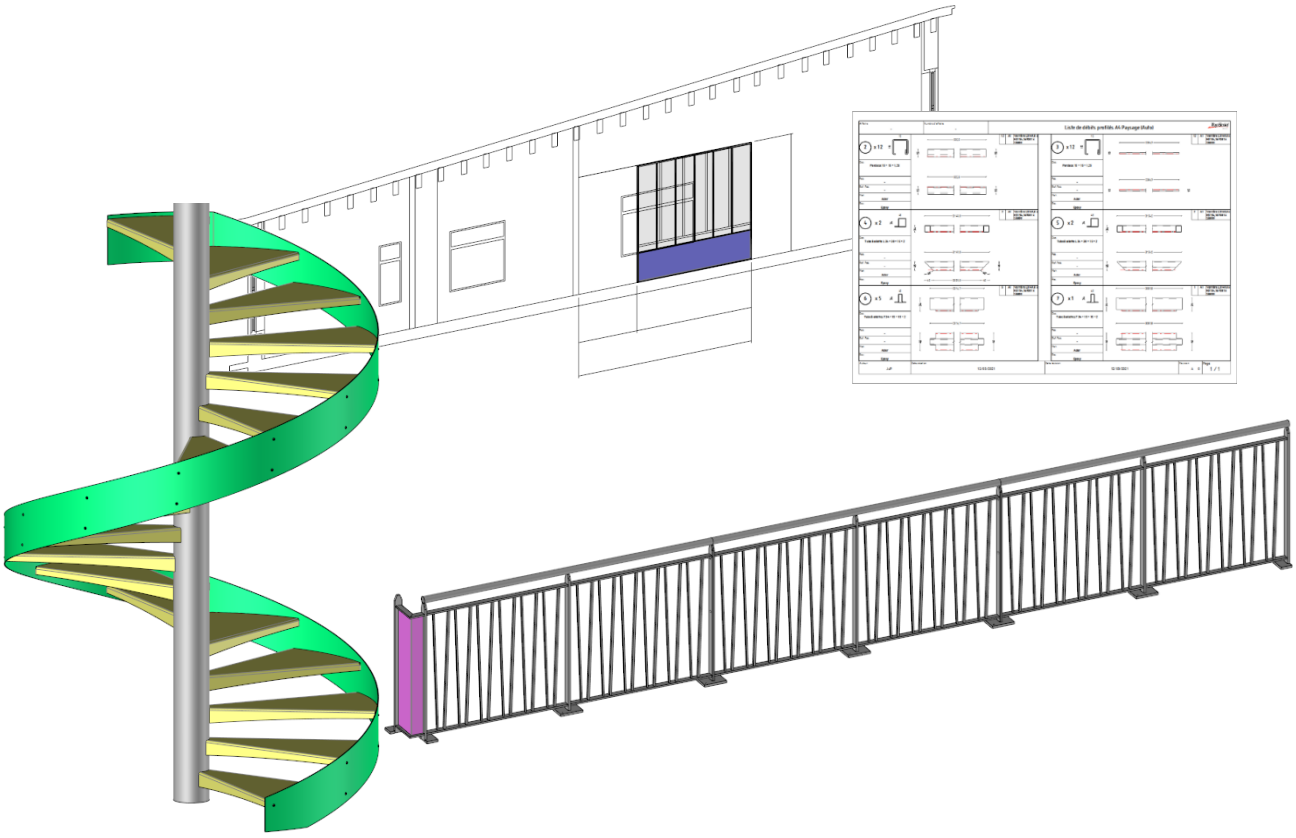
TopSolid Steel

Training Guide

TopSolid'Steel - Advanced



MASTER YOUR MANUFACTURING PROCESS

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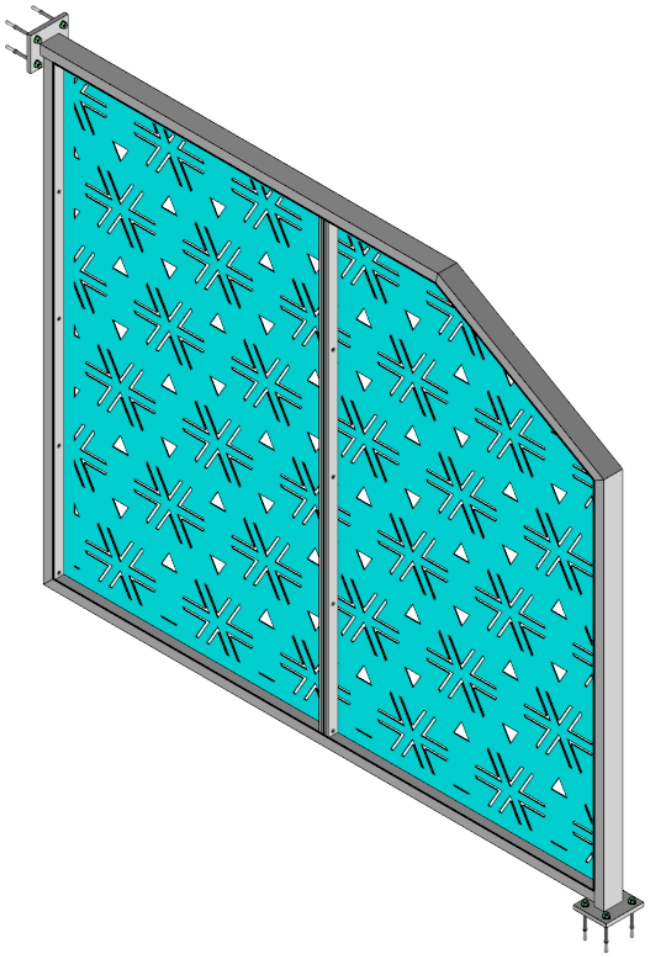
Note: If you are experiencing problems using this training guide, please feel free to send your feedback and comments at edition@topsolid.com.

Contents

Part 1 - Designing a Privacy Screen

The purpose of this exercise is to draw the following privacy screen based on an environment.



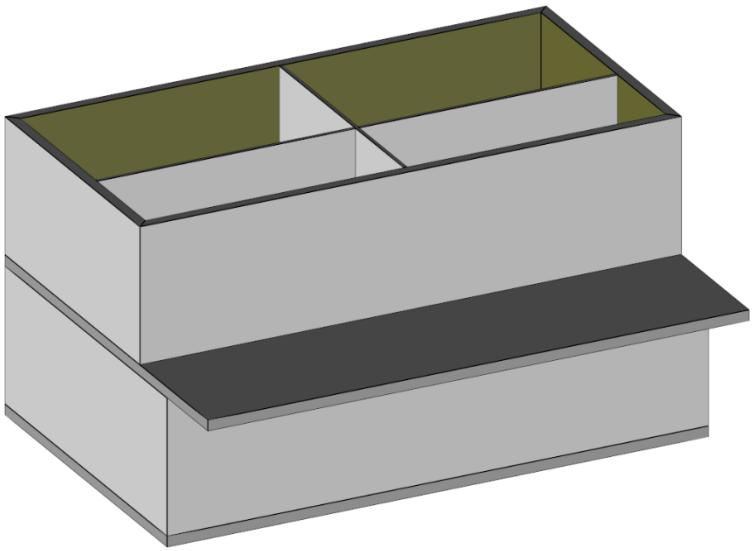
Concepts addressed:

- Using the **Building** command
- In-place design
- Using assembly kits
- Performing trimming operations while keeping the trimmed side

Creating the project

- From the **TopSolid 7** home page, click on the **New Project** icon.
- From the **Steel Templates** folder, select **Design Template**, rename the project *Advanced TopSolid'Steel Training*, then click on to **confirm** the operation.
- In the Project tree, create a first folder named *1- Privacy screen*. To do this, right-click on the *Advanced TopSolid'Steel Training* project name and select the **Folder** command.

Designing the environment using the Building command



- Right-click on the *1- Privacy screen* folder and select the **Assembly** command.
- Right-click on the new assembly document and select the **Properties** command.
- Click on the **Edit** button.
- Enter *Environment* in the **Description** field and click on to **confirm** the operation.

During the basic session of **TopSolid**, we saw how to create an environment with standard modeling commands.

TopSolid'Steel also integrates a simplified drawing tool to create buildings.

- From the **Modeling** tab, select the **Building** command.
- In the **Building** section, enter *2* in the **Level count** field and *250cm* in the **Default height under ceiling** field.
- In the **Walls** section, select the **Wall** family and enter *200mm* as the **default thickness**.
- In the **Slabs** section, select the **Slab** family and enter *200mm* as the **default thickness**.

The **Positioning** section deals with the positioning of the building.

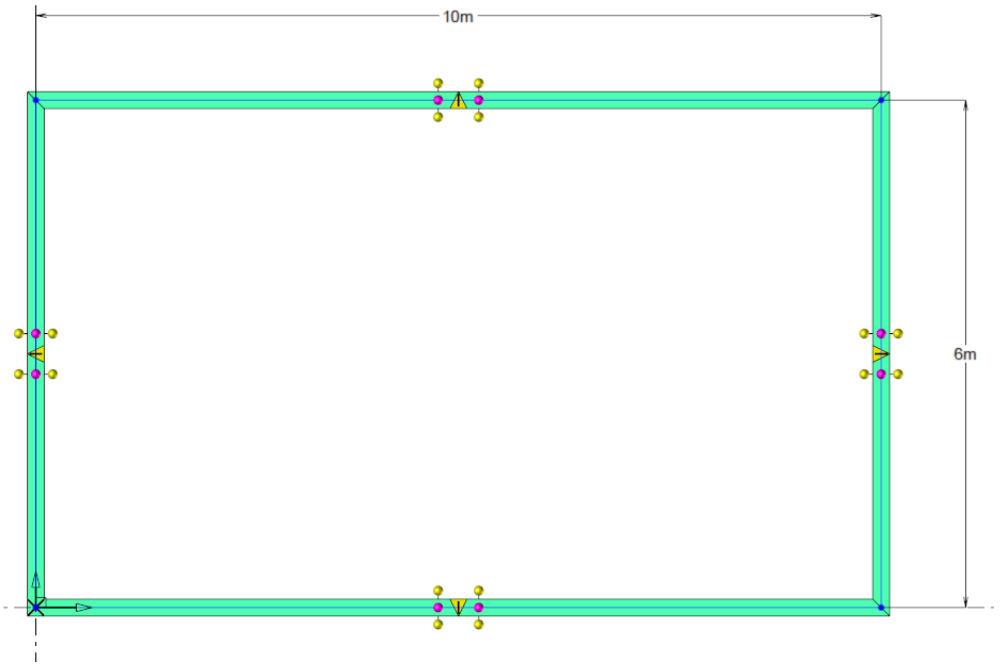
- Keep the default values.

In the **Advanced Options** section, the **Imprint walls on slabs** option allows you to divide the slab into zones. This can be useful to retrieve a surface or create a coating quickly.
The **Extend exterior walls** option allows you to hide the slab with the walls. This is purely for aesthetic purposes and does not affect the different dimensions in any way.

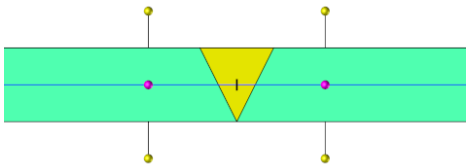
- Uncheck the **Extend exterior walls** box.
- Click on to **confirm** the **Building** command.

TopSolid switches to the sketch context for the walls. The aim here is not to complete the design of the entire building but simply the area concerned by the work.

- Select the **Rectangle** command and draw the following rectangle.



TopSolid draws the walls dynamically with graphic handles.

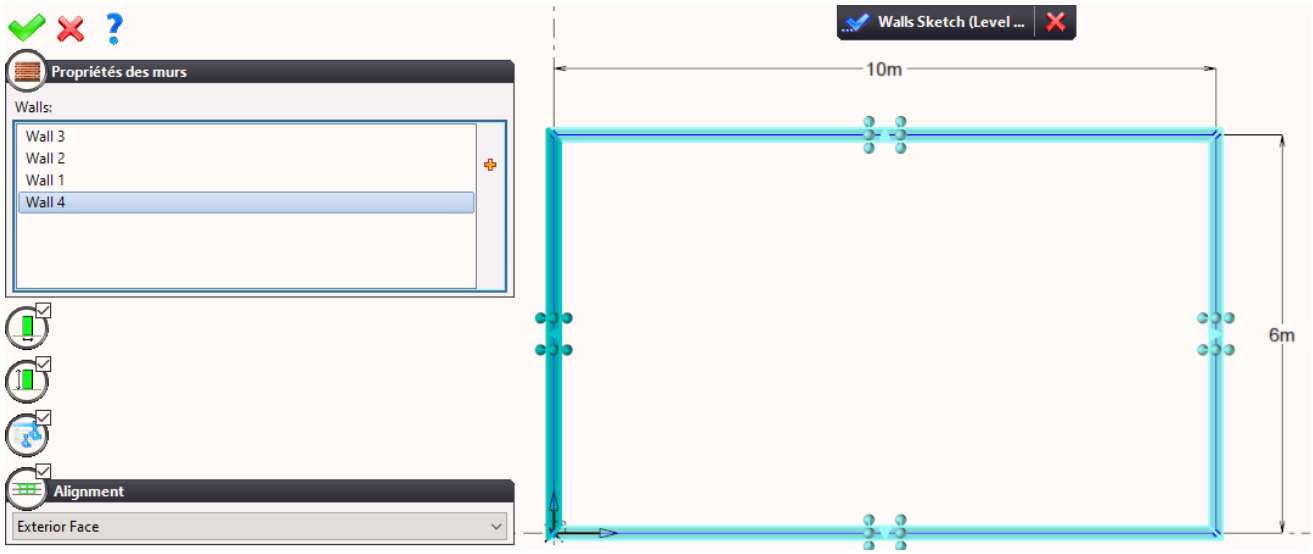


Note: The yellow arrow indicates the outside of the wall: the point corresponds to the outside and the base corresponds to the inside. By clicking on this arrow, you can flip the wall.
The spheres allow you to change the alignment of the wall in relation to the created segment.

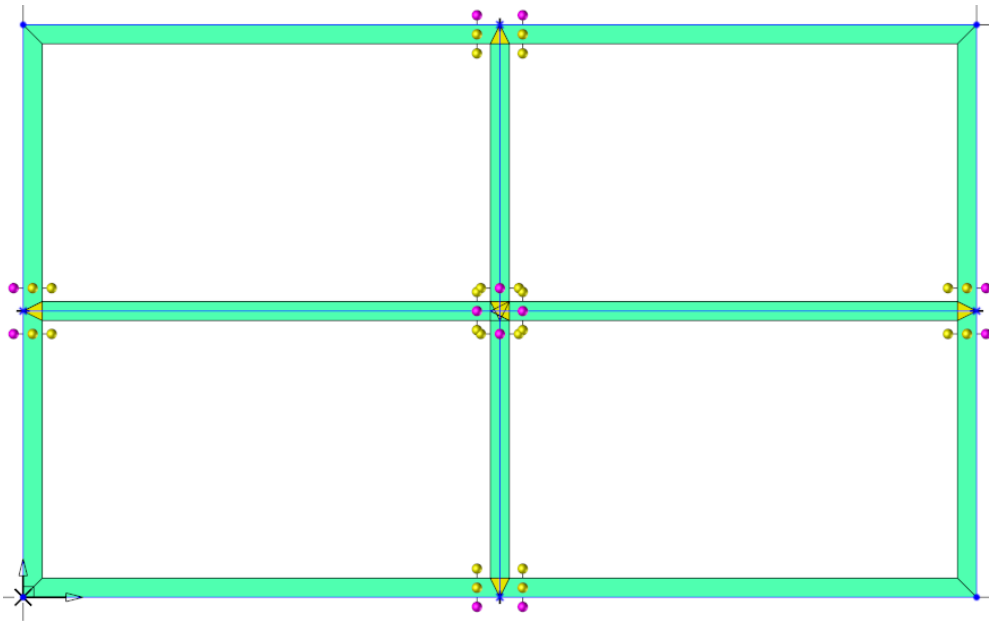
- Double-click on one of the walls to display the wall properties.

The **Wall properties** command allows you to make local or global changes. You can therefore modify the thickness, the height, the alignment or even the drivers on the **Wall** component (family).

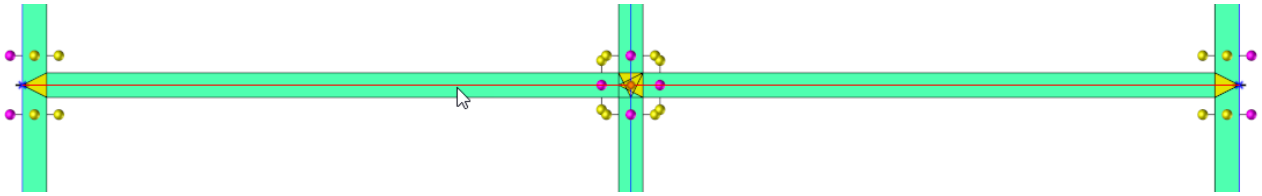
- In the **Walls** field, select all the walls.
- In the **Alignment** section, select **Exterior Face**.



- Click on to **confirm** the operation.
- Add the two segments as shown below.

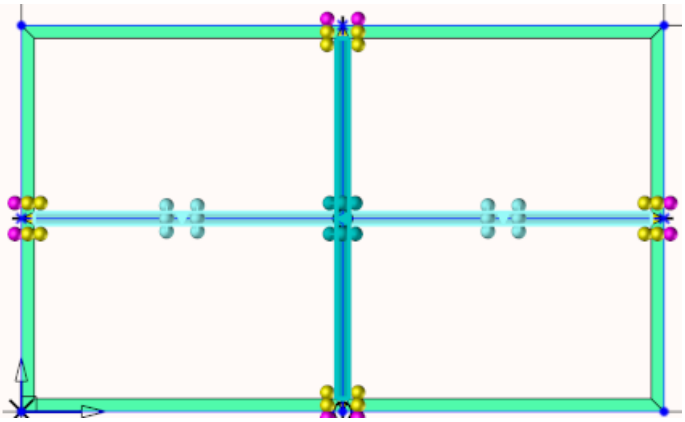


- Select the **Trim** command.
- Activate the **Divide** mode and click on the left part of the horizontal segment.



The wall is then divided into two parts.

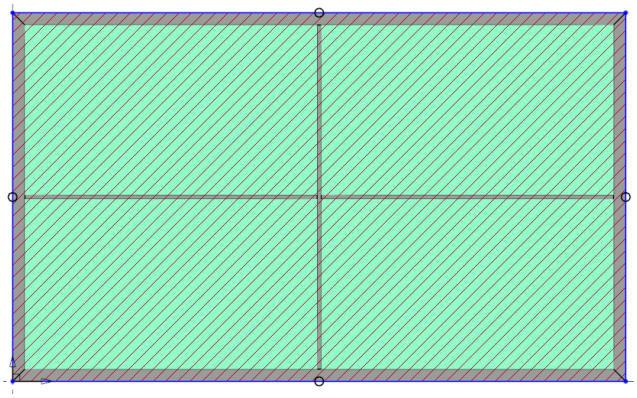
- Select the three interior walls, then right-click and select the **Wall Properties** command.
- In the **Thickness** section, select the **Manual** mode and enter *70mm*.



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

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

TopSolid automatically generates a profile that includes all the walls.

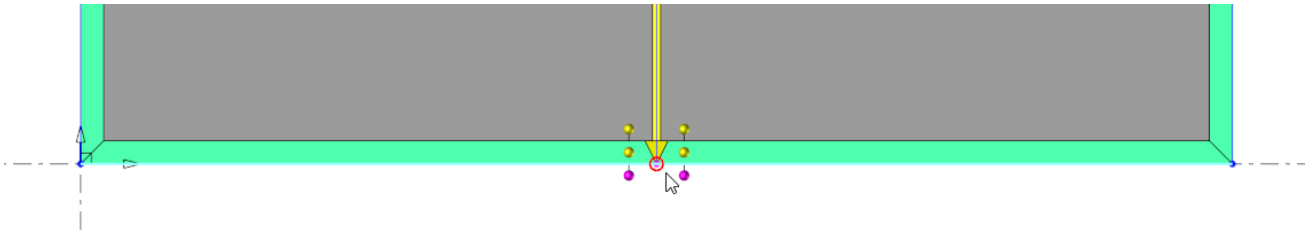


We will now move on to the next level.

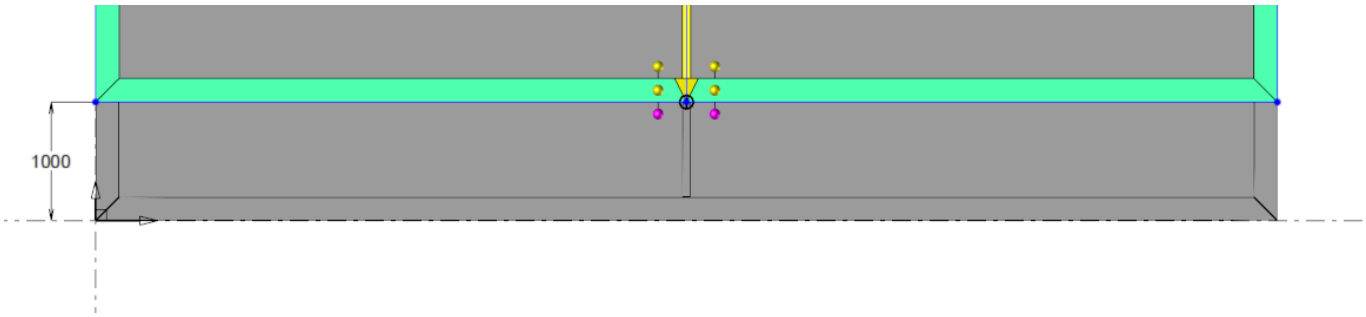
- Right-click in the graphics area and select the **Edit Walls of Upper Level** command.

TopSolid automatically uses the same walls as for the lower level. You can modify this level completely independently.

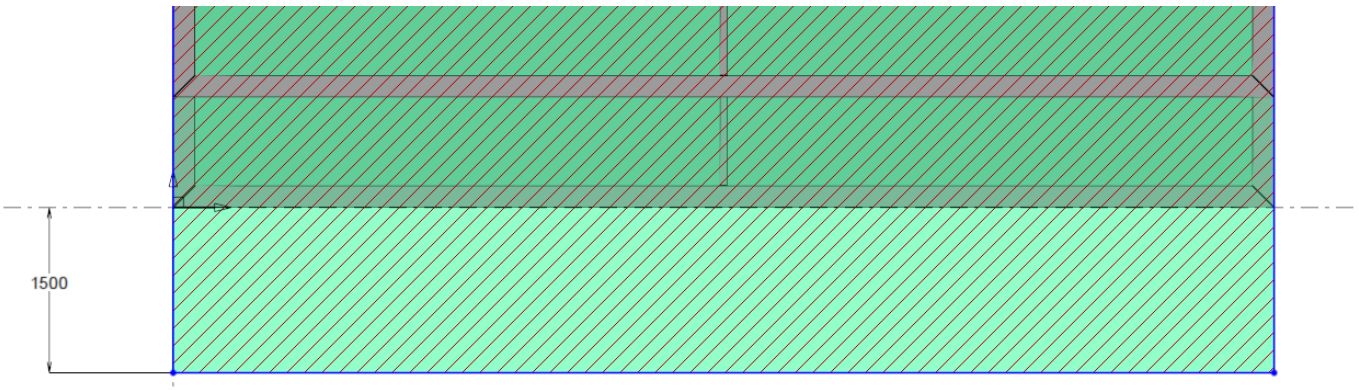
- Delete the coincidence constraint as shown below using the **Del** key on the keyboard or the **Delete** command from the contextuam menu.



- Offset the wall upward and add the dimension as shown below.

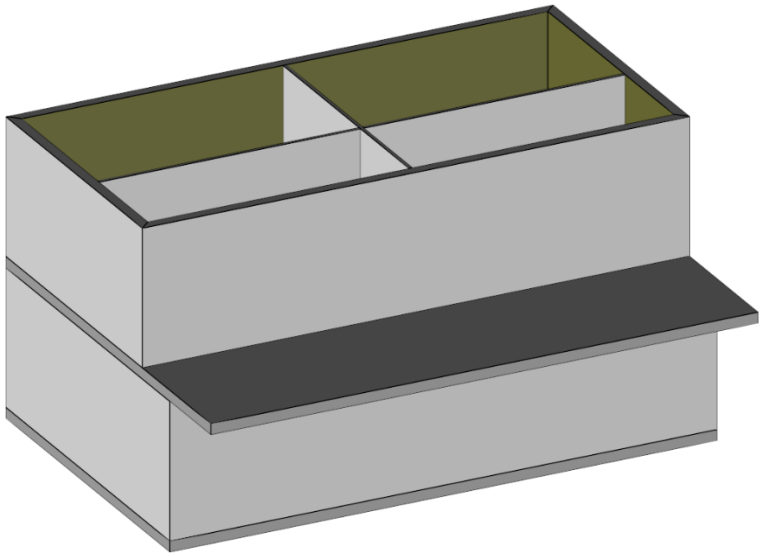


- Right-click in the graphics area and select the **Edit Slab** command.
- In the same way as for the walls, offset the edge of the slab toward the outside.

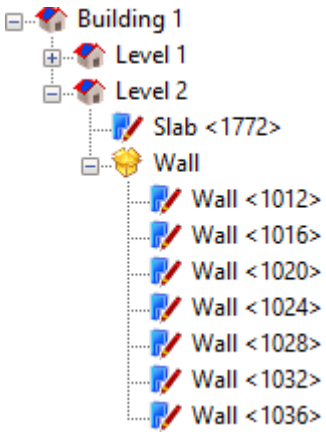


- **Confirm** the sketch.

You should obtain the following result.

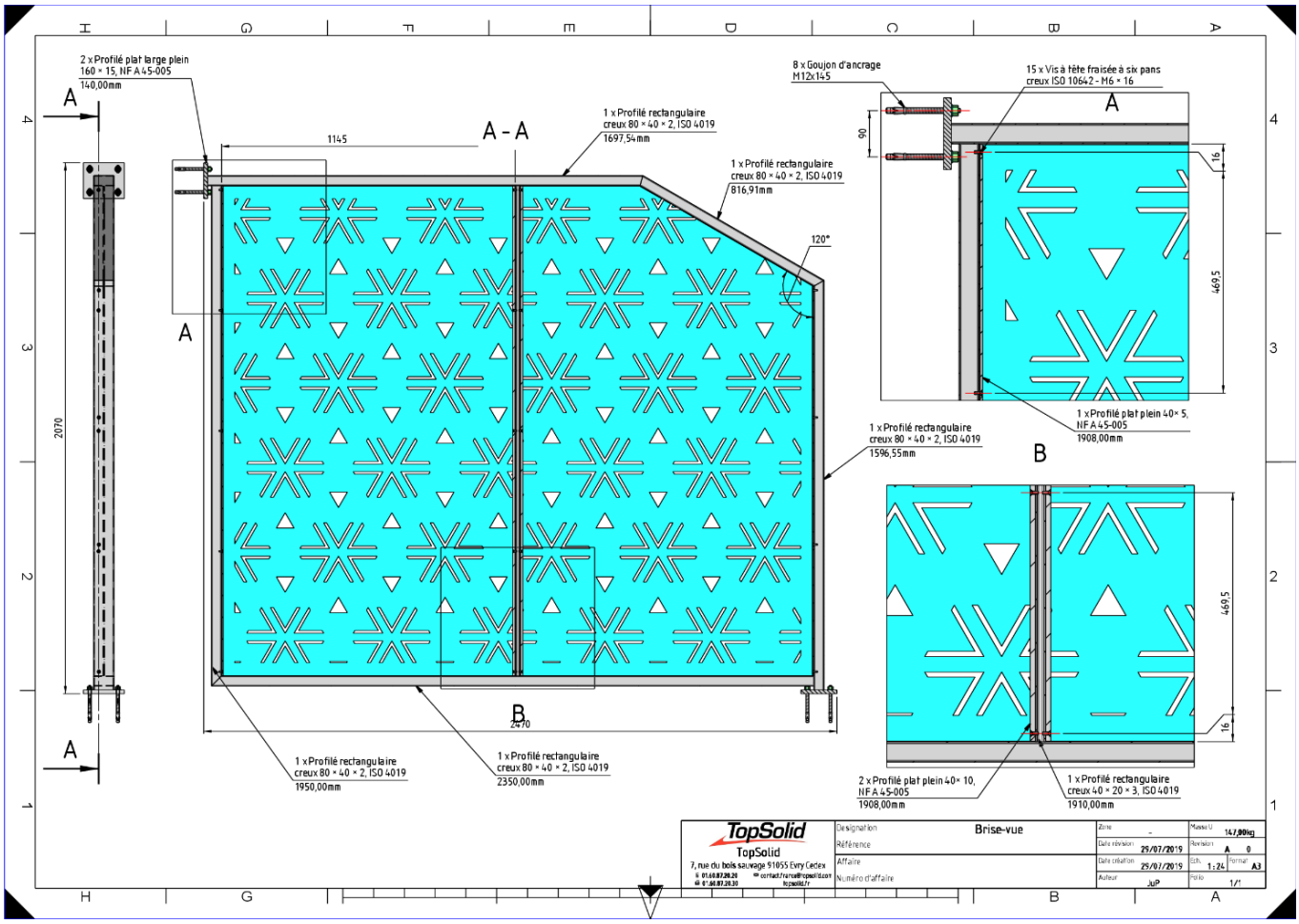


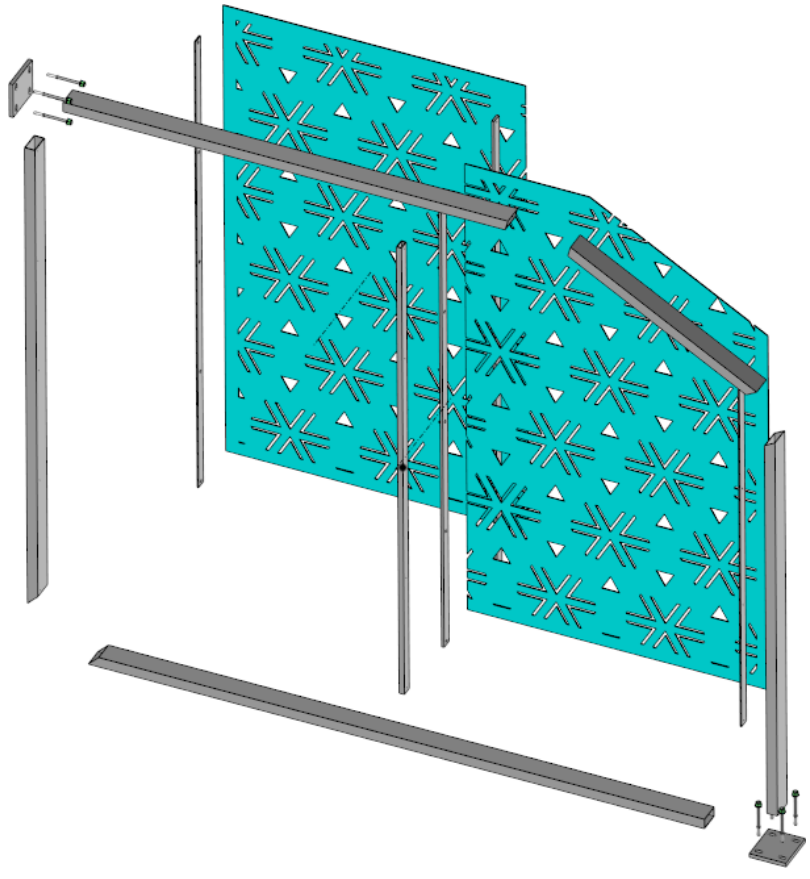
A simple and clear structure of the building is constructed in the Parts tree as well as in the Entities tree.



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

Designing the privacy screen



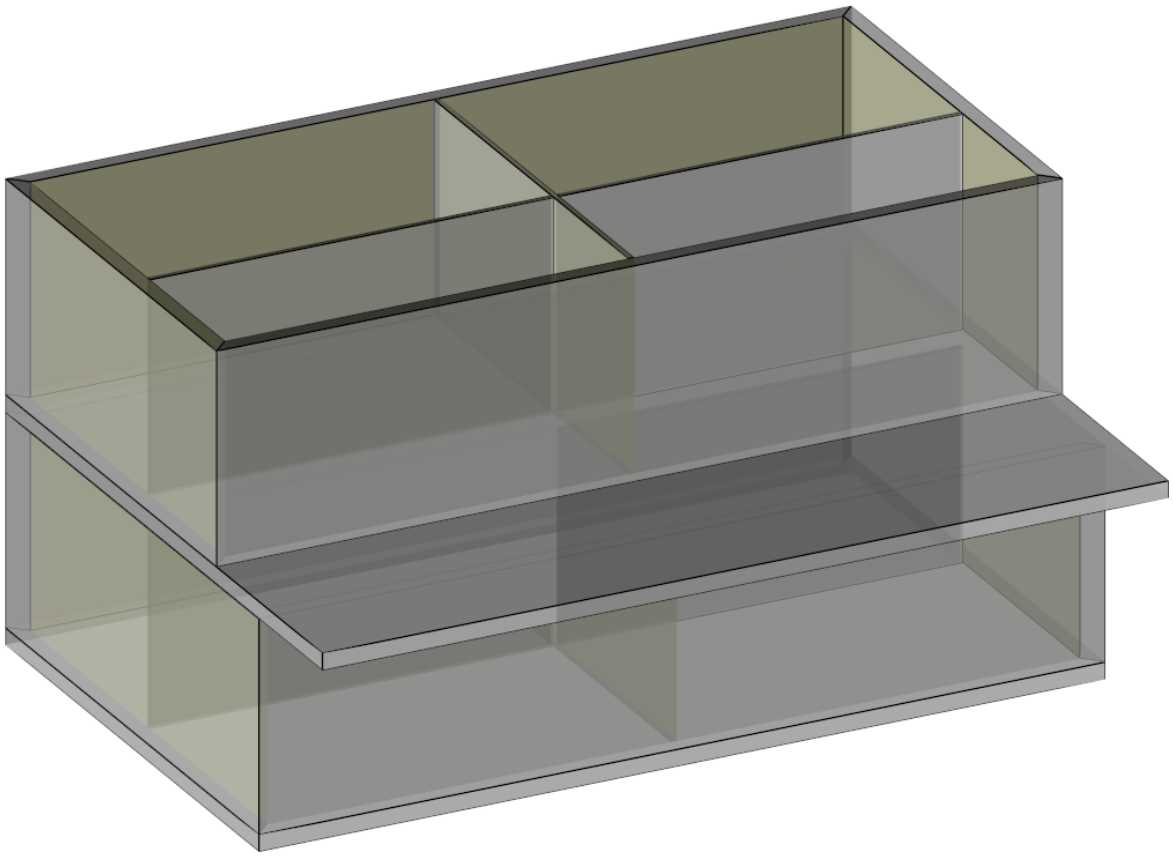


The privacy screen will be designed using parametric components whose design principles will be described throughout the rest of the training.

- Right-click on the *1- Privacy screen* folder and select the **Import/Export** > **Import Package** command.
- Select the package named *Privacy Screen Components.TopPkg*.

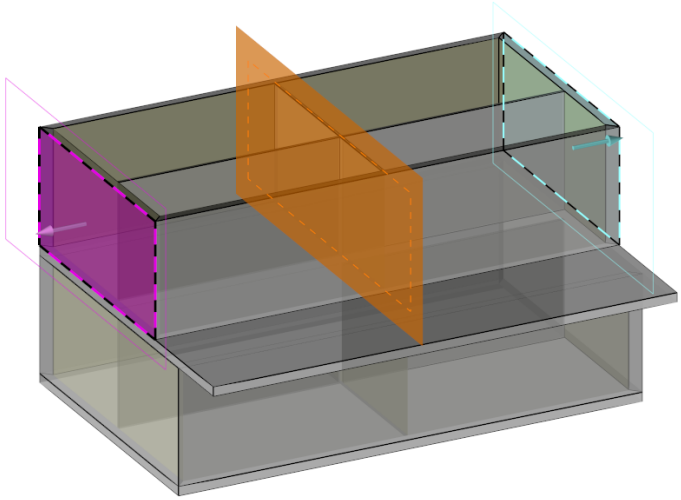
Reminder: This import method allows you to copy the contents of the package locally to a project folder. Only the elements resulting from the projects can be copied in this way. The elements from libraries are imported as libraries.

- Right-click on the *1- Privacy screen* folder and select the **Assembly** command.
- Right-click on the new assembly document and select the **Properties** command.
- Click on the **Edit** button.
- Enter *Privacy screen* in the **Description** field and click on to **confirm** the operation.
- Click on the **TopSolid 7** icon at the top left of the screen and select the **File** > **Background Document** command.
- Select the **Environment** document and click on to **confirm** the operation.

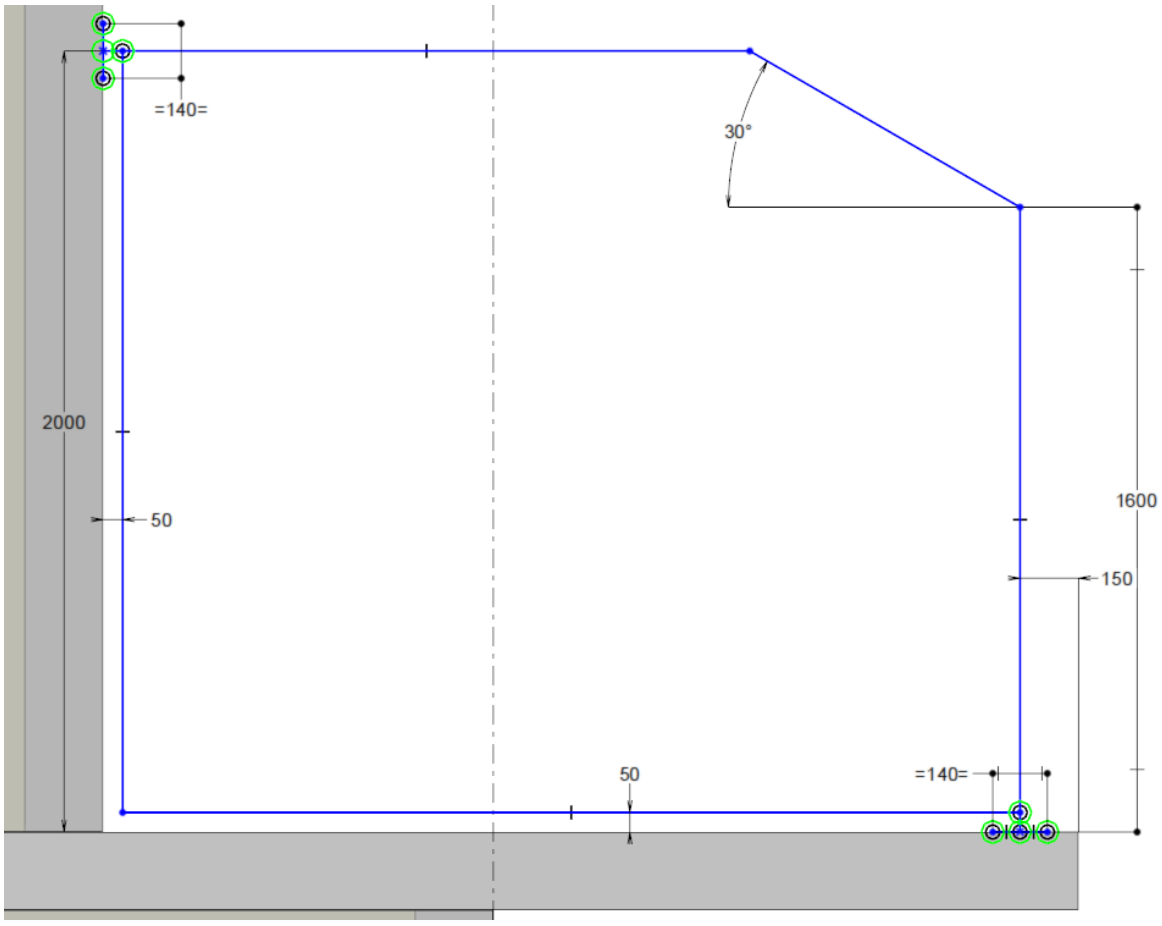


Creating the outer frame

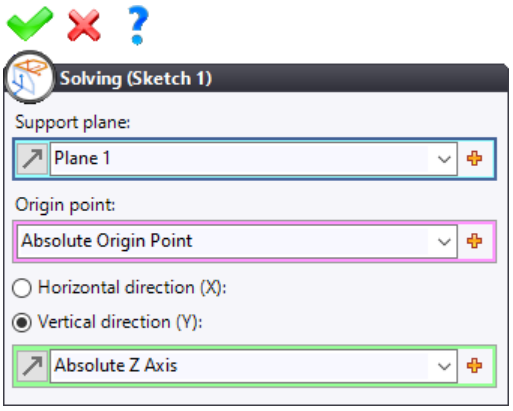
- Select the **Construction** > **Midplane** command, then select the planes as shown below.



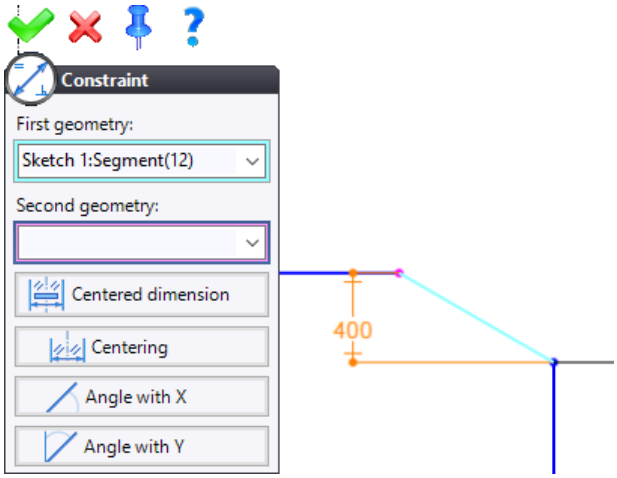
- Create the following sketch on the previously created plane.



The orientation of the sketch does not matter (wall on the left or right). However, if you want to reorientate the sketch, you only have to select the **Position Sketch** command and click on the icon to the left of **Plane 1**.

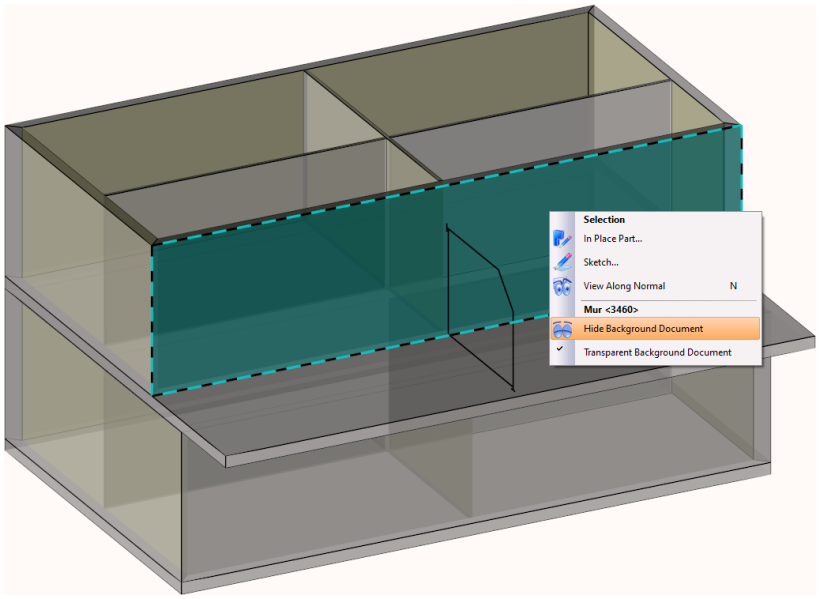


For the 30° angular dimension, you can use an existing horizontal segment or use the virtual X axis. To do this, when setting the dimension, you can either click on the **Angle with X** button in the **Constraint** dialog box or use the contextual menu.

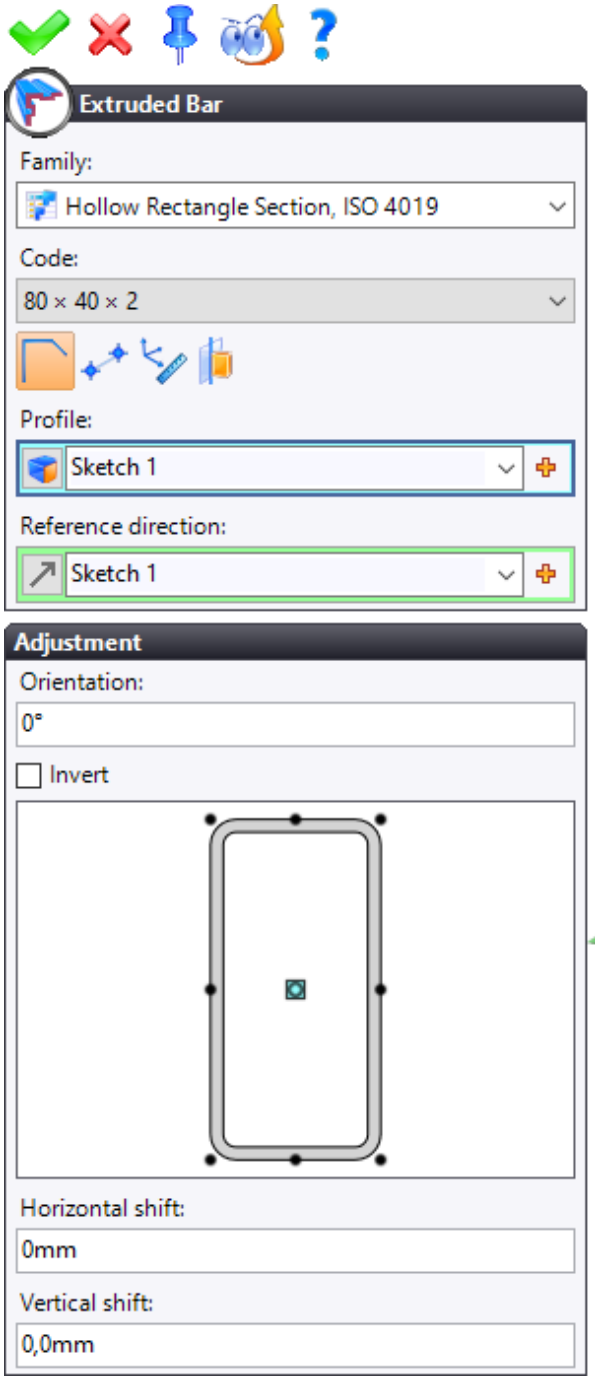


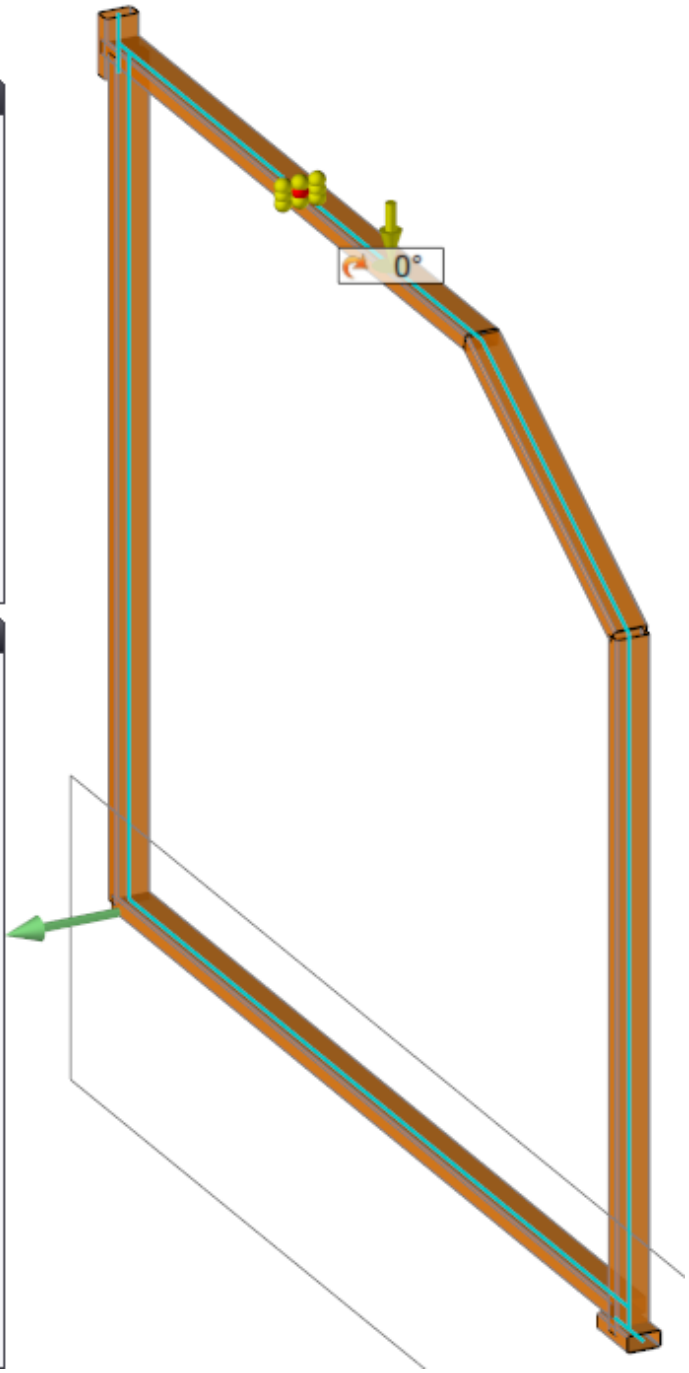
The sketch must be completely blue.

- **Confirm** the sketch.
- Hide the background document using the contextual menu or from the Entities tree.

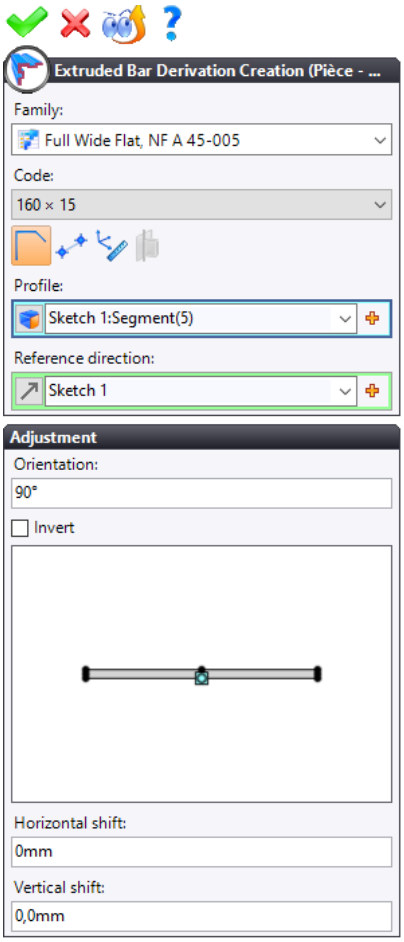


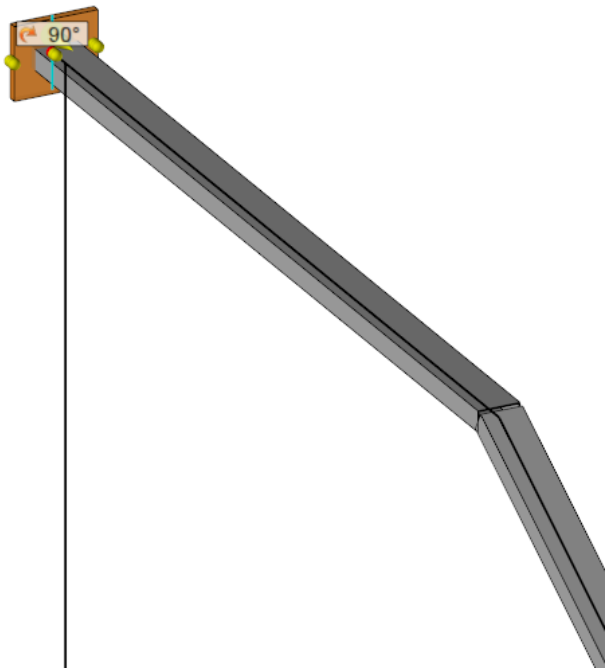
- Select the **Modeling** > **Extruded Bar** command.
- Select the whole sketch, then select the **Hollow Rectangle Section, ISO 4019** family and the **80 x 40 x 2** code.

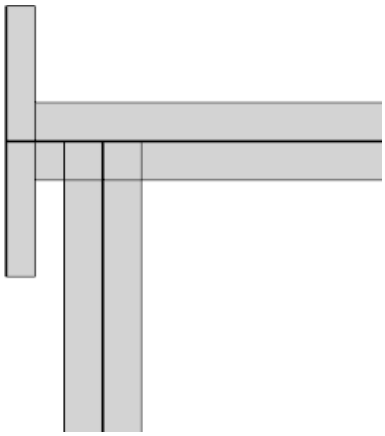




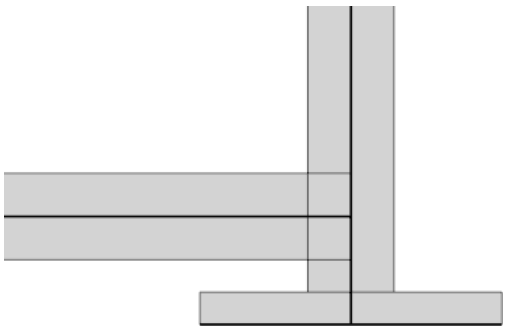
- Right-click on the extruded bar as shown below and edit it.
- Replace the family with **Full Wide Flat, NF A 45-005** and the code with **160 x 15**.
- Position the extruded bar as shown below.



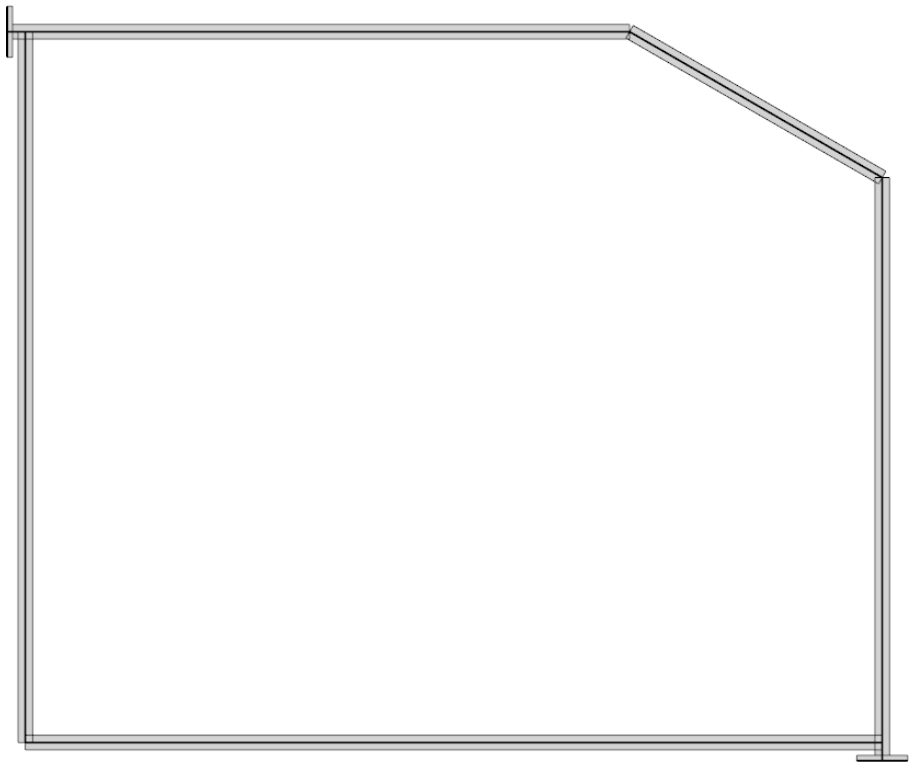




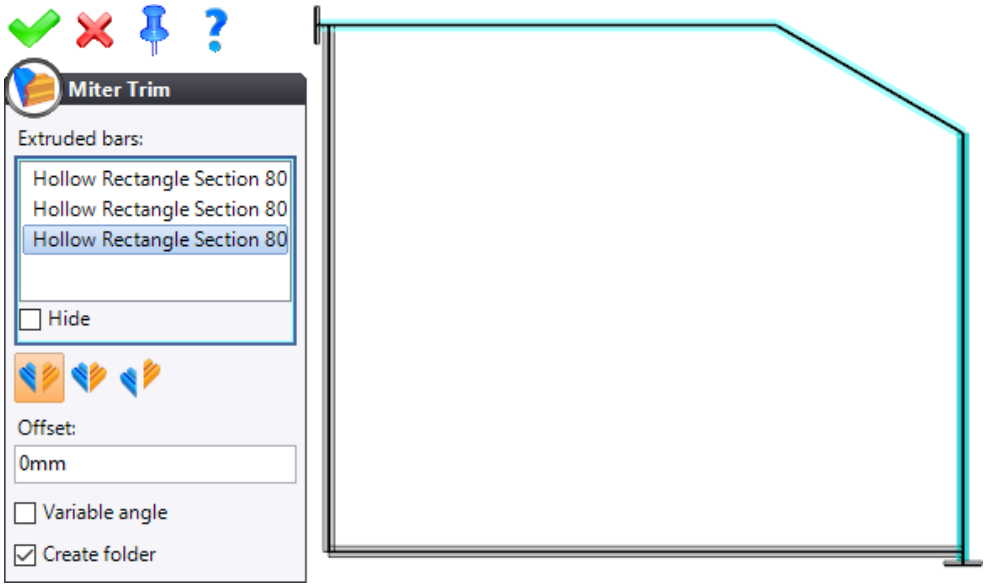
- Repeat the operation on the following extruded bar.



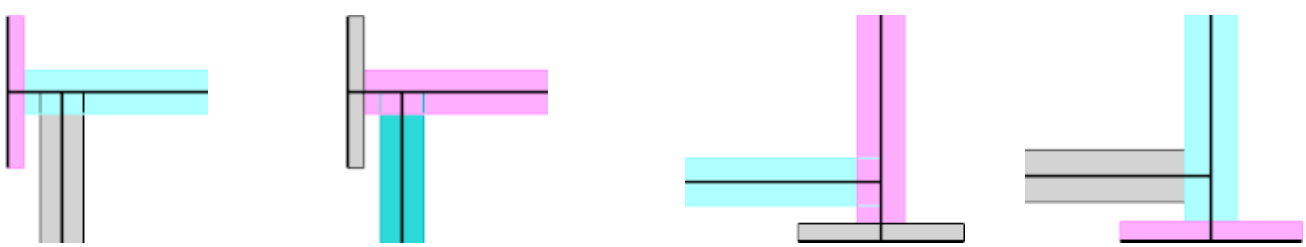
You should obtain the following result.



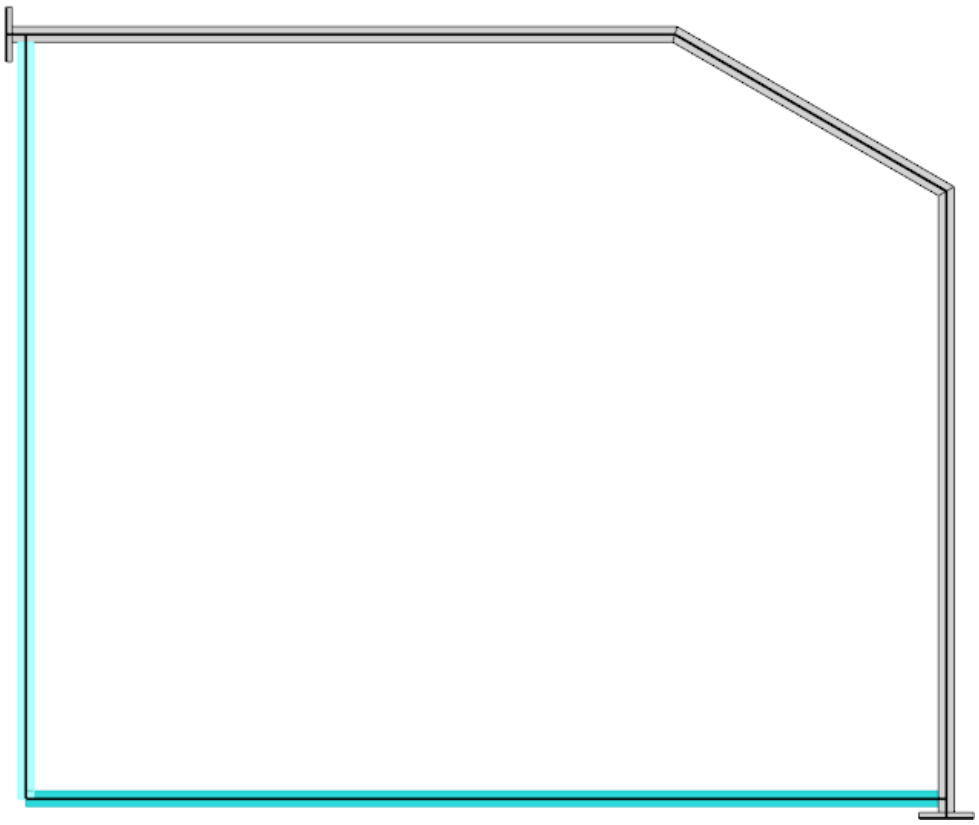
- **Save** the document (**Ctrl** + **S**).
- Right-click on one of the extruded bars and select the **Miter Trim** command.
- Select the three extruded bars as shown below.



- Perform **main trims** on the following extruded bars.



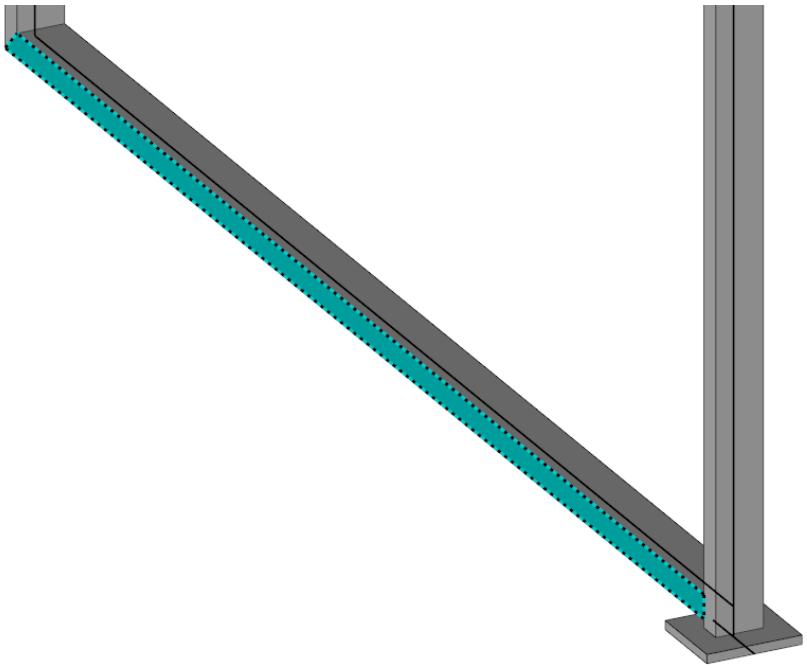
- Finish the operation by performing a miter trim.



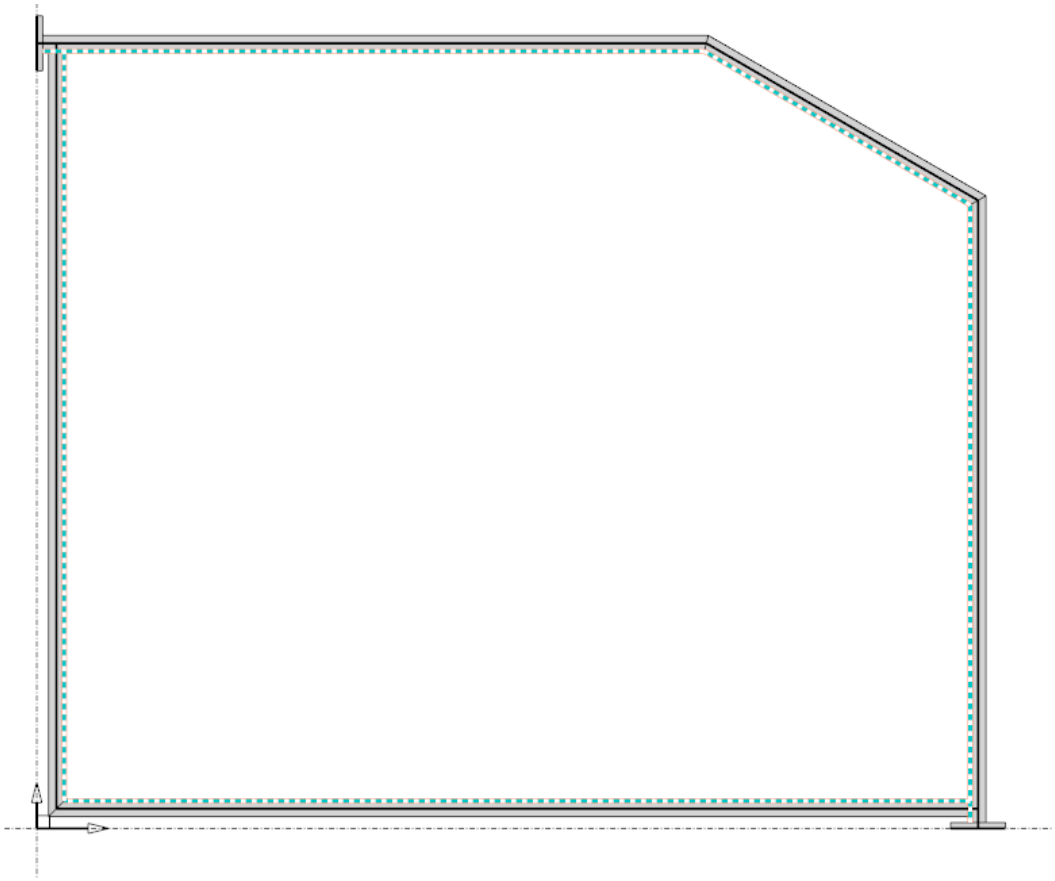
Creating the filling

We will now build the internal structure that will support the sheet metal parts.

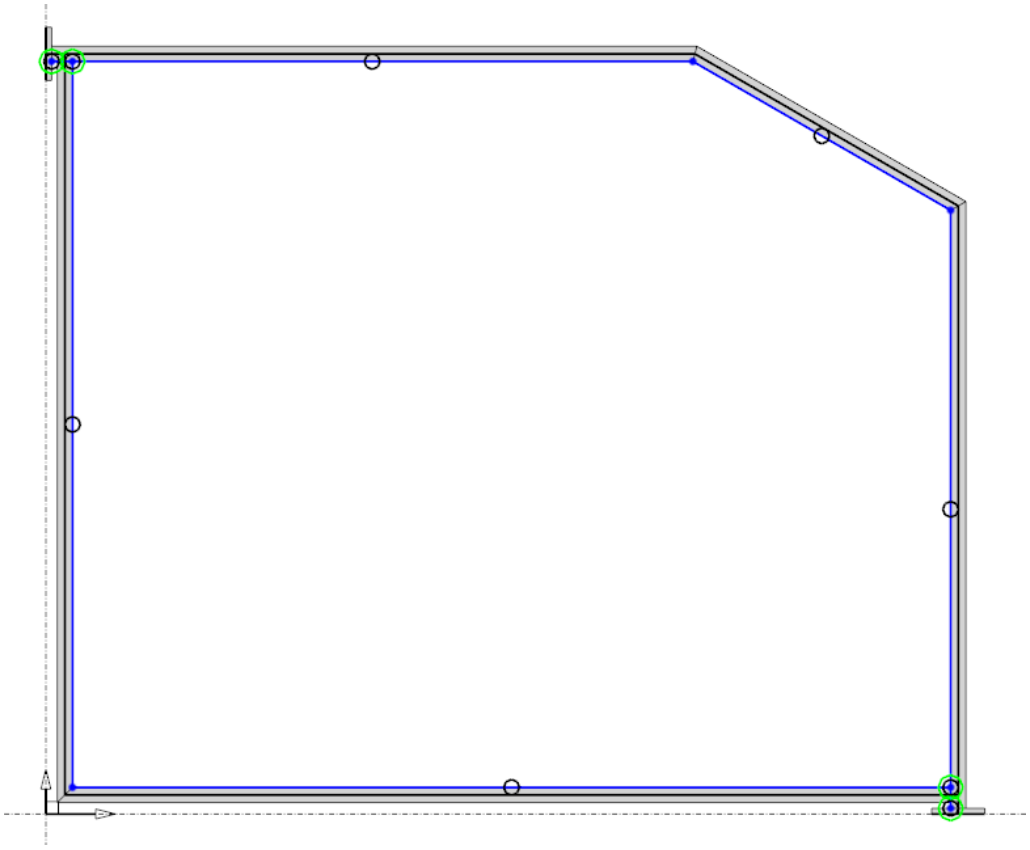
- Create a new sketch on the following plane.



- Select the **Project** command and select the following edges to form a frame.

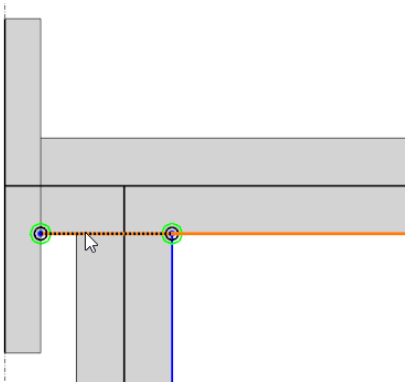


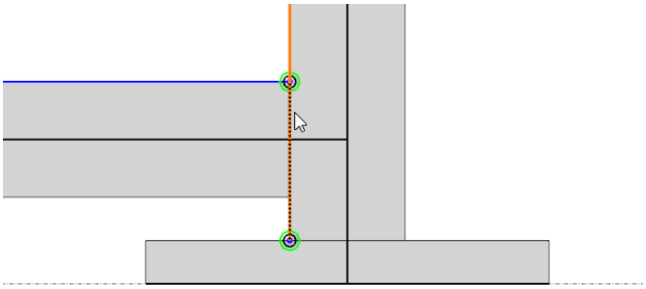
Once the operation has been confirmed, you should obtain the following result.



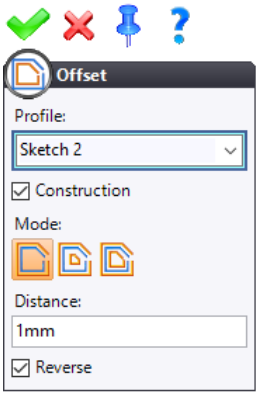
Reminder: The vertices circled in green indicate the openings of a profile.

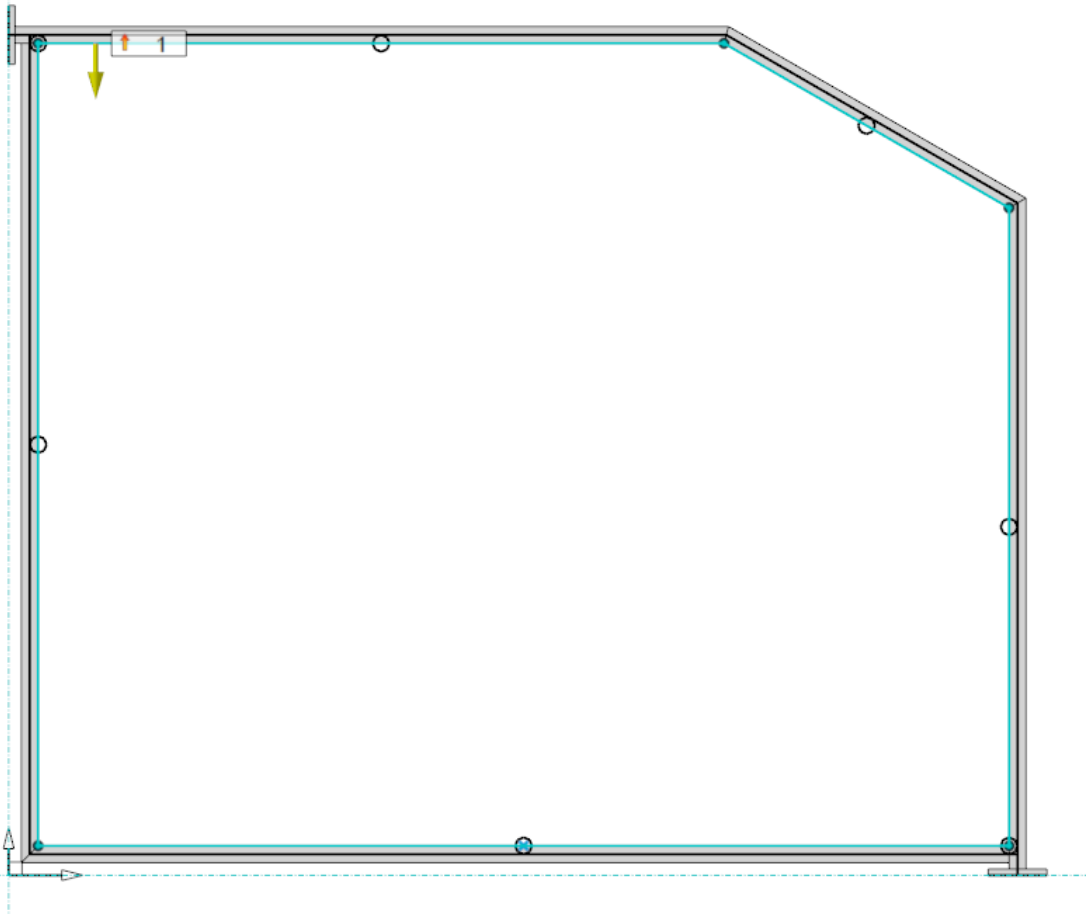
- Select the **Trim** command and click on the segments as shown below.



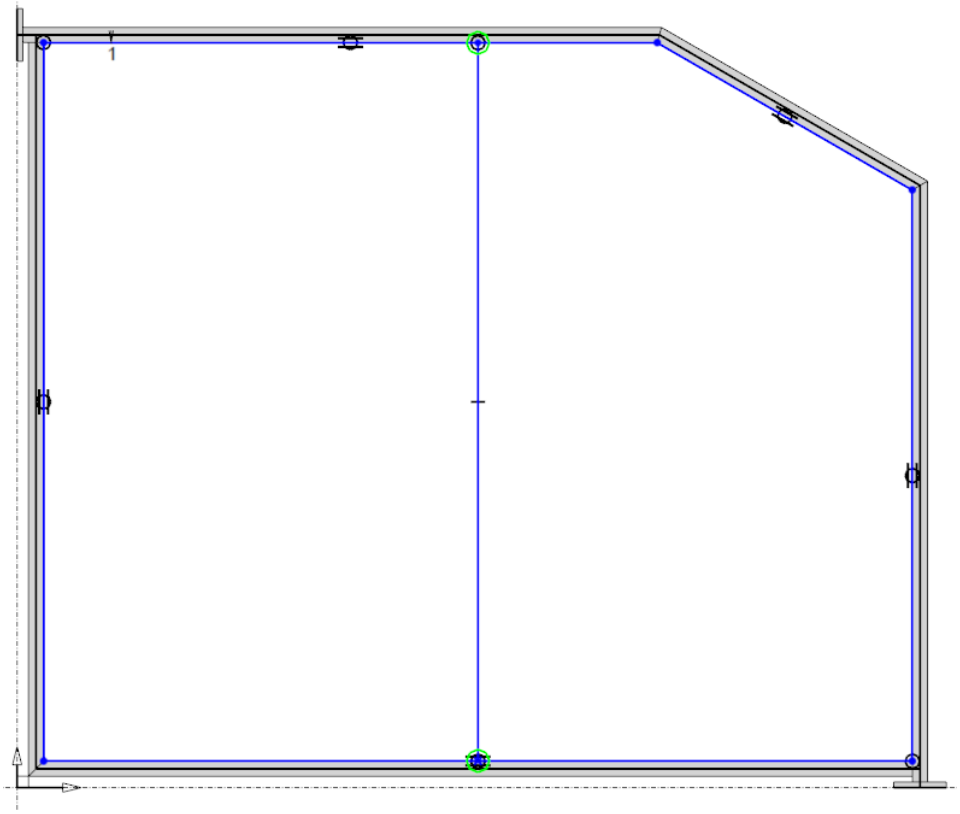


- Select the **Offset** command.
- Select the frame and enter a **distance** of *1mm*.



