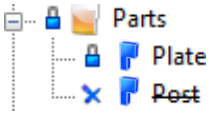
The documents are therefore placed in recycle bin of the project.

You can then empty the recycle bin if and only if the documents are not used elsewhere.



You can restore one or more documents deleted by mistake by right-clicking on the appropriate document(s) and selecting the **Restore** command.

Note: If the documents are in the vault, you will also need to put the deleted items into the vault so that they can go into the recycle bin. Before this last operation, the documents will still be in the project but simply crossed out.

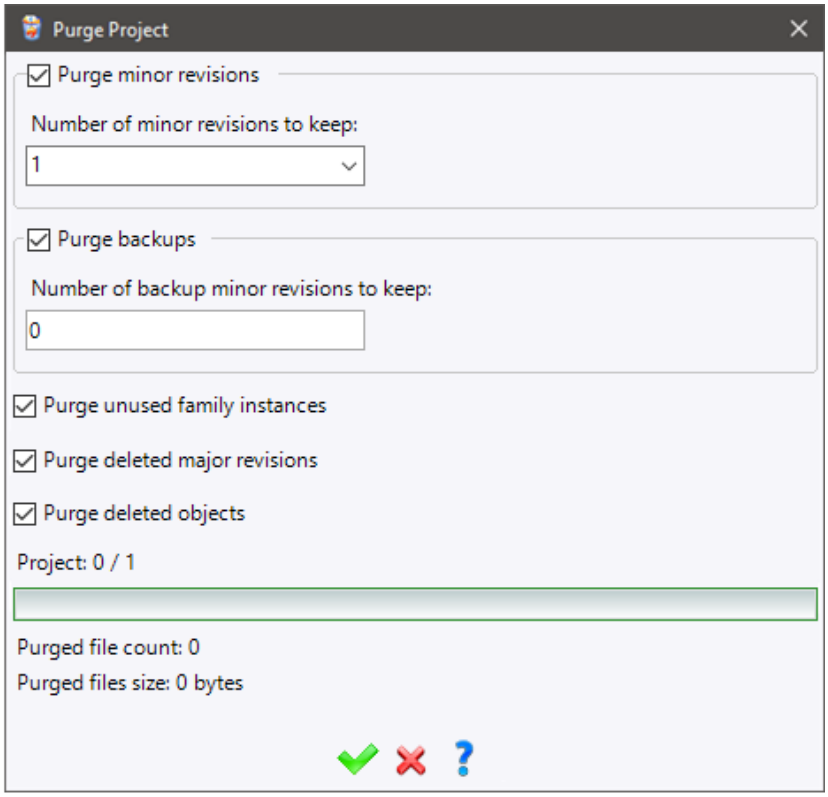


You can undo this deletion by right-clicking on the appropriate document(s) and selecting the **Undo changes** command.

Purging the project

As we have seen previously, **TopSolid** automatically creates backups, minor revisions, major revisions and documents in the recycle bin. At the end of the project, it is advisable to perform a purge which will delete all the unnecessary elements.

- Right-click on *My first project* and select the **Check In** command.
- Update the indicated documents if necessary.
- Right-click on *My first project* and select the **Deletion** > **Purge Project** command.
- Adjust the dialog box as shown below.



- Click on to **confirm** the operation.

TopSolid indicates the number of deleted files as well as the space gained.

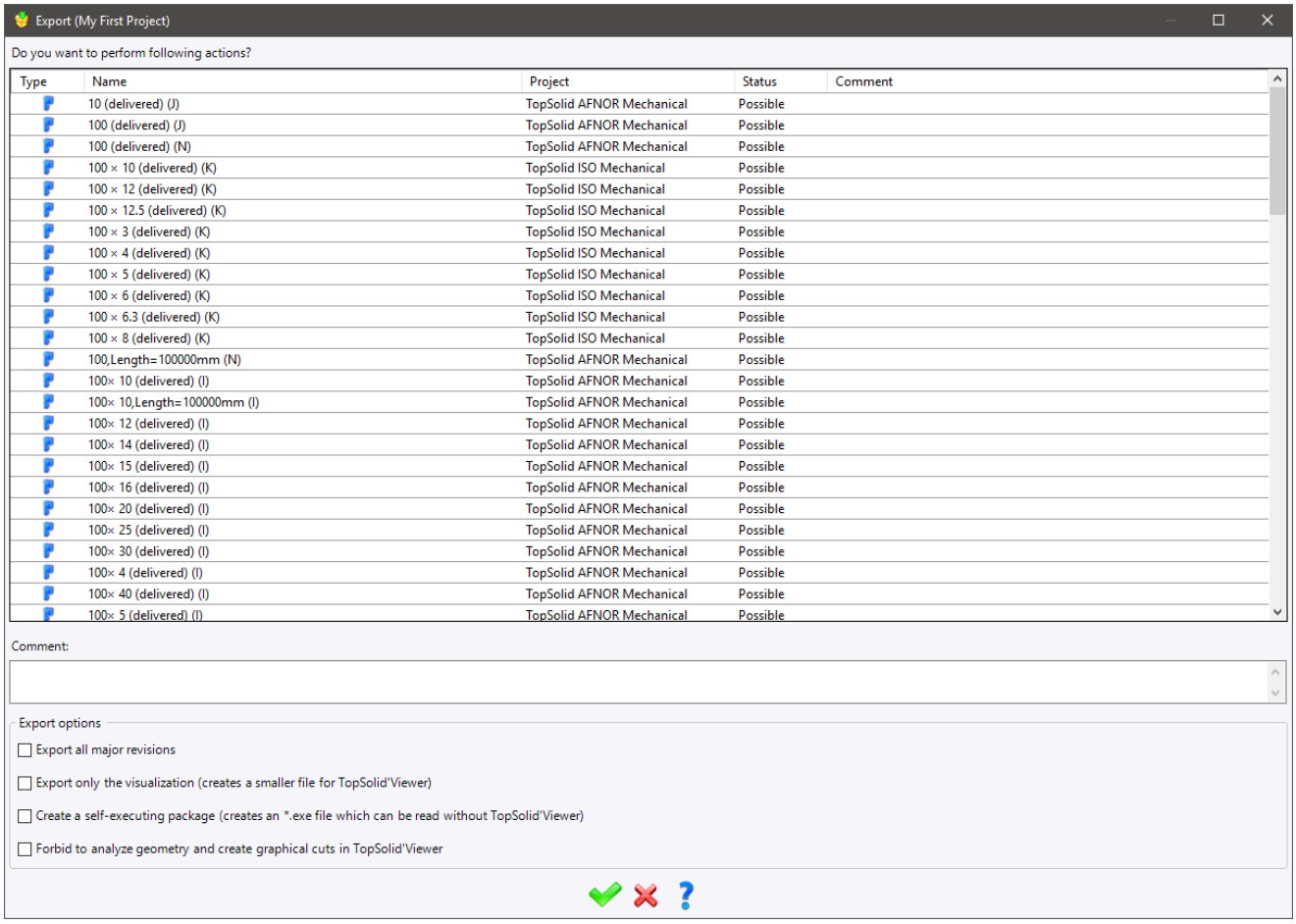
Import/export of a package

Imports/exports are used to transfer documents (customer/supplier, mobile station) from one PDM to another. You can perform an export on each document, folder or project. **TopSolid** automatically exports all the documents required for the project.

Exporting a project

- Using the **Ctrl** key, select the *Right Stair*, *Railing* and *Post + Plate* folders, then right-click on one of the folders to select the **Import/Export** > **Export Package** command.

TopSolid takes an inventory of everything it needs to export in order to import the file correctly. It also makes sure that everything is up to date and in the vault.



- Click on to **confirm** the operation.
- Place the package on the desktop.

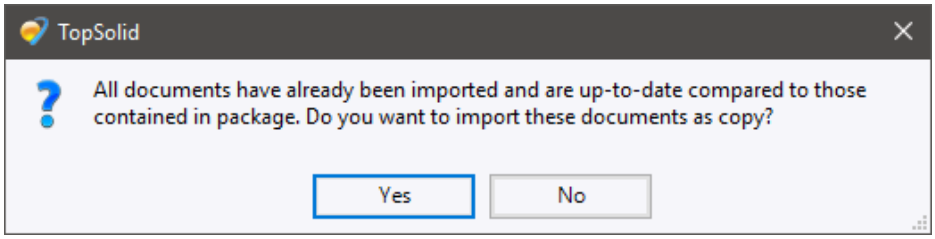
Let us say that you have been working on your project from another computer and now you want to reintegrate it into your computer. **TopSolid** offers two modes of operation:

- import in replication: **TopSolid** makes sure that the project has not been modified and updates it.
- import in copy: **TopSolid** creates a copy in a project.

Note: You can also export the *My First Project*. Everything in the project will be exported.

Import in replication

- From the **Home** tab, select the **Import Project** command.
- Select the previously created package and click on to **confirm** the operation.



TopSolid detects that there has been no modification of the project and therefore offers to make a copy.

- Click on **Yes**.

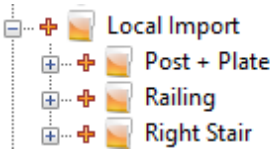
If the project had been modified, the project would have been updated.
For this to work, at the time of export, it is imperative to make sure that the documents are up to date and are in the vault.

Import in copy

Following the previous operation, a new project is created (not in the vault). It is strictly identical to the original project.

- Rename this project *My first project V2*.
- Delete the *Right Stair*, *Railing* and *Post + Plate* folders in the project.
- Create a new folder named *Local Import*.
- Right-click on the folder and select the **Import/Export** > **Import Package** command.
- Select the previously created package.

All the documents are copied and regrouped in the folder.



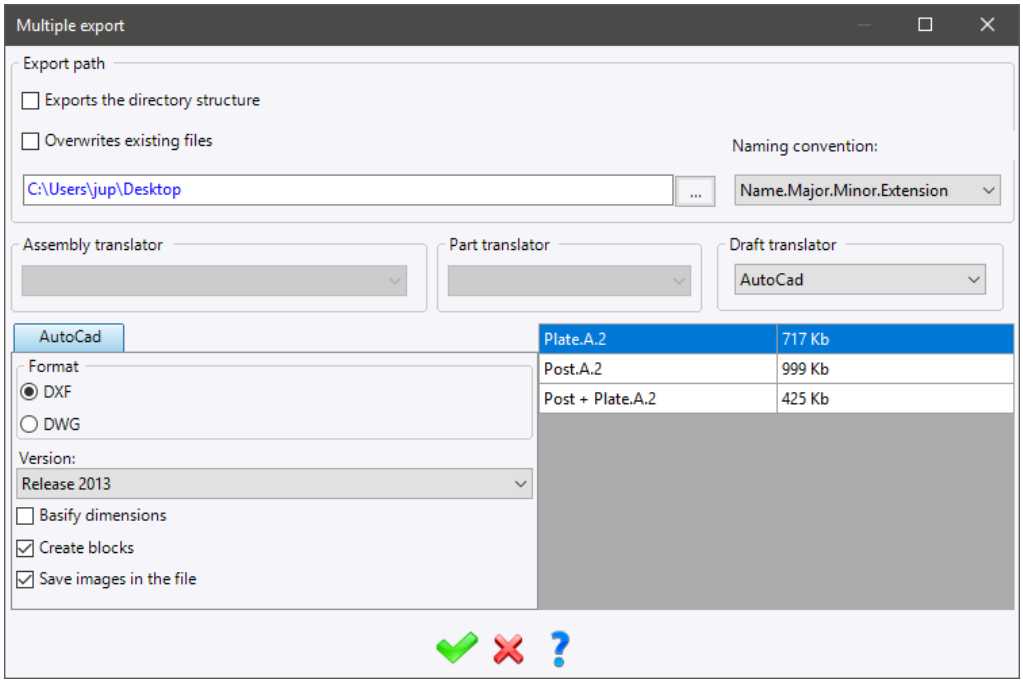
To make a copy as we have just done, at the time of export, the documents do not have to be in the vault.

- **Close** the *My first project V2* by clicking on the project's tab with the mouse wheel.

Import/export with conversion

Multiple DXF export

- In the *Post + Plate* folder, right-click on the *Draftings* folder and select the **Import/Export** > **Export several Documents with Conversion** command.
- Select the desktop as the **export path** and select **Name.Major.Minor.Extension** from the **Naming convention** drop-down list. Select the **AutoCad** translator.



- Click on to **confirm** the operation.

TopSolid highlights in green the documents that have been successfully exported.

- **Close** the dialog box.

Importing a step

Numerous websites provide access to several manufacturers' catalogues in various industries.

Here are a few of them, building oriented:

BIMcatalogs.net (Cadenas)
https://bimcatalogs.partcommunity.com/3d-cad-models/

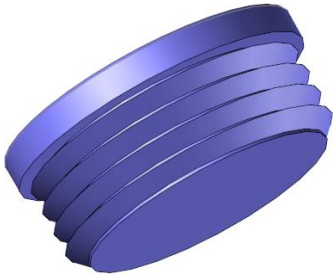
BIM&CO (TraceParts)
https://www.bimandco.com/fr/objets-bim

BIMObject
https://bimobject.com/fr/product

Polantis
https://www.polantis.com/fr/

Without forgetting the industry leaders, Cadenas and TraceParts, which have a direct interface in **TopSolid** (**File** > **Translators**).

- In the Project tree, create an *Import Step* folder, then right-click on this folder and select the **Import/Export** > **Import File with Conversion** command.
- Select the *emile_maurin_92-951-48* file.
- Click on to **confirm** the import dialog box that is then displayed.

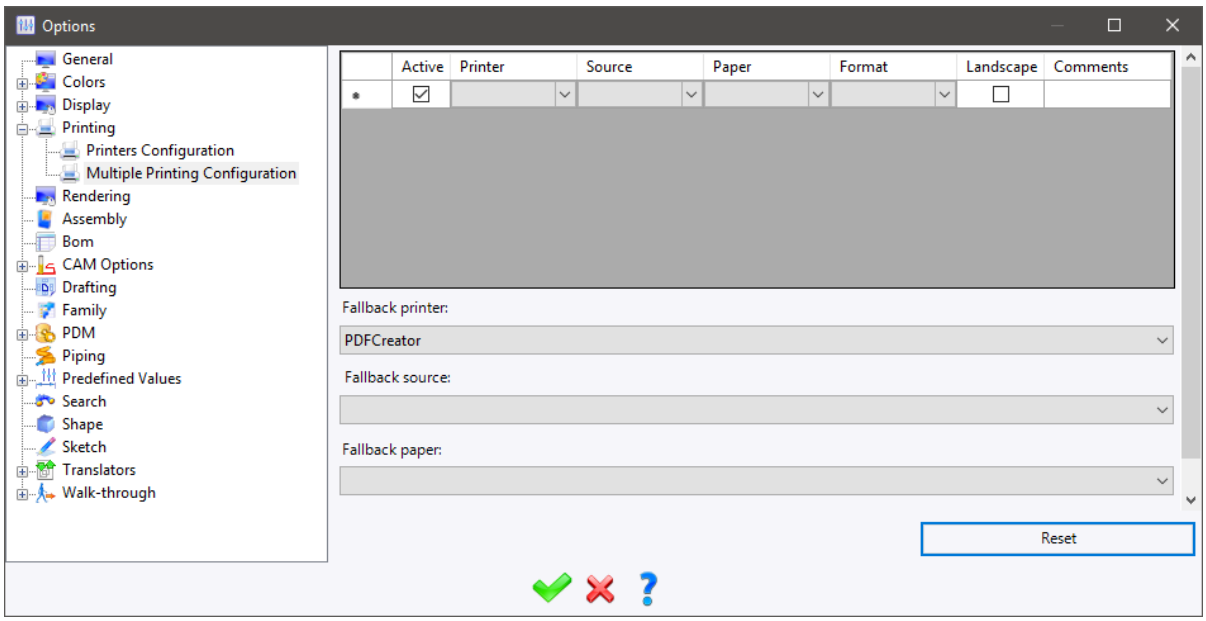


TopSolid creates a new document. You can now rename it, assign a reference to it, a material, etc.

Multiple printing configuration

TopSolid allows you to automate printing (on physical printer or PDF) on an entire project. However, automation certainly involves a bit of parameterization beforehand.

- From the **Tools** tab, select the **Options** command.
- Open the **Printing** node and select **Multiple Printing Configuration**.



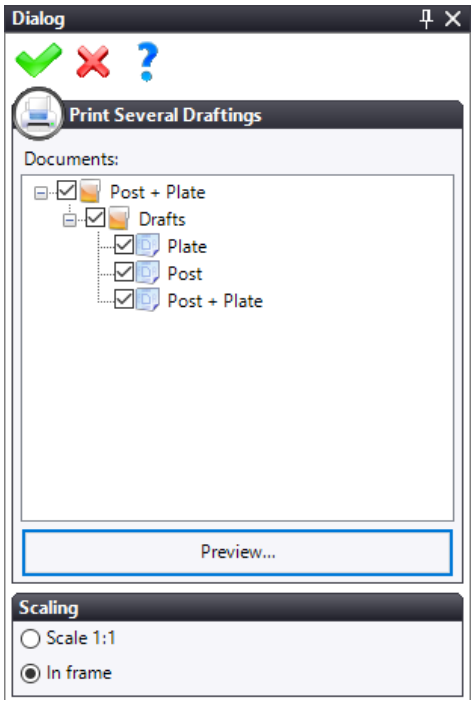
The upper table allows you to orientate the printings based on the format of the drafting. For example, A0 formats will automatically be sent to a plotter.

The **Printers Configuration** section allows you to define the thicknesses when printing.

- In the **Fallback printer** field, select **PDFCreator** and then click on to **confirm** the operation.

Note: The alternative to using a virtual PDF printer is to directly use the **TopSolid** PDF export as seen in the *Complete stage drafting* section.

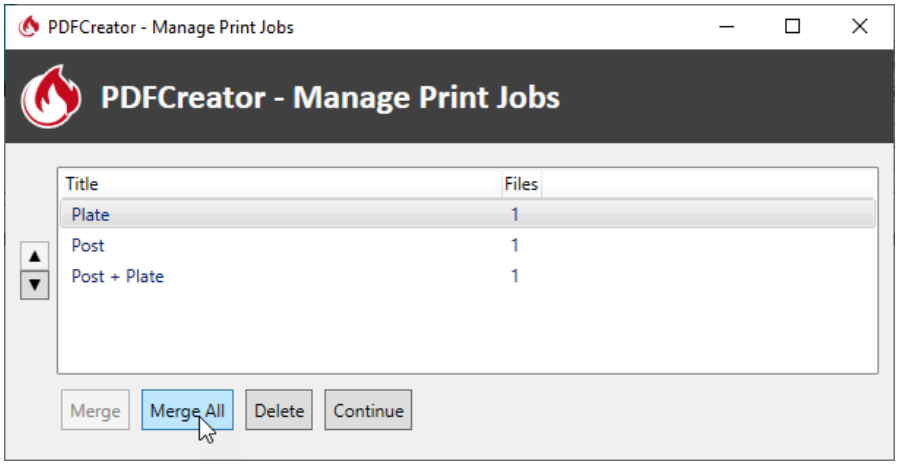
- Right-click on the *Post + Plate* folder and select the **Multiple Printing Configuration** > **Print Several Draftings** command.



- In the **Scaling** section, select the **In frame** option.
- Click on to **confirm** the operation.
- Click on **Merge**. This results in all the draftings being regrouped in a single PDF file.

The dialog box may vary from one version to another.

- Click on **Merge All**, then click on **Continue**.



- Select **Save**.

PDFCreator offers many options such as the possibility to add watermarks, modify the format of the generated file, modify the PDF compression level, etc.

Annex: Customization

Within the framework of the deployment of **TopSolid**, a customization/adaptation stage is necessary so that the product corresponds to the company's needs. We often see the following four stages:

- Customizing document templates:
  - Drafting
  - Bill of materials
  - Unfolding
  - Part
  - Assembly
- Creating a project template
- Creating standard libraries regrouping elements common to business (materials, profiles, company's standards, etc.)
- Customizing the interface (icons, keyboard shortcuts, colors, etc.)

We will cover almost all of these customization steps in this chapter, starting with document templates.

Document templates

Document templates can be stored in three different locations:

- In the projects: The templates will only be accessible in the project where they are located.
  This is only recommended for basic documents such as parts, assemblies and possibly unfoldings.
  If a document is updated, remember to update it in all the projects where the update can be useful.
- In **My Templates**: The templates will only be accessible by the user. For example, it is used in the case of a single-user installation (a **TopSolid** license in the company). These templates can be used in all projects by the user.
- In **Company Templates**: These templates are centralized and can be used in all projects by all users.

Summary:

- In the projects/**Templates of the project** → Parts, assemblies and possibly unfoldings templates.
- In **My Templates**/**Company Templates** → Bills of materials, draftings, drafting bundles templates, etc.

