TopSolid Wood

Training Guide

TopSolid'Wood Advanced

MASTER YOUR MANUFACTURING PROCESS

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Contents

Making a guitar body

The purpose of this first section is to make the body of a guitar in order to review some of the functions addressed during the first training cycle.

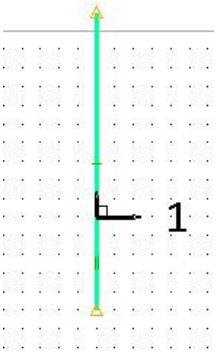
Concepts addressed:

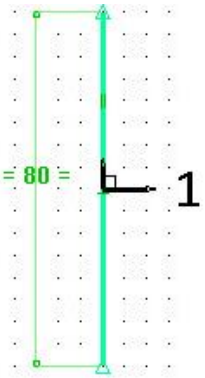
- **Sketch**: Line, arc, sketch copy, dimension and constraint
- **Extrusion**
- **Wood machinings**: Pockets, drillings and mouldings
- **Data structure**: Part and set definition

Creating the body

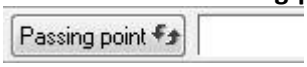
Start the sketch

- Create a **new document** of the  **Design** type.
  In the **Advanced parameters**, select **Without template**. Select the **Associative** design mode and then **Millimeters**.
- Click on **OK** to confirm.
- Activate the **Sketch** context  , then start a **new sketch**. 
- Draw a first **line 1** as shown opposite. 
- Apply a **coincidence constraint** of this line to the **Y axis**.  > 
- **Dimension** this line with a **nominal value** of 80mm. 
- After applying the value, activate the **Symmetry constraint** option in the dialog bar, then select **X** as the **symmetry axis.**

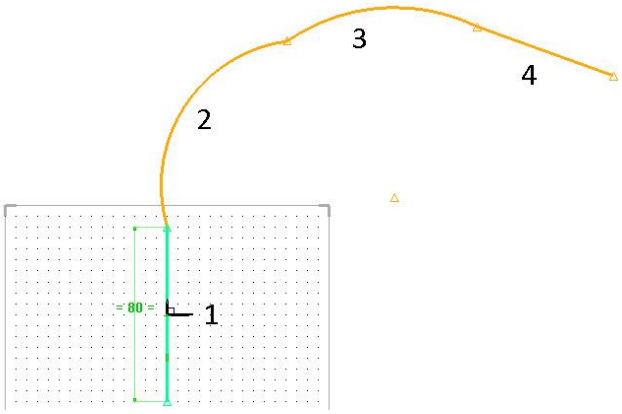




Note: Applying a **symmetry constraint** to a dimension allows the dimension to be automatically centered on the selected axis, regardless of its length.

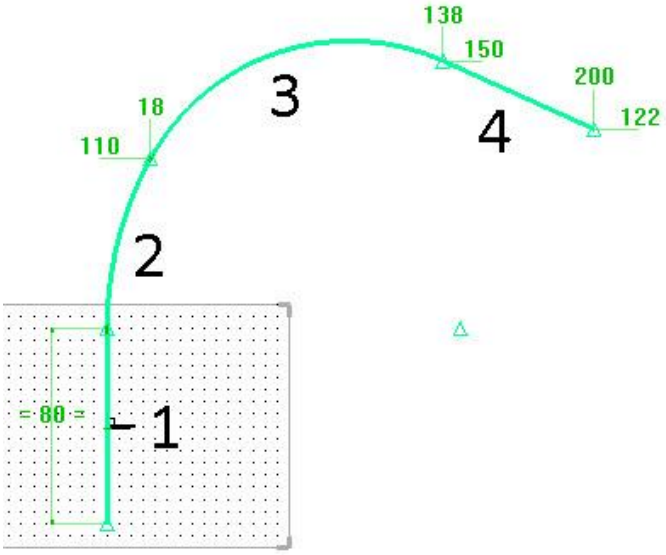
- Start the **Circle** function  in **Passing point** mode to draw the two circle arcs 2 and 3 as shown below. 

- Draw the line **4** after the circle arc **3** as shown below.

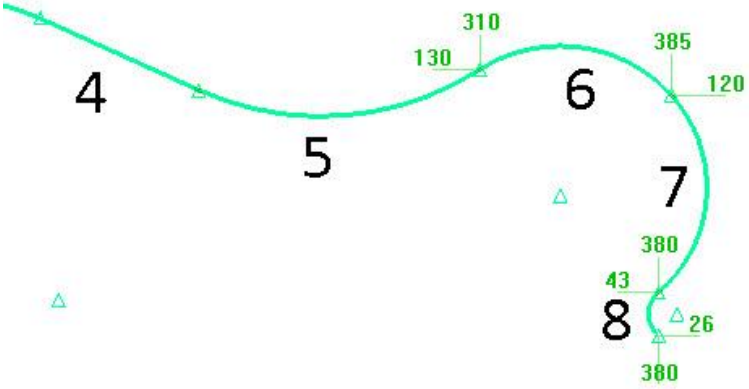


- **Dimension** the points between **2** and **3**, as well as the two points of line **4** with the values shown opposite.
- Apply a **tangency constraint** between **1** and **2** and between **2** and **3**.

Note: All the sketch elements are green, which means that the sketch is totally constrained.



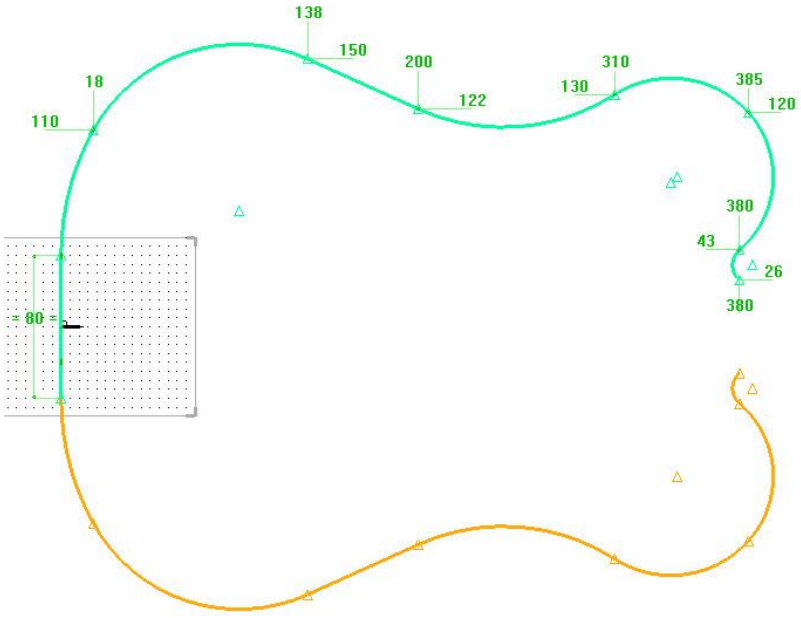
- Draw 4 **circle arcs** (5, 6, 7 and 8) in **Passing point** mode as shown below.
- Then **dimension** the different points with the given values.
- Apply **tangency constraints** between the arcs **4** and **5**, **5** and **6**, **6** and **7** and **7** and **8**.



Copy the sketch

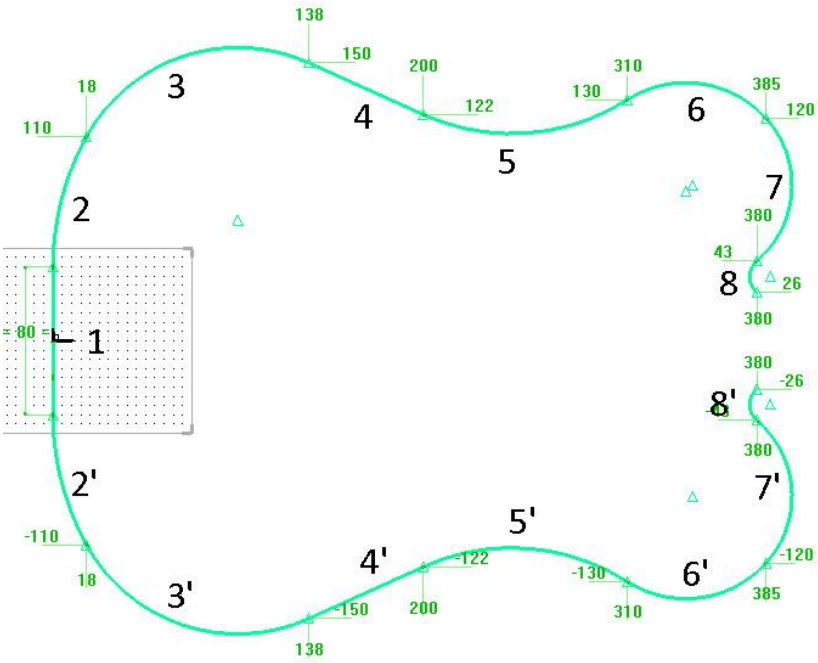
- Start the **Copy** function. 
- Using the **selection** (lasso)  , select segments **2**, **3**, **4**, **5**, **6**, **7** and **8**.

- Click on **OK** to confirm.
- In **Propagation**, select **Simple mirror** in the dropdown list.

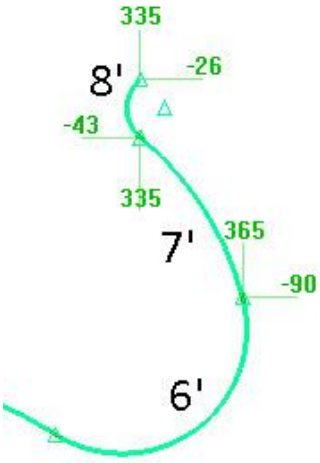


- Select the **ZX** plane as the **symmetry plane** in the dialog bar. 

- **Dimension** the different points of the copied elements.
  Do not change the values of these dimensions.
- Then apply a **tangency constraint** between **1** and **2'**.

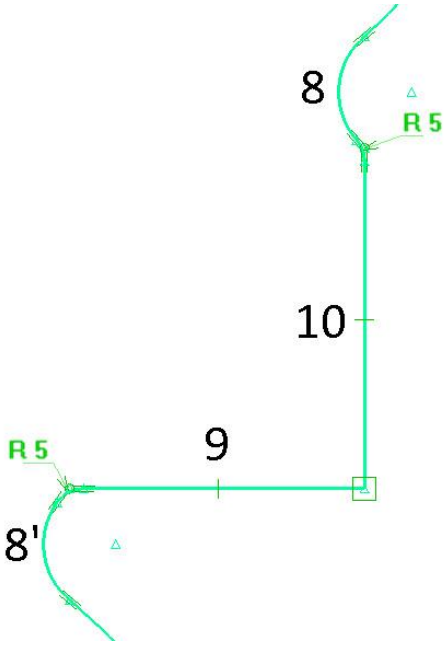


- Use the **Modify parameter** function to change the dimension values of the points of arcs **7'** and **8'** as shown opposite. 



Close the sketch

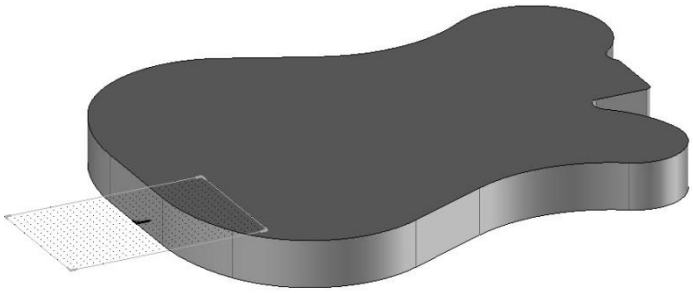
- Draw two lines **9** and **10** between the arcs **8** and **8'** as shown opposite.



- Apply:
  - an **orientation constraint along X** to the line 9;
  - a **perpendicularity** constraint between lines 9 and 10;
  - fillets with a **radius** = 5mm between 8'/9, 9/10 and 10/8.

- **End the sketch**.

Extrude the body

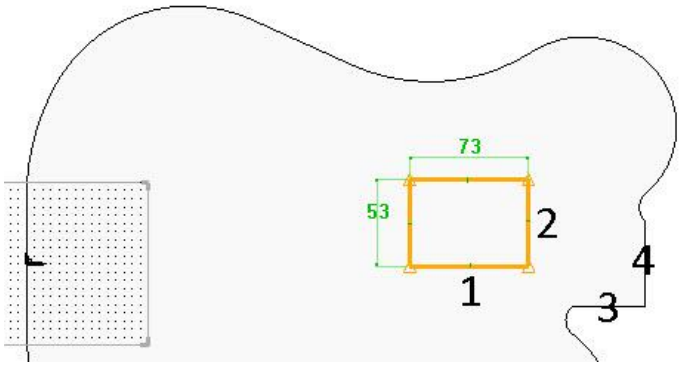


- From the **Shapes** context, start the **Extruded** function. 
- Select the previously drawn sketch in **Section curves or texts**.
- Extrude the body to a **height** of 35mm**.**

- Save this document:
  - Answer **No** to the request for a part definition.
  - Create a **new folder** called *Guitar body* and rename the file *Guitar body.*

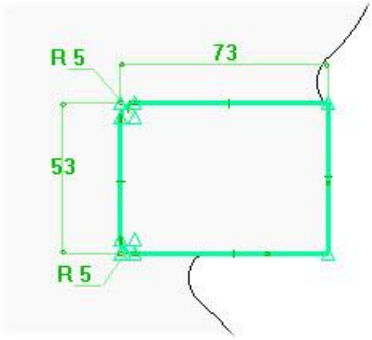
Performing the operations

Make the pockets

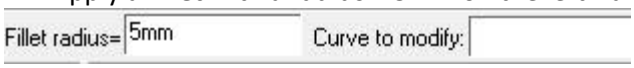


- Make **layer 1** current. 
- Start a **new sketch**, then use **Contour** > **Rectangular**.
- Draw a rectangle as shown opposite and select **Auto dimension** 

to automatically place the dimensions of the rectangle.

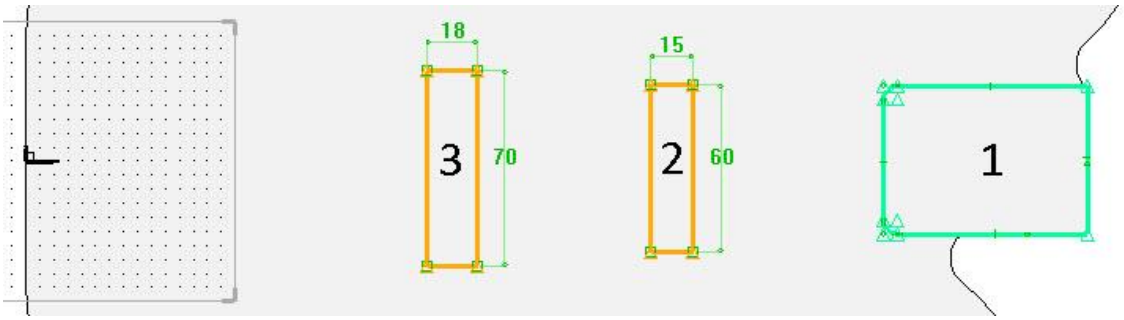


- Use the **Modify parameter** function to change the values to 73mm on the X axis and to 53mm on the Y axis.
- Apply two **coincidence constraints**:
  - between the segment 1 of the rectangle and the edge 3 of the guitar;
  - between the segment 2 of the rectangle and the edge 4 of the guitar.
- Apply a **fillet** with a **radius** = 5mm on the left-hand angles of rectangle 1.

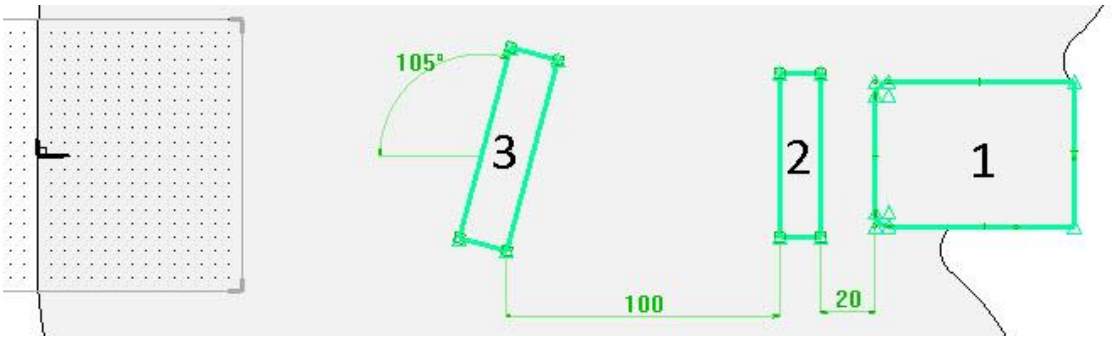


- Draw two new **rectangular contours** in **Constraints = Perpendicularity** mode.