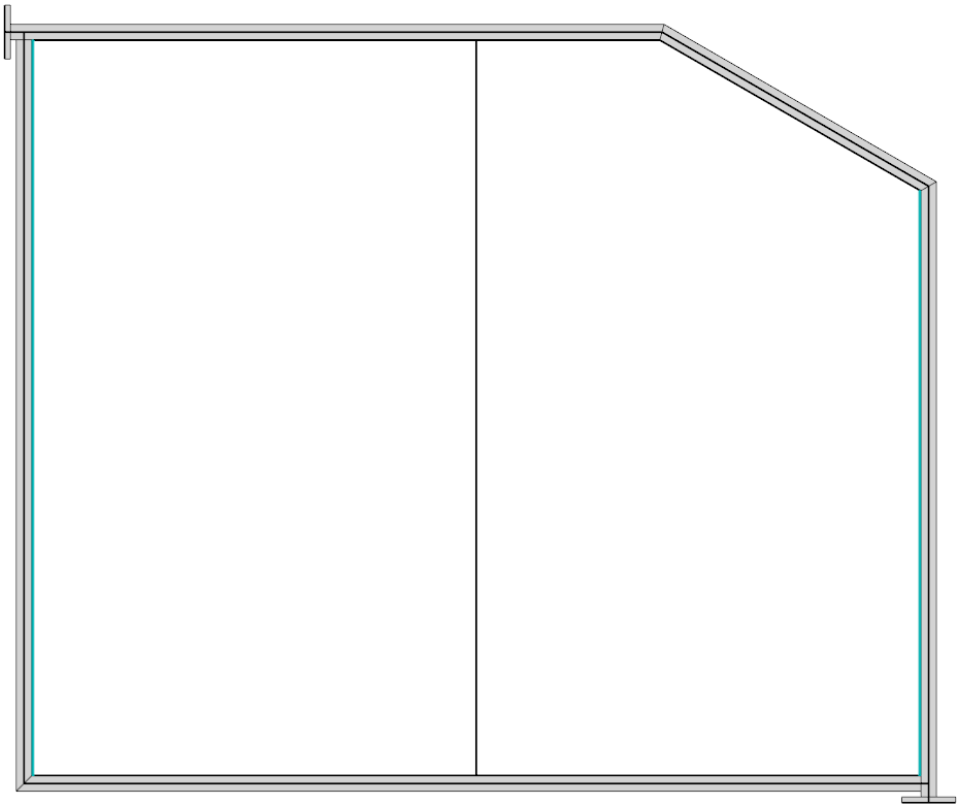


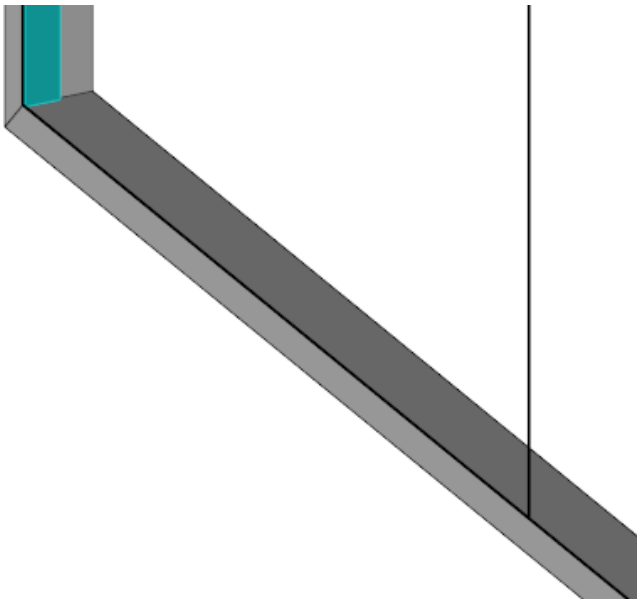
- Add a segment centered on the lower segment.



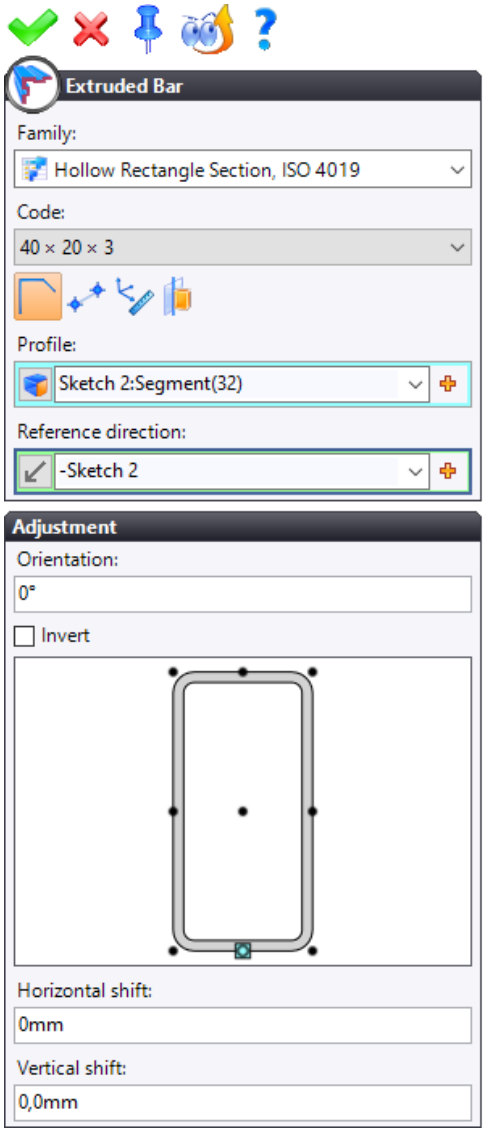
- **Confirm** the sketch.

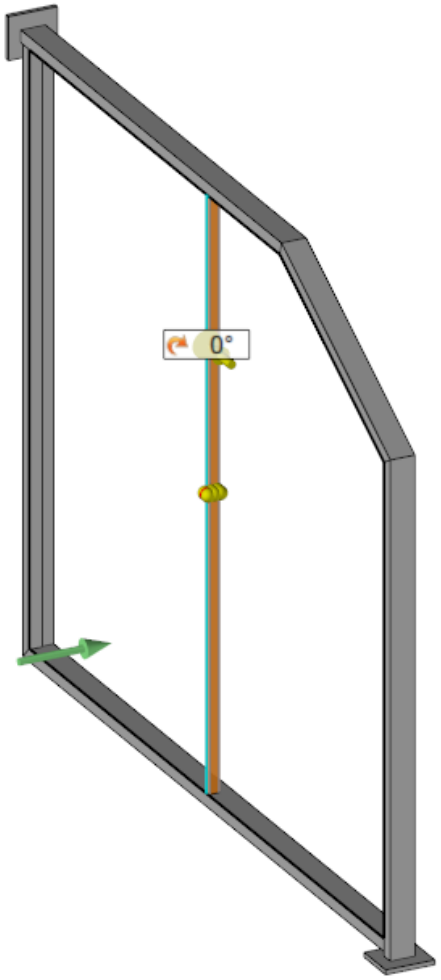
- Position two **Full Flats, NF A 45-005** extruded bars using the **40 x 5** code on the following segments.





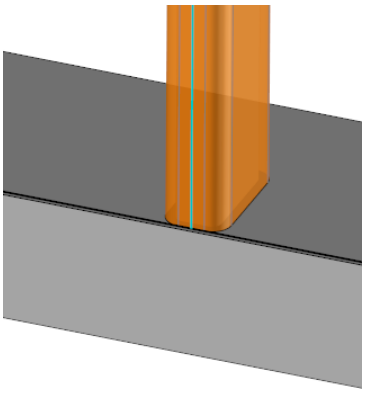
- Add a **Hollow Rectangle Section, ISO 4019** extruded bar using the **40 x 20 x 3** code on the center segment.



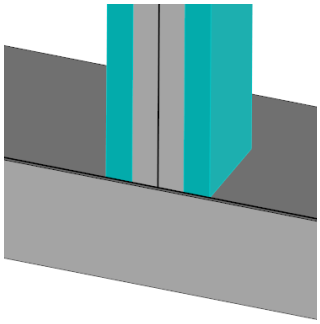


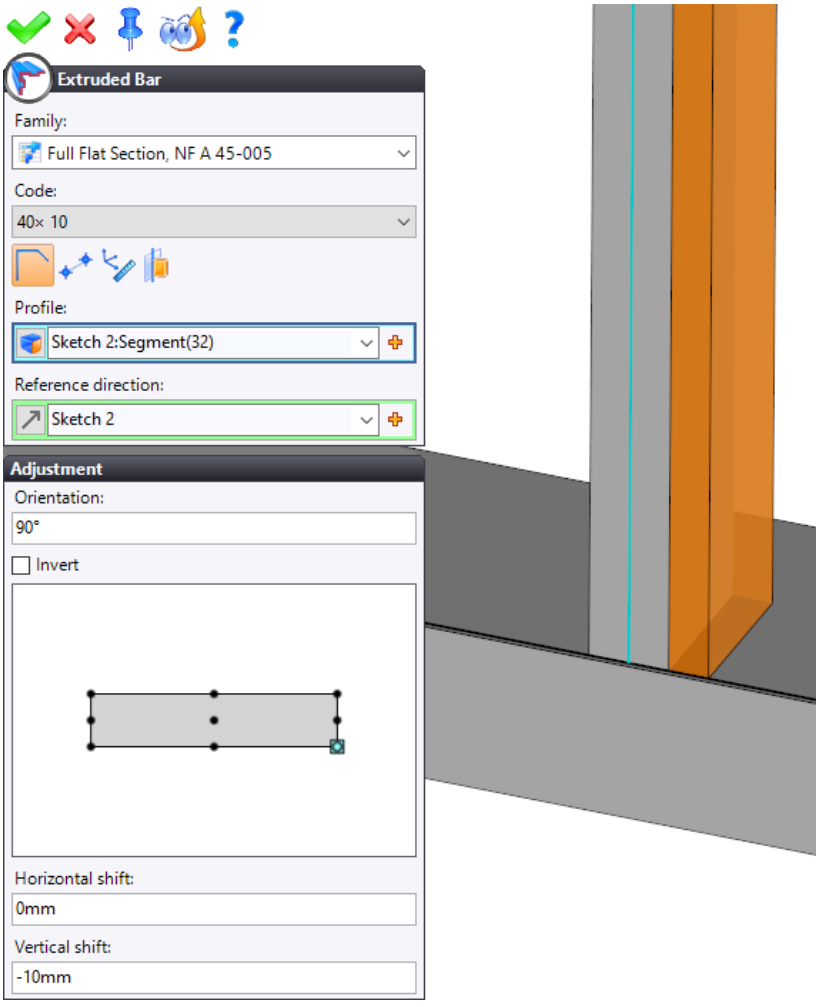
- Use the **Orientation** field to adjust the positioning angle.

The extruded bar must be centered on the segment.



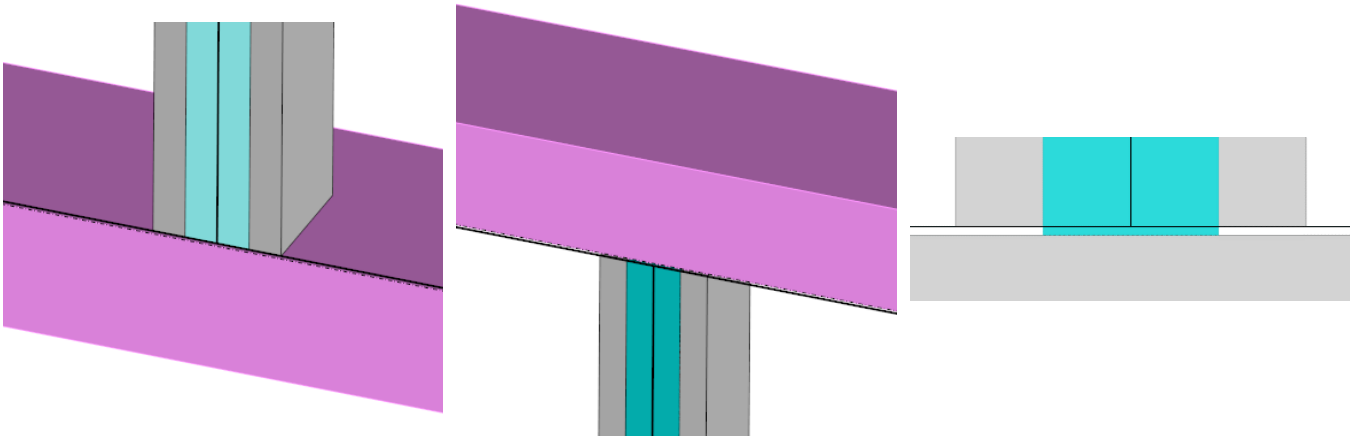
- Add two **Full Flat, NF A 45-005** extruded bars using the **40 x 10 code** on either side of the tube.
- Add a *10mm* offset if you are using the center segment of the sketch as the positioning profile.





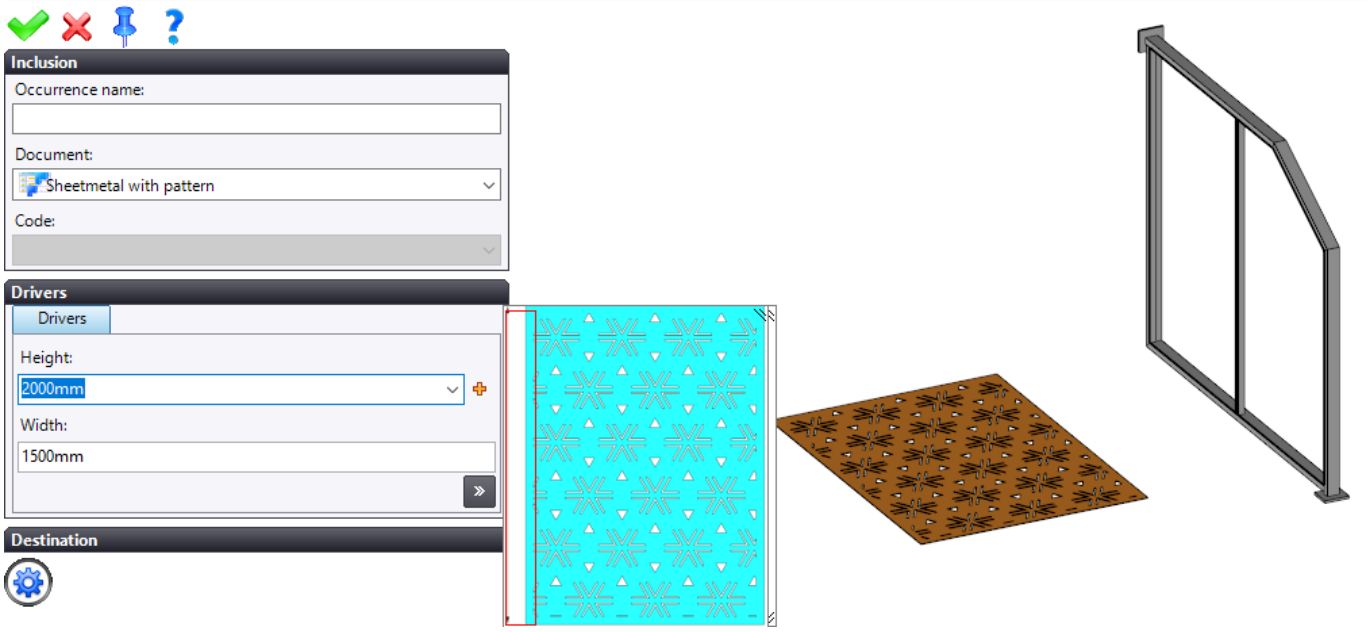
Since the rectangular extruded bar will be welded to the frame, you have to extend it at the ends so that it will be in contact with the frame.

- Right-click on the extruded bar and select the **Main Trim** command.
- Perform the operation on the top and bottom.



- From the Project tree, open the *1- Privacy screen > Privacy screen components > Sheetmetal with pattern* folders, then drag and drop the **Family** document named *Sheetmetal with pattern* into the assembly's graphics area.

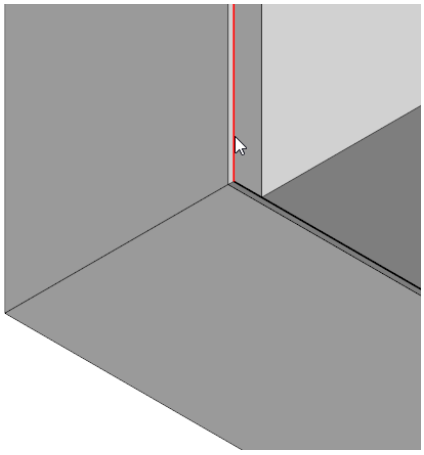
Note: Families are parameterized components in **TopSolid**, which have variables called "drivers". These drivers can be of any type (digital, text, material, coating, etc.).



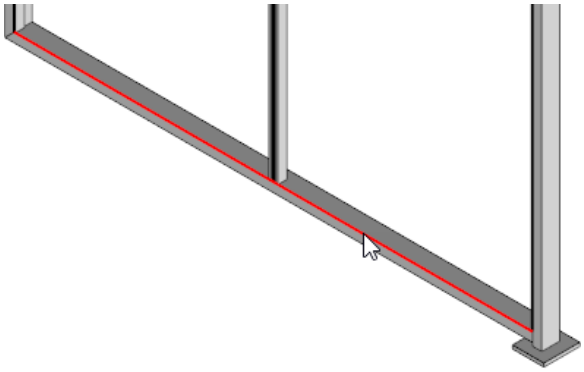
In this case, the **Sheetmetal with pattern** component allows you to select its height and width. An image is available to help you understand what these values represent and can be hidden using the » icon.

You can synchronize the **Height** and **Width** values with the frame. Modifying the dimensions of the frame will automatically modify the sheet metal part.

- In the **Height** field, click on the icon and select the **Associative value** option.
- Select the left segment of the sketch.



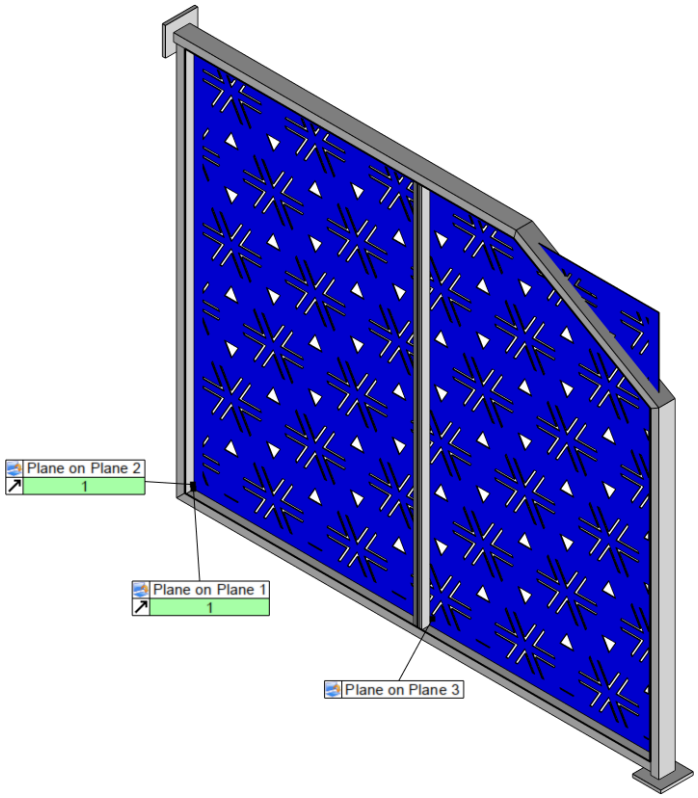
- Repeat the operation with the **Width** field by selecting the horizontal segment of the sketch.



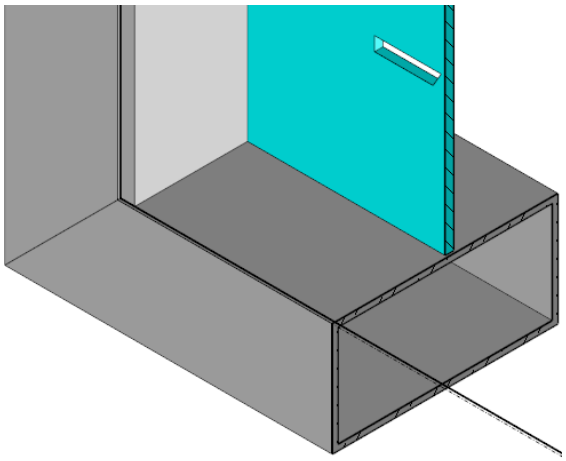
- Click on to **confirm** the inclusion.

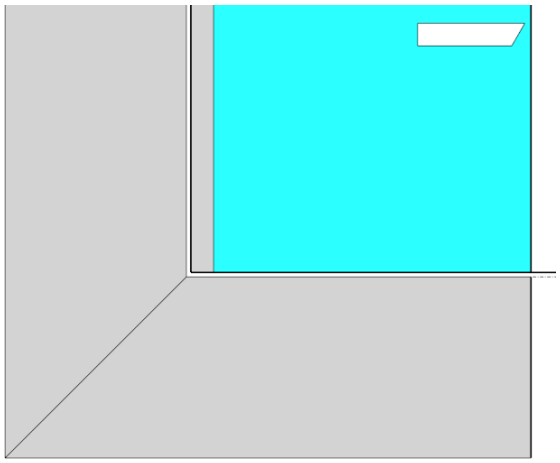
TopSolid switches to the positioning context.

- Add constraints to position the sheet metal part as shown below.

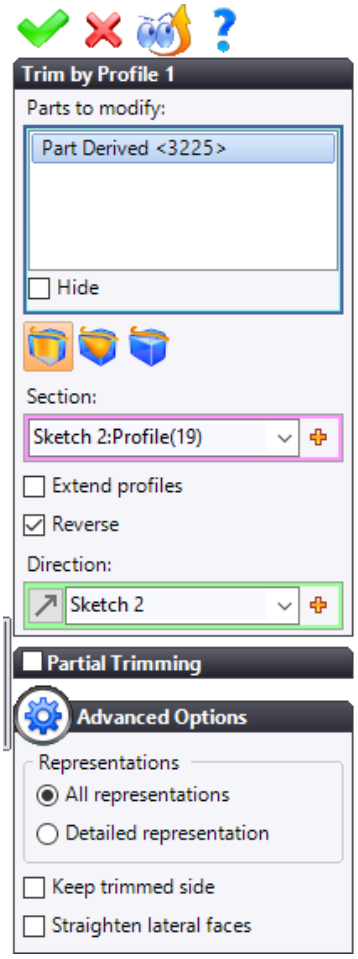


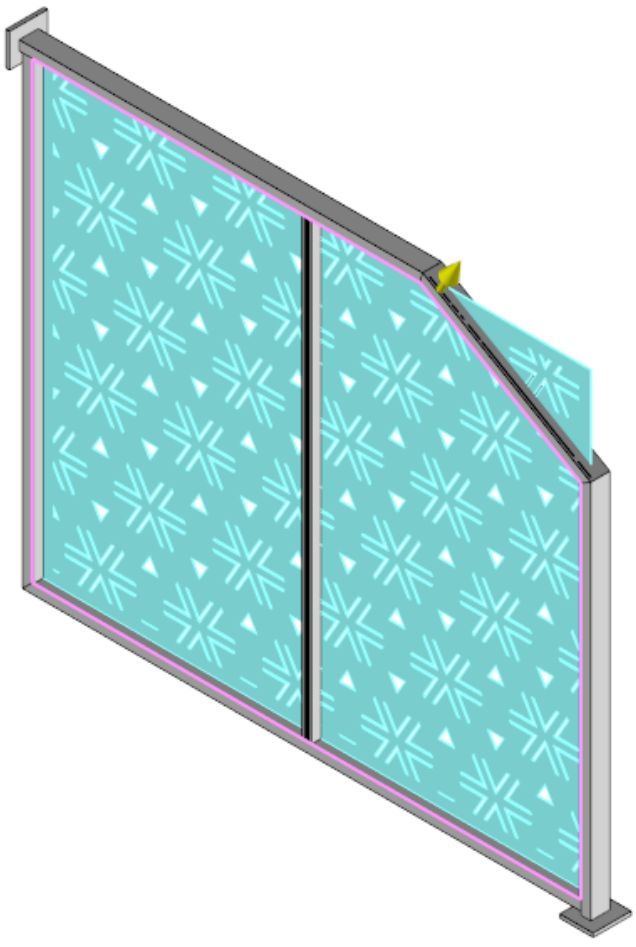
- Enter *1mm* offsets to keep a clearance.





- **Confirm** the positioning.
- Select the **Modeling** > **Trim by Profile** command.
- First, select the sheet metal part as the **part to modify**, uncheck the **Straighten lateral faces** advanced option, then click in the **Section** field and select sketch 2's frame using the rotary picking technique.



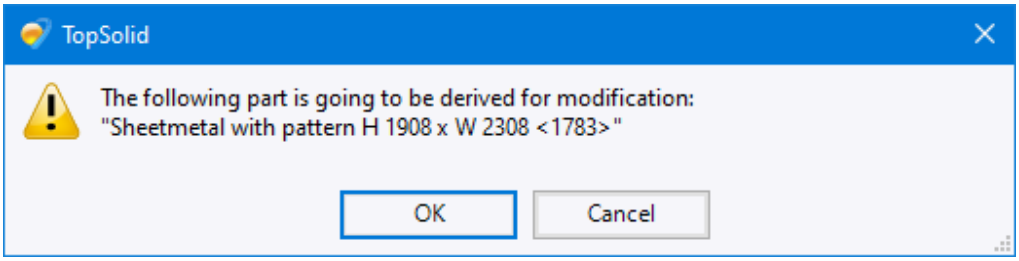


Note: Deactivating the straightening of the side faces is not mandatory, but this operation requires significant resources in the case of perforated sheet metal parts. In fact, **TopSolid** tries to straighten each face of the part and spends an unnecessary amount of time on each opening. It is therefore advisable to uncheck the box when it is not necessary.

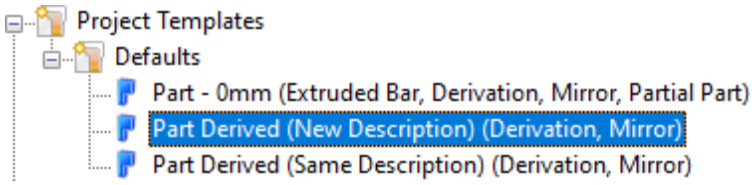
- Click on to **confirm** the trimming operation.

TopSolid indicates that the part must be derived for modification.

- Click on **OK**.

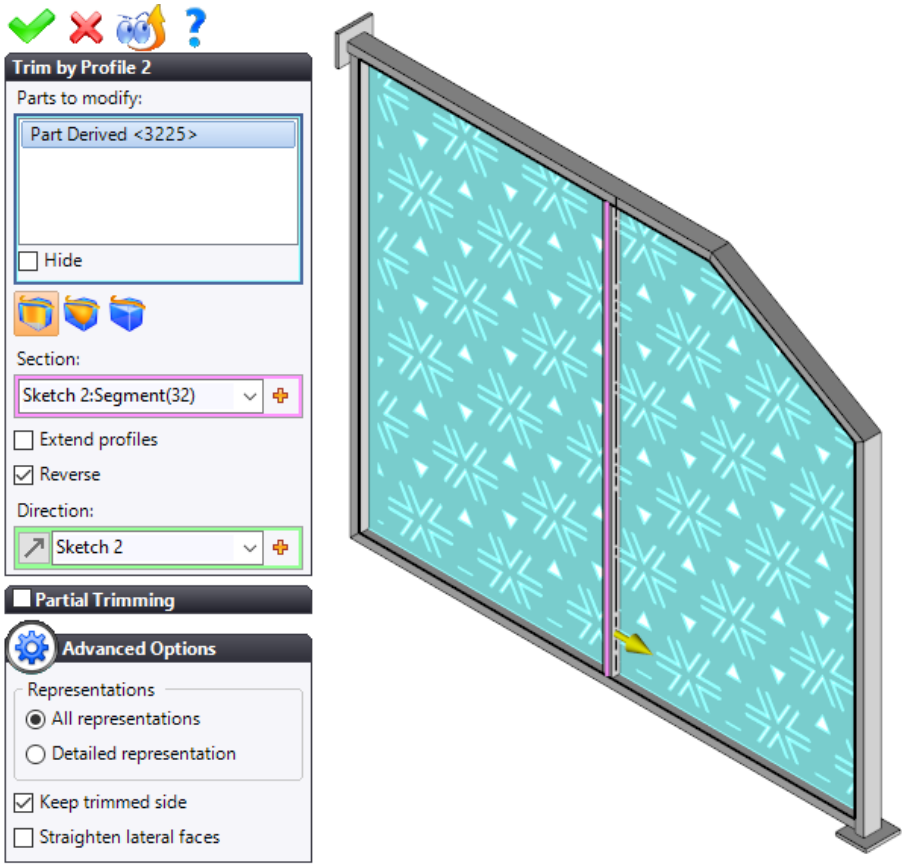


- Select the derivation template as shown below.



This template will allow you to modify the description of the sheet metal part. In this way, you will have a left sheet metal part and a right sheet metal part.

- Select the **Modeling** > **Trim by Profile** command.
- First, select the sheet metal part as the **part to modify**, uncheck the **Straighten lateral faces** box and check the **Keep trimmed side** box, then click in the **Section** field and select the vertical center segment of sketch 2 using the rotary picking technique.



- Select the **Partial Part (New description)** document template to be used for the trimmed side. A partial part means that a single part becomes several parts.

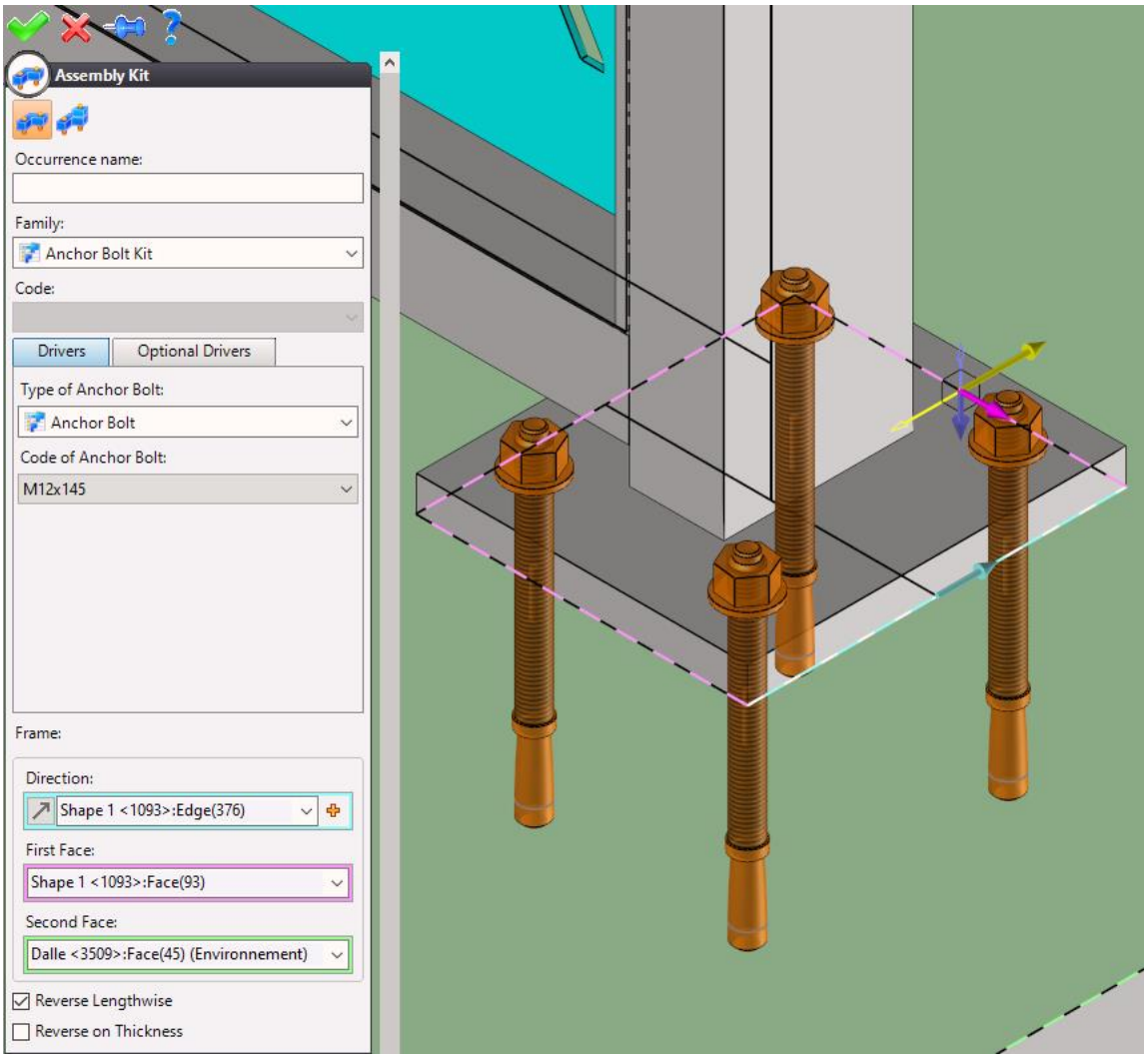
Integrating fastenings

We will now integrate the different fastening elements using the **Assembly Kit** command.

- From the Entities tree, display the background document.
- From the Project tree, open the *1- Privacy screen > Privacy screen components > Anchor Bolt Kit* folders, then drag and drop the **Family** document named *Anchor Bolt Kit* into the assembly's graphics area.

The **Assembly Kit** command starts automatically.

- Select **M12 x 145** as the code, select an edge of the lower right-hand plate as the **direction**, and then select the contact face between the plate and the environment as the **first face**.



Note: Assembly kits are tailor-made components that can be designed by everyone. The method used to create them will be explained throughout the training course.

The calculation of the assembly kit distribution is based on the Eurocodes. The distribution depends on the diameter of the anchor bolt, the thickness to be clamped and the dimensions of the element to be fixed. You can adjust the result in the **Optional Drivers** tab.

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

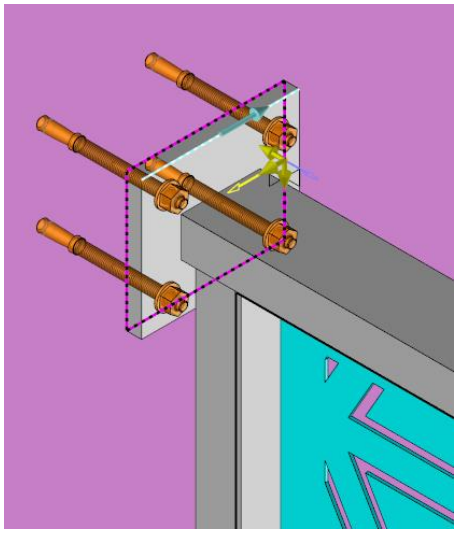
TopSolid then offers to create the associated process.

- Select **Slotted Clearance Hole** from the **Process** drop-down list.

This process has drivers that can be modified as needed. The adjustments and dimensions of the oblong hole are also based on the Eurocodes.

Warning: The Eurocodes start from size M12. Therefore, this process cannot be used on smaller diameters. However, you can edit this by copying and editing the **Clearance Hole Standard EN 1090-2** documents (**TopSolid Mechanical** library), as well as the desired processes.

- Repeat the procedure on the upper plate.

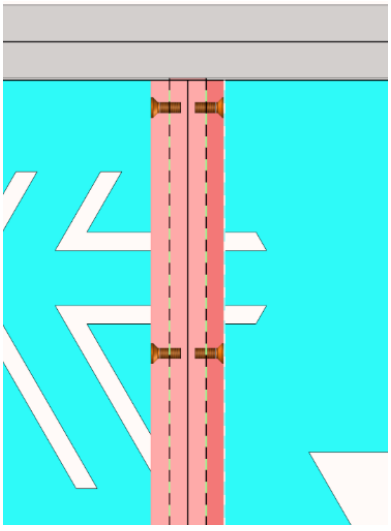


We will now fix the two flat sections on the central rectangular extruded bar. To save time, we will install several kits at the same time.

- From the Project tree, open the *1- Privacy screen > Privacy screen components > Screw Kit* folders, then drag and drop the **Family** document named *Screw Kit* into the assembly's graphics area.

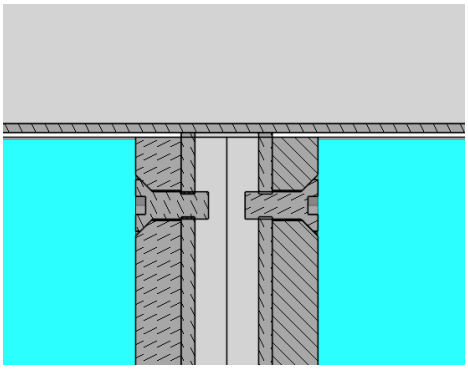
The **Assembly Kit** command starts automatically.

- Select the **Multiple** mode.
- Select the **M6 x 16** code, select a vertical edge of one of the flat sections as the **direction**, then select the two flats and the rectangular tube as the **parts to assembly**.



- In the **Optional Drivers** tab, enter *5* in the **Number of screws on height** field.
- Click on to **confirm** the inclusion and the process.

You should obtain the following result.

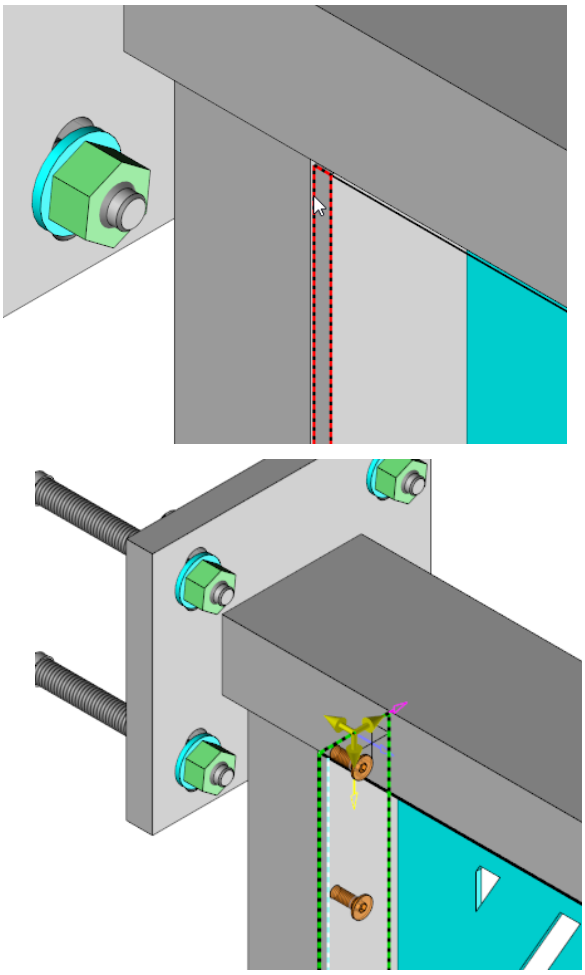


We will repeat the operation on the two flat sections that are fixed to the outer frame.

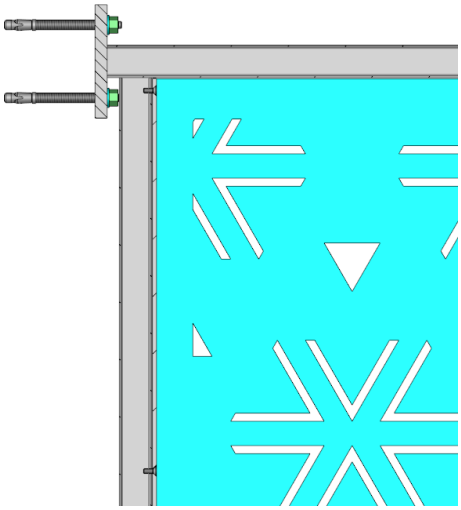
- From the Project tree, open the *1- Privacy screen > Privacy screen components > Screw Kit* folders, then drag and drop the **Family** document named *Screw Kit* into the assembly's graphics area.
- Select the **M6 x 16** code and select a vertical edge of one of the flat sections as the **direction**.

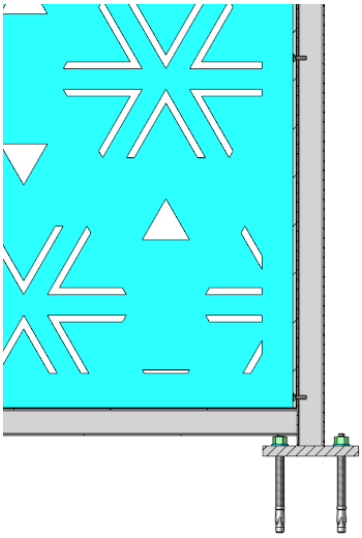
By default, the assembly kits automatically detect the contacting faces. In this case, a clearance exists between the plane and the outer frame.

- To force the system, click in the **First face** field, position the cursor as shown below, then select the contact face between the flat section and the tube using rotary picking.
- Repeat the operation with the **Second face** field.



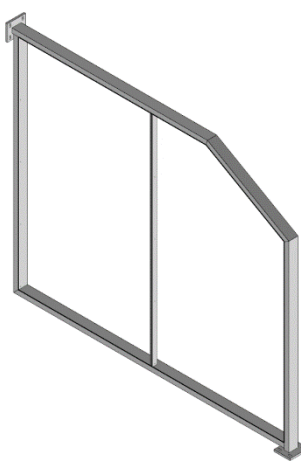
- In the **Optional Drivers** tab, enter *5* in the **Number of screws on height** field.
- Click on to **confirm** the inclusion and the process.
- Repeat the operation on the other side.

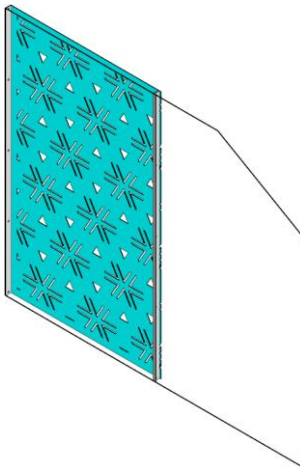


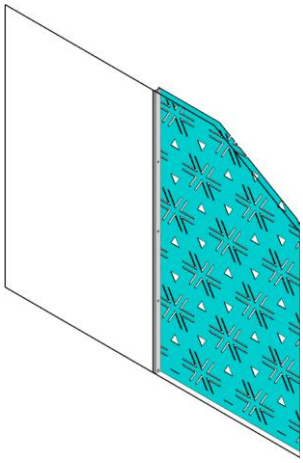


Defining the sub-assemblies (in-place assemblies)

- Select the **Modeling** > **In Place Assembly** command.
- Create the following three sub-assemblies.

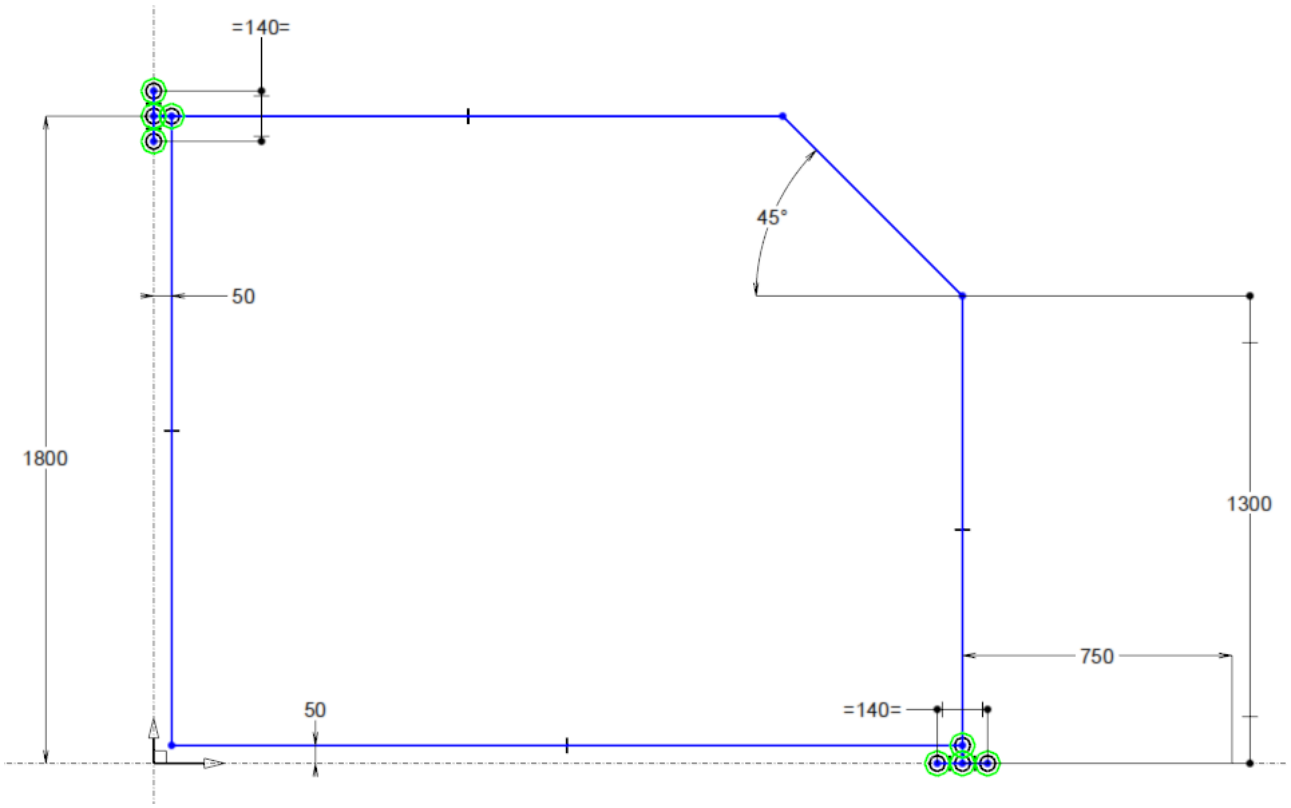






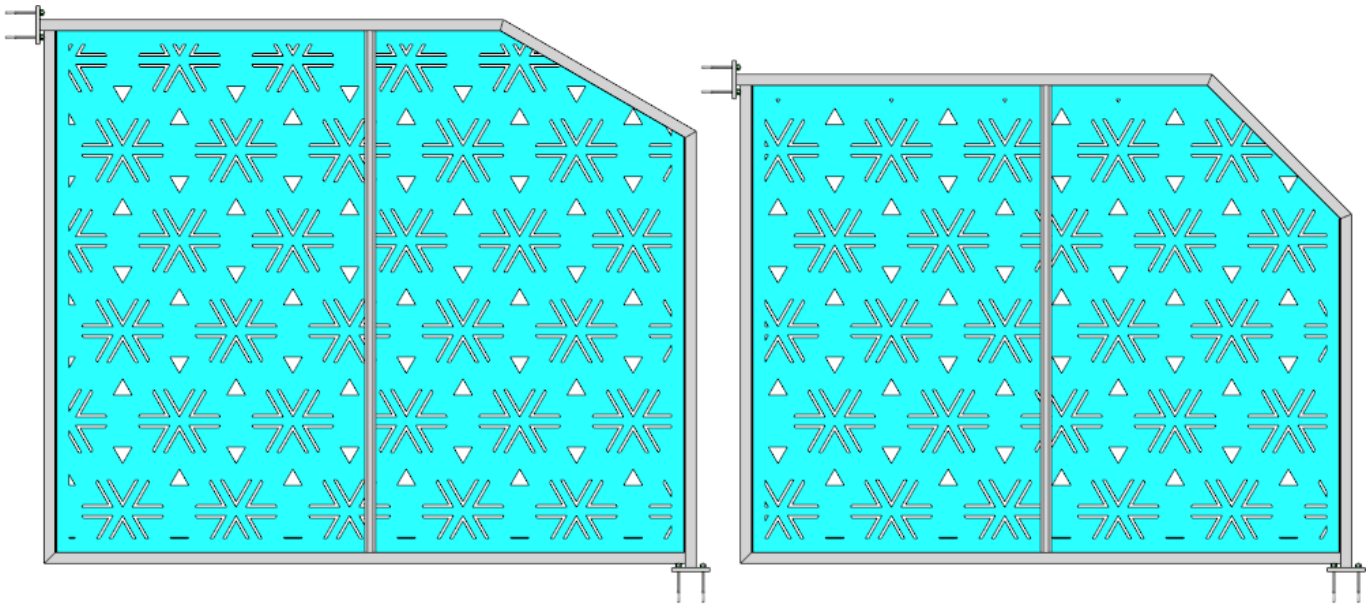
The modeling is now complete. You can vary the environment as well as the different sketches to reset the dimensions of the privacy screen as you wish.

- Edit sketch 1 and modify the following dimensions:
  - 2000mm → 1800mm
  - 1600mm → 1300mm
  - 150mm → 250mm
  - 30° angle → 45°



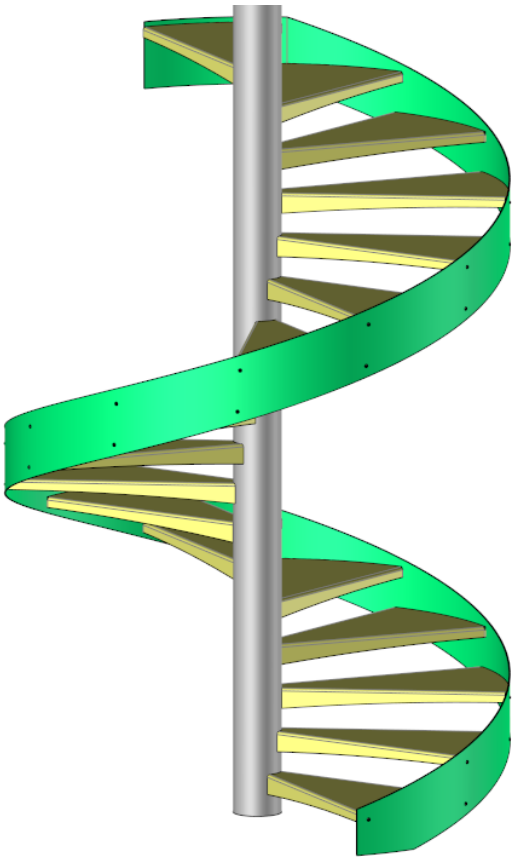
- **Confirm** the sketch.

TopSolid recalculates the entire privacy screen (structure, sheet metal part, screw distributions, etc.).



Part 2 - Designing a Spiral Staircase

The purpose of this exercise is to draw a parametric base of a spiral staircase as shown below.



Concepts addressed:

- Parameterization
- Sheet metal work
- Helical pattern

Creating the basic parameters

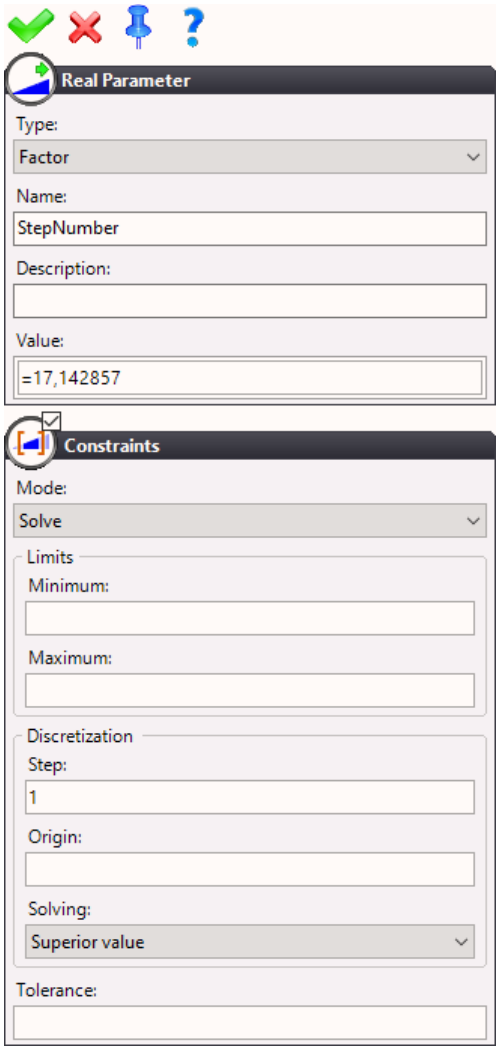
- From the Project tree, create a new folder named *2- Spiral staircase*.
- In this folder, create a new **Assembly** document named *Spiral staircase*.
- From the Entities tree, right-click on the **Parameters** folder and select the **Real Parameter** command.

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

- Select **Length** as the type, enter *TotalRise* as the **name** and a **value** of *3000mm*.
- **Pin** the command's dialog box and click on to **confirm** the operation.
- Create a *StepRise* parameter using the **Length** type and enter a value of *175mm*.
- Click on to **confirm** the operation.
- Create a *StepNumber* parameter using the **Factor** type and enter the following formula as the value: *TotalRise/StepRise.*

TopSolid integrates a system to control and constrain the results.

- Select the **Constraints** dialog box.
- Select the **Solve** mode. The value will be automatically adjusted.
- In the **Discretization** section, enter *1* in the **Step** field and select **Superior value** from the **Solving** drop-down list.



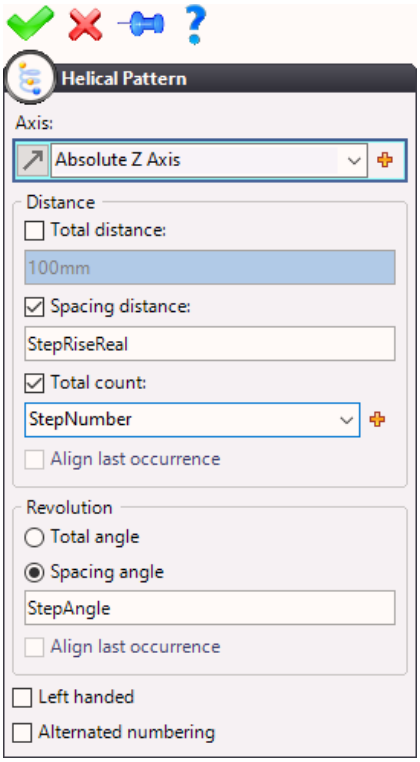
- Click on to **confirm** the operation.

Factor parameters contain values without units and can be used as coefficients, to manage quantities, etc.

- Create the following parameters.

Parameter type	Name	Value/formula
Length	*StepRiseReal*	*TotalRise/StepNumber*
Length	*Headroom*	*2000mm*
Factor	*RevolutionNumber*	*TotalRise/Headroom*
Angle	*StepAngle*	*360 * RevolutionNumber/StepNumber*
Length	*ExternalDiameter*	*2000mm*
Length	*TreadingDiameter*	*ExternalDiameter * (2/3)*
Length	*ProjectedTreadingLength*	*pi * TreadingDiameter*
Length	*StepWidth*	*ProjectedTreadingLength * RevolutionNumber/StepNumber*

- Select the **Construction** > **Patterns** > **Helical Pattern** command to create a new helical pattern and adjust the parameters as shown below.

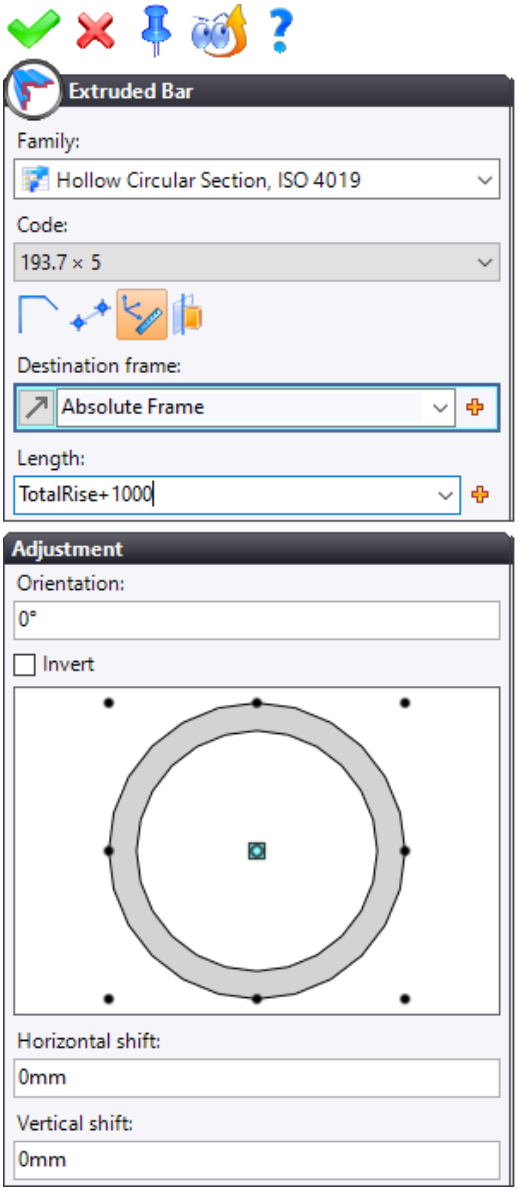


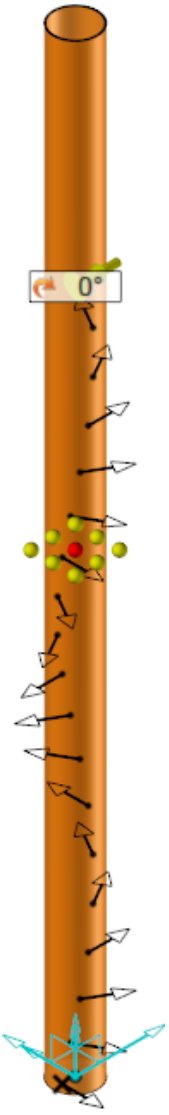
Note: Creating the pattern prior to performing the repetition offers some advantages over creating it on-the-fly when performing the repetition operation.

- This allows you to see the pattern result before designing the staircase.
- This allows you to create a parameterized base containing everything you need to create the staircase.
- If the repetition is deleted, the pattern remains.

Creating the central post and steps

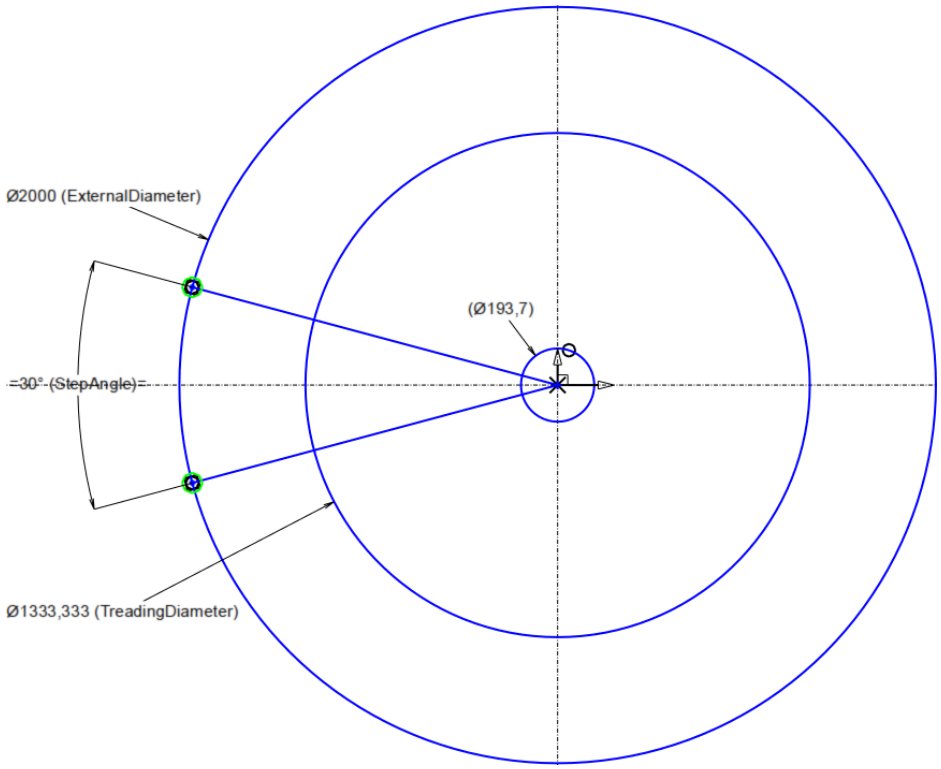
- Select the **Modeling** > **Extruded Bar** command.
- Select the **Hollow Circular Section, ISO 4019** family and the **193.7 x 5** code.
- Select the **Frame and Length** mode.
- In the **Destination frame** field, select **Absolute Frame**.
- In the **Length** field, enter *TotalRise + 1000*.





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

- Select the **Construction** > **Offset Plane** command.
- Select **XY Plane** as the **reference plane** and the **StepRiseReal** parameter as the **offset distance**.
- Right-click on the previously created plane and select the **In Place Part** command.
- Draw the following sketch.



The Ø193.7 dimension is a projection of the central post's external diameter performed using the **Project** command.

To retrieve the parameters of the assembly, you only have to:

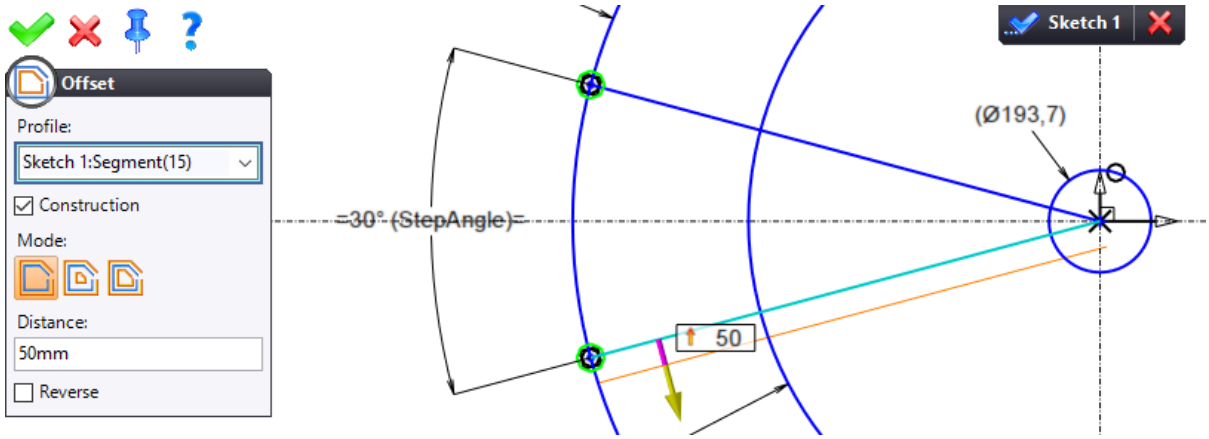
- create the dimension;
- click on the icon and select the **Real Relay Parameter** command;
- select the desired parameter from the **Parameter** drop-down list and click on to **confirm**;
- click on to **confirm** the dimension again.

Note: There is an alternative solution to the relay parameters. To retrieve a parameter from the parent assembly, you only have to enter *ParentName:ParameterName* in the **Value** field of the dimension.
Example: *Spiral staircase:TreadingDiameter*

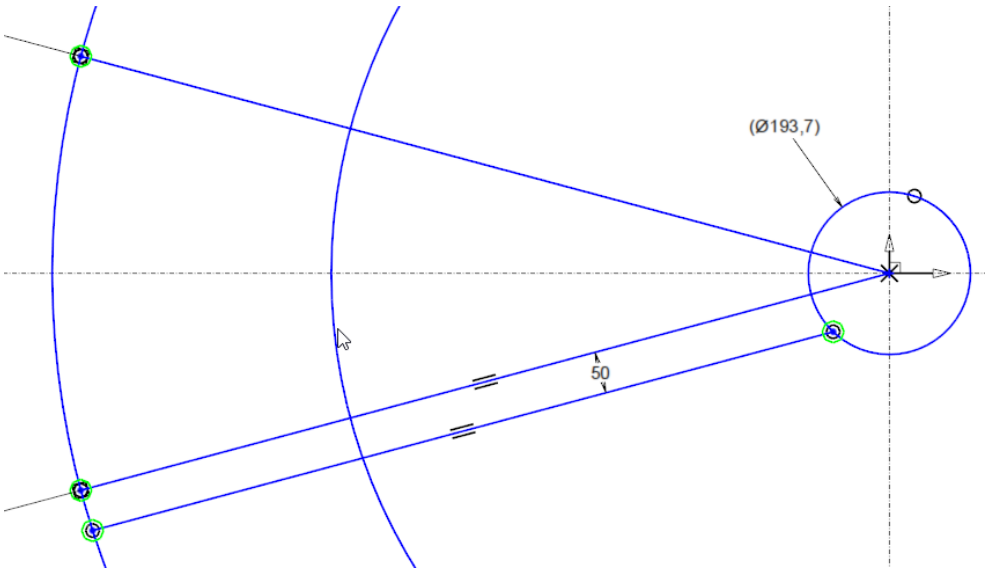
- Hide the assembly context by clicking on the icon at the top right of the graphic area, then by selecting **Hide Editing Context**.

	Hide Editing Context	Allows you to hide the assembly during in-place editing.
	Show Editing Context	Allows you to display the assembly during in-place editing without the ability to hook onto it.
	Activate Editing Context	Allows you to display and use the assembly during in-place editing.

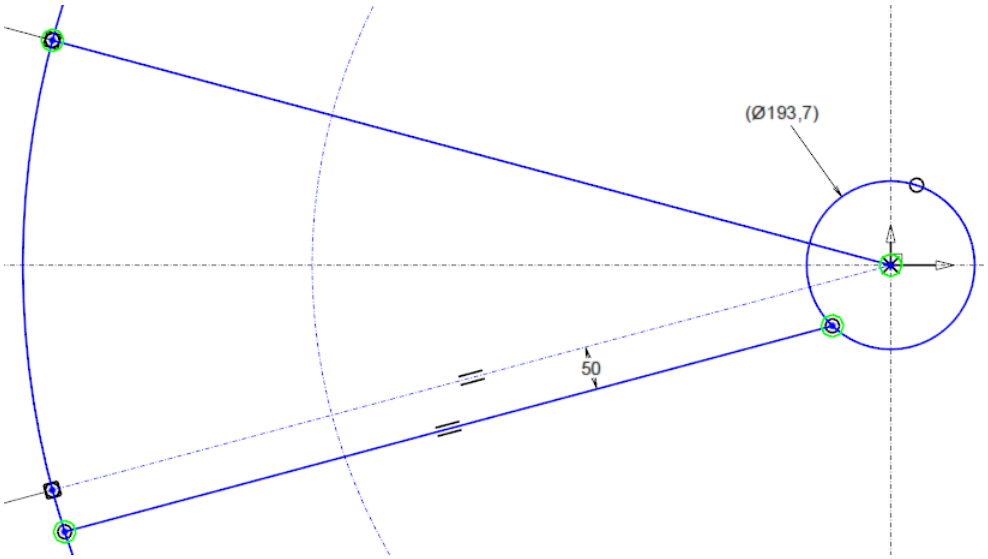
- Create an  **offset** based on the following segment. Use rotary picking to select this segment.



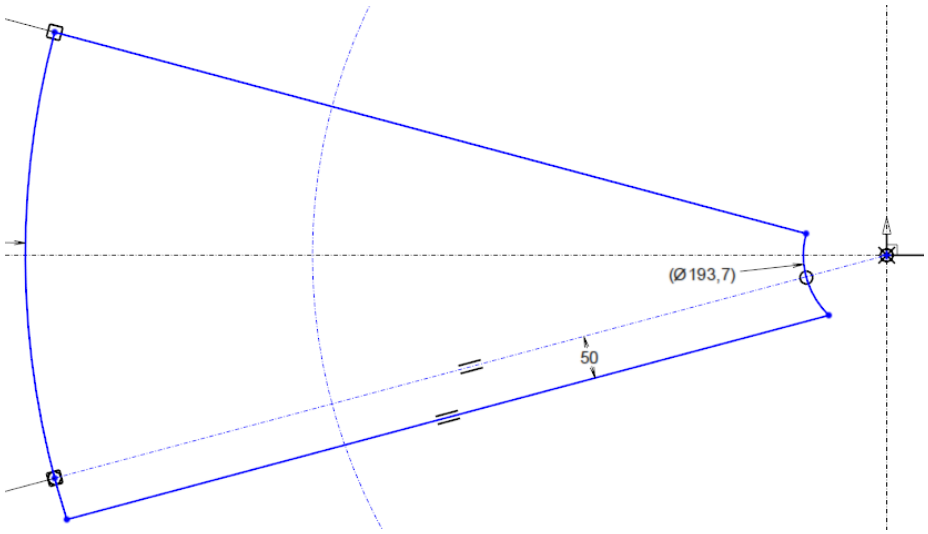
- Adjust the ends by dragging the points onto the inner and outer circles.



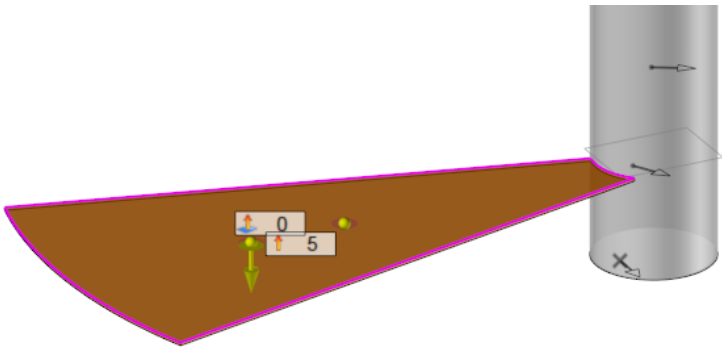
- Hold down the **Ctrl** key and select the lines as shown below, then select the  **Construction** command to convert them into construction lines.



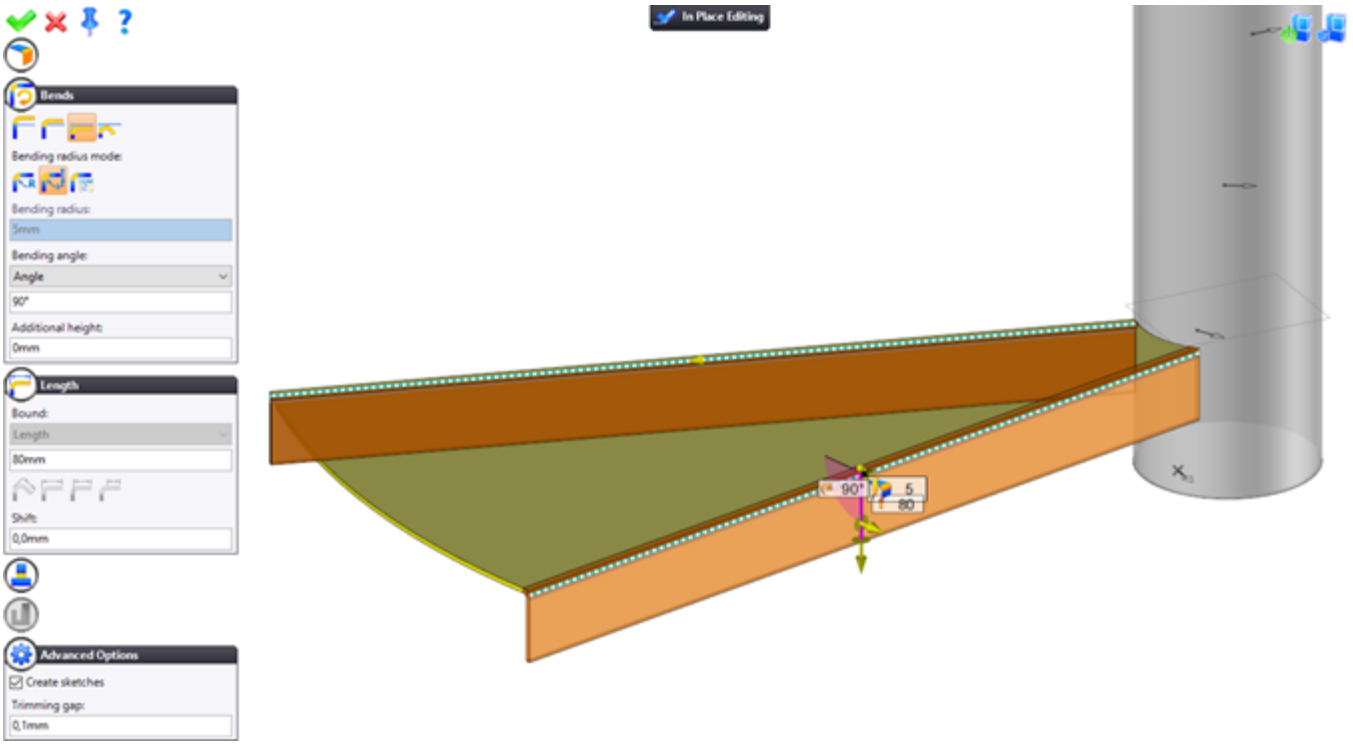
- Select the **Trim** command and delete the segments to obtain the following result.



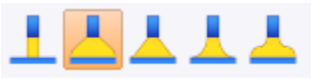
- Right-click in the graphics area with no active selection and select the **Sheet Metal on Sketch** command.
- Enter a **thickness** of *5mm* and click on to **confirm** the operation.



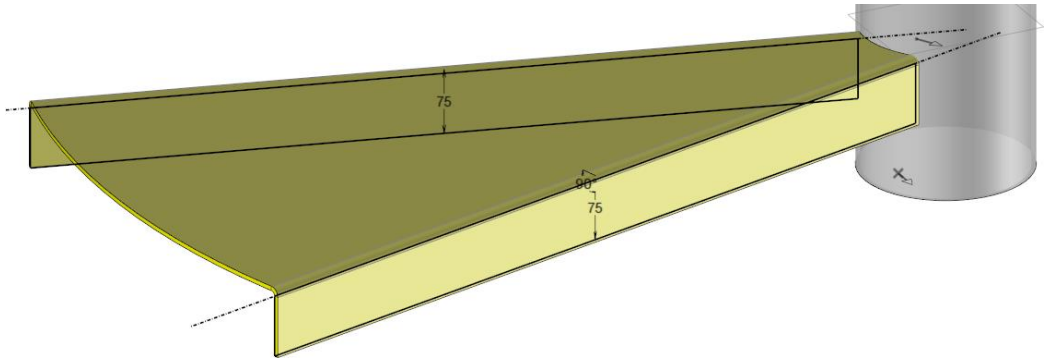
- Add two *80mm* **flanges** as shown below. Make sure you check the **Create sketches** box.