My templates/Company templates

Depending on the installation available in your company, it is advisable to use the right area for the following (**My templates** for single-user, **Company templates** for multi-user).

- Click on the  **TopSolid 7** icon and select the **File** > **Document templates** >  **Open My Templates** (or **Company templates**) command.
- Right-click on **My Templates (Documents)** and select the  **Folder** command.
- Create two folders named *Draftings* and *Bill of Materials*.

Customizing a drafting template

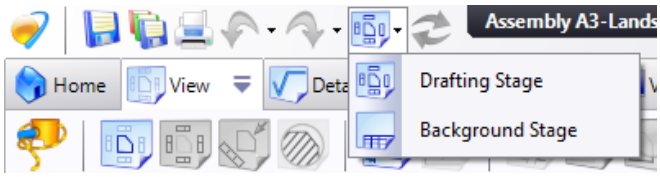
We will now customize a drafting template in your company's colors starting from a template delivered as standard.

- In the **My Templates/Company Templates** project, right-click on the *Draftings* folder and select the **Draftings** command.
- Select the **Assembly A3 ISO Landscape** template from the **Steel Standard Templates - United States** folder.

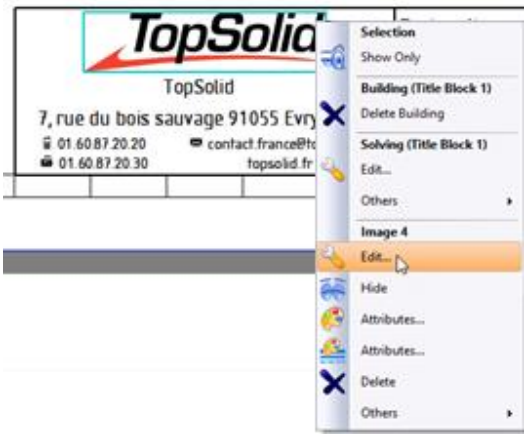
A drafting has two stages of creation:

- the background (creating the title block and the various parameters);
- the drafting (positioning the different views).

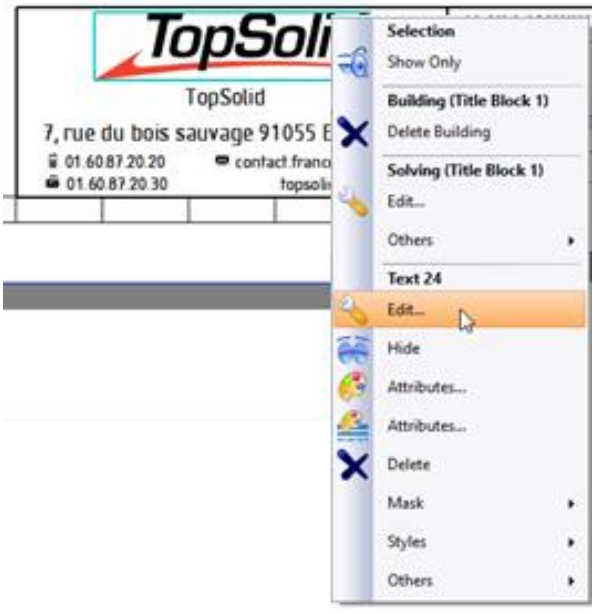
These stages can be seen here:



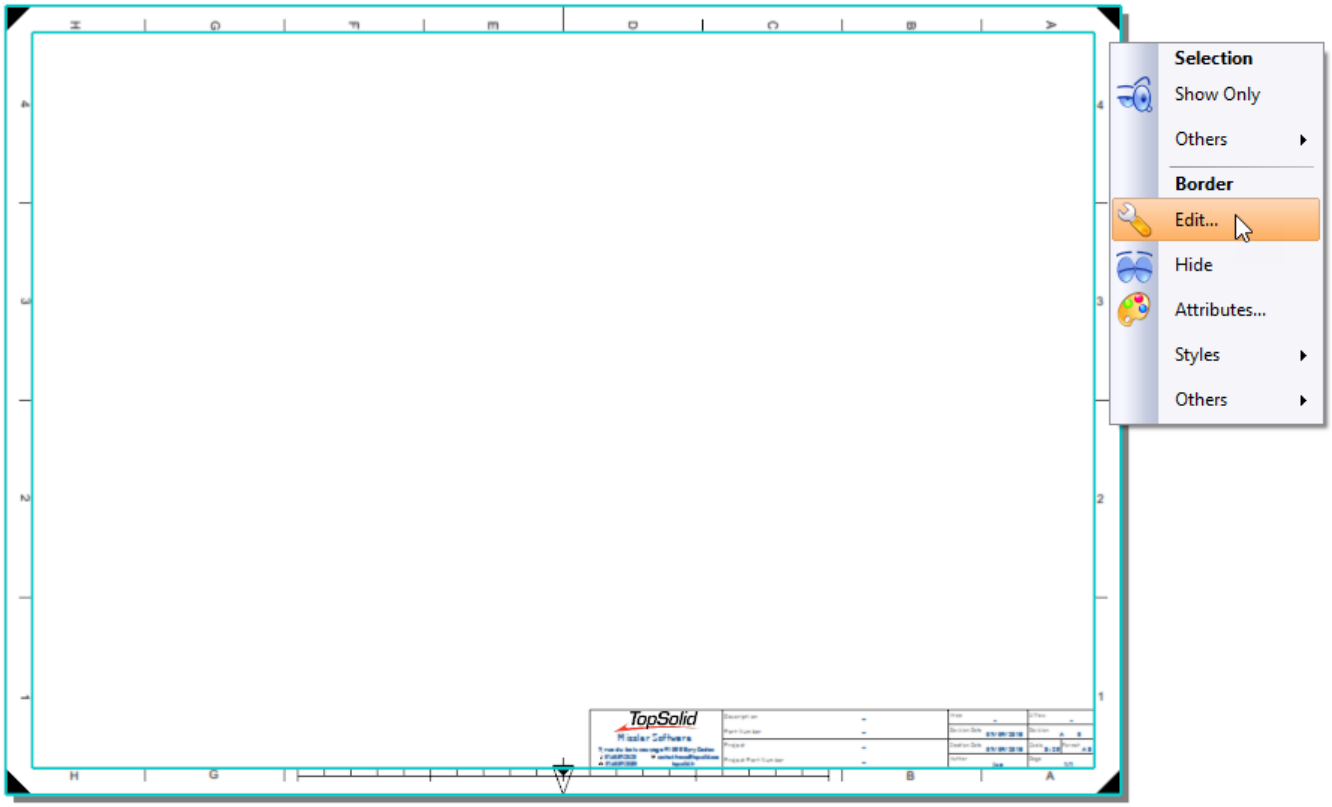
- Select the **background stage**.
- Edit the background logo. To do this, right-click on the logo and select the **Edit** command.



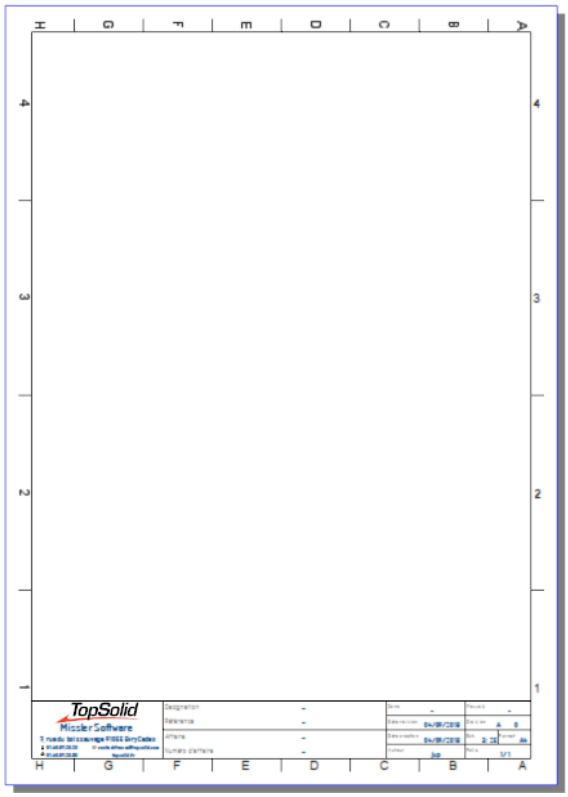
- Select your company logo.
- Edit all the texts based on your company. To do this, right-click on the text and select the **Edit** command.



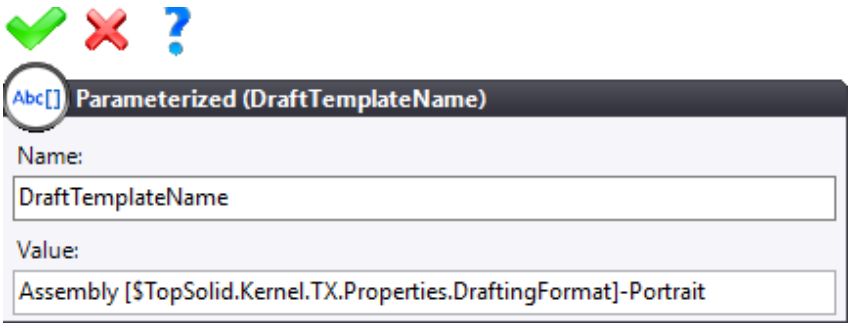
- Right-click on the drawing's frame and select the **Edit** command.



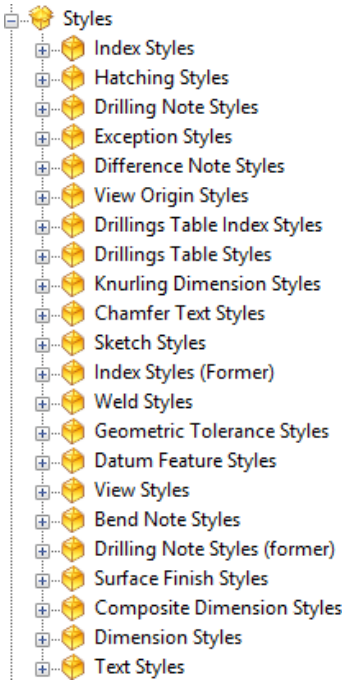
- In the **Predefined format** field, select **A4 ISO Portrait**. Uncheck **Cutting tabs**, **Centering Marks**, **Orientation Marks** and **Graduations**, then click on to **confirm** the operation.



- From the Operations tree or the Entities tree, open the **Parameters** > **Document Name Parameterization** folders and edit the **TemplateDraftingName** parameter.
- In the **Value** field, change *Landscape* to *Portrait* and click on to **confirm** the operation.

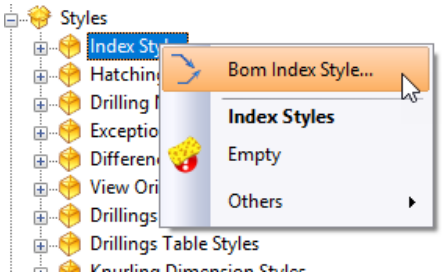


You can go further in the customization process using the styles options. The styles are specific to the document and exist in all types of document (part, assembly, drafting, etc.). They are directly available from the Entities tree.



Let us imagine that we want to create a new index style containing the description of the part, its material and thickness.

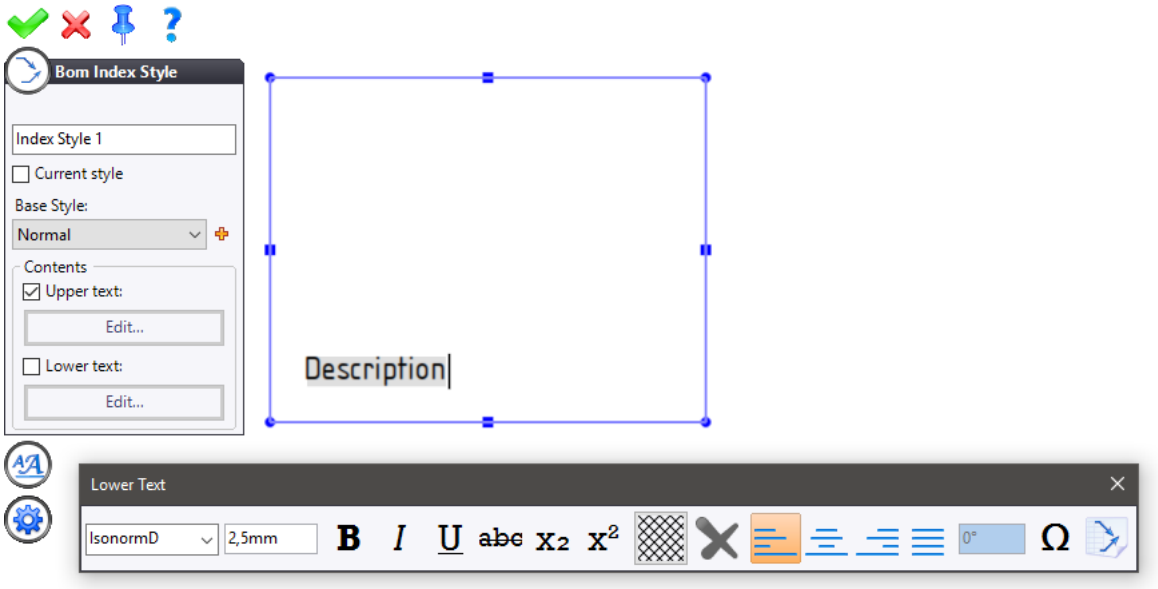
- Right-click on the **Index Styles** node and select the **Bom Index Style** command.



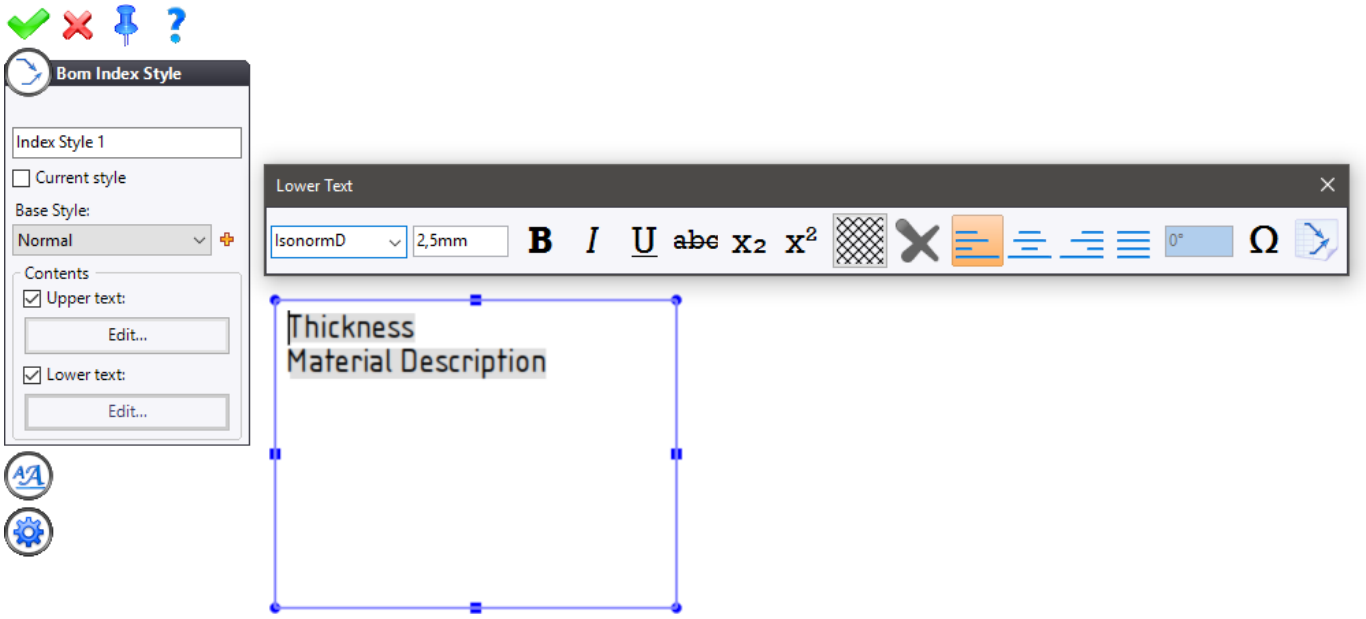
A new style is created. Each style has a base style (in this case, **Normal**). Checking an option in this dialog box means that you do not want to inherit the base style and therefore change it.

- Check the **Upper Text** box and click on the **Edit** button.
- Delete the word *Index* and click on the **Insert BOM Property** icon.

- In the dialog box that appears, select **Standard** > **General** > **Description** and click on to **confirm** the operation.



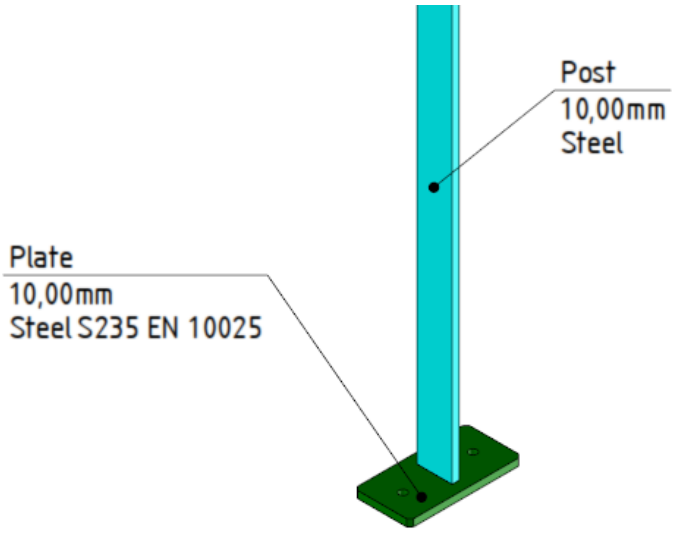
- Check the **Upper Text** box and click on the **Edit** button.
- Click on the **Insert BOM Property** icon.
- Select **Standard** > **Dimension** >**Thickness** and click on to **confirm** the operation.
- Click on to **confirm** the **Format** dialog box that appears.
- Using the **Enter** key, go to the next line to place the cursor under the *Thickness* text.
- Click on the **Insert BOM Property** icon.
- Select **Standard** > **Material** >**Material Description** and click on to **confirm** the operation.



- In the **Name** field of the style dialog box, enter *Desc_EpMat* and click on to **confirm** the operation.

The new index style is finished.

When used, this style will give the following result.



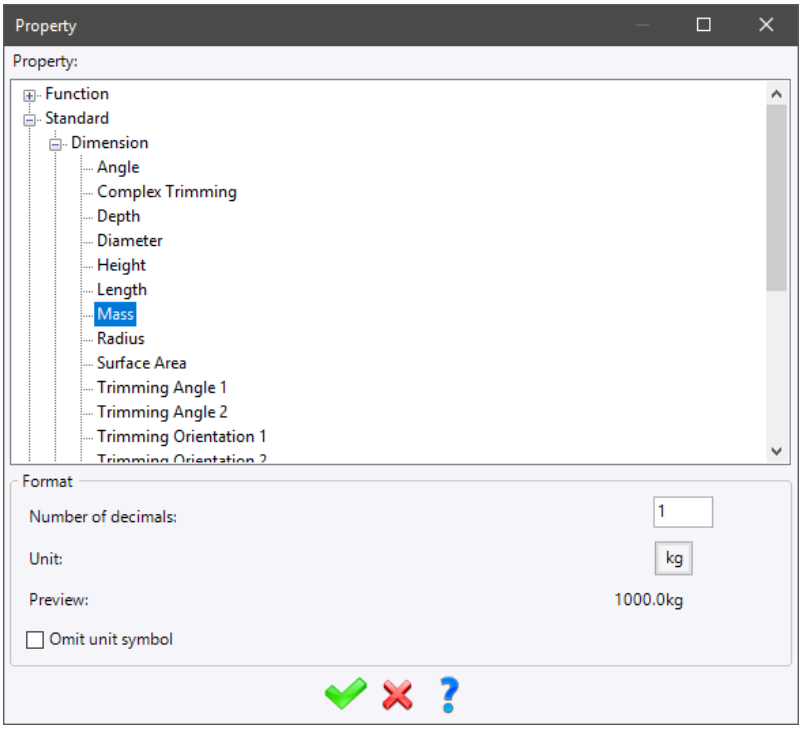
The customization of the drafting is complete.

- **Save** the drafting document, **close** it, then **check** it into the vault.

Customizing a BOM template

We will now customize a BOM template by working on the columns, the sort order and the precision of the values.

- In the **My Templates/Company Templates** project, right-click on the *Bill of Materials* folder and select the **Bill of Material** command.
- Select the **Sheet Metal** template from the **Steel Standard Templates - United States** folder.
- Right-click on the **HOLES NUMBER** column and select **Delete column**.
- Repeat the operation for the **PERIMETER** and **SURFACE AREA** columns.
- To do this, right-click on the last column and select **Add Column**. Select **Standard** > **Dimension** > **Mass**. At the bottom of the dialog box, enter *1* in the **Number of decimals** field, then click on to **confirm** the operation.