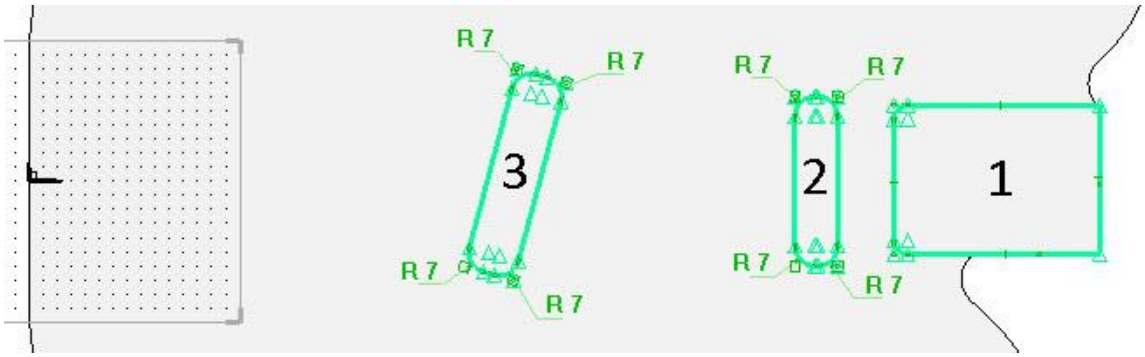
- Select **Auto dimension** to place the dimensions of the rectangles.
- Use **Modify parameter**  to change the dimensions of the rectangles:
  - 15mm on X and 60mm on Y for rectangle 2;
  - 18mm on X and 70mm on Y for rectangle 3.



- Use the **Modify element** function on the vertical dimensions of the rectangles (60 and 70mm) to apply a **constraint** to the **X axis**.
- **Dimension**:
  - the distance between rectangles 1 and 2 to 20mm;
  - the angle between the left-hand side of rectangle 3 and the **X** axis of the coordinate system to 105°;
  - the distance between the left-hand side of rectangle 2 and the bottom right point of rectangle 3 to 100mm.



- Apply **fillets** with a **radius** = 7mm on the angles of rectangles **2** and **3**.



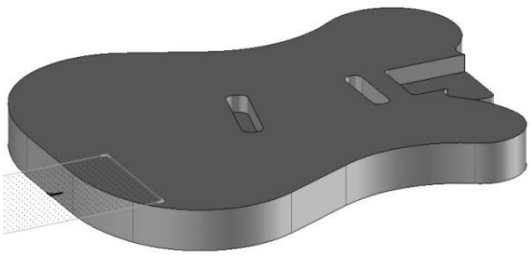
- **End the sketch**.
- In the **Wood** context, start the **Pocket** function. 
- Select the top of the guitar body as the **reference face**.
- Set **Generatrix = Hidden** and **Generatrix sketch = Global**.

Note: **Generatrix = Hidden** is used to hide the current sketch once the pocket is performed.
Generatrix sketch = Global generates a pocket for each contour of the sketch in one single operation. This means that all the pockets will have the same characteristics.

- In **Curve(s)**, select the sketch.

The **Pocket parameters** window opens.

- Set:
  - **Through**: **No**.
  - **Depth**: 15mm.



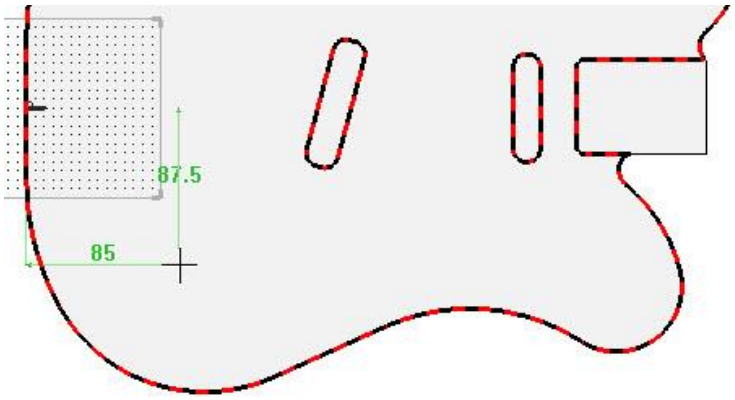
- Confirm the window with **OK** to make the pockets.

Make the drillings

- Start the **Drilling** function. 
- Set **Coordinate system = Constraint**, **Mode = Dynamic** and **Hook = Current coordinate system**.

Note: The **Dynamic mode** allows a drilling to be positioned and dimensioned at the same time.
Hook = Current coordinate system is used to place the drilling dimensions in relation to the current coordinate system.

- Position the drilling on the top face of the guitar body at 85mm in the X+ direction and 87mm in the Y- direction.

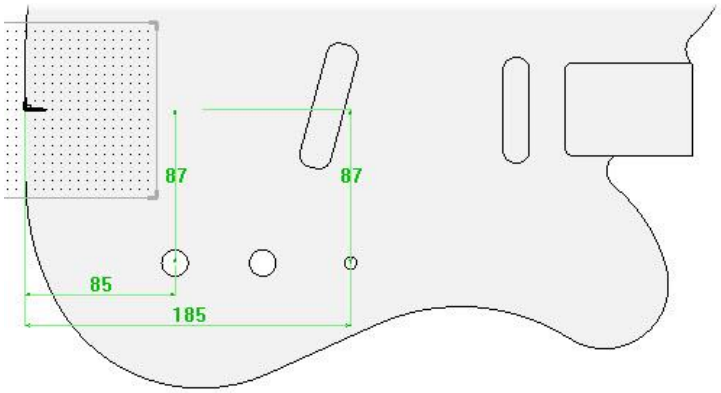


Note: The hooking of the drilling in dynamic mode is made based on the precision of the step defined in the grid.
If the value of the dimension is not exact, it will be possible to modify it later using **Modify parameter**.

- In the **Drilling models** window, select **hole** in **Standard models**, then set:
  - **Hole**: **Through one**
  - **Diameter**: 15mm

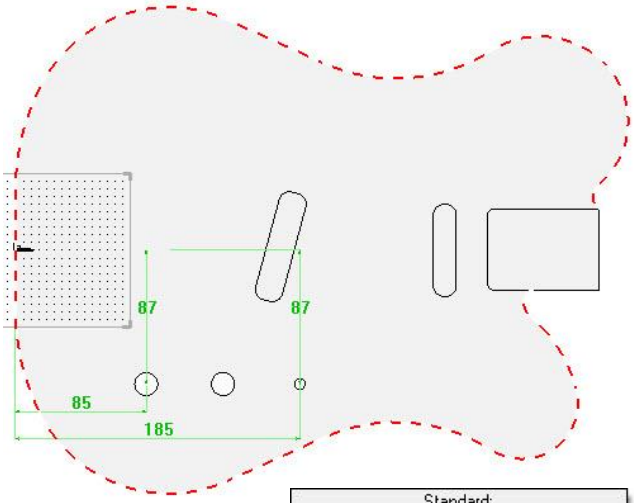
- Once the drilling has been performed, select **Propagate** in the dialog bar. 

- Select **Linear propagation** in the **X+ direction**, set the **distance per instance** to 50mm and the **total number** to 2.



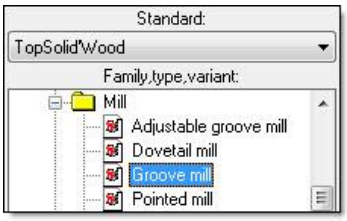
- Then make a third drilling:
  - 185mm in the X+ direction and 87mm in the Y- direction
  - **Hole**
  - **Through one**, **Diameter**: 7mm

Make the moulding



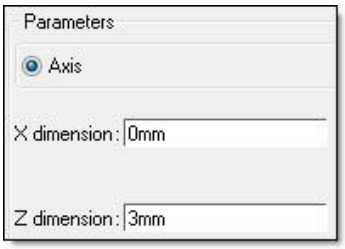
- Start the **Moulding** function. 
- Select **Sweep = Planar face** to perform a standard moulding on a planar face.
- Select the top of the guitar body as the **reference face**.

- Set **Join edges = YES** and **Follow tangent edges = YES**.
- In **Reference edge or curve for tool path**, select the contour edge of the guitar as shown opposite. Click on **Stop** to confirm the selection.



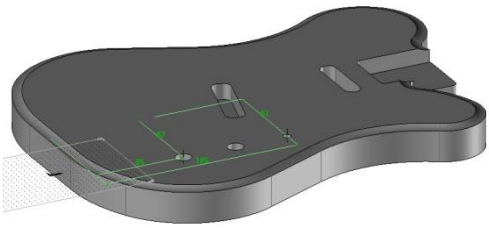
- Click on **OK** to confirm the default direction of the arrows.

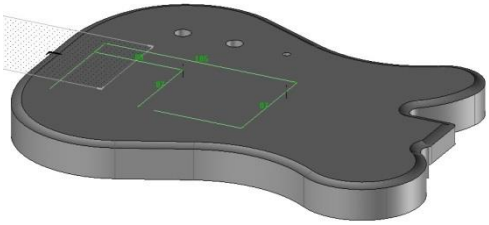
The **Parameters** window of the moulding opens.



- In **Standard TopSolid'Wood** select **Groove mill** in the **Mill** category.
- Set an **entry** and an **exit** with a **radius** = 0mm.
- Set **Parameters = Axis** and **Z dimension = 3mm**.

- Click on **OK** to perform the moulding.
- Use **Copy operation** to perform the same moulding on the base of the guitar.

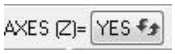




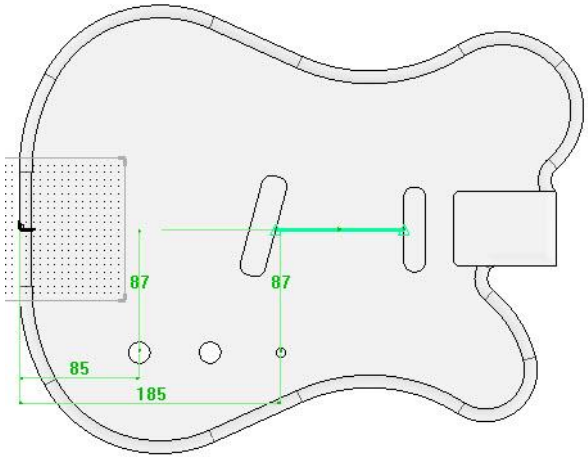
Create the groove

- Make **layer 2** current.

- Start a **new sketch.**
- Draw a horizontal **line** in **Axes (Z) = YES**.

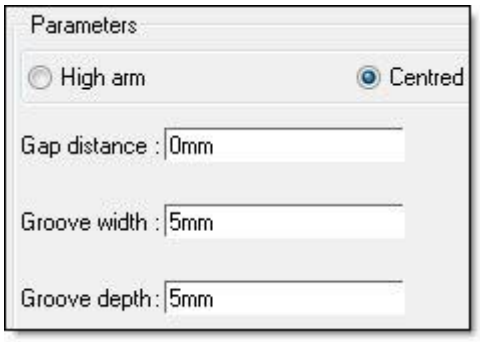


- Apply an **alignment constraint** to this line with the **X axis**.
- Apply two **coincidence constraints**: 
  - of the left-hand point of the line on the left-hand pocket;
  - of the right-hand point of the line on the middle pocket.

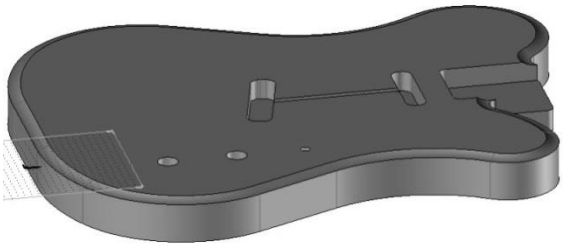


- Finish the sketch.

- Make a **groove** on the top face with the drawn line as the **tool path** and the guitar body as the **shape to profile**.
- Select **Stop.**
- In the **Parameters** window of the groove, set:
  - **Tool type**: **Routers**
  - **Standard: TopSolid'Wood**
  - **Mill** > **Simple mill**
  - **Parameters** : **Centred**
  - **Gap distance**: 0mm
  - **Groove width**: 5mm
  - **Groove depth**: 5mm



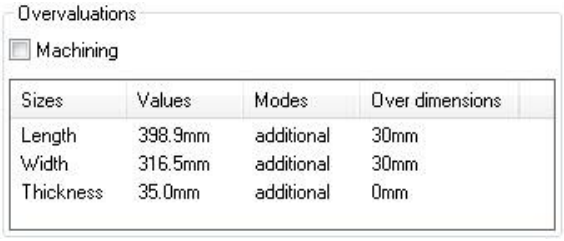
- Click on **OK** to confirm the settings.



Data definition

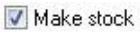
Define the part

- Use the **Wood** > **Define** > **Define part** function to define the guitar body:
  - **Designation**: *Guitar body*
  - **Material**: **wood** > **mahogany**
- In the **Cutting-up** tab, add 30mm **over dimensions** in **length** and **width**.



Note: It is possible with an option to view the stock of the part as shown below. Viewing the stock is helpful to validate it. However, it is not recommended to work with the stock displayed.

- In the **Part definition** window, open the **Stock** tab.
- Check **Make stock** and validate the window with **OK**.



Note: If the stock is not centered on the part, check the boxes **Both sides** for the **Over length** and the **Over width repartition** in the **Stock** tab.

- Start the **Define part** function.
- Select the guitar (and not the stock of the guitar) as the part to define using the rotary selection.
- Select **Modify cutting-up**.

- In the **Stock** tab, uncheck **Make stock**.

Define the set

- Use **Wood** > **Define** > **Define set** > **Characteristics** > **Modify cutting-up** to define the set:
  - **Designation**: *Guitar body*
  - **Main set**: **Single Unit**
- Save the document.

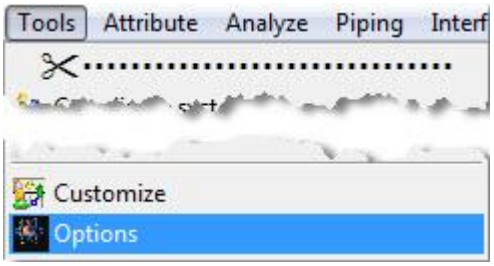
Introduction to the components

When designing design projects, the user often or always needs to use production standards. These standards can be for example:

- Hardware parts (screws, dowels, hinges, slides…)
- Interior design components (cabinets, doors, shelves, separation panels…)
- Complete furniture (coffee table, counter, door…)

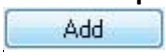
They allow information to be centralized and the component to be designed only once to help minimize the time it is used in projects.

Creating a library

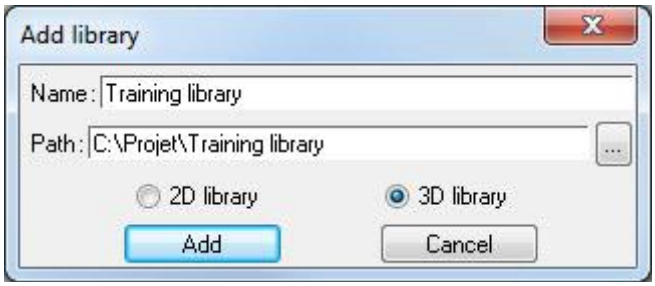


- Create a **new Design document**.
- Open the options using **Tools** > **Options**.

- In **Component** > **User library**, add a user library using the **Add** button.



- In the **Name** field, enter *Training library.*
- Select *C:\Projet* as the **path** and create a new folder called *Training library.*



- Click on **Add** to create this new library.

Name	Path	Mode
Training library	C:\Projet\Training library	3D

- Click on **OK** to confirm the **options**.

Note: In order to preserve you library components, it is not recommended that you modify and save the components of the TopSolid standard libraries (**My 3D Standard**, **TopSolid'Wood**, **AFNOR**…).
To modify and use the standard library components, it is better to save them in your own libraries.

Exercise 1: Creating a shelf

The goal of this exercise is to make the different parts of the shelf as constrained blocks, and then assemble them. This shelf will be saved in the library.