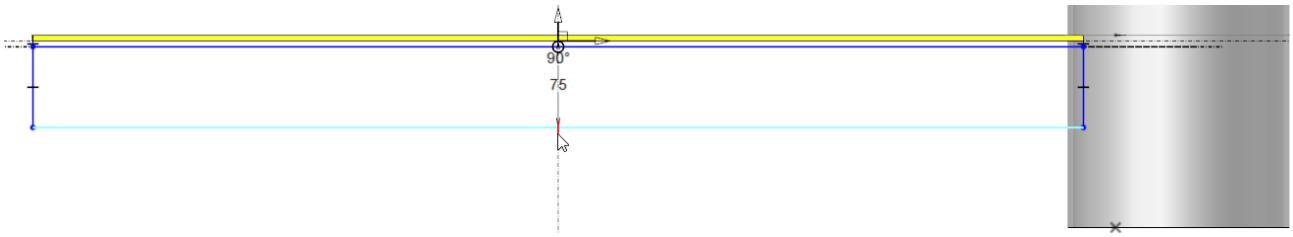
- Via the **Width** dialog box or the contextual menu, make sure that the **Bend transition** option is set to **Adjusted** mode.



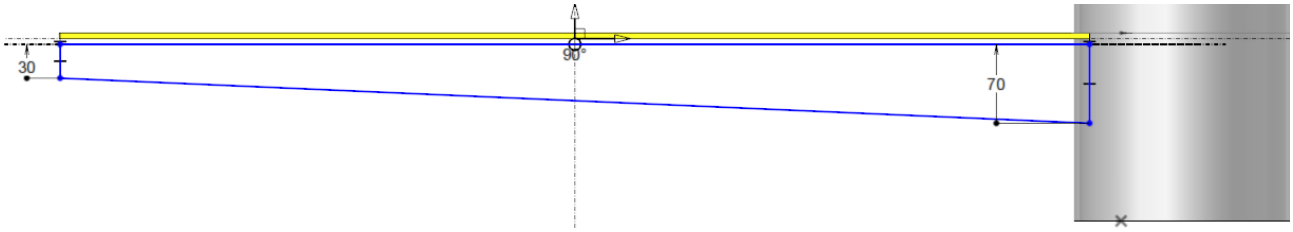
- Click on to **confirm** the operation.
- Double-click on one of the previously created flanges. A sketch is displayed.



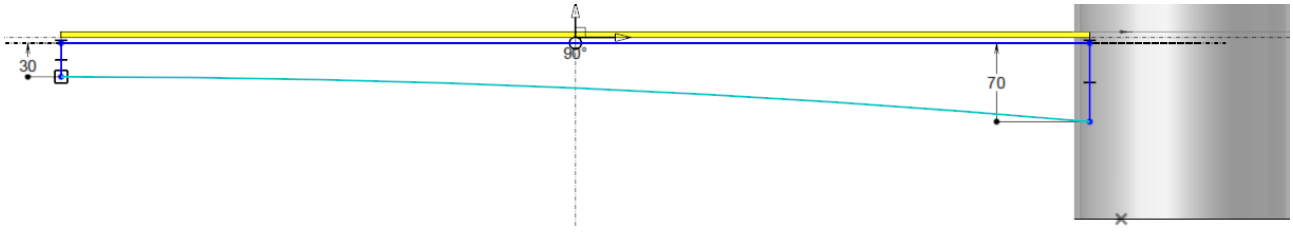
- Right-click on the sketch and select the **Edit** command.
- Remove the orientation constraint as shown below, as well as the 75mm dimension. To do this, select the element and press the **Del** key, or right-click on the element and select the **Delete** command from the contextual menu.



- Offset the line so that it is no longer horizontal, then add the two dimensions as shown below.



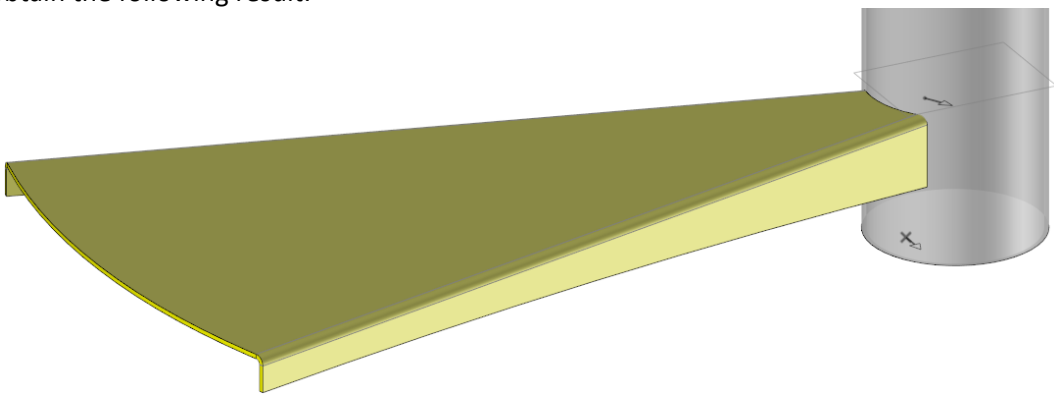
- Select the **2D Sketch** > **Deformation** command. Click on the segment as shown below. Hold down the left mouse button and move the cursor upward. The segment becomes distorted and turns into a circular arc.
- Add a perpendicularity constraint on the left vertex of the arc as shown below. To do this, right-click on the vertex and select the **Perpendicular** command.



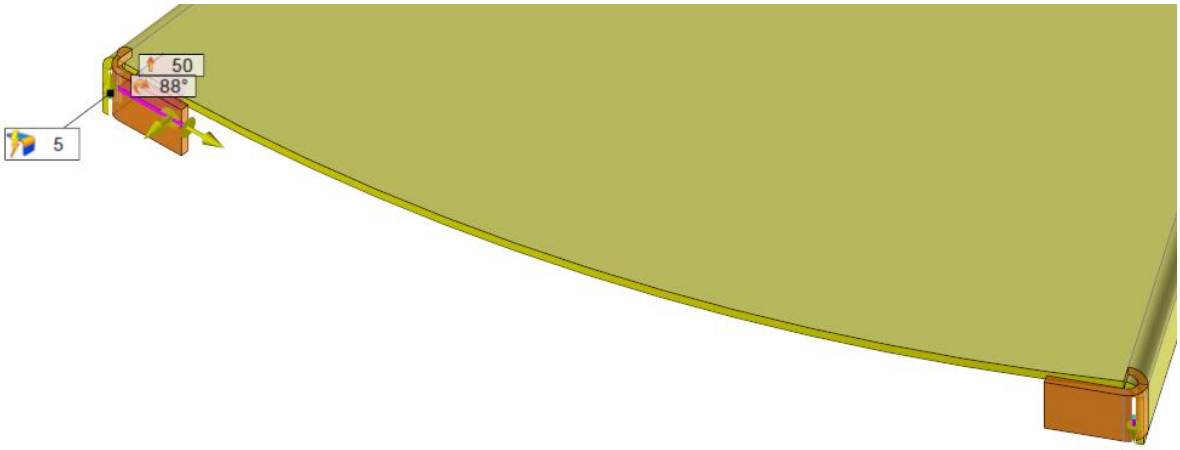
- **Confirm** the sketch.

- Repeat the operations on the second bend.

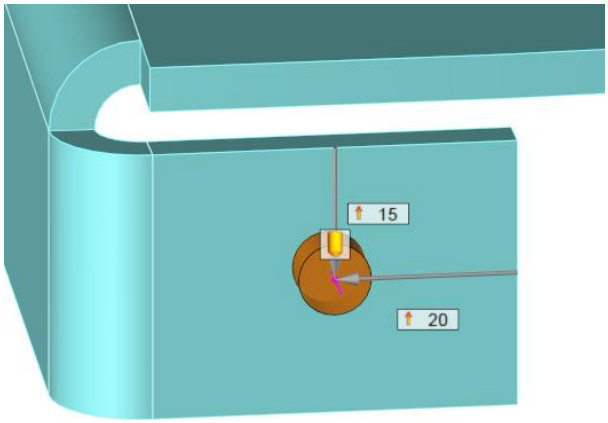
You should obtain the following result.

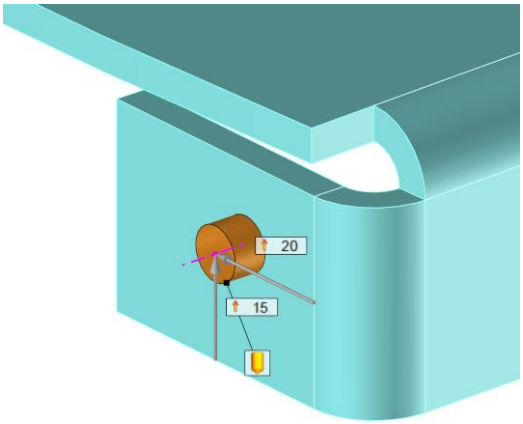


- Add two *50mm* **flanges** on the segments as shown below with an angle of *88°*.



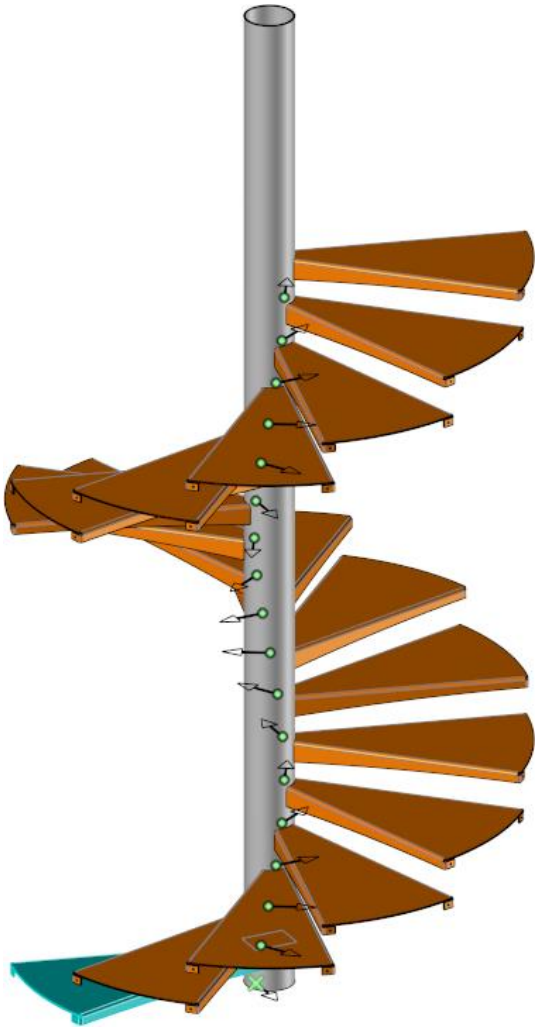
- Add a *Ø8* **drilling** centered on each of the previously created flanges.





- **Confirm** the in-place editing of the part.

- Select the **Construction** > **Repetition** command.
- Select the step as the **entity**.
- Select **Pattern 1** as the **pattern** to be used.



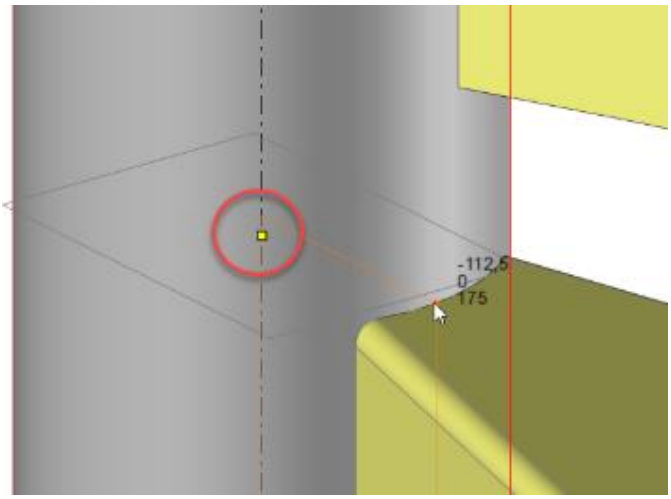
- Click on to **confirm** the repetition.

The base of the staircase is now complete.

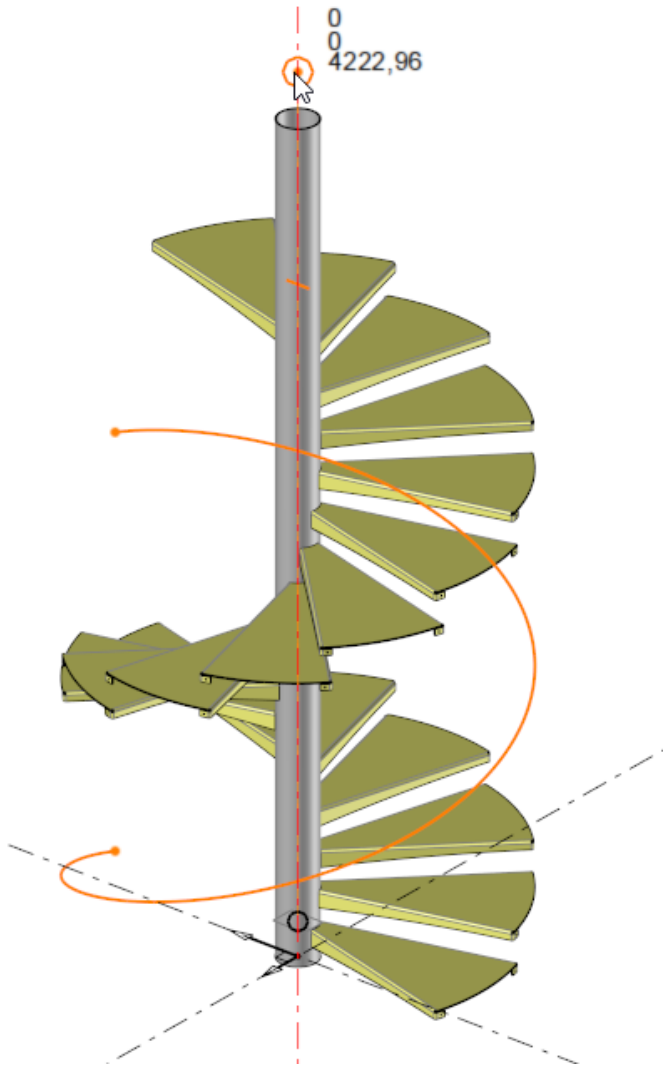
You can vary the **TotalRise**, **StepRise**, **Headroom**, and **ExternalDiameter** parameters to test your parameterization.

Creating the external stringboard

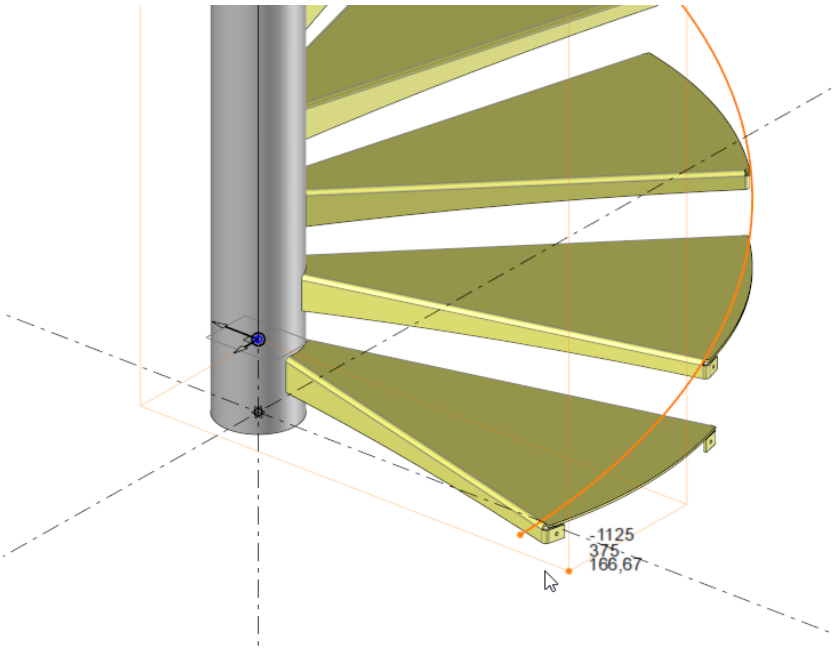
- Select the **3D sketch** > **Helix** command.
- In the **Radius** field, enter the formula *ExternalDiameter/2*.
- In the **Longitudinal pitch** field, select the **Headroom** parameter.
- Select the point shown below as the first point.



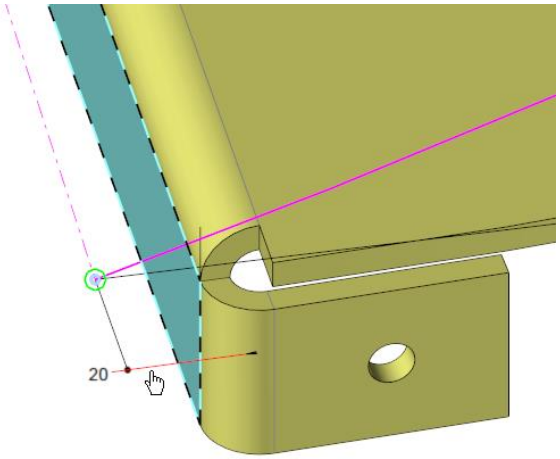
- Select the following point as the **Z point** (vertical).



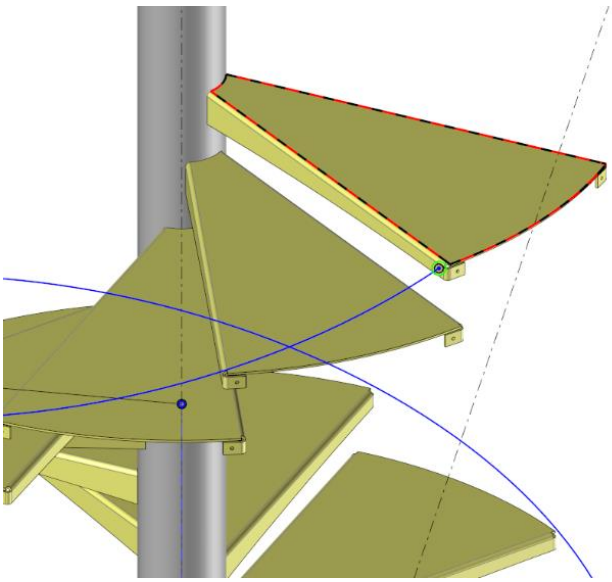
- Select a point in the void approximately in the area shown below.



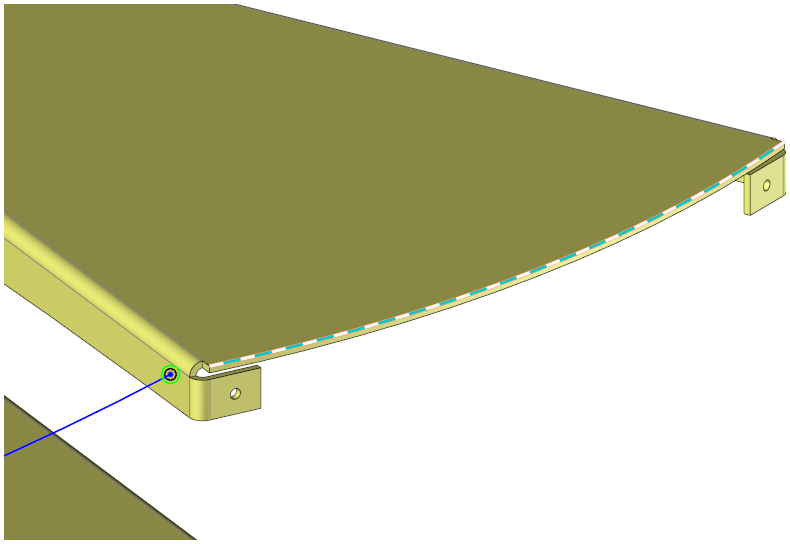
- On the base of the helix, add a *20mm* dimension between the bottom vertex of the helix and the front plane of the step.



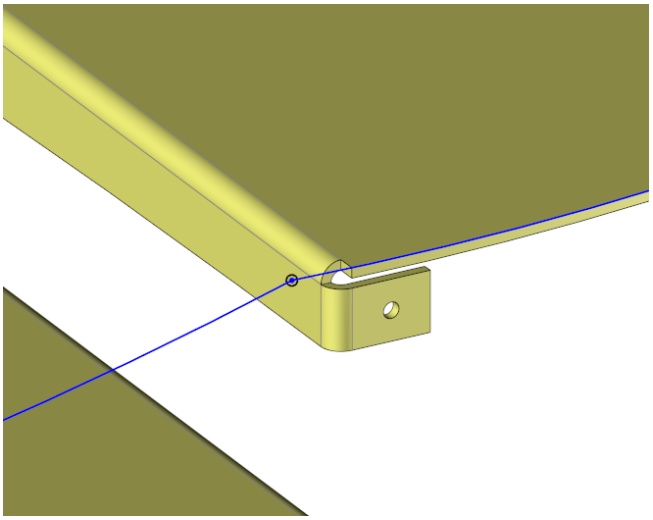
- Add a coincidence between the top vertex of the helix and the arrival plane of the staircase.



- Select the **Edge Copy** command and select the segment as shown below.

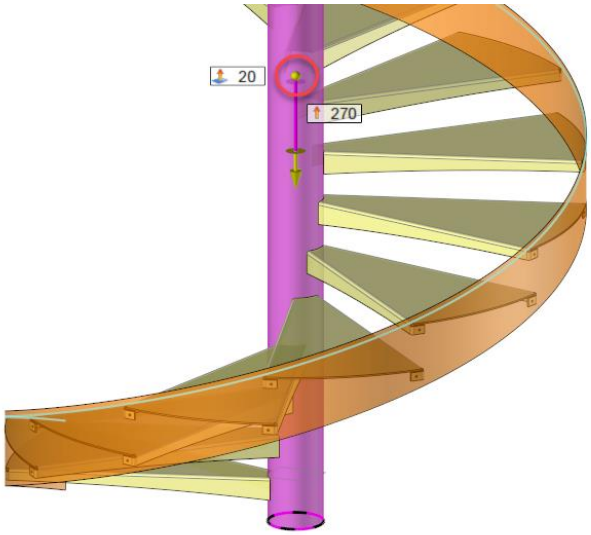


- Edit the coincidence of the previously created segment to make it coincide with the top vertex of the helix.

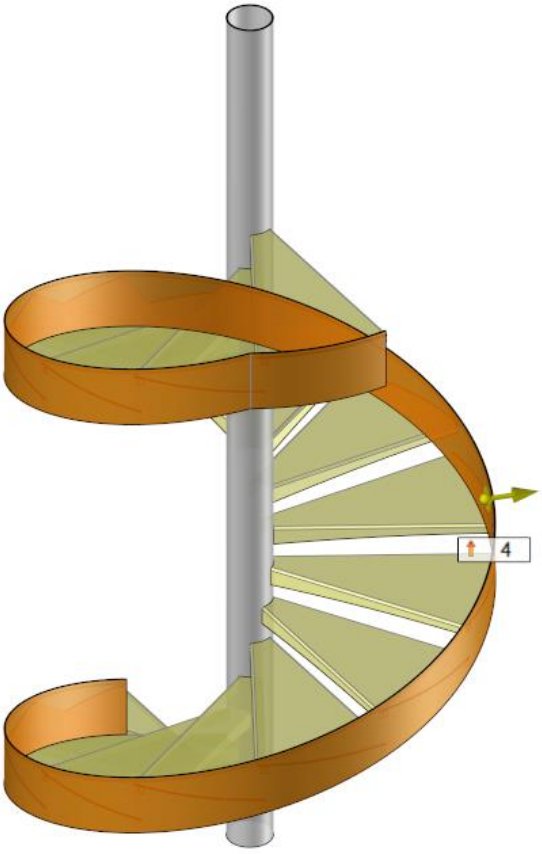


- **Confirm** the sketch.
- Right-click in the graphics area and select the **In Place Part** command.

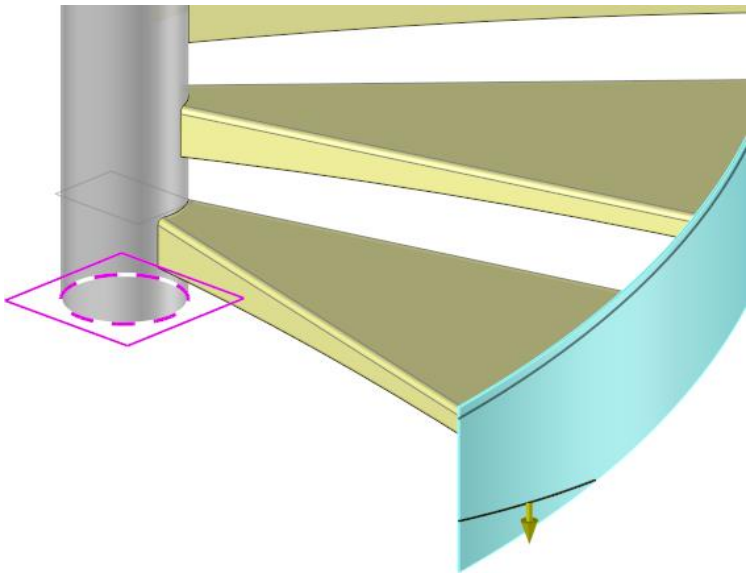
- Select the **Shape** > **Extruded** command.
- In the **Section** field, select the previously created sketch.
- In the **Direction** field, select the absolute Z axis or the central post axis.
- Extrude *270mm* downwards with an **offset** of *20mm*. For the offset, you can move the yellow sphere directly into the graphics area.



- Click on to **confirm** the extrusion.
- Right-click on the created shape and select the **Sheet Metal by Thickening** command.
- Enter a **thickness** of *4mm* toward the outside.

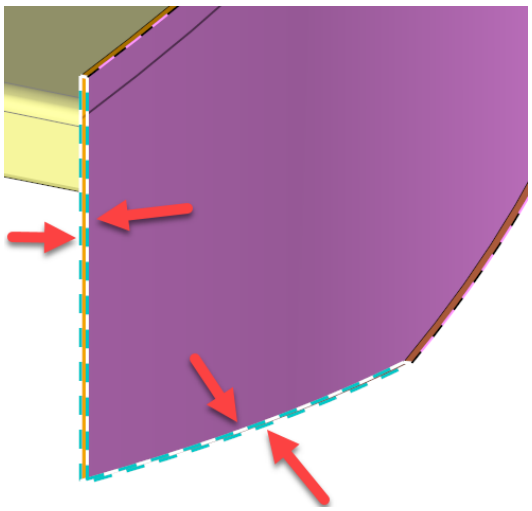


- Select the **Shape** > **Trim** command.
- Trim the stringboard in relation to the **XY** plane or the base of the central post.



For this type of part, it is often necessary to provide an extra rolling length.

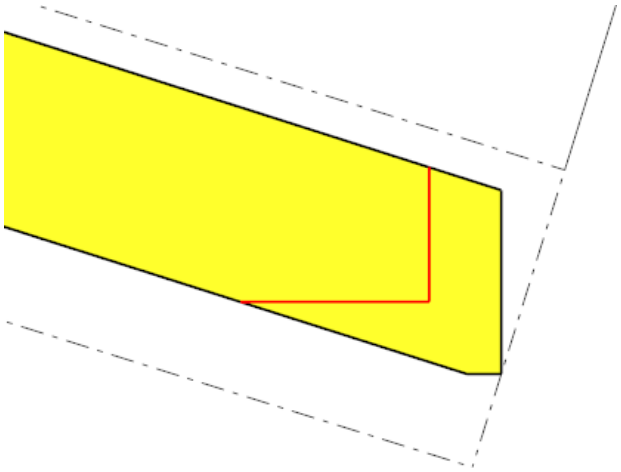
- Select the **Sheet Metal** > **Forming Flange Straightening** command.
- Select the **Extension on unfolding** mode.
- In the **Face on area to compensate** field, select the external face of the stringboard.
- In the **Boundary edges** field, select the four edges indicated below. **TopSolid** then understands which face it should extend.



- Enter *100mm* as the **extension value**.
- Make sure that the **Extend connected edge** mode is selected for the **extension of borders**.

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

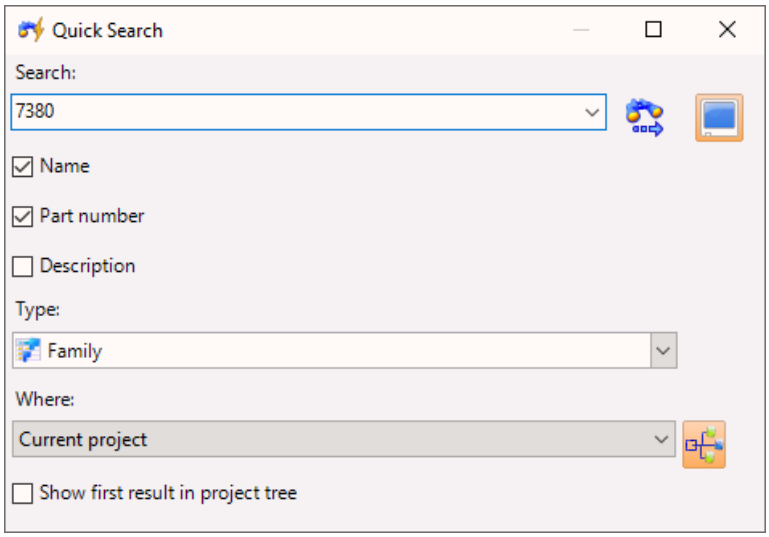
The unfolding result is as follows.



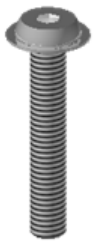
- **Confirm** the in-place editing.

We will now attach the stringboard to the steps.

- Start a  **quick search**.
- Enter *7380* as the keyword, select the **Family** type, search in the **current project** and its **part numbers**.

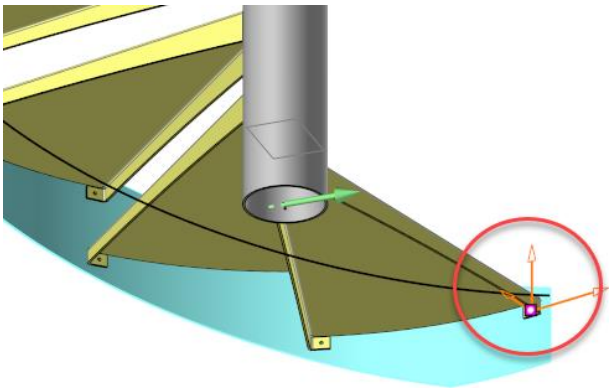


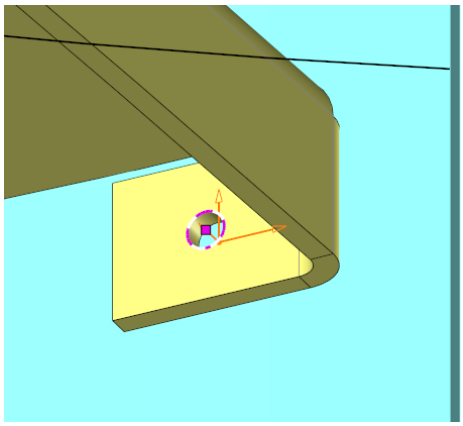
- Select the first part number and drag and drop it into the graphic area.



- Select the **M6 x 20** code.

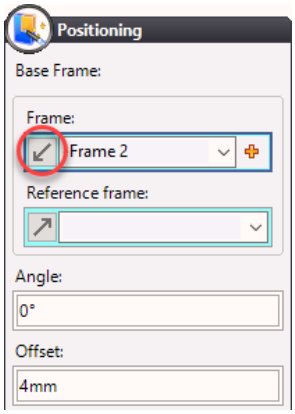
- In the **Frame** field, click on the icon and select the **Frame on Shape** command.
- In the **Shape** field, select the stringboard.
- In the **Point** field, select a point on the first drilling of the first step.

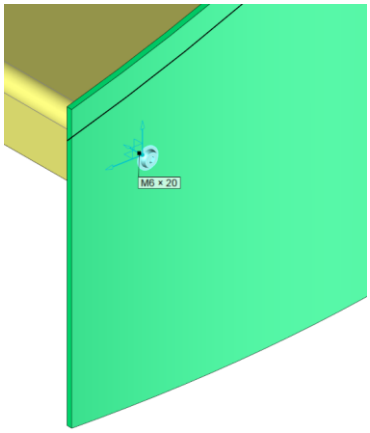




- Click on to **confirm** the creation of the frame on shape.

- In the wizard, reverse the direction of the frame using the arrow as shown below, then add an **offset** of *4mm*.

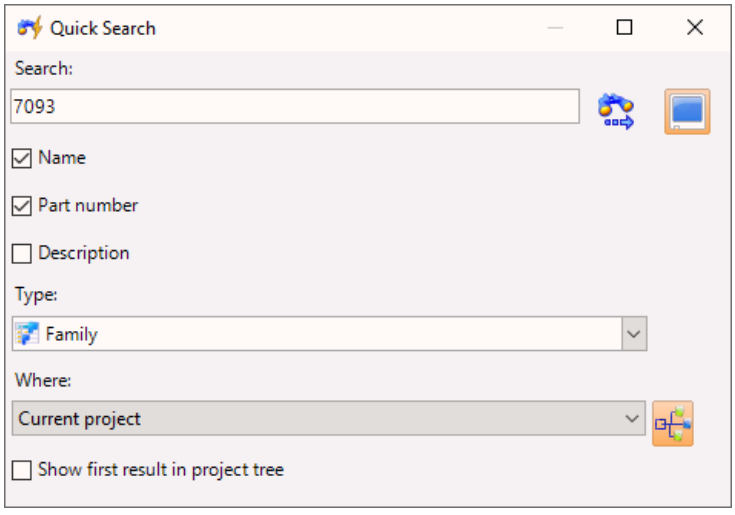




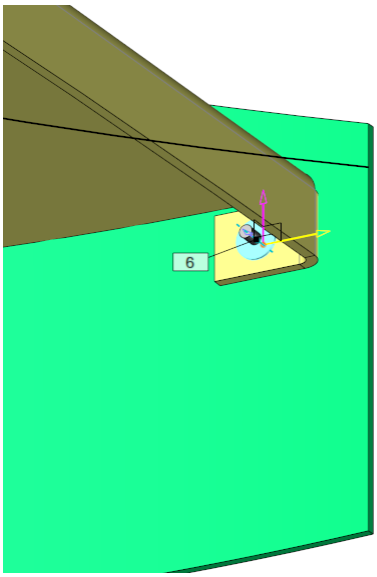
- Click on to **confirm** the wizard.
- Select **Clearance hole** as the **process**.

We will now include the washer.

- Start a **quick search**.
- Enter *7380* as the keyword, select the **Family** type, search in the **current project** and its **part numbers**.

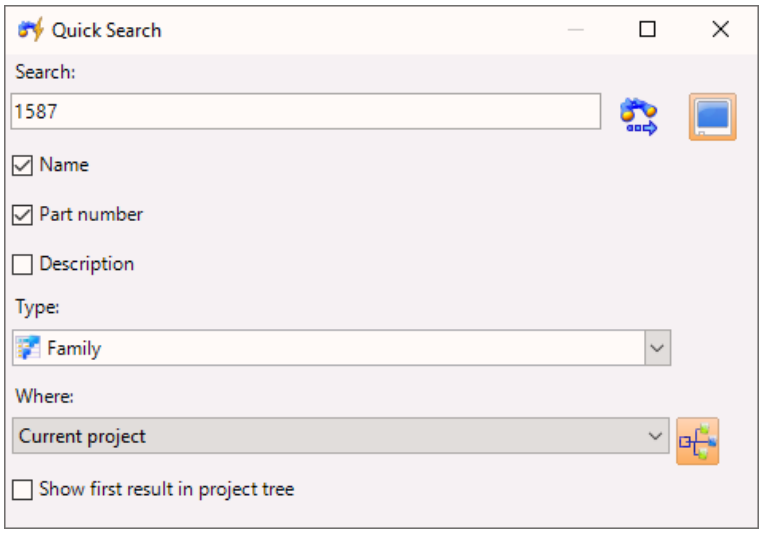


- Insert the washer, select the code **6**, then position the component as shown below.

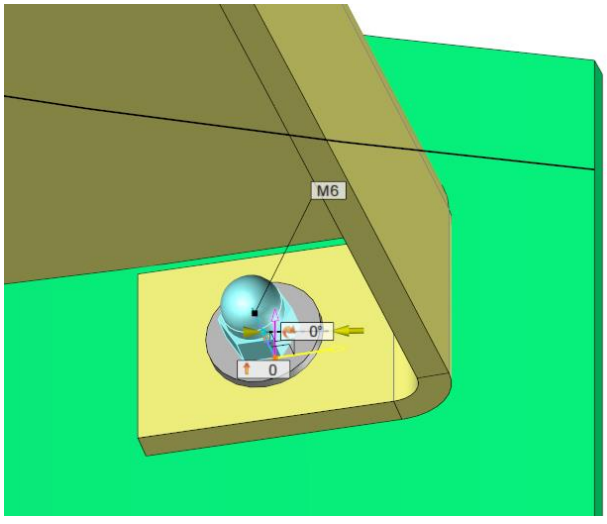


We will now include the nut.

- Start a **quick search**.
- Enter *1587* as the keyword, select the **Family** type, search in the **current project** and its **part numbers**.

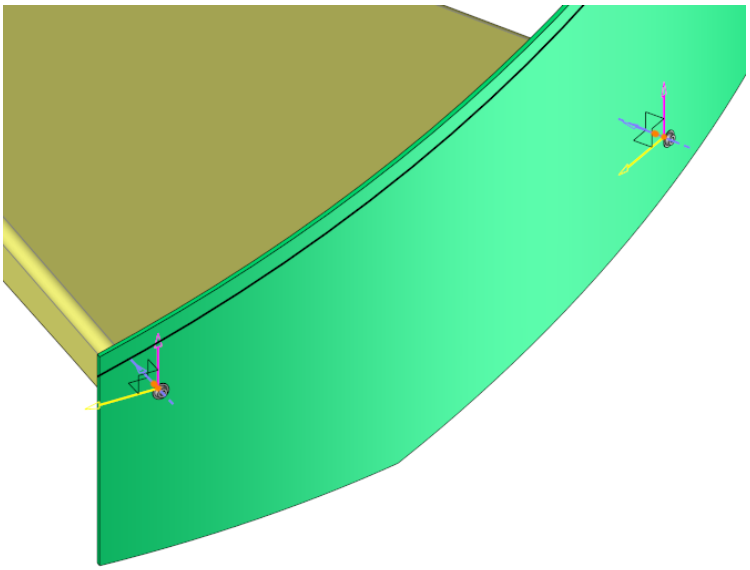


- Insert the nut, select the code **M6**, then position the component as shown below.

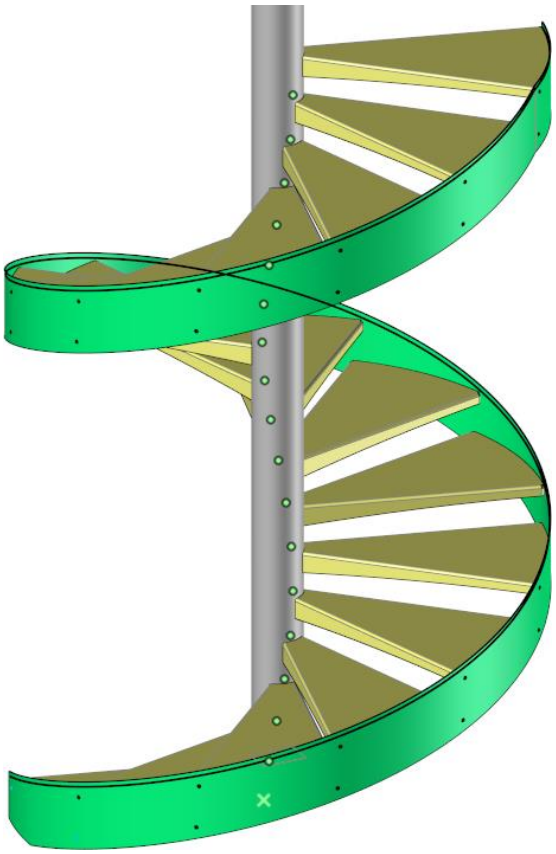


We will now add the same components on the second drilling of the step. To avoid having to search again, you can reinsert a component that is already there.

- To do this, press and hold down the **Ctrl** key on the keyboard and drag the desired component while holding down the left mouse button.



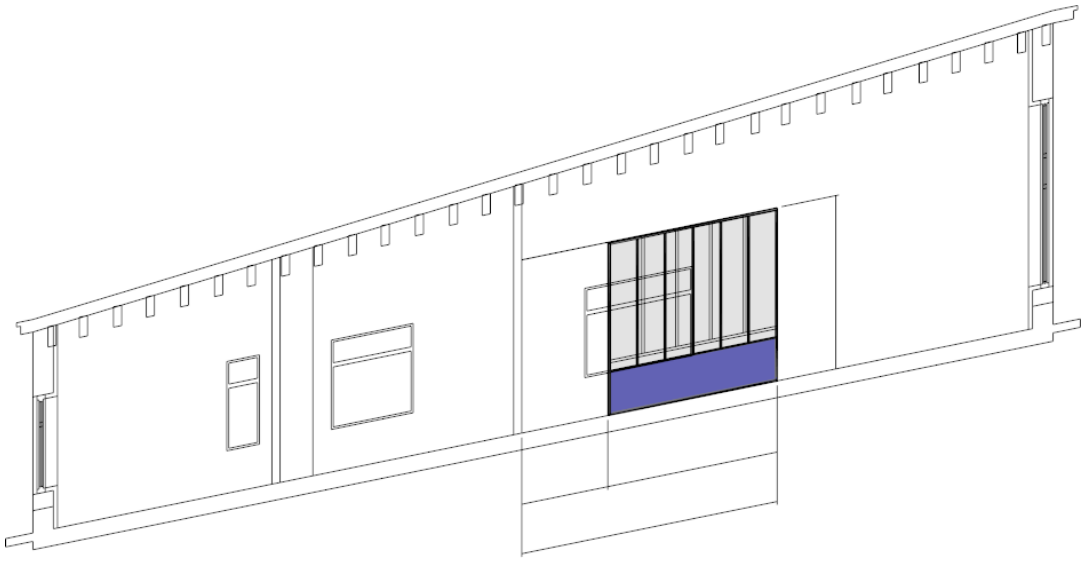
- Select the **Construction** > **Repetition** command.
- In the **Entities** field, select the two screws, the two washers and the two nuts.
- In the **Pattern** field, select the pattern of the steps.



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

Part 3 - Designing and Using a Glass Partition Component

The purpose of this exercise is to draw the following parametric glass partition, to integrate it in an environment resulting from a DXF/DWG import and to generate manufacturing drawings.



Concepts addressed:

- Importing a DXF/DWG file
- Using steel joinery extruded bars
- Creating and using families
- Using automatic drafting bundles
- Multiple unfoldings/draftings

Importing the DXF/DWG 2D environment

- From the Project tree, create a new folder named *3- Glass partition*.

The glass partition will be designed using extruded bars and document templates whose design principles will be described throughout the training.

- In the *3- Glass partition* folder, **Import the package** named *Glass Partition Files.TopPkg*.
- Right-click on the *3- Glass partition* folder and select the **Import/Export** > **Import File with Conversion** command. Select the file named *Glass Partition Project Architect Drafting.A.0.dxf*.

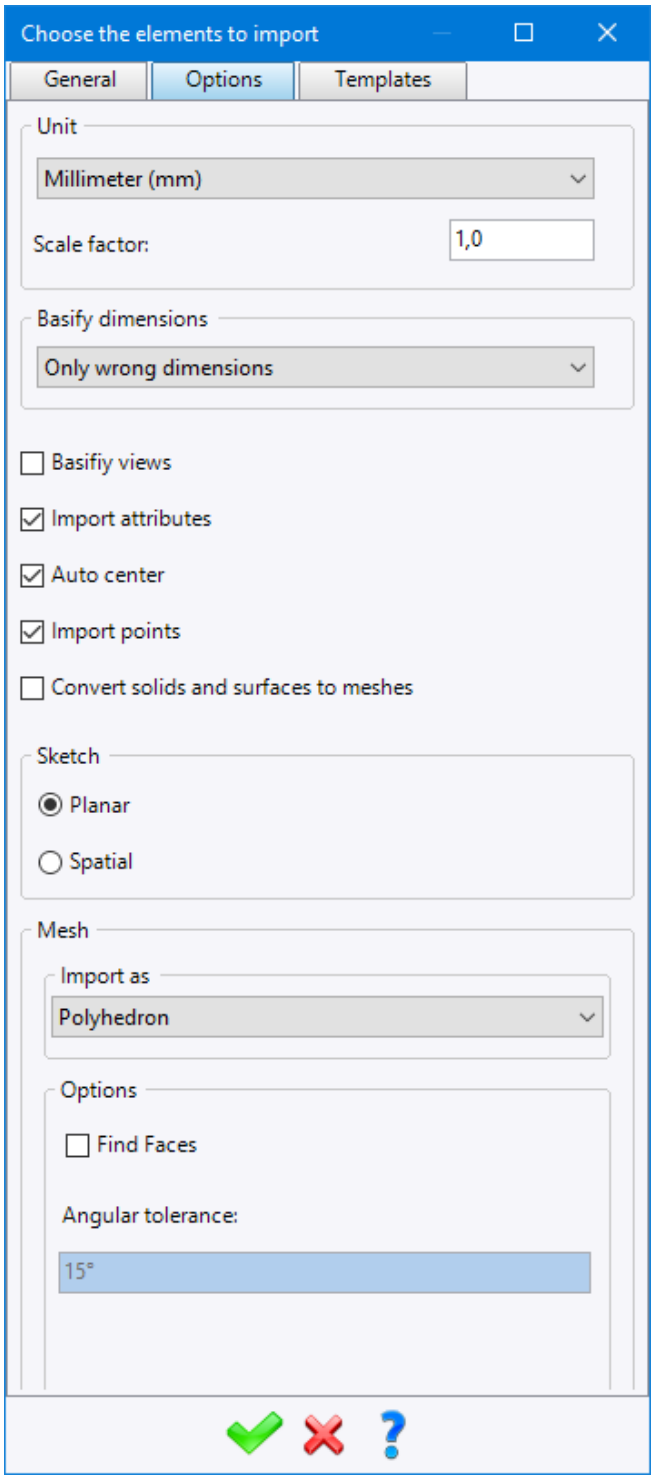
TopSolid displays the different spaces contained in the DXF/DWG file as tabs. In most cases, only the object space is required. The other tabs can therefore be unchecked.

The different layers can be selected on the left side of the screen.

- Select **Drawing** from the **Import as** drop-down list.

The drawing document is a non-associative 2D document that is perfectly suited for a DXF/DWG import. Although it is very basic, it allows you to manage large documents with better performance than when importing as a part or assembly.

- From the **Options** tab, check the settings of the following parameters.



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

The drawing document is displayed in the foreground.

It is advisable to clean the 2D imported files in order to remove the superimposed lines, to adjust the segments, etc.

- Select the **2D Sketch** > **Healing** > **Clean** command.
- Select the sketch and click on the icon.

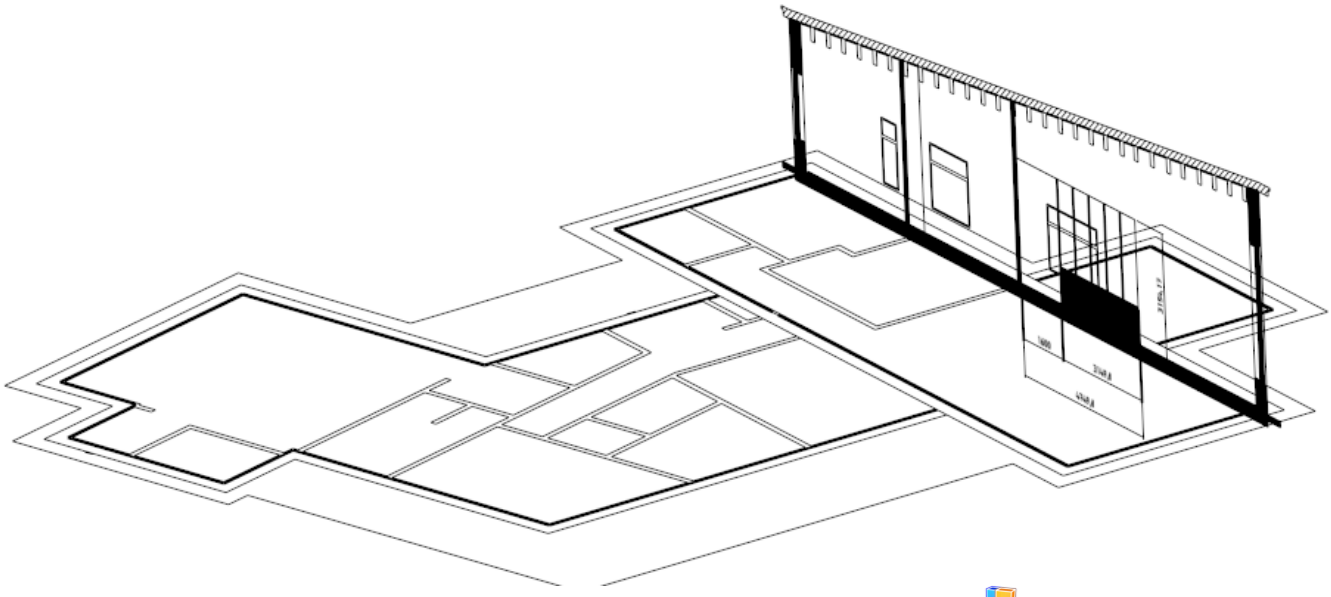
TopSolid displays the created lines and circles.

The cleaning tolerance can be adjusted as required.

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

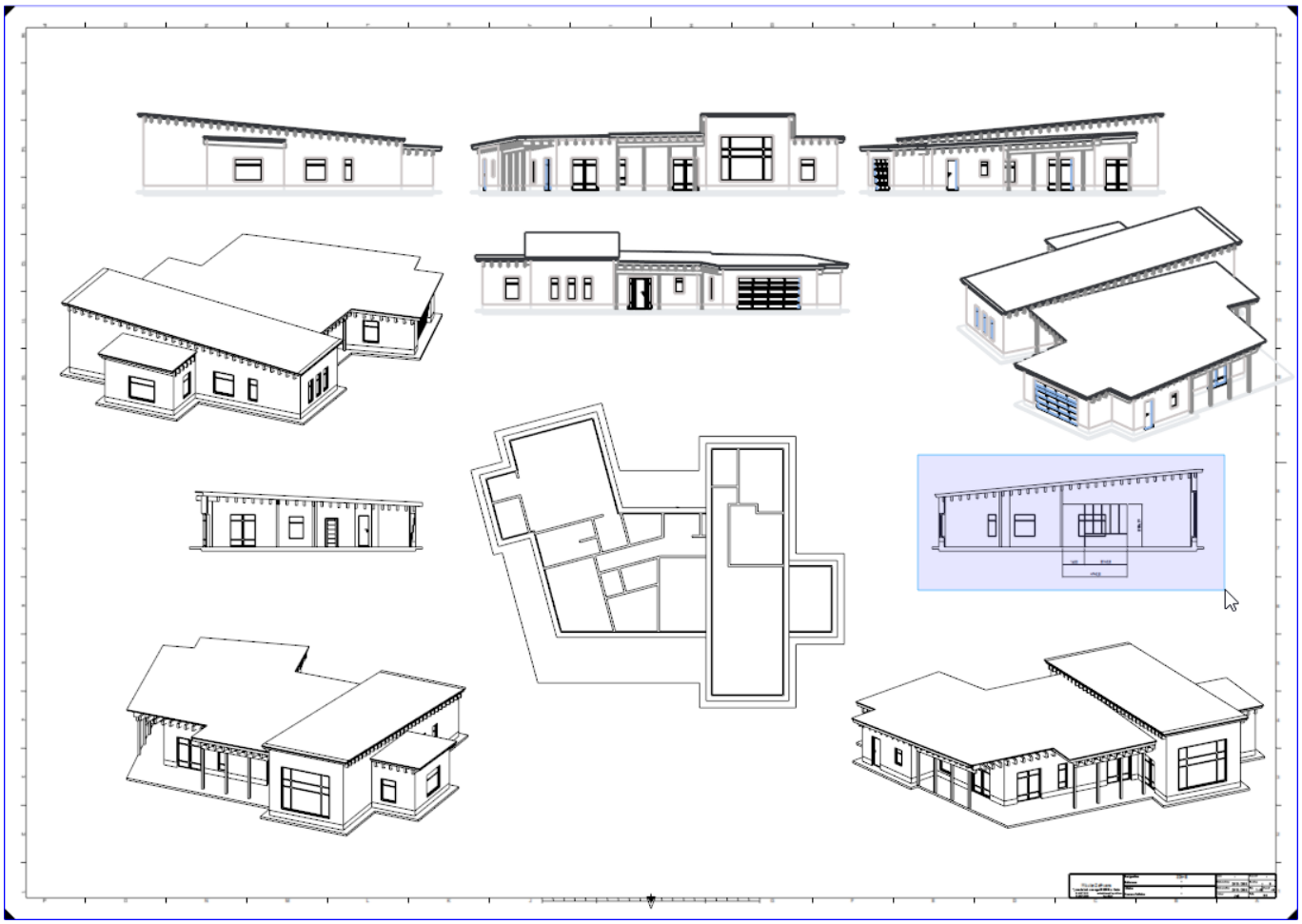
We will now isolate a cross section view to then integrate our future glass partition.

Note: In addition to the cross section view, you can also use a top view to recreate a 3D environment as shown below.

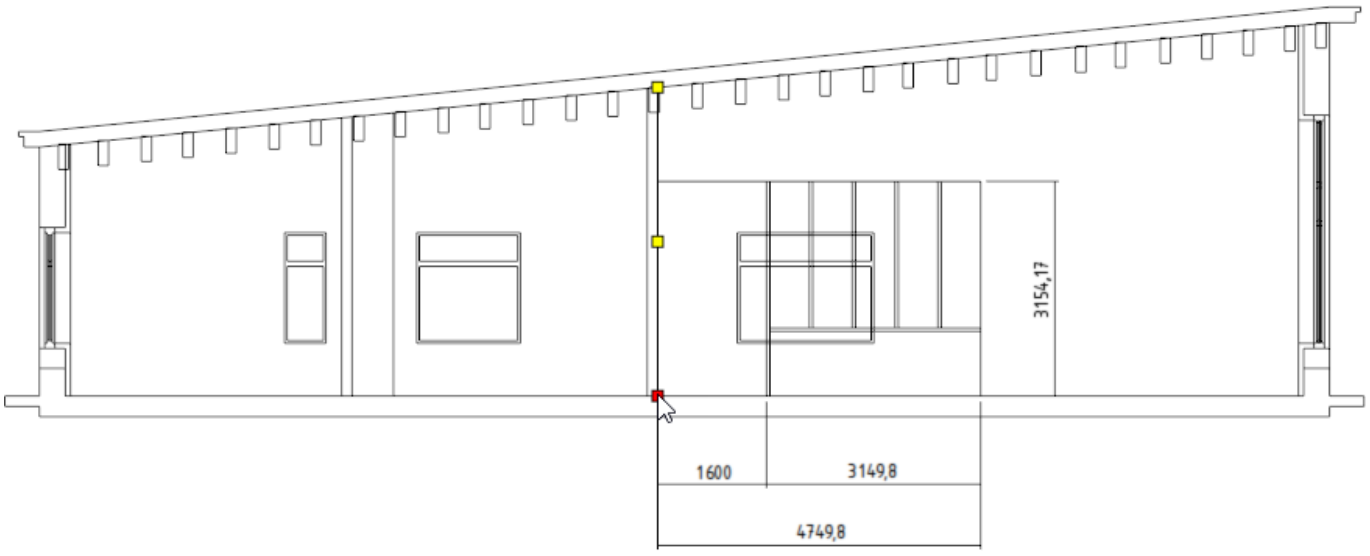


- From the Project tree, right-click on the *3- Glass partition* folder and select the  **Assembly** command.
- Rename the assembly document *Glass partition establishment*.

- In the drawing document, select the cross section view.



- Right-click in the graphics area and select the **Copy** command or use the **Ctrl + C** keyboard shortcut.
- Select the **Point** option and select the point as shown below.



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

- In the assembly document, right-click in the graphics area and select the **Sketch** command.
- **Position the sketch** on the **absolute XZ plane**.
- Right-click in the graphics area and select the **Paste** command or use the **Ctrl + V** shortcut.
- Select the **Sketch origin** option and make sure that you check the **Fix** box.
- Click on to **confirm** the operation.
- **Confirm** the sketch.

The imported drawing has a scale factor. In order to build on this drawing, you need to reset the dimensions of the sketches.

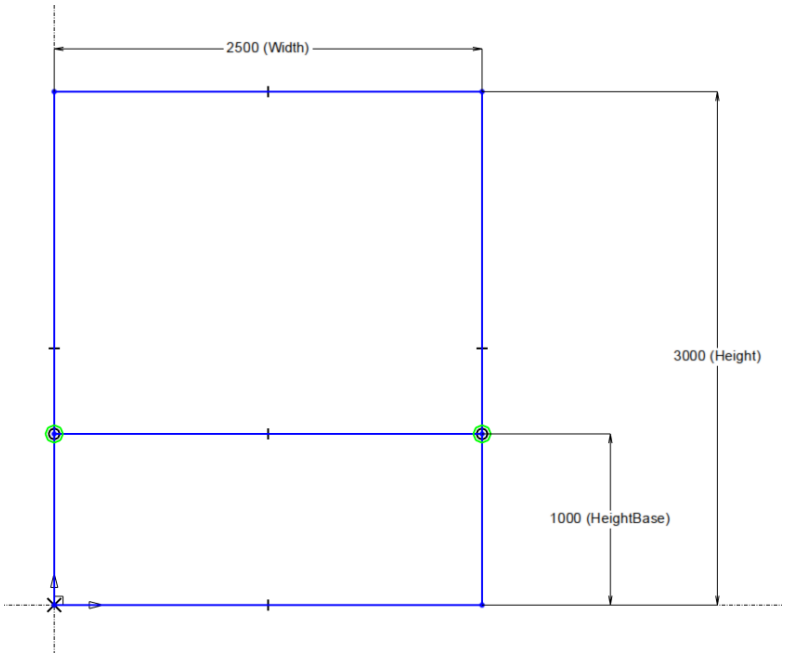
- Select the **Construction** > **Transform** command.
- Select the two sketches and the plane as the **entities to transform**.
- In the **Transform** field, click on the icon and select the **Scaling Transform** command.
- In the **Center** field, select **Absolute Origin Point** and enter a **factor** of *80*.
- Click on to **confirm** the transformation type, and then the transformation operation.
- **Save** the document (**Ctrl** + **S**).

Creating the glass partition

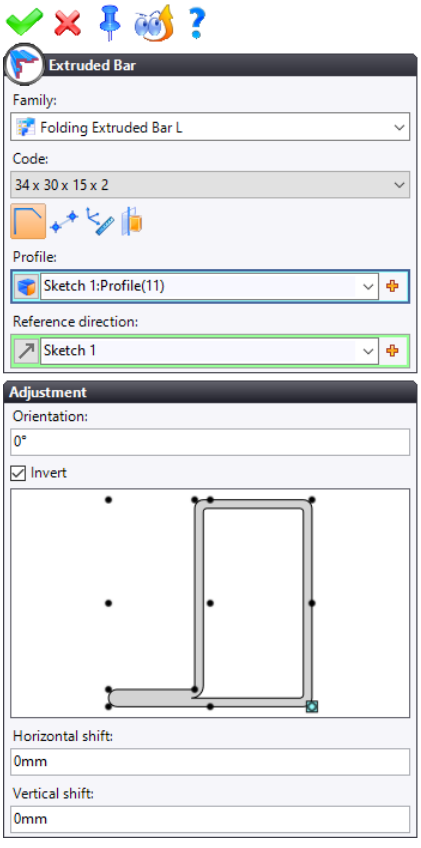
Creating the frame

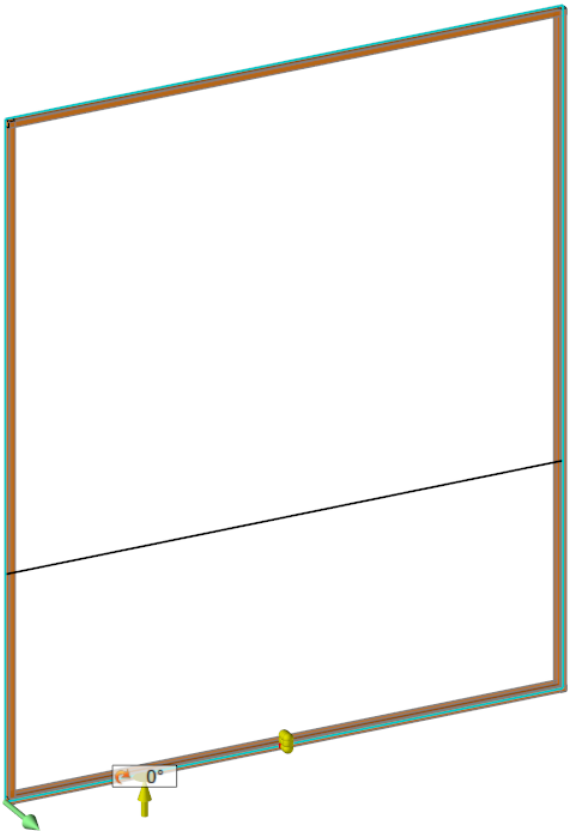
The glass partition will not be created directly on an environment. It will be independent, which will allow you to store it in a library and use it in several projects.

- In the *3- Glass partition* folder in the Project tree, create a new **Assembly** document named *Glass partition*.
- On the XZ plane, create a sketch with the parameters as shown below.



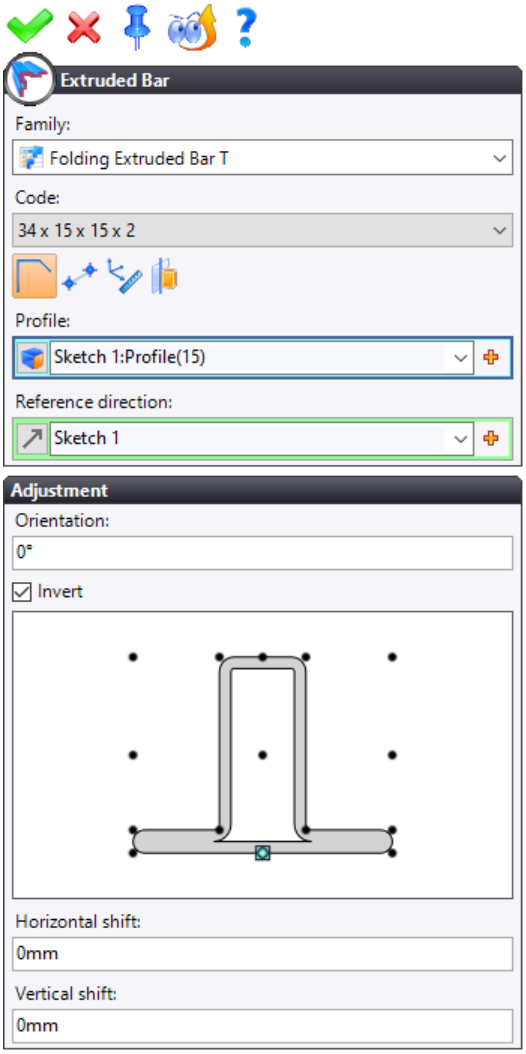
- On the outer frame, include the **Folding Extruded Bar L** extruded bars using the **34 x 30 x 15 x 2** code.

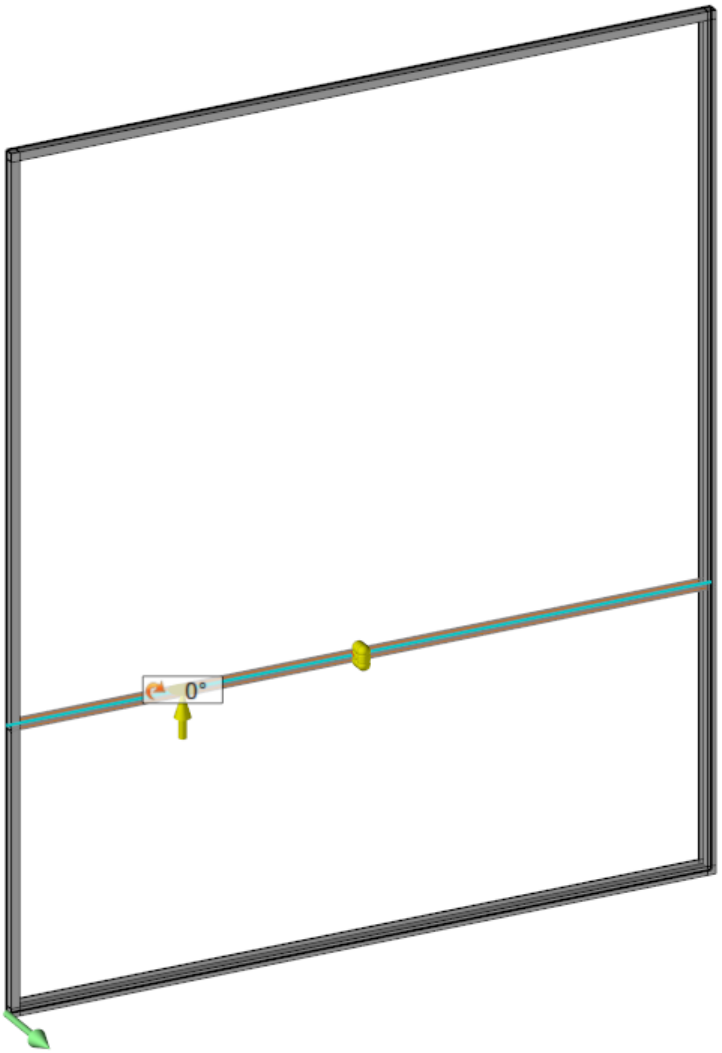




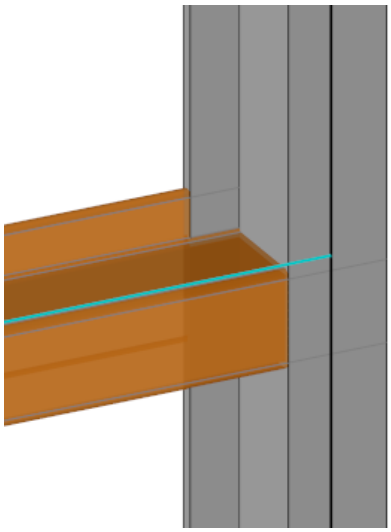
- Orient the extruded bars using the **Orientation** field, the positioning points and the **Invert** option so that the extruded bars are positioned as shown below with the blade pointing inwards.

- Click on  to **confirm** the inclusion.
- Add a **Folding Extruded Bar T** extruded bar using the **34 x 15 x 15 x 2** code on the center segment.

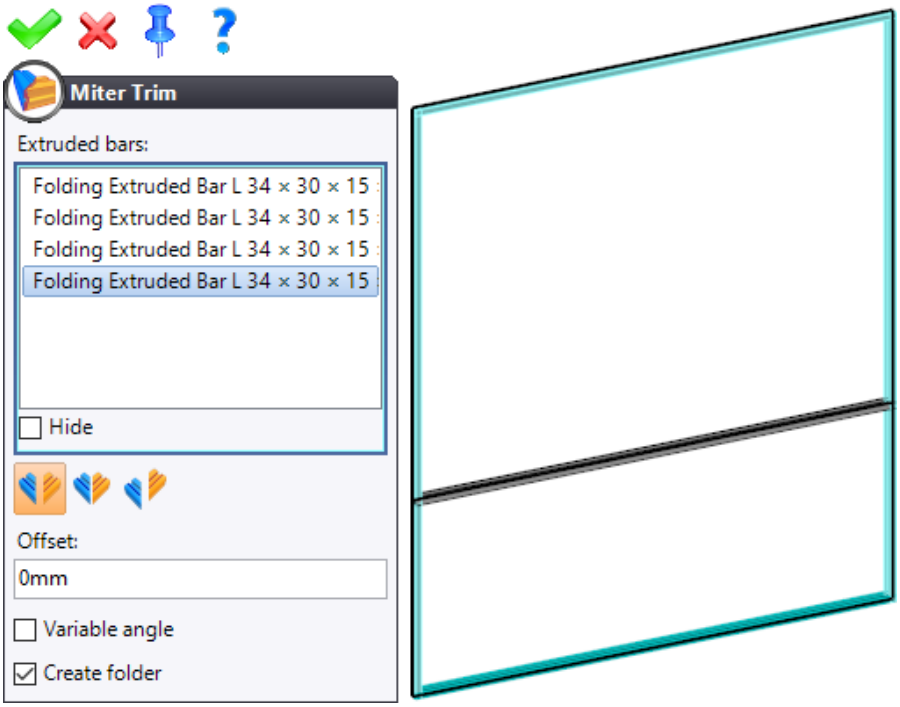




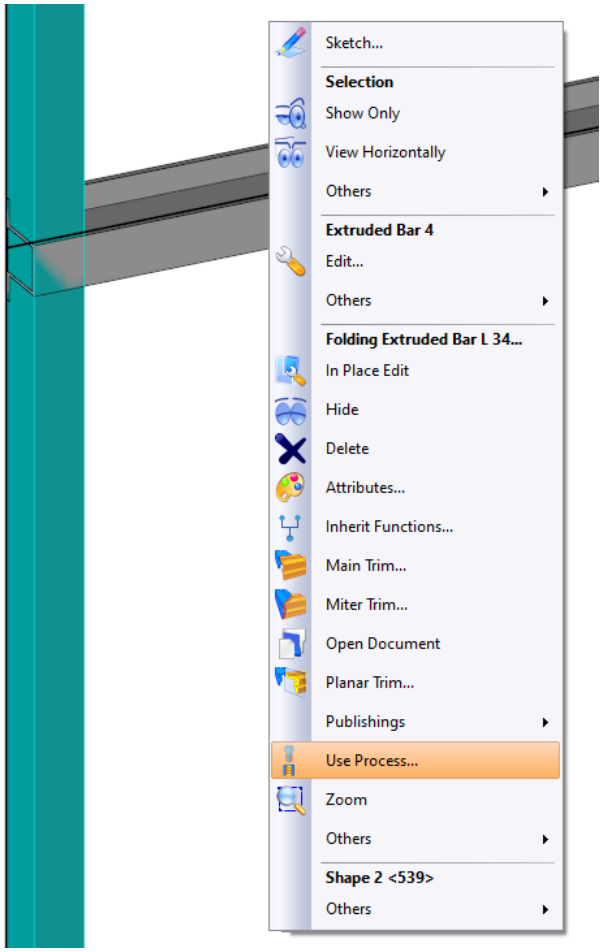
- Align the blades with those of the frame.

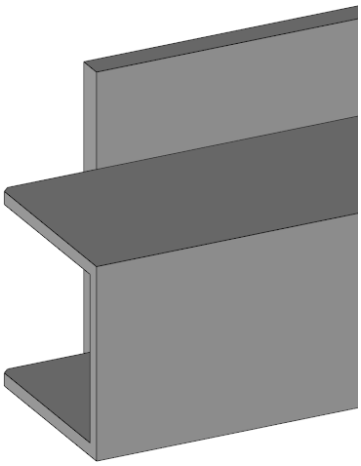


- Click on to **confirm** the inclusion.
- **Save** the document (**Ctrl** + **S**).
- Perform **miter trims** on the outer frame.



- Adjust the central crossbar with notches. To do this, right-click on one of the vertical uprights and select the **Use Process** command.

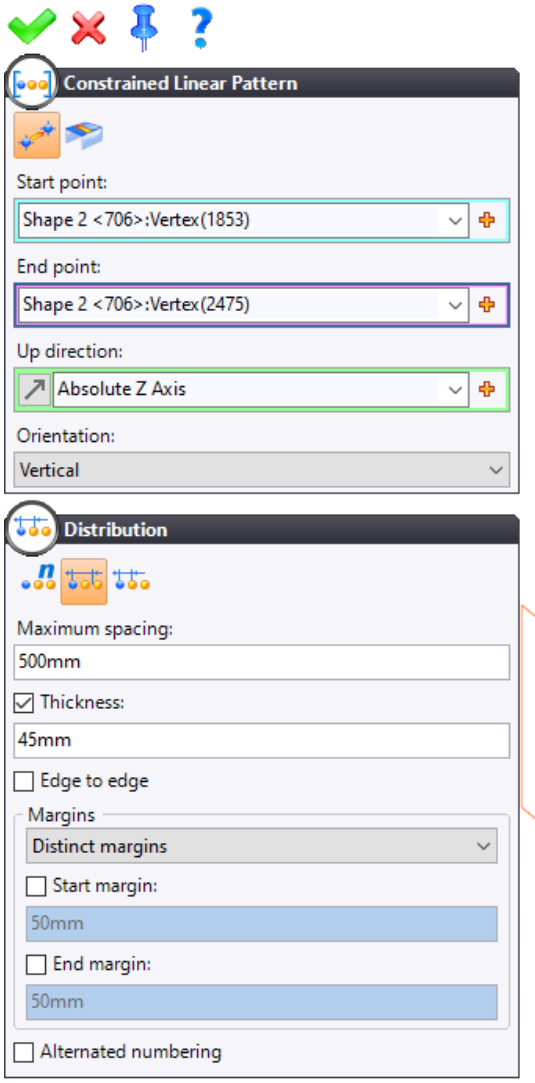


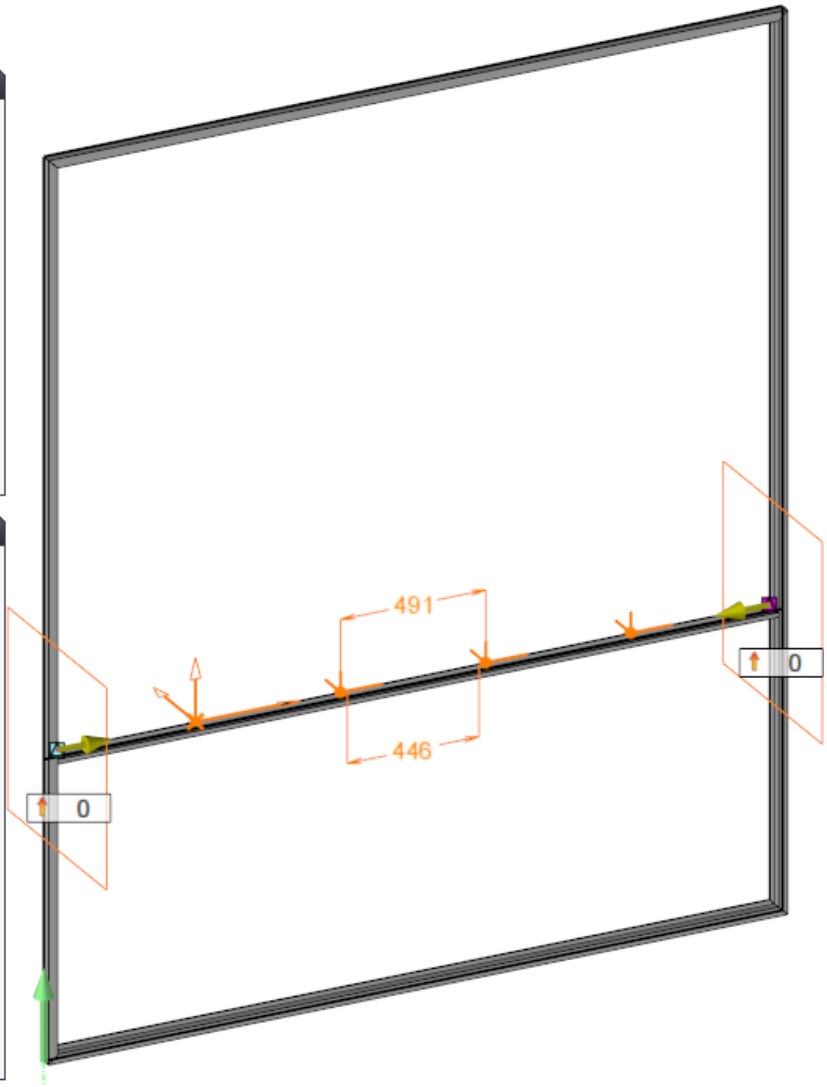


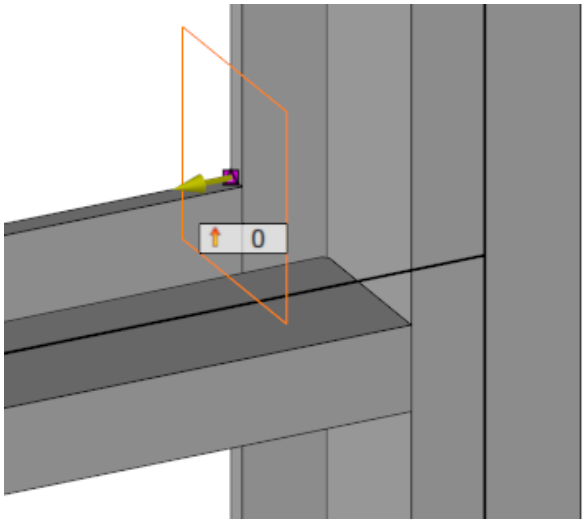
- Repeat the operation on the opposite side.