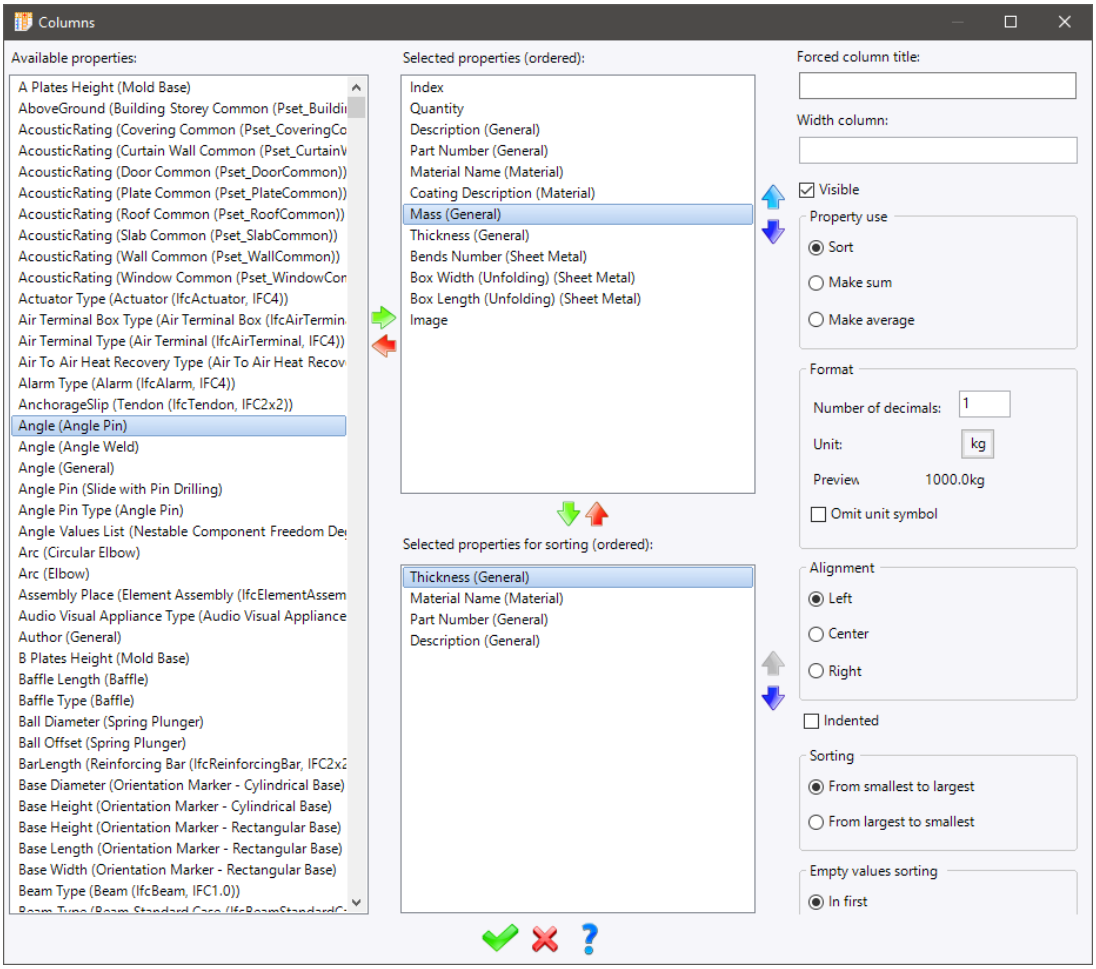
There is an advanced mode for editing the columns.

- From the **Bill of Material** tab, select the **Columns** command.

A dialog box appears. On the left side of the screen you can see all the properties available in a bill of materials. You can add them to the bill of materials using the green arrow (a property must be selected to see it) or delete them using the red arrow.

In the lower right frame, you will find the sort order. In this case, the sorting will be done first by material, then by thickness, by part number and finally by description. You can obviously change the sort order.

- In the bottom right frame, select the **Thickness** property. A light blue arrow appears on the right.
- Click on the arrow to move the property to the first position.



In the dialog box, it is also possible to force the title of a column, to adjust the precision of the values, to modify the order, to force the width of a column, etc.

- Click on to **confirm** the operation.
- **Save** the bill of materials document, **close** it, then **check** it into the vault.

Creating a project template

We have previously seen how to create document templates (drafting and bill of materials). A similar approach exists for a complete project.

You will have noticed throughout this training course that organizing documents in the project tree is important but repetitive. Therefore, it is advisable to integrate it when you are creating the project. This is what we will cover in the next exercise.

- Create a **new project** by clicking on the icon in the **Home** tab or by using the **Ctrl** + **Shift** + **N** keyboard shortcut.
- Rename the project *My project template*, select **Design Template** in the **Steel Templates** folder, then click on to **confirm** the operation.

At the beginning of the training, we addressed the subject of references, which is an essential element allowing you to use or delete libraries in your project.

A project that has too many references may overwhelm searches and the list of available extruded bars in the **Extruded bar** command.
Conversely, a project containing too few references will simply prevent you from accessing potentially useful components in your design.

- From the Project tree, open the **References** node, right-click on the **TopSolid DIN Mechanical** library and select the **Delete** command or press the **Delete** key directly on the keyboard.

All the components in this library are now inaccessible in this project.

- Right-click on the **References** node and select the **Reference Library** command.
- Using the **Ctrl** key, select the **TopSolid ANSI Mechanical** and **TopSolid DIN Mechanical** libraries and click on to **confirm** the operation.

Both libraries have been added to the references.

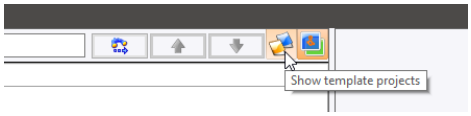
- Right-click on the project name and create the following **folders**:
  - *Unfoldings*
  - *Draftings*
  - *Bills of materials*
  - *Parts*
  - *Stringers*

Note: This part is completely free and can be adapted according to your needs.

The order is alphabetical in the Project tree. If you want to change the order, you simply have to add a number in front of the folder name (for example: *1 - Parts*, *2 - Sub-assemblies, etc.*).

- Right-click on the project name and **check** the entire project into the vault.
- Add this project to the project templates. To do this, right-click on the project name and select the **Others** > **Add to Templates** command. Depending on the installation available in your company, select **My Templates** or **Company Templates**.

This project can now be selected when creating a new project. If you wish to upgrade it, you simply have to open the project manager by clicking on the **Projects** icon in the **Home** tab and click on the **Show template projects** icon. The project templates will be visible and editable just like any other **TopSolid** project.



Customizing the interface

TopSolid's interface is very flexible. During the training course, we saw that trees can be placed anywhere on the screen, hidden, moved, etc. You can also display your favorite icons directly on the screen or to even add your own keyboard shortcuts.
Finally, all of these parameters can be exported to be saved or installed on another machine.

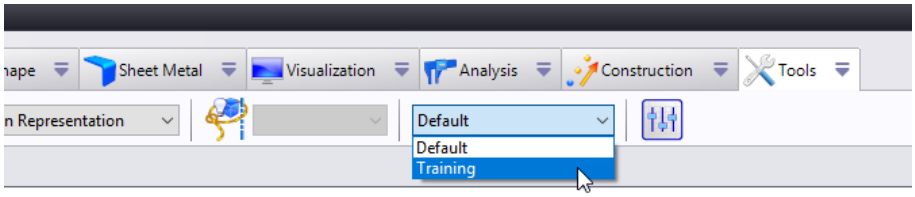
Menu creation

- Open or return to *My First Project*.
- Open or create a part (it does not matter what type).

Each type of document has its own tabs, icons and shortcuts that are integrated into the menus. Customizing the interface is therefore done document type by document type.

The first step is to create a new menu.

- From the **Tools** tab, select the **Menu** command.
- Click on the **Add** button, enter the name *Training* and click on twice to **confirm** the operation.
- From the **Tools** tab, select the previously created *Training* menu as shown below.

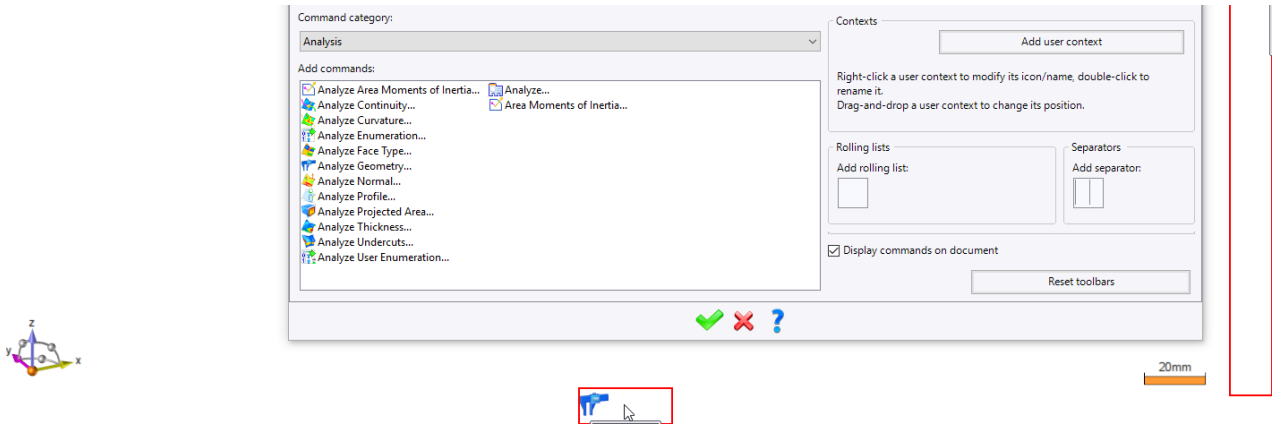


Adding shortcut icons and shortcut keys

- From the **Tools** tab, select the **Customize** command.

A dialog box appears and red areas appear on the screen. These symbolize the areas where commands can be added or deleted.

- From the **Command Category** drop-down menu, select **Analysis** and drag the **Analyze Geometry** command to the lower red area of the screen.

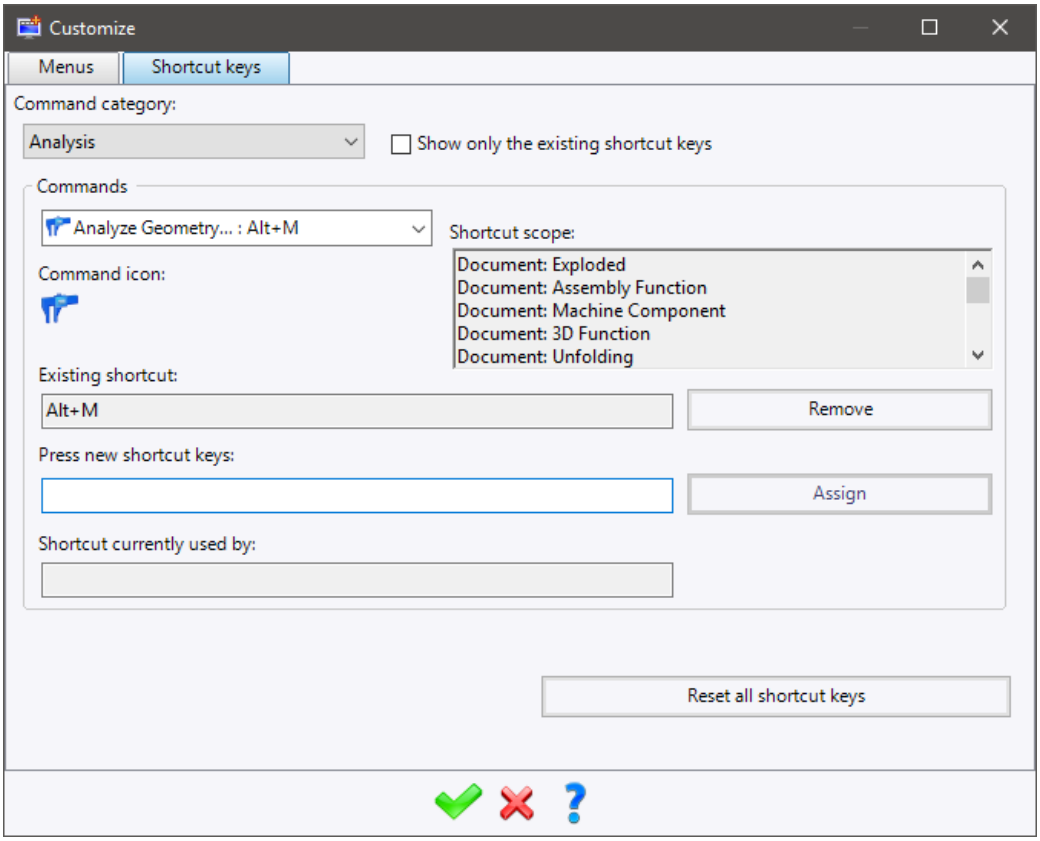


To delete a command, you simply have to drag it outside of an area.

Note: The **Add user context** button allows you to add a tab at the top of the screen. This is very useful to create a group of commands that you can use regularly.

- Click on the **Shortcut keys** tab, select the **Analysis** command category and select the **Analyze Geometry** command.

- In the **Press new shortcut keys** field, click **Alt** + **M**.
- Click on the **Assign** button to assign the keyboard shortcut.



- Click on to **confirm** the operation.

Saving and importing the settings

- From the **Tools** tab, select the **Manage Settings** command.
- Select the **Export application settings** option and click on the **Next** icon.
- Make sure that all the boxes are checked and then click on the **Next** icon again.
- Select a location to export the settings and possibly modify the default name.
- Click on to **confirm** the operation.

The import is done via the same command, as well as reinitializing the settings.

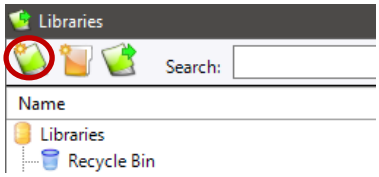
Introduction to libraries

TopSolid comes with a large number of standard element libraries. It is natural and advisable for a company to create its own libraries.

Typically, several libraries are created to avoid confusion. Here is a client example of a few libraries:

Here we will find company's standard (railings, fixings), commercial elements (Schüco, Wurth), standard elements (extruded bars), as well as a **Common** library.
The **Common** library regroups everything that will be common to libraries and projects. This can be filters (filtering of parts, assemblies according to several criteria), user properties (production mode, supply mode, sub-assembly type, etc.), functions, processes, etc.

- Open the Library Manager by clicking on the **Libraries** icon on the **Home** tab, then create a new library by clicking on the **New Library** icon.



- Rename the library *My Company - Common* and use a **blank template**.

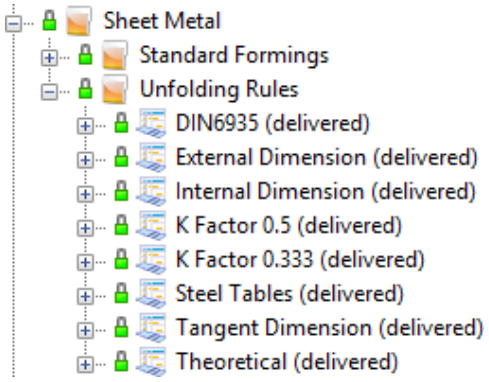
As you can see, a library is presented exactly like a project. Therefore, everything that was seen during this training course is applicable here.

Warning: This new library can only be used in your projects if it is referenced.

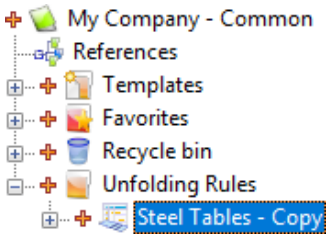
Using the unfolding rules for unfoldings

Now that the user library has been created, we will include an unfolding rule for the sheet metal. To avoid starting from a blank page, we will use an unfolding rule provided by **TopSolid** as a basis.

- Open the Library Manager by clicking on the **Libraries** icon in the **Home** tab and open the **TopSolid Mechanical** library.
- Open the *Sheet Metal* > *Unfolding Rules* folders and copy the *Steel Tables* document.

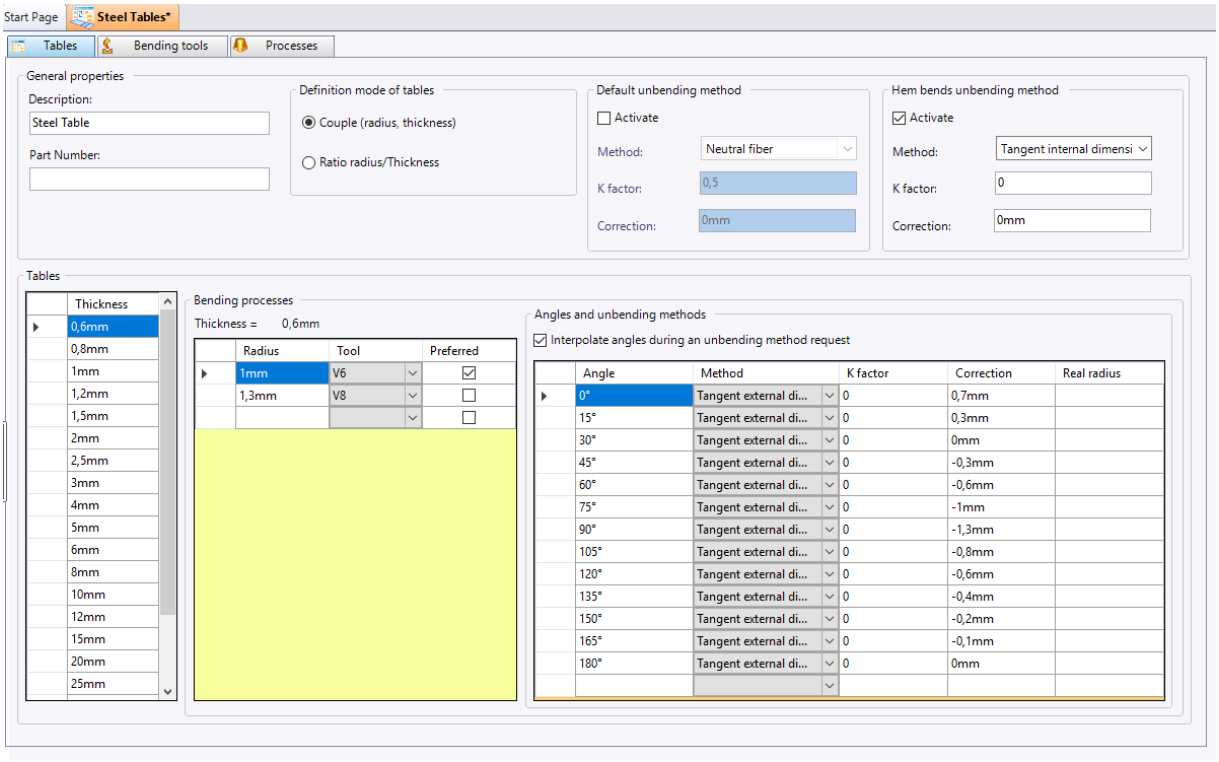


- In the *My Company - Common* library, create a new folder named *Unfolding Rules* and paste the previously copied steel table into it.



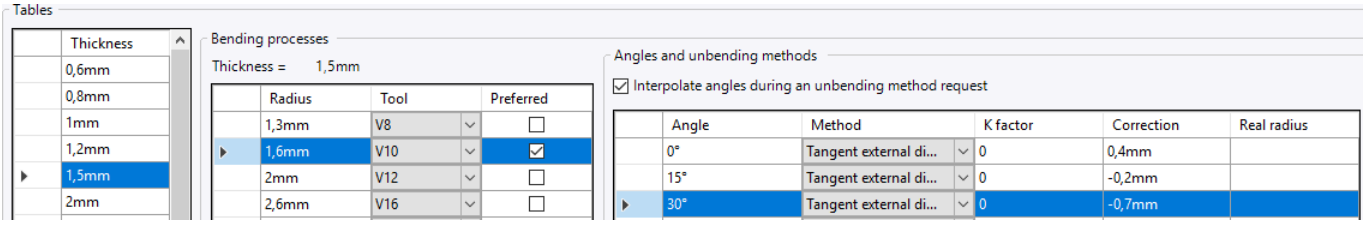
Note: You cannot edit a component and more generally, a library provided by **TopSolid**. The libraries are locked for modification so that they can be updated if necessary.
However, as you can see, they are not copy-protected. You can therefore copy or derive them according to your needs.

- Rename the document *Steel tables company* and open it.