Concepts addressed:

- Parameters
- Construction volume
- Constrained blocks
- Defining the parts and the set
- Assembling the parts
- Saving in the library

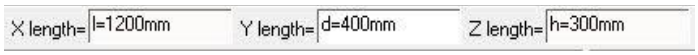
Designing the construction volume

Create the parameters

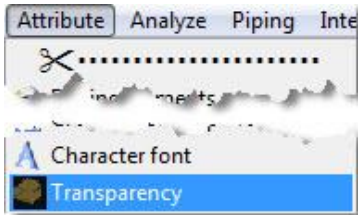
- Create a **new Design document**  . In the **Advanced parameters**, select **Without template**.
- Use the **Parameter** > **Create** function to create the following four parameters in **Unit type = Length** mode. Do not display the texts of the parameters.
  - **Value** = 1200; **Name**: *l*; **Designation**: *Shelf length*
  - **Value** = 400; **Name**: *d;* **Designation**: *Shelf depth*
  - **Value** = 300; **Name**: *h*; **Designation**: *Shelf height*
  - **Value** = 19; **Name**: *t*; **Designation**: *Panel thickness*

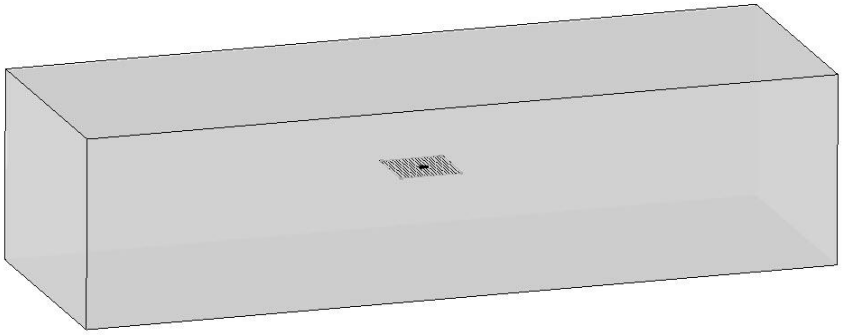
Create the construction volume

- In the **Shapes** context  , start the **Block** function  , then enter **X length = l**, **Y length =d** and **Z length = h**.



- Select the center of the absolute coordinate system as the **alignment point**.
- Start the **Attribute** > **Transparency** function to apply a transparency of **7** to this block.



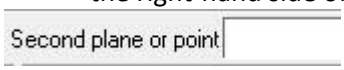


Creating the parts

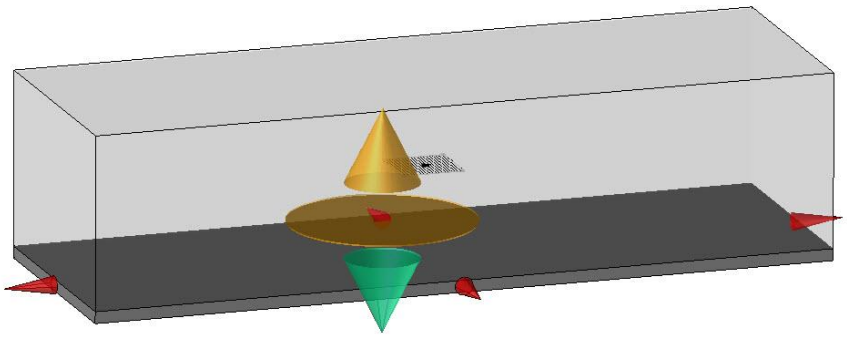
Low part

- Make **layer 1** current. 
- In the **Wood**  context, start the **Constrained block** function. 
- Set the *t* parameter as the **thickness**, then select:
  - the left-hand side of the construction block as the **first plane**.

  - the right-hand side of the construction block as the **second plane or point**.



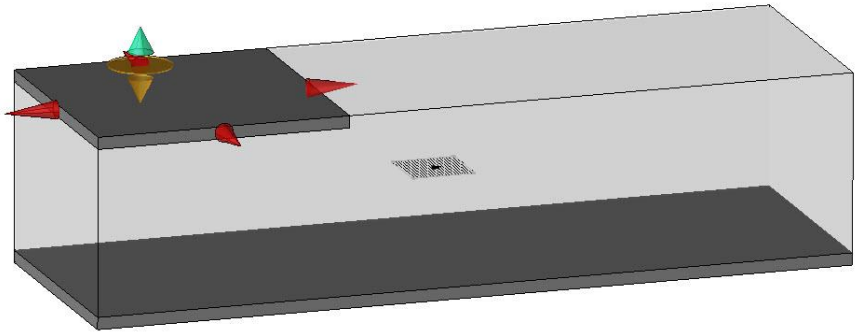
  - the front face of the construction volume as the **first plane**.
  - the rear face of the construction volume as the **second plane or point**.
  - the bottom face of the construction volume as the **positioning plane**.
- Adjust the positioning of the constrained block using the yellow arrows so as to place the part inside the construction volume.



Top part

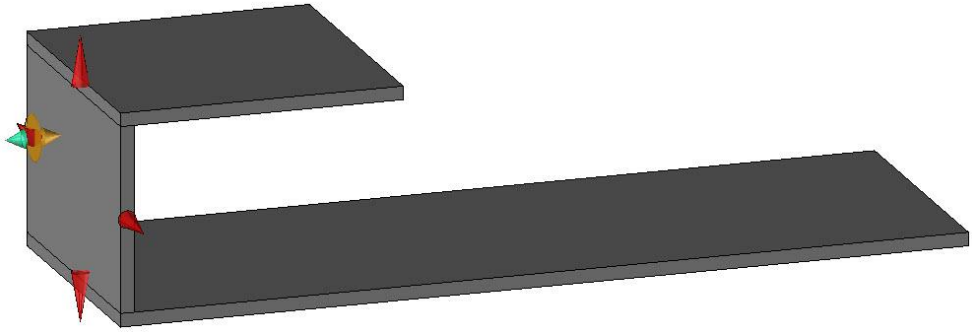
- Create another constrained block to make the part below:
  - **Thickness** = *t*
  - **First plane**: Select the left-hand face of the construction volume.
  - **Second plane**: Set **Mode = Faces**, enter **Second shift = (2/3)*l**, then select the right-hand face of the construction volume.

  - **First plane/Second plane**: Select the front and rear faces of the construction volume.
  - **Positioning plane**: Select the top face of the construction volume.
  - Position the constrained block inside the construction volume.



Side part

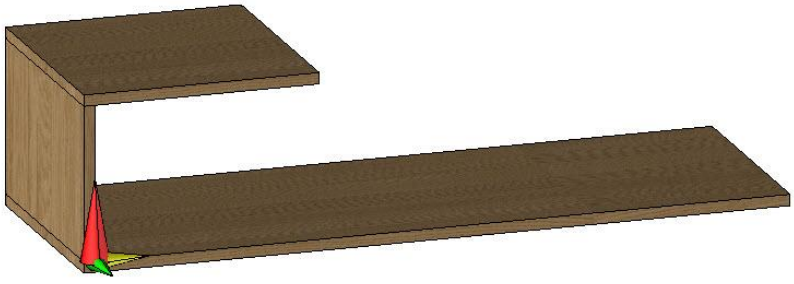
- Switch off **layer 0**.
- Using the **constrained block**, make the part below:
  - The **constrained block** has a **thickness** = *t*.



Defining the parts and the set

Define the parts

- Define the parts using **Wood** > **Define** > **Define part**.
- For the three parts, set the material to **wood** > **oak**.
- For the low part:
  - **Designation**: *Bottom shelf top*
  - **Reference**: *BST*
- For the top part:
  - **Designation**: *Top shelf top*
  - **Reference**: *TST*
- For the side part:
  - **Designation**: *Shelf side*
  - **Reference**: *SS*
  - The grain orientation of the part must be placed on the height. Use the **Invert axes** button in the **Cutting-up** tab to rotate the cutting-up axes of the part.



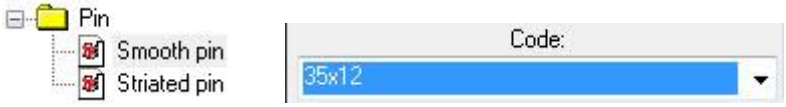
Note: In TopSolid'Wood, the length of a part always matches its grain orientation.

Define the set

- Use **Wood** > **Define** > **Define set** > **Characteristics** to define the set.
  - **Designation**: *Wall shelf*
  - **Reference**: *WS*
  - **Main set: Sub-assembly**

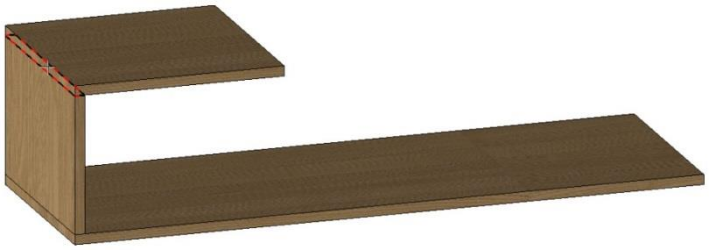
Assembling the shelf

- In the **Wood** context, start the **Dowel** function. 
- Select **Smooth pin** in the **Pin** category and select the code **35x12**.



- In the dialog bar, select the **Filter** mode and set **Propagation = YES**.

- Select:
  - the contact face between the side et the top as the **support face**. Use the **rotary selection** to select this face.

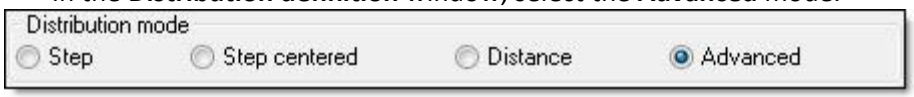


  - the front edge of the top as the **start face or edge**.
  - **Centred automatically**. 

Note: The **Centred automatically** option is used to automatically center the assembly in the thickness of the part.

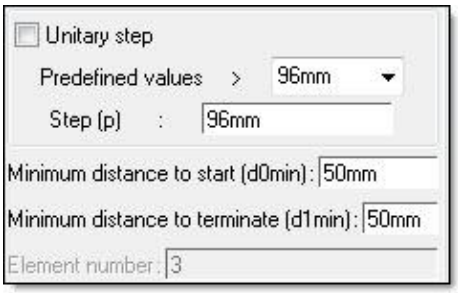
  - the rear face of the top as the **terminate face or edge**.

- In the **Distribution definition** window, select the **Advanced** mode.

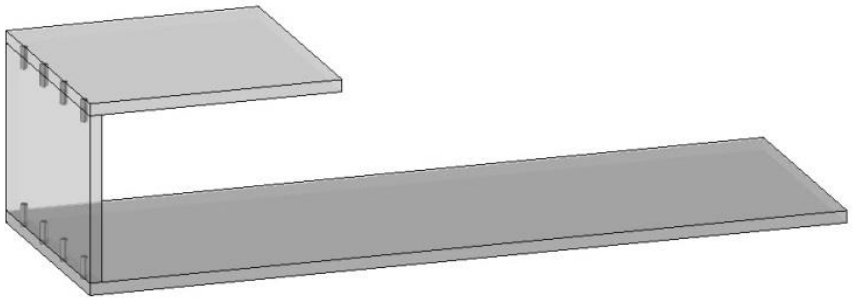


Note: The **Advanced** mode is used to provide minimum start and terminate distances for the dowels, as well as a step between the dowels. The quantity of dowels will be automatically calculated.

- Then set the following parameters:
  - **Step**: 96mm
  - **Minimum distance to start**: 50mm
  - **Minimum distance to terminate**: 50mm



- Confirm with **OK** to position the dowels.
- Use **Copy propagation**, and then repeat the operation to assemble the side with the bottom shelf top. 

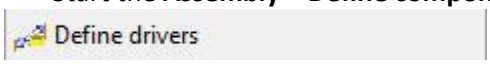


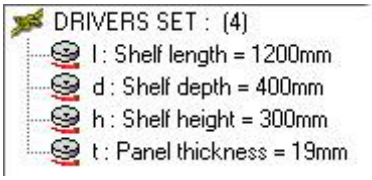
Saving in the library

Define the parameters as drivers

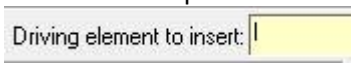
Note: Defining parameters as drivers allows their values to be modified when inserting the component in an assembly file.

- Start the **Assembly** > **Define component** > **Define drivers** function.





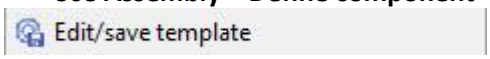
- Enter the **l** parameter as the **driving element to insert**.



- Press **Enter** to confirm.
- Press **Enter** again to confirm the parameter designation.
- Define the parameters **d**, **h** and **t** In the same way.

Save the component in the library

- Use **Assembly** > **Define component** > **Edit/save template**.



- Select **Save standard template**.

Note: **Save standard template** is used to save a file in the component library.
Edit standard template is used to edit a file already saved in the library.

- In the **Standard** dropdown list, select the library you created: **Training Library**.
- Enter:
  - **New family**: *Furniture*
  - **New type**: *Shelves*
  - **New variant**: *Wall shelf*

Note: **Family**, **Type** and **Variant** correspond to the different levels of the component library.

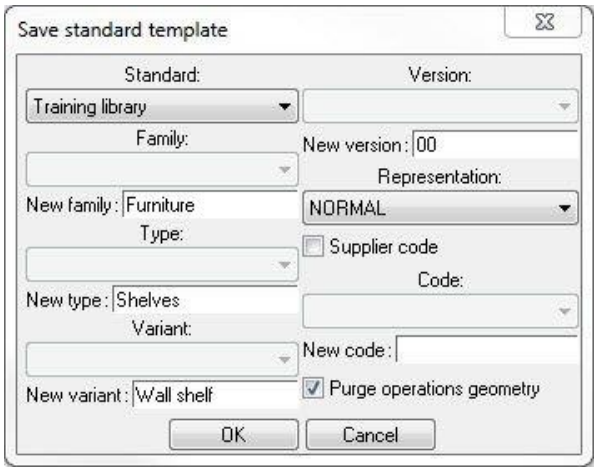
- In the **New version** field, enter *00*.

Note: A new **version** of a component is done when a number of changes have been made to that component. This new component will be saved in a new file.

Thus, the assemblies using the old version of the component are not modified and you can use the new version for the new assemblies.

- Uncheck **Supplier code** and check **Purge operations geometry**.

Note: The **Purge operations geometry** option improves the calculation performance of the components.



Note: Once the component has been saved in the library, $STD=**LibraryName** is shown before the file name in the software banner.

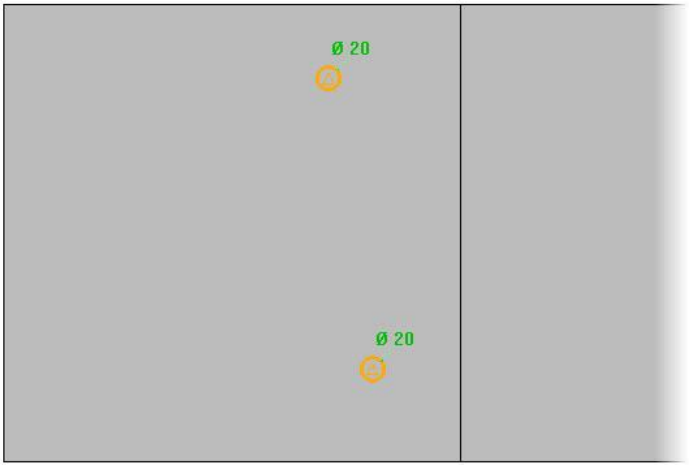
[Design : $STD=Training library\Furniture\Shelves#V=Wall shelf#I=00#R=NR.top

Once the file has been saved in the library, click on the disk icon to save the changes made to the file. 

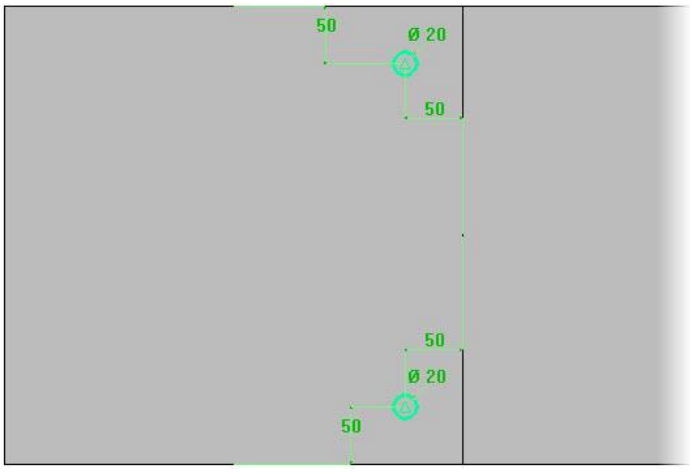
Supplement: Making the supports

Create the shelf supports

- Make **layer 2** current.
- Start a **new sketch** and draw two **circles** 20mm in **diameter** as shown opposite.

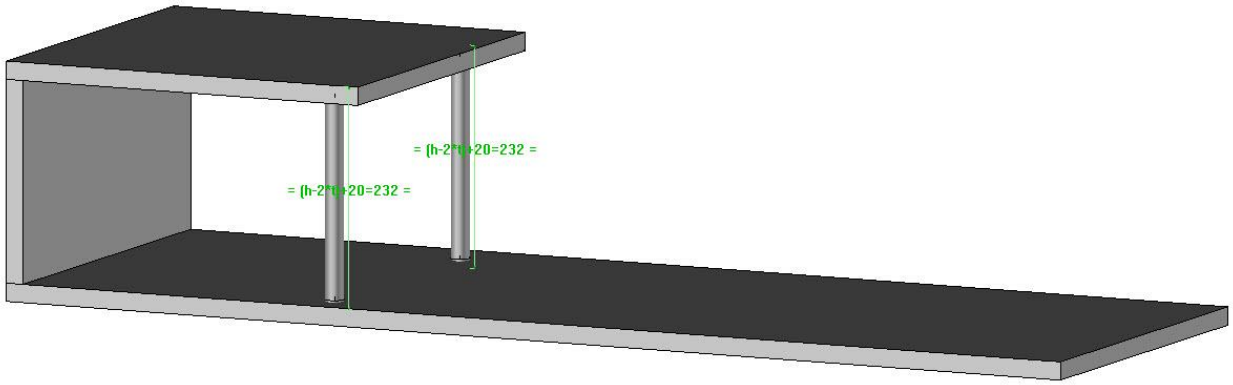


- Use the **Dimension** function to dimension the center of the circle with the top shelf top edges to 50mm as shown opposite. 