Creating the top filling

- Create a **constrained linear pattern** between points by entering a **maximum spacing** of *500mm* and a **thickness** of *45mm*.
- Check the **Create parameter** box in the **advanced options**.





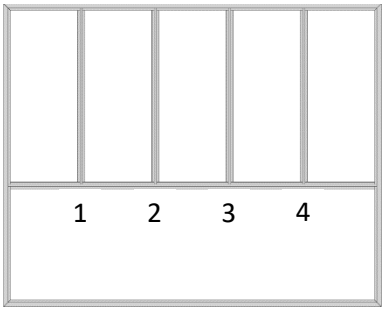


Notes: All repetitions have an **Alternated numbering** option which affects the way **TopSolid** will number the occurrences resulting from the repetition. This option will have a very strong impact on the way you design according to the selected mode.

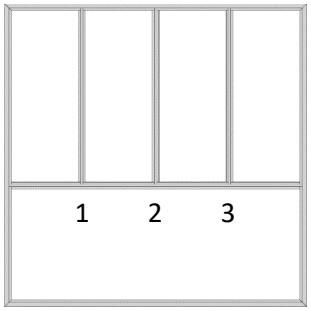
Examples:

- **Alternated numbering** box unchecked:

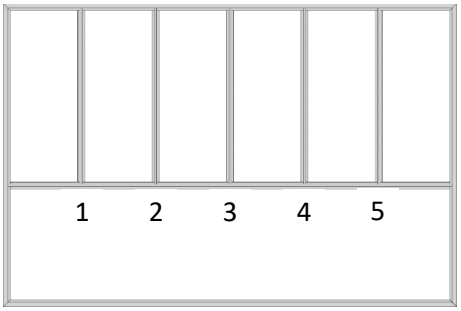
Width = 2500mm



Width = 2000mm



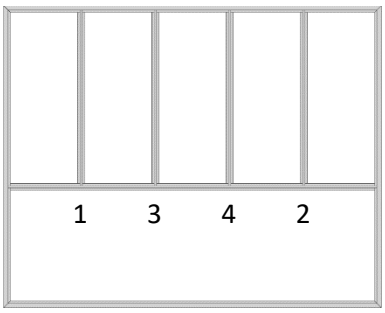
Width = 3000mm



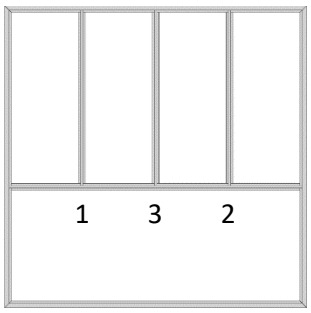
Standard continuous numbering.
If the user performs an operation on the last extruded bar with the number 4 and the width changes from 2500mm to 2000mm, **TopSolid** will go into error since extruded bar 4 would no longer exist.

- **Alternated numbering** box checked:

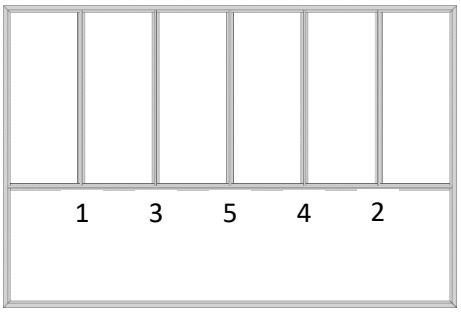
Width = 2500mm



Width = 2000mm



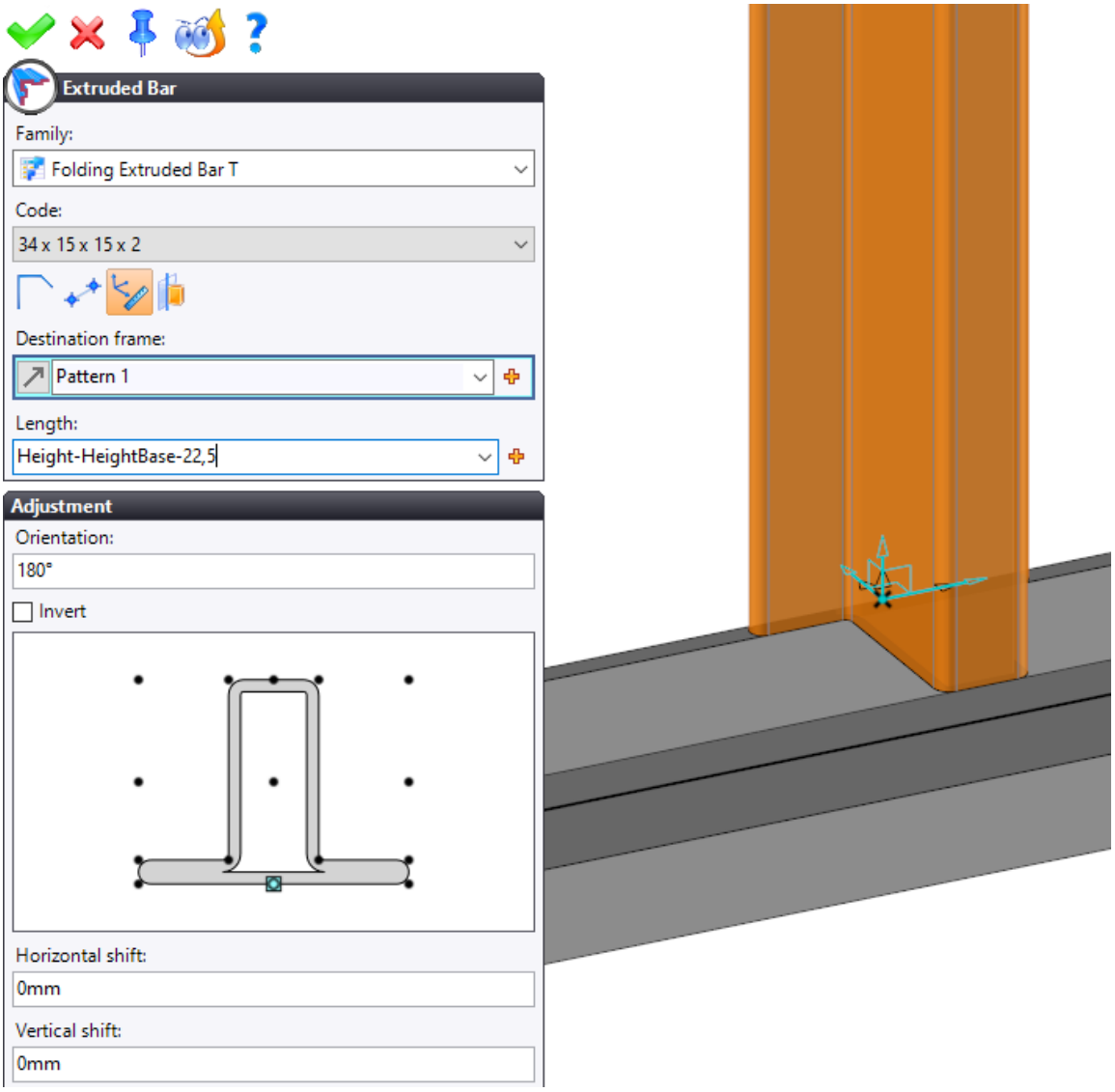
Width = 3000mm



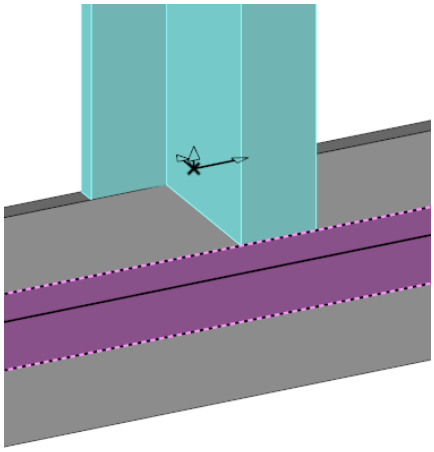
Numbering that gives priority to the ends.
If the user performs an operation on the last extruded bar with the number 2 and the width is changed from 2500mm to 2000mm, no error will appear.
In another case, the third extruded bar with the number 4 is modified on the 2500mm width. If the width increases to 3000mm, extruded bar number 4 moves to the fourth position instead of the third position.

Alternated numbering also has some disadvantages. There is no ideal mode. Each mode has its own areas of application with the related advantages and constraints.

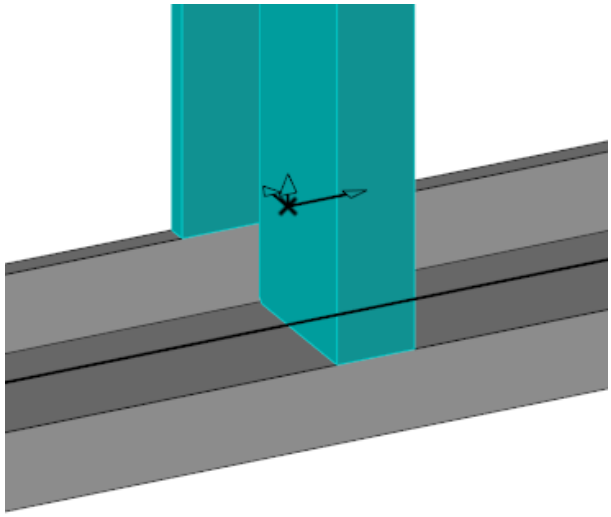
- Select the **Modeling** > **Extruded Bar** command.
- Select the **Folding Extruded Bar T** family and the **34 x 15 x 15 x 2** code.
- Select the **Frame and Length** mode.
- In the **Length** field, enter the *Height - HeightBase - 22.5* formula.
- Use the **Orientation** field to align the blades.



- Adjust the extruded bar on the central crossbar. To do this, right-click on the extruded bar and select the **Planar Trim** command.

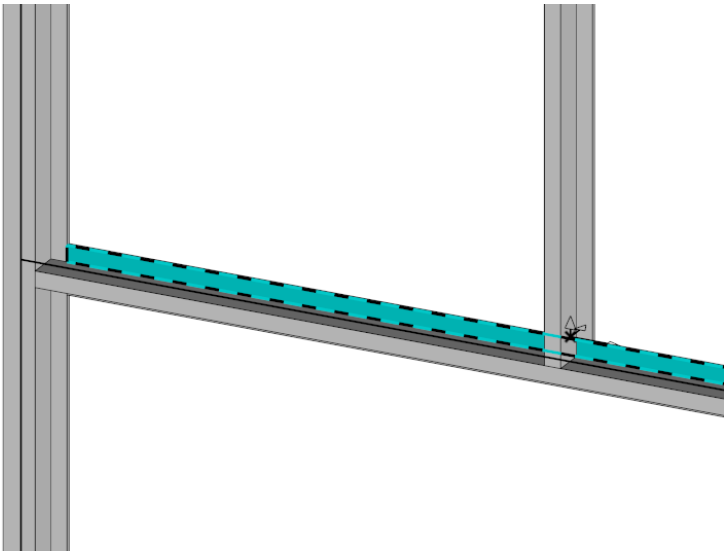


- Add a notching operation to both ends. To do this, right-click both on the central crossbar and top crossbar and select the **Use Process** command.

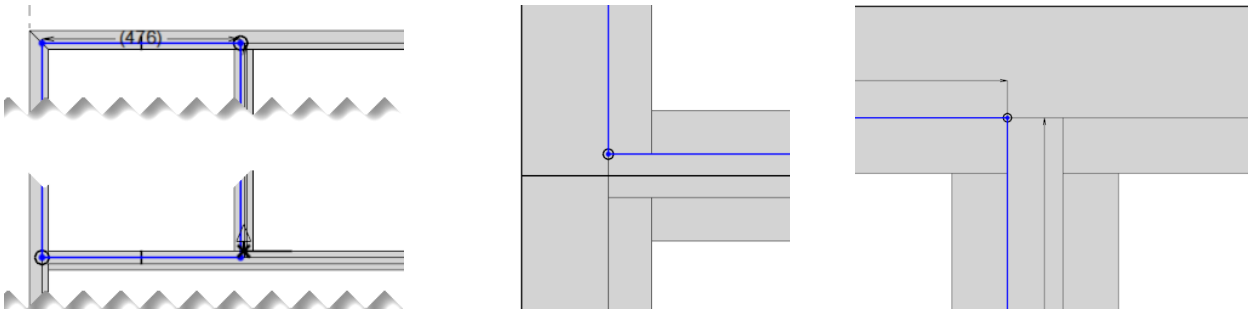


We will now integrate the glass.

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



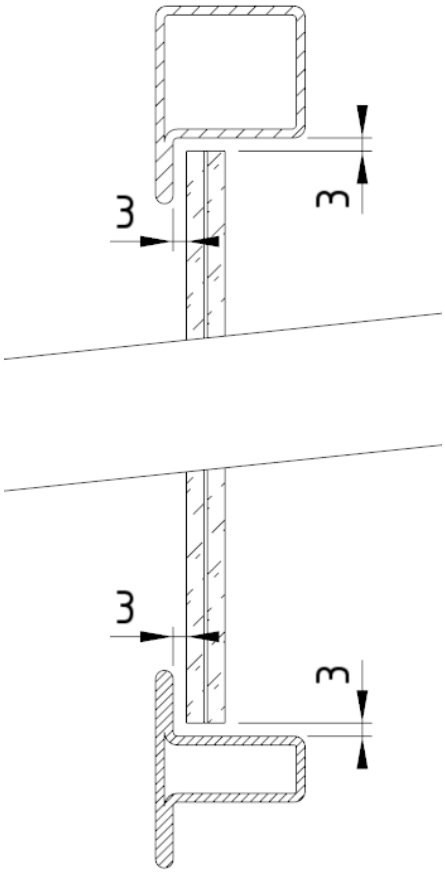
- Create a **rectangle** as shown below.



- **Confirm** the sketch.

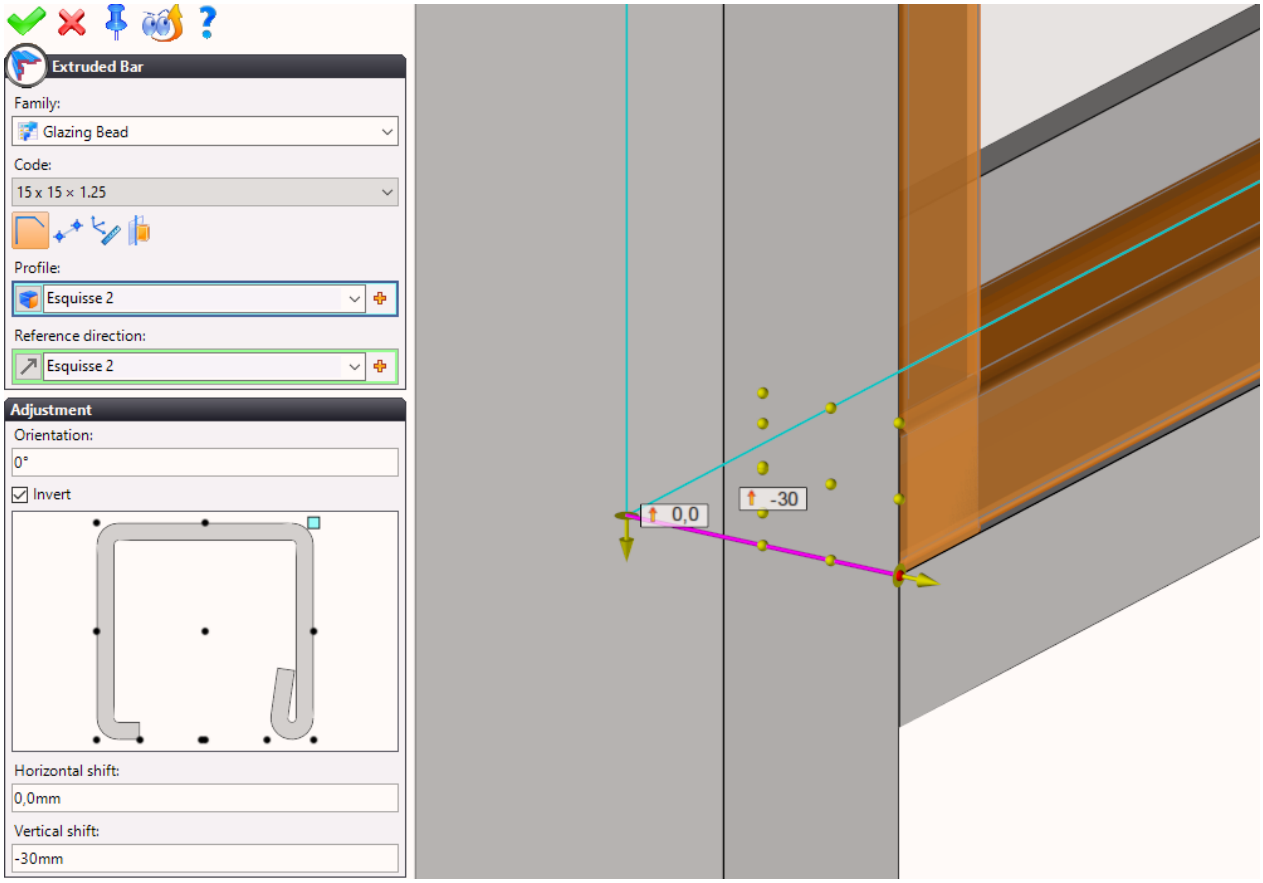
- From the Project tree, open the *3- Glass partition > Glass partition files > Laminated glass* folders, then drag and drop the **Family** document named *Laminated glass – Geometric drivers* into the graphics area.
- Select the **44.2 (8.8mm)** code.
- In the **Profile** field, select the previously created sketch.
- Enter a **clearance** of *3mm* and an **offset** of *3mm*.
- Click on to **confirm** the inclusion.

The glass must be positioned as shown below.

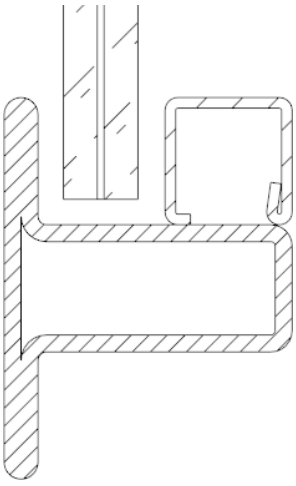


We will now integrate the glazing beads.

- Use the sketch to insert four **Glazing Bead** extruded bars using the **15 x 15 x 1.25** code.
- Enter a **vertical shift** of *30mm*.

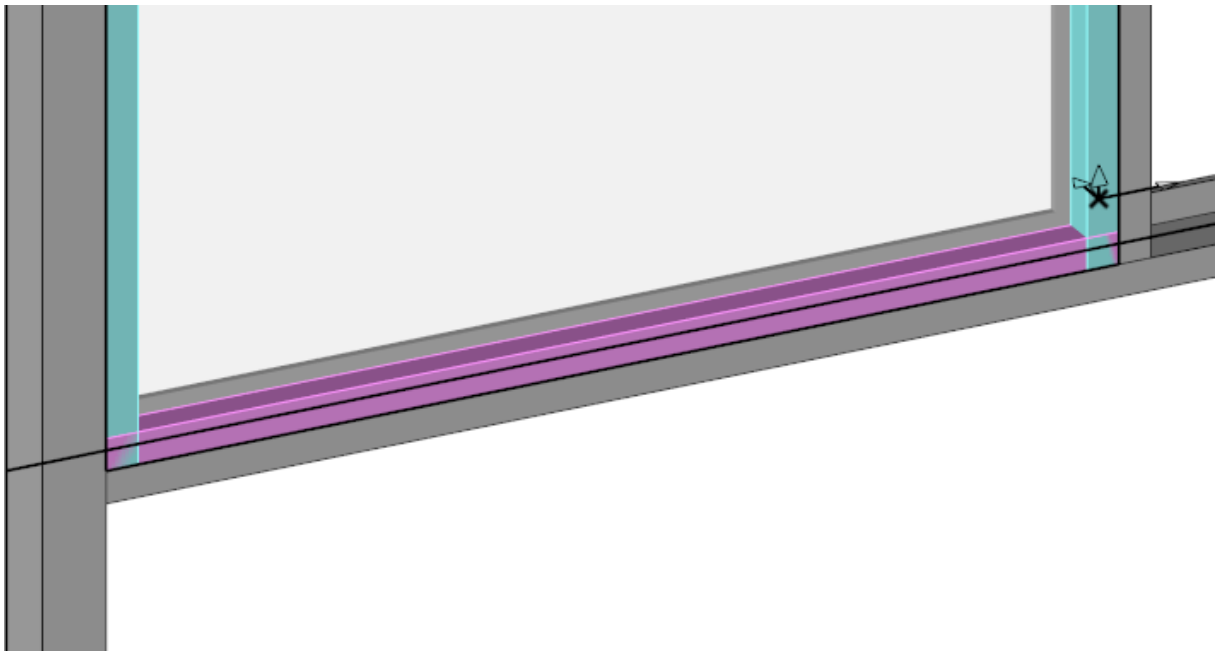


- Make sure that the positioning is correct.

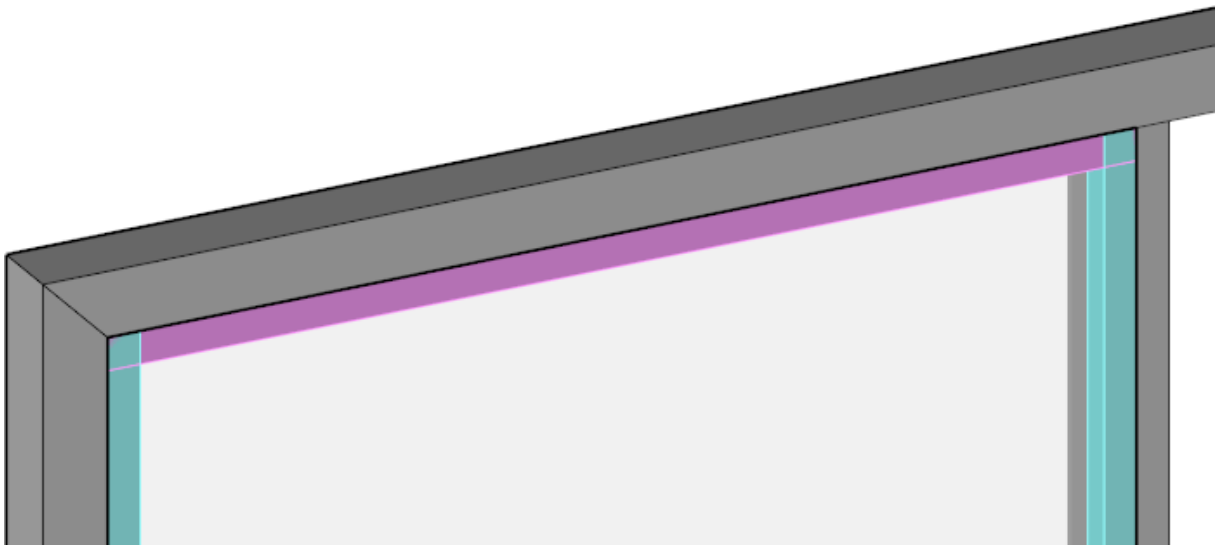


- **Save** the document (**Ctrl** + **S**).

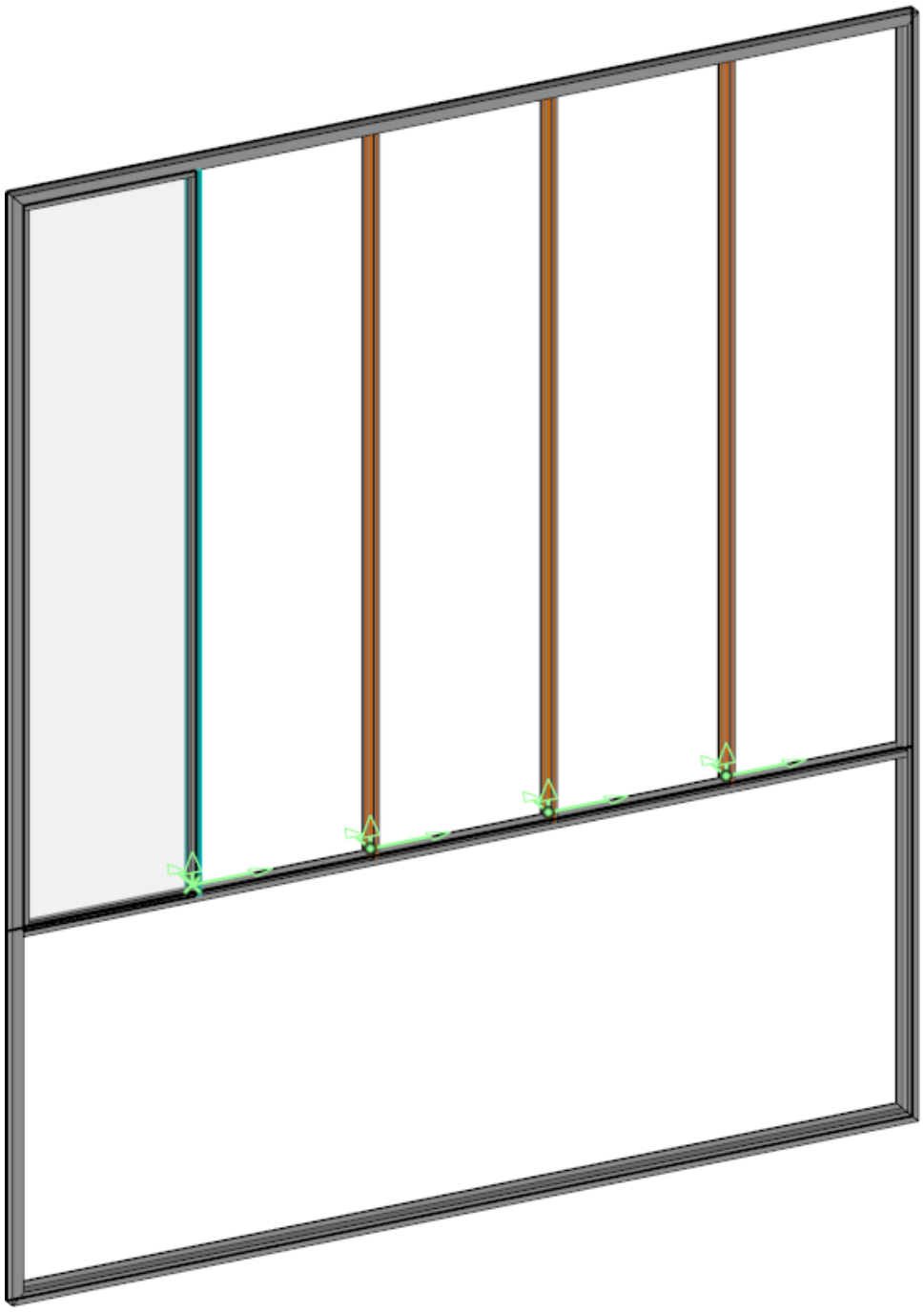
- Perform **main trims** on the vertical glazing beads in relation to the lower horizontal glazing bead.



- Perform **main trims** on the vertical glazing beads in relation to the upper horizontal glazing bead.



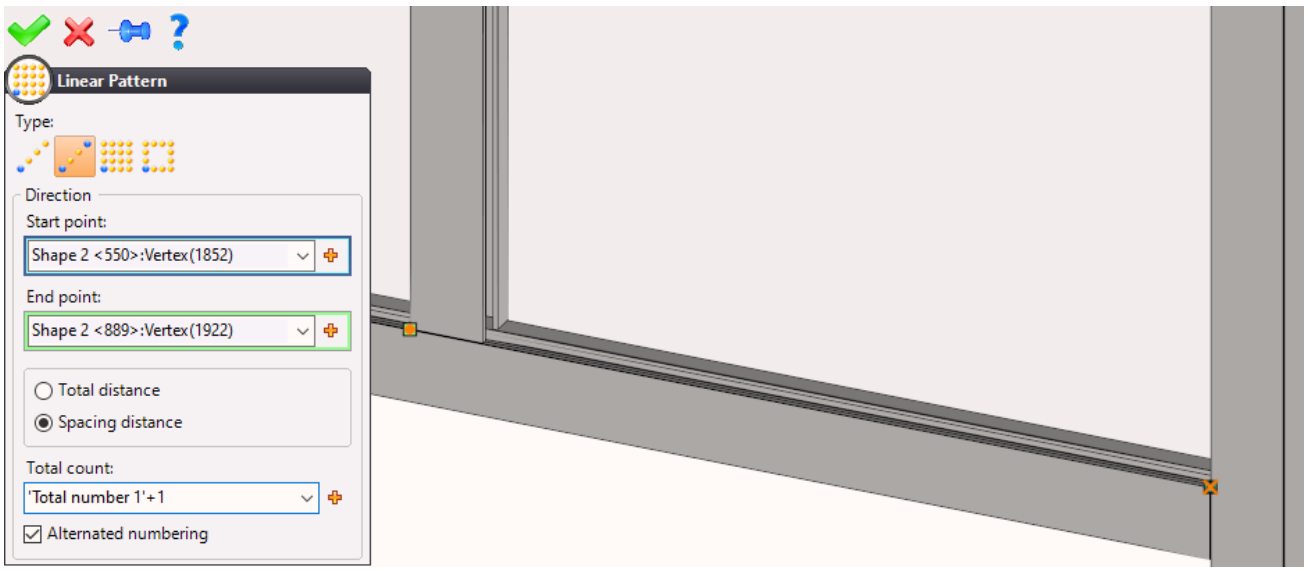
- Select the **Construction** > **Repetition** command.
- Select the following folding extruded T bar as the **entity** and select the constrained linear pattern (in this case, **Pattern 1**).



The glass and glazing beads will have an additional repetition occurrence compared to the T extruded bars.

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

- Select the glass and the glazing beads as the **entities**.
- In the **Pattern** field, click on the icon and select the **Linear Pattern** command.
- Select the **Line by two points** mode and select the points as shown below (starting point on the right).



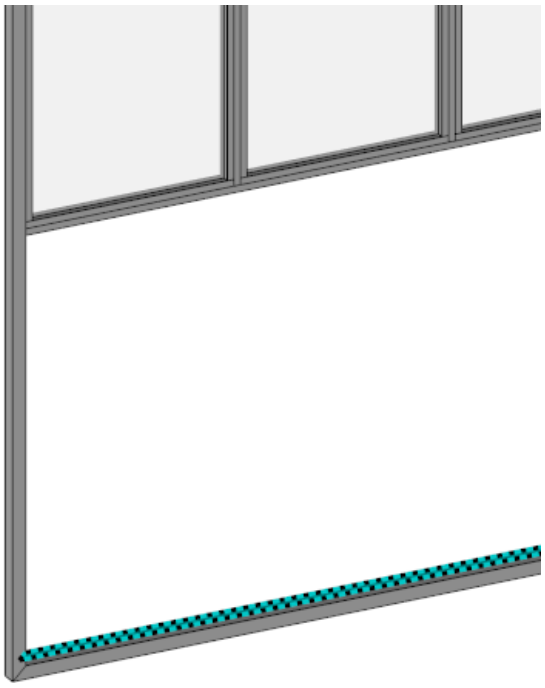
- In the **Total count** field, select the parameter created by the pattern of the T extruded bars from the drop-down list and add *1*.

We add 1 because the basic pattern is that of the interior uprights and it is necessary to have an additional glass.

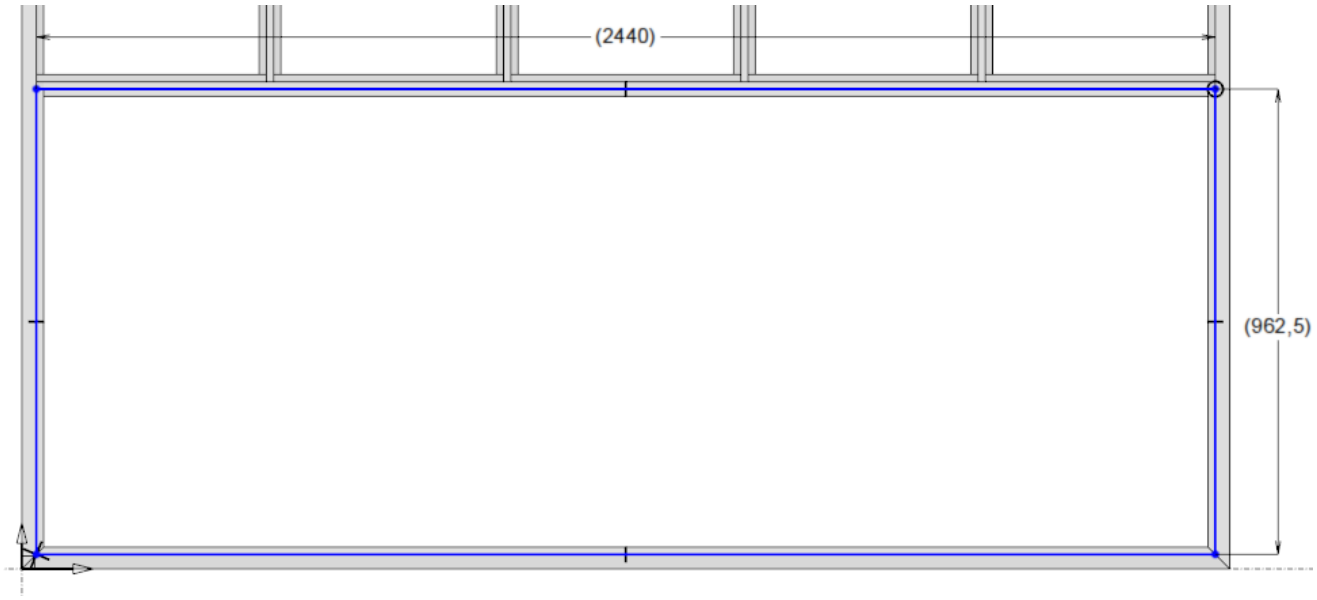
- Click on to **confirm** the pattern and the repetition.

Creating the bottom filling

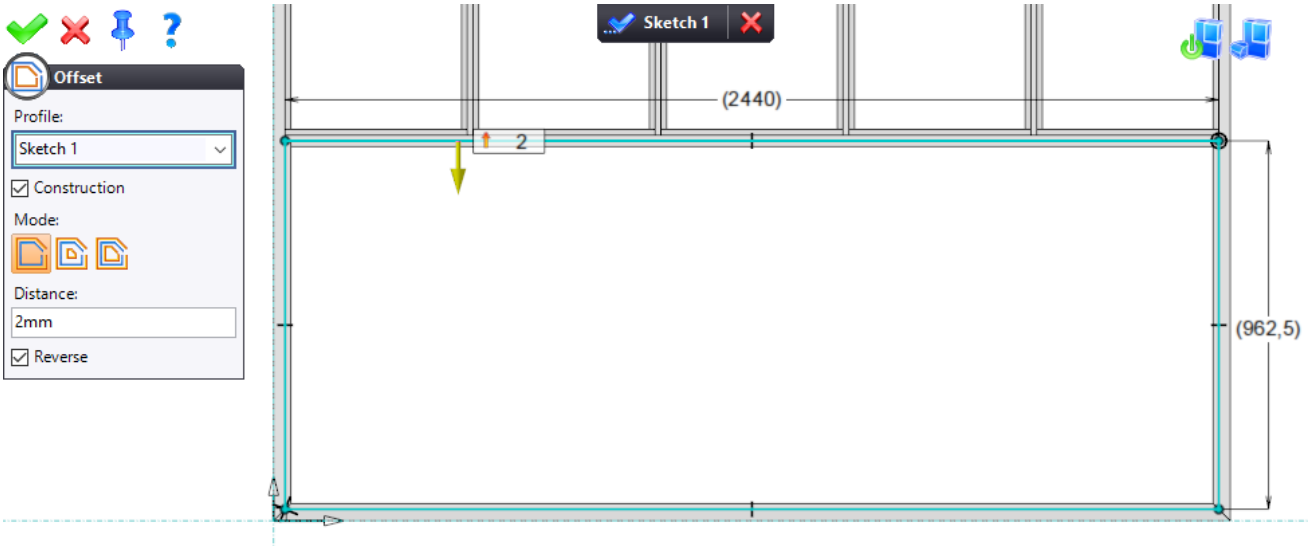
- Right-click on the following face as shown below and select the **In Place Part** command.



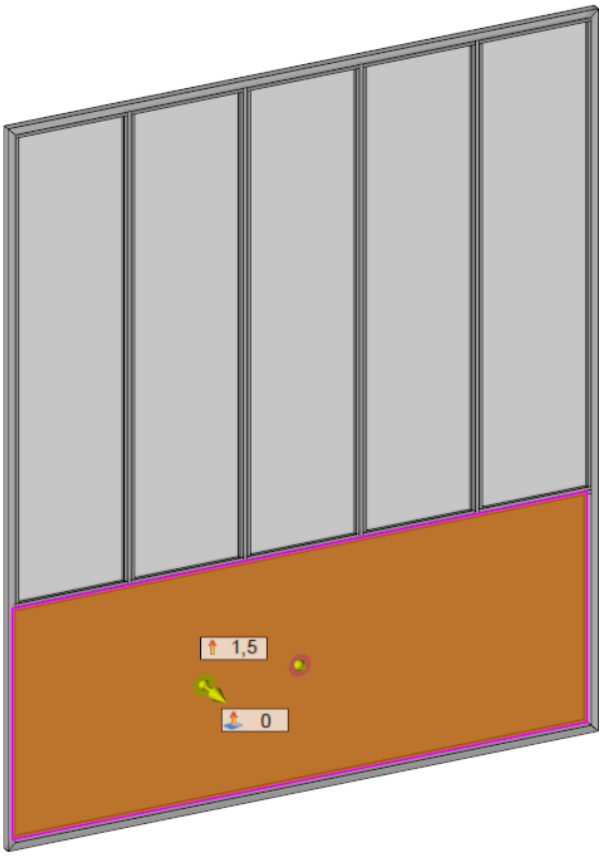
- Draw the following rectangle.



- Add an **offset** profile with a distance of *2mm* toward the center of the rectangle. Make sure you check the **Construction** box.

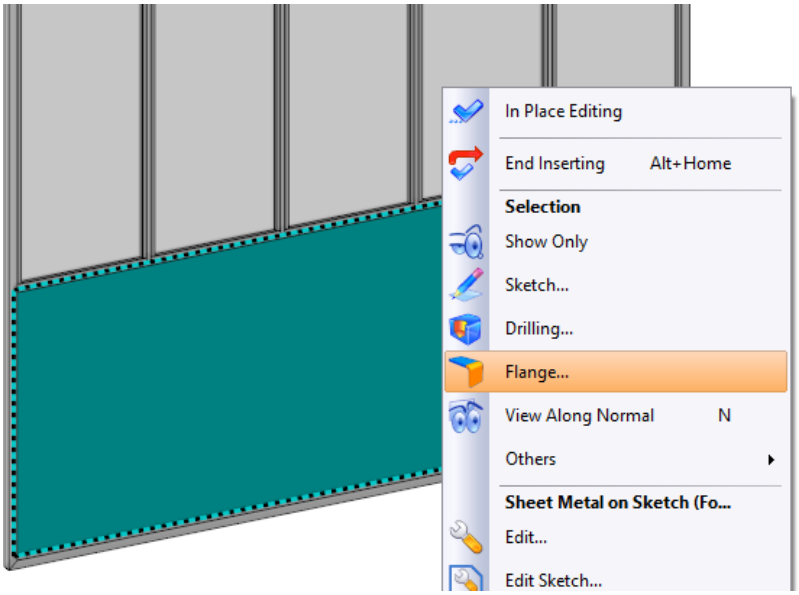


- Right-click in the graphics area with no active selection and select the **Sheet Metal on Sketch** command.
- Check the direction and enter a **thickness** of *1.5mm*.

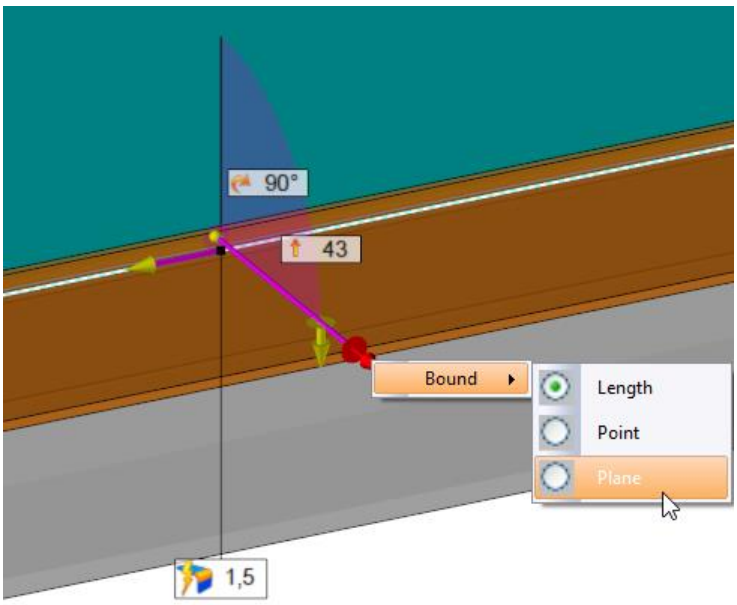


- Click on to **confirm** the operation.

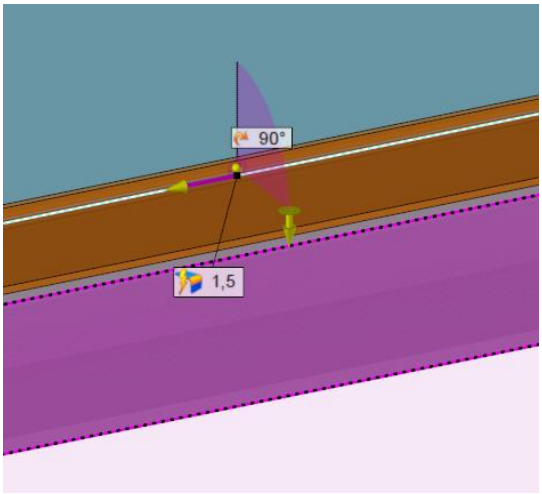
- Right-click on the following face as shown below and select the **Flange** command.



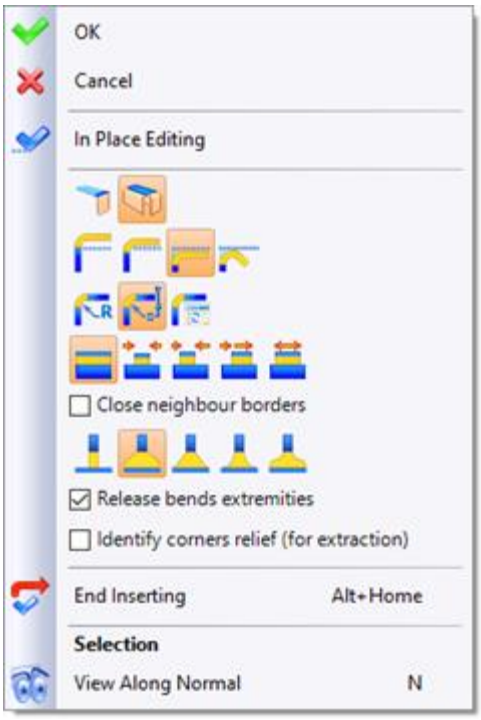
- Right-click on the length arrow and select **Bound** > **Plane**.



- Select the plane as shown below, then add an **offset** of *2mm*.



- Right-click in the graphics area with no active selection and make sure that the following options are selected.

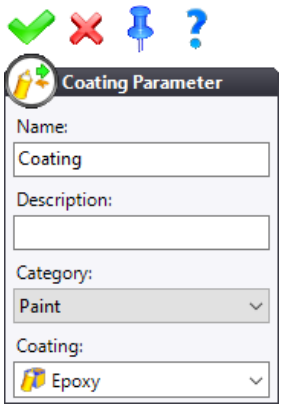


- Click on **OK** to **confirm** the operation.
- **Confirm** the in-place editing of the part.

Parameterizing the coating

The purpose is to create a standard parameterized glass partition. It may be interesting to be able to manage its color.

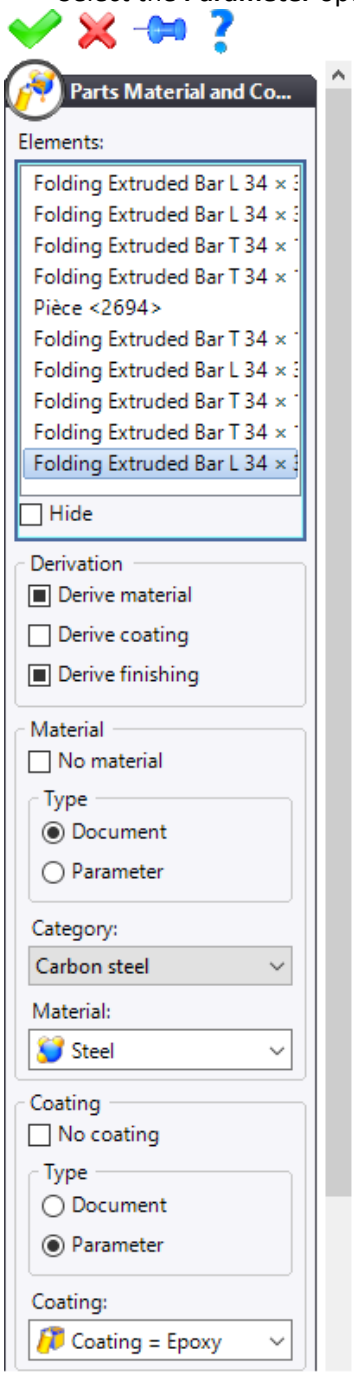
- Select the **Construction** > **Parameters** > **Other Parameters** > **Coating Parameter** command.
- Adjust the dialog box as shown below.

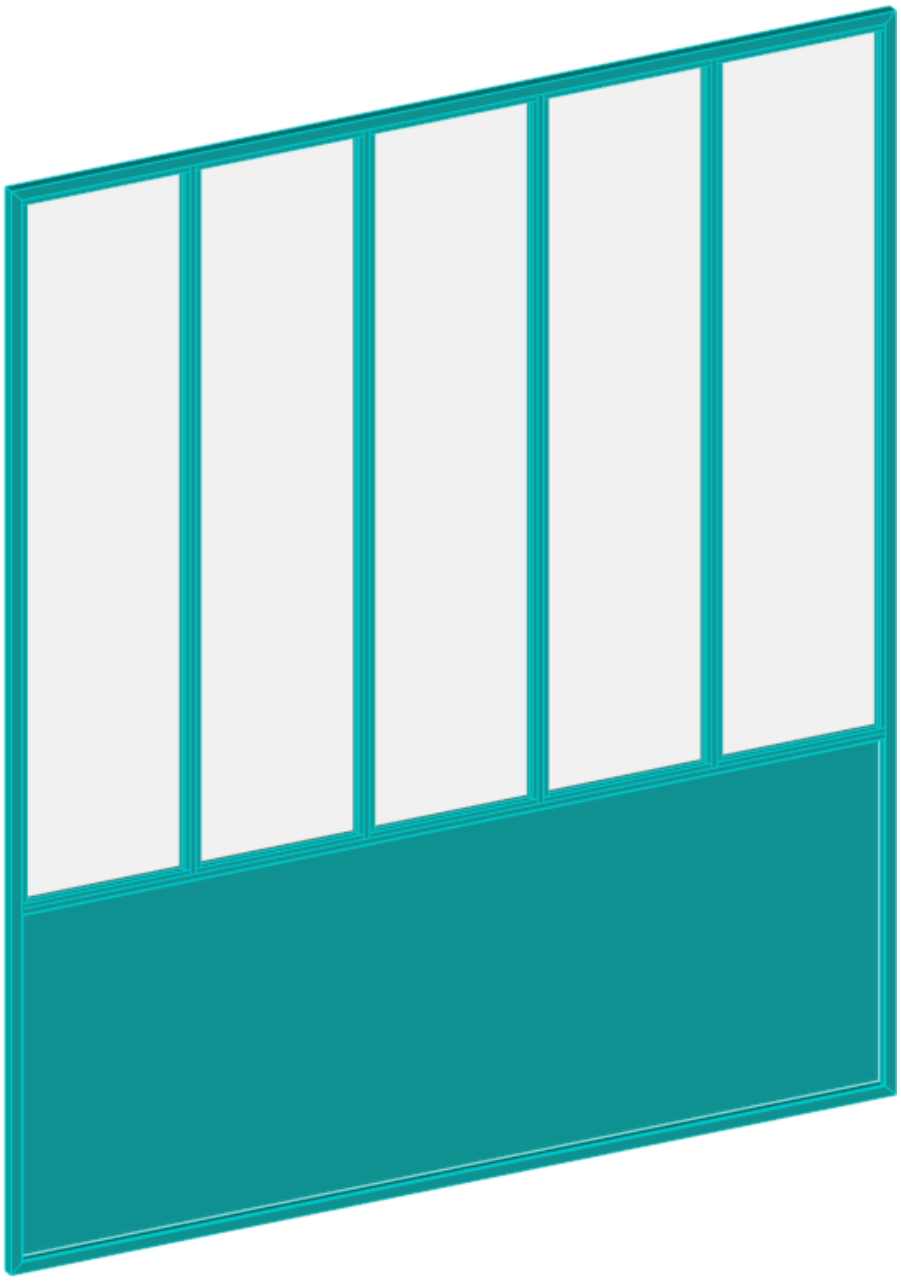


- Click on to **confirm** the operation.

We will now define the parts on which we will use this parameter and therefore apply this coating.

- Select the **Tools** > **Parts Material and Coating** command.
- Adjust the following parameters:
  - In the **Elements** field, select all parts except the glasses.
  - Uncheck the **Derive coating** box. This indicates that you do not want to use the original coating of the extruded bar.
  - Uncheck the **No coating** box.
  - Select the **Parameter** option and the **Coating** parameter.





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

Creating the family

Before creating the family, it can be interesting to parameterize the description. In a project, if several glass partitions of different dimensions are inserted, the parameterization will allow you to know their dimensions in the bill of materials and thus to distinguish between them.

- From the Operations tree, open the folder named *Document name parameterization*.
- Right-click on the **Parameterized (Name)** operation and select the **Others** > **Insert Before** command.
- From the Entities tree, open the **Parameters** > **System Parameters** folders.
- Right-click on the **Description = "Glass partition"** entity and select the **Others** > **Parameterized** command.
- In the **Value** field, enter the *Glass partition W[Width] x H[Height] x B[HeightBase]* formula.

The convention for parameterization is as follows:

- for parameters: [ParameterName];
- for **TopSolid**'s system parameters: [$EnglishParameterName].

The list of system parameters is available in the **TopSolid** online help by pressing the **F1** key when parameterizing the description or any other text.

[$Author]	Retrieves the author of the document.
[$Code]	Retrieves the code of the document.
[$CreationDate]	Retrieves the creation date of the document.
[$Description]	Retrieves the description of the document.
[$ComplementaryPartNumber]	Retrieves the complementary part number of the document.
[$MajorRevision]	Retrieves the major revision of the document.
[$MinorRevision]	Retrieves the minor revision of the document.
[$ModificationDate]	Retrieves the modification date of the document.
[$Name]	Retrieves the name of the document.
[$PartNumber]	Retrieves the part number of the document.
[$Thickness]	Retrieves the thickness parameter of the created sheet metal part.

- Click on to **confirm** the description parameterization.
- Click on the **End inserting** icon in the document's tab.
- **Save** the document (**Ctrl** + **S**).

The tab's name should now include the dimensions of the glass partition.

- Right-click on the assembly document's tab and create a  **Family** document.

The family documents will allow the glass partition to be declined in different dimensions/configurations.

The following two strategies are possible:

- **Using codes**: Predefined codes are integrated. Then, you only have to select the desired code and **TopSolid** will provide the right glass partition. This operation is ideal for users who want to have a catalog of references.
- **Using drivers**: The designer of the family selects the parameters he wants to convert into drivers. Then, the driver values are provided and the family generates the associated component.

For more flexibility, you can define codes as well as drivers. If a parameter is used in both cases, priority is given to the drivers.
You can also have optional drivers. These optional drivers allow you to break free from the rigid framework of codes.

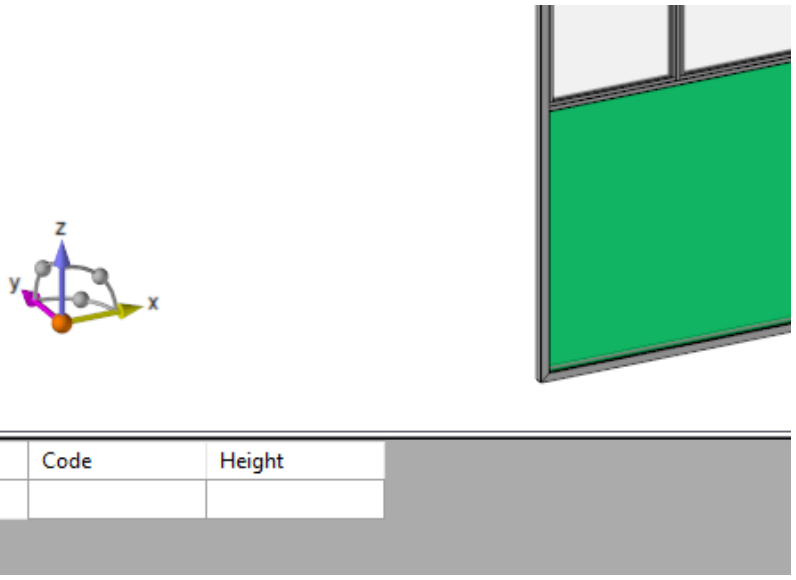
Example:

The glass partition has two codes:

- Model A:
  - Height = 2500
  - Width = 2000
  - Base = 1000
- Model B:
  - Height = 2000
  - Width = 1500
  - Base = 800

From time to time, it may be necessary to use the **Model A** code, but you may want to enter a special **width**. In this case, the **Width** parameter will be both in the codes AND the optional drivers. Therefore, priority is given to the codes.

- From the Entities tree, open the **Generics** folder which contains all the parameters of the glass partition.
- From the Entities tree, drag and drop the **Height** parameter to the right of the **Code** column in the table at the bottom of the screen.



- Repeat the operation with the **Width** and **HeightBase** parameters.
- In the code table, enter the following values:

	Code	Height	Width	HeightBase
▶	2500 x 2000	2500mm	2000mm	1000mm
	2000 x 1500	2000mm	1500mm	800mm
*				

- To view the code, right-click on the first column and select **View Instance**.

TopSolid generates the component in the graphics area.

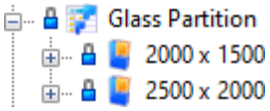
We want to be able to select the coating each time the component is used.

- From the Entities tree's **Generics** folder, right-click on the **Coating** parameter and select **Move to Drivers**.
- In the **Drivers** folder, double-click on the **Coating** parameter.
- In the **Description** field, enter *Glass partition coating*. This allows an alternative name to be exposed to the parameter name to the family user.

As explained above, we want to be able to enter a special width if necessary.

- From the **Generics** folder, right-click on the **Width** parameter and select **Move to Optional Drivers**.
- Repeat the operation for the **Height** parameter.
- Rename the family *Glass partition* and **check it into the vault**.

TopSolid checks all the referenced documents into the vault and also generates the various created codes.
The instances generated by the family can be found under the family document in the Project tree.



Future instances will be added to the sequence.

Reminder: Here is a reminder of the different terms used:

- **Generic**: Part or assembly document (or 2D modeling) in which the component is modeled and parameterized.
- **Family**: Document that generates the different instances according to the defined codes and drivers.
- **Instance**: Family result. It is very similar to the generic documents, except that it integrates the values selected by the user.

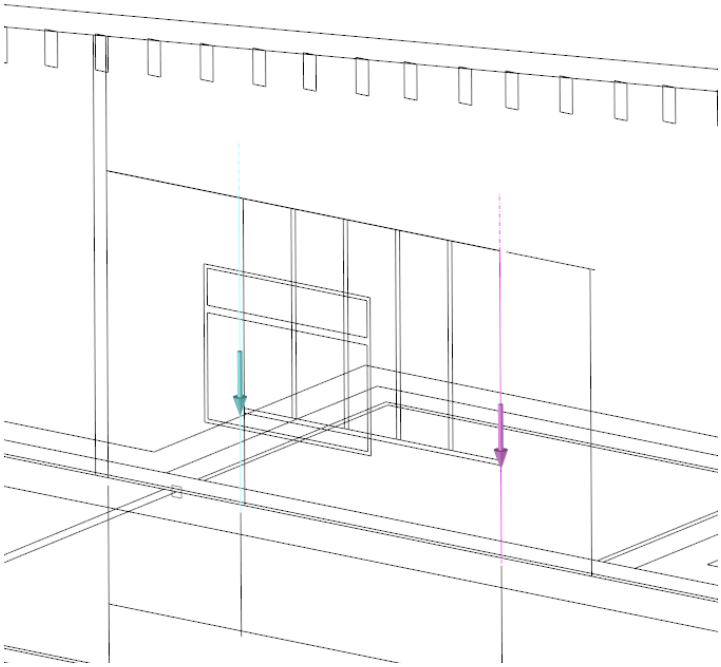
- Close the *Glass partition* family and its generic *Glass partition W2500 x H3000 x B1000*.

Using the glass partition component

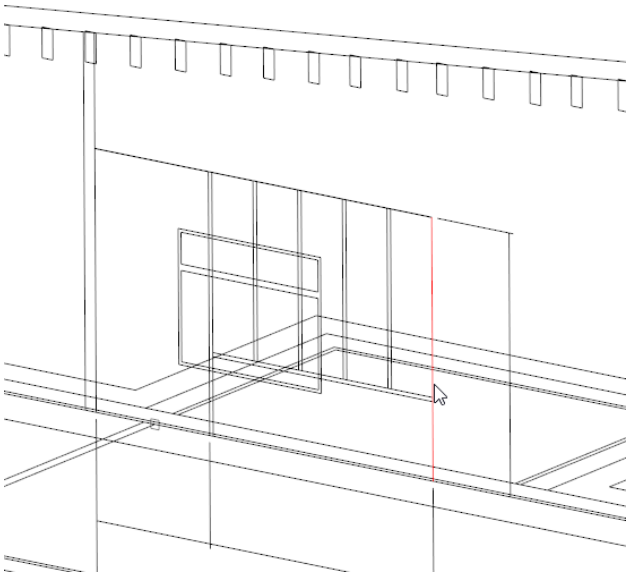
- Open the *Glass partition establishment* assembly that was created at the beginning of the exercise.
- From the Project tree, drag and drop the *Glass partition* family document into the graphics area.

The architect designed a completely tailor-made glass partition. Codes are therefore useless in this context.

- Open the **Optional Drivers** tab.
- In the **Width** field, click on the icon and select the **Distance Parameter** command.
- Enter *GlassPartitionWidth* in the **Name** field, select the **Axis** type and select the two axes as shown below.



- In the **Height** field, click on the icon and select the **Associative value** option.
- Select the segment as shown below.



Reminder: The **Associative value** option makes the value associative to the selection. If the architect's drawing is modified, the glass partition will also be modified.

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

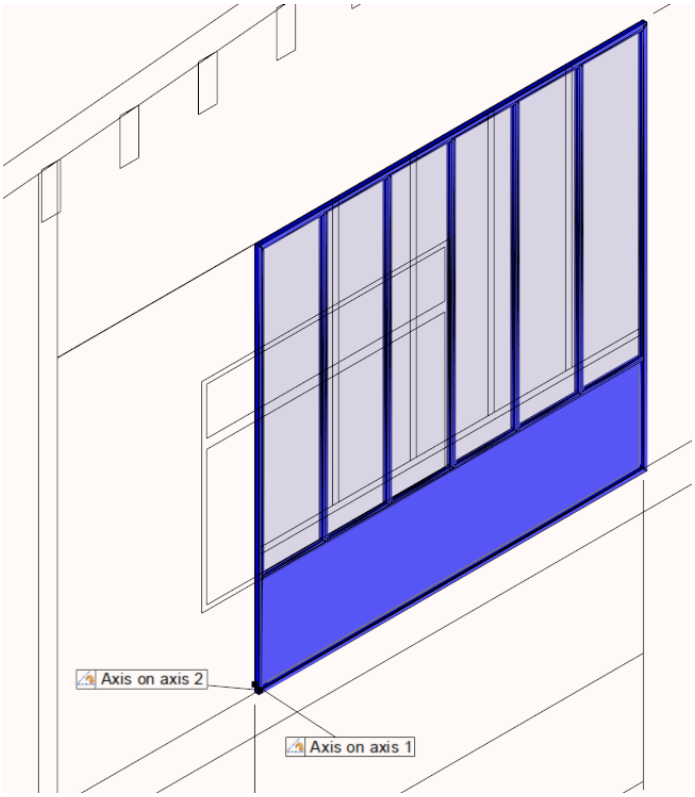
TopSolid switches to the positioning context.

The glass partition is displayed in gray since it is the first element that was inserted in the assembly. Accordingly, it is positioned at the origin and is fixed.

- Right-click on the glass partition and select the **Unfix** command.

The glass partition turns pink.

- Using the **Assembly** > **Axis on Axis** command, position the glass partition in place of the drawing as shown below.



- **Confirm** the positioning.
- **Save** the document (**Ctrl** + **S**).

The glass partition is complete and positioned in its environment.

We will now produce some of the drafting documents.

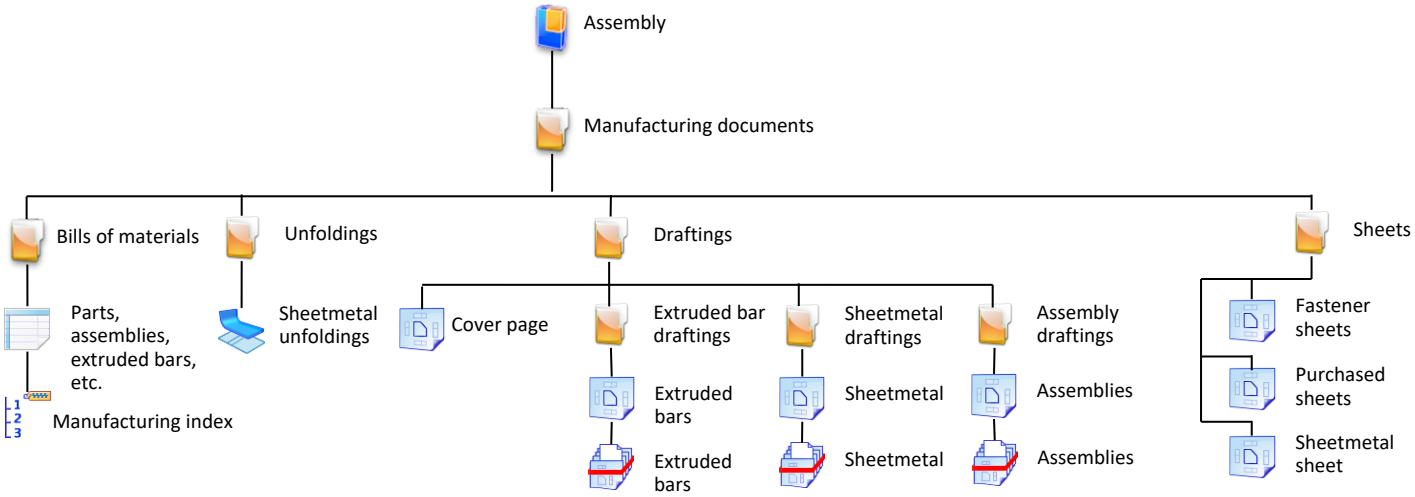
Automatic generation of the manufacturing folder (Work)

Introduction

TopSolid has a **Work** document which allows you to automate various actions:

- Create folders
- Export (PDF, DXF, DSTV, etc.)
- Create documents:
  - Bills of materials
  - Draftings
  - Unfoldings
  - Drafting bundles
  - Machining/cutting
  - Nesting
  - Etc.
- Perform actions in certain documents:
  - Create manufacturing indexes
  - Reinitialize the indexes
  - Etc.

These actions call for processing and can be linked to each other. **TopSolid'Steel** is delivered with an example of how to create the structure below.



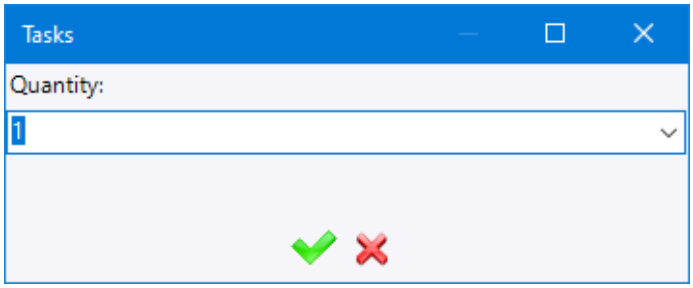
This work starts directly on an assembly and runs automatically. It will:

- Create a structure with folders directly in the Project tree.
- Generate the following bills of materials:
    - **Flat - Parts**: Allows you to regroup parts with identical geometry and properties, then to generate the manufacturing index for the parts.
    - **Flat - Parts for Manufacturing**: Same as above, but displays the manufacturing indexes.
    - **Flat - Assemblies**: Allows you to regroup assemblies with identical properties, then generate manufacturing indexes for the assemblies.
    - **Flat - Assemblies for Manufacturing**: Same as above, but displays the manufacturing indexes.
    - **Multi-level**: Describes the hierarchical structure of the assembly with its sub-assemblies, etc.
    - **Multi-level for Manufacturing**: Same as above, but displays the manufacturing indexes.
    - **Parts - Back References**: Allows you to know in which assembly(s) each part is assembled.
    - **Assembly - Back References**: Same as above but for assemblies.
    - **Sheet Metal for Manufacturing**: Allows you to isolate the sheet metal parts with their manufacturing indexes. It also allows you to generate unfolding documents.
    - **Extruded Bars for Manufacturing**: Allows you to isolate the extruded bars with their manufacturing indexes. It also allows you to generate an automatic drafting document of the extruded bars.
- Generate sheet metal unfoldings from the **Sheet Metal for Manufacturing** bill of material document.
- Generate a drafting document by unfolding, then put all the elements in a drafting bundle.
- Generate a drafting document by extruded bar from the **Extruded Bars for Manufacturing** bill of material document, then put all the elements in a drafting bundle.
- Same but for the assemblies from the **Flat - Assemblies for Manufacturing** bill of material document.
- Generate a cover page for the folder in the form of a drafting document in A3 format.
- Create the sheets (BOM in drafting format:
    - fasteners used;
    - purchased elements (with the **Production Type** = **Purchased** property);
    - sheet metals.

Note: This automated document package can be copied to be customized/completed.

Use

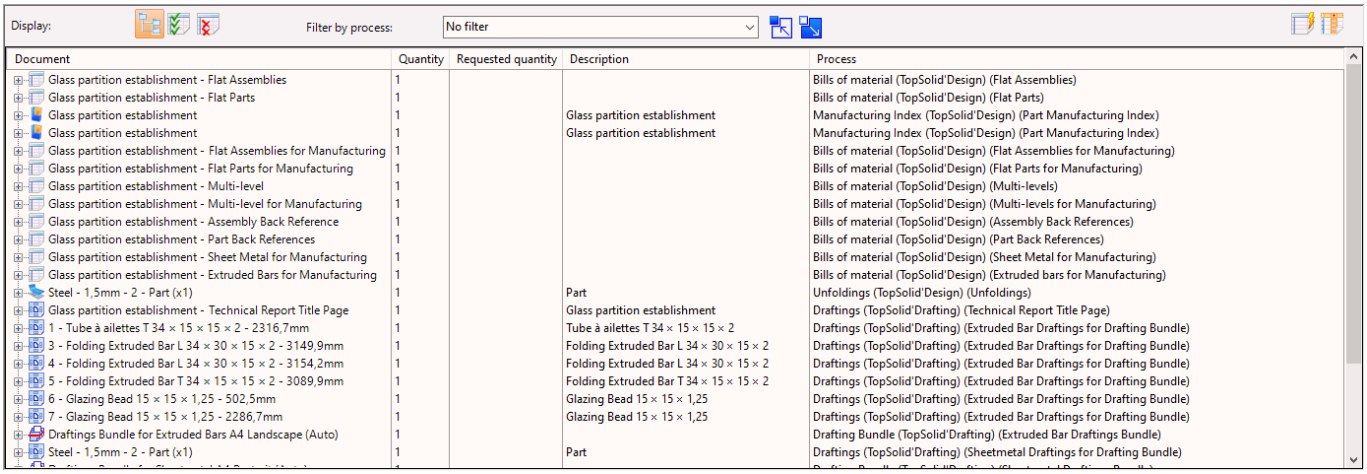
- Right-click on the *Glass partition establishment* assembly document's tab and select the  **Work** command. You can create the work document from the *Glass partition establishment* document in the Project tree.
- Select the **TopSolid'Steel** – **Bill of materials** - **Unfoldings** - **Draftings** - **Sheets** - **Drafting bundles** template.
- **Confirm** the default task quantity of 1.



Note: The quantities are useful for nesting or using the optional **TopSolid** cutting modules. This dialog box can no longer be displayed by selecting **Tools** > **Options** > **Work Manager** > **Tasks** and unchecking the **Manage nesting data** box.

TopSolid creates a new work document in the project, retrieves all the necessary templates and starts the process.

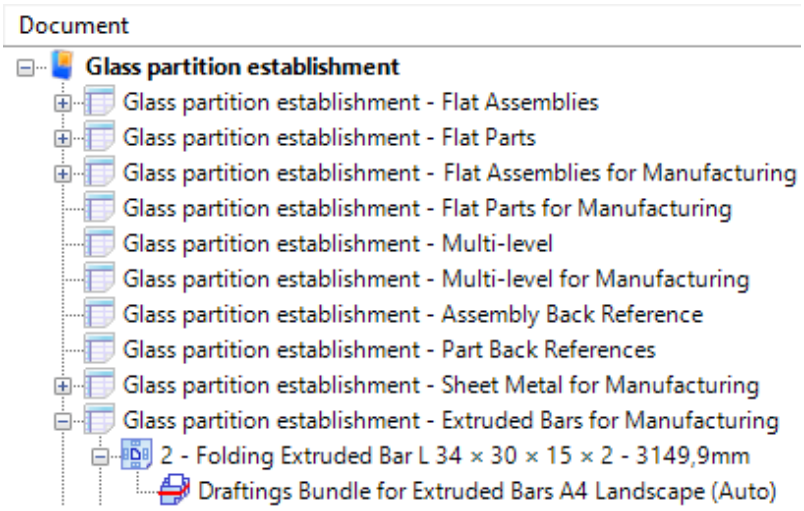
Once completed, **TopSolid** displays the result of the work document, i.e. all of the documents produced automatically.