There are two main methods of calculation:

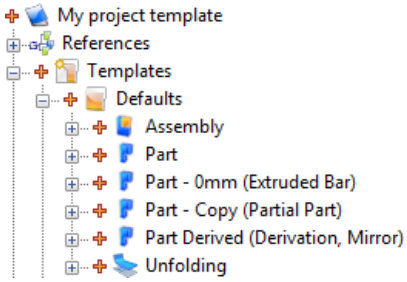
- **Thickness/Angle** couple: This is the current setting. The table reads from left to right: for a sheet metal thickness of 1.5mm, **TopSolid** will use a V of 10 by default which will produce a radius of 1.6mm. For a 30° angle, the correction will be -0.7mm.



- **Radius/Thickness** ratio: Coefficients and a K factor are entered and for each of them. For example, for a **radius/thickness** ratio of 1, the K factor is 0.27. For a **radius/thickness** ratio of 4, the K factor is 0.5.

- Make the modifications according to your needs.
- **Save** the unfolding rule document, **close** it, then **check** it into the vault.

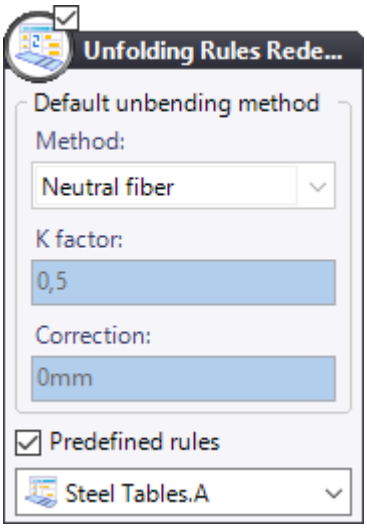
Note: To be able to use the unfolding rule in your default unfoldings, you have to edit the unfolding template in your project template. By default, it is located in the **Templates** > **Defaults** folders of your project template.



To assign the unfolding rule to an unfolding document, you simply have to proceed as follows.

- Reference the library. To do this, right-click on the **References** folder and select the **Reference Library** command.
- Open the *Unfolding* document.
- Drag the unfolding rule from your library into the graphics area.

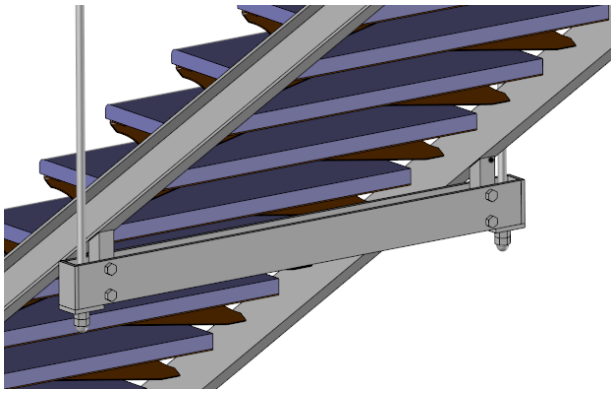
You can make sure that the unfolding rule has been taken into account by editing the **Part inclusion** operation from the Operations tree (if a message is displayed, confirm it). The **Unfolding rules redefinition** icon in the dialog box and the option **Predefined rules** must be checked and the rule must be selected.



- **Save** the unfolding document, **close** it, then **check** it into the vault.

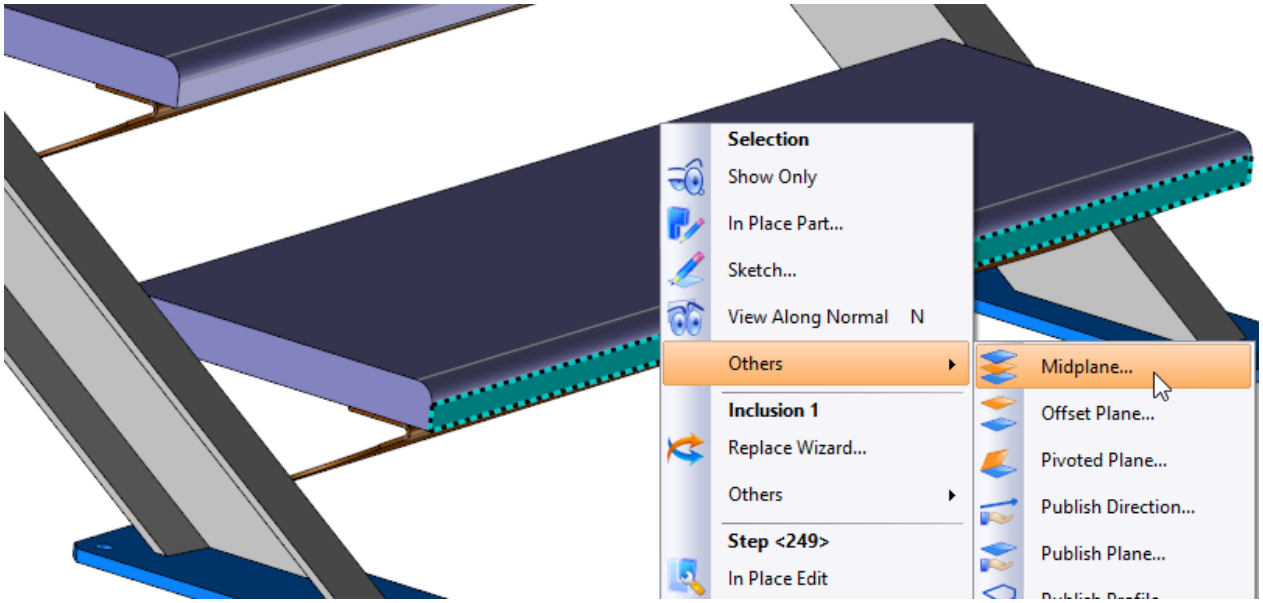
Annex: Creating the Central Support For the Right Stair

In this exercise, we will finish the design of the right stair previously created. The small section of the UPNs used to create the stringers require support in the center of the stair. This support should remain in the center regardless of the angle, backward movement or height of the stair.

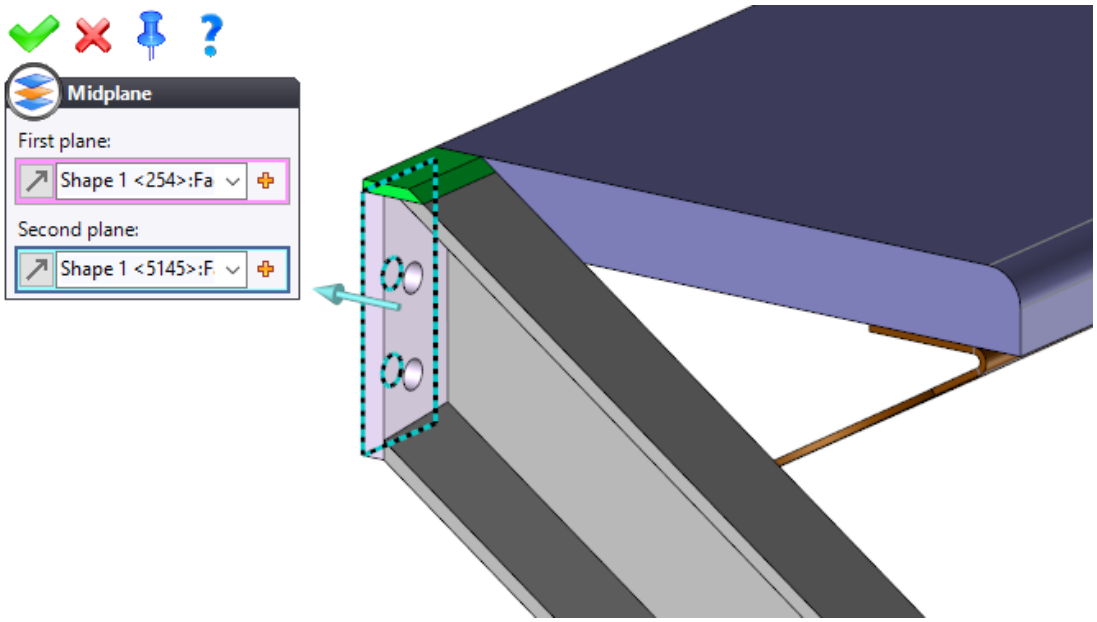


Creating the structure

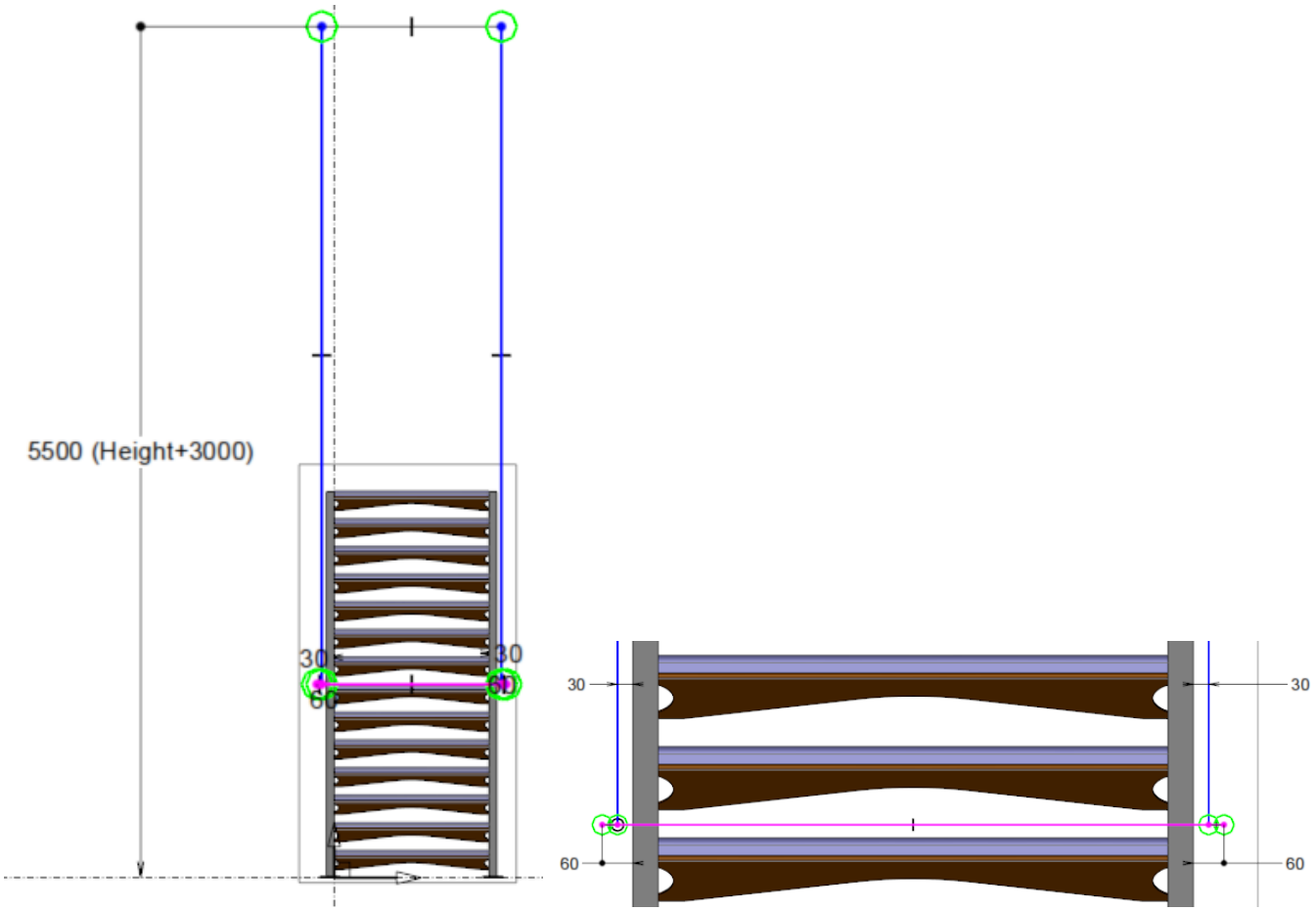
- Open or return to the *Right stair* assembly document.
- Right-click on the front face of the first step and select the **Others** (in the **Selection** section) > **Midplane** command.



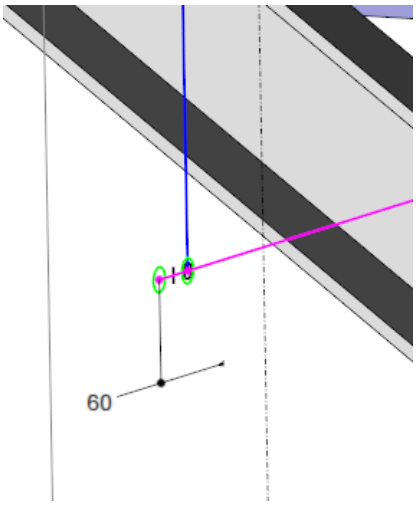
- Select the face as shown below of the up plate as the **second plane**.



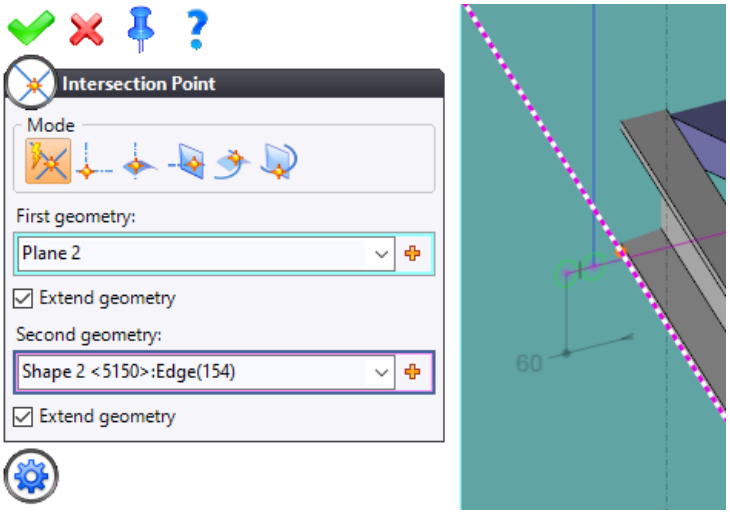
- Right-click on the previously created plane and select the **Sketch** command.
- Draw the sketch as shown below.



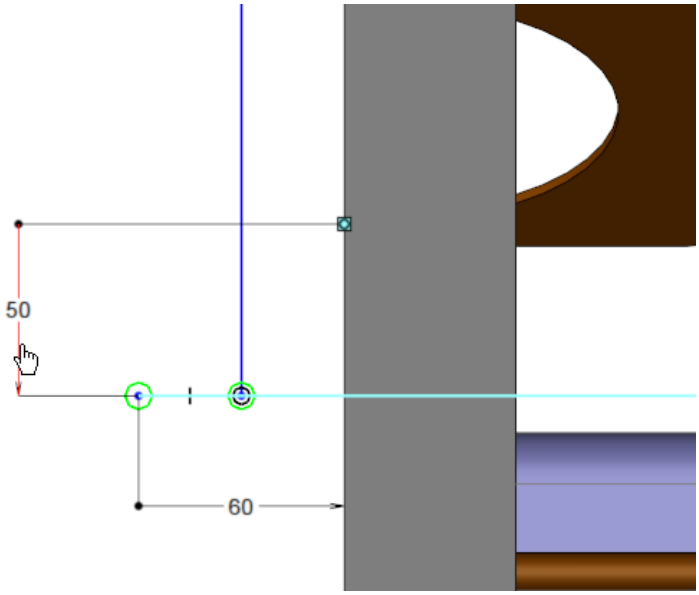
As you can see, the position of the support is complicated to dimension because it depends on the stringers and the position of the plane.



- From the **Construction** tab, select the **Intersection Point** command still from the sketch context.
- Select the sketch plane as the **first geometry** and the low outer edge of the stringer as the **second geometry**.

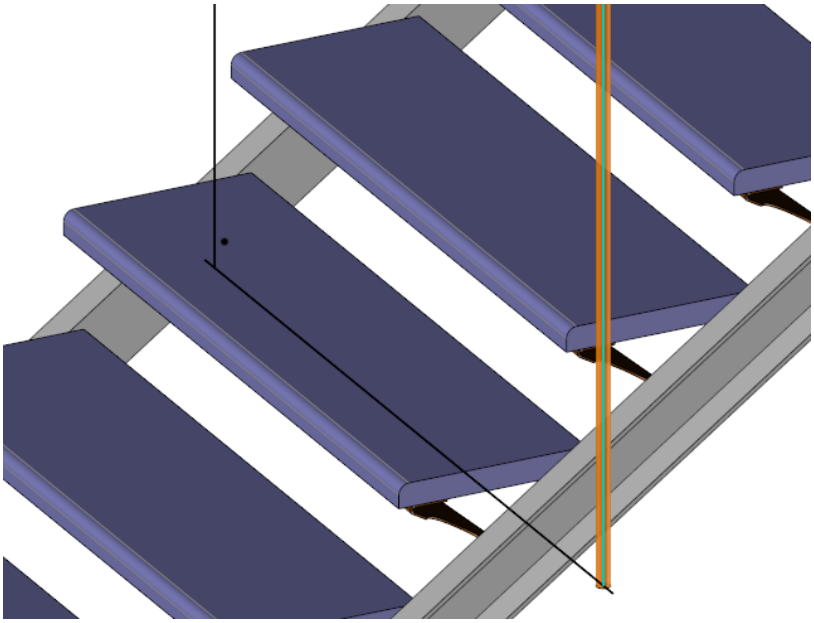


- Complete the sketch by adding the dimension as shown below.

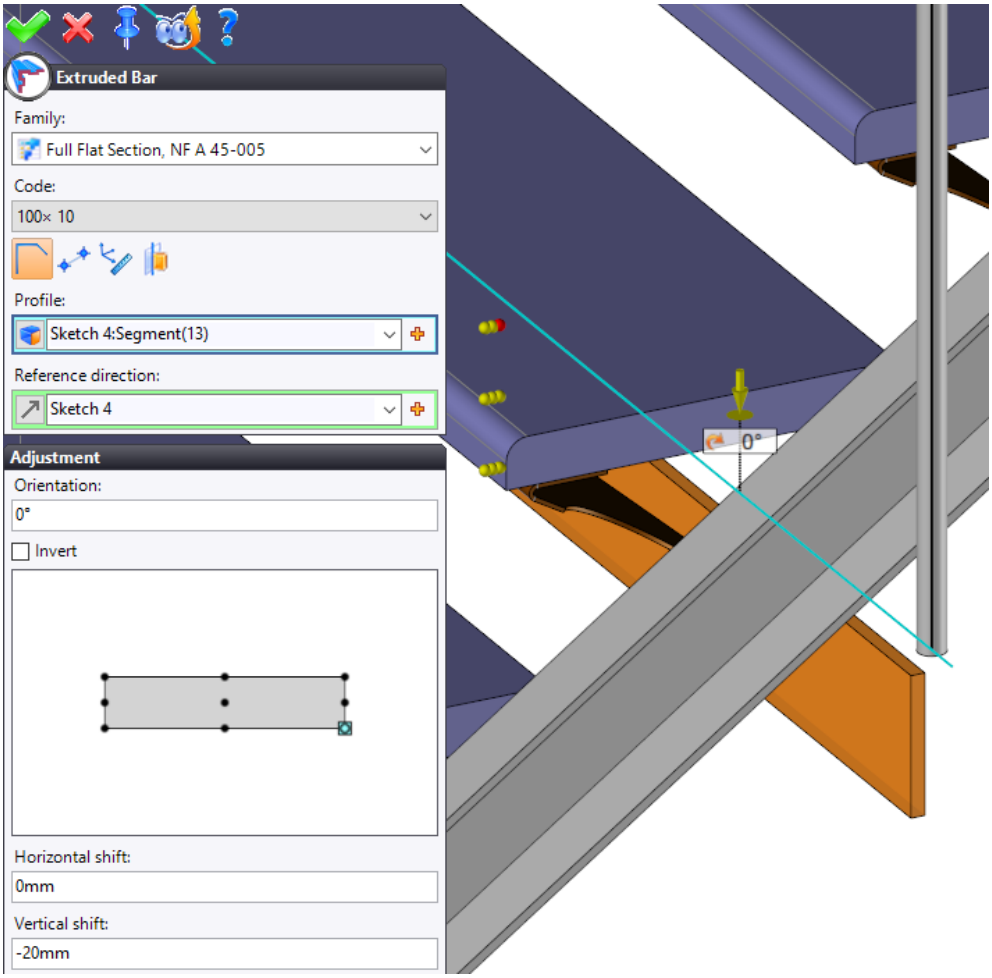


- **Confirm** the sketch.