- **End the sketch**, then extrude it in **Generatrix sketch = Global** and **Result = One shape per profile** modes.

- Once the sketch selected, extrude it to a **height** = (h-2*t)+20 in **Alignment = Centered** and **Type = Solid** modes.



Define the supports

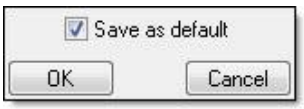
- Using **Wood** > **Define part**, define the two supports previously created.
- Select the cylinder of the support as the **length axis** and **Y+** as the **width axis**.
- Set:
  - **Designation**: *Left support/Right support*
  - **Reference**: *LS/RS*
  - **Material**: **metal** > **aluminum**

Make the drillings

- Start the **Wood** > **Drilling** function. 
- Set **Mode = Non dynamic**, then select the upper face of the bottom shelf top as **the face to drill**.

- Select the cylinder of a support as the **first alignment face or edge**.

Note: When making a drilling, selecting an existing cylinder, circle or axis allows you to automatically orient the drilling with the selected element.



- In the **Drilling models** window, select **Hole** in **Standard models**.
- Check **Save as default**, then click on **OK** to confirm.

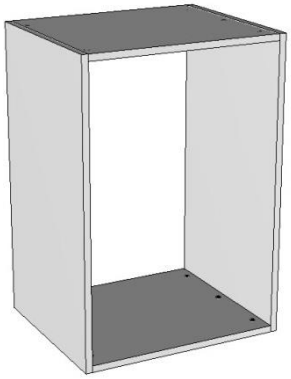
Note: Checking **Save as default** saves the drilling values for the following drilling operations.

- Set the drilling:
  - **Hole**: **Blind**
  - **Diameter**: 20mm
  - **Depth**: 11mm
  - **Bottom angle**: 0°

- Save the file using the disk icon.

Note: Once the file has been saved in the library, simply click on the disk icon to save the modifications.

Exercise 2: Creating a cabinet component



The goal of this exercise is to create the cabinet in a driver block, and then assemble it with dowels and cams.

Concepts addressed:

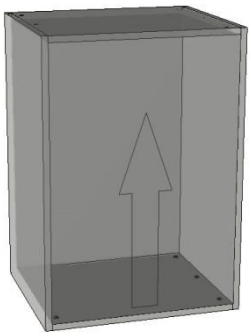
- Driver block
- Propagations
- Panels

Creating a driver block component

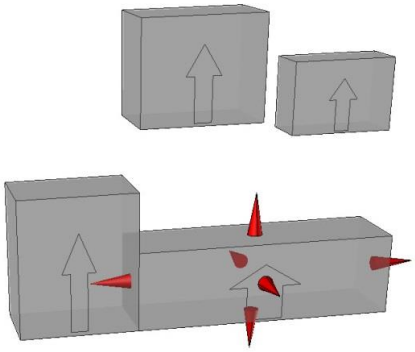
Note

The **driver block** is used to drive a component from a block.
This makes it easier to create and use all kinds of components that can be inserted in a rectangular block: cabinet, back of furniture, door, shelves…

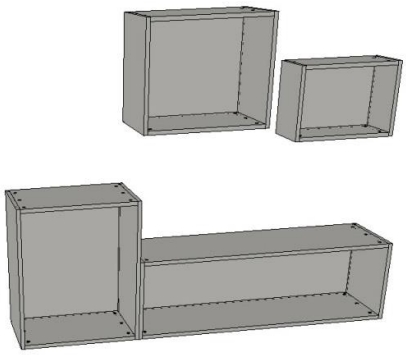
Cabinet component as driver block



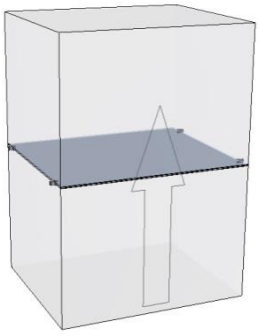
Insertion of the cabinet as driver block



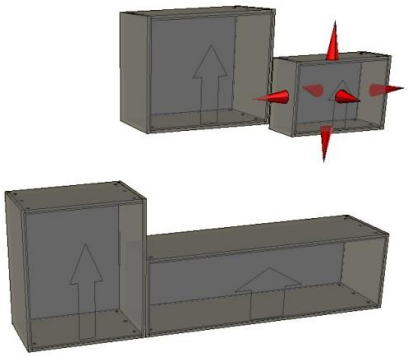
Assembly result



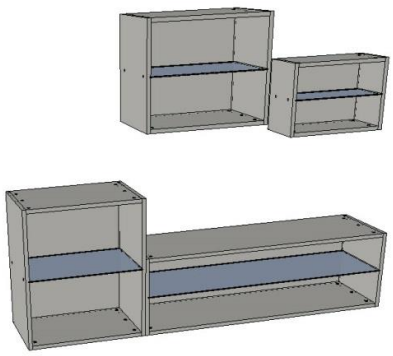
Shelf component as driver block



Component insertion



Result



Create the driver block

- Create a **new Design document**.
- Create the driver block using the **Assembly** > **Define component** > **Define drivers** > **Driver block** function.

- Enter *db* as the **name of driving element**.

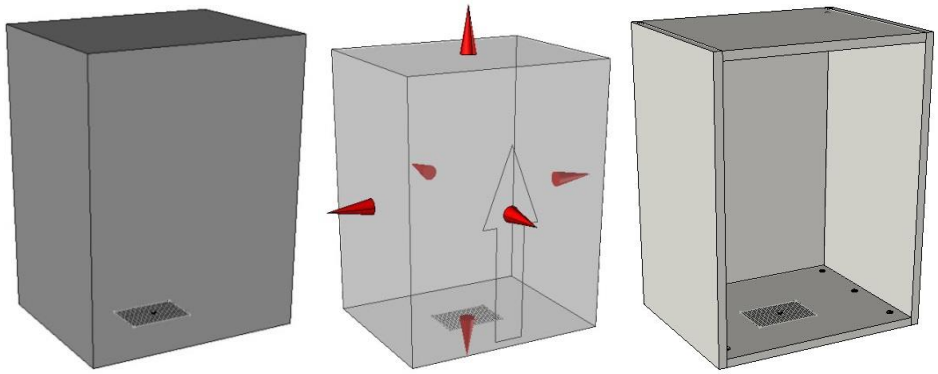
Note: The **name** and the **designation** of a driver block work the same as for a parameter.

The **name** is the system name used by the parameter. It must be simple and cannot contain any spaces. The **designation** is what the user sees when using the parameter. It must be explicit and can contain spaces.

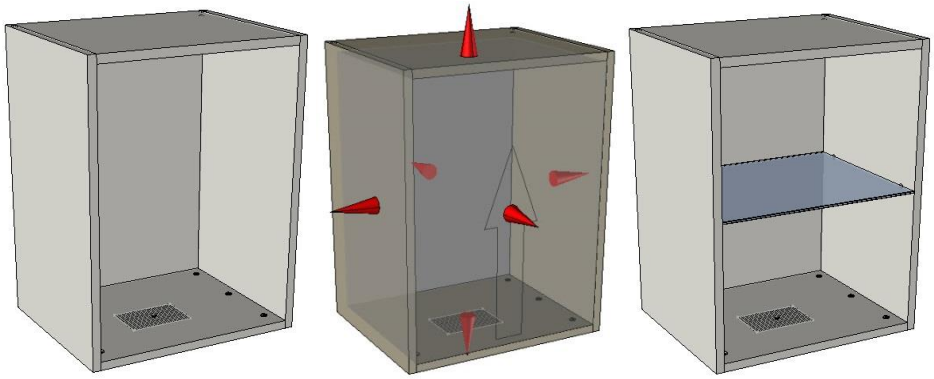
- The **lengths** on **X**, **Y** and **Z** correspond to the default dimensions of the driver block. Keep the default dimensions.
- Choose **Default housing mode = Inside a block**.

Note: A driver block component can be inserted in two different ways:

- **Inside a block**:



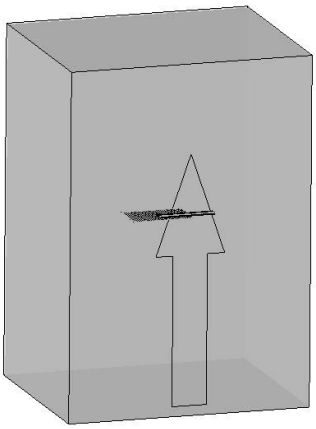
- **Housing**:



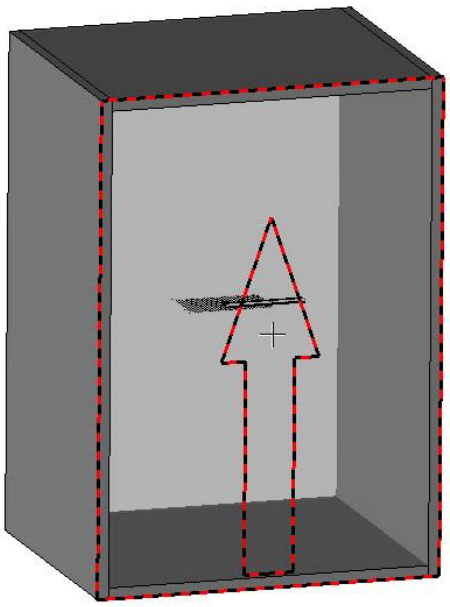
- Select **Current coordinate system** to position the **driver block**.

- Enter *Cabinet block* as the **designation of the driving element** and click on **OK** to confirm.

Note: The arrow on the driver block's face represents the front of the driver block and shows the top of the driver block.



Create the cabinet parts



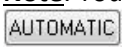
- Make **layer 1** current.
- Create a **length parameter**:
  - **Value** = 19mm
  - **Name**: *t*.
  - **Designation**: *Panel thickness*.

Name	Designation	Value
t	Panel thickness	19mm

- Using the **constrained block**, create the 4 parts shown opposite.

  - The constrained blocks have a **thickness = t**.
  - The constrained blocks are placed inside the driver block.
  - The left-hand and right-hand sides of the cabinet cover the top and the base.

Note: You can use the **Automatic** option to create the **constrained blocks** more quickly.

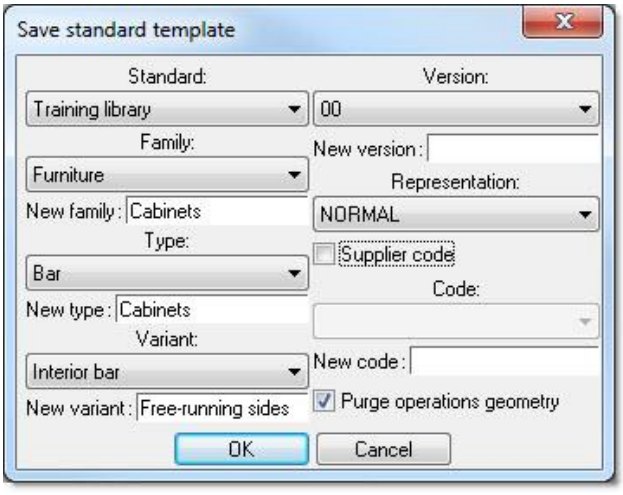


- Define the four parts of the cabinet:
  - **Designation**: *Top / Base / Left-hand side / Right-hand side*
  - **Reference**: *TO / BA / LS / RS*
  - **Material**: **wood** > **particule board**
  - **Coating**: **paint** > **mat white paint**

- Define the cabinet set:
  - **Designation**: *Free-running sided cabinet*
  - **Reference**: *FRS-CAB*
  - **Main set**: **Sub-assembly**

Save the cabinet

- Define the **t** parameter as a driving parameter using the **Assembly** > **Define component** > **Define drivers** function.
- Use **Assembly** > **Define component** > **Edit/save template** to save the file in the library.
- Select **Save standard template**.
- In the window, set:
  - **Standard**: **Training library**
  - **New family**: *Cabinets*
  - **New type**: *Cabinets*
  - **New variant**: *Free-running sides*
  - Uncheck **Supplier code**.
- Click on **OK** to confirm the window.

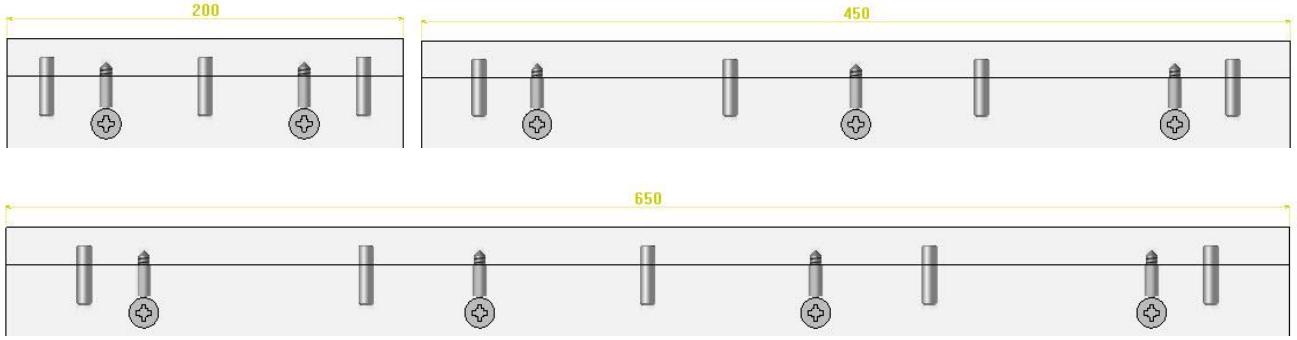


Assembling the cabinet

Create predefined propagations

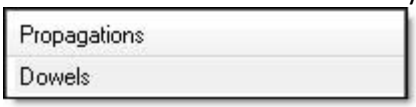
Note

As seen before, it is possible during assembly functions to use a certain type of propagation to configure the assembly.
Predefined propagations can also be configured so that the propagation parameters don't have to be filled for each assembly.
These predefined propagations also define different propagation rules according to the length to assemble.



- In a **Design document**, open **Tools** > **Options** > **TopSolid'Wood Configuration** > **Propagations configuration**.
- Select **Add propagation** to create a new propagation. 

- Then double-click on the newly created line **Untitled** and name this propagation *Dowels.*



- In the propagation list, modify the **Right bound** value on the first line to 80mm.
- Use the **Add range** button to create the different ranges below. 

Left bound	Right bound
0mm	*80mm*
80mm	*200mm*
200mm	*400mm*
400mm	*650mm*
650mm	**Infinite**

- Then set the following propagations according to the ranges.

Dowels

Left bound	Right bound	Type	Step	d0	d1	Element nb	Optimize
[0mm	80mm[	Step centered	0mm	-	-	1	
[80mm	200mm[	Distance	-	20mm	20mm	2	-
[200mm	400mm[	Distance	-	20mm	20mm	3	-
[400mm	650mm[	Distance	-	30mm	30mm	4	-
[650mm	Infinite	Distance	-	50mm	50mm	5	-

- Add a new propagation using **Add propagation**.
- Name this propagation *Cams.*
- Recreate the same ranges as for the *Dowels* propagation.
- Set the different propagations as shown below.

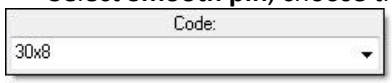
Cams

Left bound	Right bound	Type	Step	d0	d1	Element nb	Optimize
[0mm	80mm[	Not any	-	-	-	-	-
[80mm	200mm[	Step centered	0mm	-	-	1	
[200mm	400mm[	Distance	-	50mm	50mm	2	-
[400mm	650mm[	Distance	-	60mm	60mm	3	-
[650mm	Infinite	Distance	-	70mm	70mm	4	-

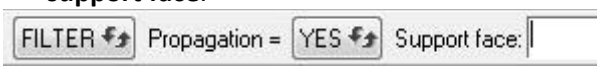
- Click on **OK** to confirm the window.

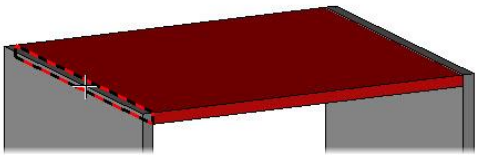
Assemble the cabinet

- In the free-running sided cabinet template, start the **Dowel** function. 
- Select **Smooth pin**, choose the code **30x8**, and then click on **OK** to confirm.