Document	Quantity	Requested quantity	Description	Process
Glass partition establishment - Flat Assemblies	1			Bills of material (TopSolid'Design) (Flat Assemblies)
Glass partition establishment - Flat Parts	1			Bills of material (TopSolid'Design) (Flat Parts)
Glass partition establishment	1		Glass partition establishment	Manufacturing Index (TopSolid'Design) (Part Manufacturing Index)
Glass partition establishment	1		Glass partition establishment	Manufacturing Index (TopSolid'Design) (Part Manufacturing Index)
Glass partition establishment - Flat Assemblies for Manufacturing	1			Bills of material (TopSolid'Design) (Flat Assemblies for Manufacturing)
Glass partition establishment - Flat Parts for Manufacturing	1			Bills of material (TopSolid'Design) (Flat Parts for Manufacturing)
Glass partition establishment - Multi-level	1			Bills of material (TopSolid'Design) (Multi-levels)
Glass partition establishment - Multi-level for Manufacturing	1			Bills of material (TopSolid'Design) (Multi-levels for Manufacturing)
Glass partition establishment - Assembly Back Reference	1			Bills of material (TopSolid'Design) (Assembly Back References)
Glass partition establishment - Part Back References	1			Bills of material (TopSolid'Design) (Part Back References)
Glass partition establishment - Sheet Metal for Manufacturing	1			Bills of material (TopSolid'Design) (Sheet Metal for Manufacturing)
Glass partition establishment - Extruded Bars for Manufacturing	1			Bills of material (TopSolid'Design) (Extruded bars for Manufacturing)
Steel - 1,5mm - 2 - Part (x1)	1		Part	Unfoldings (TopSolid'Design) (Unfoldings)
Glass partition establishment - Technical Report Title Page	1		Glass partition establishment	Draftings (TopSolid'Drafting) (Technical Report Title Page)
1 - Tube à ailettes T 34 × 15 × 15 × 2 - 2316,7mm	1		Tube à ailettes T 34 × 15 × 15 × 2	Draftings (TopSolid'Drafting) (Extruded Bar Draftings for Drafting Bundle)
3 - Folding Extruded Bar L 34 × 30 × 15 × 2 - 3149,9mm	1		Folding Extruded Bar L 34 × 30 × 15 × 2	Draftings (TopSolid'Drafting) (Extruded Bar Draftings for Drafting Bundle)
4 - Folding Extruded Bar L 34 × 30 × 15 × 2 - 3154,2mm	1		Folding Extruded Bar L 34 × 30 × 15 × 2	Draftings (TopSolid'Drafting) (Extruded Bar Draftings for Drafting Bundle)
5 - Folding Extruded Bar T 34 × 15 × 15 × 2 - 3089,9mm	1		Folding Extruded Bar T 34 × 15 × 15 × 2	Draftings (TopSolid'Drafting) (Extruded Bar Draftings for Drafting Bundle)
6 - Glazing Bead 15 × 15 × 1,25 - 502,5mm	1		Glazing Bead 15 × 15 × 1,25	Draftings (TopSolid'Drafting) (Extruded Bar Draftings for Drafting Bundle)
7 - Glazing Bead 15 × 15 × 1,25 - 2286,7mm	1		Glazing Bead 15 × 15 × 1,25	Draftings (TopSolid'Drafting) (Extruded Bar Draftings for Drafting Bundle)
Draftings Bundle for Extruded Bars A4 Landscape (Auto)	1			Drafting Bundle (TopSolid'Drafting) (Extruded Bar Draftings Bundle)
Steel - 1,5mm - 2 - Part (x1)	1		Part	Draftings (TopSolid'Drafting) (Sheetmetal Draftings for Drafting Bundle)

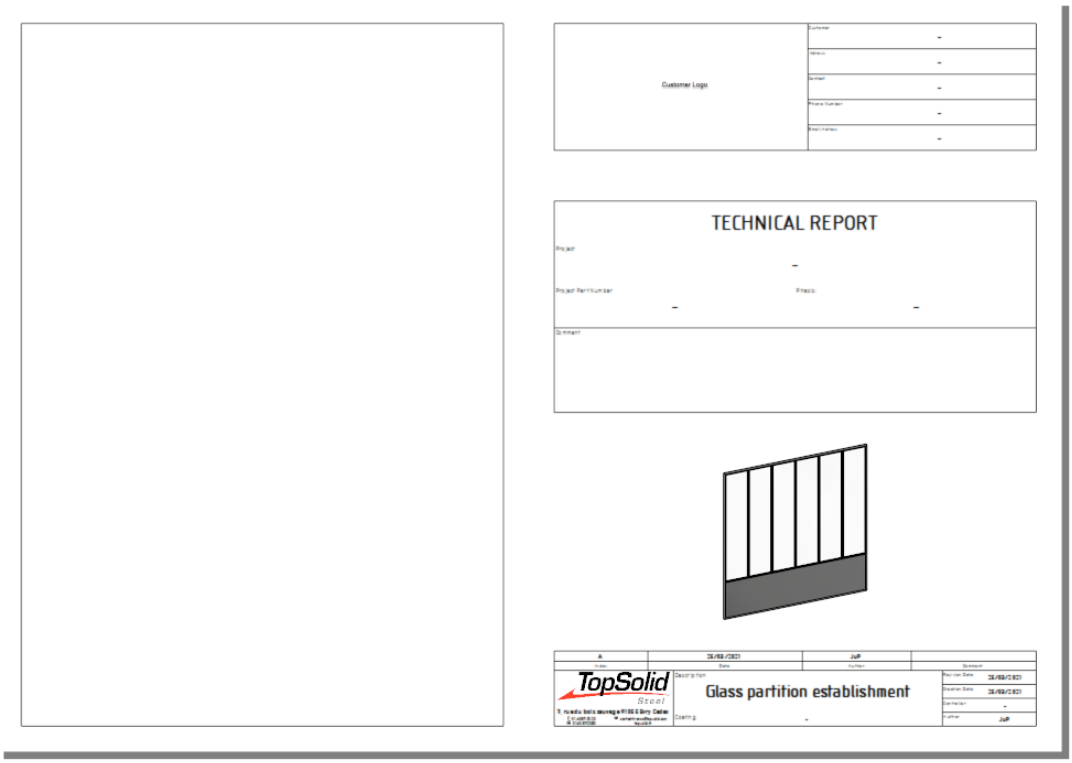
You can display one view per treated task. This makes it possible to better understand the process sequences.

- Select the  **Treated tasks** display mode.
- Expand the **Glass partition establishment** node, then the **Glass partition establishment – Extruded bars for Manufacturing** node and finally the index **2** node.

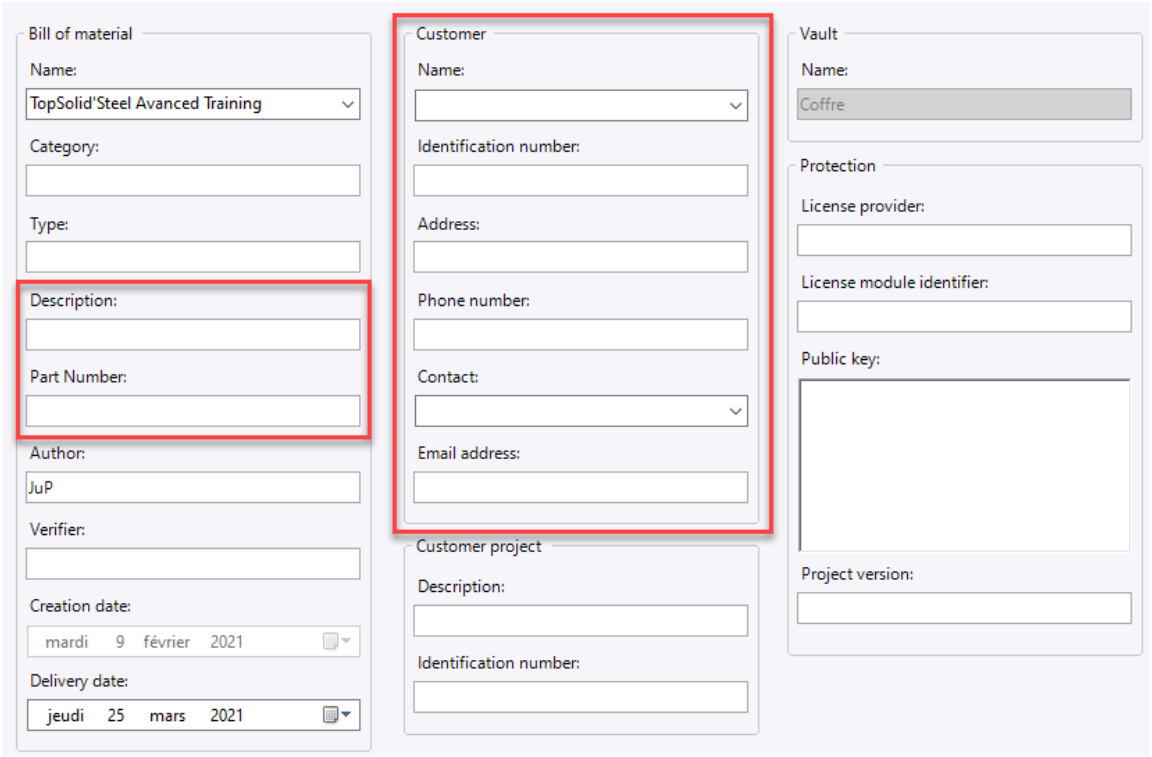


You will see that **TopSolid** generated a bill of materials from which it generated drawings, which it then inserted into a drafting bundle.

- Click on the *Glass partition establishment* drafting document's tab.



The different fields are automatically filled in with the parameters entered in the project document.



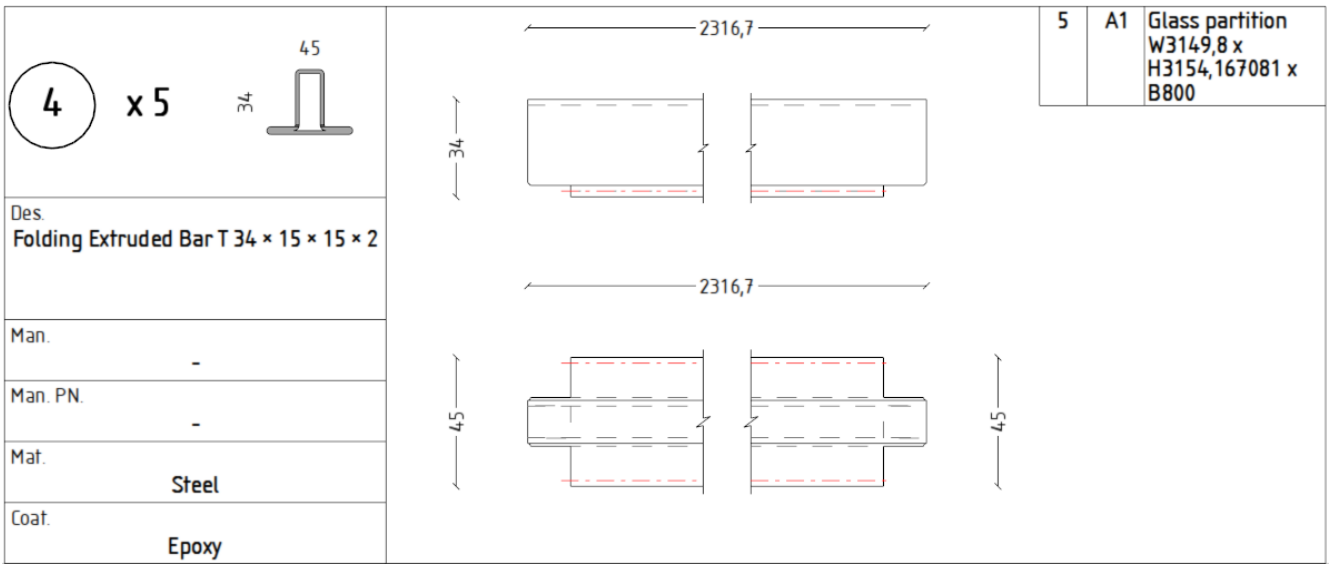
- Click on the *Draftings Bundle for Extruded Bars A4 Landscape (Auto)* document's tab. This document is a drafting bundle.

A drafting bundle is a set of draftings. **TopSolid** automatically regroups them to facilitate printing, checking and management.

In this case, the draftings have a special format in order to insert a maximum of drawings on an A4 format while keeping a certain level of readability. Each drawing remains fully editable.

- Double-click on the drafting document in the lower left corner corresponding to the folding extruded bar (index 6).

The drawing switches to in-place editing.



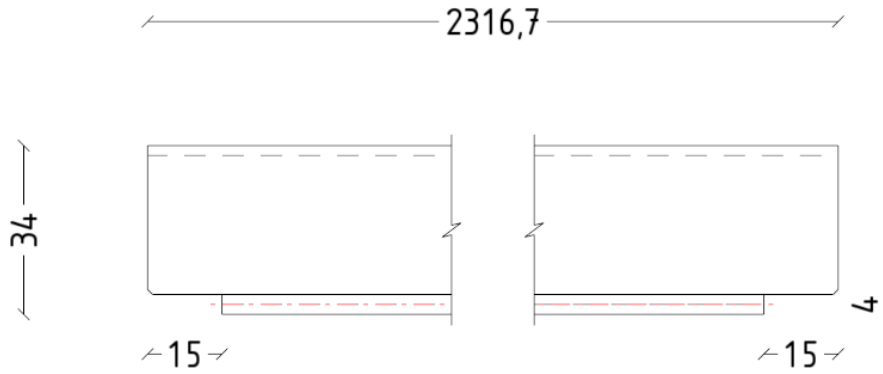
In the first upper left frame, we will find the manufacturing index, the quantity and the section.
Below are the main properties.
In the center, there are two views of the extruded bar with:

- Automatic interruption of the view (this avoids the operated areas such as drillings, cuts, etc.).
- Automatic dimensioning of length/cutting angles.
- Automatic dimensioning of the position of the drillings.

The rest of the dimensioning must be done manually.

Finally, on the right, there is a table of back references that allows you to indicate in which assembly(s) the extruded bars are assembled. In this case, out of the five extruded bars, all five will go into A1 assembly.

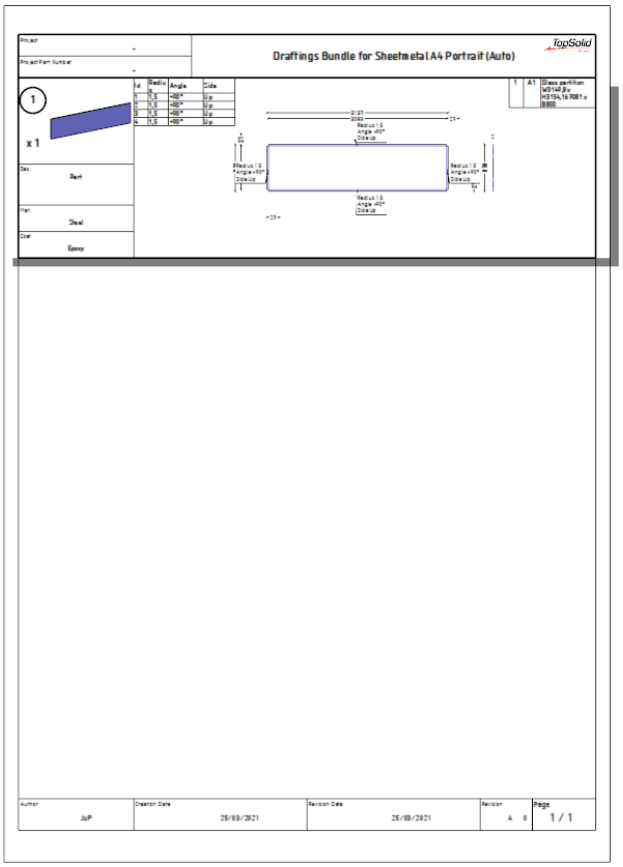
- Add three dimensions for the notches.



- **Confirm** the edition.

The bundle is updated.

- Click on the *Draftings Bundle for Sheetmetal A4 Portrait (Auto)* document's tab.



Here we find the unfoldings with a view, the manufacturing index, the quantity, the back references, etc.

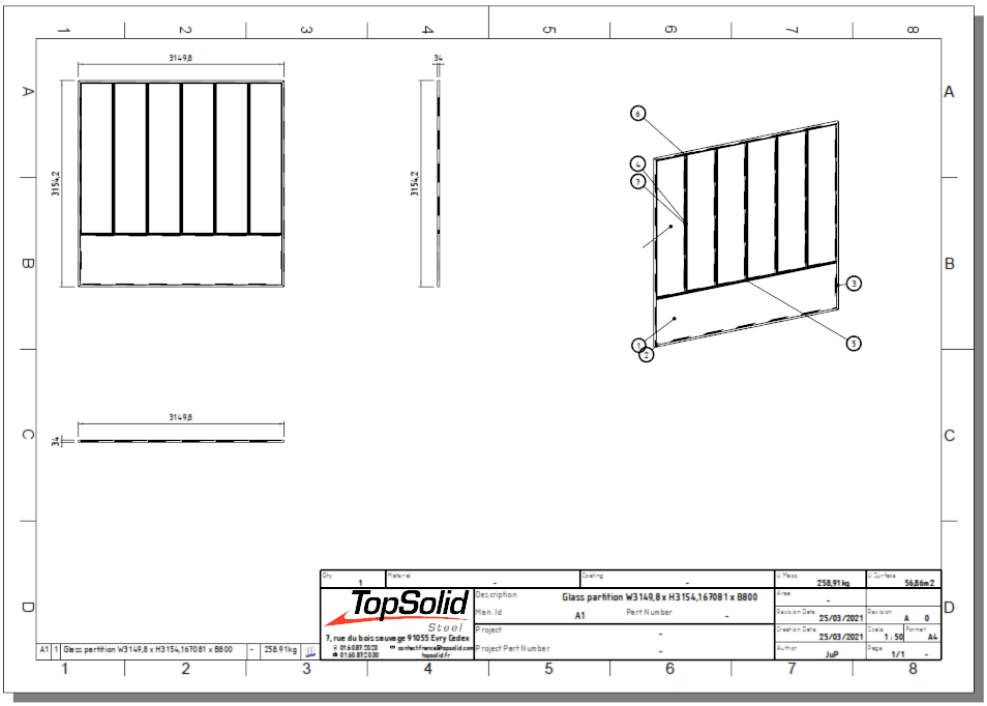
The main view includes:

- the enclosing dimensionings;
- the bending dimensionings;
- bending notes in table form;
- the automatic dimensioning of the position of the drillings.

The rest of the dimensioning must be done manually.

As for the extruded bars, the table of back references is also on the right.

- Click on the *Draftings Bundle for Assemblies A4 Landscape (Auto)* document's tab.



This last bundle concerns the sub-assemblies.

An automatic indexing is created on the isometric view.

The table at the bottom left displays the back references. If the assembly has less than 20 parts, a BOM table is displayed above the title block.

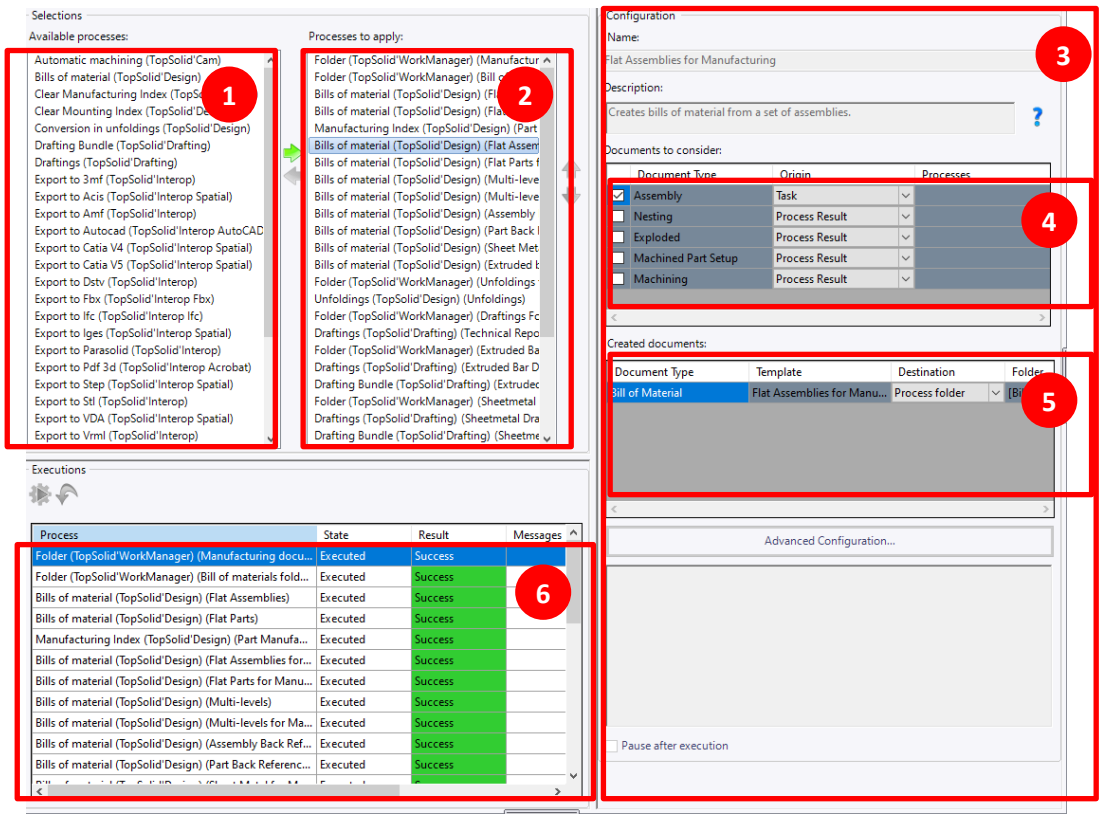
The other three documents below are sheets. They are drafting documents that contain only one bill of materials.

- *Fastener sheet*: Contains all the elements having a screw, nut, bolt function, etc.
- *Purchased sheet:* Contains all the elements having the **Production Type** = **Purchased** property.
- *Sheetmetal sheet:* Contains all the sheet metals.

As seen previously, the work document allows you to generate different manufacturing documents. To date, it does not allow the update of these documents in case of modification of the 3D. If needed, you must add the drawings manually or restart the complete generation.

To cancel (= destroy) the previous documents, you simply have to proceed as follows:

- Return to the work document named *Work 1*.
- Click on the **Processes** tab.



1: Available and usable processes
2: Processes to be applied
3: Configuration of the selected process
4: Source(s) to be considered
5: Template to be used and destination folder
6: Process status

- In the **Executions** (**6**) section, select all the processes (**Ctrl** + **A**) and click on the **Cancel** icon.

TopSolid warns that the documents will be destroyed and changes made to the documents will therefore be lost.

- Click on to **confirm** the cancellation of the processes.

You can now restart the generation of the processes by using the **Execute** command.

Part 4 - Designing a Railing with Managed Sections

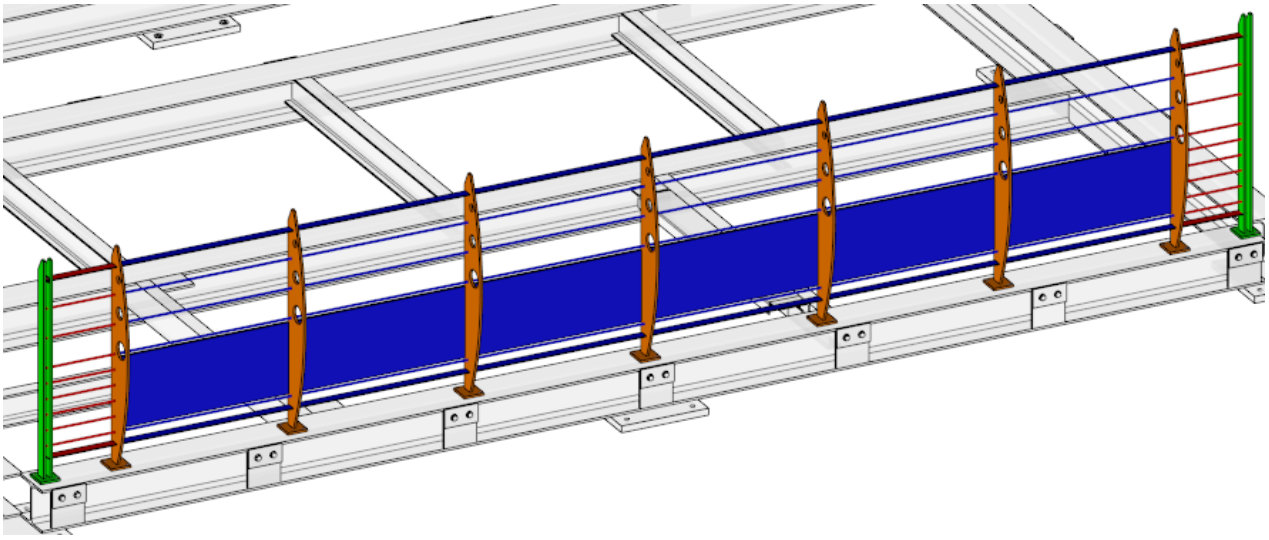
The purpose of this exercise is to design a parametric railing while managing the sections thanks to the distribution command.

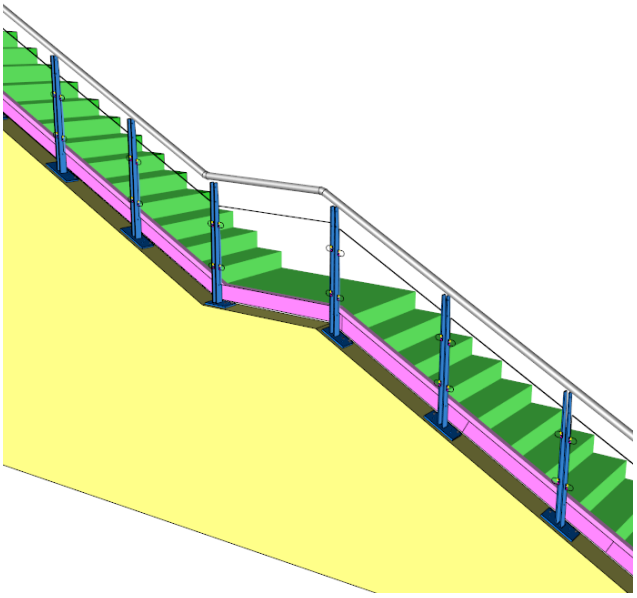
Concepts addressed:

- Using the **Distribution** command
- Providing functions
- Parameterization:
    - Setting conditions of parameters (when)
    - Setting conditions of operations
    - Using associative deletion

Introduction

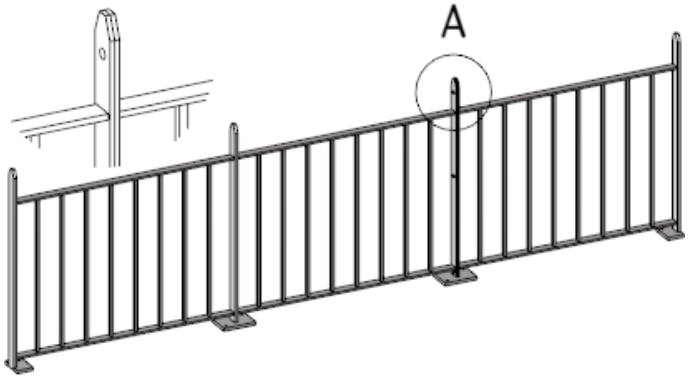
The **Distribution** command allows you to distribute one or two components alternatively with very numerous possibilities.
In practice, on a railing, component 1 could be a post model and component 2 a filling model. At any time, these models can be replaced with other models that you would have designed.



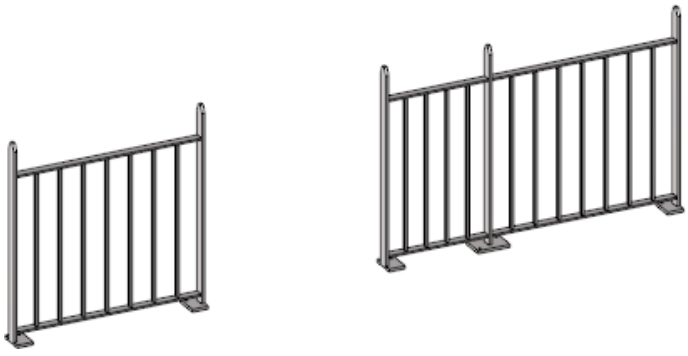


The type of distribution also offers many possibilities. At any point, these variations can occur in the project in a few minutes.
For example, you can (non-exhaustive list):

- edit a type of post in the middle of the distribution (to create a section, for example);



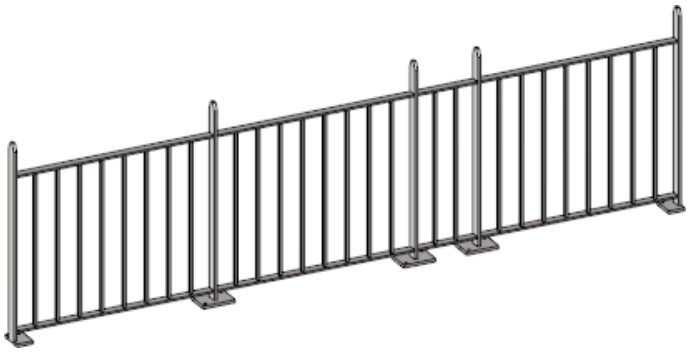
- remove a filling and modify the posts on either side;



- distribute at a fixed distance; the rest will also be either symmetrically distributed or stored on one side;

- distribute with a maximum distance;

- force the width of a filling.



The disadvantage of this is a longer implementation time than what was seen with the railing in the Basics training.

This parameterization becomes profitable in the following cases:

- case with a railing model used in different configurations/dimensions;
- work with a catalog of standardized posts and fillings;
- case requiring great flexibility in implementation;
- etc.

In order to work, the distribution will require the following elements:

- At least one component. This component can be a part or an assembly. In both cases, it must have a family.
- Providing a special function to the generic document. This will allow you to define the positioning points, as well as the orientation and length variable of the component.
- Providing a special function to the family. This will identify this family as distributable.

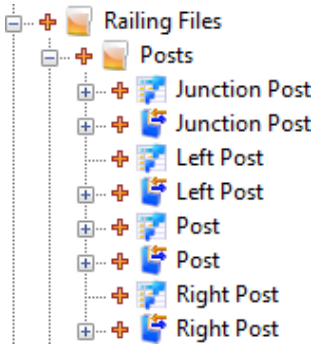
Importing components

- From the Project tree, create a new folder named *4- Railing with sections*.

In order to focus on the distribution command and its use, the posts have already been modeled.

- In the *4- Railing with sections* folder, **import the package** named *Railing Files.TopPkg*.

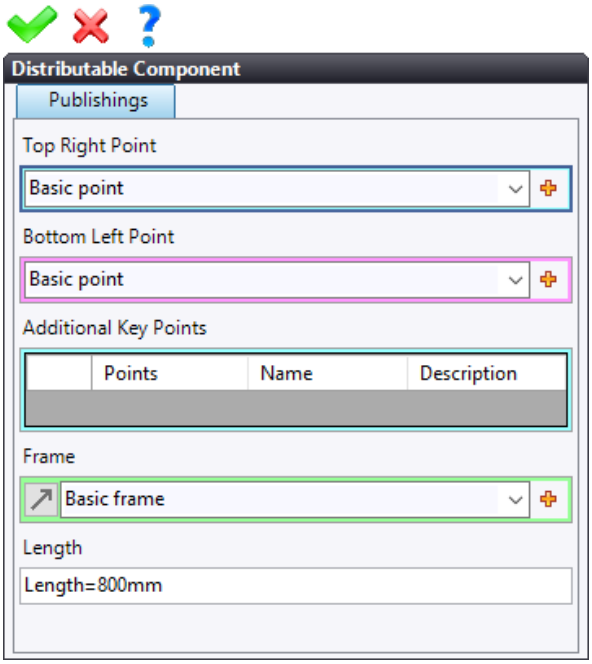
The package contains the following documents.



Configuring the post

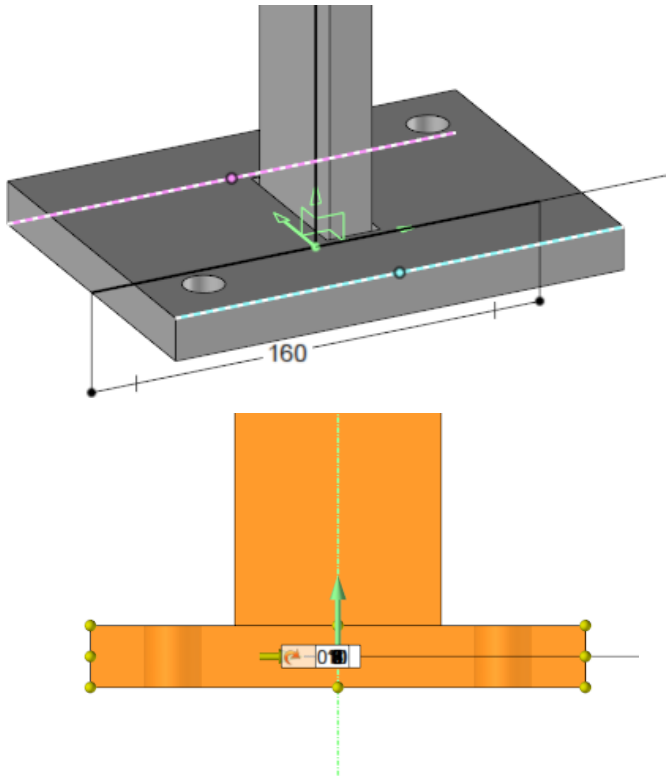
- From the *Railing files* > *Posts* folders, open the *Post* assembly document.
- Select the **Tools** > **Functions** > **Provide Function** command.
- Select the **Distributable Component** function, check the **Create parameter** box, then click on to **confirm** the operation.

The following dialog box appears.

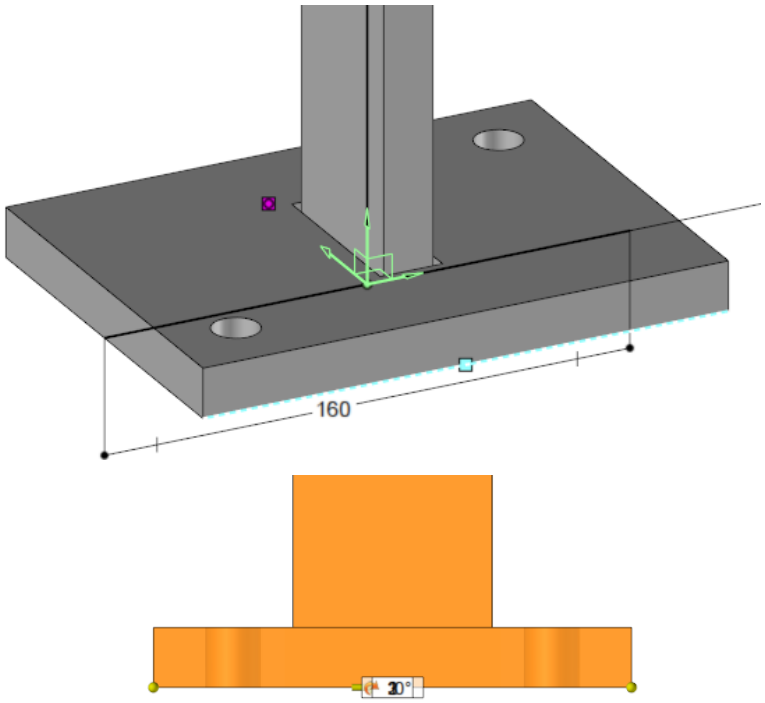


The **Top Right Point** and **Bottom Left Point** fields will allow **TopSolid** to generate the different positioning points (9 in total).
The **Additional Key Points** field allows you to add more positioning points.
The **Frame** field defines the orientation of the component.
The **Length** field defines the usable length of the component when performing the distribution.

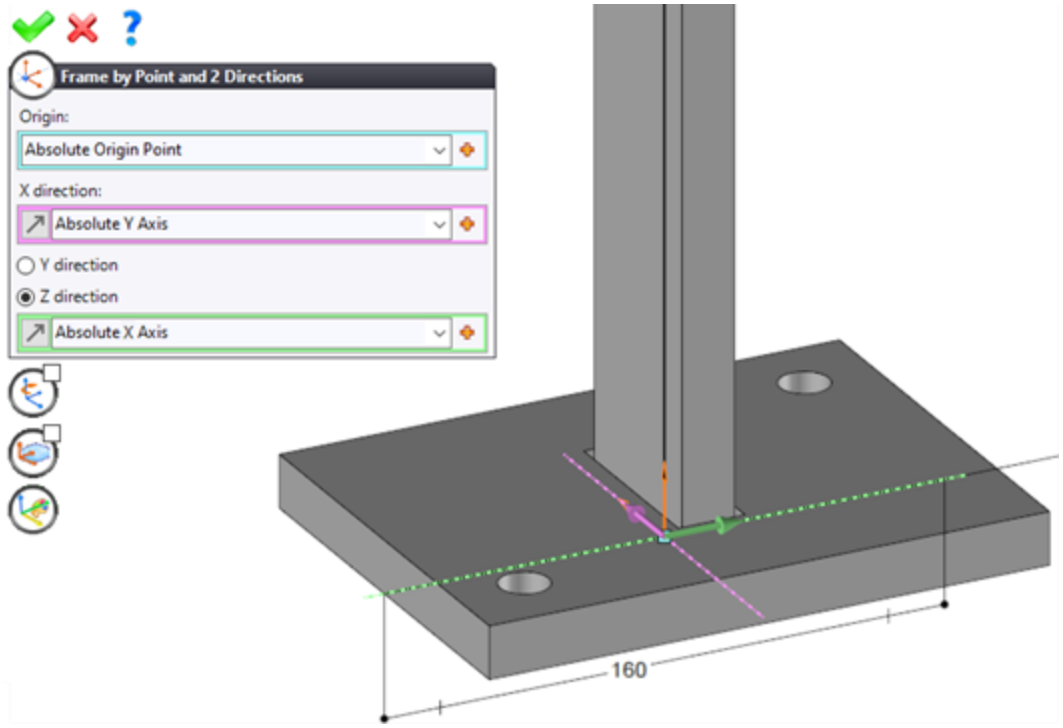
- Select the following two points on the plate.



If the points had been aligned, only three positioning points would have been visible.



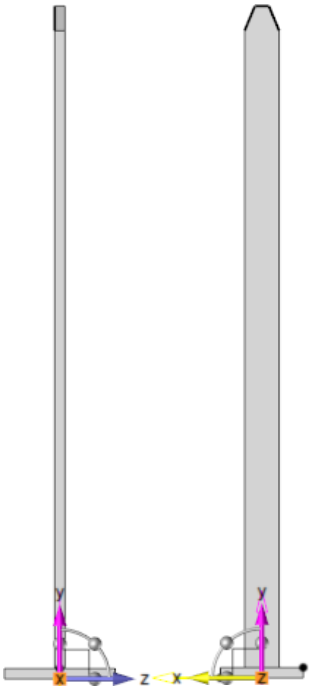
- In the **Frame** field, click on the icon and select the **Frame by Point and 2 Directions** command.
- Adjust the fields as shown below.



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

The convention to be respected is the following:

- The Z axis of the frame must be in the distribution direction.
- The Y axis describes the vertical orientation.
- The X axis describes the horizontal orientation.



- In the **Length** field, you can find the parameter that was created when providing function. Enter a value of *15mm*.

In the case of a post, this value corresponds to the thickness of the post separating the fillings.

- Click on to **confirm** the operation.
- Open the *Post* family document.
- Select the **Tools** > **Provide Driving** command.
- Select **Distributable Family** and click on the icon to move to the next step.
- Select **<Unspecified>**.

Indeed, we do not want **TopSolid** to ask us for the length. This is calculated by the component itself. In our case, it is fixed at 15mm.

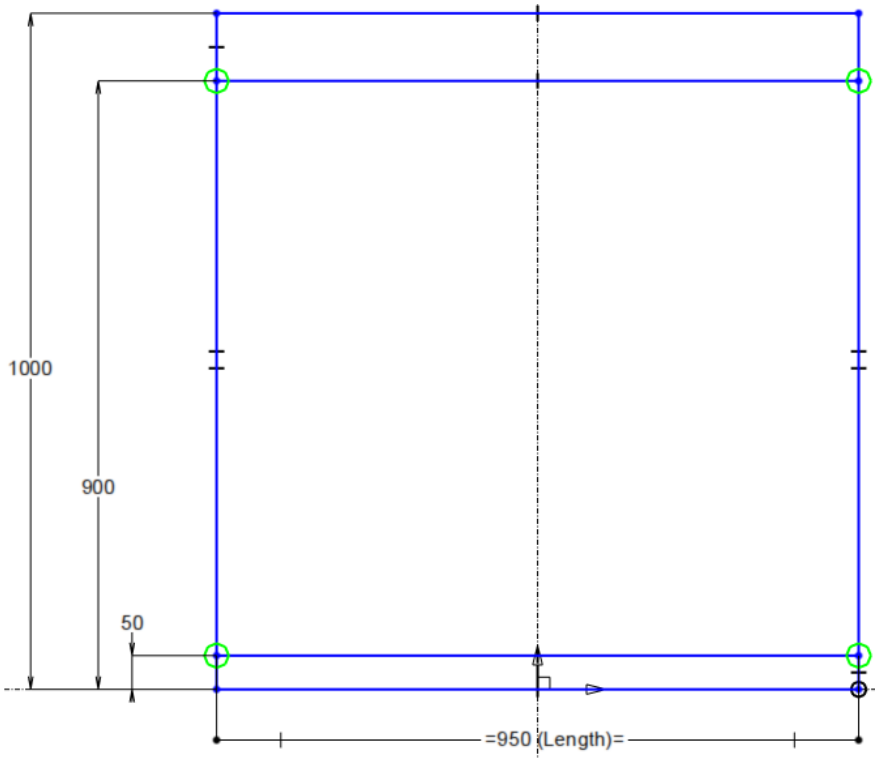
- Click on to **confirm** the operation.

Providing the **Distributable Family** driving indicates to **TopSolid** that this family is not a standard family. When the distribution command is launched, it will appear in the possible choices and, if the family is directly included in an assembly, the distribution command will be automatically launched.

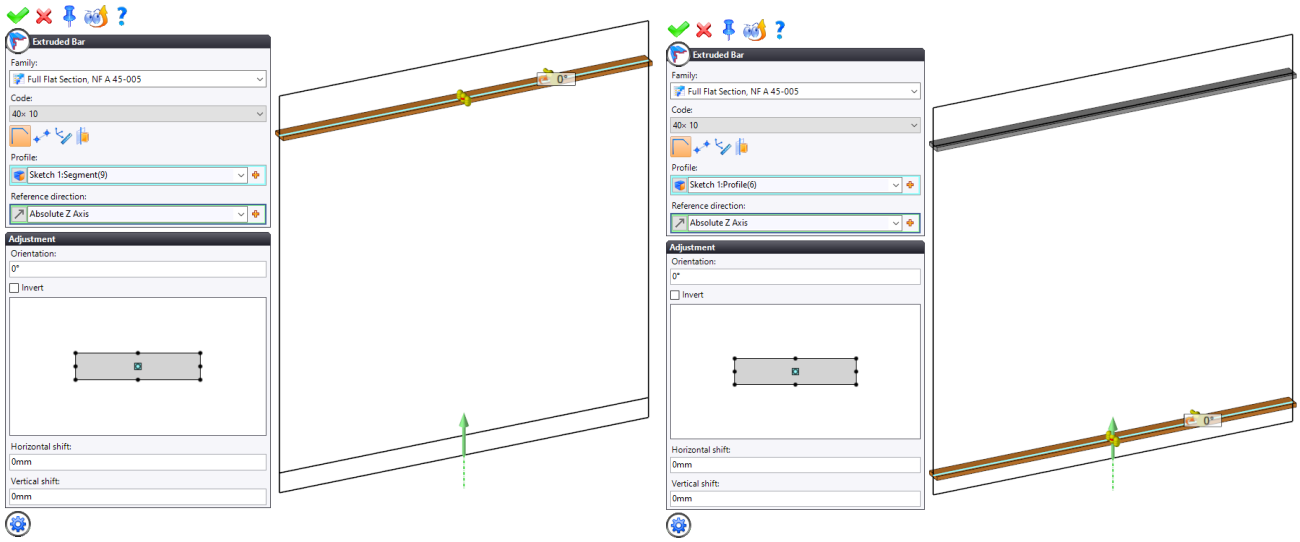
- **Save** (**Ctrl + S**) and close the family and its generic.

Creating the filling

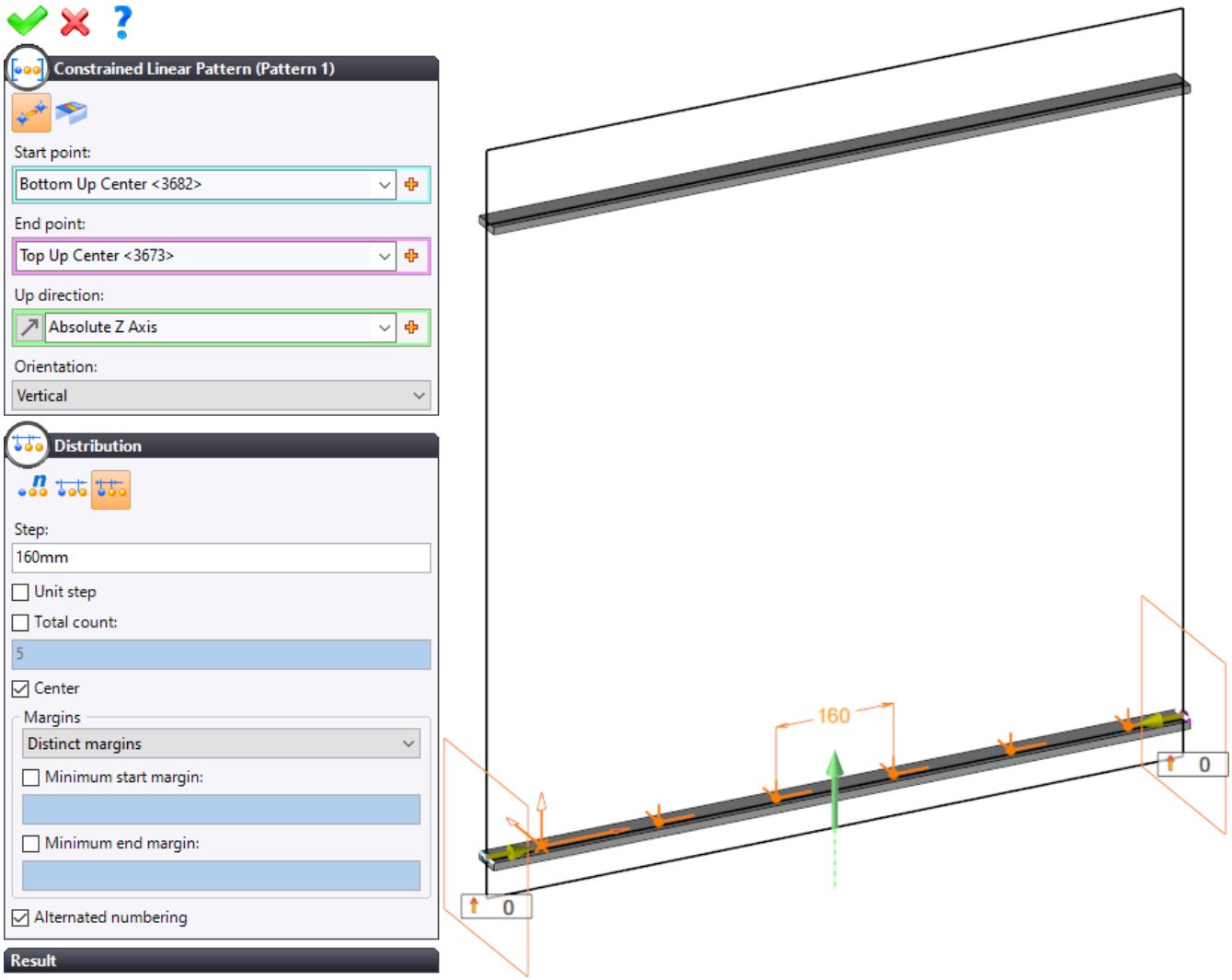
- From the Project tree, create a new folder named *Fillings* in the *4- Railing with sections > Railing files* folder.
- Right-click on the *Fillings* folder and create an **Assembly** document.
- Rename the assembly document *Alternating square bar filling*.
- Draw the following sketch on the XZ plane.



- Select the **Modeling** > **Extruded Bar** command.
- Select the **Full Flat Section, NF A 45-005** family and the **40 x 10** code, then select the following segments.

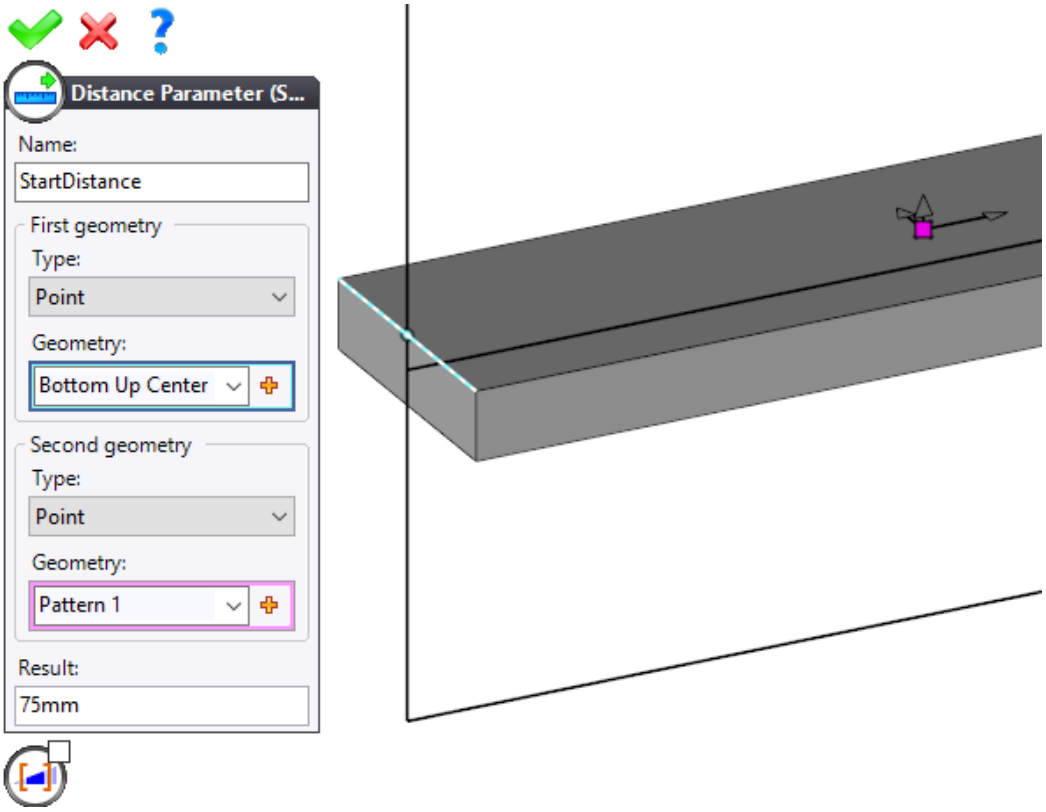


- Create the following **constrained linear pattern** between points.
- Select the **Step** distribution mode, enter a **step** of *160mm* and check the **Center** and **Alternated numbering** boxes.



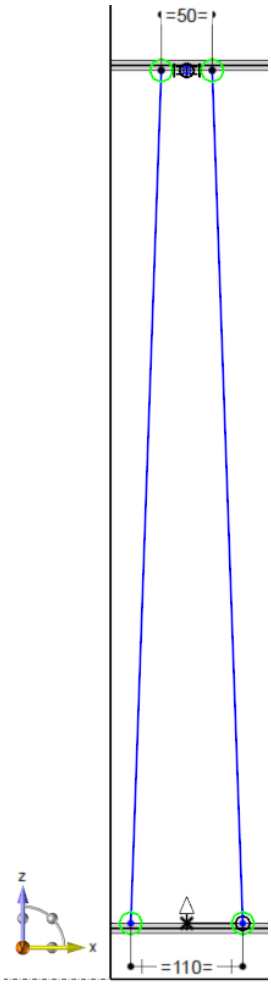
Since the pattern is composed of two extruded bars, it will be necessary, in some cases, to remove the extruded bars located on the outside.
To identify these particular cases, you have to measure the pattern.

- Select the **Construction** > **Parameters** > **Distance Parameter** command.
- Select the **Point** mode and select the two points as shown below.



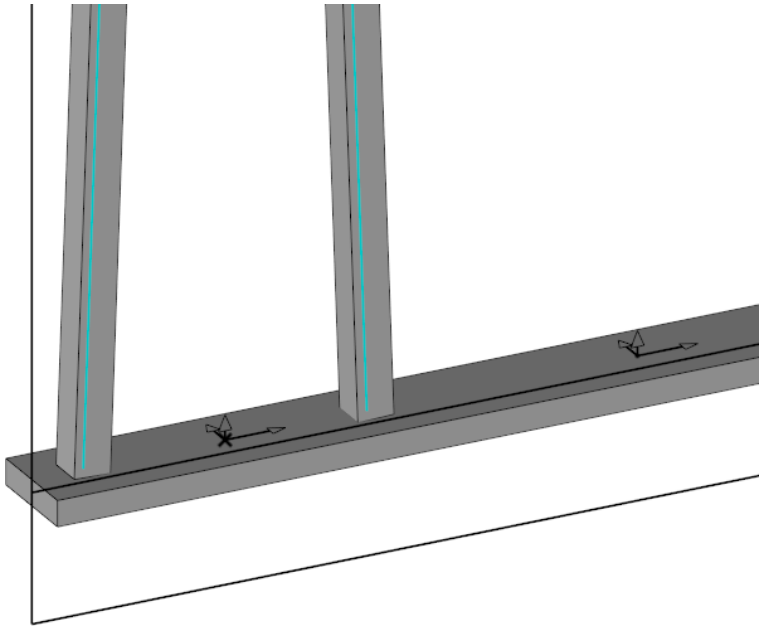
The 75mm value allows you to define rules. For example, if the value is less than 61.5mm, the left extruded bar will be deleted.

- Draw the following sketch on the XZ plane.



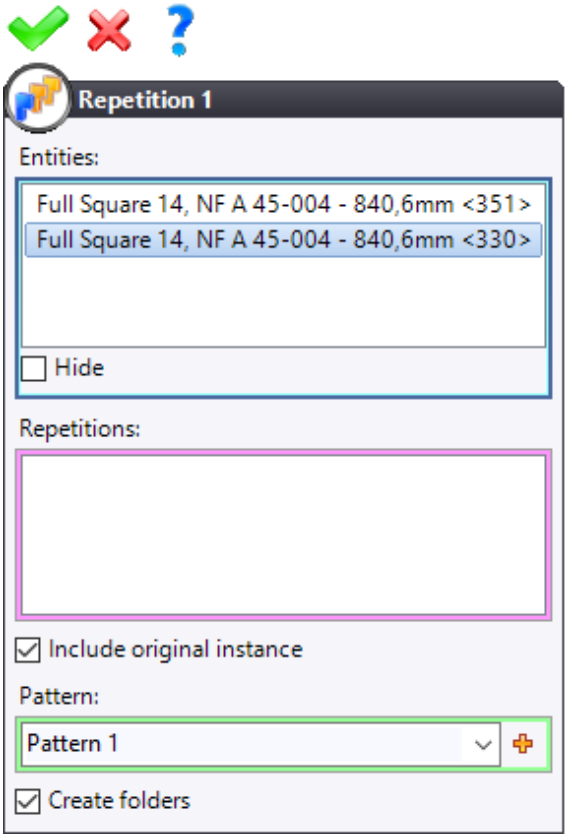
The 50mm and 110mm dimensions are centered on the first occurrence of the constrained linear pattern.

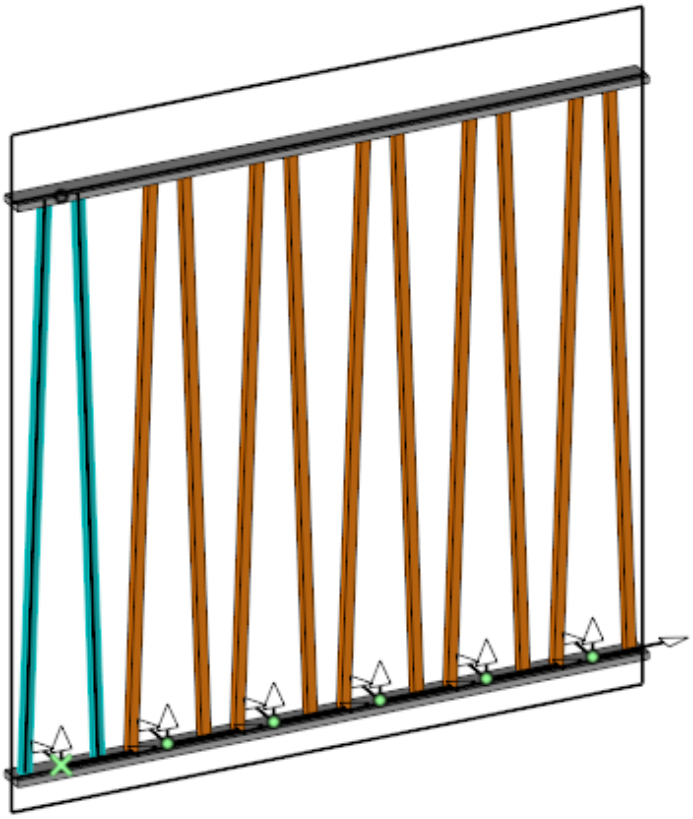
- Select the **Modeling** > **Extruded Bar** command.
- Select the **Full Square Section, NF A 45-004** family and the **14** code, then select the following segments.



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

- Select the **Construction** > **Repetition** command.
- Select the two full square extruded bars as the **entities**.
- Select **Pattern 1** as the **pattern** to be used.





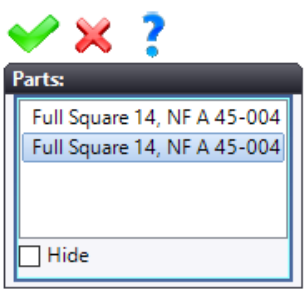
- Click on to **confirm** the operation.

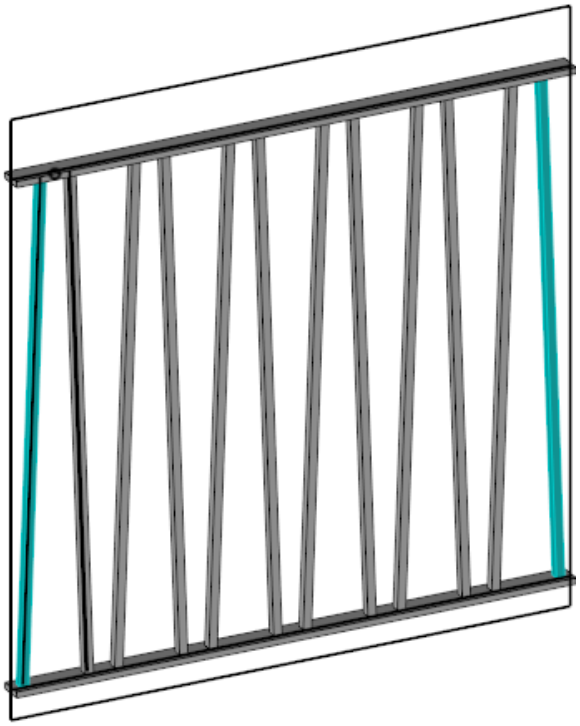
We will now have to deal with the particular case where the starting distance is less than 61.5mm. This is possible thanks to the distance parameter that was previously created.

Note: You are probably wondering, why 61.5mm? How to know and anticipate this particular case?

In the vast majority of cases, the answer is simple: you only have to use the component. Indeed, certain particular cases are obvious and can easily be anticipated. The other cases are discovered as the component is used.
As a result, your component has to evolve, be corrected and optimized. This is an integral part of the parameterization work that could even be called "development".

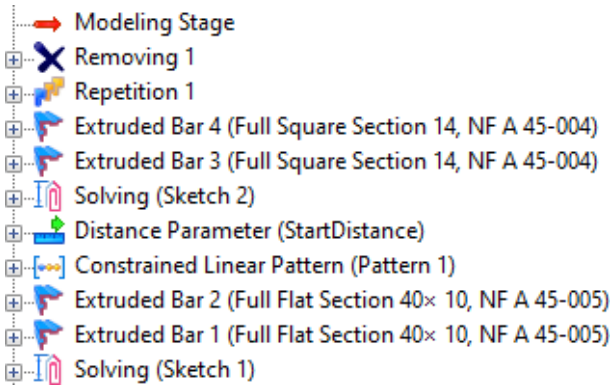
- Select the **Assembly** > **Removing** command.
- Select the following two extruded bars.





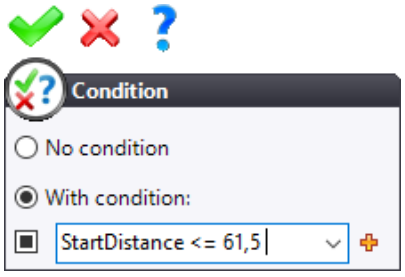
- Click on to **confirm** the operation.

This command, also called "associative deletion", creates a **removing** operation.



In other words, this deletion is chronological. If the **Modeling stage** cursor is placed before the **Removing 1** operation, the extruded bars will be displayed. If the removing operation is deleted, they will be displayed again as well.

- Right-click on the **Removing 1** operation and select the **Others** > **Condition** command.
- Select the **With condition** option and enter the *StartDistance<=61.5* formula.



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

A suffix is added before the operation:

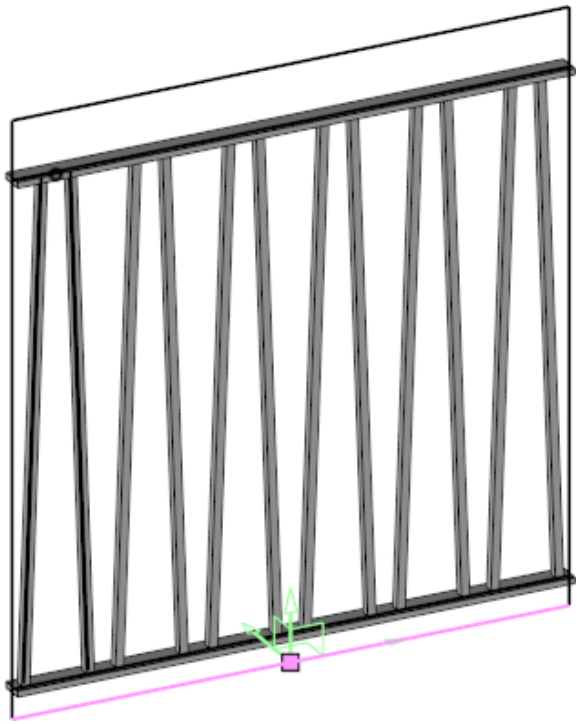
- (?): The operation has a conditioning. The operation is active because its condition is also active.
- (x): The operation has a conditioning. The operation is inactive because its condition is not respected.

In this case, the operation is disabled because the **StartDistance** parameter value is greater than 61.5mm.

The basic parameterization is now complete. You can vary the **Length** parameter for testing purposes.

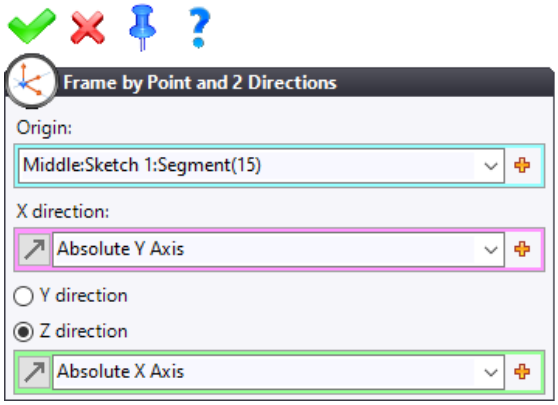
As with the post, you now have to make the filling distributable.

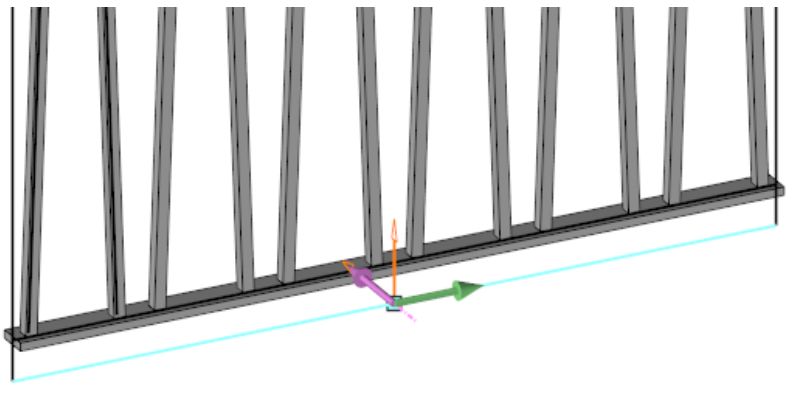
- Select the **Tools** > **Functions** > **Provide Function** command.
- Select the **Distributable Component** function, uncheck the **Create parameter** box, then click on to **confirm** the operation.
- For the **Top Right Point** and **Bottom Left Point** fields, select the single point indicated below.



Selecting the same point for these two fields can be useful when the object has only one positioning method, which is the case here.

- In the **Frame** field, click on the icon and select the **Frame by Point and 2 Directions** command.
- Fill in the fields as shown below.





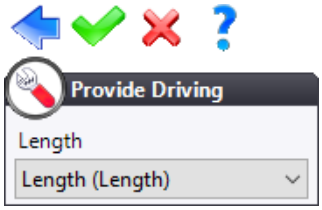
- Click on to **confirm** the operation.

Reminder:

- The Z axis of the frame must be in the distribution direction.
- The Y axis describes the vertical orientation.
- The X axis describes the horizontal orientation.

- In the **Length** field, select the **Length** parameter.
- Click on to **confirm** the operation.

- From the *Alternating square bar filling* assembly document, create a **Family** document.
- From the Entities tree's **Generics** folder, right-click on the **Length** parameter and select **Move to Drivers**.
- Select the **Tools** > **Provide Driving** command.
- Select **Distributable Family** and click on the icon to move to the next step.
- Select the **Length** parameter.

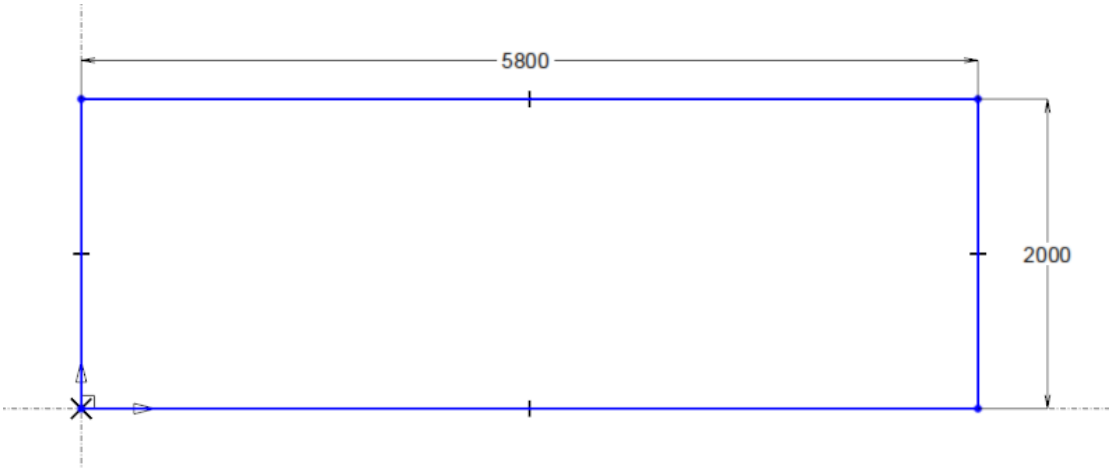


- Click on to **confirm** the operation.
- **Save** (**Ctrl + S**) and close the family and its generic.

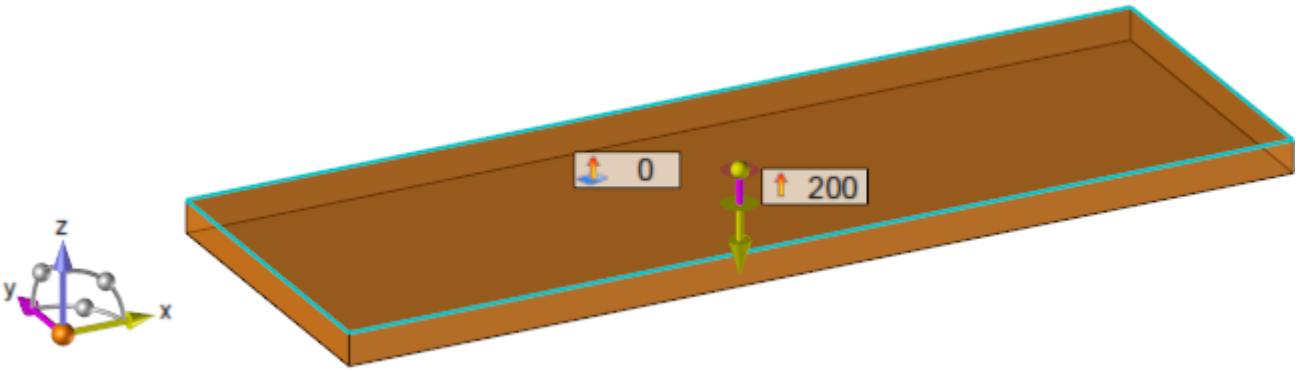
Using the distribution command

We will now test our components and discover the possibilities of the distribution command.

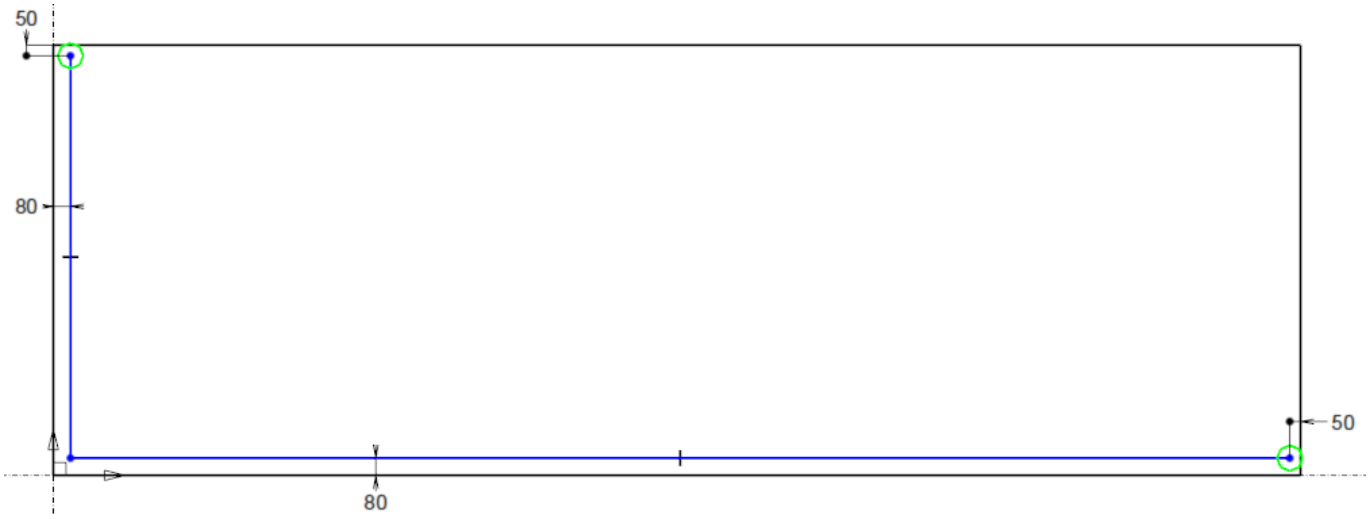
- From the Project tree, right-click on the *4- Railing with sections* folder and create an **Assembly** document.
- Rename the assembly document *Railing with sections*.
- Draw the following sketch on the XY plane.



- Select the **Modeling** > **In Place Extruded Parts** command.
- In the **Sections** field, select the previously created sketch.
- Enter a **length** of *200mm*.



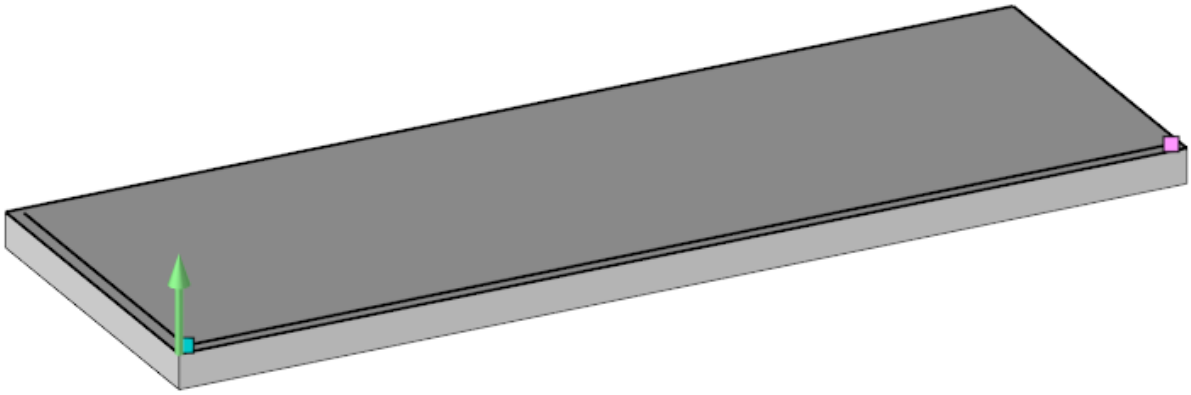
- Still on the XY plane, create the following sketch.



- Select the **Modeling** > **Distribution** command.

The **Distribution** command has the same positioning system as the **Extruded Bar** command: on profile, between points and frame + length.

- Select the **Two points** mode and then select the points as shown below.



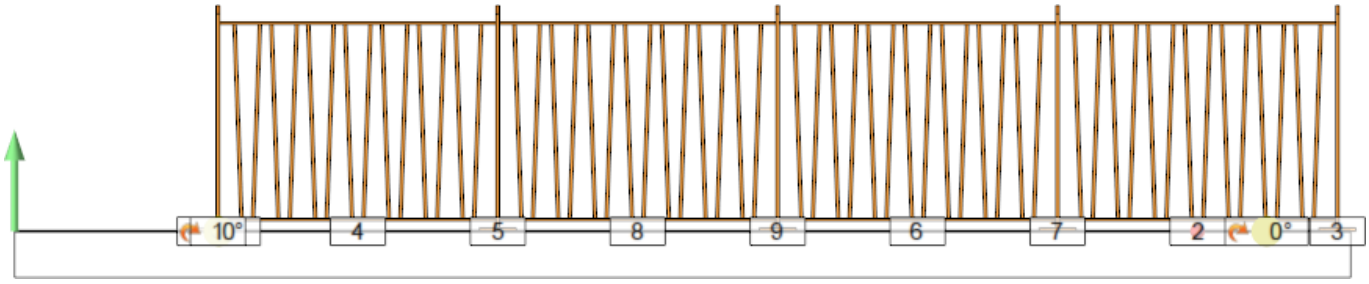
The following two distribution modes are available:

- **Mark out**: When you indicate an element length, **TopSolid** distributes the element according to some strategies described below.
- **Distribute**: You indicate a quantity of elements to be distributed.

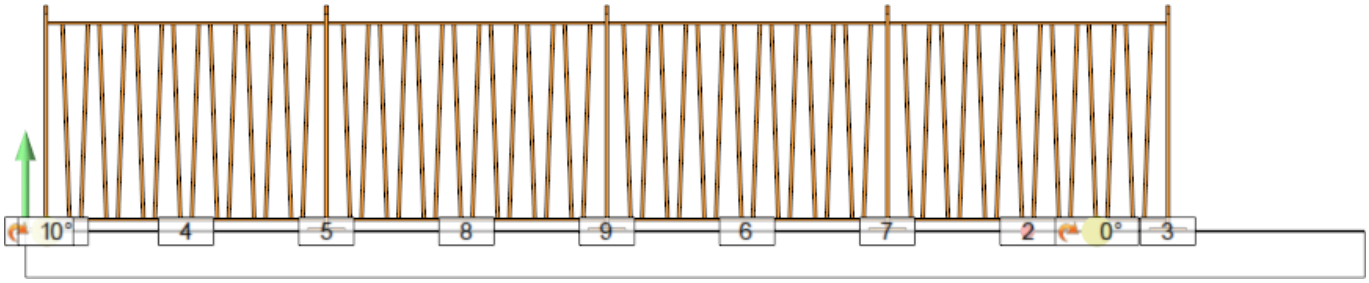
- Select the **Mark out** mode.