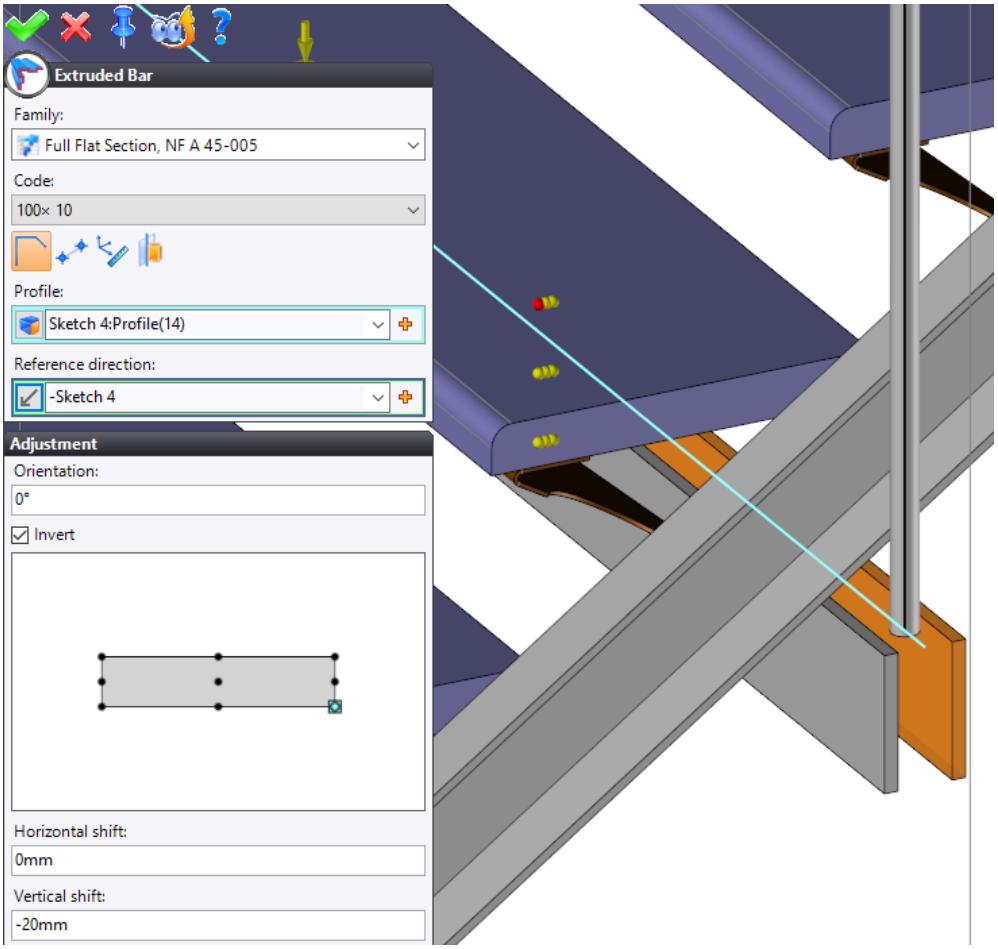
- On one of the vertical lines, position a **full circular section NF A 45-003** with the **20** code.



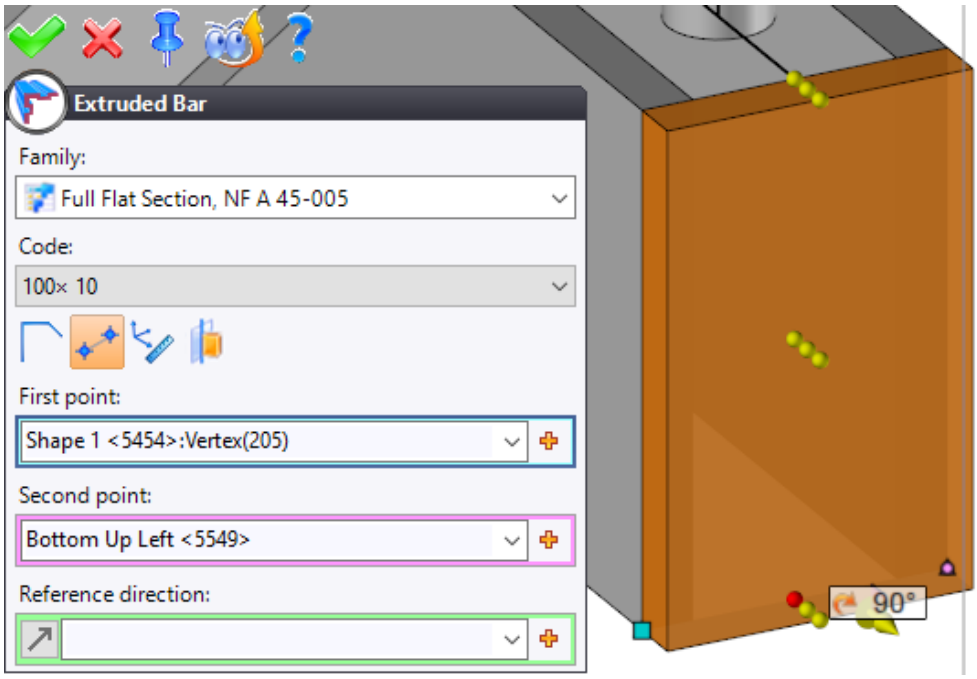
- On the horizontal segment below, add a first **full flat section NF A 45-005** with the **100 x 10** code and a **vertical shift** of *-20mm*.



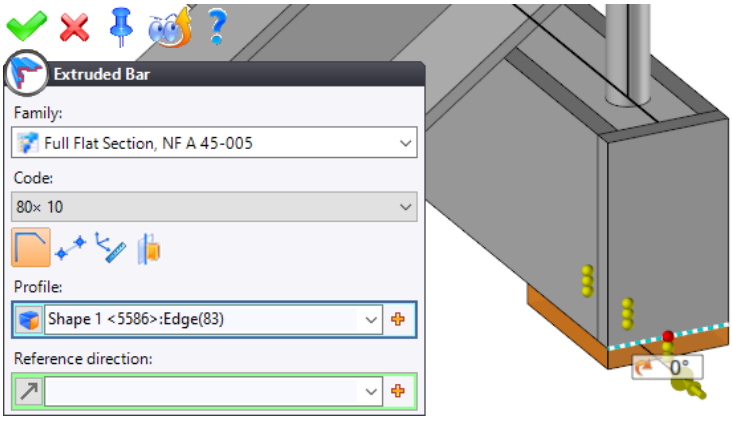
- Add the second **full flat section NF A 45-005** with the **100 x 10** code and a **vertical shift** of *-20mm*. To do this, select the line again, click once on the **Reference direction** arrow and check the **Invert** box.



- Add a **full flat section NF A 45-005** with the **100 x 10** code in **Two points** mode.



- Add a last **full flat section NF A 45-005** with the **80 x 10** code.

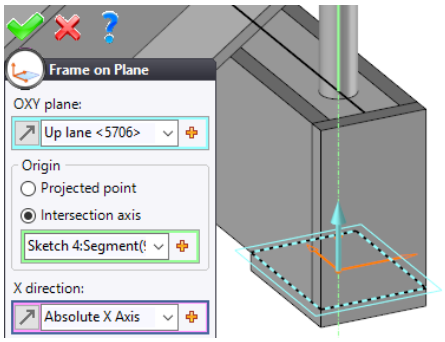


Adding fixings

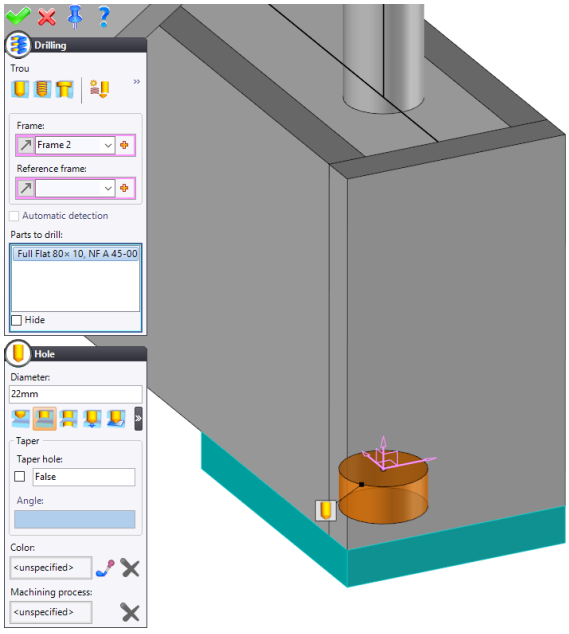
- From the **Modeling** tab, select the **Drilling** command.

The goal is to drill the plate in line with the full circular section. To facilitate this operation, we will create a frame.

- In the **Frame** field, click on the icon and select the **Frame on Plane**.
- Select the face as shown below as the **OXY plane** and the axis of the full circular section as the **intersection axis**.

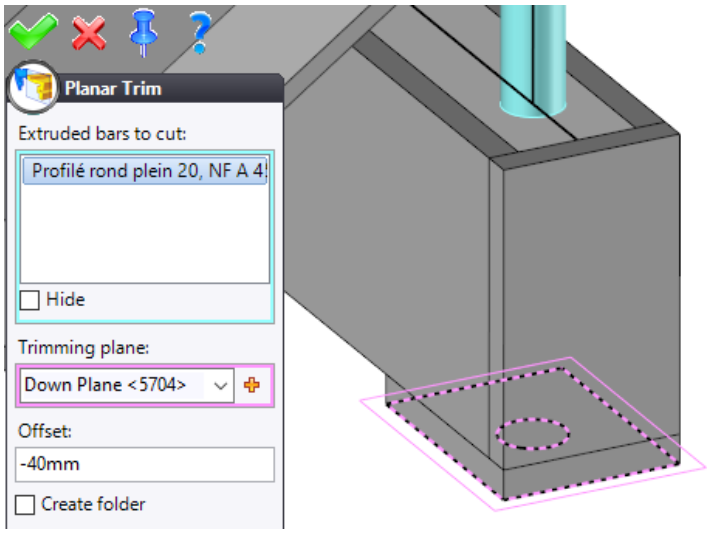


- Click on to **confirm** the frame.
- Select the full flat 80 x 10 section as the **parts to drill** and enter a **diameter** of *22mm*.



- Click on to **confirm** the drilling.

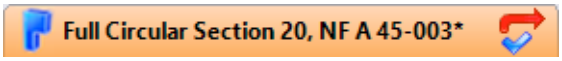
- Right-click on the circular extruded bar and select the **Planar Trim** command.
- Select the face as shown below and enter an **offset** of *-40mm*.



- Double-click on the circular extruded bar to edit the part in place.

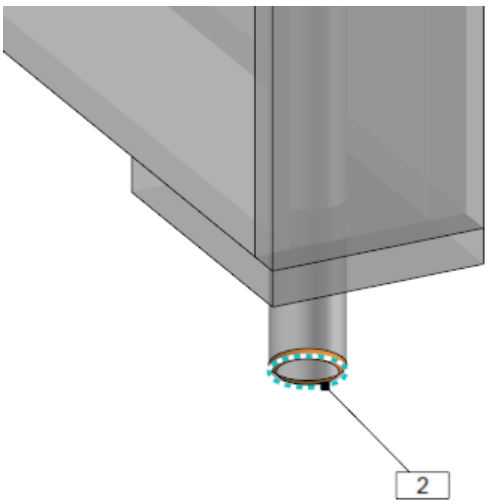
TopSolid displays the extruded as it is inserted; the other extruded bars and the extension are not visible.

- To correct this, click on the **End inserting** icon in the document's tab.

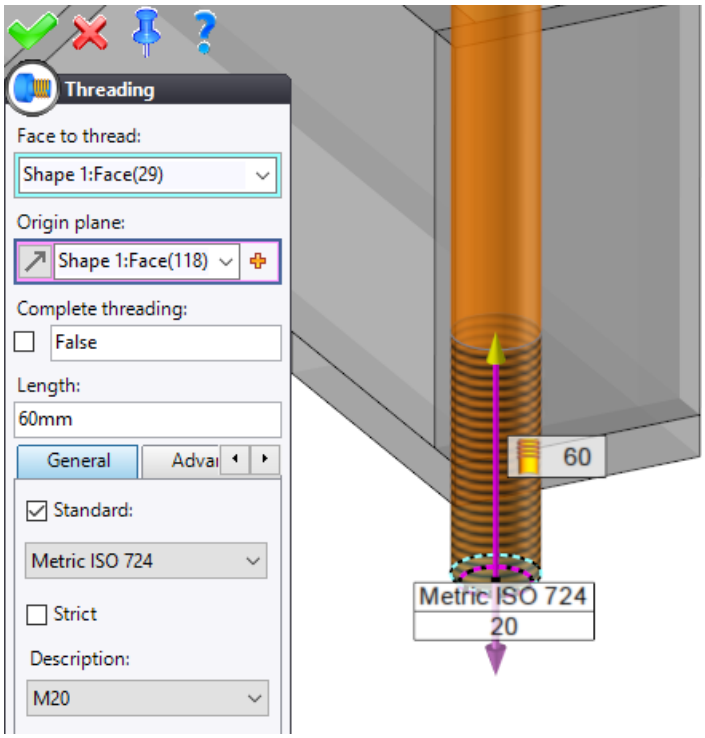


Note: To edit this operation by default, you simply have to select the **Tools** > **Options** > **Assembly** command and check the **Show posterior context when editing in-place part**. This mode of operation is better suited to the metalwork and locksmith trades.

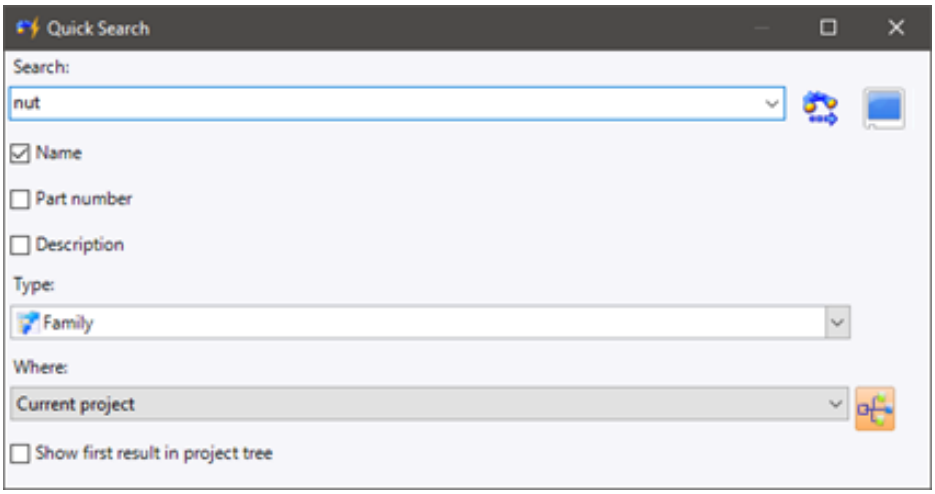
- Right-click on the edge as shown below and select the **Chamfer** command.
- Enter a value of *2mm*.



- From the **Shape** tab, select the **Threading** command.
- Select the face of the circular extruded bar as the **face to thread**, enter a **length** of *60mm* and select the **Metric ISO 724** standard.

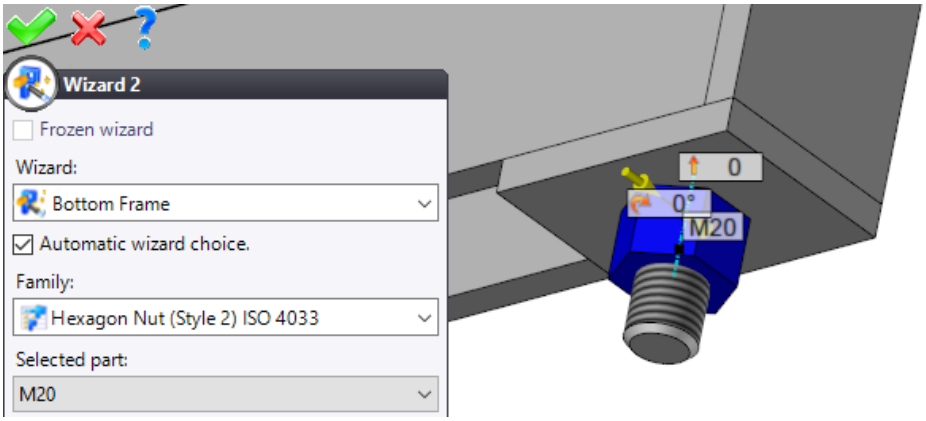


- Click on to **confirm** the operation.
- **Confirm** the in-place editing context.
- At the top right of the screen, click on the **Quick Search** icon.
- Adjust the settings as shown below.

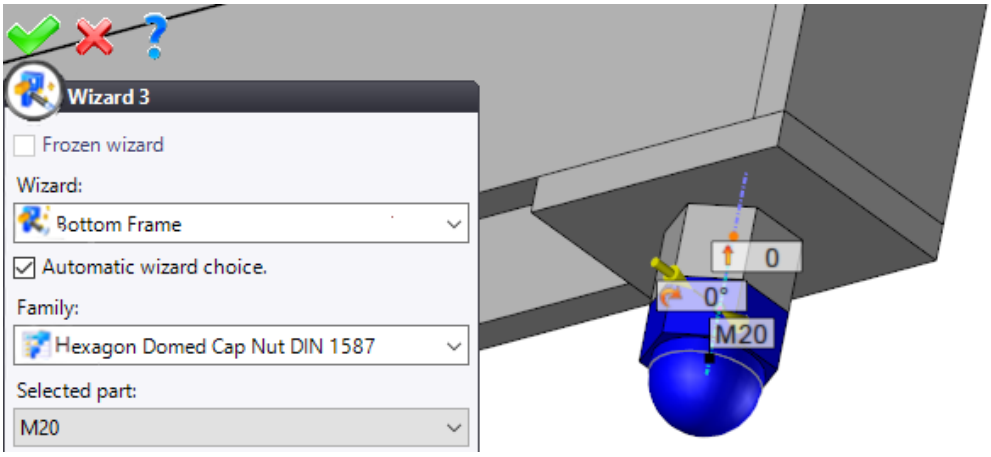


- Click on the icon to launch the search. **TopSolid** displays the search results.
- **Close** the quick search dialog box.

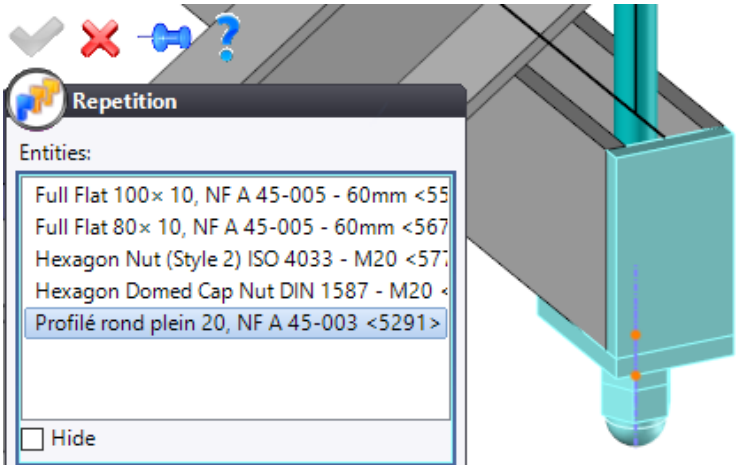
- Add a **hexagonal nut (style 2) ISO 4033**.
- Select the **M20** code.



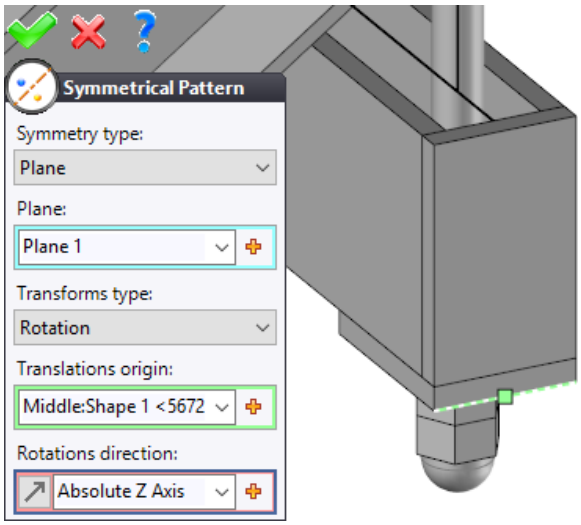
- Position the nut. To do this, you simply have to bring the cursor near the drilling. **TopSolid** will automatically hook onto it.
- Similarly, add a **hexagon domed cap nut DIN 1587** with the **M20** code.



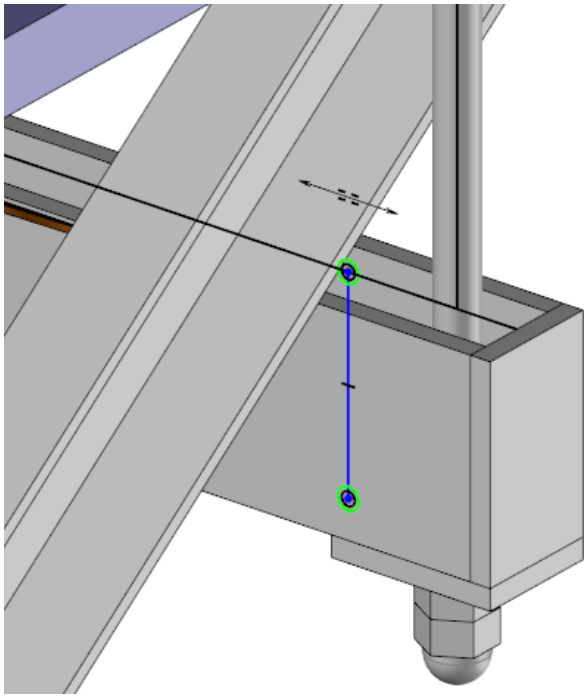
- From the **Construction** tab, select the **Repetition** command.
- Select the elements as shown below as the **entities** to be repeated.