- Select the **Filter** mode and set **Propagation = YES**.
- Select the face between the **top** and the **left-hand side** as the **support face**.

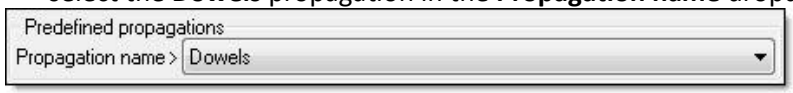


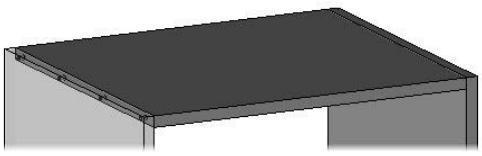


Note: When the support face of the parts to assemble is rectangular, the **Automatic** option can be used to detect the start, terminate and centred automatically faces.

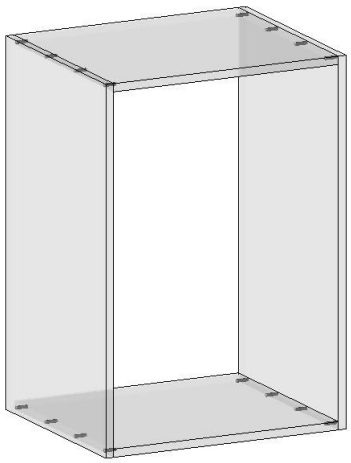
- Select **Automatic**. 

- The **Distribution definition** window opens. To use here the previously configured predefined propagations, select the **Dowels** propagation in the **Propagation name** dropdown list.

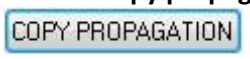




- Confirm the window with **OK** to assemble the parts.

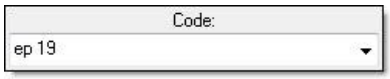


- Use **Copy propagation** to assemble other parts with the same propagation.

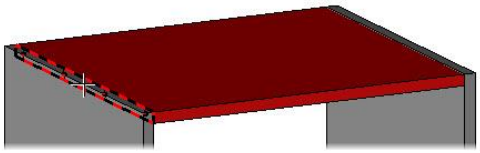


- Repeat the previous steps to **dowel assemble** the four parts.

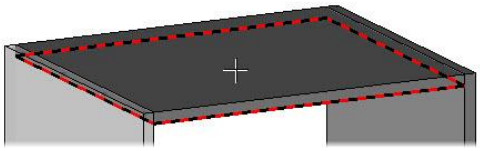
- Start the **Cam** function. 
- Select **Wood eccentric assembly hinge** with the **code**: **ep 19**.



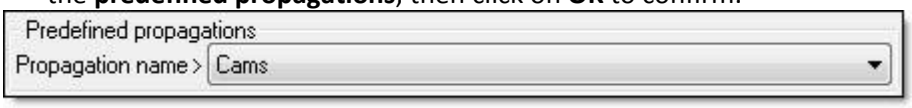
- Select the **Filter** mode and set **Propagation = YES**.
- Select the contact face between the **top** and the **left-hand side** as the **support face**.

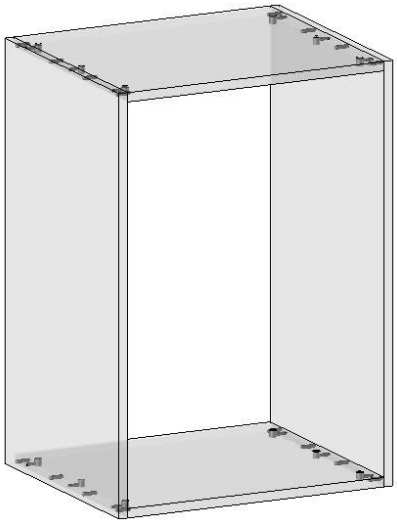


- Then select the bottom face of the **top** as the **face to drill for the case** of the cam.



- Use the **Automatic** option.
- In the **Distribution definition** window, choose the **Cams** propagation as the **predefined propagations**, then click on **OK** to confirm.





- Use **Copy propagation** to position the three other assemblies.
- Save the cabinet.

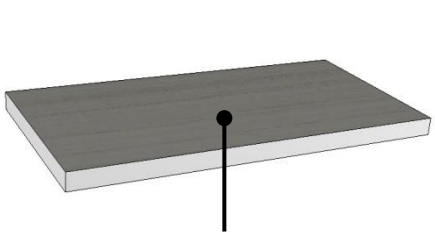
Creating the panels

Note

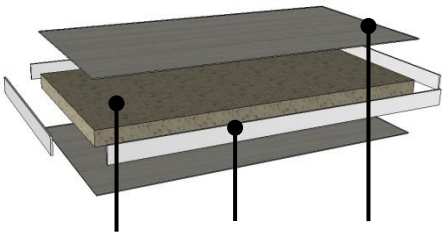
Creating panels allows edges and laminates to be placed on parts.
Several elements are then generated:

- A **panel entity**: Includes the part, the edges and the laminates.
- A **support**: This is the part on which edges and laminates are placed.
- **Edges** and **laminates**.

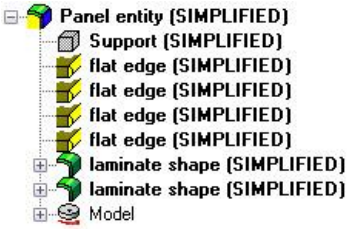
It is then possible to use the **panel entity** or the **support**, depending on the manufacturing processes. For example, for the machining, if the part is machined before the edges are placed, the support will be used. But if the part is machined after the edges are placed, the panel entity will be used.



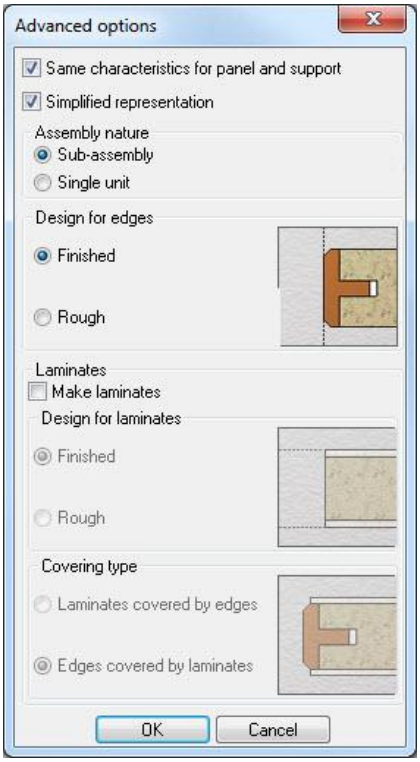
Panel entity



Support - Edges - Laminates



- In the **Wood** context, start the **Panel** function. 
- Open the panel **advanced options**. 
- Set:



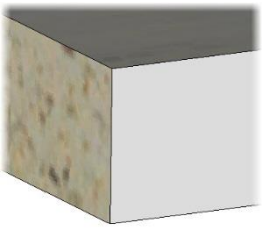
- **Same characteristics for panel and support**.

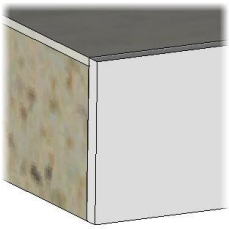
This option automatically applies the same characteristics (**designation**, **reference**…) to the **support** and the **panel entity**. Thus, the part definition is done only once.

- **Simplified representation**.

The simplified representation is used to view only the edge textures and the laminates without displaying them in 3D.

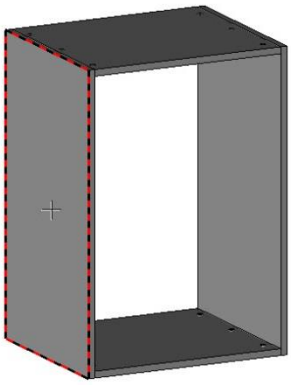
Note: For design purposes, it is strongly recommended that you always work in **simplified representation** when designing. This improves overall graphics performance during the designing.
The **detailed representation** is used for example in the draft to view the real edges.




- **Assembly nature**: **Sub-assembly**.

The **Sub-assembly** mode for the panels displays the BOM of the edges and the laminates on additional lines.

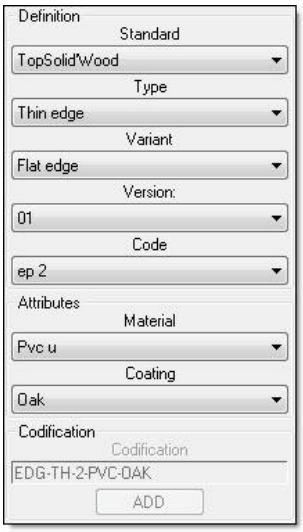


- **Design for edges**: **Finished**.
- Do not **make laminates**.

- Click on **OK** to validate the parameters.
- Select the main face of the **left-hand side** as the **reference face**.

The panel configuration window opens.

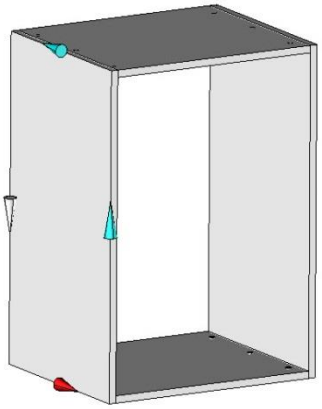
- Double-click in the **Edge type – code** box on the first line to select an edge:
  - **Standard**: **TopSolid'Wood**
  - **Type**: **Thin edge**
  - **Variant**: **Flat edge**
  - **Version**: **01**
  - **Code**: **ep 2**
  - **Material**: **PVC u**
  - **Coating**: **Oak**
  - **Codification**: *EDG-TH-2-PVC-OAK*



- Select **Add** to create the new edge codification, then click on **OK** to validate the edge.

Note: Edge codifications are used to provide the manufacturing reference of edges for the bill of material.
It is possible to use edges without codifying them by unchecking the **All edges and laminates must have one codification** option in **Tools** > **Options** > **TopSolid'Wood Configuration** > **Edge/ Laminate**.

Each line corresponds to an edge of the part. By clicking on a line, the arrow corresponding to the selected edge turns red.



- Uncheck the line of the rear edge so as not to place an edge on the rear edge of the panel.

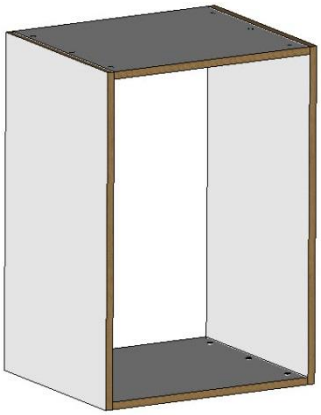
Its arrow then turns transparent.

- Double-click in the **Beginning cut type** box to set the **covering long side**.

Edges

	N°	Codification	Edge type - code	Length	Beginning cut ...	End cut...
☑	1	EDG-TH-2-PVC-OAK	Flat edge - ep 2	167.5mm	Covering	Covering
☑	2	EDG-TH-2-PVC-OAK	Flat edge - ep 2	72.5mm	Covered	Covered
☐	3	EDG-TH-2-PVC-OAK	Flat edge - ep 2	167.5mm	Covering	Covering
☑	4	EDG-TH-2-PVC-OAK	Flat edge - ep 2	72.5mm	Covered	Covered

- Confirm the window with **OK** to make the panel.
- Repeat these operations to apply edges to the three other parts:
  - On the right-hand side, the rear edge is not placed.
  - On the top and the base, only the front edge is placed.

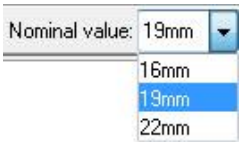


- Save the cabinet.

Supplement: Predefined values

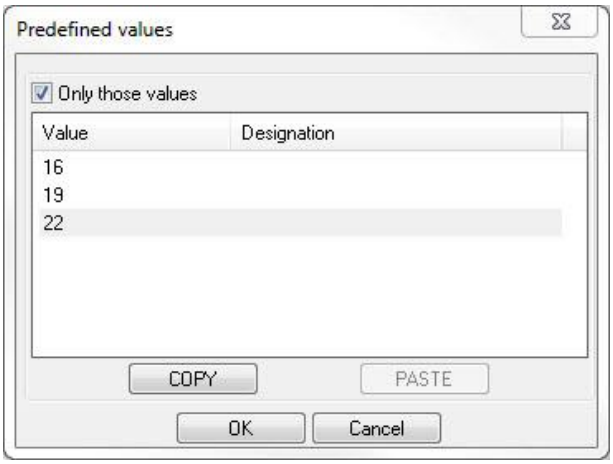
Note

Predefined values can be defined for a parameter.
This means that, when using this parameter, it is possible to use one of the predefined values.
This helps, for example, to set the panel thickness to certain values only.



- Start the **Parameter** > **Modify parameter** function.
- Enter *t* as the **parameter to modify** and press **Enter** to confirm.
- Open the **advanced parameters >>**. 
- Select **Predefined values=0**. 

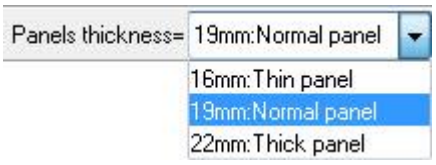
The **Predefined values** window opens.



- Enter the three **values 16**, **19** and **22**.
- Check **Only those values**.
- Click on **OK** to confirm the window.

Note: The **Only those values** option allows you to enter for this parameter only the predefined values indicated here.
The **Designation** field is used to associate a designation to the predefined values.

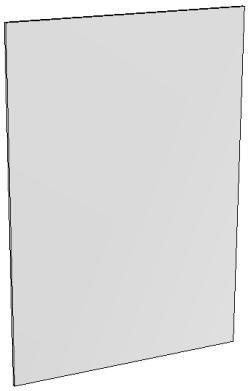
Value	Designation
16	Thin panel
19	Normal panel
22	Thick panel



- Save and close the cabinet.

Exercise 3: Creating a back of furniture in grooves

In a library component, it is possible to define machinings that will be performed when the component is inserted.
This is e.g. the case for cams and dowels which perform drillings after their insertion.
This process will be used in the "Back grooves" component which will automatically perform the grooves once inserted.



Concepts addressed:

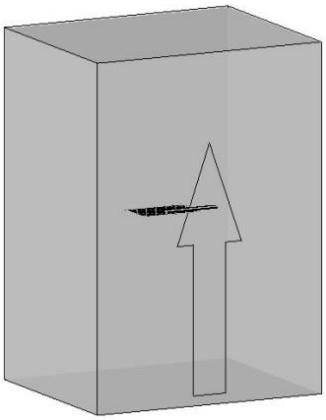
- Negative shifts on constrained blocks
- Martyr parts
- Component processes

Creating the component

Create the driver block

- Create a **new Design document**.
- Create the driver block using **Assembly** > **Define component** > **Define drivers** > **Driver block**:

DRIVER BLOCK



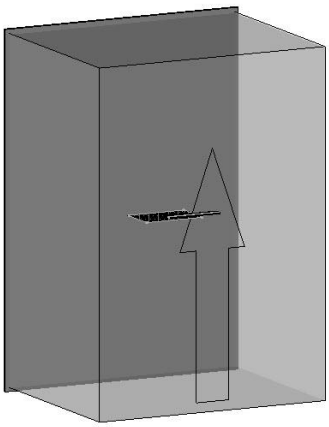
- **Name of driving element**: *db*
- Keep the default dimensions
- **Default housing mode = Housing**
- **Current coordinate system** to position the driver block on the current coordinate system.

  CURRENT COORDINATE SYSTEM

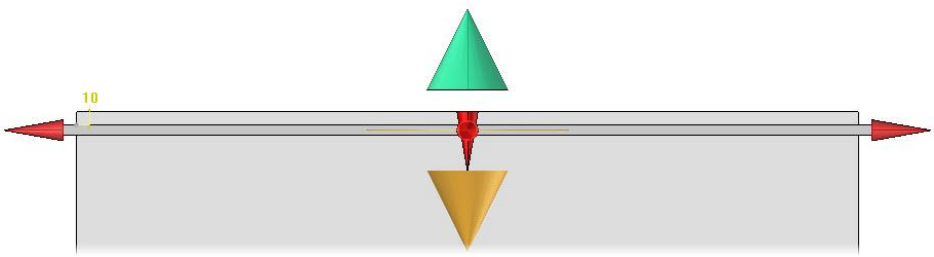
- **Designation of the driving element**: *Cabinet block*

  OK Designation of the driving element: Cabinet block

Create the back



- Make **layer 1** current.
- Use the **Constrained block** function to create the back: 
  - **Thickness** = 8mm
  - **Shift** = -5mm on the 4 selected faces
  - **Selected planes**: 4 sides of the driver block
  - **Positioning shift**: 10mm
  - **Positioning plane**: Rear face of the driver block



Define the part and the set

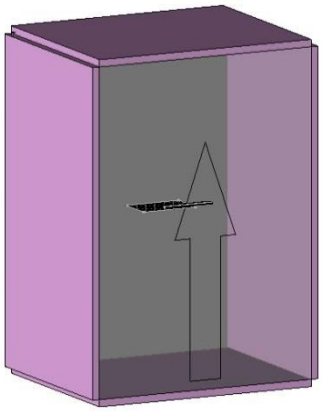
- Define the back using the **Define part** function:
  - **Designation**: *Part*
  - **Material**: **wood** > **particule board**.
  - **Coating**: **paint** > **mat white paint**.
- Define the set using the **Define set** function:
  - **Designation**: *Back grooves*
  - **Reference**: *BACK-GR*
  - **Main set**: **Single Unit**
- Save the component in the library using **Assembly** > **Define component** > **Edit/save template** > **Save standard template**:
  - **Standard**: **Training library**
  - **Family**: **Cabinets**
  - **New type**: *Backs*
  - **New variant**: *Back grooves*

Creating the processes

Create the martyr parts

Note: The goal of creating the **Back grooves** component is that it can perform its grooves automatically when inserted in a cabinet. To do this, the grooves must be made in the component on "**martyr**" parts: these parts will be created without being defined so that they are not included with the component.