Several **remainder distribution** modes are available:

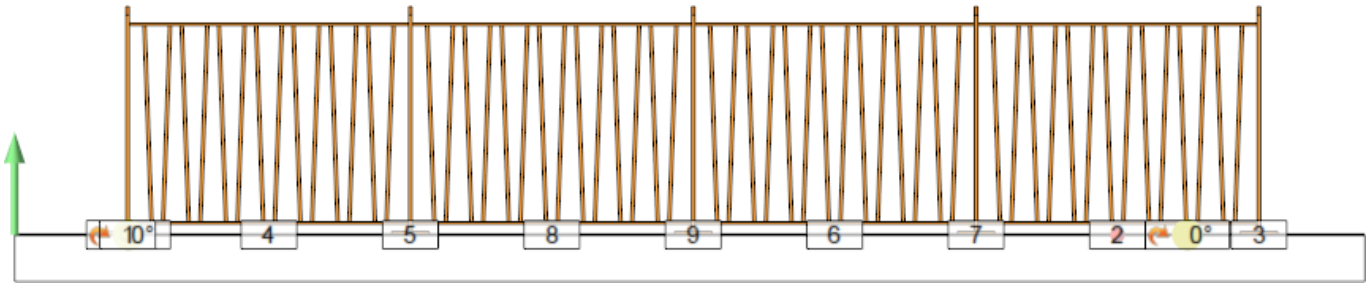
- **Do not distribute**: Fillings of 1200mm will be positioned and the remainder will remain empty. You can determine its position using the **Remaining length** option.
  - **Start**



  - **End**

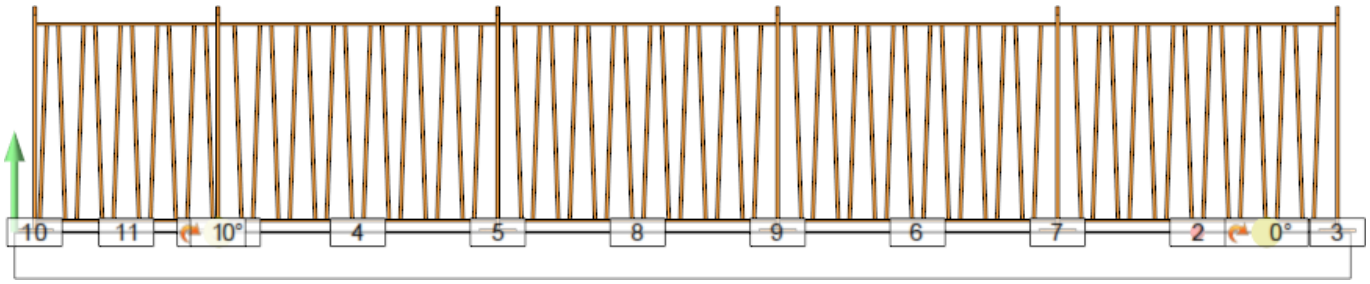


- **Symmetric**

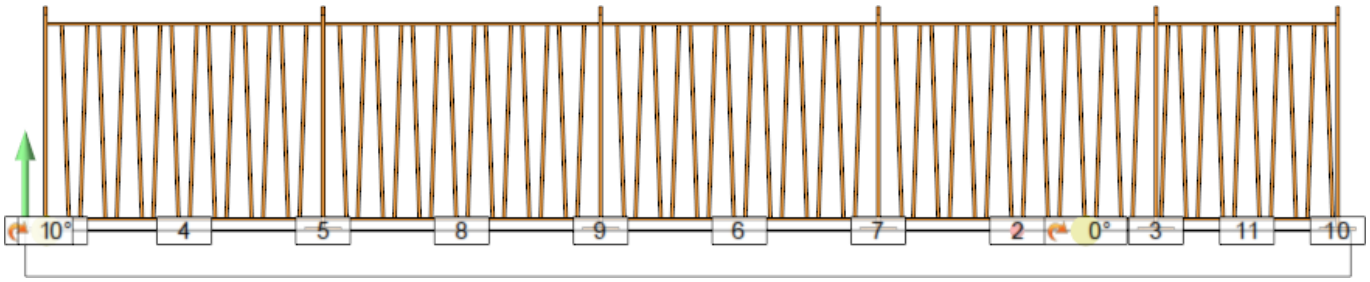


- **Fill remainder**: Fillings of 1200mm will be positioned and the remainder will be filled with a shorter instance. You can indicate the minimum filling distance as well as the position of the remaining length.

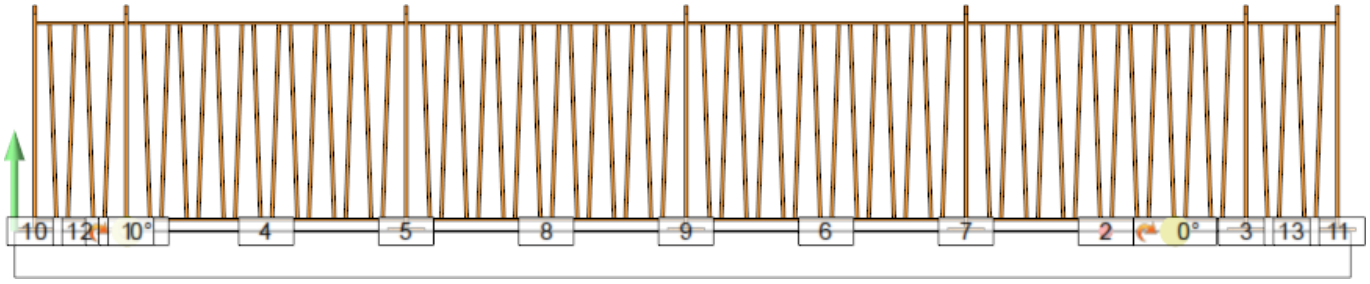
- **Start**



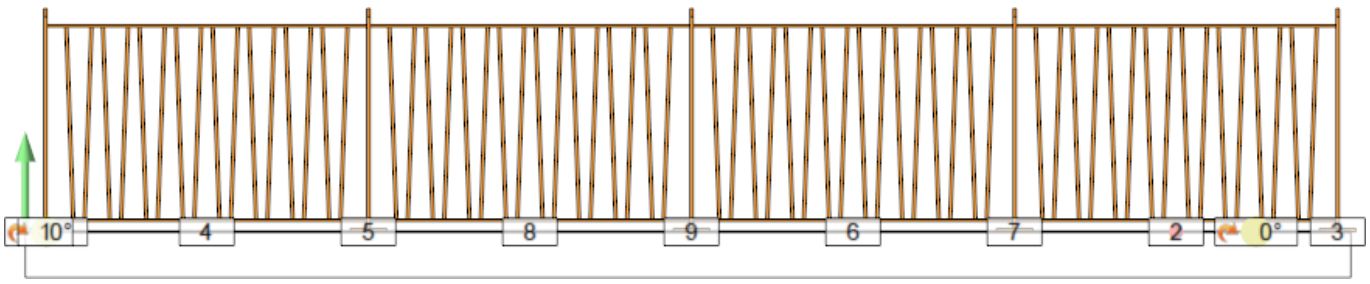
- **End**



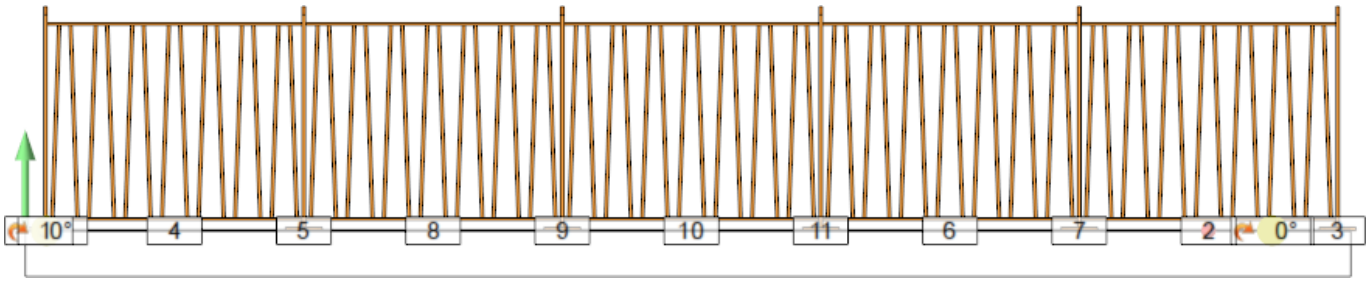
- **Symmetric**



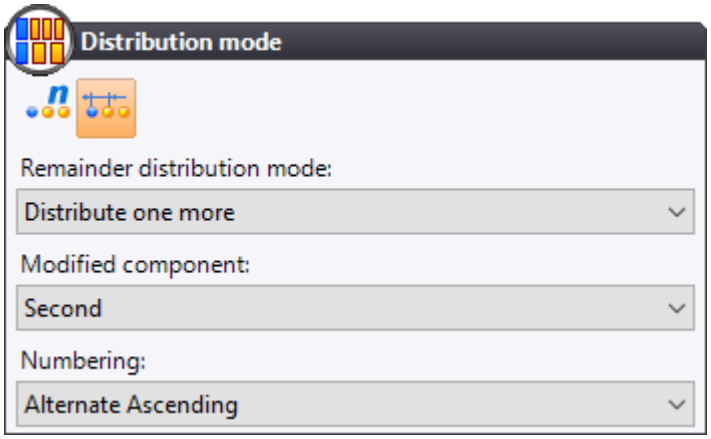
- **Distribute remainder**: Fillings with a length greater than or equal to 1200mm will be positioned.



- **Distribute one more**: Fillings with a length less than or equal to 1200mm will be positioned.



- Select the options below.

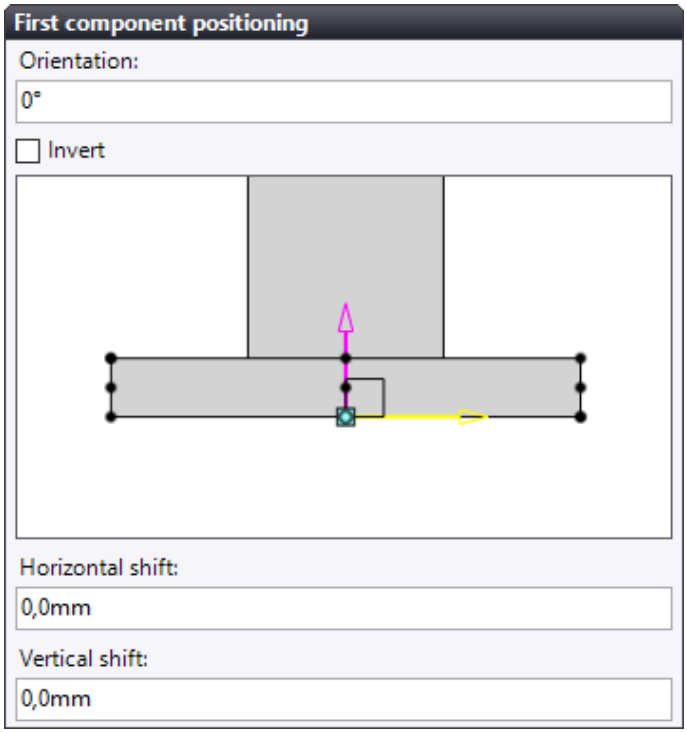


The distribution is able to manage the distribution of two components simultaneously. In our example, the first component will be the post, the second will be the filling.

- In the **First component** section, select **Post** as the **family**.

TopSolid preselects the center point. In our case, it is the center of the post.

- Zoom in to select the point as shown below.

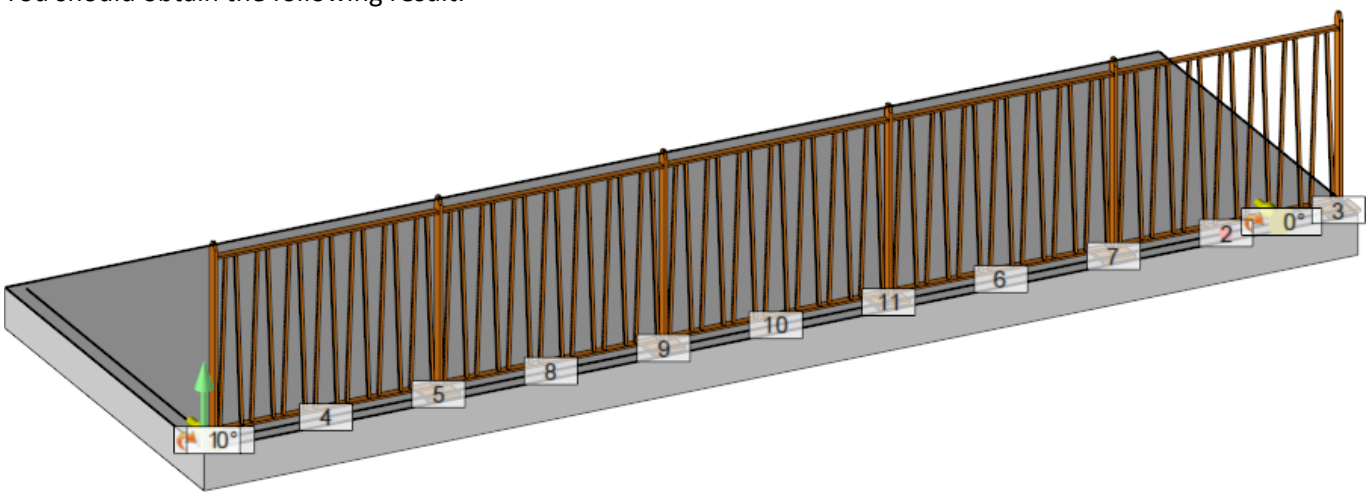


In the **Second component** section, the following three modes are available:

- **None**: Only one component is distributed.
- **Family**: A family is distributed.
- **Space**: Empty spacings are managed.

- Select the **Family** mode.
- Enter a **length** of *1200mm* (filling length).
- Select the **Alternating square bar filling** family.

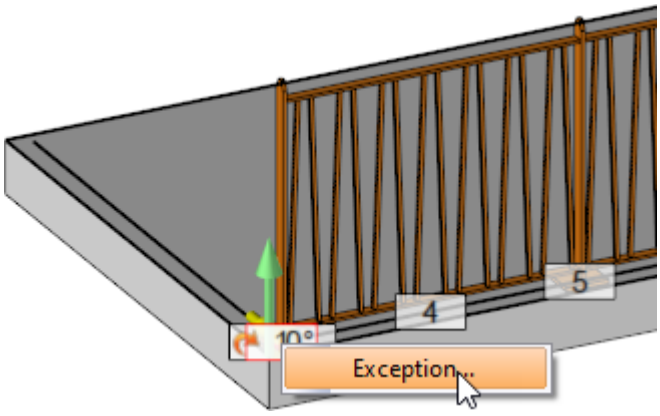
You should obtain the following result.



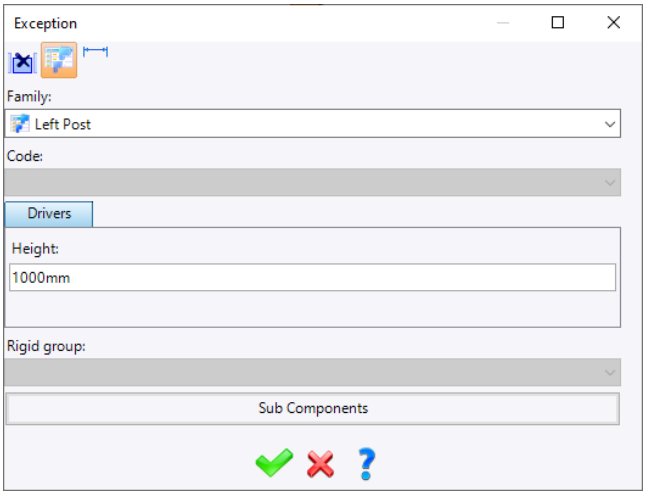
Another characteristic of the distribution command is its ability to handle particular cases. You can therefore perform actions on each of the elements to be distributed such as:

- replacing an element with a space;
- replacing an element with another family;
- forcing the length of an element;
- modifying an element's parameters.

- Right-click on label **1** and select **Exception**.

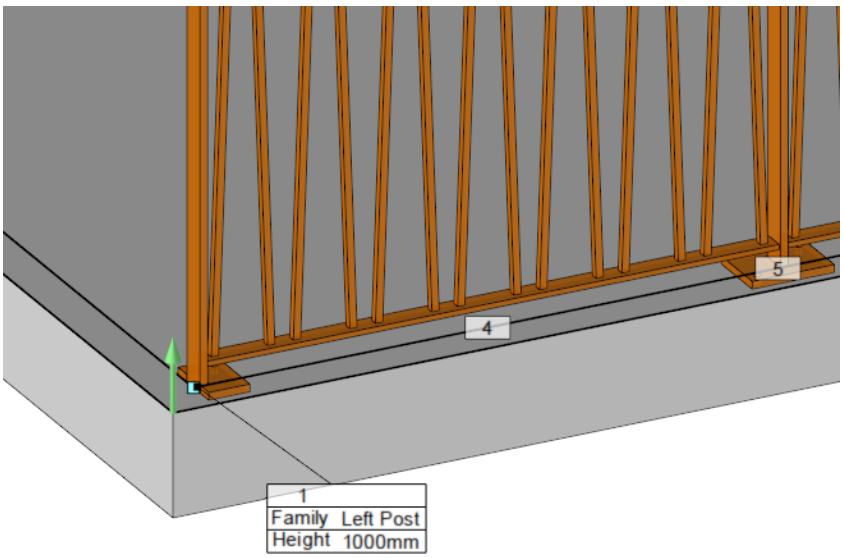


- Select the **Family** mode and select the **Left post** family.

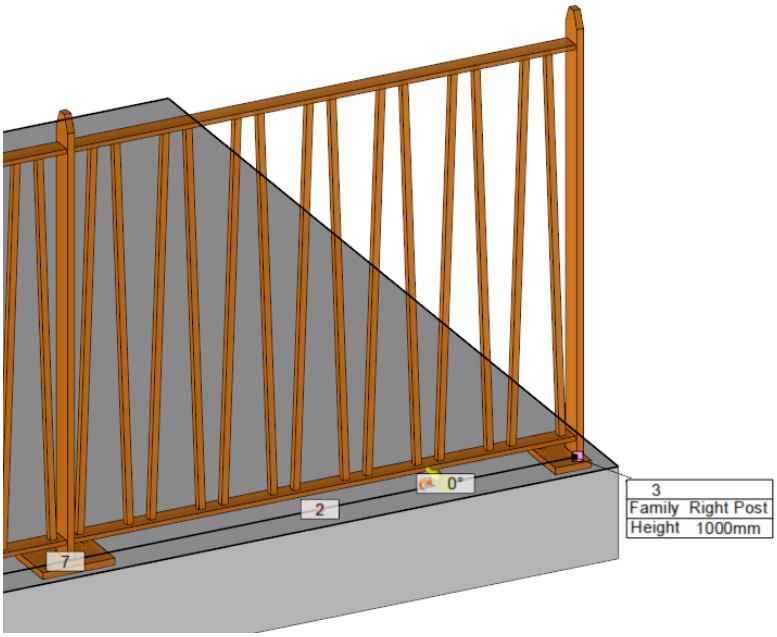


- Click on to **confirm** the operation.

TopSolid replaces the post and adds a label that represents the exception.

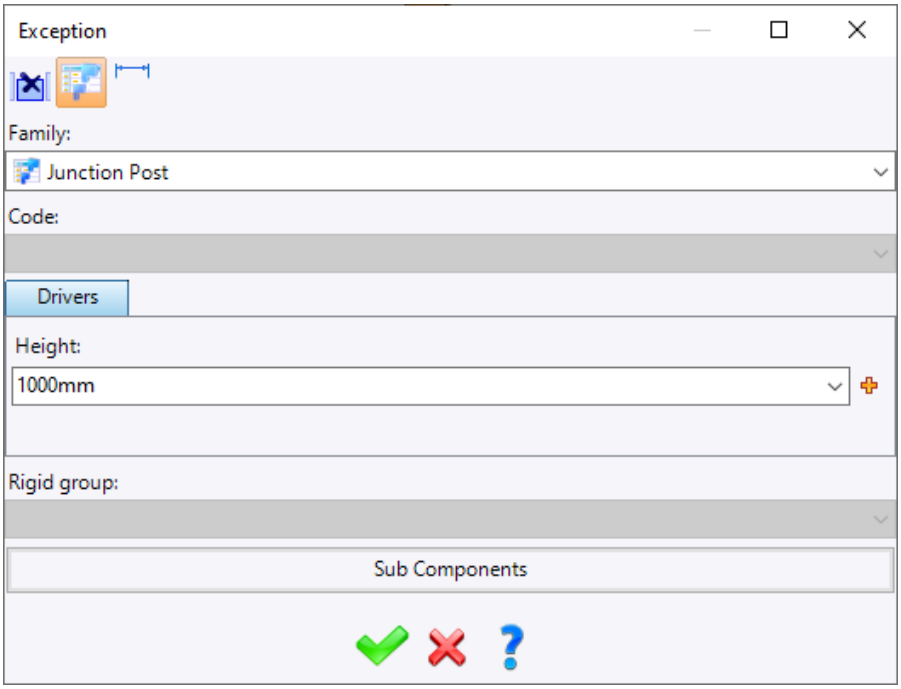


- Add an **exception** to label 3 by selecting the **Right post** family.

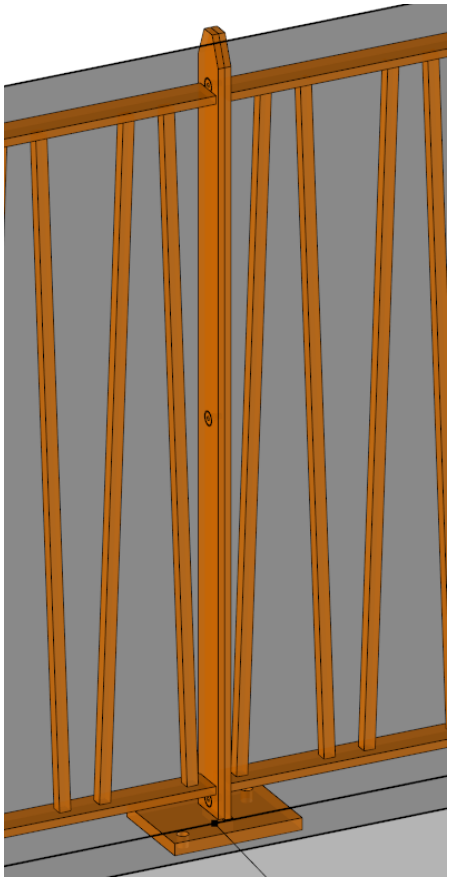


One solution to manage the railing sections is to use junction posts.

- Add an **exception** to label 11 by selecting the **Junction post** family.



TopSolid recalculates the entire distribution to obtain homogeneous fillings in order to take into account this difference in thickness.



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

- Modify the concrete length to *6500mm*. The distribution is recalculated.

To obtain correct railing sections, you have to replace a post with a junction post.

- Edit the distribution operation.
- Add an **exception** to label 9 by selecting the **Junction post** family.
- Enter a **length** of *16mm*.
- Leave the distribution command active.

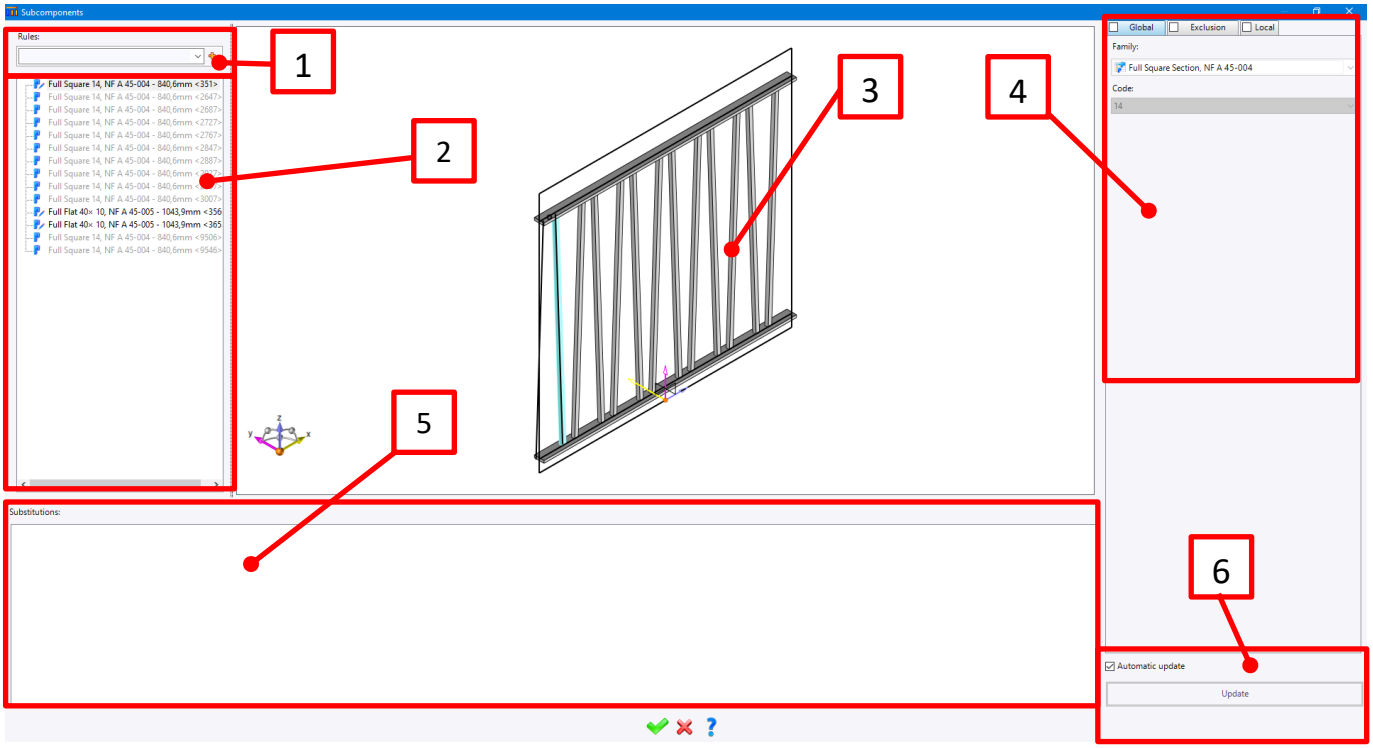
Using sub-components

To go even further in terms of flexibility and parameterization, you can modify certain sub-components without parameterization. This may involve modifying a driver parameter, a code or even the family directly.

Let us take the case of our filling's bars. We want to switch to a 12mm section instead of 14mm.

- In the  **Second component** section, click on the **Sub-components** button.

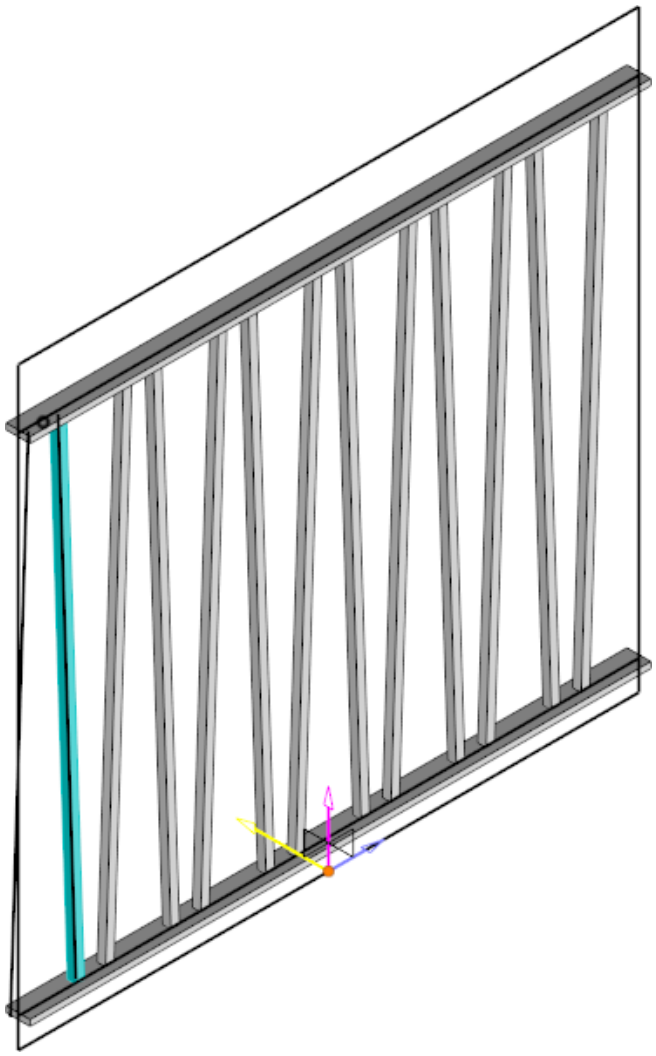
The following dialog box appears.



1 - **Rules**: You can define substitution rules using the functions and filters (see online help).
2 - **List of sub-components**: The components displayed in gray cannot be modified directly. The other components can be selected and edited from the editing area (4).
3 - **Preview**: It allows you to view the selected components, as well as the changes made.
4 - **Editing area**: This area allows you to make changes to the components. The following three modes are available:
Global: All parts with the same function will be modified.
Exclusion: Allows you to define exclusions as part of a global modification.
Local: Only the selected element will be modified.
5 - **Substitutions**: This area will contain the list of changes made.
6 - **Update**: Two modes are available: **Automatic** or **Manual**.

For more information, you can consult the **TopSolid** online help by clicking on the  icon or by pressing the **F1** key on your keyboard.

- Select the extruded bar as shown below. Check the **Local** tab and select the **10** code.



- Check the **Automatic Update** box. All the right bars are updated.

Note: To modify the left bar, you have to resize the railing so that it appears or you have to make a global modification.

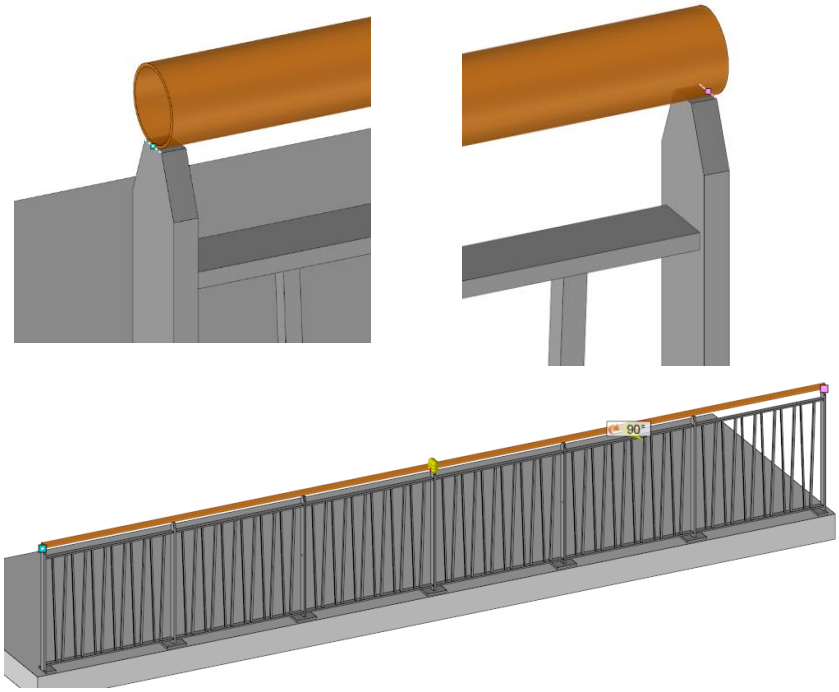
- **Confirm** to finish editing the sub-components.

All fillings are modified. If we had wanted to modify only one of them, it would have been necessary to create an exception and then use the **Sub-components** option.

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

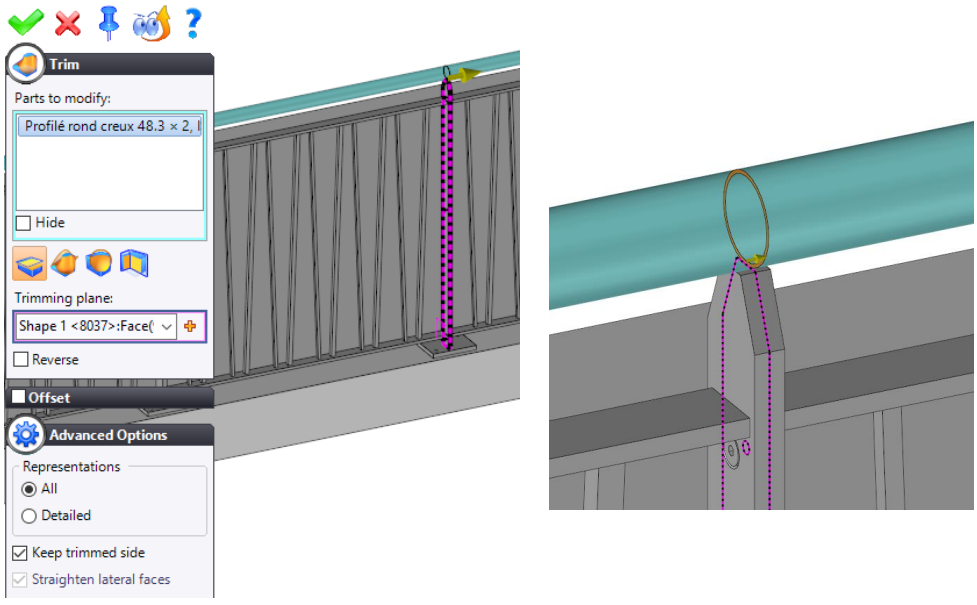
Adding the handrail

- Select the **Modeling** > **Extruded Bar** command.
- Select the **Hollow Circular Section, ISO 4019** family and the **48.3 x 2** code.
- Select the **Between points** mode and position the extruded bar as shown below.



It is now necessary to trim the handrail at each junction post. Another possibility would have been to include three extruded bars, one for each section.
This second method, although slower, is more interesting if you plan to modify the ends of the handrail by performing miter trims or other means. Indeed, a trimming operation by keeping the trimmed side is faster, but makes it impossible to use the cutting commands.

- Select the **Modeling** > **Trim** command.
- Select the **Plane** mode, check the **Keep trimmed side** box, and then select the middle face of the first junction post.



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

TopSolid asks you to select a derivation template.

- Select the **Part - 0mm (Extruded bar, Derivation, Mirror, Partial part)** template.
- Click on  again to **confirm** the operation.

Note: The selected template is specific to extruded bars. It integrates many parameterizations for only extruded bars. For example, it can parameterize the name of the extruded bar by integrating its length (see the Project tree), calculate the cutting angles and lengths, produce the symbol for the orientation of the cuts, etc.
Make sure you select the right template because, once you have used it, you can no longer change it. In our case, if an error occurs, the only solution is to remove the trimming operation and perform it again using the right template.

Important: In case of derivation for modification or trimming while keeping the trimmed side, it is advisable to use the **Part - 0mm (Extruded bar[...])** template.

- Repeat the trimming operation on the second junction post. There should now be three handrails.

Defining the sub-assemblies (in-place assemblies)

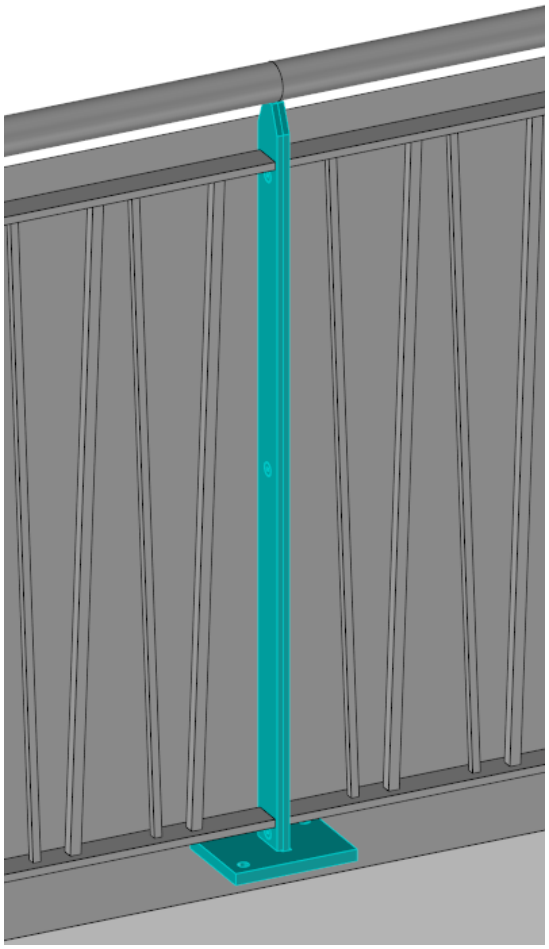
- Right-click on the *Railing with sections* assembly document's tab and select the **Bill of Material** command.
- Select the **Multi-level** template from the **Steel Standard Templates - United States** folder and click on to **confirm** the operation.
- In the **Assembly** dialog box, make sure that the **Groups by properties** box is checked and click on to **confirm** the operation.
- Rename the bill of materials *Railing with sections - General*.

As you can see, the bill of materials contains all the elements without any real organization. Indeed, the **Distribution** command is transparent; it does not produce a sub-assembly. This allows the grouping of parts/sub-assemblies as desired.

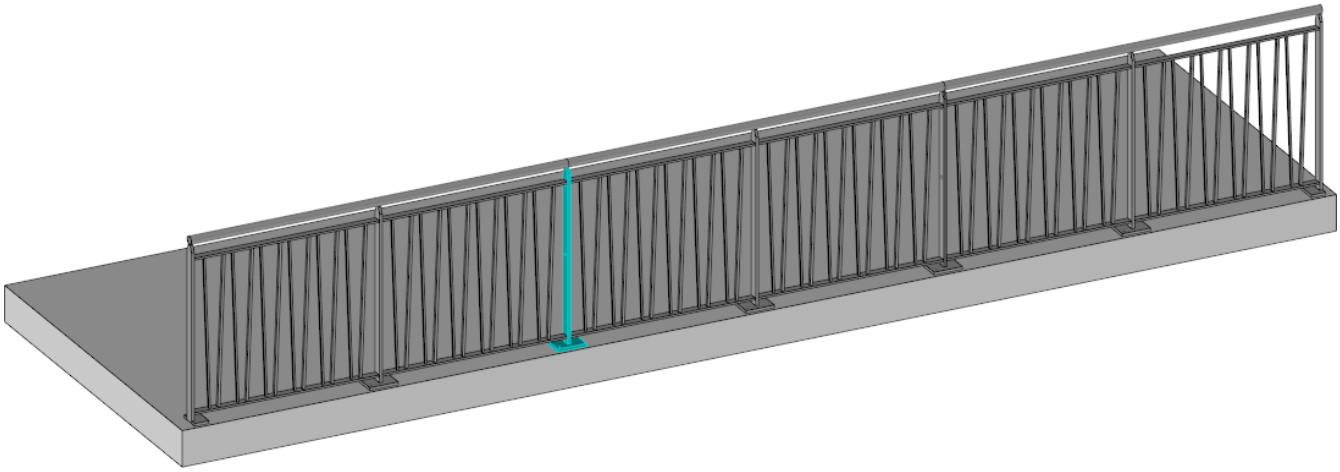
We will exclude the concrete part from the bill of materials.

- Right-click on the **Part** line and select the **Open Document** command.
- Select the **Construction** > **Parameters** > **Type for BOM** command.
- Select the **Absent** type.
- Click on to **confirm** the operation.
- **Save** (**Ctrl** + **S**) and **close** the document.
- Return to the *Railing with sections* assembly document.

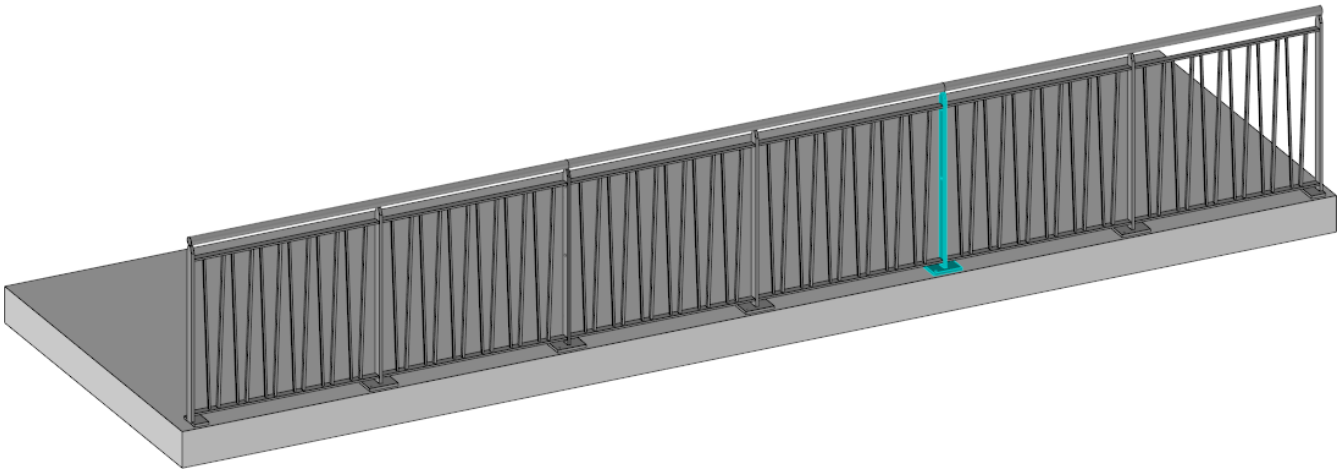
The junction posts were brought in one go to save time and facilitate the calculation of the distribution. You have to “split” them up since one portion will be welded on the left section and the other portion on the right section.



- Select the **Modeling** > **Disassembly** command.
- Select first junction post.

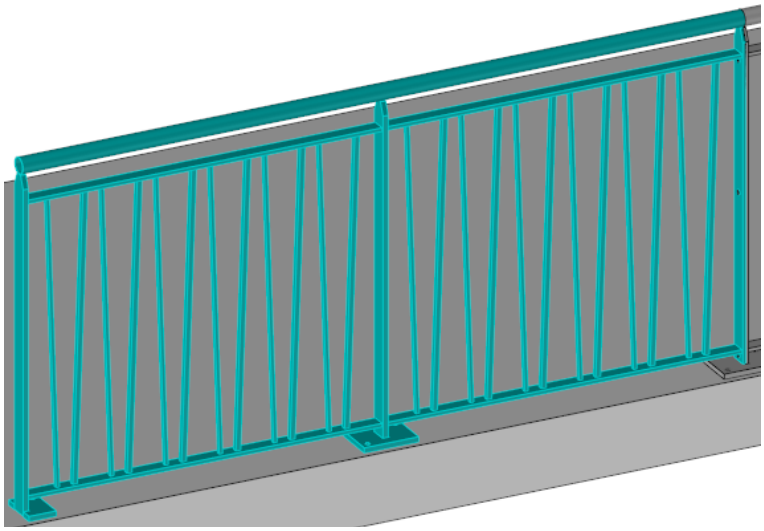


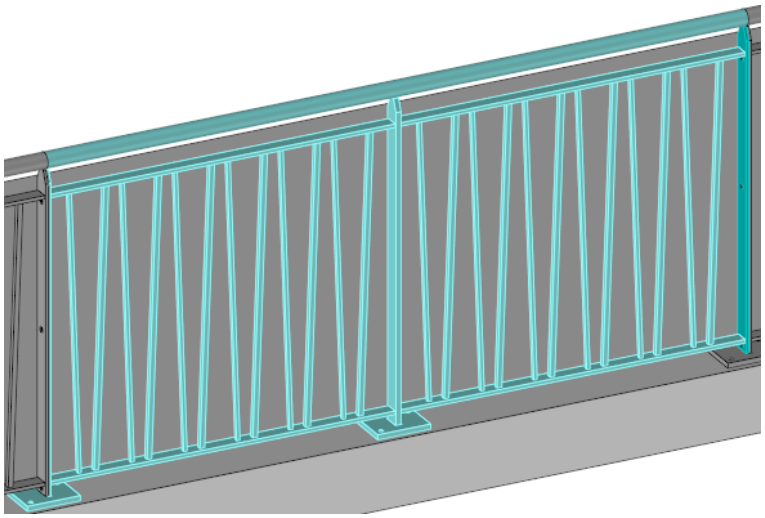
- **Pin** the command's dialog box and click on to **confirm** the operation.
- Disassemble the second junction post.

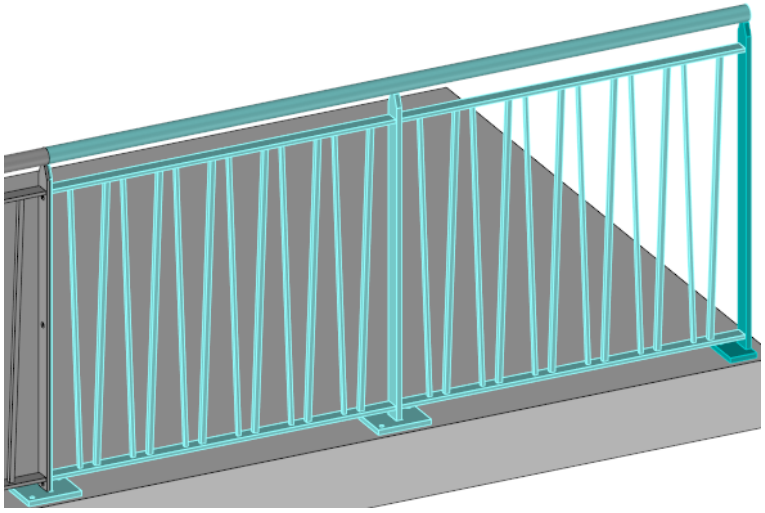


We will now create the sub-assemblies.

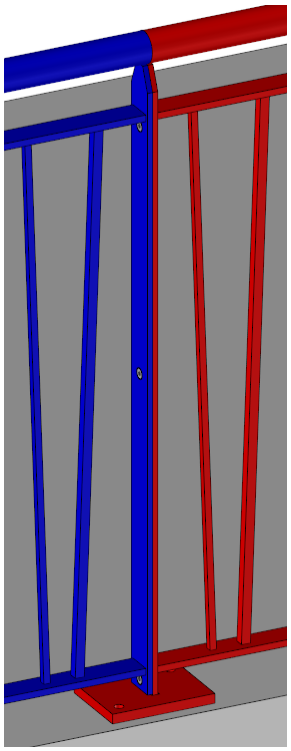
- Select the **Modeling** > **In Place Assembly** command.
- Create the following three sub-assemblies.







The parts of the junction posts are divided between the sections as shown below.



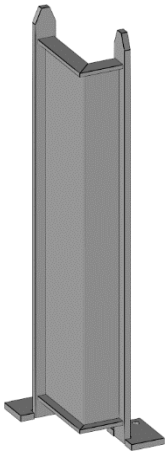
- Create another in-place assembly which contains the three sections.

Warning: If the length of the railing is modified, you will have to edit the in-place assemblies to add the new parts. You may also need to add/remove junction posts and disassemble them.

- **Save** the document (**Ctrl** + **S**).
- Return to the BOM document.
- Rename the three sub-assemblies *Start section*, *Intermediate section* and *End section*, as well as the railing assembly.

Additional exercises

Creating a side return

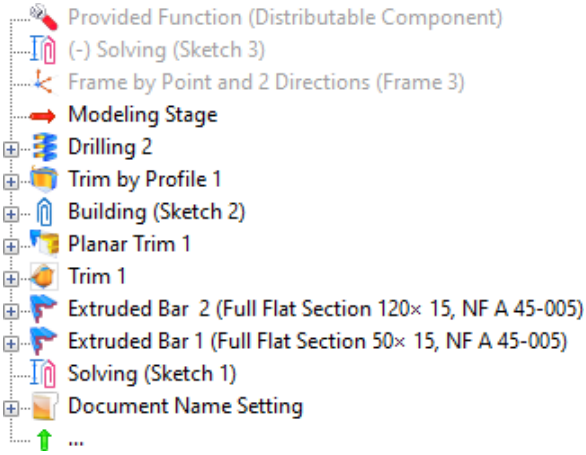


Railings very regularly have returns on their ends. These are very varied and, depending on the needs, so is the method for designing them.

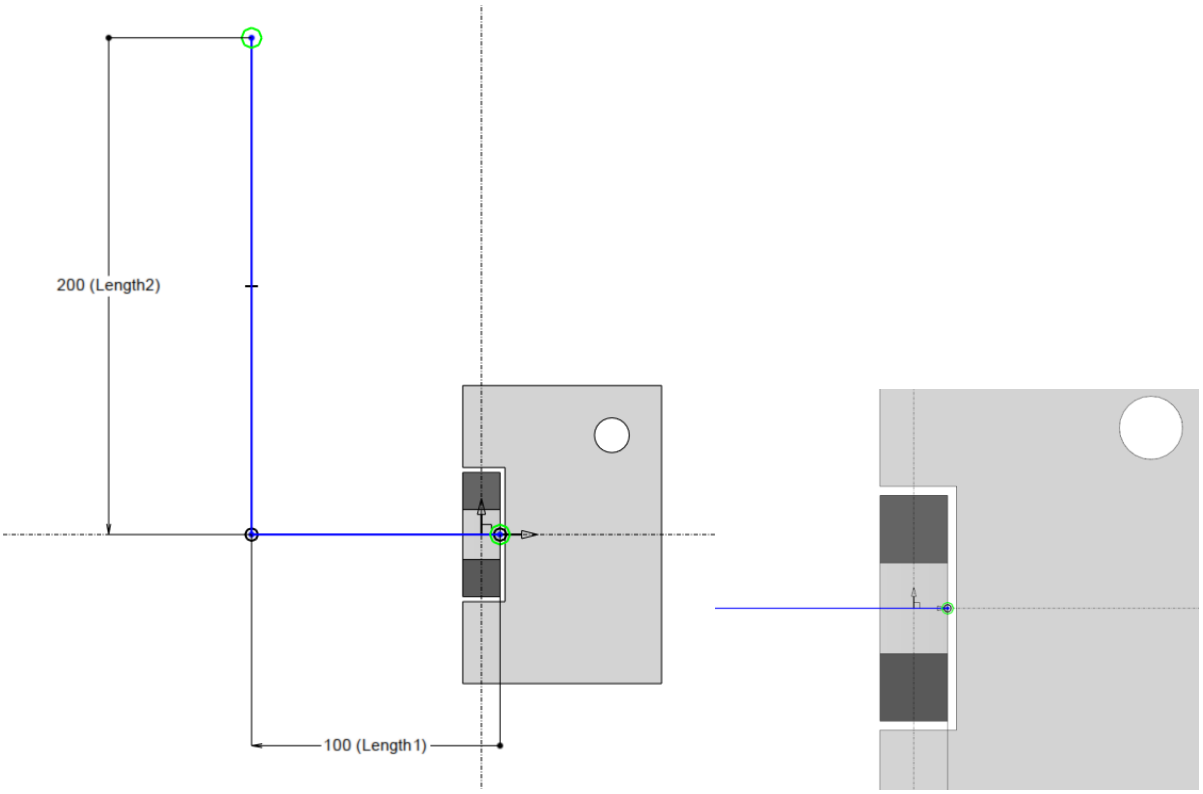
Two strategies are possible:

- integrate the angle into the distribution (exercise);
- put the angles first, then create the distributions on each side.

- From the Project tree, create a new folder named *Returns* in the *4- Railing with sections > Railing files* folder.
- From the *4- Railing with sections > Railing files > Posts* folders, copy the *Left post* family and assembly documents.
- Paste these documents into the previously created *Returns* folder, then rename them *Left return*.
- Open the *Left return* assembly document.
- From the Operations tree, move the **Modeling Stage** cursor before the **Frame by Point and 2 Directions (Frame 3)** operation.

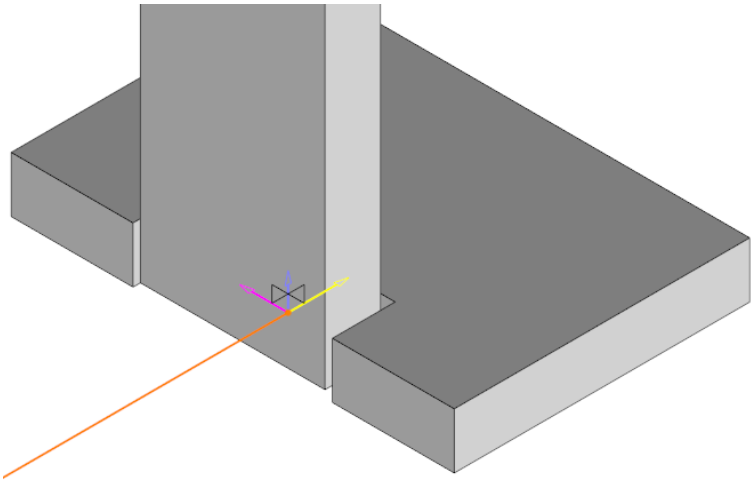


- Create the following sketch on the XY plane.

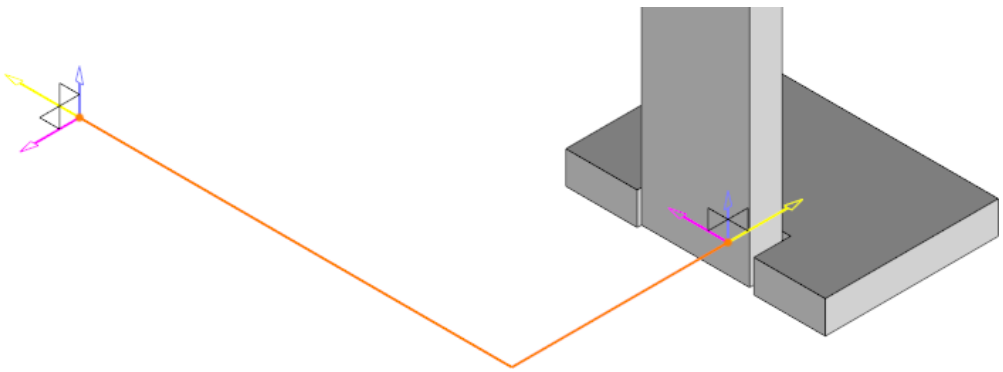


Note: Do not forget to create the two **Length1** and **Length2** parameters.

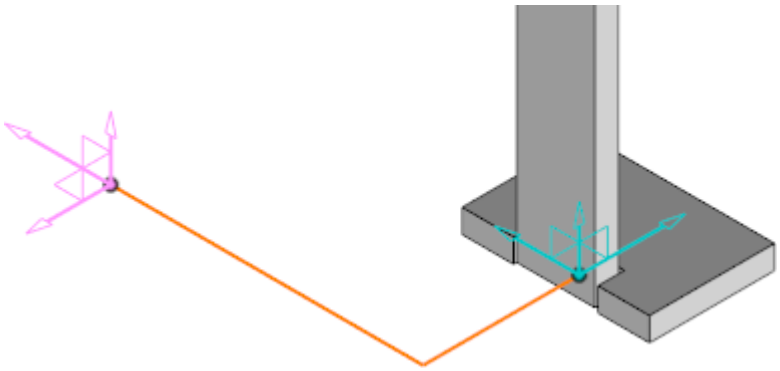
- Select the **Construction** > **Frame by Point and 2 Directions** command and create the following frame (Z on the absolute Z axis and X on the absolute X axis).



- Repeat the operation by creating the following frame (Z on the absolute Z axis, X on the absolute Y axis).

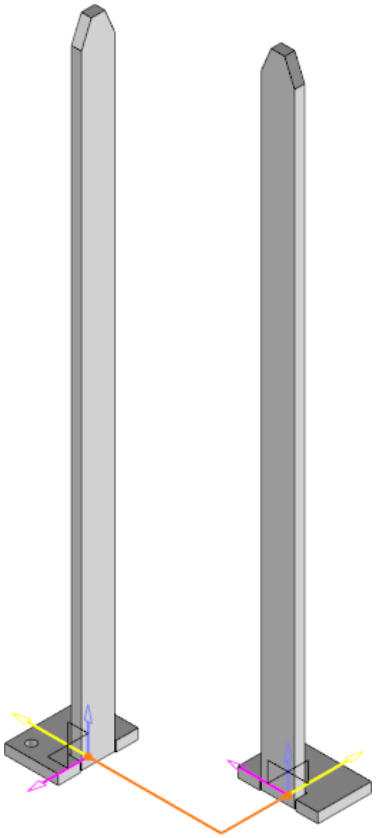


- Select the **Construction** > **Repetition** command.
- Select the post and its plate as the **entities**.
- Create a **pattern on frames**.
- Select the first frame as the **source frame** and the second frame as the **destination frame**.

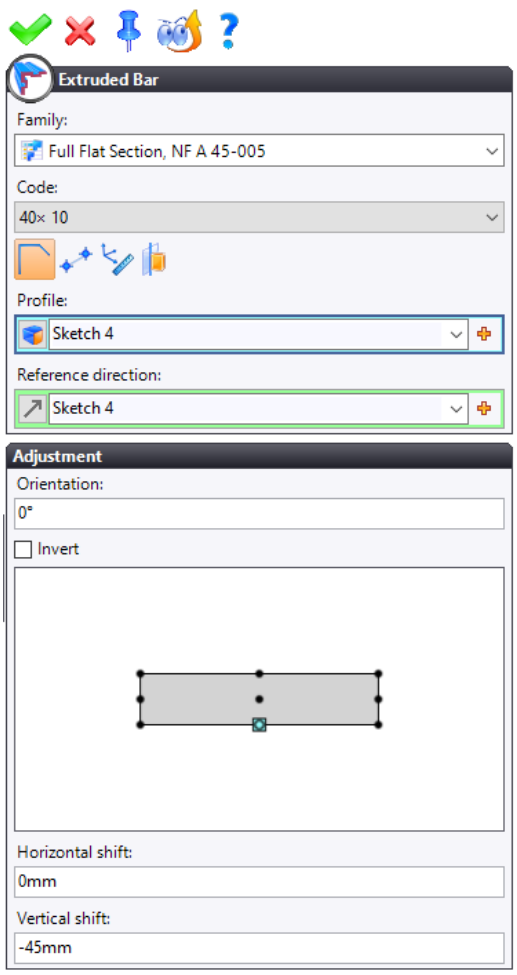


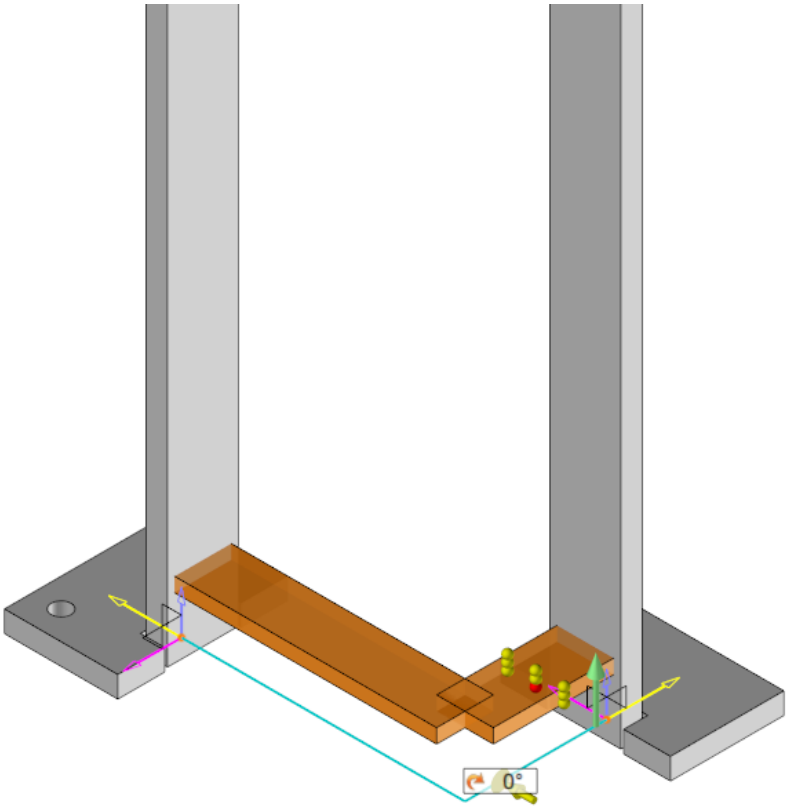
- Click on to **confirm** the pattern, and then the repetition.

You should obtain the following result.



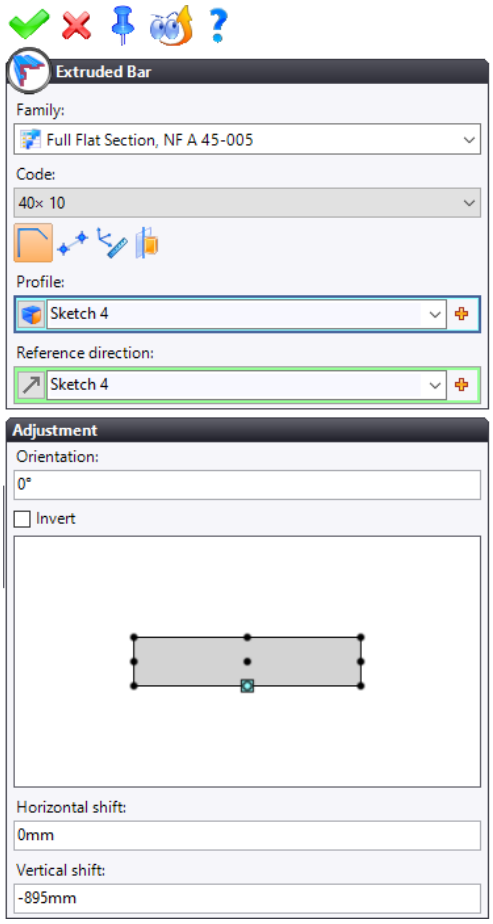
- Select the **Modeling** > **Extruded Bar** command.
- Select the **Full Flat Section, NF A 45-005** family and the **40 x 10** code, then select the sketch.

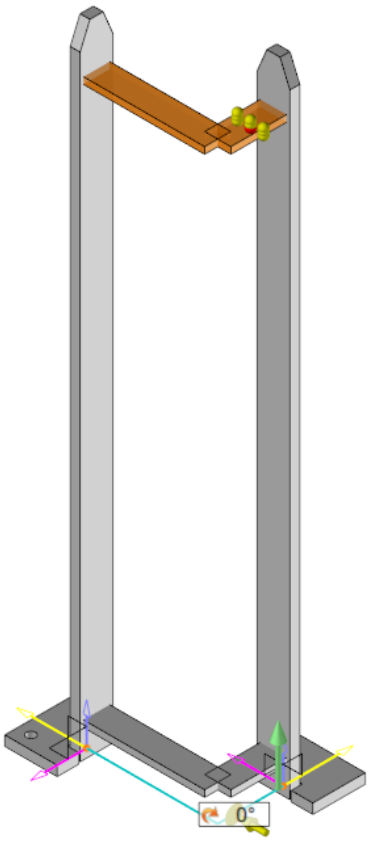




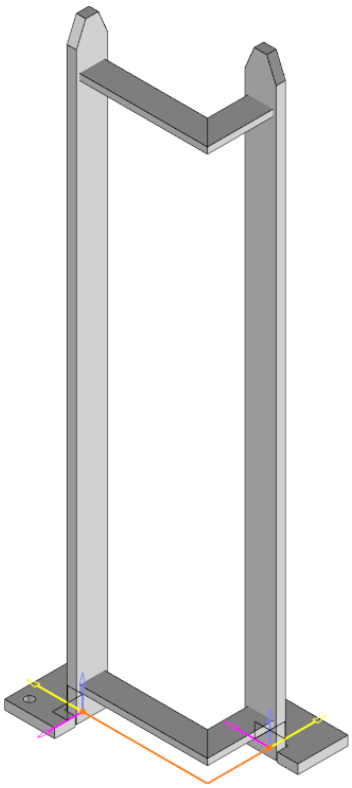
- Add a **vertical offset** of *45mm* (or *-45mm* according to the indicated direction).

- Repeat the operation for the upper extruded bars by entering an offset of *895mm*.

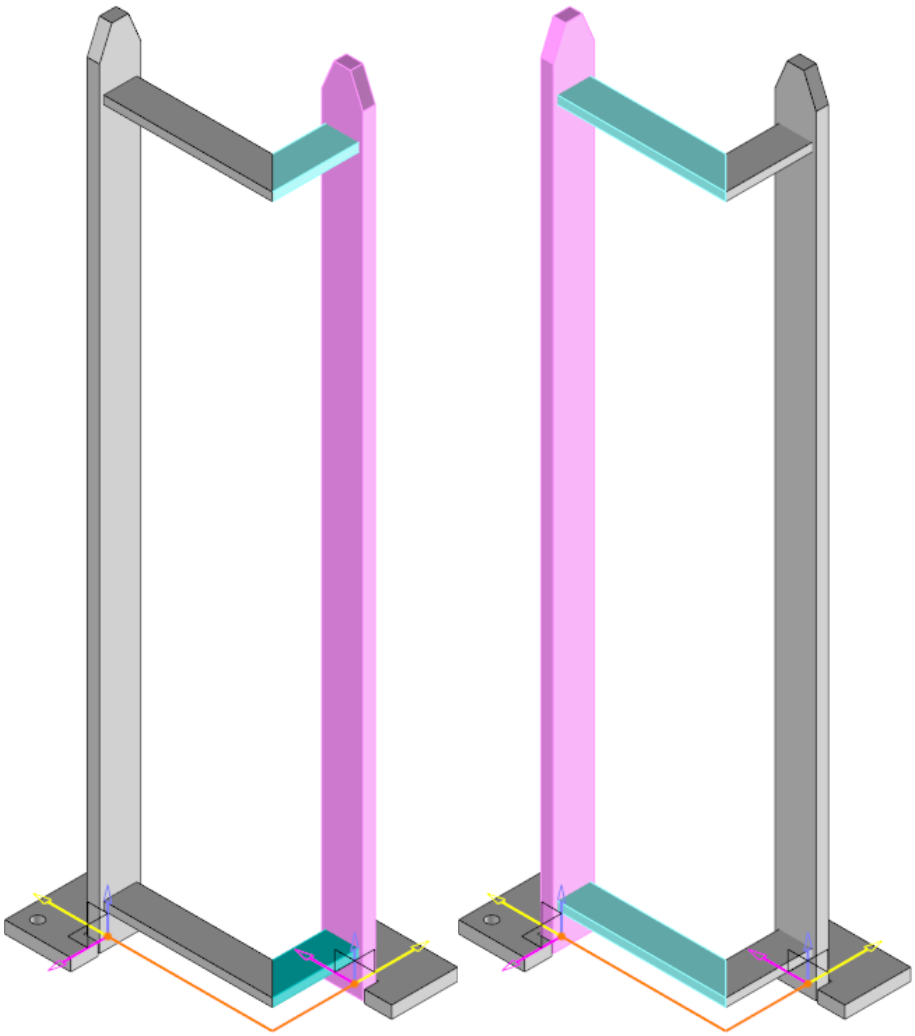




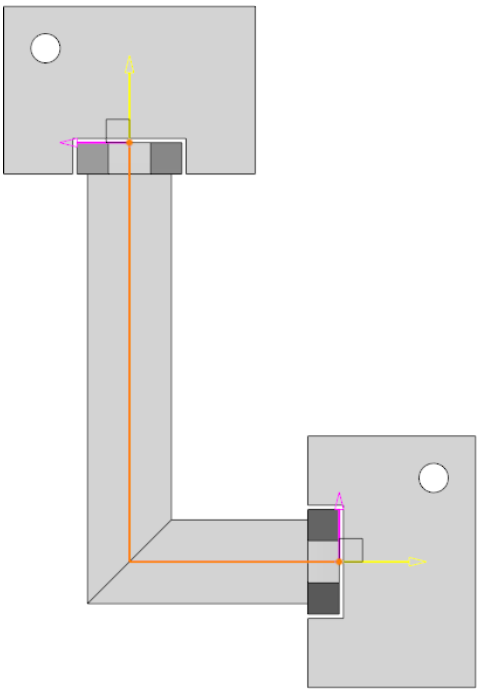
- Add two **miter trims**.



- Add **main cuts** in relation to the posts.

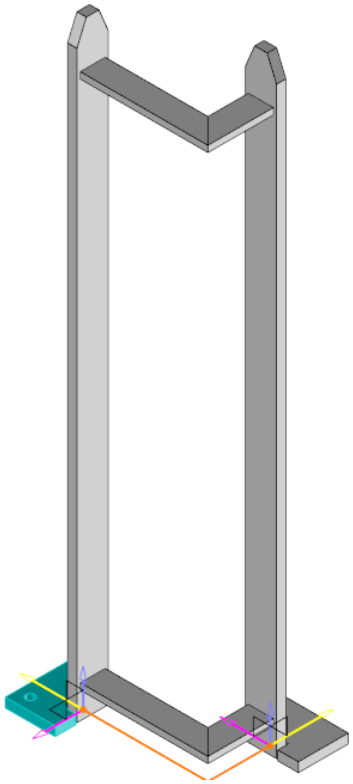


You should obtain the following top view result.

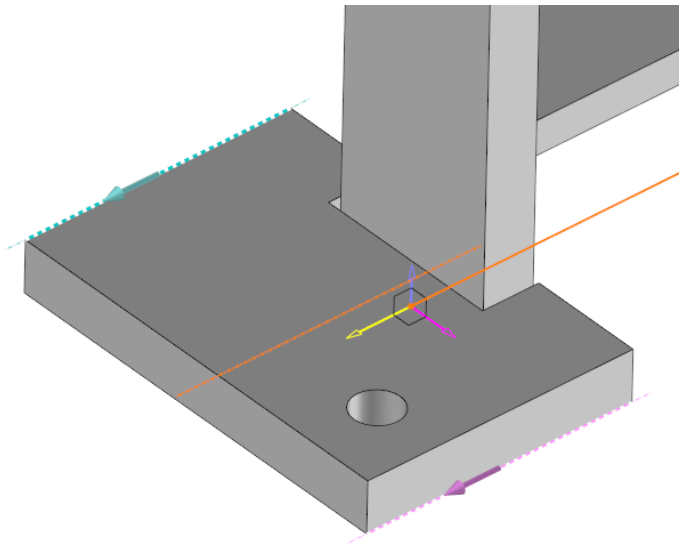


To make it easier to fix the second post, it would be ideal if the drilling of the plate was oriented toward the inside of the railing. **TopSolid** has quick modification tools to “transform” the objects.

- Select the **Construction** > **Transforms** > **Transform** command.
- Select the second plate as the **entity to transform**.

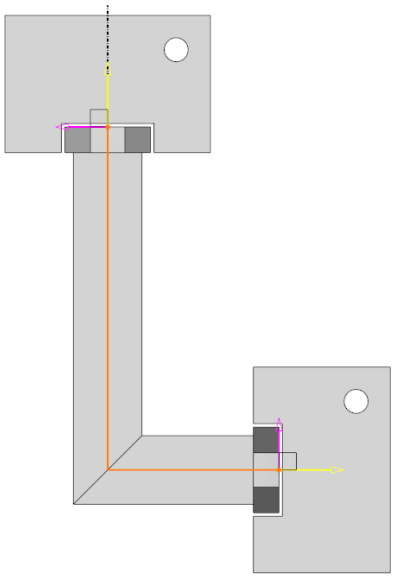


- In the **Transform** field, click on the icon and select the **Rotation Transform** command.
- As the rotation axis does not exist, you have to create it. To do this, in the **Axis** field, click on the icon and select the **Middle Axis** command.
- Select the two following axes.

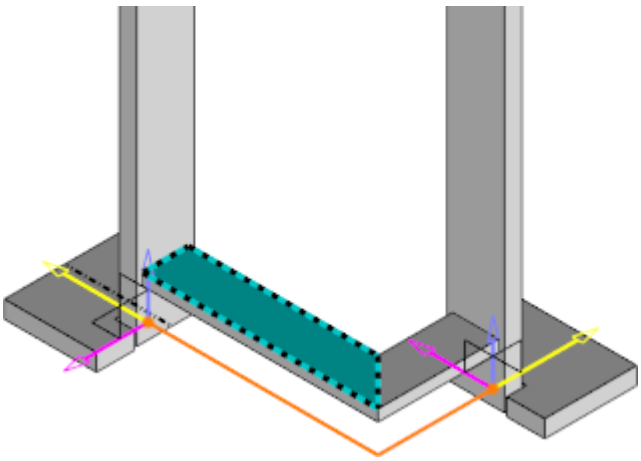


- Click on to **confirm** the middle axis.
- In the **Angle** field, enter *180°*.
- Click on to **confirm** the axis, the transformation pattern and the transformation operation.

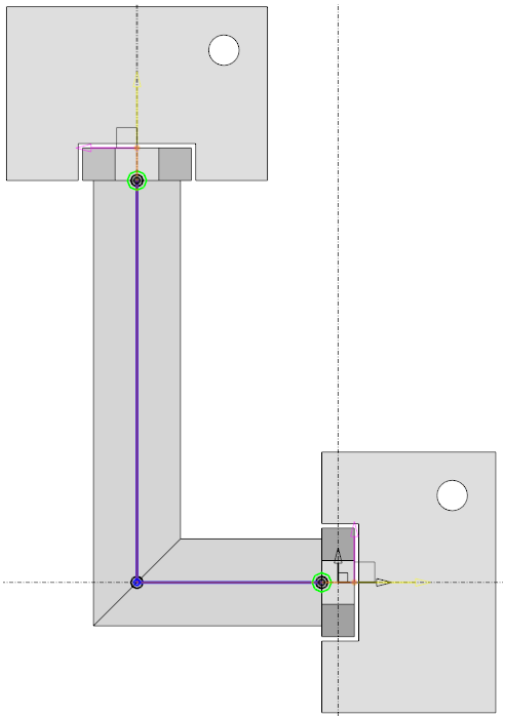
You should obtain the following result.



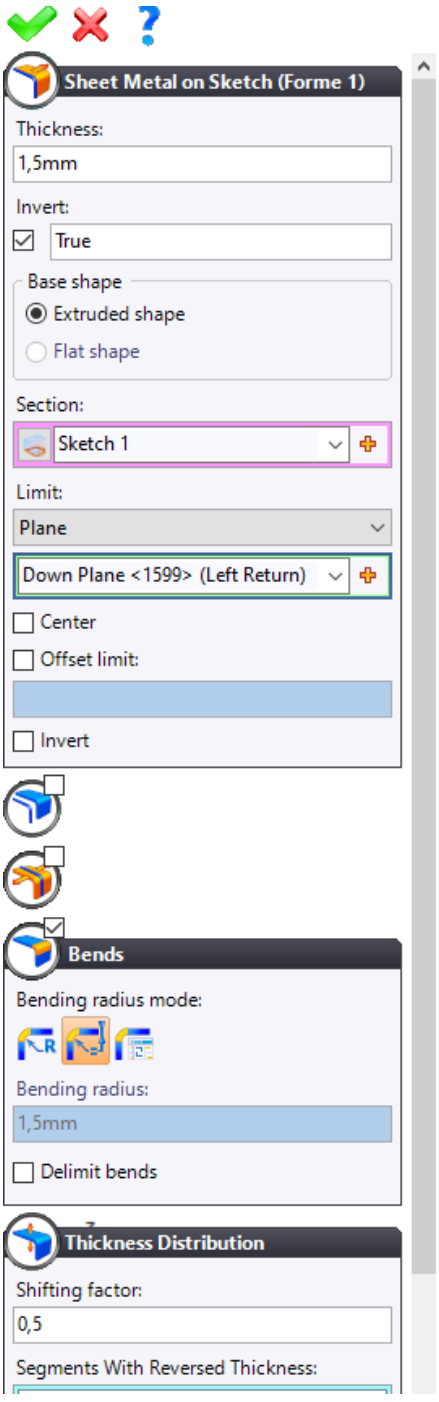
- Right-click on the following face and select the **In Place Part** command.