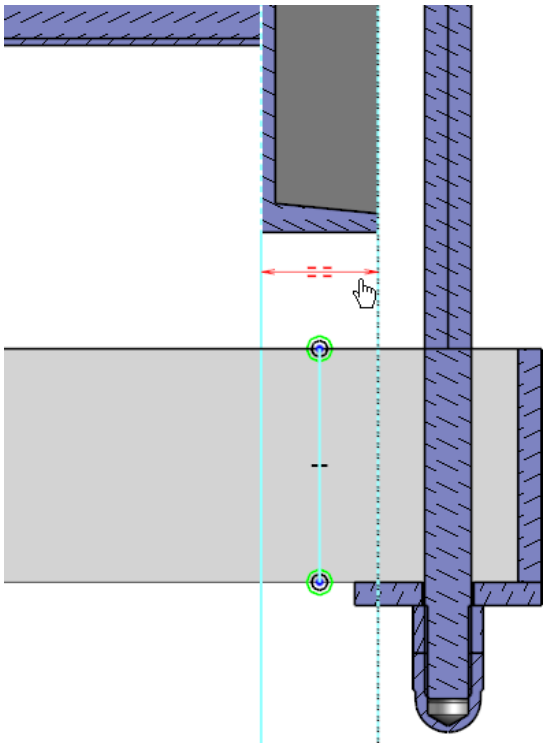


- In the **Pattern** field, create a **symmetrical pattern**.
- Select the central plane of the stair (**Plane 1** normally) as the **plane**.
- Select **Rotation** as the **transforms type**. Select a point as shown below as the **translations origin** and select **Absolute Z Axis** as the **rotations direction**.

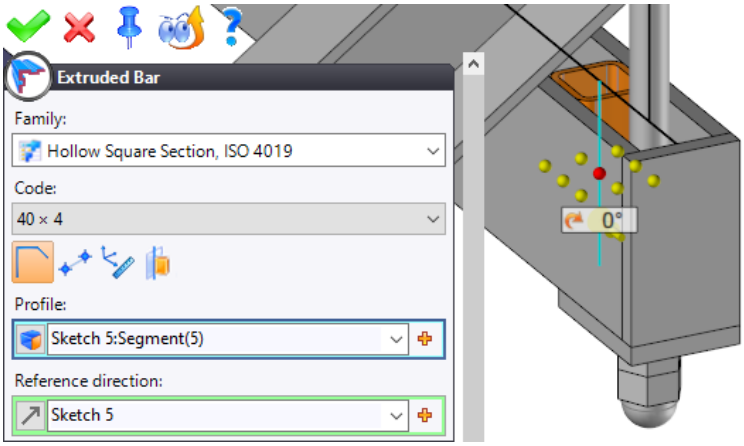


- Click on to **confirm** the pattern.
- Click on to **confirm** the repetition.
- Create the following sketch on the central support plane. Use the **Centering** constraint.

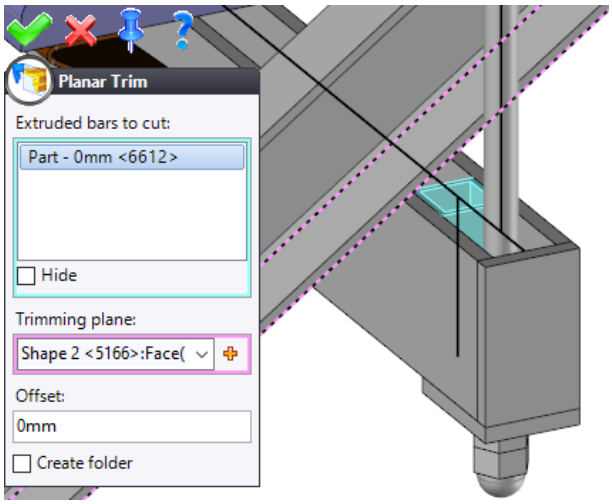




- Add a **hollow square section, ISO 4019** with the **40 x 4** code as shown below.

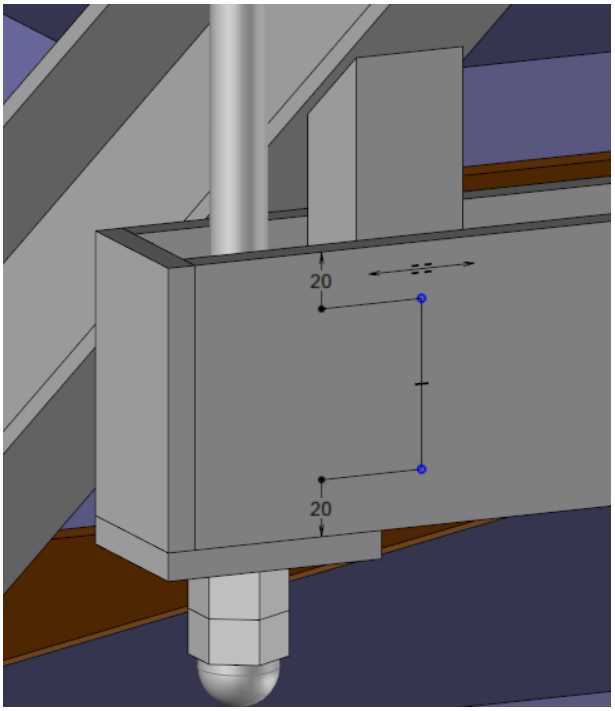


- Perform a **planar trim** operation by selecting the bottom of the stringer as the **trimming plane**.

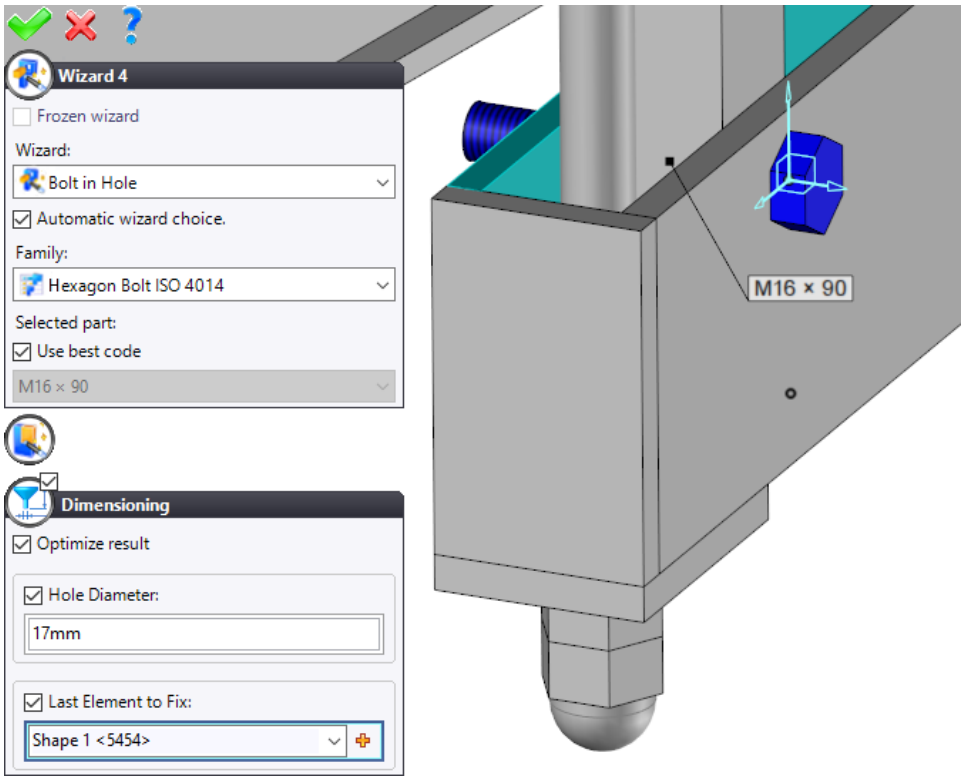


We will now add the fixings.

- Draw the following sketch.

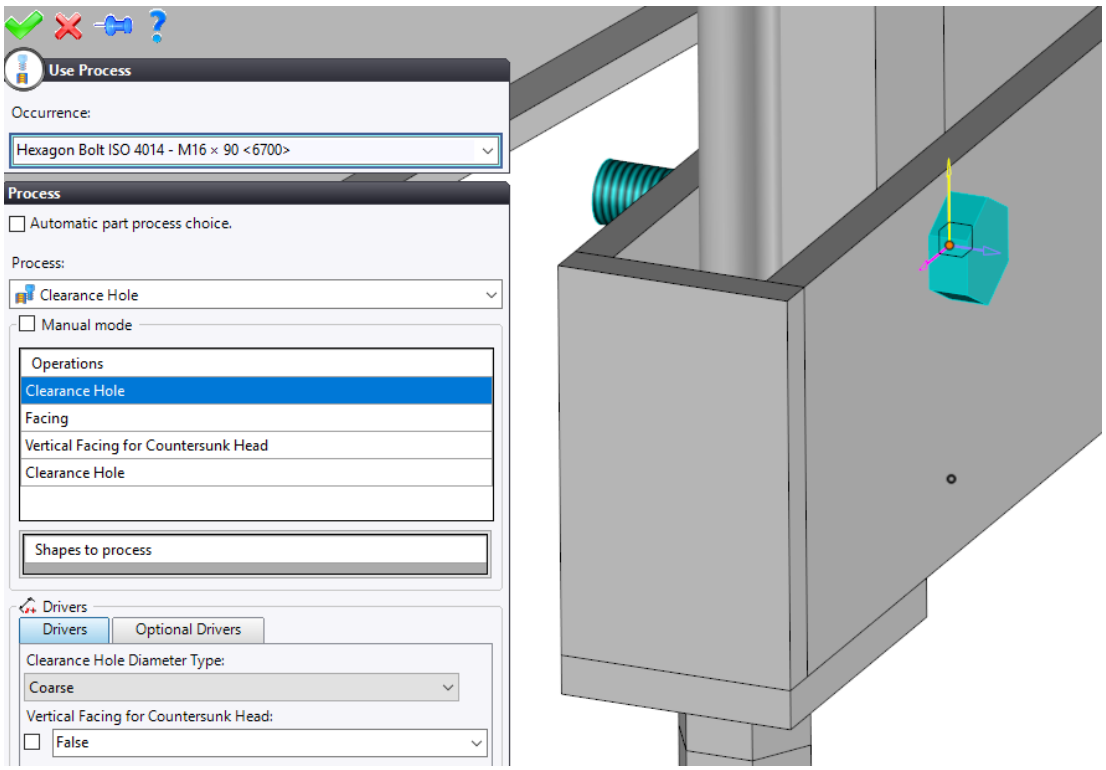


- Use the **quick search** to find a **hexagon screw ISO 4017**.
- Add and position the screw. To do this, create a **frame on plane** using the top point of the sketch.
- Select the **Bolt in Hole** wizard. Enable the **Dimensioning** icon in the dialog box, enter a **hole diameter** of *17mm*, then select the flat as shown below as the **last element to fix**.

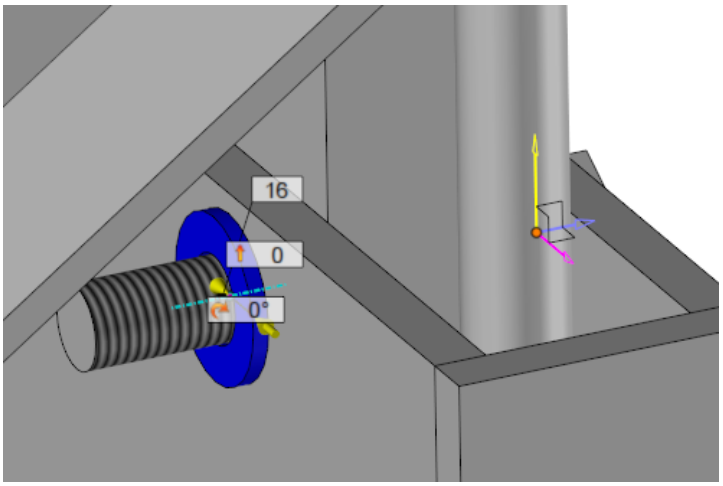


TopSolid looks for the right dimension in the family to tighten the different parts and add a nut.

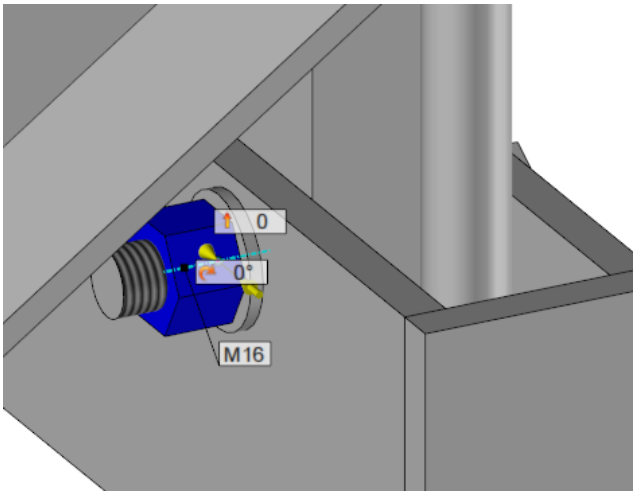
- In the **Use Process** dialog box, select the **Clearance Hole** process, then select **Coarse** from the **Clearance Hole Diameter Type** drop-down menu.



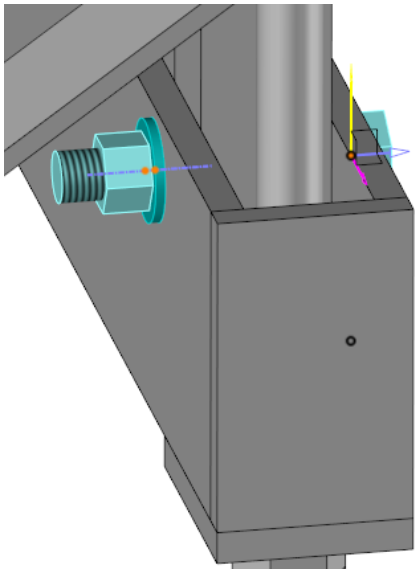
- Add a **flat washer - M, NF E 25-513** with the **16** code.



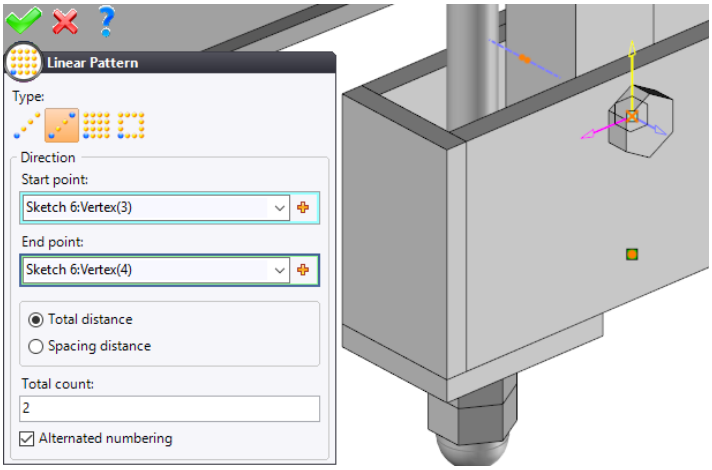
- Add a **hexagonal nut (style 2) ISO 4033** with code **16**.



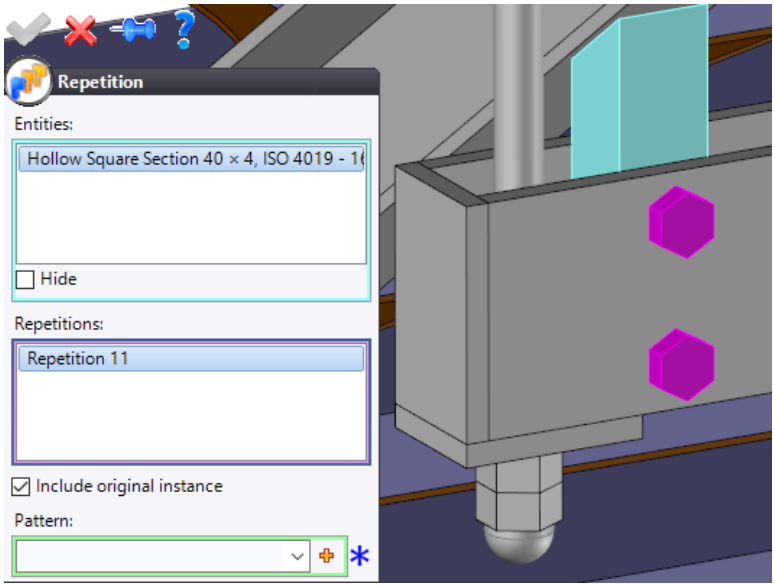
- From the **Construction** tab, select the **Repetition** command.
- Select the screw, washer and nut.



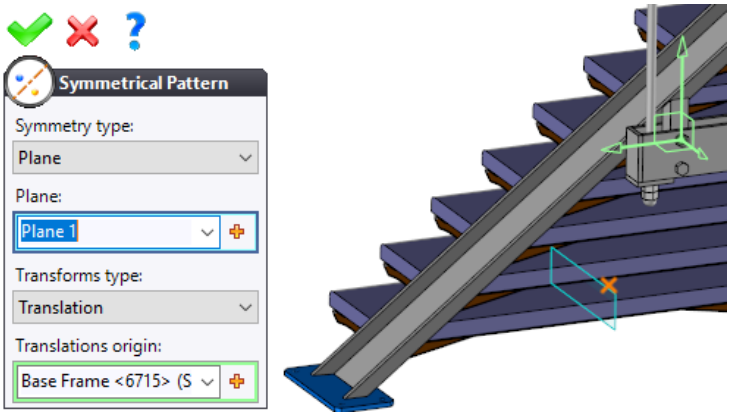
- Create a **linear pattern** in **Line by two points** mode.
- Select the two points of the sketch as shown below.



- Click on to **confirm** the pattern.
- Click on to **confirm** the repetition.
- From the **Construction** tab, select the **Repetition** command.
- Select the small section as the **entities** and select the bolt as the **repetition**.

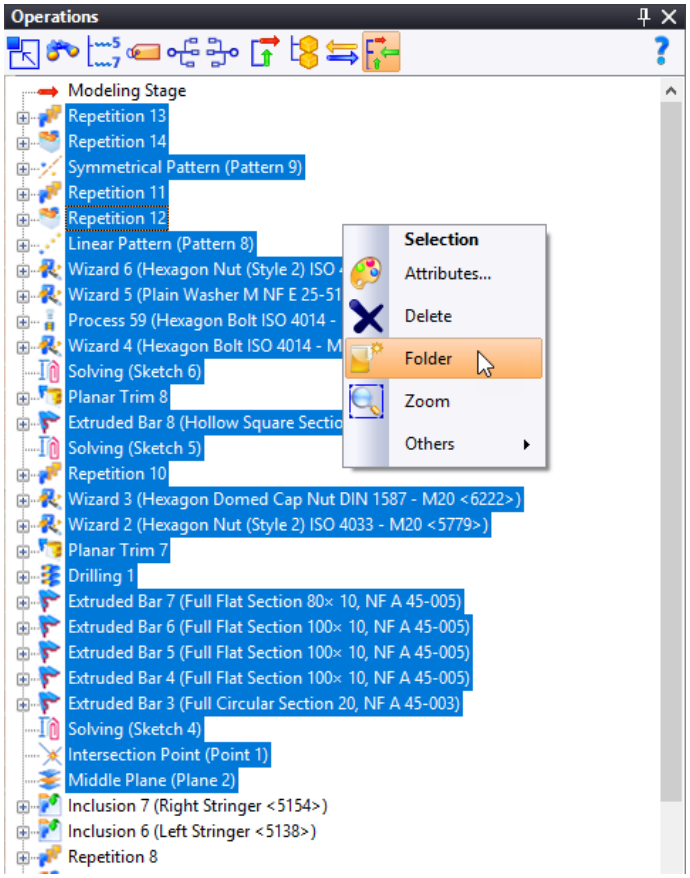


- Create a **symmetrical pattern**. Select the **Plane** mode and select the symmetry plane of the stair. Select **Translation** as the **transforms type**. Select one of the two points of the sketch as the **translations origin**.