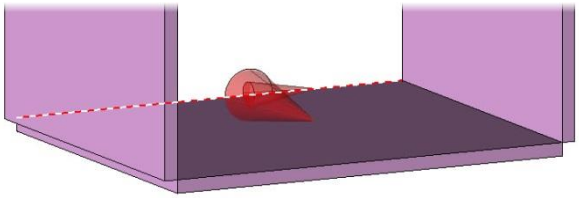
- Make **layer 2** current.
- Change the **design color** to differentiate the martyr parts. 
- Use the **Constrained block** function to automatically create the 4 martyr parts on the sides of the driver block:
  - **Thickness** = 19mm
  - **Selected planes**: 4 sides of the driver block
  - **Positioning**: Outside the driver block

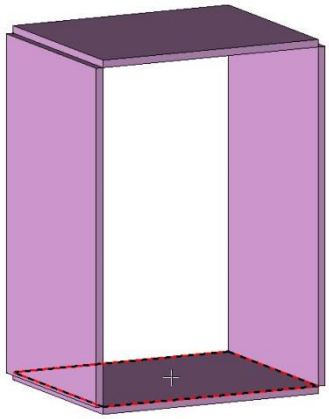


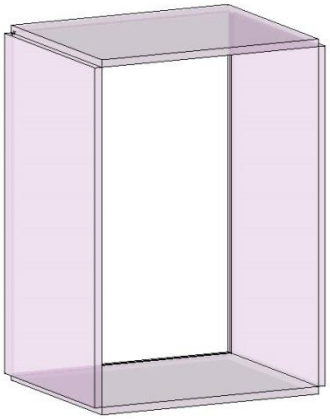
Note: The martyr parts of a component should not be panel entities.

Making the grooves

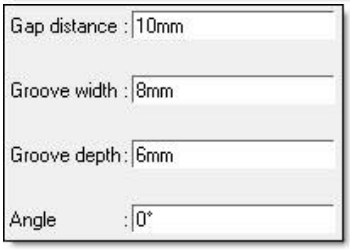
- Switch off **layers 0** and **1**.
- Start the **Groove** function. 
- Set **Sweep = Planar face** and select the inner face of the base martyr part as the **reference face**.
- Then select the rear edge (Y+) of the martyr part as the **reference edge or curve for tool path**.
- The gap of the groove symbolized by the red arrow must be towards the inside of the part. If needed, click on the arrow to invert.
- Click on **Stop** to validate the groove path.



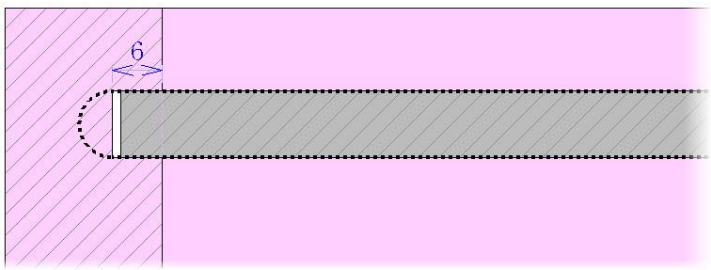




- Set the groove:
  - **Tool type**: **Routers**
  - **Standard**: **Simple mill**
  - **Entry/Exit**: **Radius**
  - **Distance to start point/from terminate point**: -6mm
  - **Parameters**: **High arm**
  - **Gap distance**: 10mm
  - **Groove width**: 8mm
  - **Groove depth**: 6mm



Note: The **distance to start point/from terminate point** of -6mm is used to extend the groove by 6mm towards the outside of the part to match the extension of the back with a shift of 1mm.

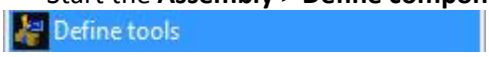


- Confirm the window with **OK** to make the groove.
- Use **Copy operation** to perform the same groove on the three other martyr parts.

Define the grooves as component processes

Note: Defining the grooves as component processes (also called component tools) allows these machinings to be performed when inserting the component.

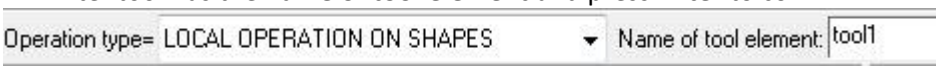
- Start the **Assembly** > **Define component** > **Define tools** function.



- Set **Operation type = Local operation on shapes**.

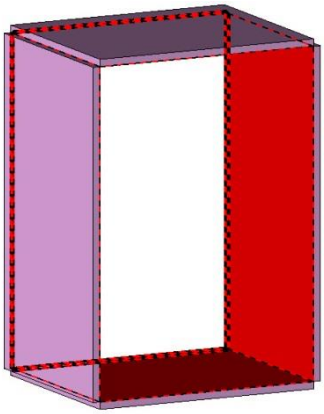
Note: A **local operation on shapes** corresponds to a performed machining.
If the process is not a machining, it is possible to choose a shape **subtraction** or **union**.

- Enter *tool1* as the **name of tool element** and press **Enter** to confirm.



- Modify the designation to *Back grooves* and press **Enter** to confirm.

- In **Local operation to insert in tool**, select one of the grooves previously performed.



Note: To select a machining, simply select a geometry produced by the machining: side edge, back face…
If a geometry belongs to several machinings, it is possible to select the machining to be inserted.

- Confirm the default **local operation name**.
- Then select the three other grooves.

Note: The selected operations are shown in red.

- Once the four grooves have been selected, click on **Stop** to validate.

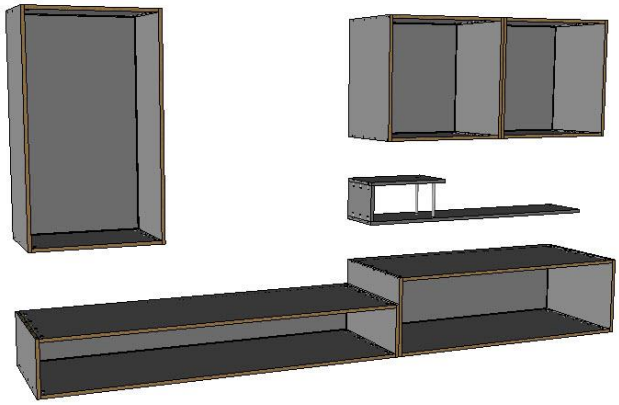
- Confirm the **Operation as tool definition** window without changing anything.
- Make **layer 1** current, then switch off **layers 0** and **2**.
- **Save** and **close** the document.

Exercise 4: Using the components in an interior design

The goal of this exercise is to create an interior design, and then insert the standard components previously created.

Concepts addressed:

- Importing a DWG file
- Using the layers
- Making the room to fit out in 3D from 2D
- Inserting standard components



Importing and using a DWG file

Import the file

Importing a file in another format can be done directly using the **Open** function of TopSolid.

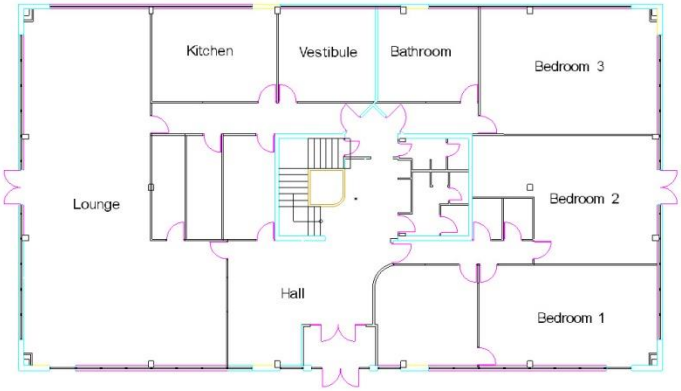
- Start **File** > **Open** and click on the file *Missler.DWG*. 
- Always in the Open File window, set **Open as**: **TopSolid'Design – Design**.

Note: It is possible to configure how to open a DMG file using the **Configure** button. 

- Then in the dialog bar set the file's import parameters:

  - **Unit**: Set the unit to use for this document (m, mm, Km…).
    Select **Unit = Automatic** to automatically detect the unit of the document.
  - **Standard**: Set the type of drawing standard to use for this file (Iso, Ansi…).
    Select **Standard = Automatic** to automatically detect the standard of the document.

- Click on **OK** to validate the parameters and close the Import Options window.

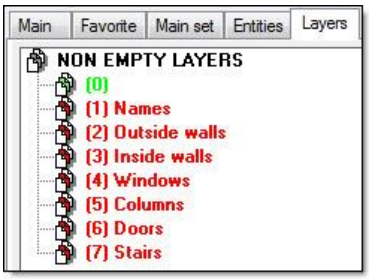


Note: When imported, the file is automatically converted to the **TopSolid** format (.top) in a new document.

- Save this document in a folder called *Interior design* and rename it *Interior design*.
- Answer **No** to the **Part definition** window.

Explore the imported file

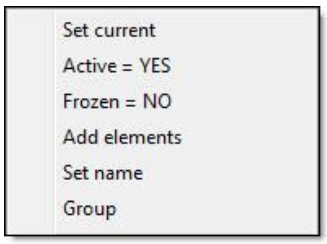
Note: When a DXF or DWG file is imported in TopSolid, the drawing colors are automatically retrieved. Moreover, the overlays are imported on the TopSolid **layers**. The names of these overlays are also imported via the names of the **layers**.



- Open the construction tree using **Ctrl + ²**.
- Go to the **Layers** tab.

Note: The **Layers** tab of the construction tree displays the different layers of the document.

Right-clicking on a layer allows you to make it **current** or **active**, give it a **name** or **add elements**.



By default, only the **non-empty layers** are displayed.

- **Right-click** > **Definition** on the first line **Non empty layers** and set:
  - **Display = All layers**.
  - **Sort = Numerical** and click on **OK** to confirm.

Make the walls

- Make **layer 9** current and name it *Walls*.
- Switch off **layer 5** by double-clicking on it.

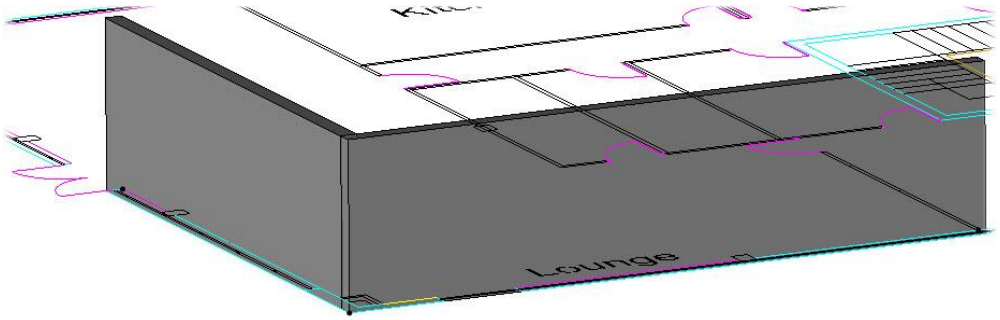
- Start the **Extruded** function from the **Shapes** context. 
- Set **New contour = Sketch** and select **New contour**.

- Then select **Rectangular**.

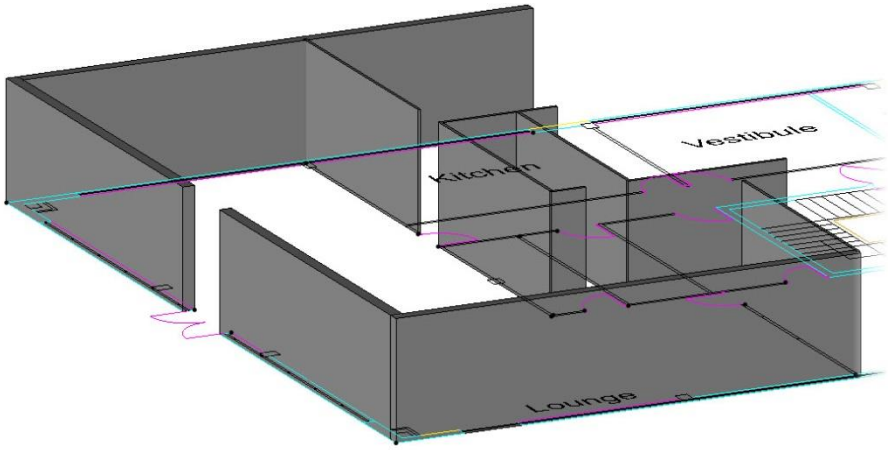
- Select the bottom left-hand corner of the **lounge** as the **first diagonal point**.

- Select the intersection point between the exterior wall (light blue) and the interior wall (black) as the **second diagonal point**.

- Extrude this wall in the **Z+** direction to a **height** of **2500mm**.
- Repeat this operation to extrude a second wall perpendicular to the first one to the double door (purple).

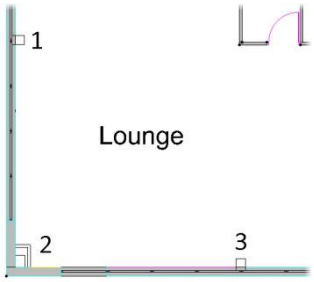


- Make several walls of the house in the same way.



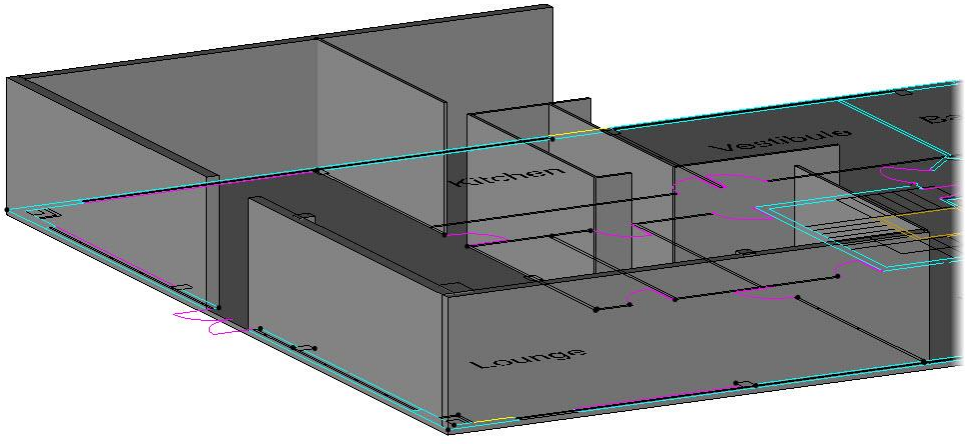
Make the columns

- Use the same method to extrude the 3 columns of the lounge.



Make the ground

- Make **layer 10** current and name it *Ground*.
- Use the same method to extrude the ground on the whole plane in the **Z-** direction to a **height** of 100mm.