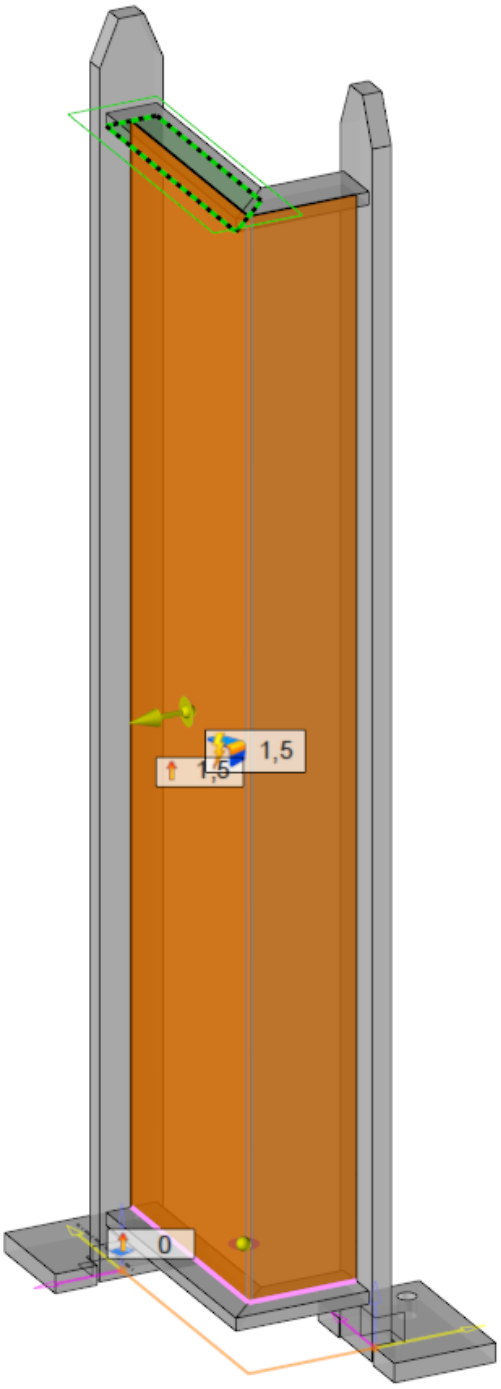


- Create the following sketch.



- Right-click in the graphics area with no active selection and select the **Sheet Metal on Sketch** command.
- Enter a **thickness** of *1.5mm.*
- In the **Limit** field, select **Plane** and select the lower face of the top rails.
- Check the **Bends** section and select the **Thickness** mode.
- In the **Thickness Distribution** section, enter a **shifting factor** of *0.5mm*.

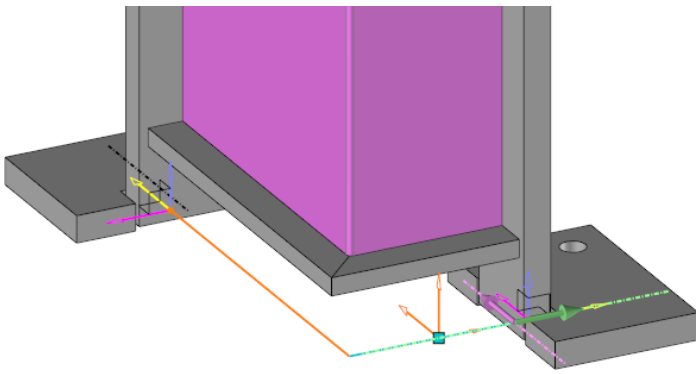




- Click on to **confirm** the operation.
- **Confirm** the in-place editing.
- Return to the assembly document and click on the icon in the document's tab.

In order for the component to be inserted at the right position, you have to reposition the distribution frame.

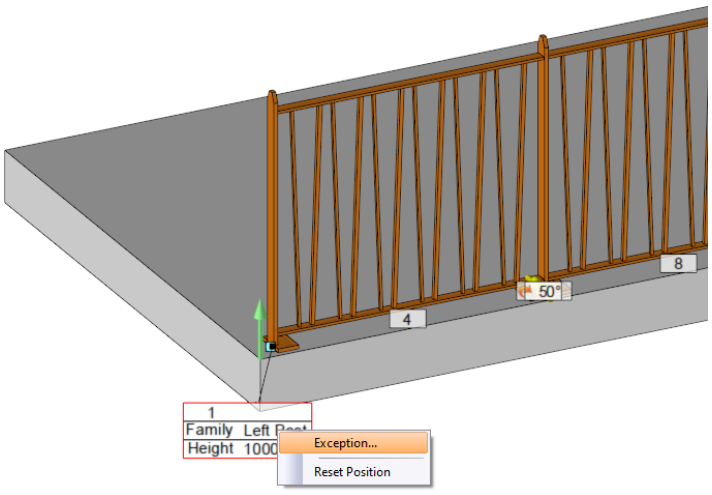
- To do this, edit the **Frame by Point and 2 Directions (Frame 3)** operation.
- Select the point as shown below as the **origin**.



- Click on to **confirm** the operation.
- Edit the operation providing the **Distributable Component** function.
- In the **Length** field, enter *Length1*.
- Click on to **confirm** the operation.
- **Save** (**Ctrl** + **S**) and **close** the document.
- Open the *Left return* family document.
- From the Entities tree's **Generics** folder, right-click on the **Length1** and **Length2** parameters and select **Move to Drivers**.
- **Save** (**Ctrl** + **S**) and **close** the document.

Using the side return

- Return to the *Railing with sections* assembly document.
- From the Operations tree, edit the **Distribution 1** operation.
- Right-click on the exception with number 1 and select **Exception**.



- In the **Family** field, select **Left return**.
- Click on to **confirm** the operation.

The next operations consist in repeating the previously performed steps (distribution, exception, handrail and creation of sub-assemblies).

A variant of the side return could have been possible if we had wanted it to be disassembled. Instead of starting from the left post, we could have used the junction post as the base file. The return would then have been disassembled.

Part 5 - Steel Furniture

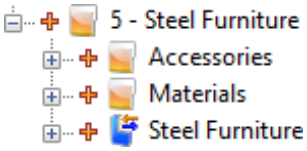
The following exercise is an introduction to the staging of objects. This first step allows you to become familiar with the creation of specific materials with textures and coatings or to add a mechanism to an object, in this case a door.

Concepts addressed:

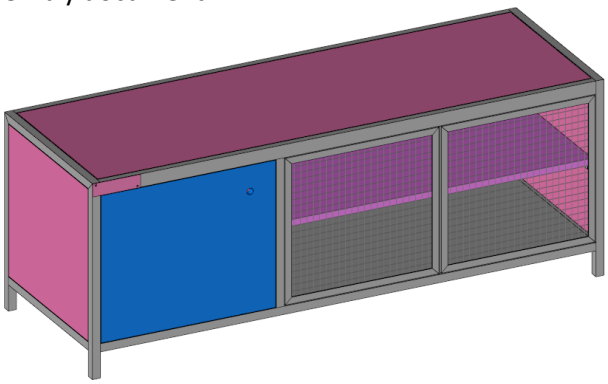
- Creating and applying textures, materials, coatings and finishings
- Cutting a sheet metal part from a vectorized text
- Applying a coating locally on a face
- Creating a mechanism on an imported hardware
- Creating configurations and projection in a drafting document

Importing the steel furniture

- In the Project tree, **import the package** named *5- Steel Furniture.TopPkg*.

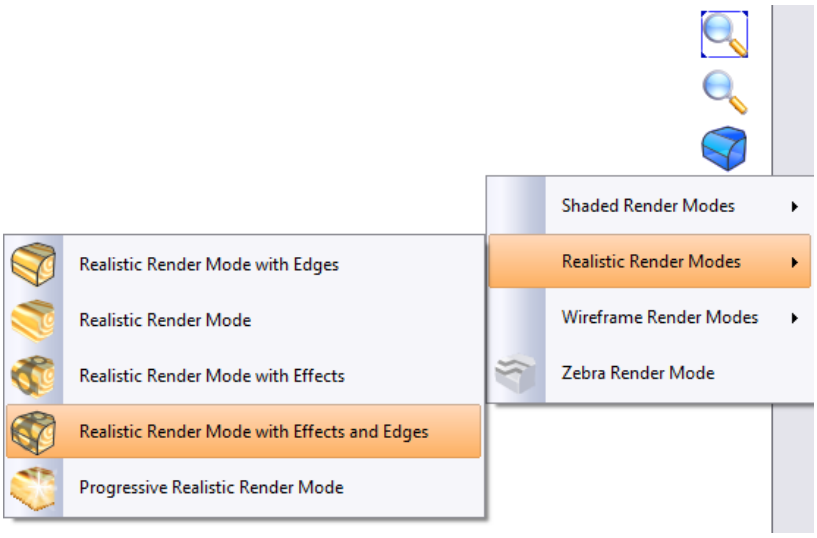


- Open the *Steel furniture* assembly document.

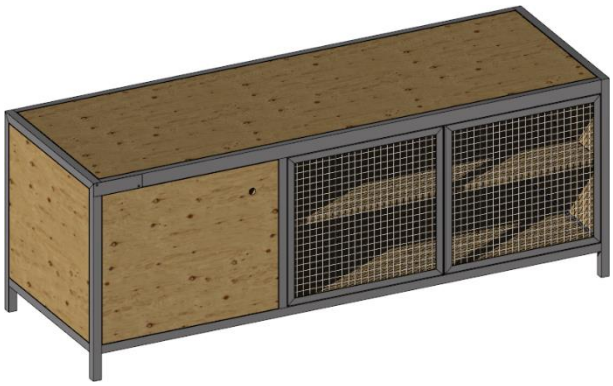


TopSolid has several render modes. The shading render mode is the default render mode and allows you to apply colors and degrees of transparency to the parts, regardless of the materials and coatings.

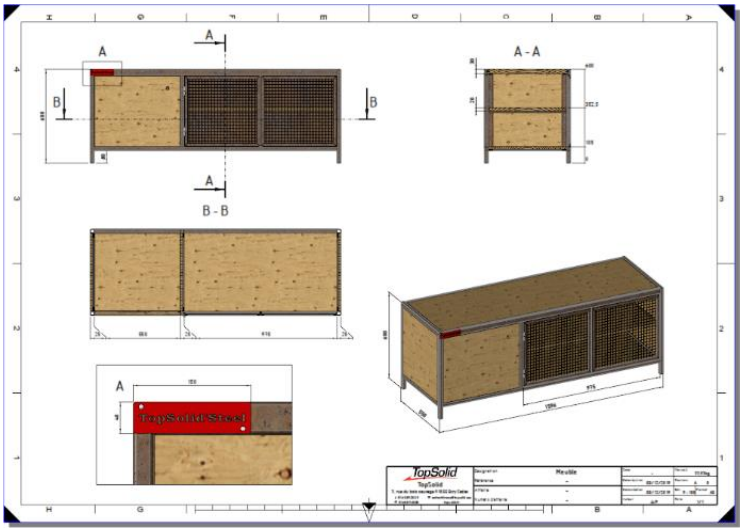
- Select the **Realistic Render Mode with Effects and Edges** mode.



Note: If you encounter display problems or if the **Realistic Render Modes** menu is not available, you have to check the compatibility of your graphic card with the **TopSolid** recommendations.
The realistic render mode consumes a lot of graphic resources. Depending on the hardware installed, delays may occur.



The selected rendering level makes it possible to see the final design. It can also be used in a drafting document.



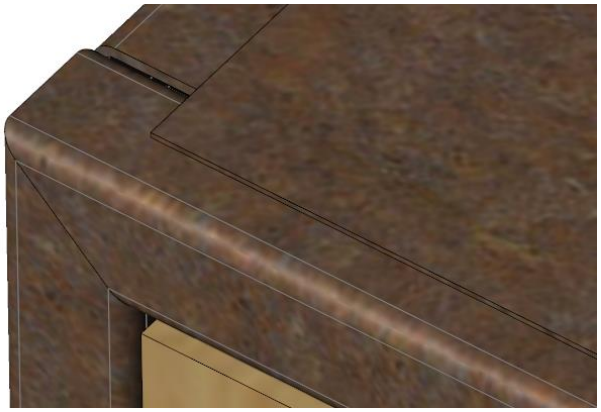
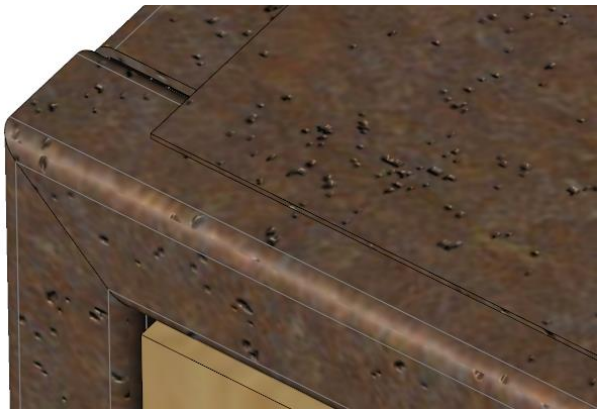
Creating a new textured material

A material is created in two steps:

- Creation of the texture documents: one for the color and one for the relief (optional).
- Creation of the material document in which the physical properties of the material as well as its rendering are defined.

- Right-click on the *5- Steel furniture > Materials* folder and select the **Document** command. From the **Advanced** tab, select **Texture** and use a **blank template**.
- Rename the document *Corten Steel.*
- In the **Parameters** section, click on the ... button to the right of the **Bitmap** field and select the *Corten Steel.jpg* file.
- Modify the **width** to *500mm*. This value corresponds to the actual size of the image.
- **Save** (**Ctrl** + **S**) and then **close** the document.

We will now create the texture that will give relief to our texture. Depending on the materials, the relief strongly changes the rendering.

Without relief	**With relief**
	

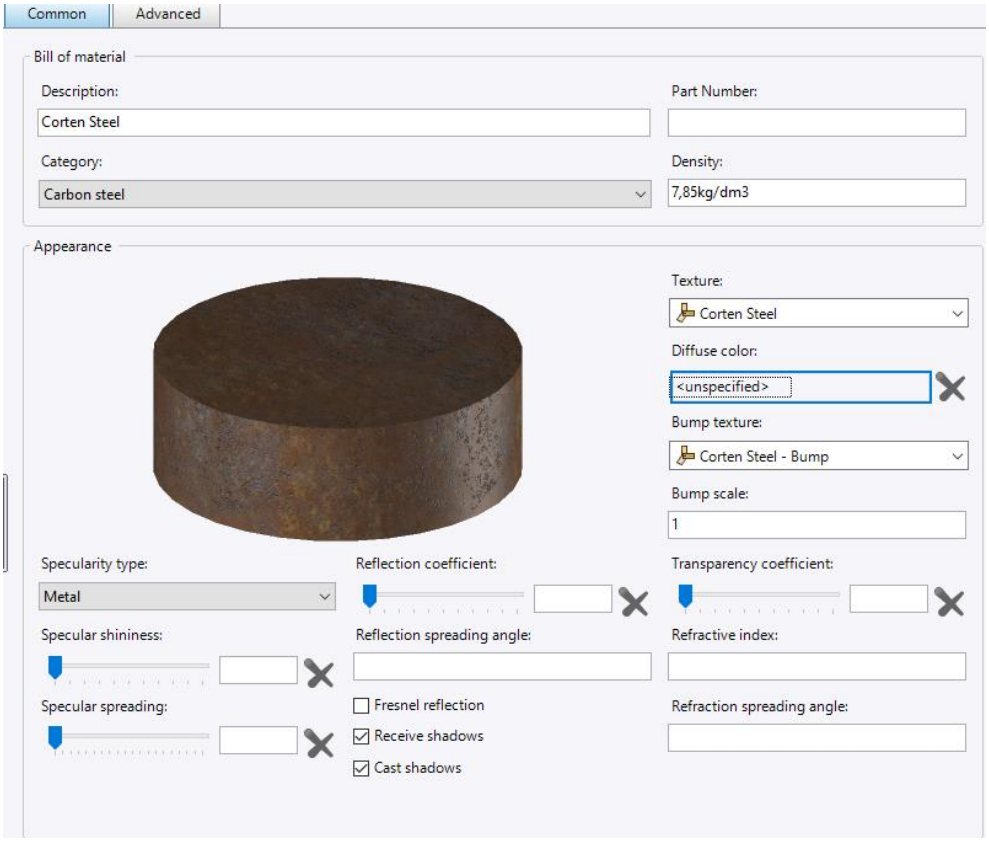
- Right-click on the *5- Steel furniture > Materials* folder and select the **Document** command. From the **Advanced** tab, select **Texture** and use a **blank template**.
- Rename the document *Corten Steel - Bump*.
- In the **Category** field, select **Bump**.
- In the **Parameters** section, click on the ... button to the right of the **Bitmap** field and select the *Corten Steel - Bump.jpg* file.
- Modify the **width** to *500mm*. This value corresponds to the actual size of the image.
- **Save** (**Ctrl** + **S**) and **close** the document.

We will now create the material that will contain the two previously created textures.

- Right-click on the *5- Steel furniture > Materials* folder and select the **Document** command.
- From the **Advanced** tab, select **Material** and use a **blank template**.
- Rename the document *Corten Steel.*
- In the **Description** field, enter *Corten Steel*.
- In the **Category** field, select **Carbon steel**.

The category choice is important. The category provides quick access to the desired material and defines the hatching style in the drafting document.

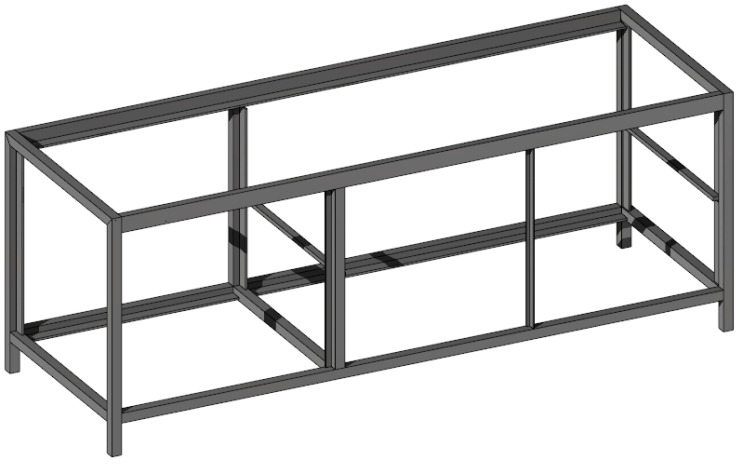
- In the **Density** field, enter *7.85kg/dm3.*
- In the **Texture** field, select **Corten Steel**.
- In the **Bump texture** field, select **Corten Steel - Bump**.
- In the **Bump scale** field, enter *1*. This value allows you to accentuate the bump effect. The higher the value, the more visible the relief will be.
- In the **Specularity type** field, select **Metal**.



- **Save** (**Ctrl** + **S**) and **close** the document.

Using the new material

- Return to the *Steel furniture* assembly document.
- From the Parts tree, click on the **Hide All** icon.
- In the column, check **Frame**.



- Select the **Tools** > **Parts Material and Coating** command.
- Select all the parts that are visible on the screen.
- Uncheck the **Derive material** box. As a reminder, this allows you not to use the material used when creating the library.
- In the **Category** field, select **Carbon steel**.
- In the **Material** field, select **Corten Steel**.
- Click on to **confirm** the operation.

- From the Parts tree, click on the **Show All** icon.
- **Save** the document (**Ctrl** + **S**).

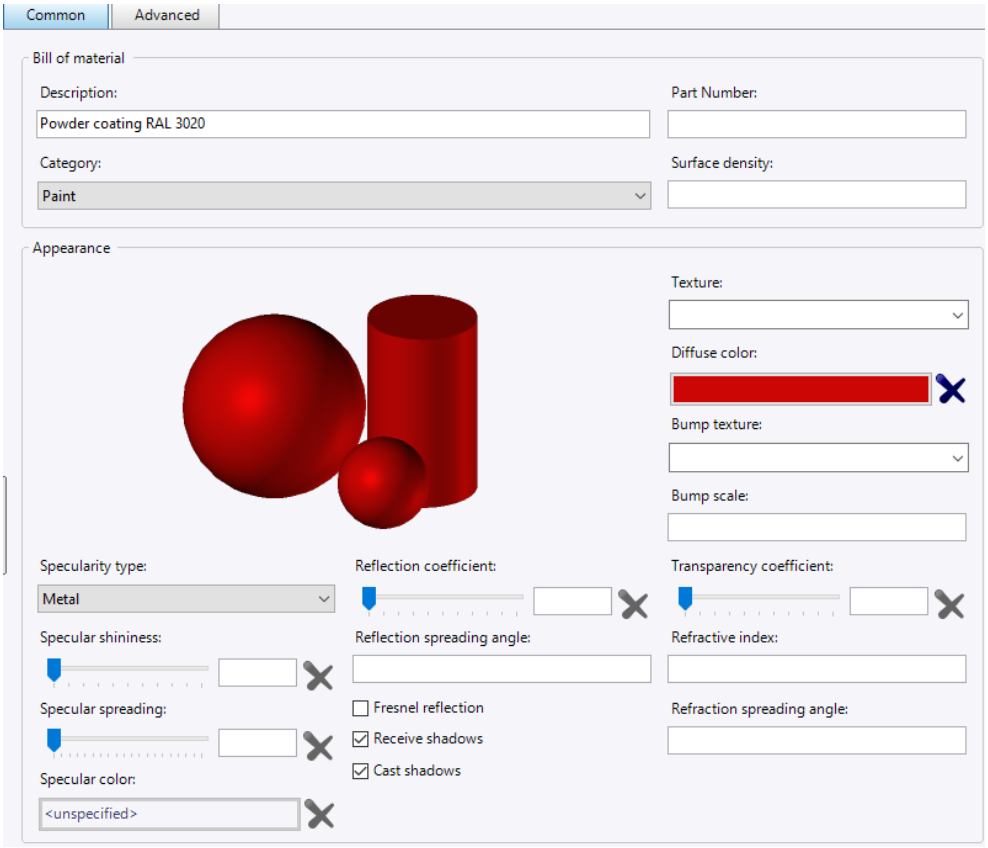
Creating a new coating

TopSolid allows you to apply a layer called "coating" on a material. This type of document is used to add a galvanization, a paint or a specific treatment.
This coating can be applied globally on one or more parts (by using the same command as before) or locally on a batch of faces belonging to the same part.

- Right-click on the *5- Steel furniture > Materials* folder and select the **Document** command.
- From the **Advanced** tab, select **Coating** and use a **blank template**.
- Rename the document *Powder coating RAL 3020*.
- In the **Description** field, enter *Powder coating RAL 3020*.
- In the **Category** field, select **Paint**.
- Click on the **Diffuse color** button and enter the following values:
  - **Red**: *204*
  - **Green**: *6*
  - **Blue**: *5*

These values are available on the Internet by typing the keywords "RAL RGB".

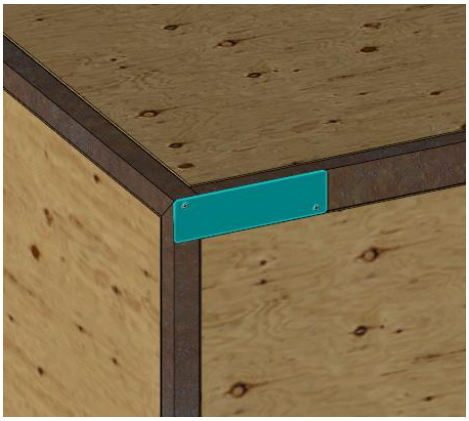
- In the **Specularity type** field, select **Metal**.



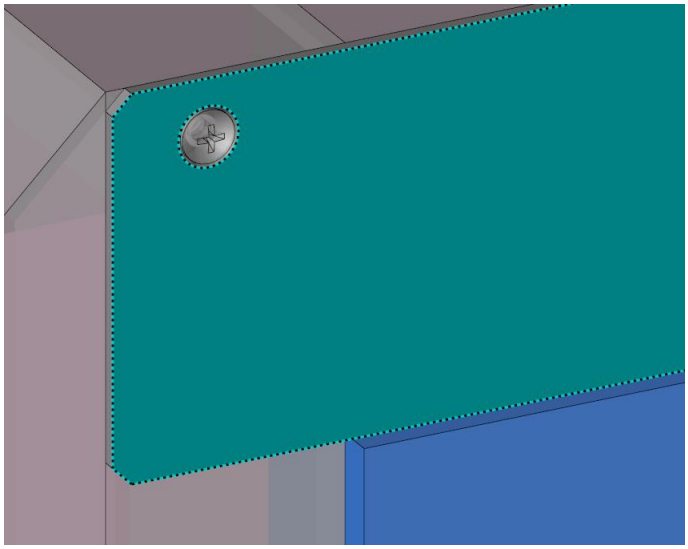
- **Save** (**Ctrl** + **S**) and close the document.

Applying a local coating

- Return to the *Steel furniture* assembly document.
- Double-click on the following part to edit it in place.

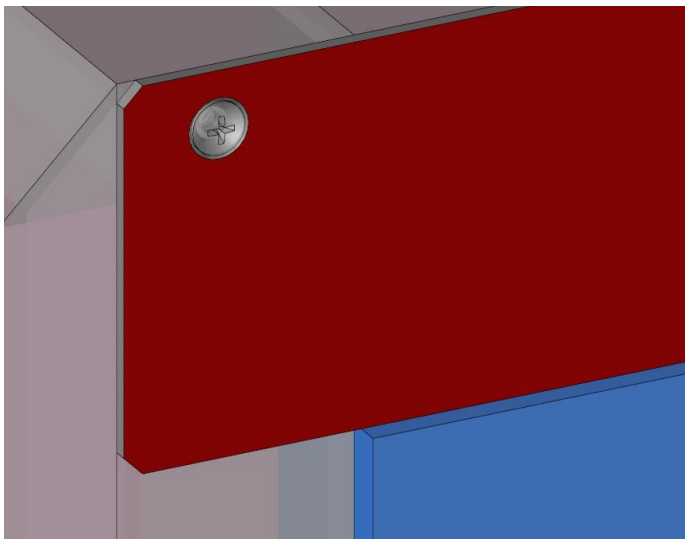


- Select the **Shape** > **Other Operations** > **Coating** command.
- In the **Faces** field, select the face as shown below.
- Select **Paint** in the **Category** field and select **Powder coating RAL 3020** in the **Coating** field.



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

You should obtain the following result.



Cutting a sheet metal part from a vectorized text

Cutting a sheet metal part from a text requires the use of a special Windows font. In fact, standard fonts cannot be used because some sections of the different letters would fall off (O, P, Q, B, etc.).

- In Windows Explorer, in the training folder, double-click on the *LaserCutOldStyle.ttf* file.
- Click on the **Install** button at the top left of the window. This action adds a new font to Windows. This font will be available in all applications using Windows fonts (**TopSolid** is one of them).

Note: On some computers, it may be necessary to restart **TopSolid**, sometimes even the computer itself.

- In **TopSolid**, switch back to the in-place editing context of the red sheet metal part.
- Create a new sketch on the face painted in red.
- From the **2D Sketch** tab, select the **Text** command.
- In the **Text** field, enter *TopSolid'Steel* or any other text of approximately the same length.
- Check the **Profiles** box. This option allows you to convert a text into profiles (vectorization).
- In the **Format** section, check the **Font** box and click on the button below.
- Select the **LaserCutOldStyle** font, enter a **height** of *10mm* and click on to **confirm** the operation.
- Check the **Expansion** box and enter *1.2*. This option enlarges the text by 20% (x1.2).
- Position the text approximately in the center of the plate.

- Click on to **confirm** the text creation.
- Right-click on the text and select the **Annotation Centering** command.
- Select the left vertical edge of the red plate, and then the right vertical edge.

TopSolid moves to the next centering.

- Select the text, the upper horizontal edge, then the lower horizontal edge.

The text should turn blue.

- Right-click in the graphics area with no active selection and select the **Trim by Profile** command.

The arrow indicates the direction of the trimmed side.

- Make sure that it is pointing toward the inside of the text.

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

Additional exercise: Importing a hinge and creating a mechanism

In metalworking, you sometimes need to simulate a mechanism such as the opening of a door, a gate, a trap door, etc.

There are many benefits:

- explain a movement to your customer;
- facilitate the teams' understanding of a mechanism;
- know enclosing dimensions, displacements, etc.;
- facilitate the design of complex mechanisms.

In most cases, these mechanisms are given by accessories (hinges, rack and pinions, slides, etc.). The scenario to animate a design is the following (with the example of the piece of furniture):

- Importing the hinge.
- Creating the rigid groups (groups of parts that do not move with each other). There are two rigid groups in a hinge.
- Adding the joint between the rigid groups. We can define which movement is possible between the rigid groups.
- Designing the furniture with its hinges.
- Creating the rigid groups to link the different groups of accessories with the different parts of the furniture.
- Creating configurations (open doors, closed doors, half-open doors, etc.).

- From the Project tree, create a new folder named *Hinge* in the *5- Steel furniture > Accessories* folder.
- Right-click on this new folder and select the **Import/Export** > **Import File with Conversion** command.
- Select the file named *Welded Steel Hinge - Fixed Axis - Steel Axis 37-720-60.A.0.stp*.

The import dialog box is displayed.

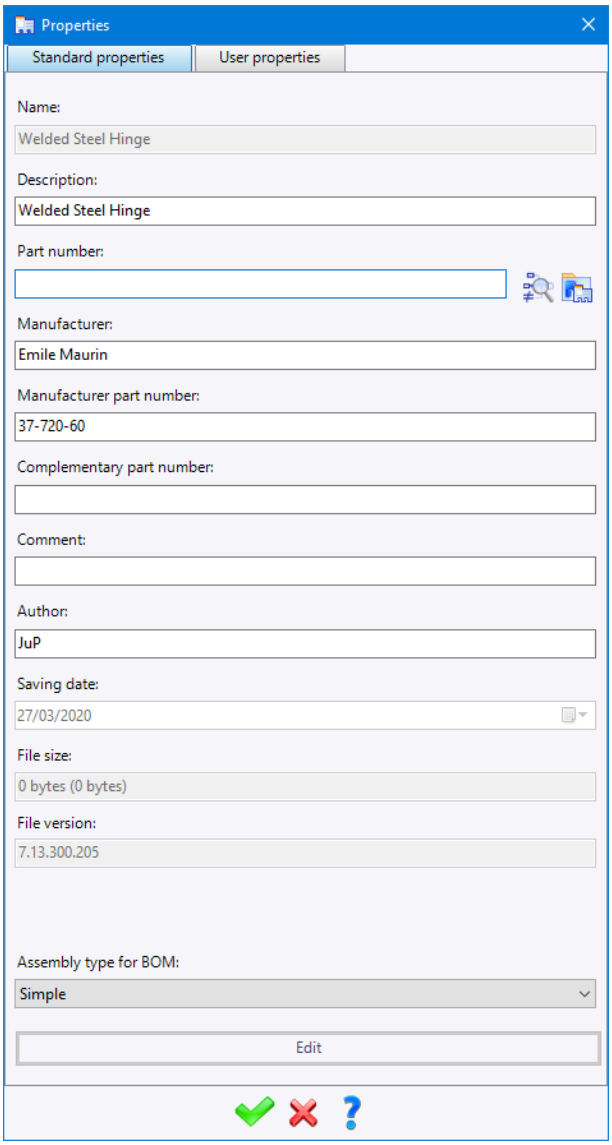
- In the **Simplification and sewing** tab, check the **Simplify geometry** box.
- In the **Templates** tab, click on the ... button to the right of **Part template**.
- Select **Project Templates** > **Defaults** > **Part** and click on to **confirm** the operation.
- Click on the ... button to the right of **Assembly template**.
- Select **Project Templates** > **Defaults** > **Assembly** and click on to **confirm**.
- Click on to **confirm** the import.

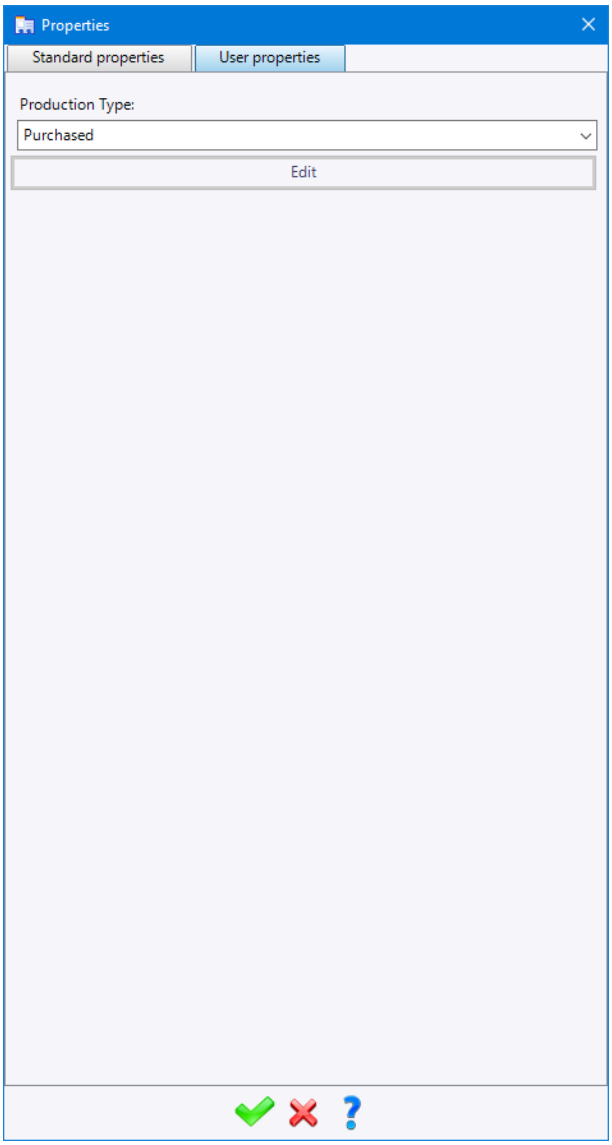
Note: To import the manufacturers' 3D components, we recommend the following formats:

- Step (.stp)
- Parasolid (.x_t)
- AutoCad 3D (.dxf/.dwg)

Other formats can be used but the geometries can be altered, which may hinder their use in **TopSolid**.

- Edit the properties of the imported assembly and fill in the fields as shown below.





The **Assembly type for BOM** property allows you to select the assembly structure in the bill of materials. The following four modes are available:

- **Absent**: The hinge will not be included in the bill of materials. It will still be part of the parent assembly. It will be projected in the drafting document and its mass will be taken into account.
- **Composite**: The hinge and the parts that make it up will be visible in the bill of materials. It is the default mode used in **TopSolid**.
- **Simple**: Only the hinge will be included. This is the ideal mode for commercially available components. This allows you to simply display the line containing the manufacturer's part number and not the detail of the sub-parts which have no interest in a bill of materials.
- **Transparent**: Only the parts making up the hinge will be visible. This mode is widely used to create groups of parts that are easier to insert in one go. For example, a screw and its washer, an extruded bar and its seal. The assembly is not necessary in a bill of materials. Its parts and sub-assemblies are instead preferred.

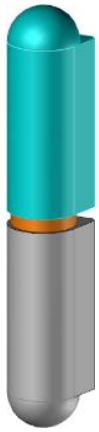
In the **User properties** tab, the **Production Type** property allows you to perform sorts and filters. For example, the result can be a bill of materials of all the purchased components, sorted by manufacturer and by reference.

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

- Open the assembly named *Welded Steel Hinge 37-720-60.*
- Select the **Mechanism** > **Mechanism** command.
- Click on to **confirm** the **Mechanism** dialog box.

A **Ground Group** rigid group is created by default. It is the reference element of our assembly. All parts are currently located in this rigid group.

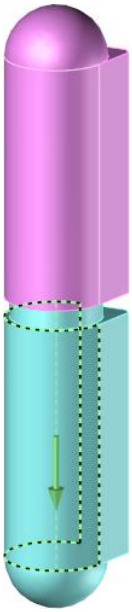
- Select the **Mechanism** > **Rigid Group** command.
- Select the entity as shown below.



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

We will now define the joint between the two rigid groups.

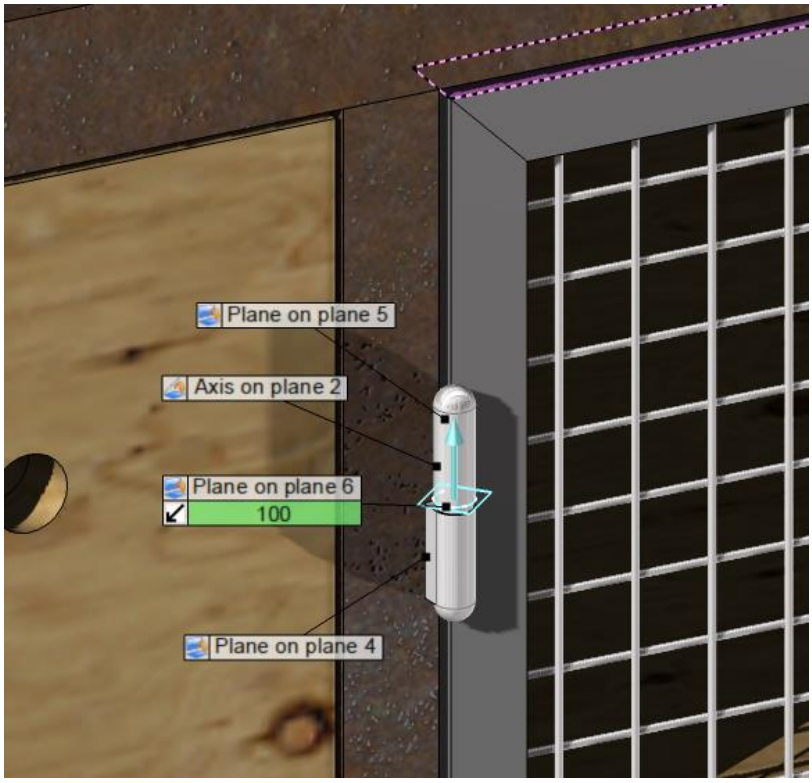
- Select the **Mechanism** > **Revolute** command.
- Select **Ground Group** from the drop-down list or directly from the graphics area as the **first rigid group**.
- Select **Group 1** as the **second rigid group**.
- Select the hinge axis as the **rotation axis**.



- Click on to **confirm** the operation.
- **Save** (**Ctrl** + **S**) and then close the document.

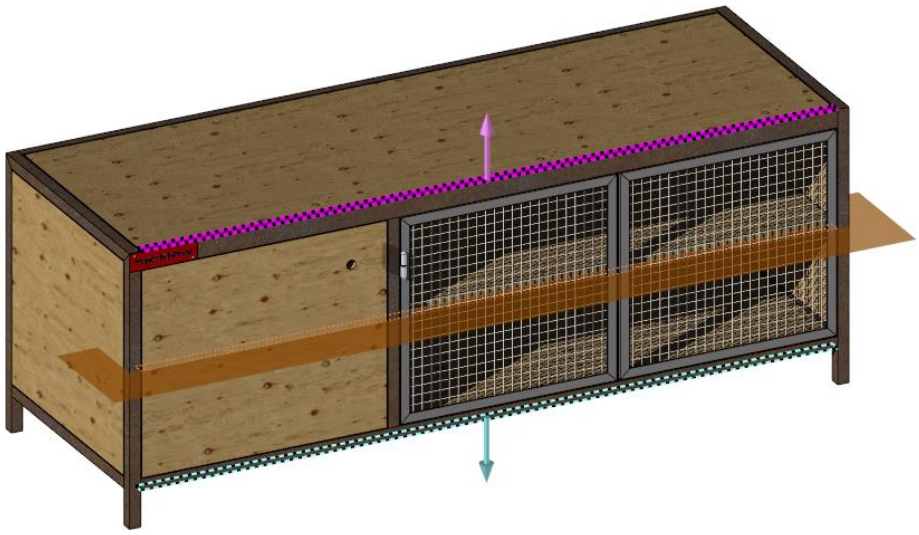
Using the hinge and the mechanism

- Return to the *Steel furniture* assembly document.
- Insert the hinge to be welded and position it as shown below.



TopSolid uses the degrees of freedom of the mechanism. This allows you to orientate each part of the hinge toward the two supports (the frame and the door) on which it will be welded.

- Select the **Construction** > **Repetition** command.
- Select the steel hinge to be welded as the **entity**.
- Create a **symmetrical pattern**.
- Select **Plane** as the **symmetry type**, then create a **midplane** between the upper face and the lower face of the furniture.



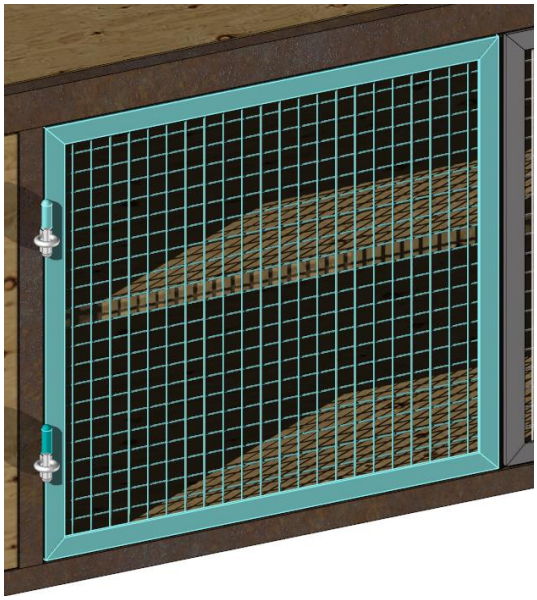
- Select **Translation** as the **transform type**, then select the **transform origin** as shown below.

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

- Repeat all the operations for the hinges of the right door:
  - inclusion and positioning of the hinge;
  - repetition of the hinge by reusing the previously created pattern.

We will now create the mechanism.

- Select the **Mechanism** > **Mechanism** command.
- Click on to **confirm** the operation.
- Select the **Mechanism** > **Rigid Group** command.
- In the **Name** field, enter *Left door*.
- Select the left door and the two upper sections of the hinges.



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

- Repeat the procedure on the right side by naming the rigid group *Right door*.

The revolute joints are already defined since they have been retrieved from the different hinges. Nevertheless, you need to inform **TopSolid** that the lower sections of the hinges are fixed to the frame.

- Select the **Mechanism** > **Merge Rigid Groups** command.
- Select the ground as the **rigid group to keep** and select the lower section of one of the hinges as the **rigid group to absorb**.

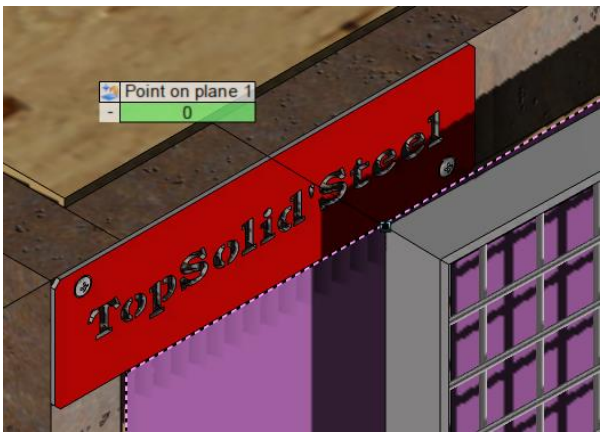
- Repeat the procedure for the three other hinges.

We will now open the doors to be able to use different configurations in the drafting document.

- Select the **Mechanism** > **Constrained Configuration** command.

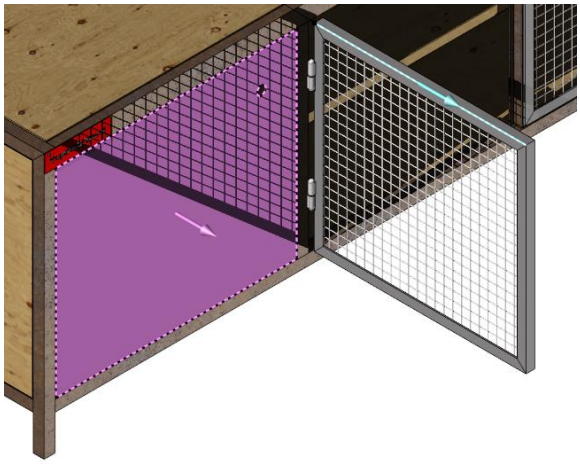
You can perform actions directly on the doors by sliding them.

- Add a **Point on plane** positioning constraint as shown below.



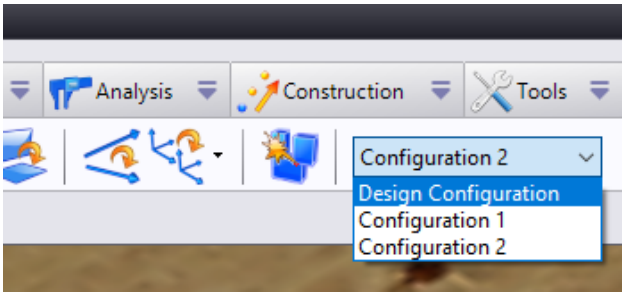
The objective is to put the door in contact with the frame to represent the maximum opening.

- **Confirm** the configuration context.
- Select the **Mechanism** > **Constrained Configuration** command to create a new configuration.
- On the same door, add an **orientation** constraint by selecting the following two elements.



- **Confirm** the configuration context.

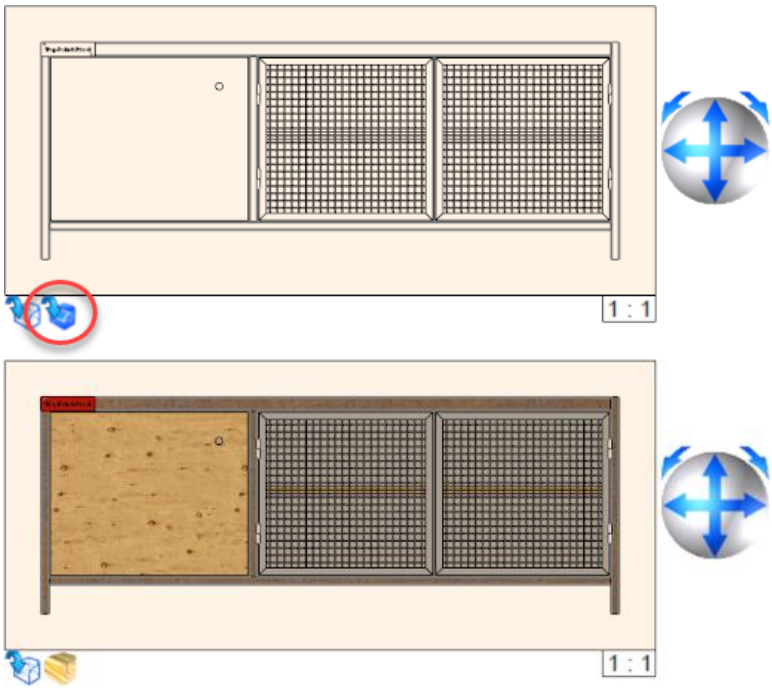
From the **Mechanism** tab, you can switch from one configuration to another using the drop-down menu on the right.



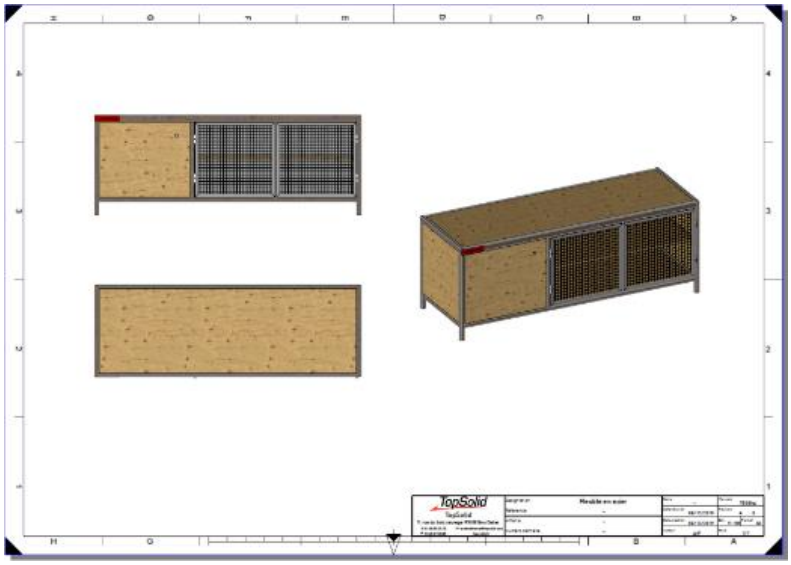
- From the Entities tree's **Mechanism** > **Configurations** folders, rename configuration 1 *Open left door* and configuration 2 *Half-open left door*.

Drafting the constrained configurations

- Create a drafting from the steel furniture using the **Assembly A3 ISO Landscape** template from the **Steel Standard Templates - United States** folder.
- Position the first view.
- Before confirming the operation, click on the icon shown below until the view switches to realistic rendering mode.



- Click on to **confirm** the main view.
- Add the following two views.



The design configuration is projected by default. At any time, you can use another configuration or even overlay other configurations.

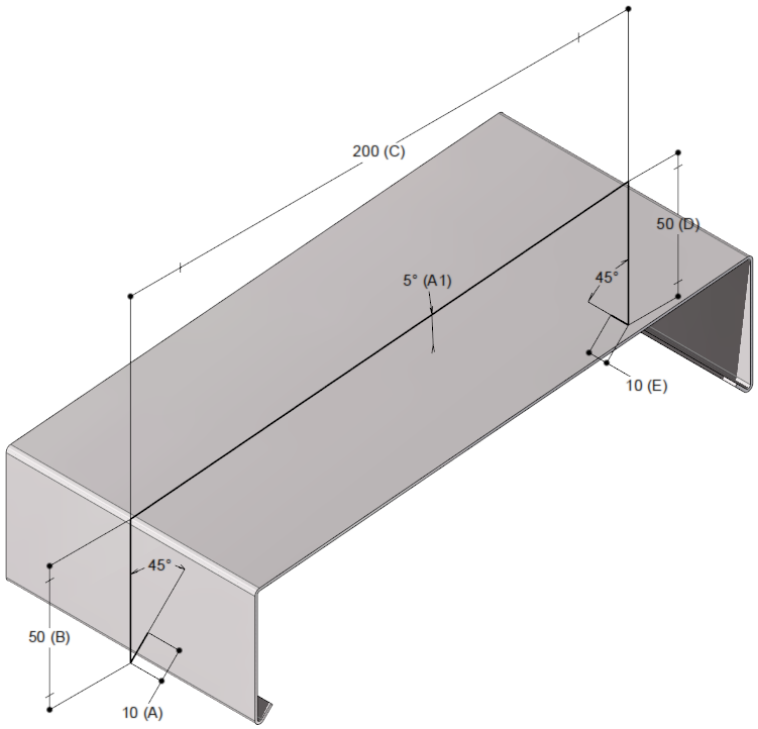
- Right-click on the main view and select the **Edit Set** command.
- Click on the **Auxiliary configurations** button.
- Select the two configurations on the left and click on the icon.
- Click on to **confirm** the operation, and then the set editing.

You should obtain the following result.

You can set dimensions to these representations and configure their appearance using the view style (auxiliary line).

Annex 1 - Creating a Sheet Metal Extruded Bar (Coping)

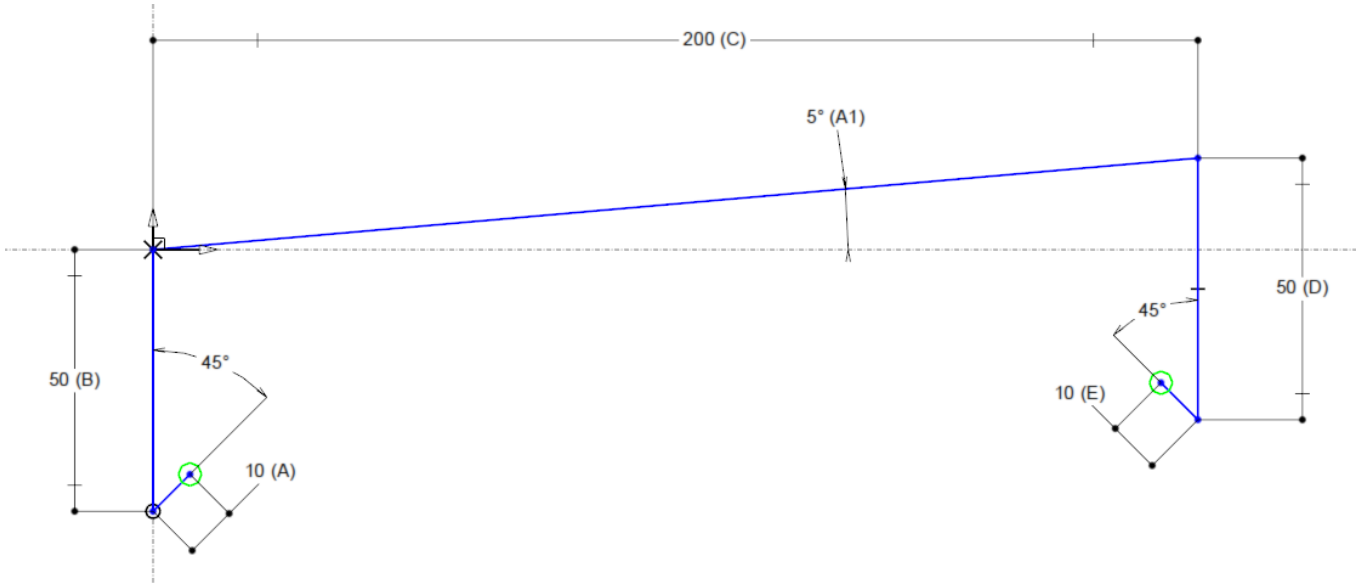
The purpose of this exercise is to design a sheet metal extruded bar using different dimensional parameters. The example will be a coping of the following type.



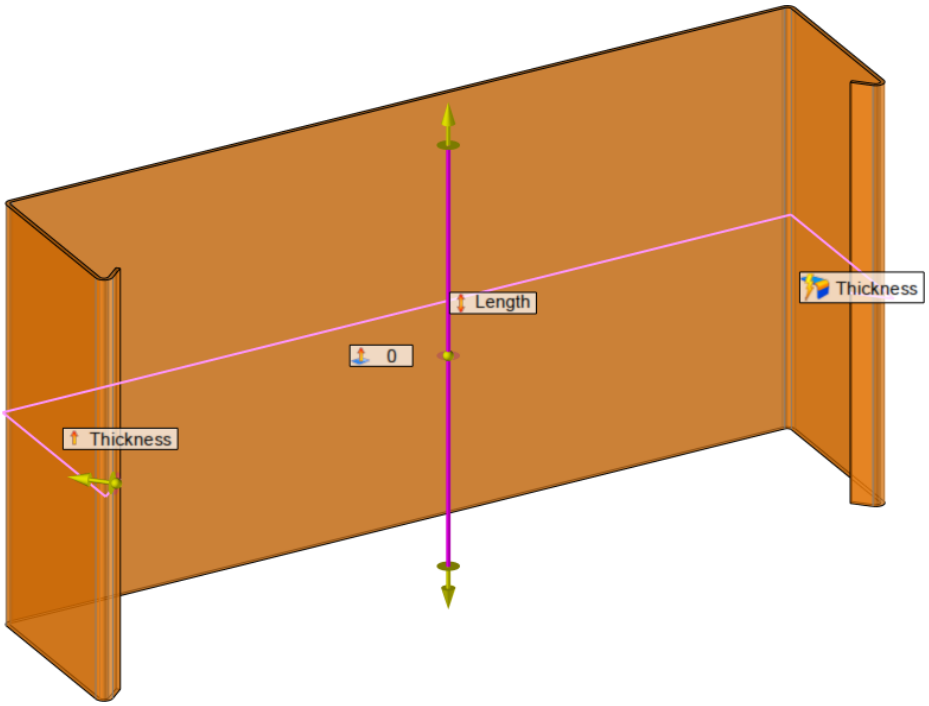
Concepts addressed:

- Creating a parameterized sketch
- Sheet metal commands
- Creating an extruded bar using the **Extruded bar** function

- From the Project tree, create a new folder named *A1- Coping*.
- Create a **Part** document in the *A1- Coping* folder.
- Rename the part document *Coping*.
- Create the following sketch.



- Right-click in the graphics area with no active selection and select the **Sheet Metal on Sketch** command.
- Enter *Thickness=1mm* in the **Thickness** field.
- Enter *Length=100mm* in the **Length** field.
- Check the **Bends** section and select the **Thickness** mode.



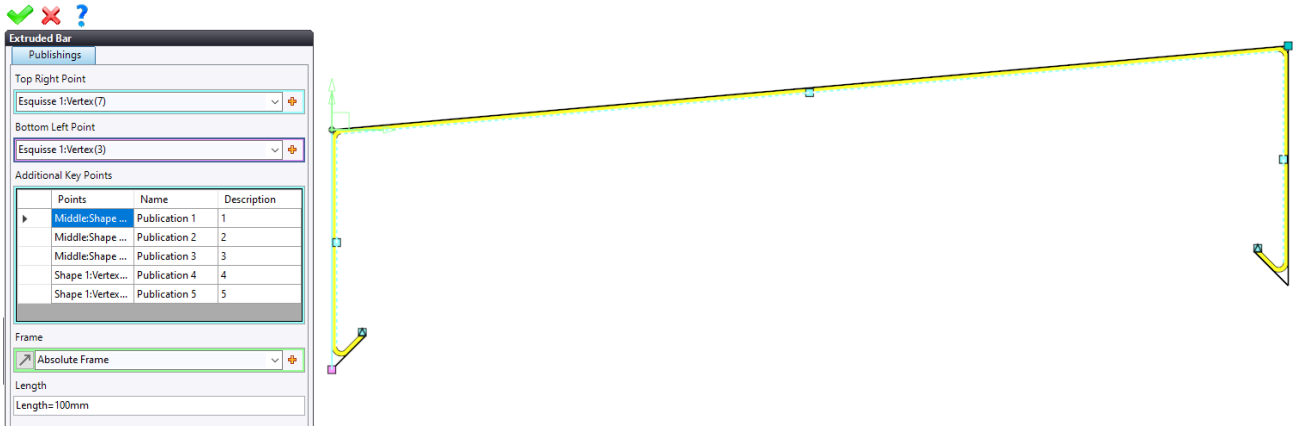
- Click on to **confirm** the operation.
- Select the **Tools** > **Functions** > **Provide Function** command.
- Select the **Extruded Bar** function.

As with the distributable component, **TopSolid** needs two points to facilitate the positioning.

- From the Entities tree, display the sketch that was used to create the sheet metal part (**Sketch 1** normally).
- Select the following points as the **top right point** and **bottom left point**.

To facilitate the positioning of the extruded bar, you can select additional positioning points.

- Click in the **Additional Key Points** table and add the following points.

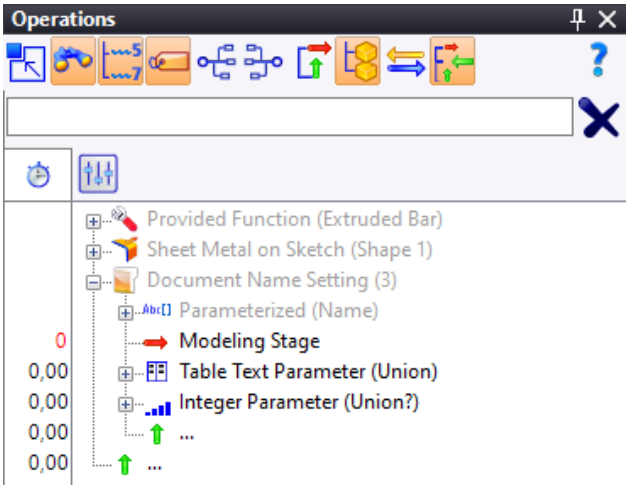


TopSolid asks you to enter a description for each point.

- Simply add a number.
- Select **Absolute Frame** as the **frame** and the **Length** parameter as the **length**.
- Click on to **confirm** the function.

We will now parameterize the description of the component in order to integrate the **Code** parameter.

- From the Operations tree, move the **Modeling Stage** cursor as shown below.



- From the Entities tree's **Parameters** > **System Parameters** folders, right-click on the **Description** parameter and select the **Others** > Abc[] **Parameterized** command.
- In the **Value** field, enter *Coping [$Code]*.

We could have also added all the part's dimensions:

Coping [A] x [B] x [C] x [D] x [E] Th[Thickness]

- Click on to **confirm** the description parameterization.
- In the document's tab, click on the **End inserting** icon.
- **Save** the document (**Ctrl** + **S**).

- Right-click on the part document's tab and create a **Family** document.
- From the Entities tree's **Generics** folder, drag the **C** parameter into the table containing the codes.
- Enter the following values in the table.

	Code	C
	200	200mm
	270	270mm
▶	400	400mm
*		

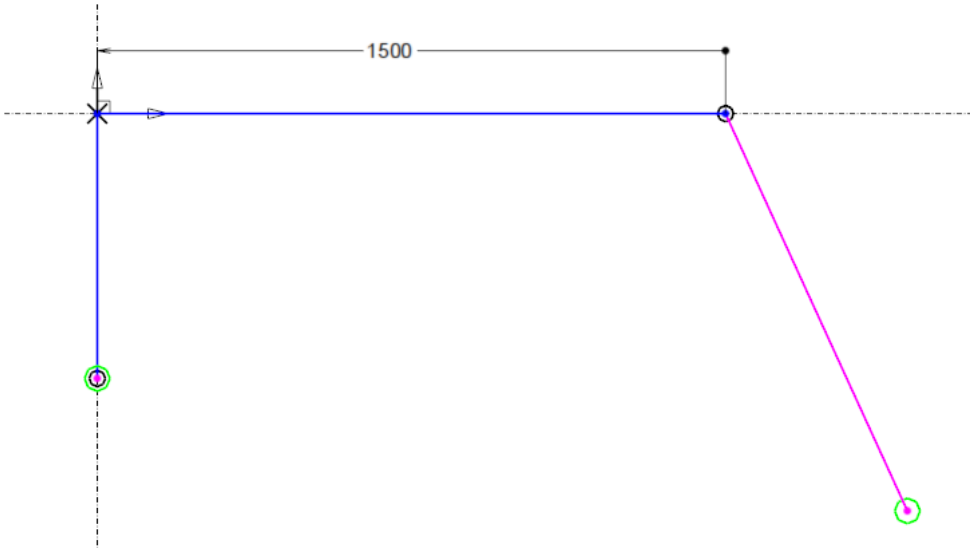
- From the Entities tree, select the **Length**, **Thickness**, **A1**, **A**, **B**, **D** and **E** parameters.
- Right-click and select **Move to Drivers**.
- Right-click on the **C** parameter and select **Move to Optional Drivers**.

As a reminder, this last manipulation will allow you, if necessary, to create widths that are not included in the catalog codes.

- **Save** (**Ctrl** + **S**) and close the family document and its generic.

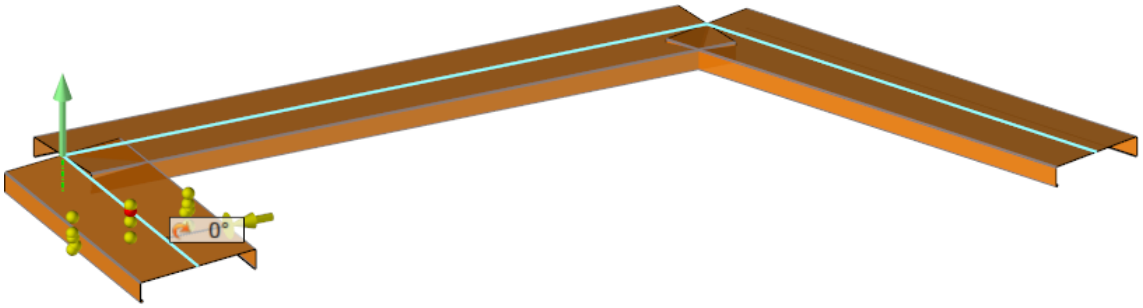
We will now use the new extruded bar.

- In the *A1- Coping* folder, create an **Assembly** document.
- Draw the following sketch.

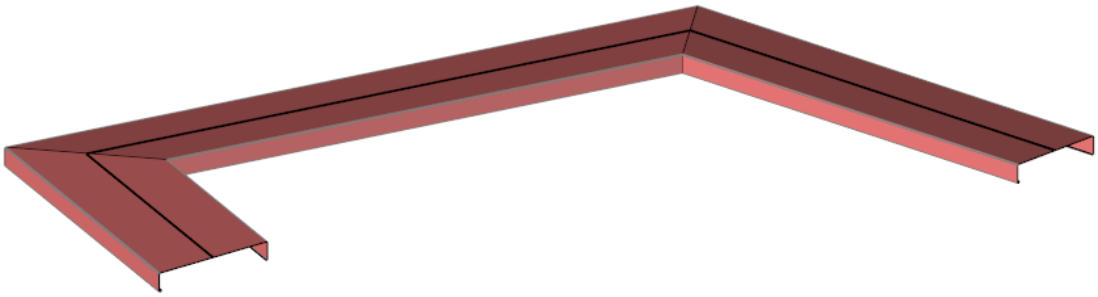


Note: Values are not important.

- Add the **Coping** extruded bars as shown below.



- Add two miter trims.

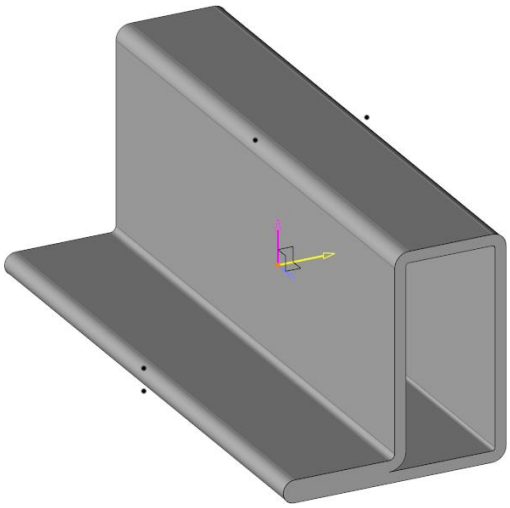


- **Save** (**Ctrl** + **S**) the assembly document.
- Produce a sheet metal **bill of materials** using the **Sheet Meta**l template from the **Steel Standard Templates** folder.

You should obtain a cutting list for the different copings. The bill of materials could have been of the extruded bar type in the case of standard copings purchased in lengths.

Annex 2 - Creating an Extruded Bar from a DXF Section

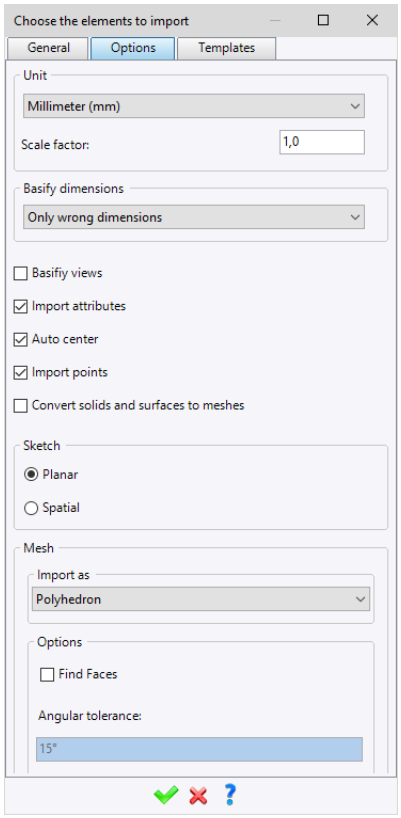
The purpose of this exercise is to design an extruded bar based on a DXF/DWG section from a process development engineer.



Concepts addressed:

- DXF/DWG import
- Creating an extruded bar using the **Extruded bar** function

- From the Project tree, create a new folder named *A2- Folding extruded bar L*.
- Right-click on the *A2- Folding extruded bar L* folder and select the **Import/Export** > **Import File with Conversion** command.
- Select the *Folding Extruded Bar L.dwg* file.
- Select **Part** from the **Import as** drop-down list.
- From the **Options** tab, check the settings of the following parameters.



- In the **Templates** tab, click on the button to the right of **Part template**.
- Select **Project Templates** > **Defaults** > **Part** and click on to **confirm** the operation.
- Click on to **confirm** the import.

The drawing document is displayed in the foreground.

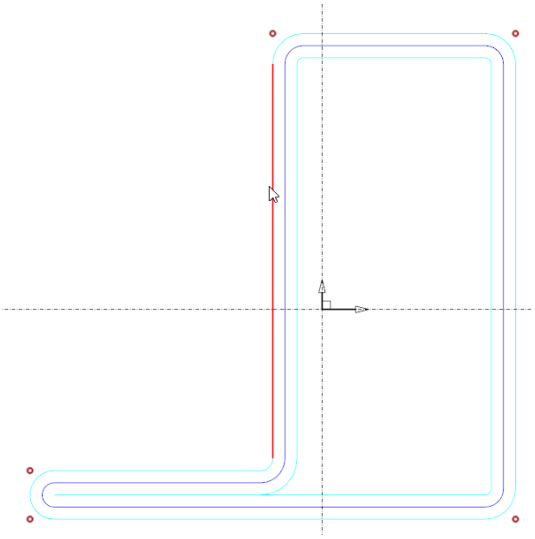
- Select the **2D Sketch** > **Healing** > **Clean** command.
- Select the sketch and click on the icon.

TopSolid displays the created lines and circles.

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

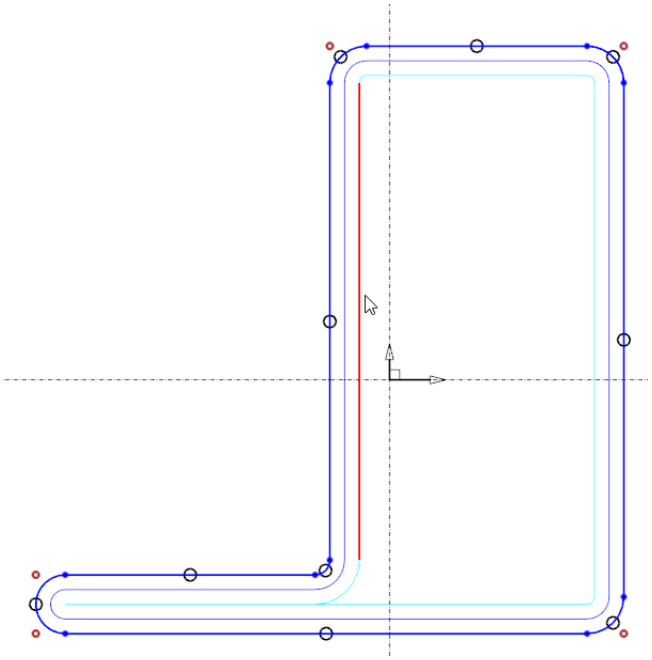
The imported DWG file contains many lines that are not necessary to build the 3D extruded bar.

- Create a new sketch and select the **Contour Wizard** command.
- Click on one of the outer contour's edges.

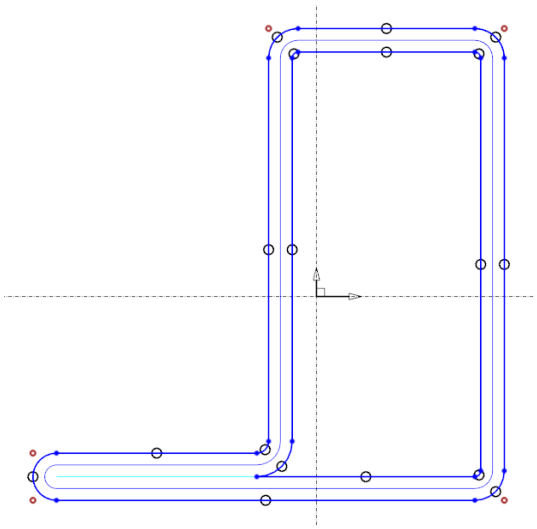


TopSolid automatically detects the contour.

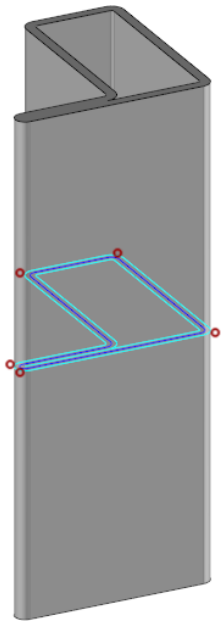
- Repeat the operation with a segment of the inner contour.



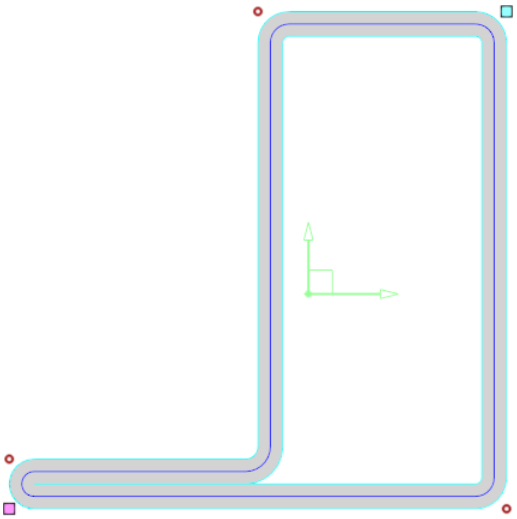
You should obtain the following result.



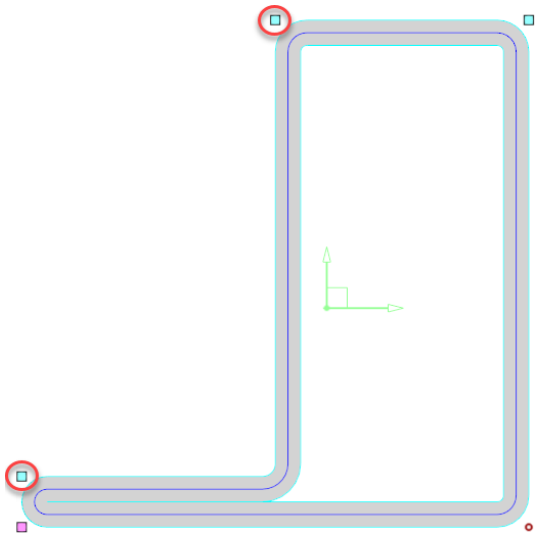
- Right-click in the graphics area with no active selection and select the **Extruded** command.
- In the **Length** field, enter *Length=100mm* and check the **Center** box.



- Select the **Tools** > **Functions** > **Provide Function** command and select the **Extruded Bar** function.
- Select the following points as the **top right point** and **bottom left point**.



- Click in the **Additional Key Points** table and add the following points.