Organizing the Operations tree

- In the Operations tree, identify the first operation that corresponds to the creation of the central support (the second middle plane normally, after the inclusion of the right stringer).
- Select all the operations above, right-click and select the **Folder** command.

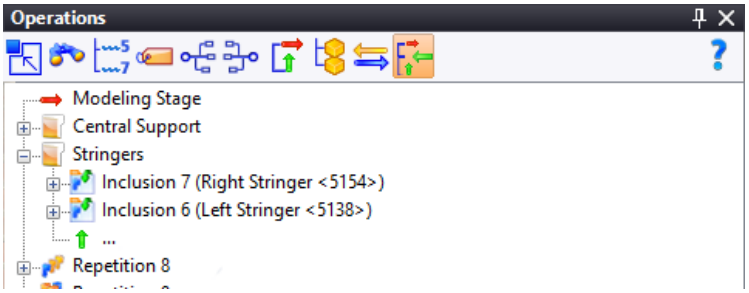


- Rename the folder *Central Support*.

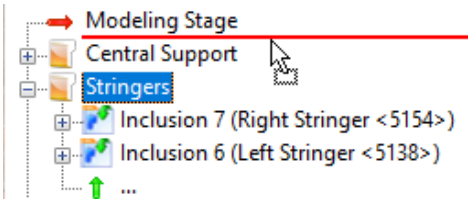
Error management

We have seen previously that the sub-assemblies should be created at the end of the design as far as possible. In our case, the small added section belongs to the stringers; therefore, it must be in the stringers sub-assembly. Editing the operation will not be sufficient since the creation of the left and right stringer assemblies was done chronologically before the central support.

- In the Operations tree, create a folder that contains the two inclusion operations for the right and left stringers.

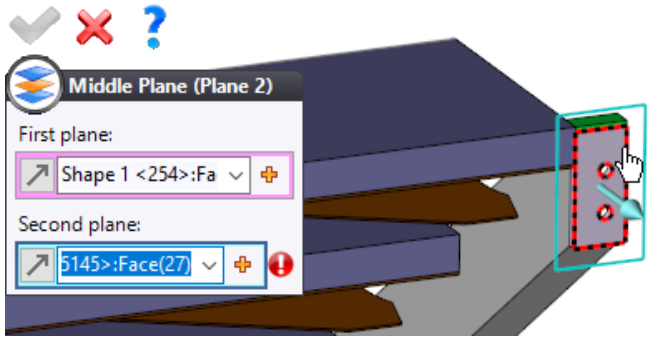


- Drag this folder over the *Central Support* folder.



TopSolid displays an error message and this icon appeared on the document's tab. This means that one or more operations are in error. The operation in error is displayed in red in the Operations tree.

- Click on **OK** to **close** the error message.
- Click on the icon on the document's tab. **TopSolid** directly opens the operation in error and identifies the precise origin of the problem. In our case, we had used a plane of the right stringer sub-assembly except that it no longer exists at this stage.
- Click in the **Second plane** field and select the plane as shown below again.

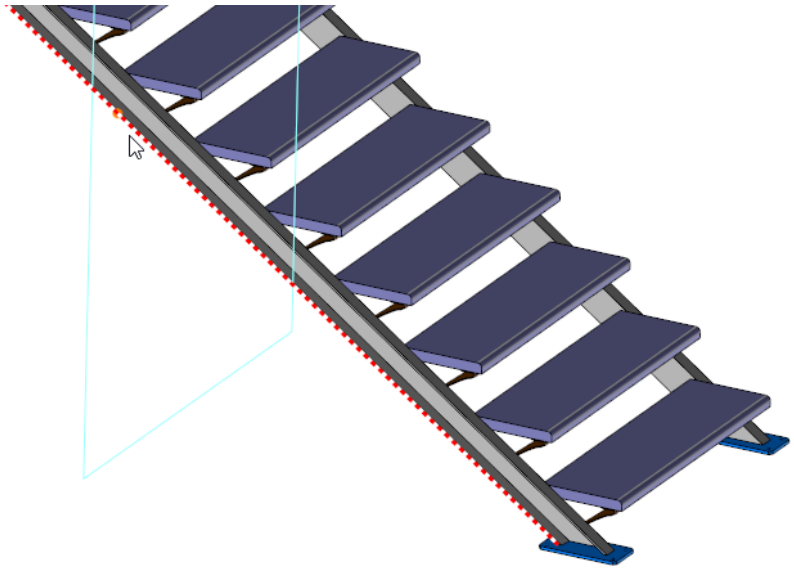


- Click on to **confirm** the operation.

A new error message appears.

- Click on **OK** to **close** the error message.
- Click on the icon on the document's tab. The intersection point was also based on the sub-assembly.

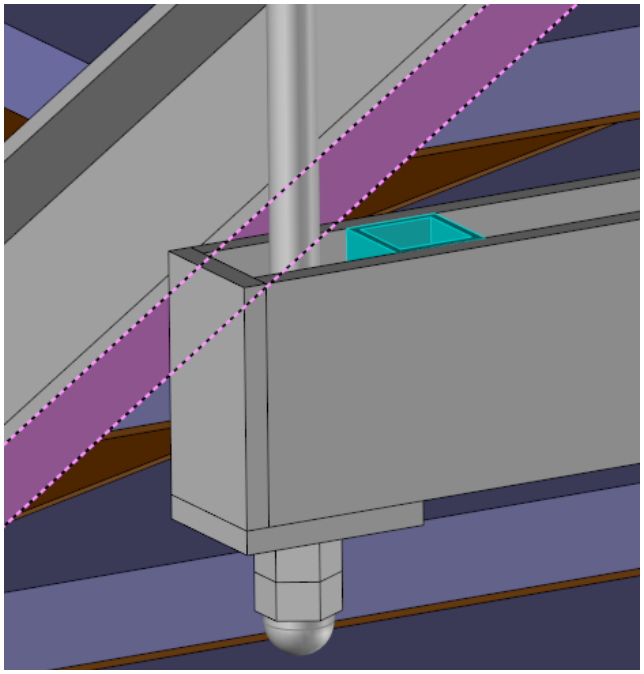
- Click in the **Second geometry** field and reconnect the edge of the stringer.



- Click on  to **confirm** the operation.

A new error message appears.

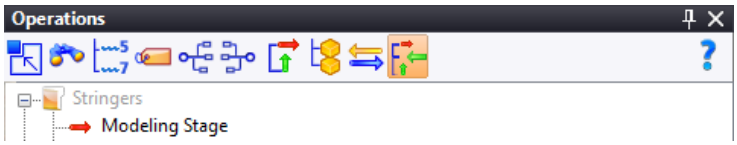
- Click on **OK** to **close** the error message.
- Click on the  icon on the document's tab.
- Reconnect the trimming plane for the small section as shown below.



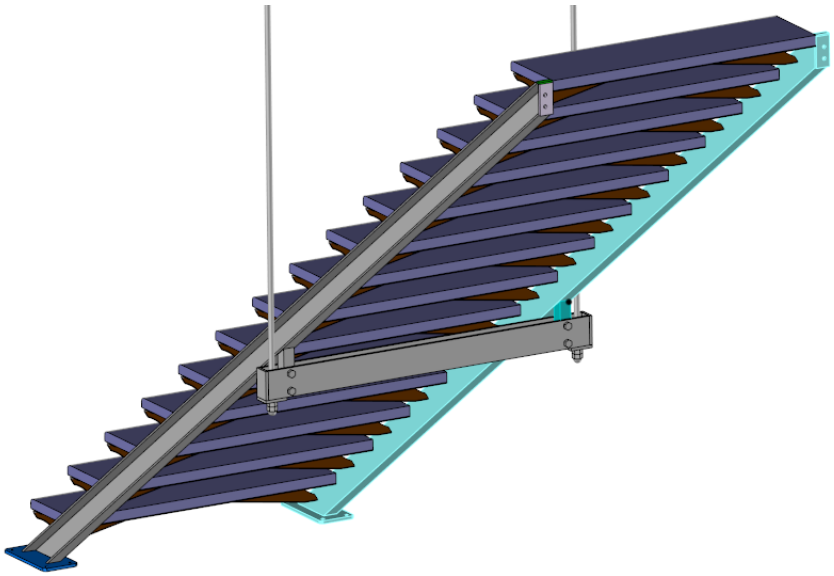
The repair is now complete.

Editing the stringers

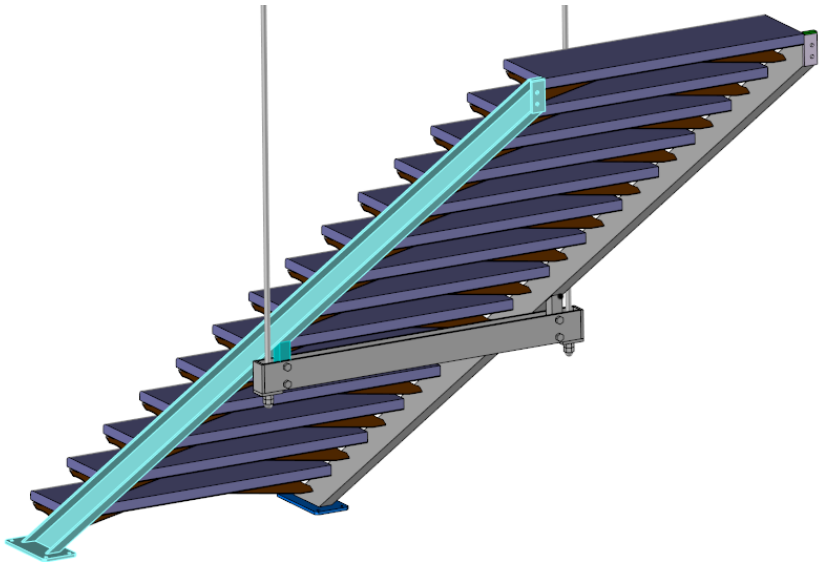
- In the Operations tree, place the **Modeling Stage** cursor inside the *Stringers* folder.



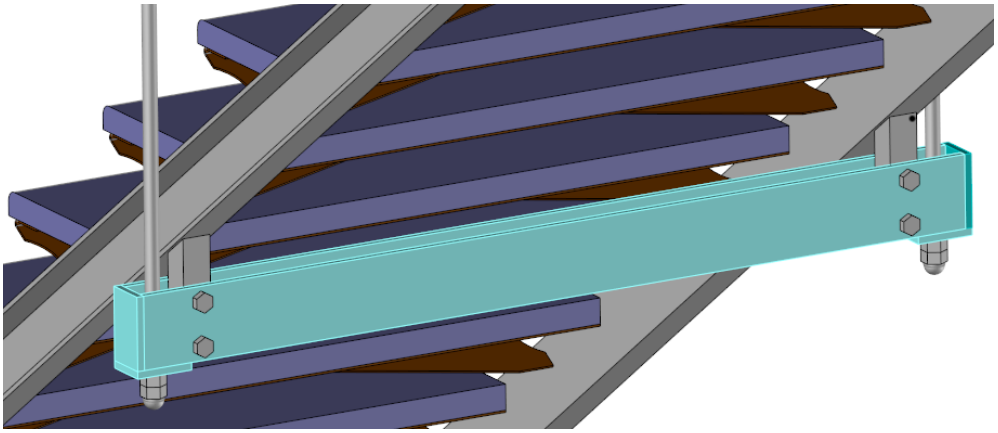
- Edit the left stringer inclusion operation and add the small section cut at an angle.



- Repeat the procedure on the right-hand side.



- Create a new **In place assembly** with all of the parts as shown below.



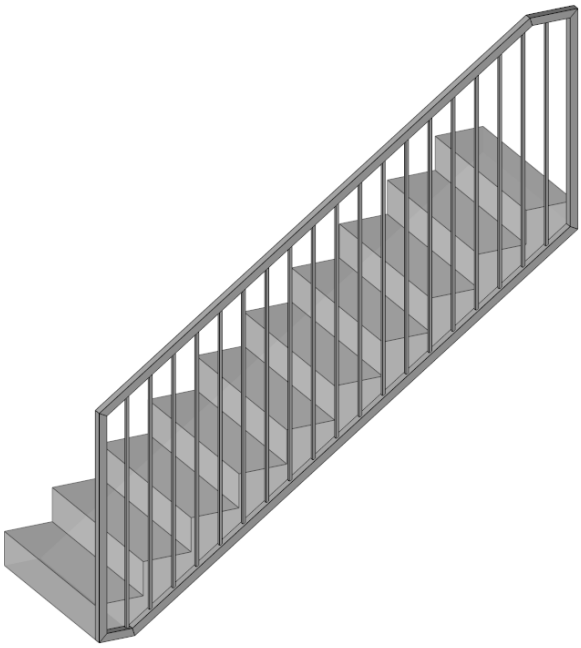
- Click on the **End inserting** icon to move the cursor to the top of the tree.
- **Save** the assembly document.

What follows is a repetition of the previous exercises:

- Bills of material with the names of the different parts/sub-assemblies
- Detail draftings
- Assembly draftings
- etc.

Annex: Ramp Railing

The objective of this exercise is to design a ramp railing on a concrete stair.



Concepts addressed:

- Using a constrained linear pattern to design a ramp railing
- Converting a list to local parts

Importing the concrete stair

- From the Project tree, create a new folder named *Ramp railing*.

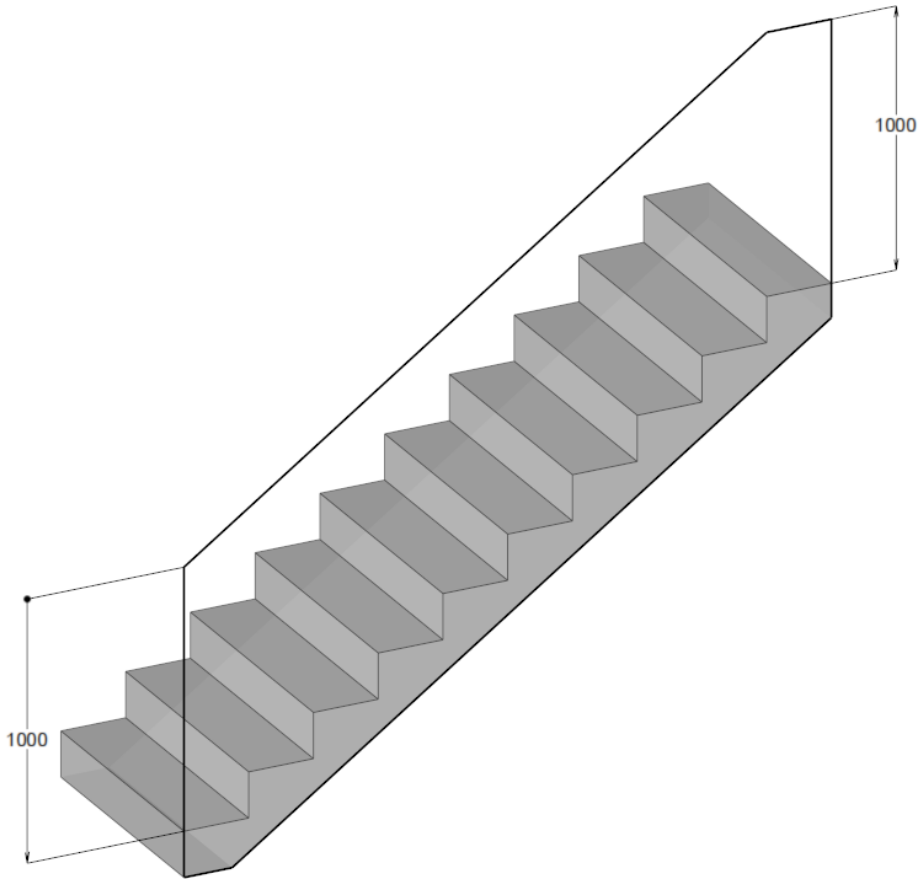
In order to focus on the modeling of the railing, the stair has already been modeled.

- Right-click on the *Ramp railing* folder and select the **Import/Export** > **Import Package** command. Select the *Parameterized Concrete Stair.TopPkg* package.
- Open the *Parameterized Concrete Stair* part document.

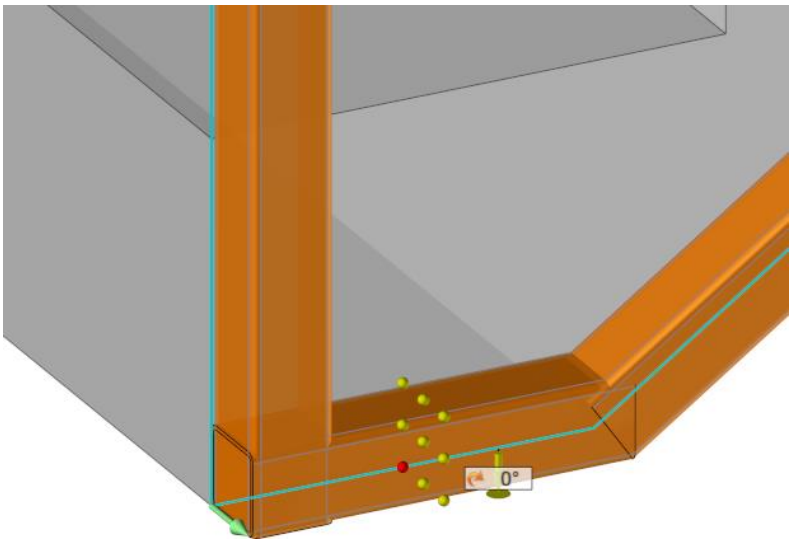
Creating the railing

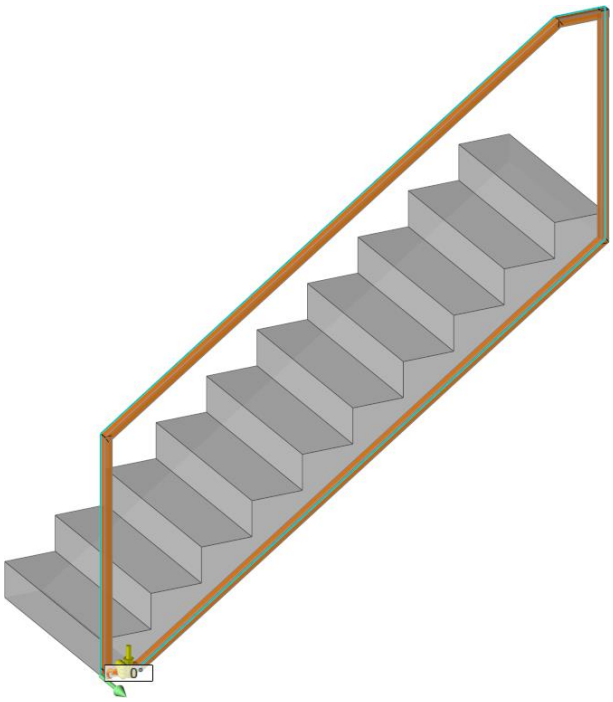
- Right-click on the *Ramp railing* folder and select the **Assembly** command.
- Right-click on the new assembly document and select the **Properties** command.
- Click on the **Edit** button. In the **Description** field, enter *Ramp railing* and then click on to **confirm** the operation.
- At the top left of the screen, click on the **TopSolid 7** icon and select the **File** > **Background Document** command.
- Select the *Parameterized Concrete Stair* document.

- Draw the following sketch.

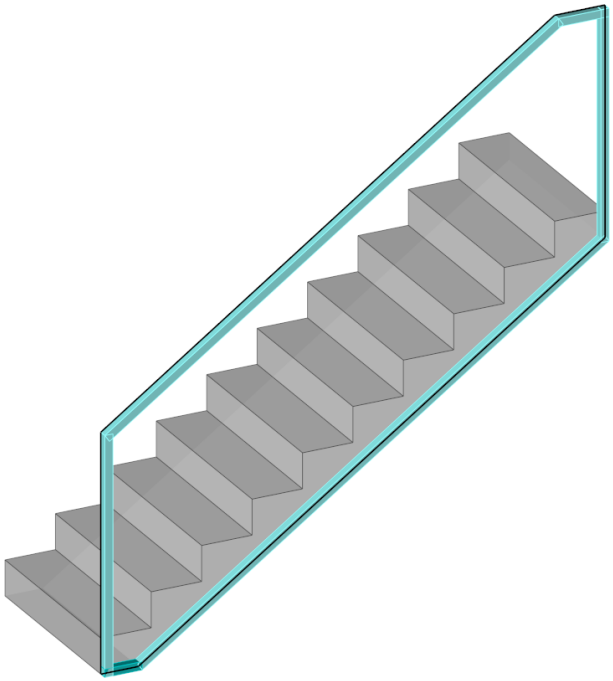


- **Confirm** the sketch.
- Right-click on the sketch and select the **Extruded bar** command.
- Select the **Hollow square section, ISO 4019** family and the **40 x 2** code.
- Select the entire sketch.
- Position the extruded bars as shown below.