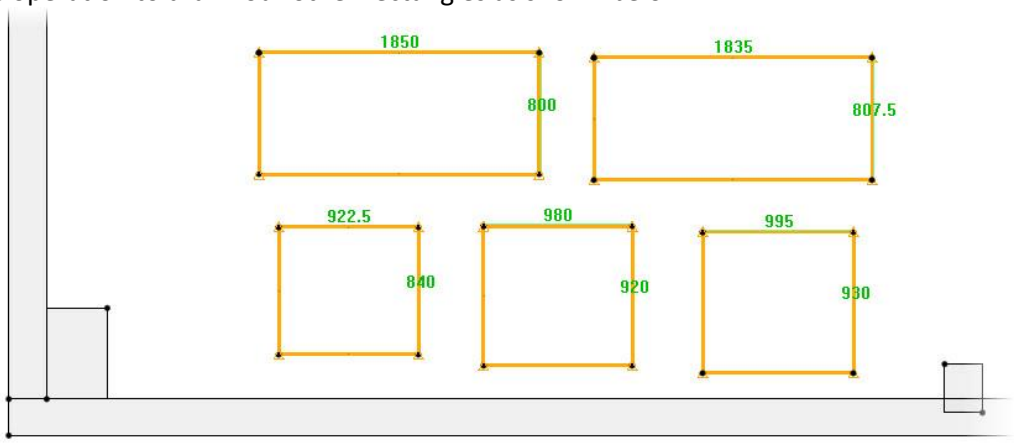
Using the standard components

Draw the construction volumes

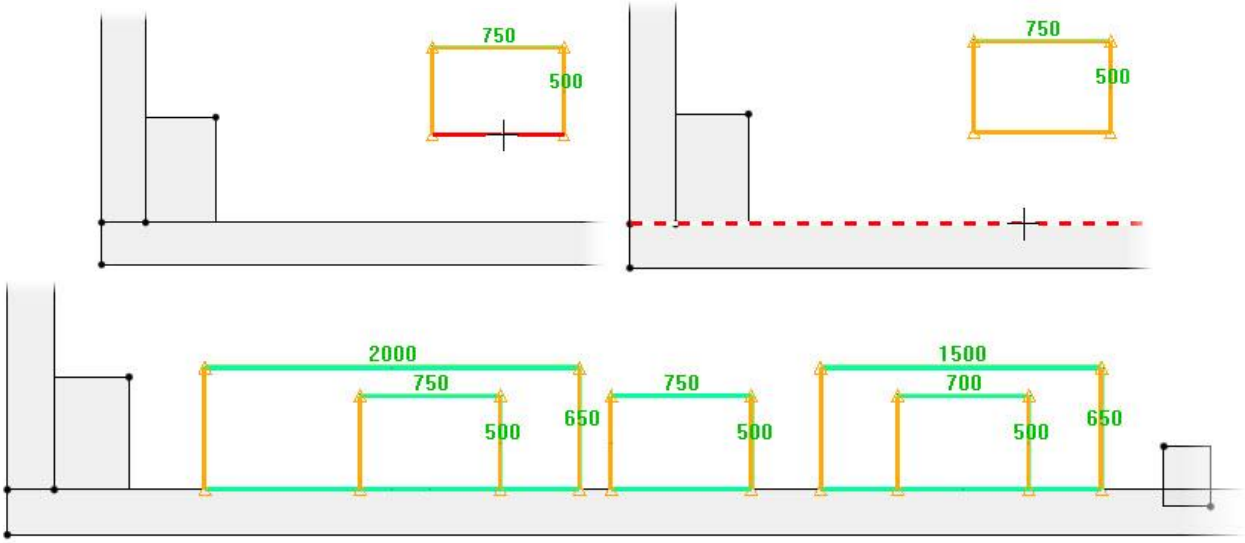
- Make **layer 11** current and **name** it *Construction volumes*.
- Switch off **layers 0** to **8**.
- Create a **new sketch**.
- Use **Contour** > **Rectangular** to draw a rectangle. 

- Then click on **Auto dimension** to automatically place the dimensions of the rectangle.

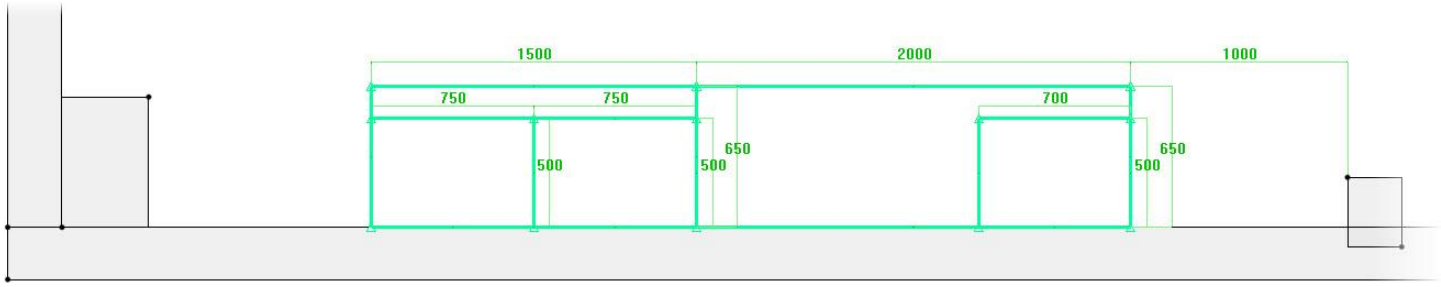
- The next dialog is used to place symmetry constraints on the dimensions. Click on **OK**.
- Repeat the operation to draw four other rectangles as shown below.



- Use **Modify parameter** to set the rectangle dimensions to the following values:
  - X=2000; Y=650
  - X=1500; Y=650
  - X=750; Y=500
  - X=750; Y=500
  - X=700; Y=500
- Then place an **alignment constraint** between the lower segments of the rectangles and the inner edge of the low wall.



- Place a dimension between the rectangle of 2000mm and the right-hand column to a **value** of 1000mm.
- Place **alignment constraints** between the rectangles to obtain the following result.

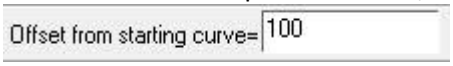


Extrude the construction volumes

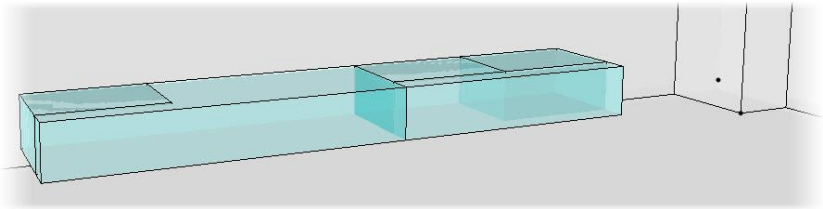
- **End the sketch**.
- Change the **design color** to **cyan**.
- Start the **Extruded** function from the **Shapes** context.
- Set **Generatrix sketch = Global** and **Result = One shape per profile**.

Note: **Generatrix sketch = Global** is used to extrude any sketch in one go.
Result = One shape per profile is used to obtain a distinct shape per drawn contour.

- Then select the sketch.
- In the advanced parameters **>>**, set **Offset from starting curve** = 100mm, then click on **OK** to confirm.



- Extrude the blocks in the **Z+** direction to a **height** of 300mm.
- Using **Attribute** > **Transparency**, apply a transparency of **7** to the extruded blocks.

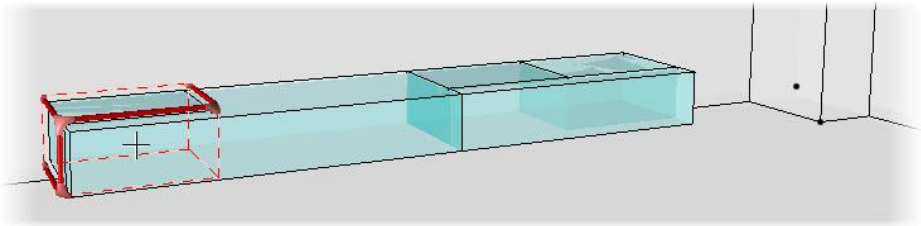


Modify the construction volumes

- Open the construction tree using **Ctrl + ²**.
- In the **Main** tab, **right-click** > **Edit**.

Note: The Edit function is used to modify an element in the construction tree. This makes it easier to find all the elements and parameters used by this element.

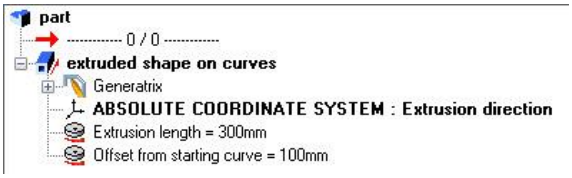
- Select the left-hand construction volume as the **element to edit**.



The extruded block is now open in the construction tree.

- Click on **+** to develop **extruded shape on curves**.

All the elements that were used to create the extruded shape are shown here:

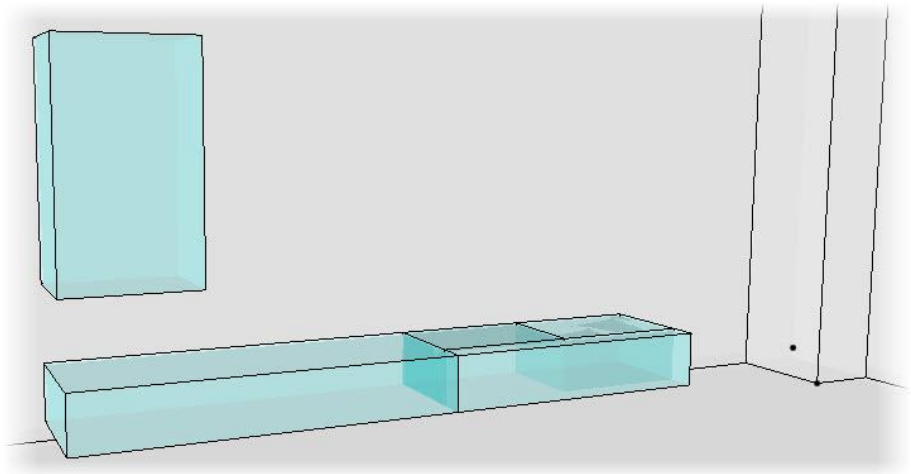


- The **generatrix**: This is the sketch used for the extrusion.
- The **absolute coordinate system**: This is the coordinate system which allowed the definition of the extrusion direction.
- The **extrusion length**.
- The **offset from starting curve**.

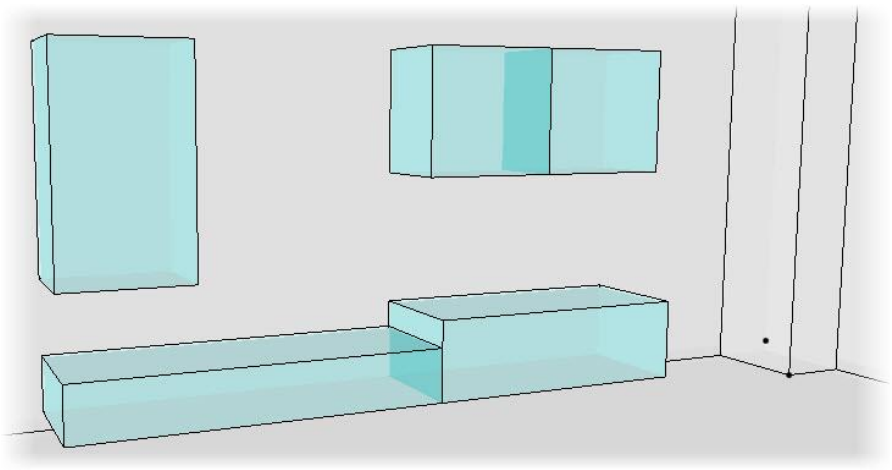
Note: All the construction volumes have been extruded in a single operation. This means that the **extrusion lengths**, as well as the **offsets from starting curves** are merged between the blocks.
To modify a volume without changing the others, the parameter must be **replaced**.

Note: It is also possible to modify an extruded shape locally without changing the others using the **Modify element** function. 

- On **Offset from starting curve**, **right-click** > **Replace**.
- Set **Replacement = Local** and **Replacement parameter** = 800.
- Also replace the **extrusion length** by 1200.



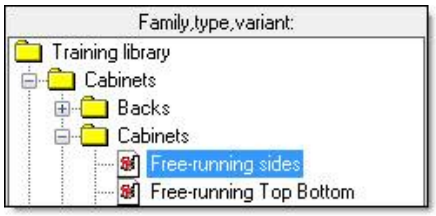
- **Edit** the two identical right-hand blocks:
  - **Replace** their **extrusion length** by 700mm
  - **Replace** their **offset from starting curve** by 1300mm.
- Finally, **edit** the right-hand block to **replace** its **extrusion length** by 450mm.



Include the standard components

- Make **layer 12** current and **name** it *Cabinets.*

- To include standard library components, use the **Include standard** function from the **Assembly** context. 
- In the **Standard component inclusion** window:
  - Select **Training library** in the dropdown list.
  - Select **Cabinets** > **Cabinets** > **Free-running sides**.
  - Click on **OK** to confirm.



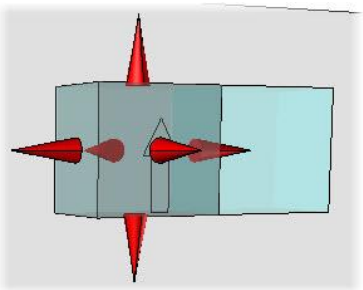
The **Cabinet** component is a **Driver block** component. When including this component, you will be asked to select the destination volume of the **Driver block**.

- Set **Housing mode** = **Inside a block** and **Hide block = NO**.

Note: The **Housing mode = Inside a block** mode has been set by default in the **Driver block** component to avoid setting it during its insertion. It is therefore possible to modify its inclusion mode.
The **Hide block** option automatically hides the construction volume used.

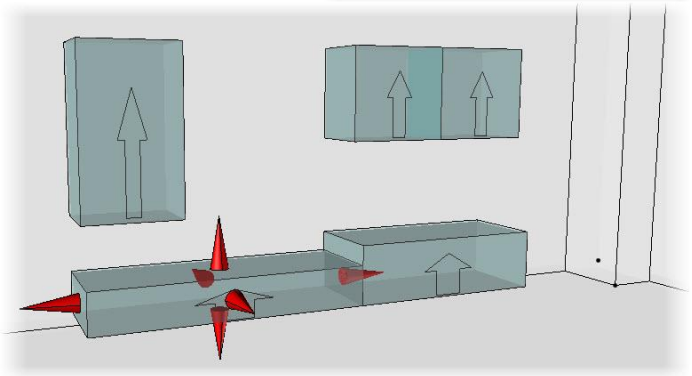
- Select the front face of one of the construction volumes as the **cabinet block**.

The **Driver block** is automatically included in the construction volume.

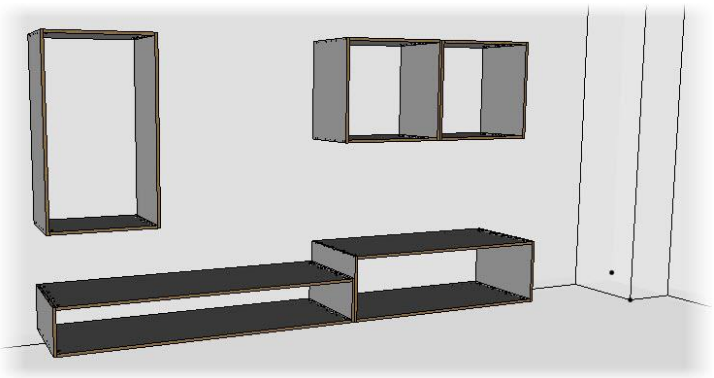


Note: The red arrows are used to modify the support planes of the driver block.

- Set **Multiple inclusion = YES** to include the same **Driver block** component several times.
- Select the front faces of the four other construction volumes, and then validate with **OK**.



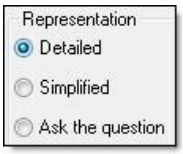
- Set **Thickness of panels = 19mm** and click **OK**.
- Switch off **layer 11** to hide the construction volumes.



Set the display mode

Note: By default, the components of a standard library are inserted in **detailed representation**.
As it is not recommended to work in this representation, the representation mode of the inserted elements must be changed.

Note: To insert the standard components in **simplified representation**, select **Tools** > **Options** > **Components** > **Components management** > **Representation**: **Simplified**.



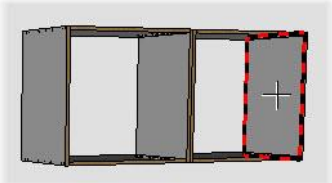
- Open the construction tree using **Ctrl + ²**.
- In the **Main set** tab, right-click on the line **ASSEMBLY**, then click on **Representation**.
- Set **Representation** = **Simplified**.

Note: **Representation = Mixed** means that some elements are in **simplified** mode and others in **detailed** mode.

Include the groove backs

- Use **Include standard** to include the standard component **Back grooves**. 
- Set **Housing mode = Housing**.

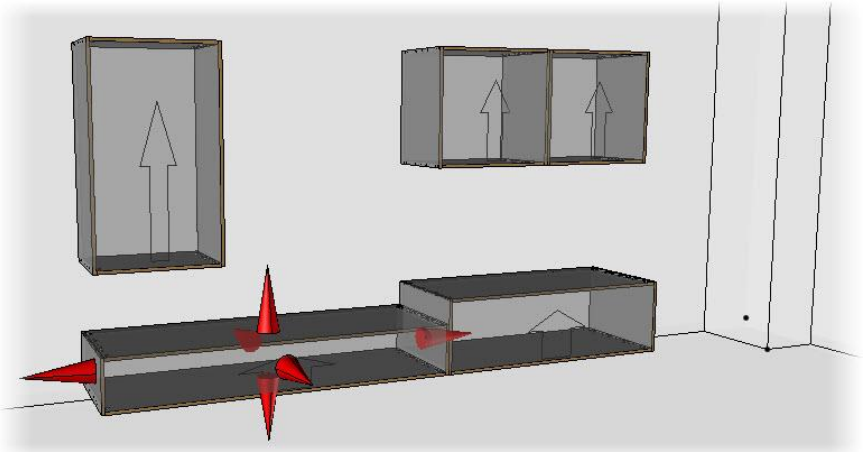
Note: To insert a **Driver block** component in housing mode like inside a cabinet, simply select one of the 4 inner faces of the cabinet.