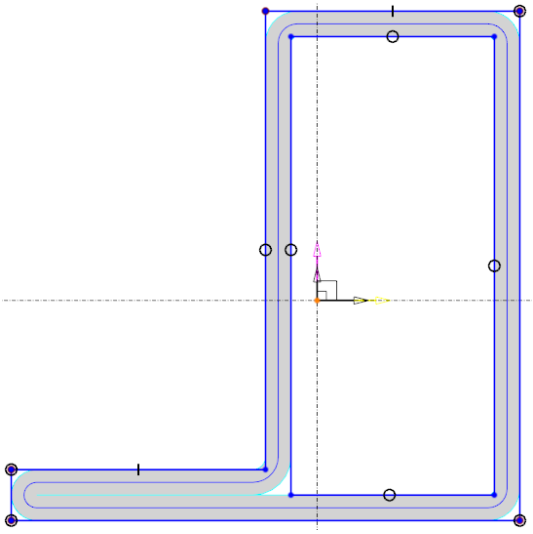


TopSolid asks you to enter a description for each point.

- Simply add a number.
- Select **Absolute Frame** as the **frame** and the **Length** parameter as the **length**.
- Click on to **confirm** the function.

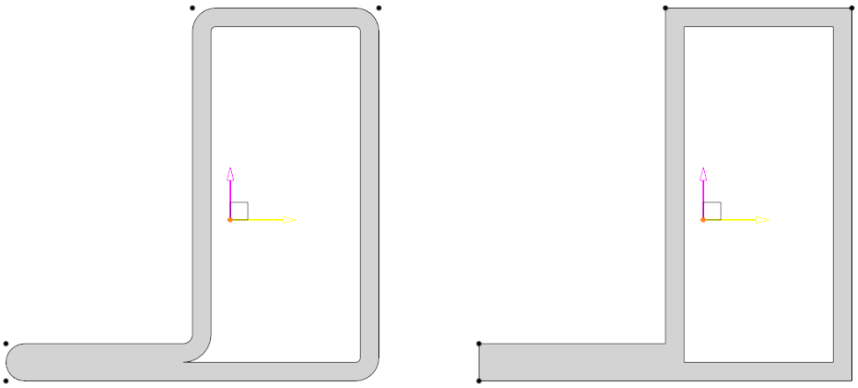
To make it easier to design with extruded bars, you can add a simplified representation.

- To do this, create the following sketch.



- Right-click in the graphics area with no active selection and select the **Extruded** command.
- In the **Length** field, enter *Length=100mm* and check the **Center** box.
- From the Entities tree's **Representations** folder, edit the **simplified representation**.
- Uncheck **Shape 1** and check **Shape 2**, then click on to **confirm** the operation.

From the **Tools** tab, you can then switch from one representation to another.

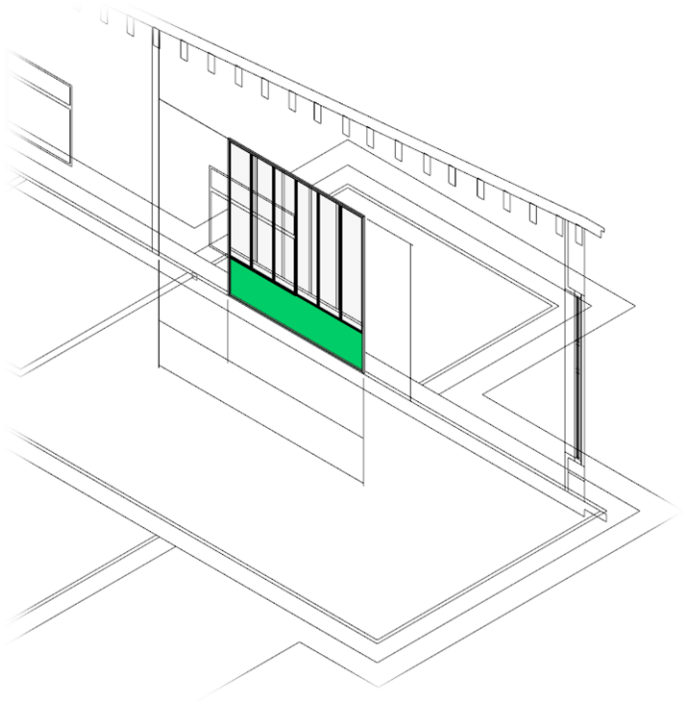


- **Save** the document (**Ctrl** + **S**).
- Right-click on the part document's tab and create a **Family** document.
- From the Entities tree, right-click on the **Length** parameter and select **Move to Drivers**.
- **Save** (**Ctrl** + **S**) and close the family document and its generic.

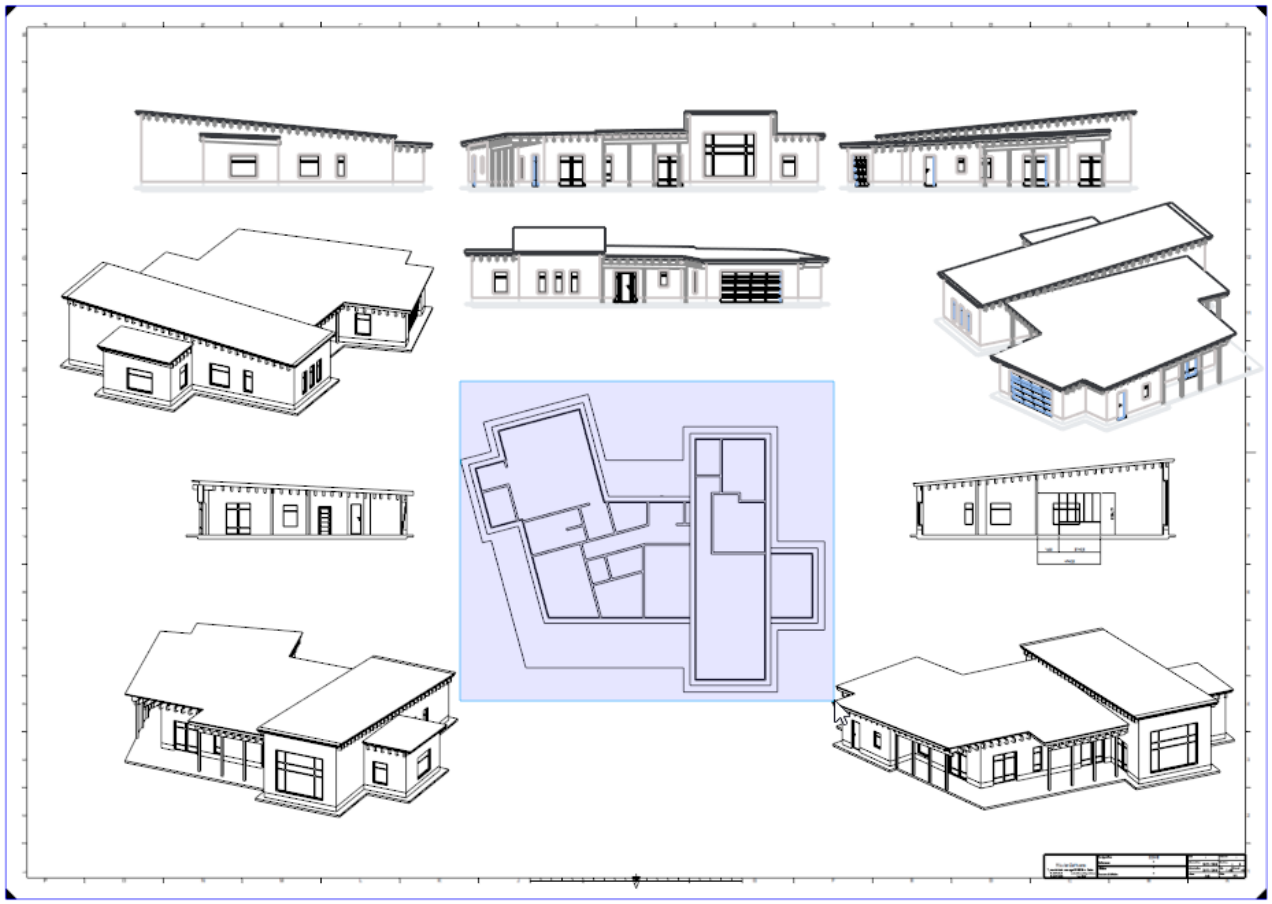
This extruded bar can be used as in the previous annex and as the other **TopSolid** standard extruded bars.

Annex 3 - Creating a 3D Environment from 2D Drawings

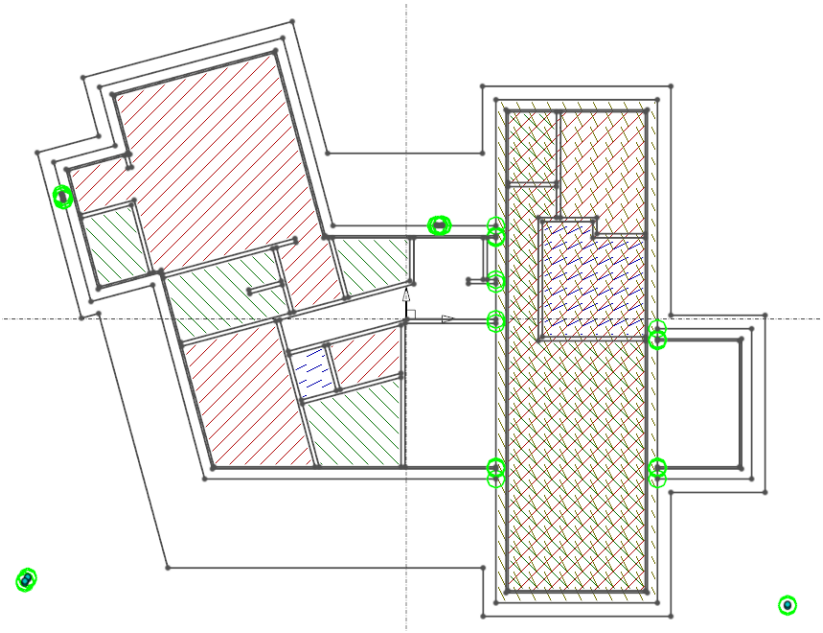
In the glass partition exercise (part 03), we saw how to work from a 2D DXF document. You can assemble several 2D views to create a 3D document.



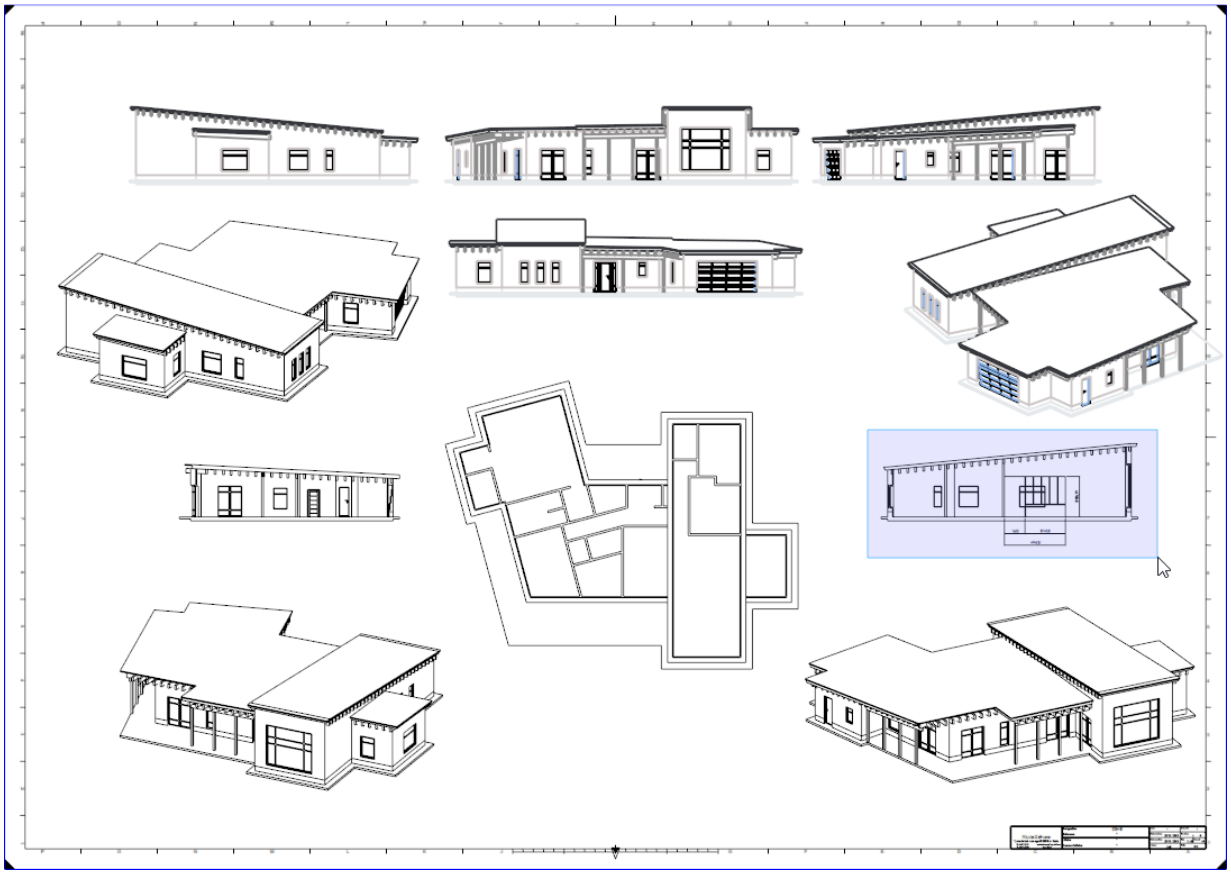
- Open the *Glass Partition Project Architect Drafting.A.0* document located in the *3- Glass partition* folder.
- Select the view shown below.



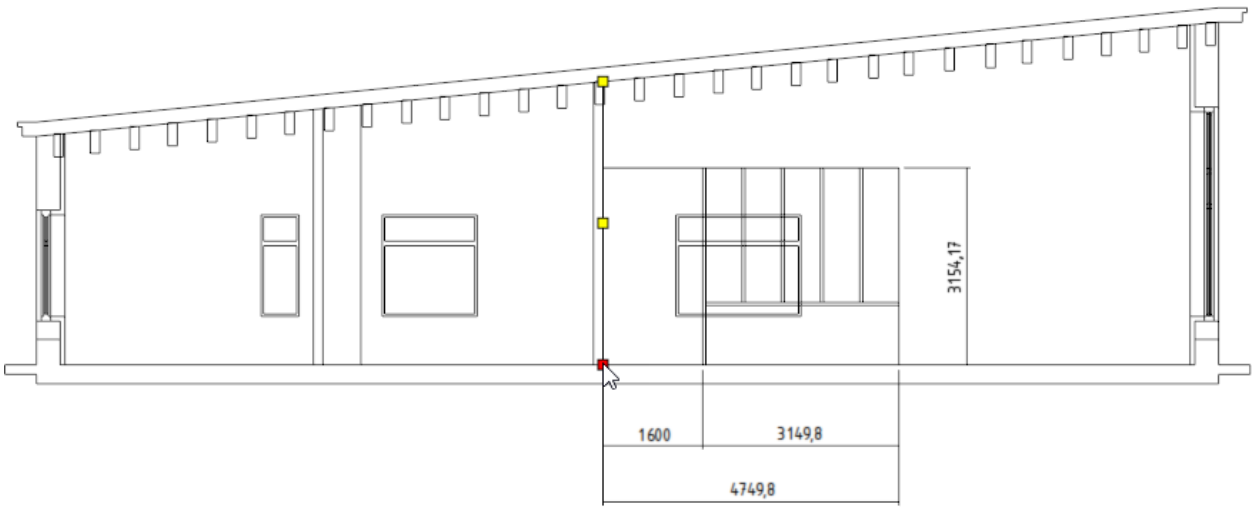
- Right-click in the graphics area and select the **Copy** command or use the **Ctrl + C** keyboard shortcut.
- Select the **Selection center** option and click on to **confirm** the operation.
- In the new assembly document, right-click in the graphics area and select the **Sketch** command.
- Right-click in the graphics area and select the **Paste** command or use the **Ctrl + V** shortcut.
- Select the **Sketch origin** option and make sure that you check the **Fix** box.
- Click on to **confirm** the operation.
- Delete any lines from the adjacent views (in this case, at the bottom left and right).



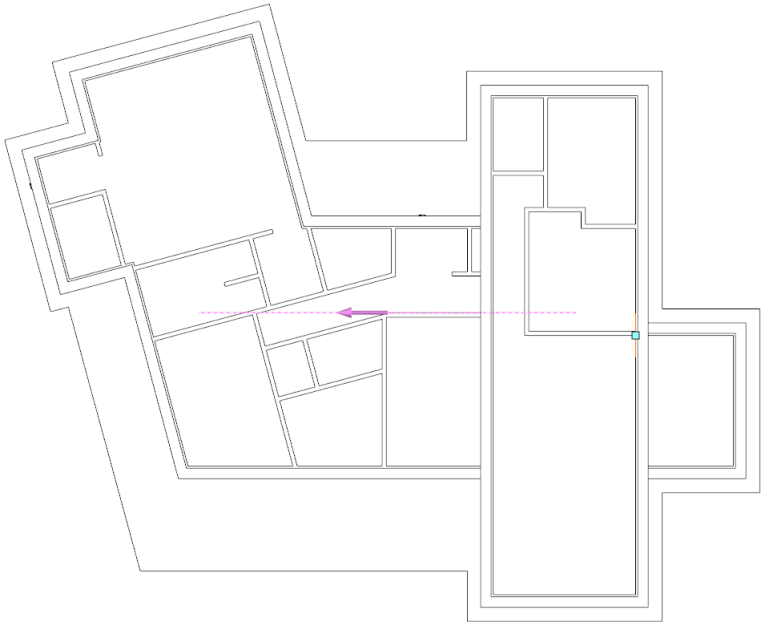
- **Confirm** the sketch.
- In the drawing document, select the section as shown below.



- Right-click in the graphics area and select the **Copy** command or use the **Ctrl + C** shortcut.
- Select the **Point** option and select the point as shown below.



- Click on to **confirm** the operation.
- In the assembly document, select the **Construction** > **Plane by point and normal** command.
- Select the point as shown below and the **-X axis** as the **normal**.



- Right-click on the previously created plane and select the **Sketch** command.
- Right-click in the graphics area and select the **Paste** command or use the **Ctrl + V** shortcut.
- Select the **Sketch origin** option, make sure that you check the **Fix** box, and then click on to **confirm**.

The imported drawing has a scale factor. In order to build on this drawing, you need to reset the dimensions of the sketches.

- Select the **Construction** > **Transform** command.
- Select the two sketches and the plane as the **entities to transform**.
- In the **Transform** field, click on the icon and select the **Scaling Transform** command.
- In the Center field, select **Absolute Origin Point** and enter a **factor** of *80*.
- Click on to **confirm** the transformation type, and then the transformation operation.
- **Save** the document (**Ctrl** + **S**).

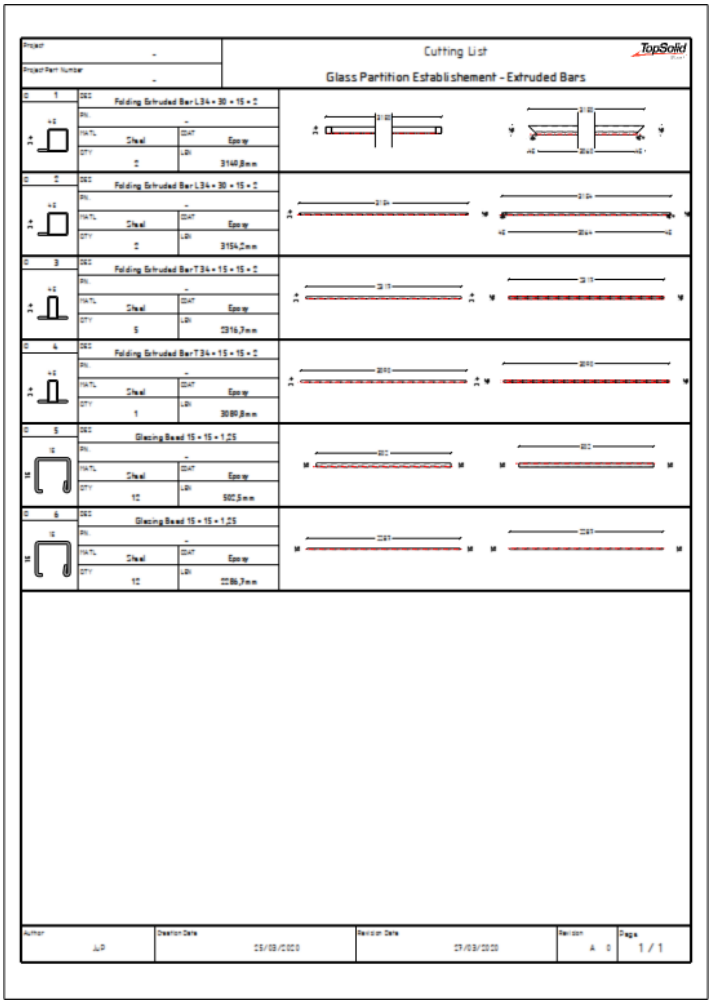
Annex 4 - Creating a Cutting List of Extruded Bars

TopSolid allows you to create single drawings as well as assemblies of several drawings thanks to the **Drafting Bundle** document.

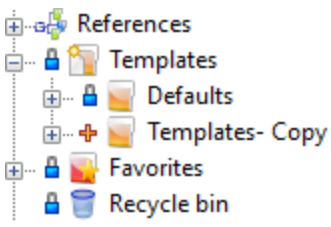
There are many benefits:

- grouping of several drawings to facilitate printing;
- creation of technical files;
- creation of A0 drawings composed of drawings of different formats (A4, A5, A3, etc.);
- creation of cutting files;
- creation of labels.

The purpose of this exercise is to obtain the following result.



- From the Project tree, open the *3- Glass partition > Glass partition files* folders and copy the *Templates (annexes)* folder (the **Ctrl + C** keyboard shortcut or selecting the **Copy** command via the contextual menu).
- From the Project tree, paste the folder into the **Templates** folder (the **Ctrl + V** keyboard shortcut or selecting the **Paste** command via the contextual menu).



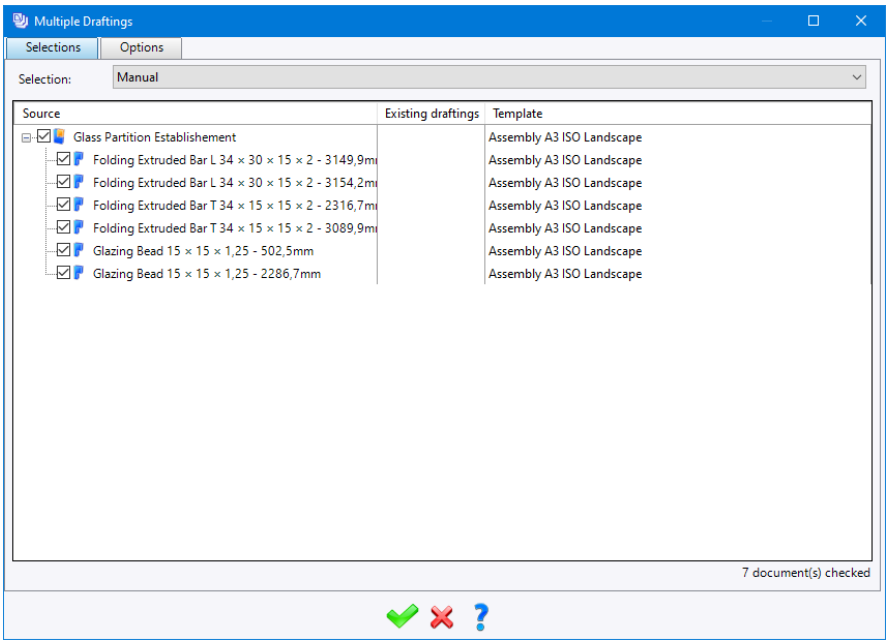
- Rename the folder *Templates - Training Technical File*.

In the basic training, we saw that it is better to store the document templates in the **My Templates** or **Company Templates** folders in the case of a server installation. Indeed, it is the recommended location for templates that concern the company.

Nevertheless, you will sometimes have to create specific document templates for a case (custom title block, layout imposed by the customer, etc.). These templates will have to be stored in the project, which has just been done.

- Right-click on the *Glass partition establishment* assembly document's tab and select the **Bill of Material** command.
- Select the **Extruded Bar** template from the **Steel Standard Templates - United States** folder, then click on to **confirm** the operation.
- In the **Assembly** dialog box, make sure that the **Groups by properties** box is checked and click on to **confirm** the operation.
- Rename the bill of materials *Glass partition establishment - Extruded bars*.
- Select the **Multiple Draftings** command.

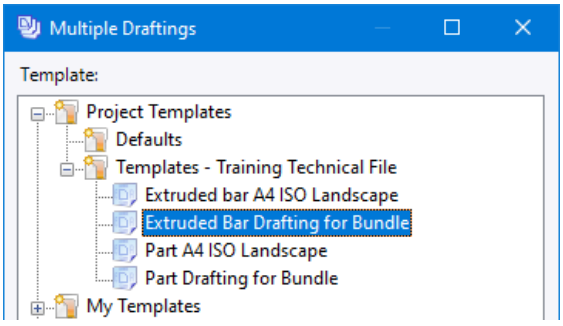
The following dialog box appears, listing all the lines of the bill of materials. **TopSolid** detects if a drawing already exists or if you have to create it.



- Uncheck **Glass partition establishment**.
- Double-click on the template of the first part.

TopSolid invites you to select the drafting template to be used for this part.

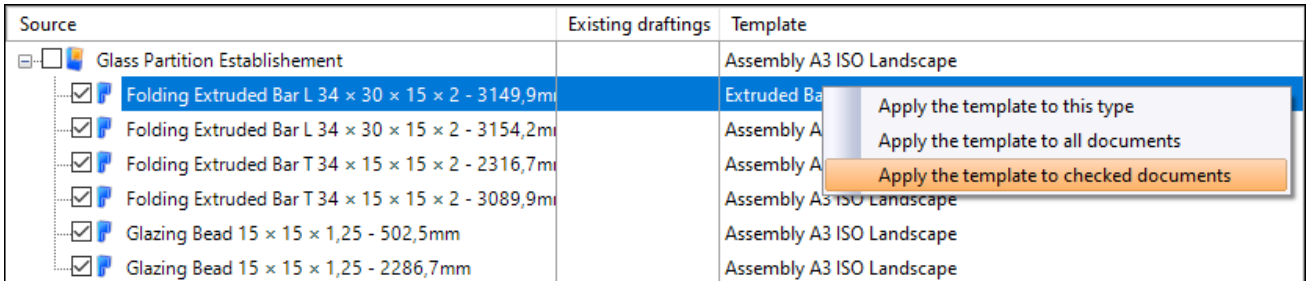
- Select **Project Templates** > **Templates - Training Technical File** > **Extruded Bar Drafting for Bundle**.



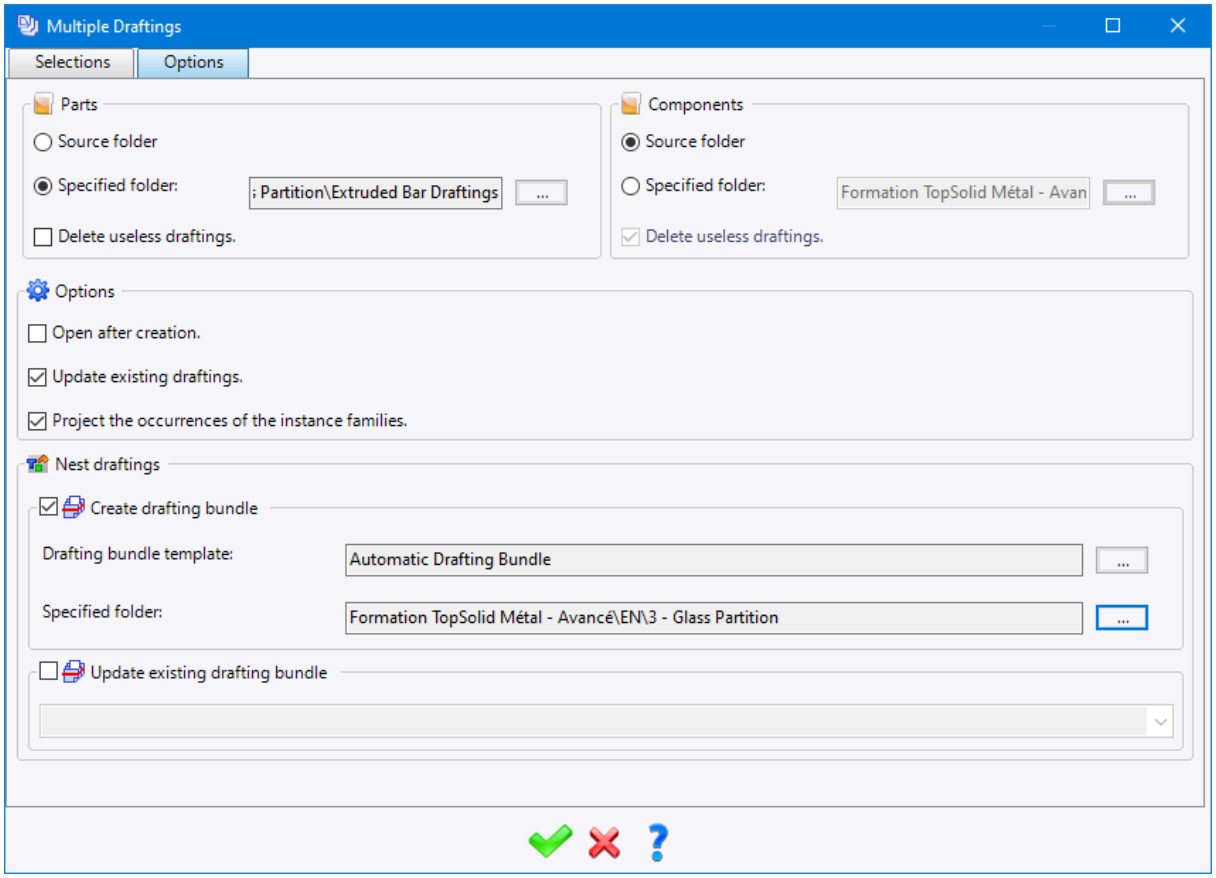
- Click on to **confirm** the operation.

You now have to apply this template to all parts.

- Right-click on the template on the first line and select **Apply the template to checked documents**.



- Click on the **Options** tab.
- In the **Parts** section, select the **Specified folder** option and click on the ... button.
- In the *3- Glass partition* folder, create a new folder named *Extruded bar draftings* and click on to **confirm** the operation.
- In the **Nest draftings** section, check the **Create drafting bundle** box and click on the ... button to the right of **Drafting bundle template**.
- Select **Project Templates** > **Templates - Training Technical File** > **Automatic Drafting Bundle** and click on to **confirm** the operation.



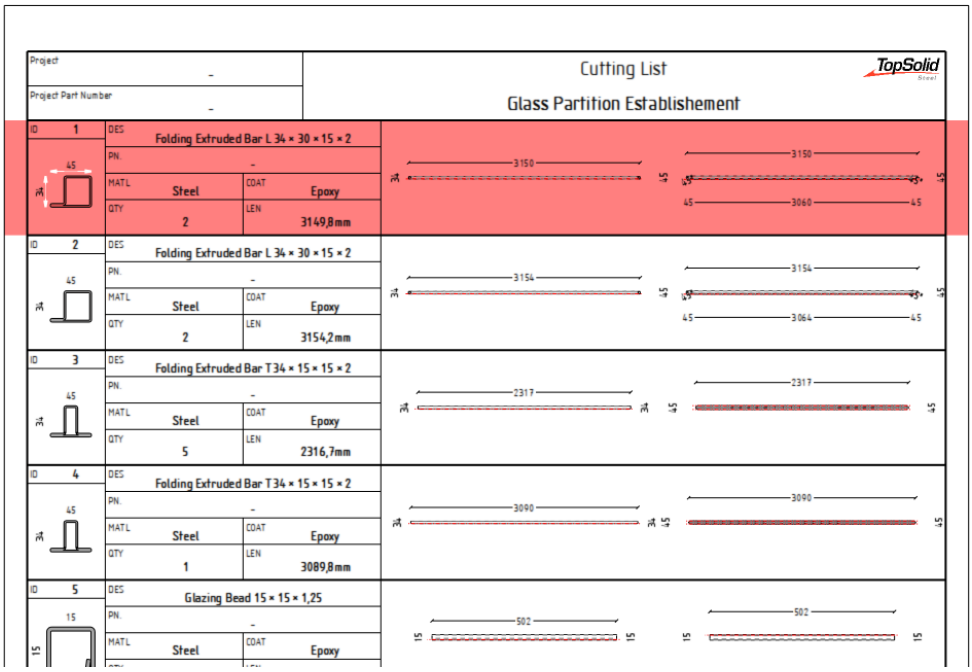
As you can see with the different options, this command also allows you to update the bundle by adding, deleting or updating the different drawings.

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

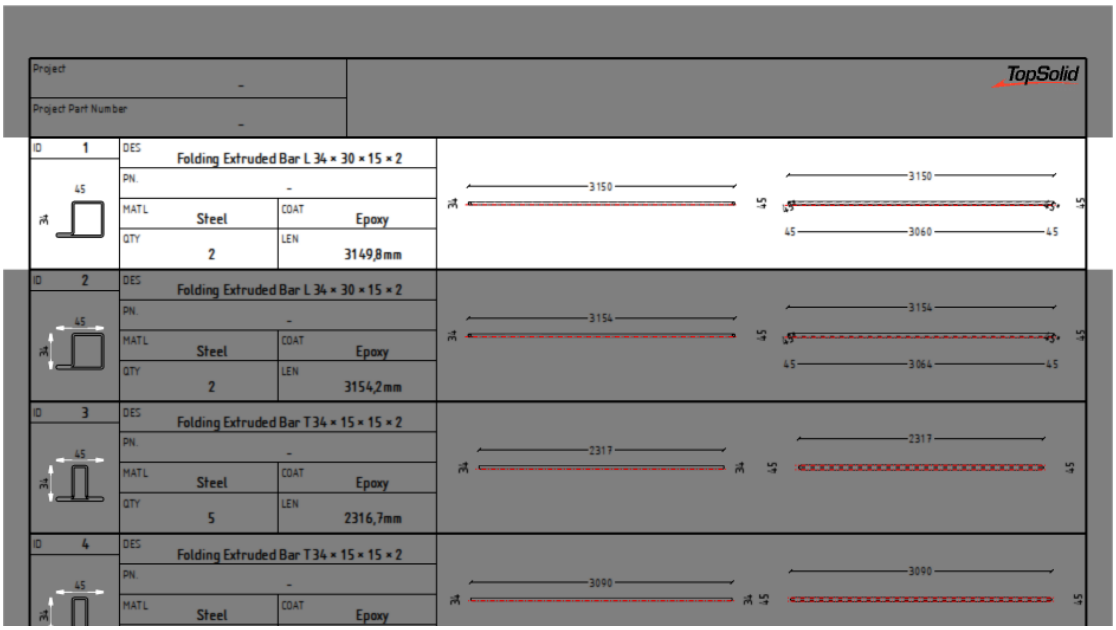
- Open the *Glass partition establishment - Extruded bars* drafting bundle from the *3- Glass partition* folder.

A drafting bundle can be compared to a classic assembly in which each part would be a drawing. As with an assembly, you can therefore edit a drawing in place by double-clicking on it or open it by right-clicking on it and selecting the **Open Document** command.

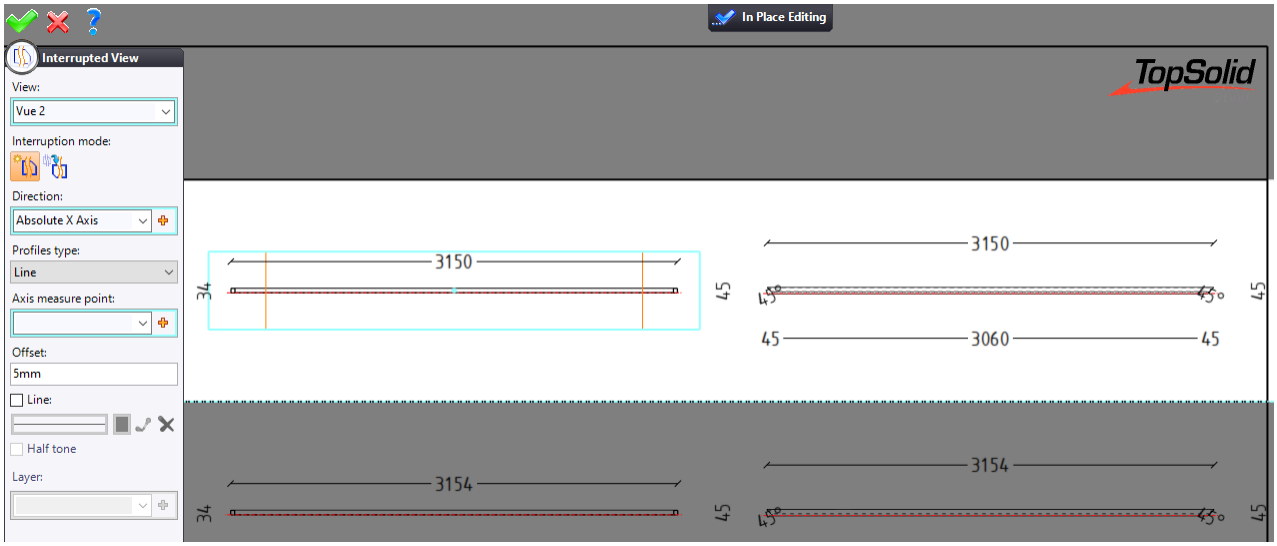
- Double-click on the first drawing from the top to edit it in place.



The drawing switches to the in-place editing mode.

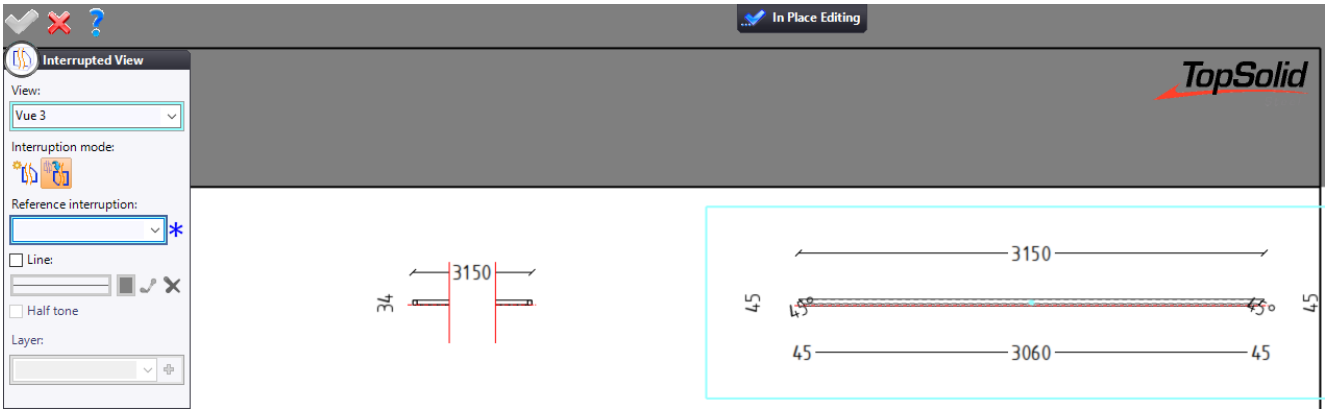


- Select the **Interrupted View** command.
- In the **View** field, select the left view.
- In the **Profiles type** field, enter *Line*.
- Position the lines as shown below.



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

- Select the command again and repeat the operation for the right view.
- Select the **Interrupt like** mode using the left view as the reference.

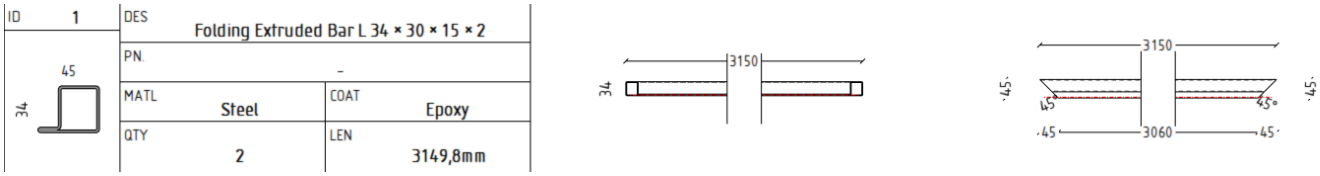


- Click on to **confirm** the operation.

You can optimize the scaling factor according to the view layouts defined in the document.

- Select the **Optimize View Layout** command.

You should obtain the following result.



- **Confirm** the in-place editing to return to the bundle document.
- **Save** the document (**Ctrl** + **S**) and close it.

Annex 5 - Creating and Exporting Sheet Metal Unfoldings (Multiple Unfoldings)

TopSolid allows you to generate multiple unfoldings from a simple bill of materials. These unfoldings can be exported very easily in DXF/DWG format. This is what we will see in this exercise.

- Right-click on the *Glass partition establishment* assembly document's tab and select the **Bill of Material** command.
- Select the **Sheet Metal** template from the **Steel Standard Templates - United States** folder and click on to **confirm** the operation.
- In the **Assembly** dialog box, make sure that the **Groups by properties** box is checked and click on to **confirm** the operation.
- Rename the bill of materials *Glass partition establishment - Sheetmetal*.
- Select the **Multiple Unfoldings** command.

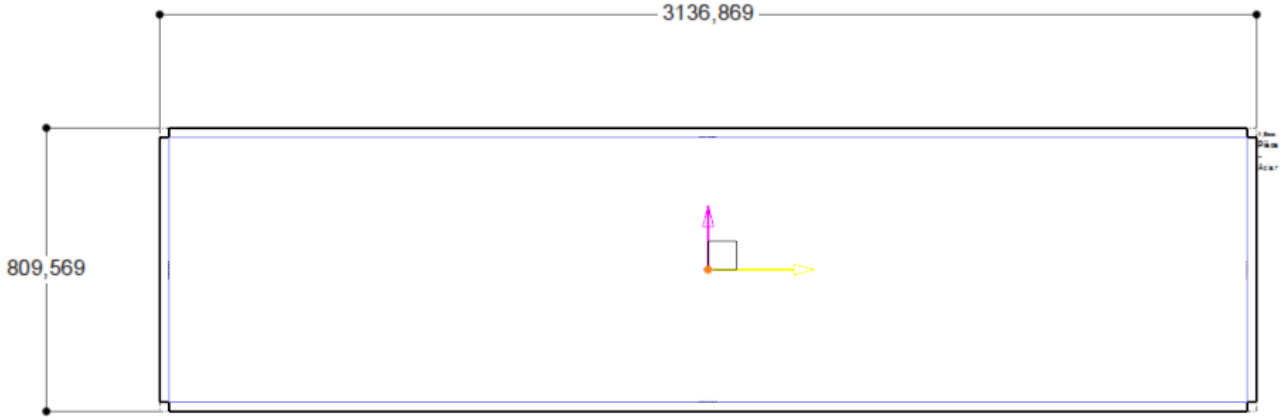
The following dialog box appears.

Source	Thickness	Material	Existing Unfoldings	Template
Glass Partition Establishement				
Part	1,5mm	Steel		Unfolding

The bill of materials already contained only sheet metal parts. If it had not been the case, the command would have proposed only the sheet metal parts to create the unfoldings.

As with the multiple drafting operation, **TopSolid** checks if an unfolding does not already exist and offers a selection of document templates to be used.

- In the **Destination** section, select the **Specified folder** option and click on the ... button.
- In the *3- Glass partition* folder, create a new folder named *Unfoldings* and click on to **confirm** the operation.
- In the **Options** section, make sure that the **Open after creation** option is checked, then click on to **confirm** the operation.

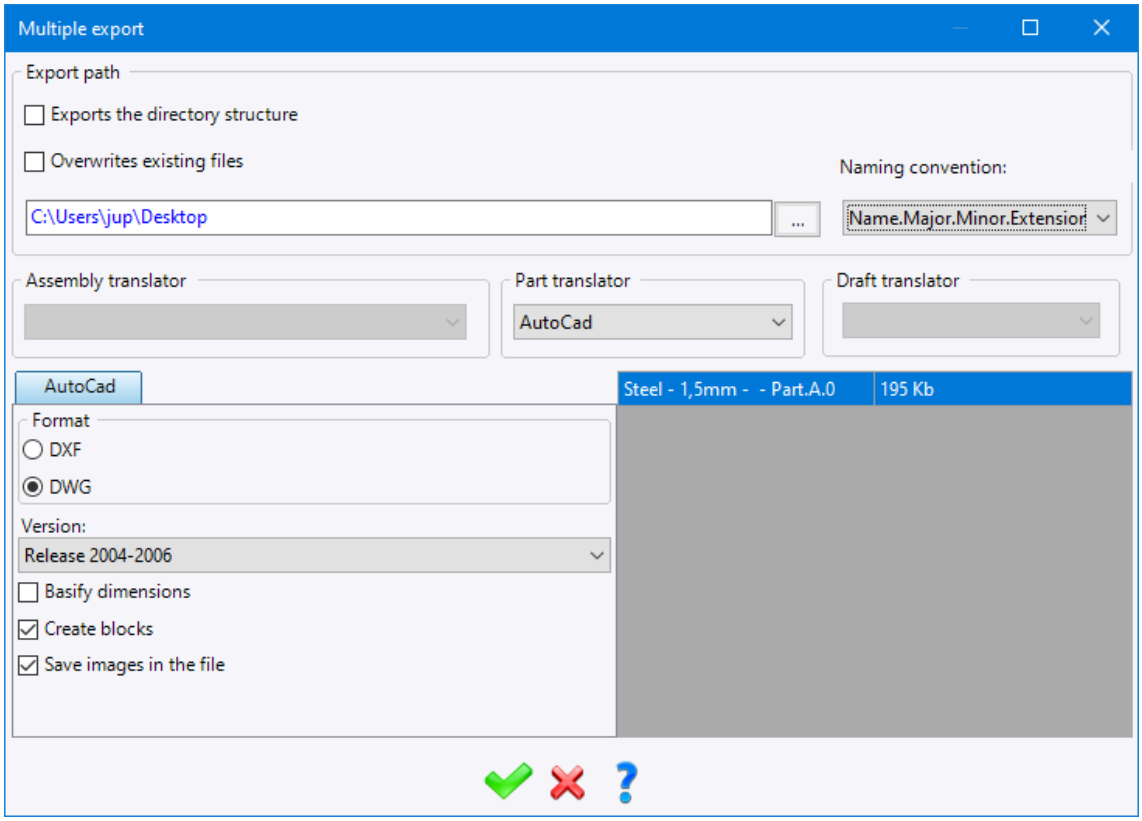


TopSolid generates as many unfolding documents as there are sheet metal parts.

You can export the complete batch in the desired format (DXF/DWG for example).

- To do this, from the Project tree, open the *3- Glass partition* > *Unfoldings* folders, then right-click and select the **Import/Export** > **Export Several Documents with Conversion** command.

- Select the desktop as the destination, select **Name.Major.Minor.Extension** as the **naming convention**, select **AutoCad** as the **part translator**, and select **DWG** as the **format**.



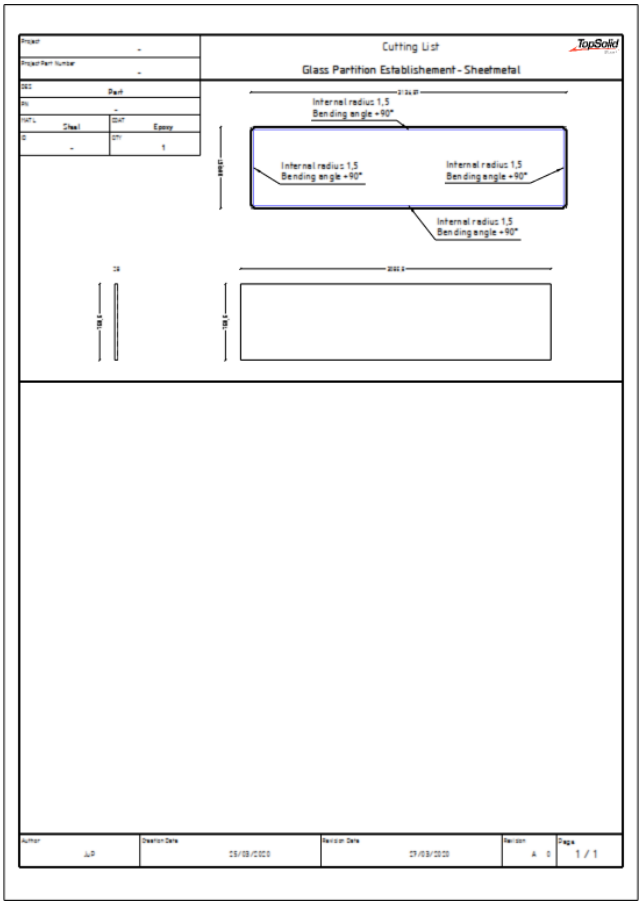
- Click on  to **confirm** the operation.

The files are created.

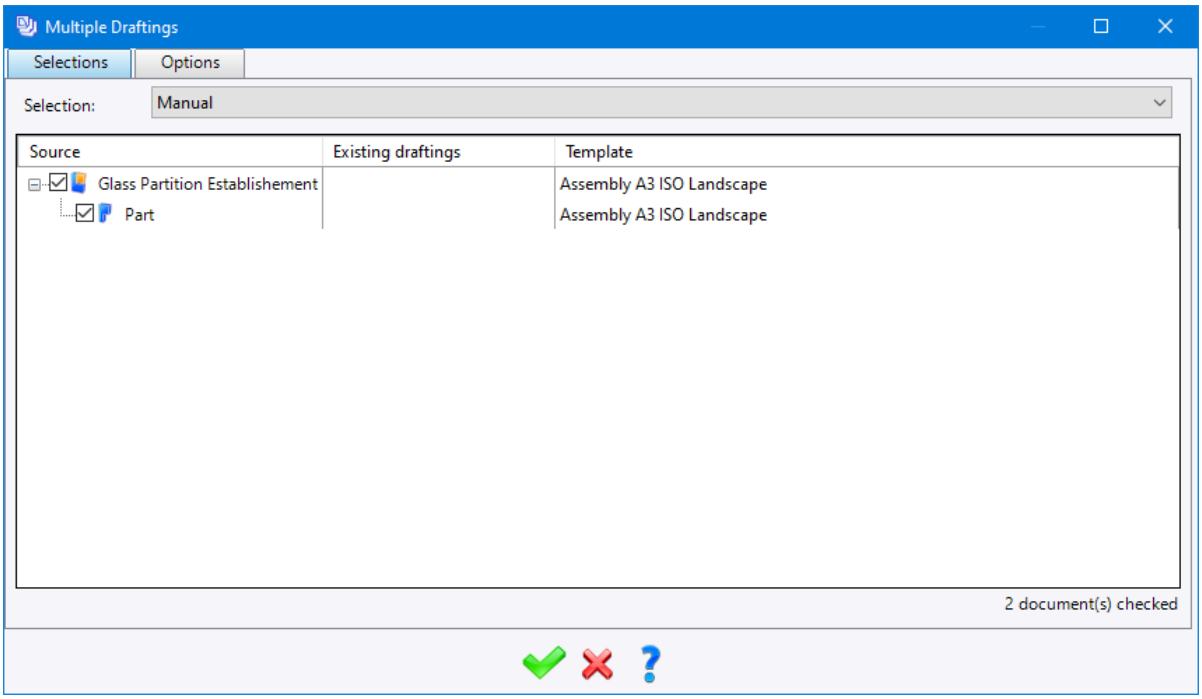
- Click on  to **confirm** the operation a second time to exit the dialog box.

Annex 06 - Creating a Cutting List of Sheet Metal Parts

The objective is now to produce the following drafting bundle.

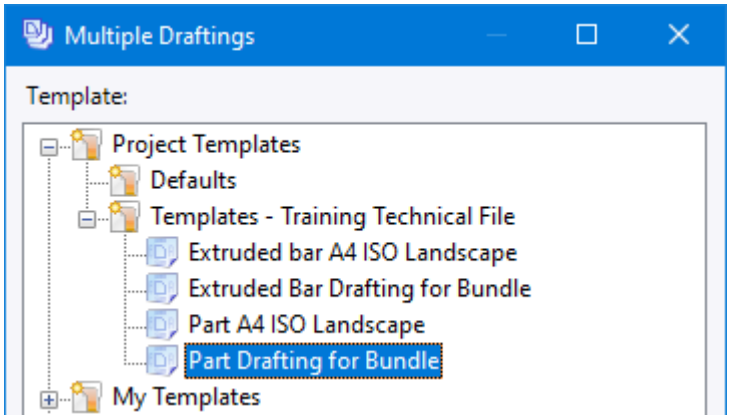


- Right-click on the *Glass partition establishment - Sheetmetal* BOM document's tab and select the **Multiple Draftings** command. The following dialog box appears.



- Uncheck **Glass Partition Establishment**.

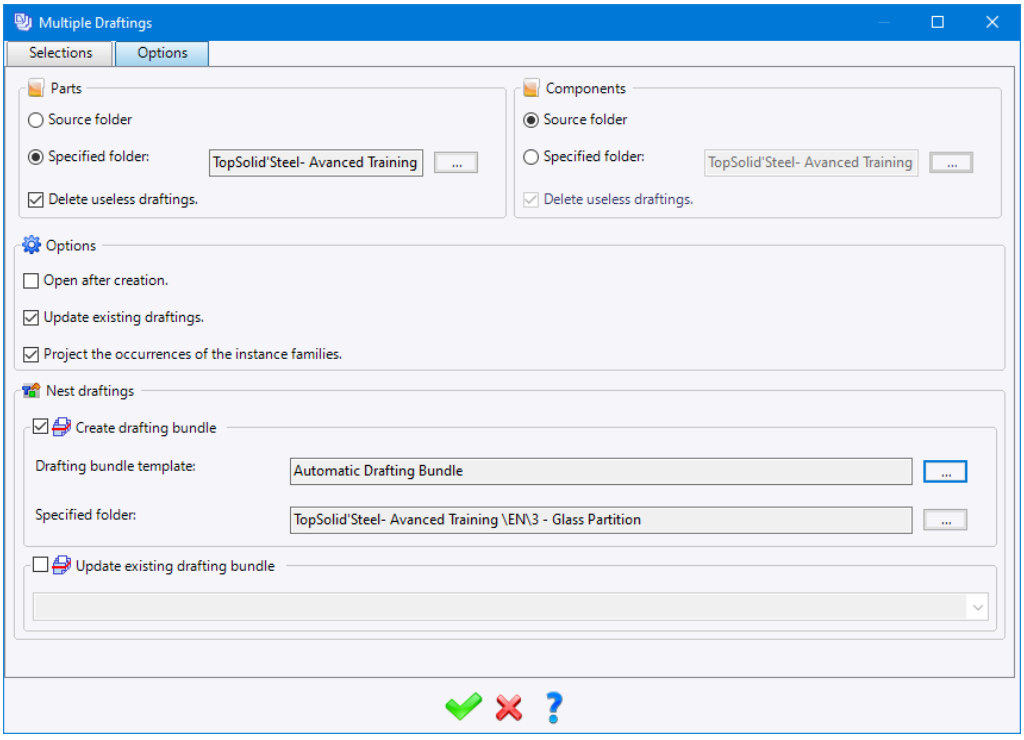
- Double-click on the template of the first part. **TopSolid** invites you to select the drafting template to be used for this part.
- Select **Project Templates** > **Templates - Training Technical File** > **Part Drafting for Bundle**.



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

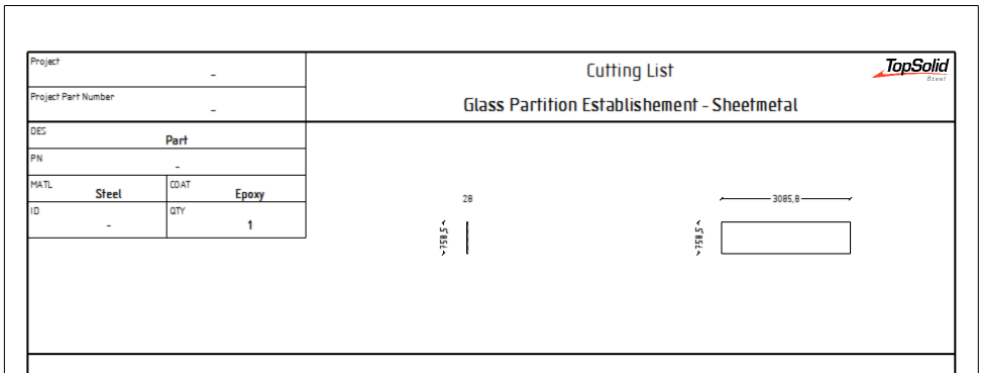
Since the bill of materials contains only one part, you do not have to apply this template to all the parts.

- Click on the **Options** tab.
- In the **Parts** section, select the **Specified folder** option and click on the button.
- In the *3- Glass partition* folder, create a new folder named *Sheetmetal draftings* and click on to **confirm**.
- In the **Nest draftings** section, check the **Create drafting bundle** box and click on the button to the right of **Drafting bundle template**.
- Select **Project Templates** > **Templates - Training Technical File** > **Automatic Drafting Bundle** and click on to **confirm**.

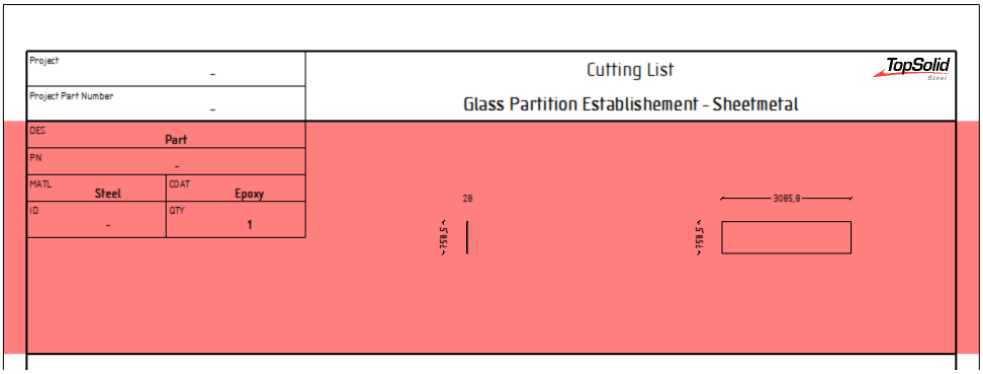


- Click on to **confirm** the operation.

- Open the *Glass partition establishment - Sheetmetal* drafting bundle from the *3- Glass partition* folder.

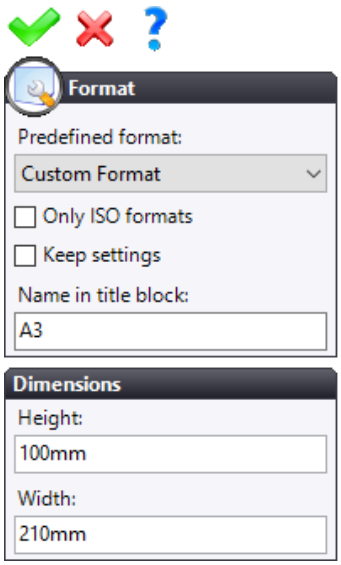


- Double-click on the drawing.



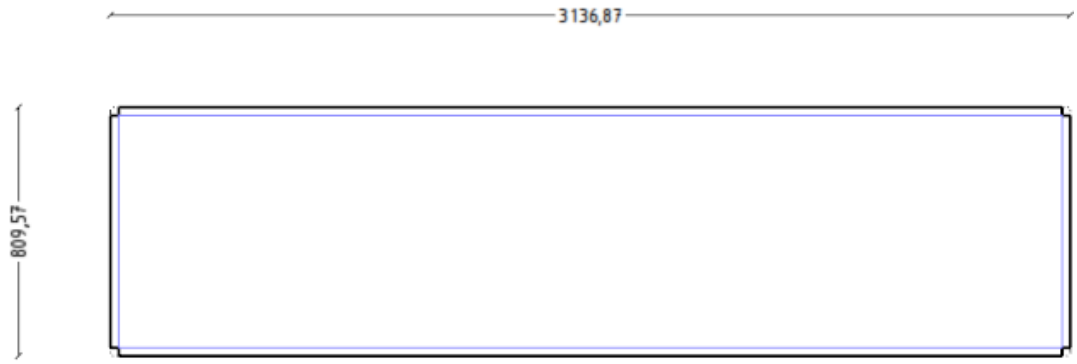
A drawing contained in a bundle can be resized at any time. The whole document will be automatically recalculated.

- Right-click in the graphics area with no active selection and select the **Format** command.
- In the **Dimensions** section, adjust the **height** to *100mm* and click on  to **confirm** the operation.

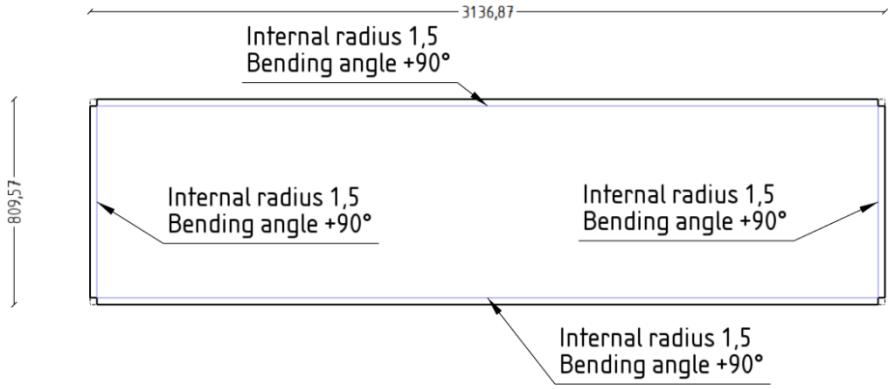


- **Confirm** the in-place editing. The positioning is recalculated.
- Edit the drawing in place again.
- Right-click in the graphics area with no active selection and select the  **Scale** command.
- Adjust the scale to *1/30*.

- From the Project tree, drag and drop the unfolding document into the drafting document.
- Select the **Automatic Dimensions** command on the unfolding to display its enclosing dimensions.



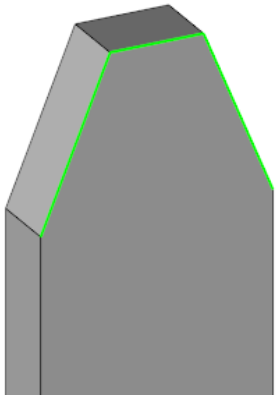
- Select the **Automatic Bend Notes** command and position the notes as shown below.



- **Confirm** the in-place editing to return to the bundle document.
- **Save** the document (**Ctrl** + **S**) and close it.

Annex 7 - Creating a 2D Symbol of the Post Cutting

The purpose of this exercise is to draw a 2D parametric component to cut the railing posts.

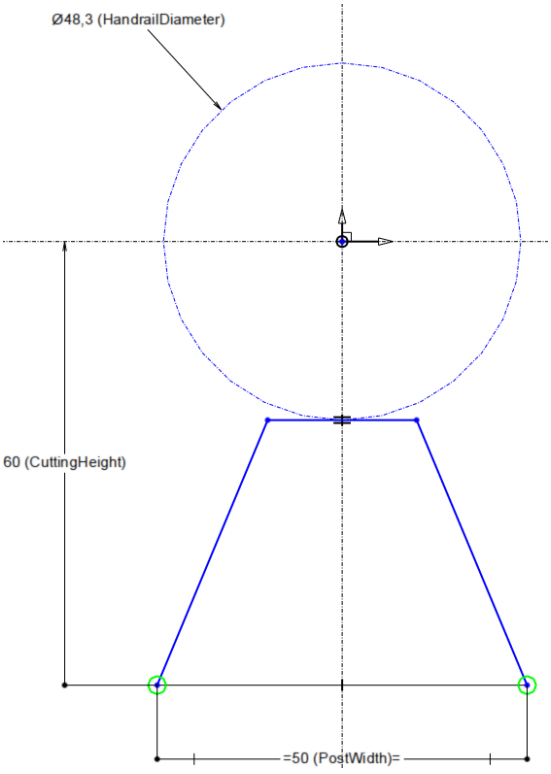


Concepts addressed:

- Creating a 2D parametric symbol
- Creating a family
- Using a 2D symbol for design

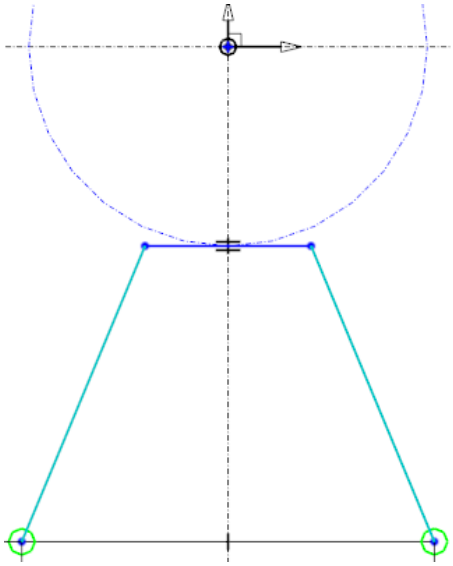
Creating a 2D parametric symbol

- From the Project tree, create a new folder named *6- 2D cutting symbol*.
- Right-click on this folder and select the **Document** command.
- From the **Special** tab, select **2D Modeling**, then select the **2D Symbol** template from the **Standard Templates - United States** > **Standard Components** folder.
- Rename the document *Cutting symbol for post*.
- Right-click in the graphics area with no active selection and select the **Sketch** command.
- Draw the following sketch.



Here are some additional guidelines to create the sketch:

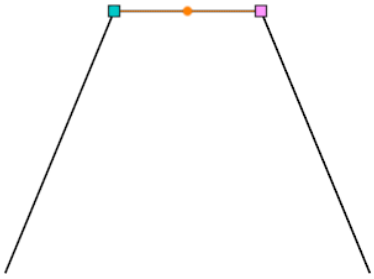
- The circle must be in construction mode. To do this, right-click on the circle and select the **Construction** command.
- The *PostWidth* dimension is centered on the Y axis.
- The two segments below coincide with the center of the circle.



- **Confirm** the sketch.

By default, the symbol is positioned via the absolute frame's point. You can add other positioning points.

- Select the **Construction** > **Midpoint** command to create the following point.



- Right-click on the previously created point and select the **Others** > **Publish Point** command.
- In the **Name** and **Description** fields, enter *Cutting height point*, then click on to **confirm** the operation.
- From the Entities tree's **Publishings** folder, edit the **Center (Center)** publishing.
- In the **Name** and **Description** fields, enter *Handrail center*, then click on to **confirm** the operation.

Note: To modify the default positioning point, you simply have to move the publishing up or down. The lowest publishing is the one by default (in this case, **Handrail center**).

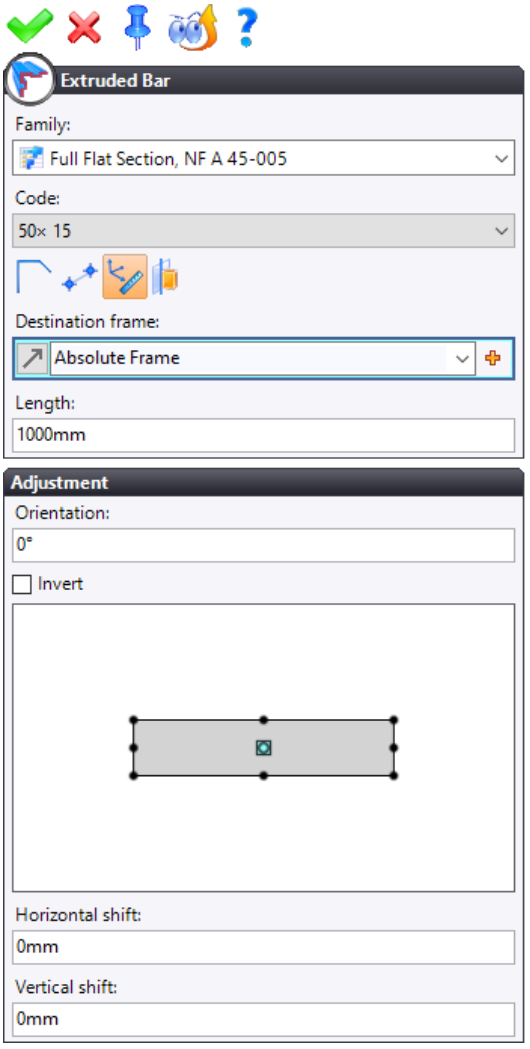
- Select the > **File** > **Virtual Document** command.
- Check the **Virtual Document** box, then click on to **confirm** the operation.

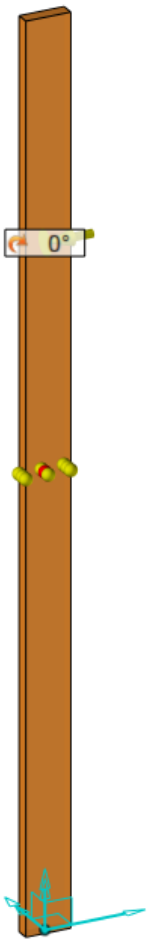
This option allows you to hide the document in the drop-down menus and searches. Indeed, the user should only see the family and not the family and the generic.

- Right-click on the *Cutting symbol for post* document's tab and create a **Family** document.
- From the Entities tree's **Generics** folder, hold down the **Ctrl** key and select the **CuttingHeight**, **PostWidth** and **HandrailDiameter** parameters, then right-click and select **Move to Drivers**.
- **Save** (**Ctrl** + **S**) and close the family document, as well as the 2D modeling document.

Using the 2D parametric symbol

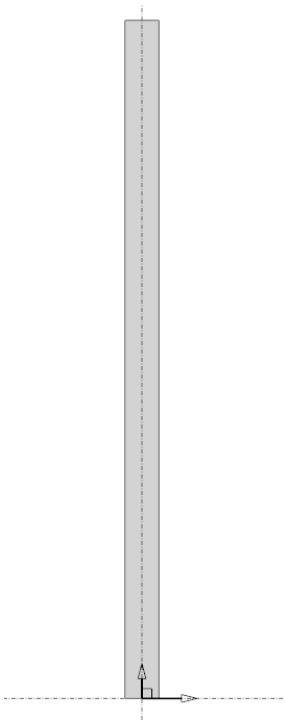
- From the Project tree, right-click on the *6- 2D cutting symbol* folder and create an **Assembly** document.
- Select the **Modeling** > **Extruded Bar** command.
- Select the **Full Flat Section, NF A 45-005** family and the **50 x 15** code.
- Select the **Frame and Length** mode.
- In the **Destination frame** field, select **Absolute Frame**. In the **Length** field, enter *1000mm*.



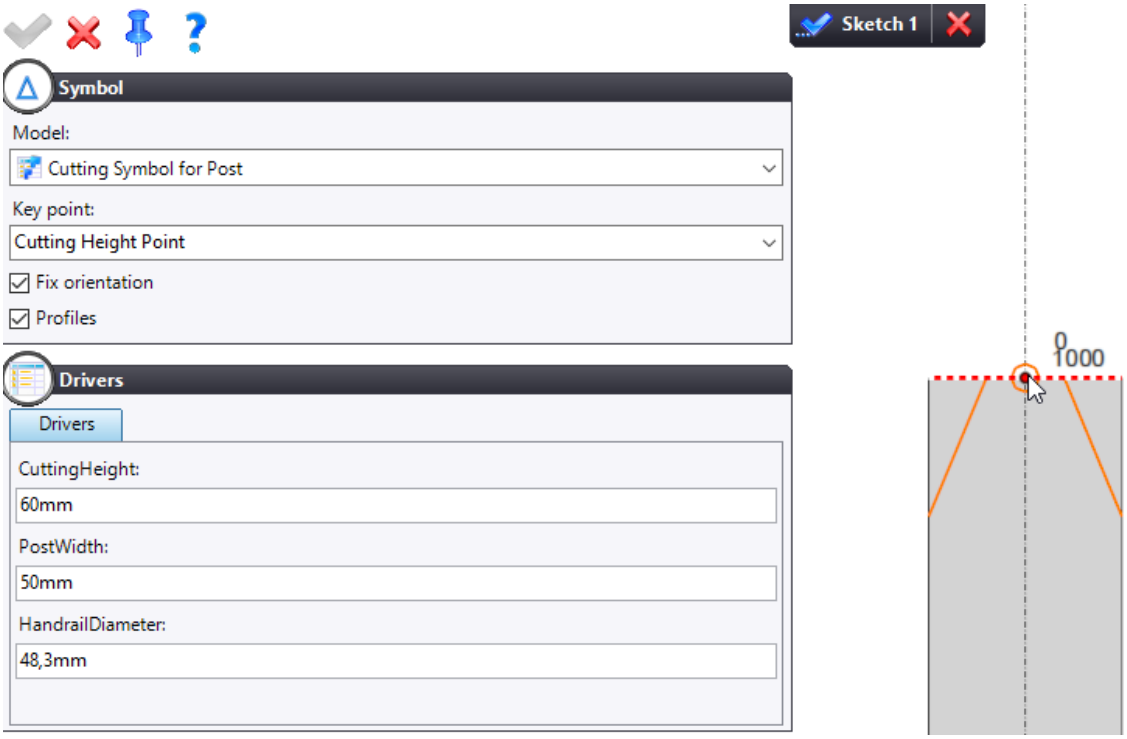


- Click on to **confirm** the operation.

- Create a sketch on one of the following faces.

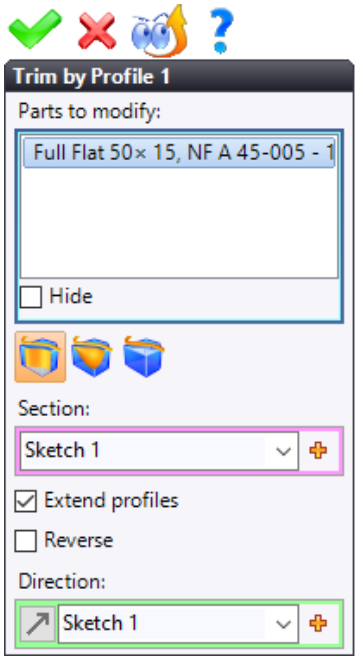


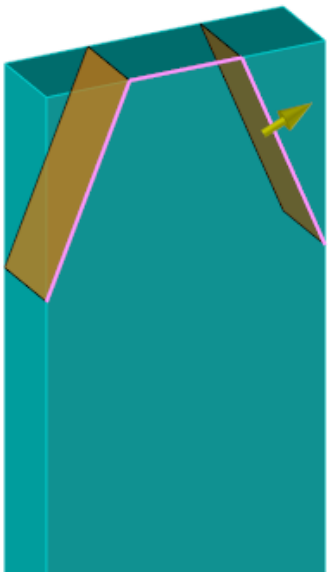
- Select the **Symbol** command.
- Select **Cutting symbol for post** as the **model** and select **Cutting height point** as the **key point**.
- Make sure that the **Fix orientation** and **Profiles** boxes are checked.
- Position the symbol as shown below.



- **Confirm** the sketch.

- Right-click on the sketch and select the **Trim by Profile** command.





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

At any time, you can change the values of the symbol or even replace it with another one.
Symbols can also be used in a drafting document. They can either be scaled to the projection set's scale, or to the paper's scale.

Notes

Individual Course Evaluation Form

(To be completed and returned to the training instructor at the end of the course)

TopSolid'Steel - Advanced

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: .....