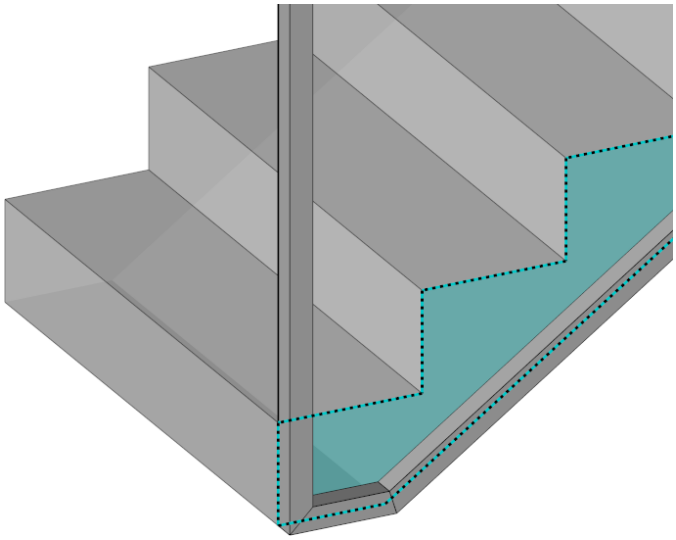


- Click on to **confirm** the operation.
- Right-click on one of the extruded bars and select the **Miter Trim** command.
- Select all the extruded bars and then click on to **confirm** the operation.

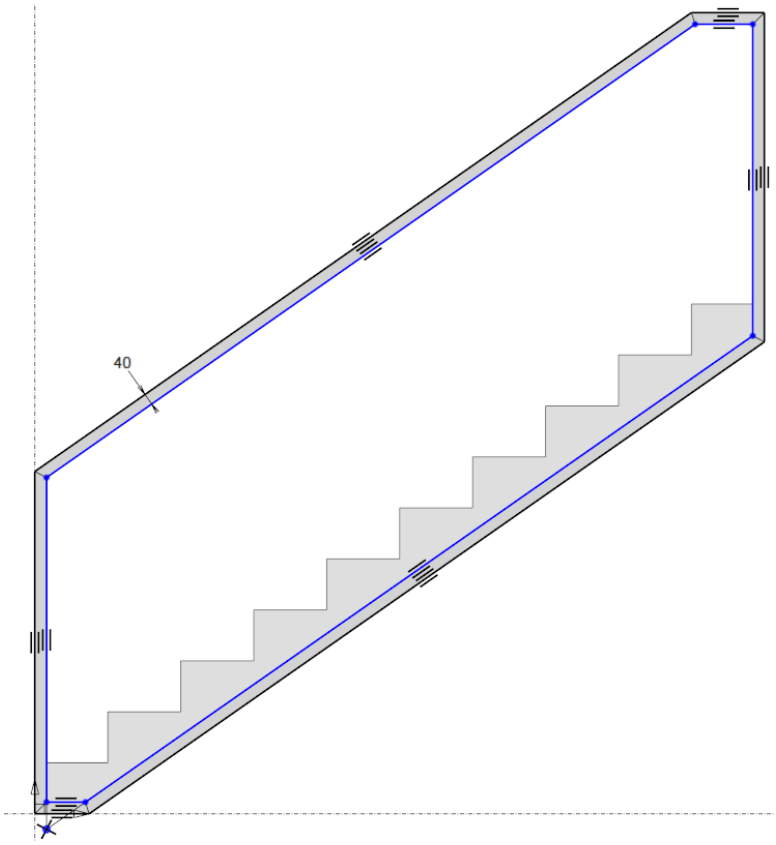


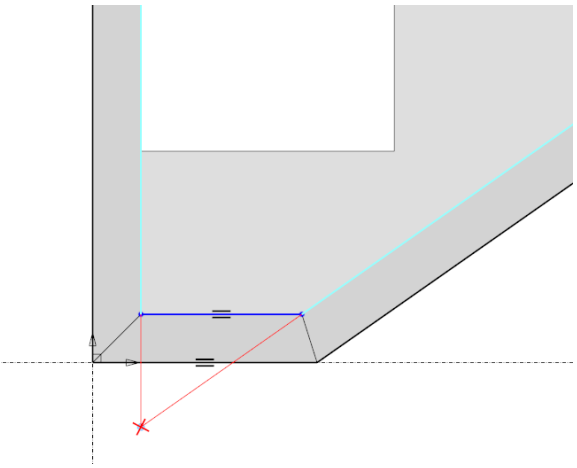
We will now create the sketch that will be used to trim the bars.

- Right-click on the according to extruded bars plane and select the **Sketch** command.

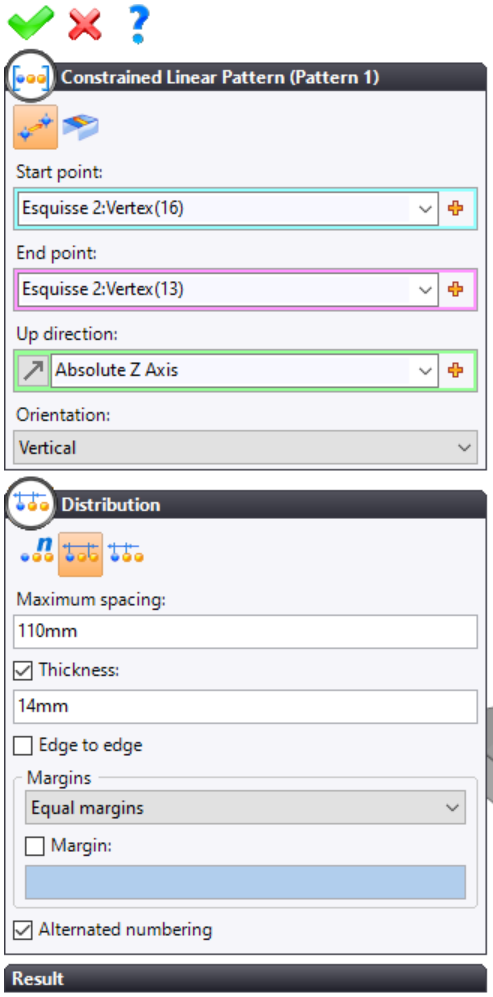


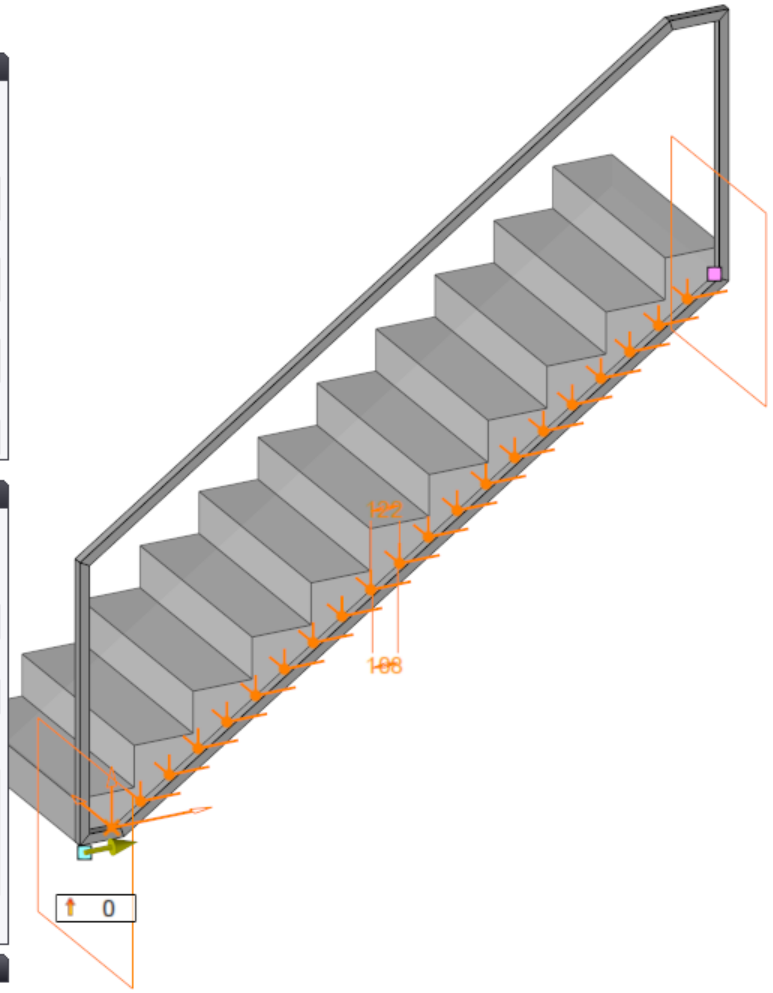
- Draw the following sketch (parallel to *40mm* from the previous sketch + point).

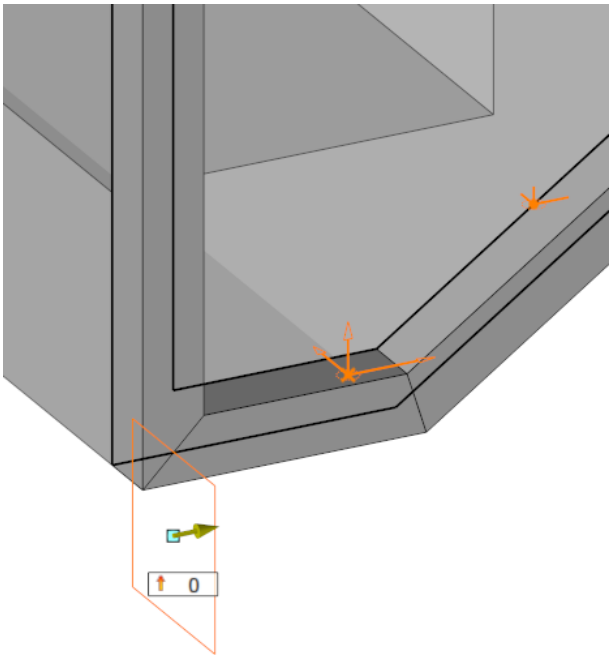


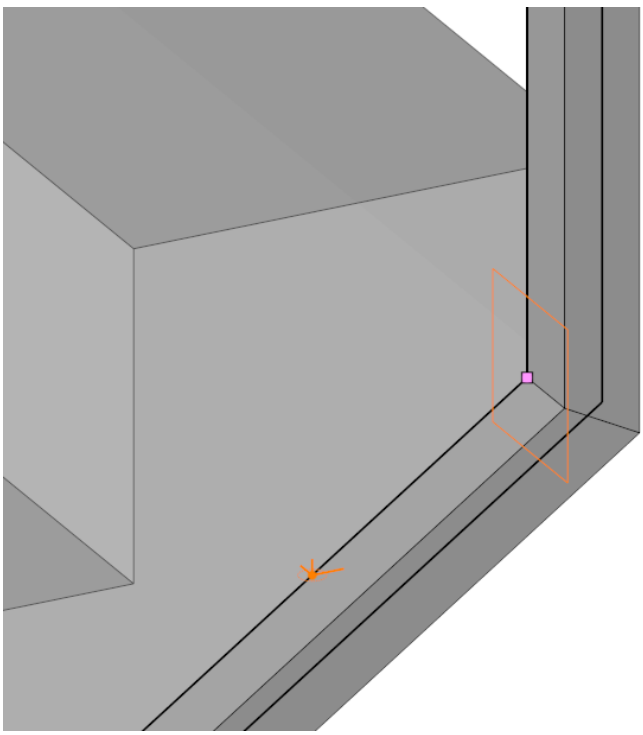


- From the **Construction** tab, select the **Constrained Linear Pattern** command.
- Select the **Between points** mode and adjust the dialog box as shown below.



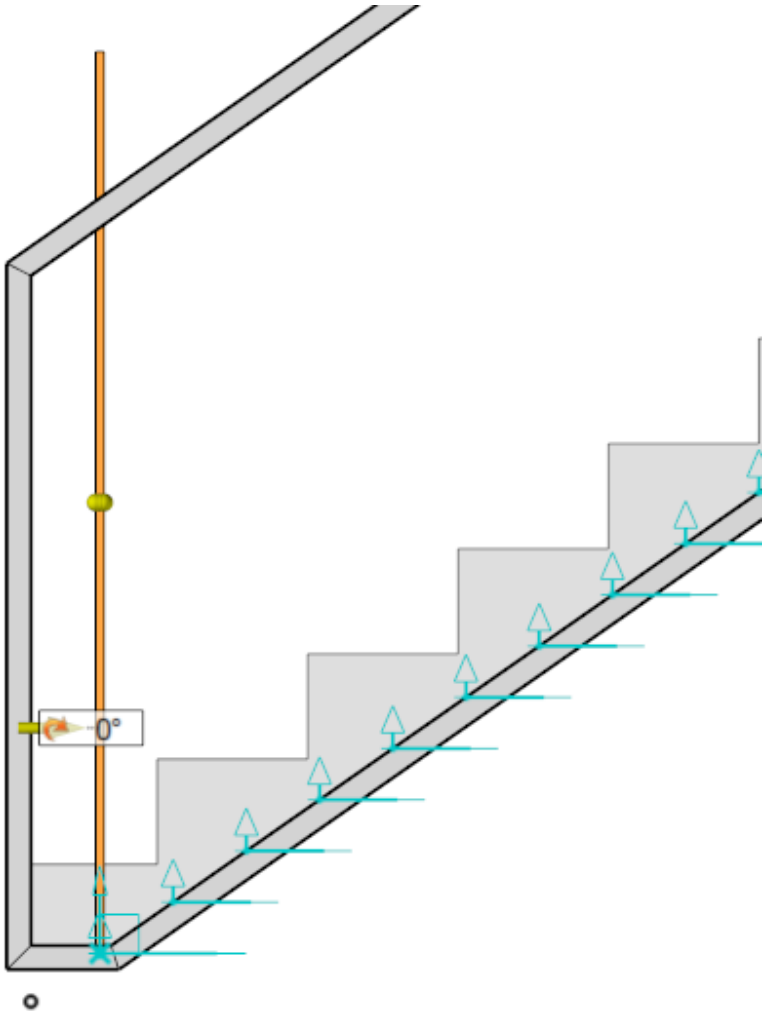




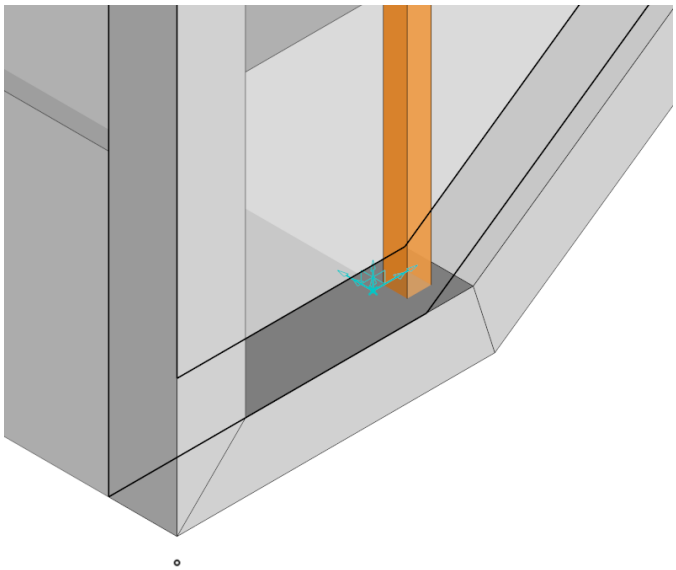


- Click on to **confirm** the pattern.

- From the **Modeling** tab, select the **Extruded Bar** command.
- Select the **Full Square Section, NF A 45-004** family and the **14** code.
- Select the **Frame and Length** mode.
- In the **Destination frame** field, select the frame of the constrained linear pattern.

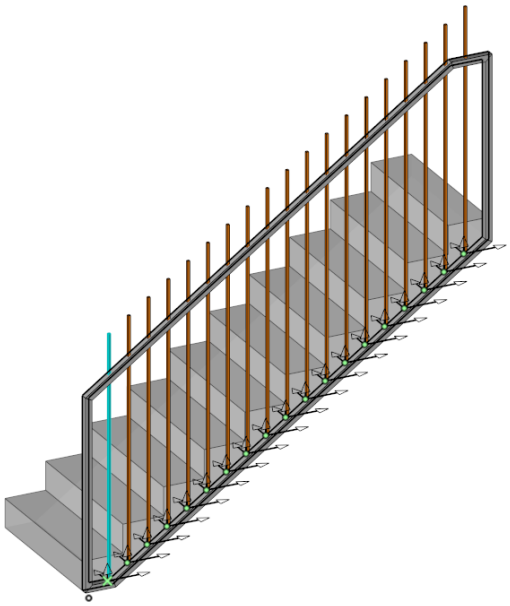


- Enter a **length** of *1500mm* to make sure that the section exceeds the border in all circumstances.
- Add a **vertical shift** of *20mm* to center the bar on the hollow square sections as shown below.

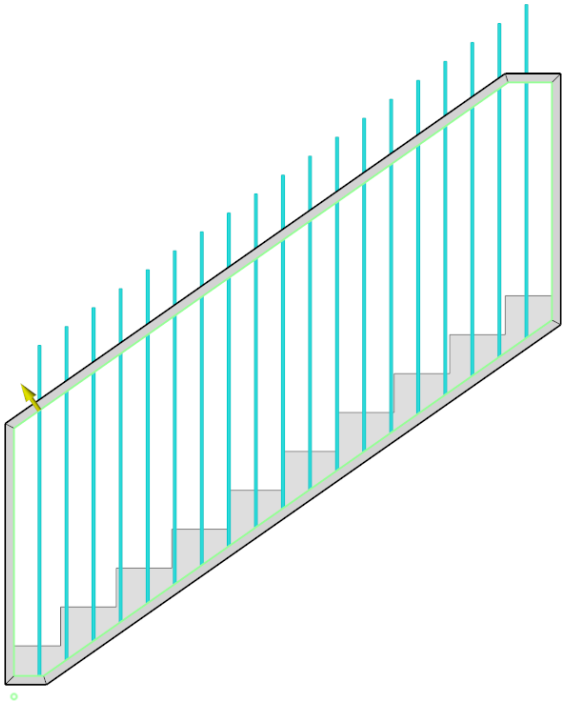


- Click on to **confirm** the extruded bar.

- From the **Construction** tab, select the **Repetition** command.
- Select the extruded bar previously included as the **entities** to be repeated.
- Click in the **Pattern** field to enable it. Select a point of the constrained linear repetition as shown below.



- Click on to **confirm** the repetition.
- Right-click on second sketch (the one with the 40mm parallel) and select the **Trim by Profile** command.
- In the **Part to modify** field, select the repetition of the bars, then click on to **confirm** the operation.



TopSolid converts all of the parts into local parts. This means that these parts do not have a document in the Project tree. Their management is done entirely from the assembly stage.

- Open the *Parameterized Concrete Stair* part document.
- From the Entities tree, open the **Parameters** folder, then work on the *Tread*, *UnitRise* and *StepNumber* parameters to test the railing.

Notes

Individual Course Evaluation Form

(To be completed and returned to the training instructor at the end of the course)

TopSolid'Steel - Basics

Name : .....

Company : .....

Date(s) from ..... to .....

By completing this individual evaluation form, you are helping to improve the quality and usefulness of the training provided in the future. Please complete it carefully.

Number of people during the course: Onsite at your company? YES ☐ NO ☐

GENERAL ASSESSMENT	Poor	Average	Good	Excellent
Overall, this course has been:	☐	☐	☐	☐
What grade would you assign?	0 1 2 3 4 5 6 7 8 9 10			

LOGISTIC	Poor	Average	Good	Excellent
Orientation (quality, organization, user-friendliness, etc.)	☐	☐	☐	☐
Physical setup (room, materials, etc.)	☐	☐	☐	☐

TRAINING	Poor	Average	Good	Excellent
Instructor's teaching method	☐	☐	☐	☐
Group relationship (participation, sharing of experiences)	☐	☐	☐	☐
Quality and clarity of educational materials (documentation)	☐	☐	☐	☐
Balance between Theory and Practice	☐	☐	☐	☐
Consistent presentations with what has been announced	☐	☐	☐	☐
Training Content	☐	☐	☐	☐

DURATION	No	Somewhat no	Somewhat yes	Yes
Does the overall duration of the course seem appropriate?	☐	☐	☐	☐
If no, was it?	Too short ☐		Too long ☐	

PACE	No	Somewhat no	Somewhat yes	Yes
Does the overall pace of the course seem appropriate?	☐	☐	☐	☐
If no, was it?	Too slow ☐		Too fast ☐	

USE OF ACQUIRED KNOWLEDGE IN THIS TRAINING	No	Somewhat no	Somewhat yes	Yes
Have you found this training to be useful in your work?	☐	☐	☐	☐
Do you think you can put the acquired knowledge into use quickly?	☐	☐	☐	☐
Do you believe that you have achieved your objectives upon completion of this course?	☐	☐	☐	☐

Comments and suggestions: .....

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