- Select one of the inner faces of a cabinet as the **cabinet block**.

The **driver block** is automatically included in the inner volume of the cabinet.

- Set **Multiple inclusion = YES**, then position the **driver block** in the four other cabinets.



- Click on **OK** to validate **the driver blocks**.
- Then create the processes of the **Back** component using the **Automatic** function.

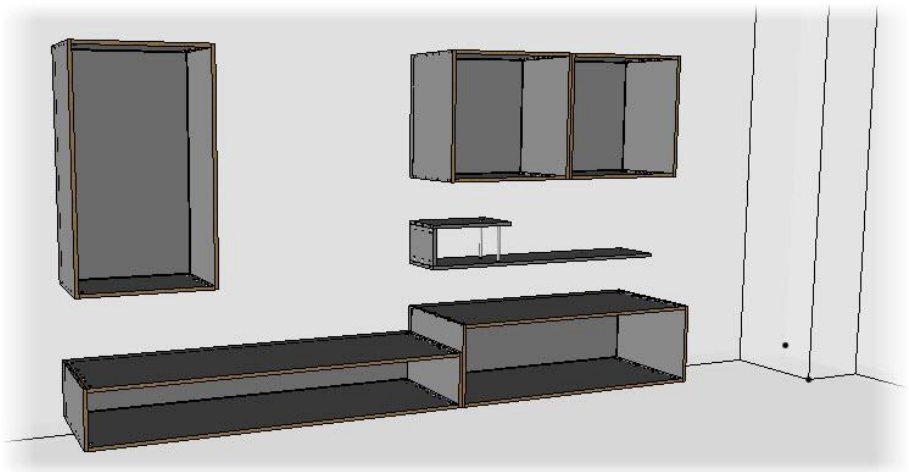
Note: To make the processes of a component after including it, use **Assembly** > **Use process** and select the component with the processes.
The **Automatic** processes are only made on defined parts.

Include the shelf

- Use **Include standard** to include the **Wall shelf** component. 
- Set the different parameters:
  - **Shelf length** = 1500mm
  - **Shelf depth** = 300mm
  - **Shelf height** = 250mm
  - **Panel thickness** = 19mm

The **Wall shelf** component is calculated, and then inserted according to the parameter values that you entered.

- Left-click in the space to release the component.
- Then place three constraints on this component to position it:
  - Rear edge of the shelf / Wall, **Distance** = 0mm.
  - Base of the shelf / Top of the lower right-hand cabinet, **Distance** = 250mm.
  - Left-hand side of the shelf / Left-hand side of the lower right-hand cabinet, **Distance** = 0mm.

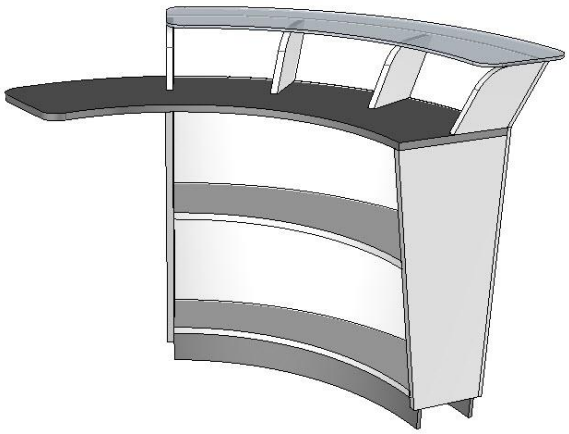


Define the set

- Define the set using **Wood** > **Define** > **Define set** > **Characteristics**:
  - **Designation**: *Lounge fitting.*

Exercise 5: Creating a counter with geometric drivers

The goal of this exercise is to create a "Counter" component driven by a path, and then insert it in the interior design.



Concepts addressed:

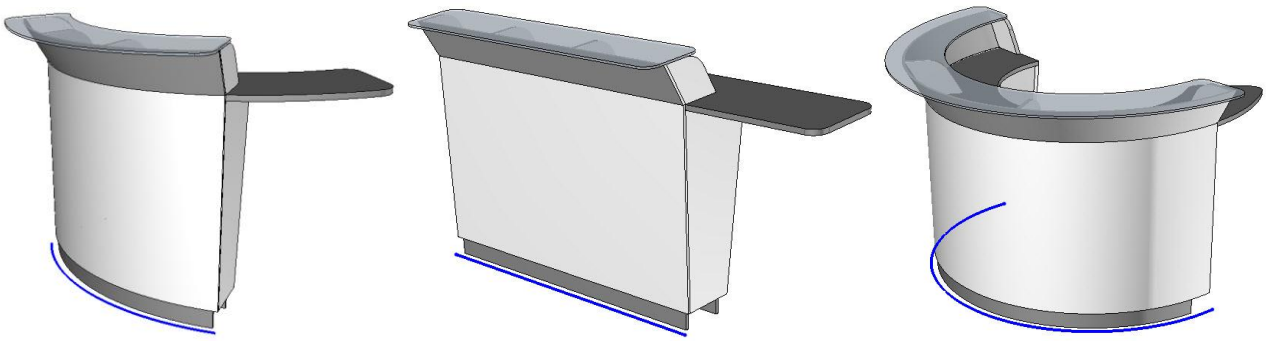
- The component driven by a path or ***Component with geometric drivers***
- The pipe shape
- The trimming by planes
- The unbent parts

Creating the component with geometric drivers

Note

The component driven by a path or ***Component with geometric drivers*** is used to create a component which will take the desired shape when inserted in an assembly.

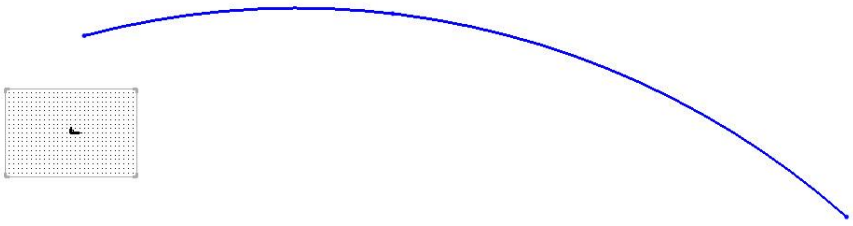
Here, the **Counter** component is designed according to the blue driving path.
This means that any line or circle arc can be defined as a driving path when inserting the component. The component will take the desired shape.
Below are two examples that show the insertion of the left-hand component on a line and a circle arc.



Create the driving path

- Create a **new Design document**.
- Change the **design color** to **blue (12)**.
- Start the **Curves** > **Circle** function, and then draw a circle arc in **Passage point** mode.

Note: The circle arc must be rather large to facilitate the design.



- Start the **Edit** > **Break associativity** function.

Note: **Breaking the associativity** of an element breaks the links it has with its construction elements. It is therefore completely independent.
This operation should be performed on the driving path of the component to avoid any errors when assembling.

- In **Elements to break their associativity**, select the circle arc.

The circle arc then becomes completely independent.

- Start the **Edit** > **Name** function, and then select the circle arc.
- Enter **Name**: *t* and **Designation**: *Trajectory.*

Note: **Naming** an element makes it easier to find it in the design.

- Make **layer 1** current.
- Start the **Tools** > **Coordinate system** > **Coordinate system on curve and point** function. 

Note: A coordinate system on curve and point is used to automatically place a coordinate system perpendicular to the path of a curve passing through a point.

- Select the circle arc as the **reference curve**, then the right end of the circle arc as the **origin point**.
- Select **Quit**, and then **Set as current** to be positioned along this coordinate system.

Create other parameters

- Using the **Parameter** > **Create** function, create the parameters shown opposite in **Unit type = Length** mode.

Name	Designation	Value
td	Trajectory shift	100mm
h	Top height	800mm
sh	Shelf height	350mm
p	Top extension	500mm

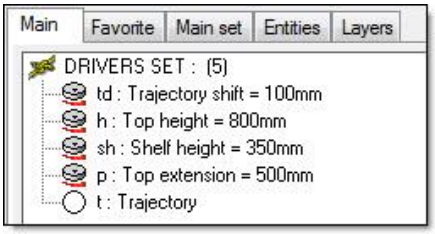
Driver
Yes
Yes
Yes
Yes

- Open the list of parameters using **Parameter** > **Edit list**.
- Make the parameters as drivers by changing **No** to **Yes** in the **Driver** column.

Define the path as driver

- Start the **Assembly** > **Define component** > **Define drivers** function.
- Enter **Driving element to insert**: ***t*** and press **Enter** to confirm.
- Confirm the default **designation of the driving element**.

The ***t*** path is now a driver of the component.



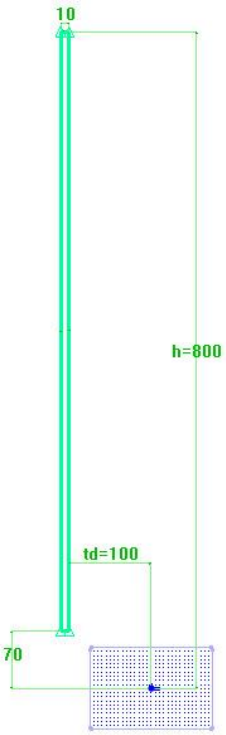
Save the component

- Save the component in the library: **Assembly** > **Define component** > **Edit/save template** > **Save standard template**.
- Set:
  - **Standard**: **Training library**
  - **Family**: **Furniture**
  - **New type**: *Counter*
  - **New variant**: *Interior counter*
- Click on **OK** to validate the window.

Drawing the parts

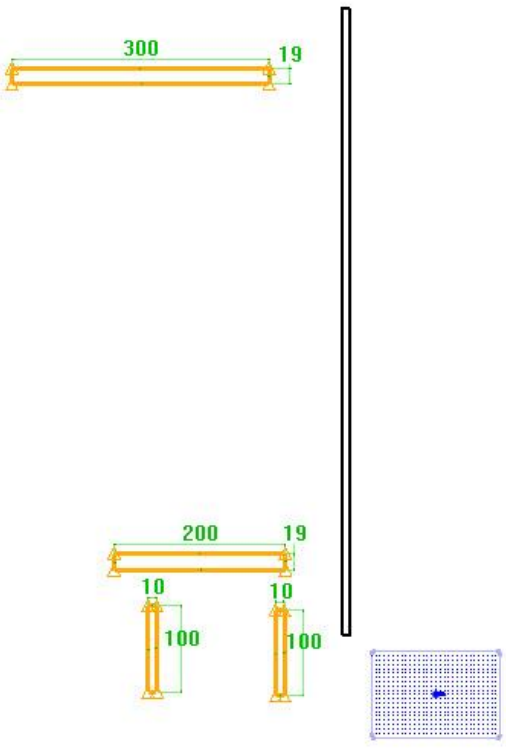
Draw the bent front

- Switch off **layer 0** and set the **design color** to **black**.
- Start a **new sketch** and draw a **rectangular contour** to the left of the coordinate system. Use **Auto dimension** to automatically apply the dimensions.
- Delete the height dimension of the rectangle. 
- Use **Modify parameter** to modify the width of the rectangle to 10mm**.** 
- Then dimension: 
  - from the top of the rectangle to the X axis with a **nominal value** = h;
  - from the bottom of the rectangle to the X axis with a **nominal value** = 70mm;
  - from the right-hand side of the rectangle to the Y axis with a **nominal value** = td.
- **End the sketch**.

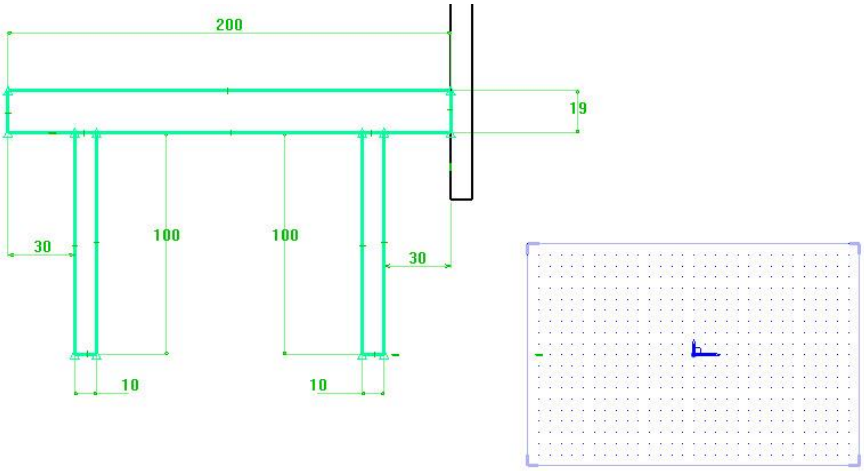


Draw the tops and the baseboards

- Make **layer 2** current.
- Draw four **rectangular contours** as shown opposite in **Orientation constraint** mode. Use **Auto dimension** to place the dimensions automatically.
- Use **Modify parameter** to modify the rectangle dimensions to the dimensions shown opposite.

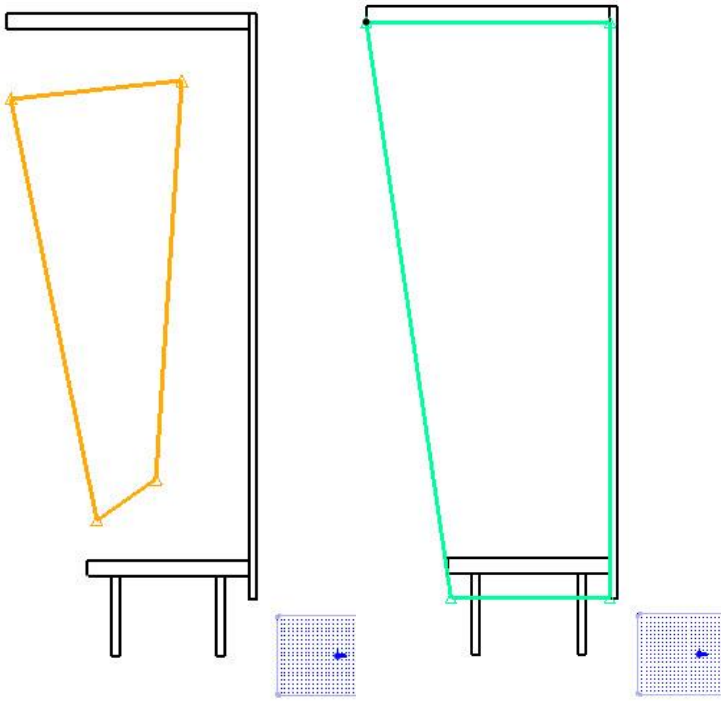


- Apply:
  - an **alignment constraint** between the bottom of the baseboards (rectangles 100x10) and the X axis;
  - an **alignment constraint** between the bottom of the base (rectangle 200x19) and the top of a baseboard;
  - an **alignment constraint** between the right-hand side of the base and the left-hand side of the bent front;
  - a **coincidence constraint** between the top right-hand point of the top and the top left-hand point of the bent front.
- Then dimension the shift of the baseboards in relation to the base to 30mm**.**

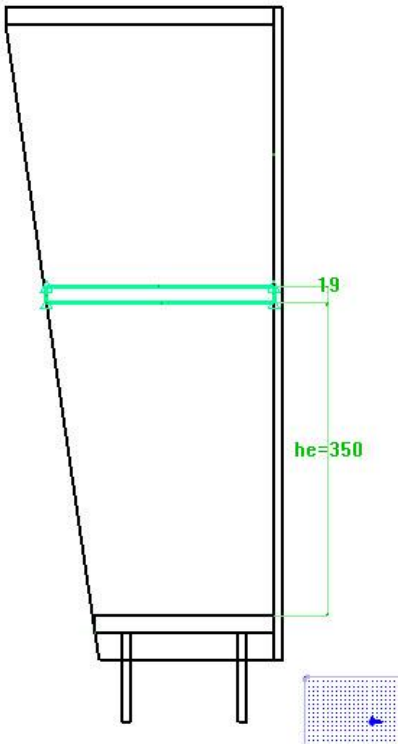


Draw the side

- Make **layer 3** current.
- In a **new sketch**, draw a four-segment **contour**.
- Apply:
  - a **coincidence constraint** between the two upper points and the two lower points of the top.
  - a **coincidence constraint** between the bottom right-hand point and the bottom left-hand point of the bent front
  - an **orientation constraint** of the lower segment **along X**
  - a **coincidence constraint** between the left-hand segment and the bottom left-hand point of the base
- **End the sketch**.



Draw the shelf



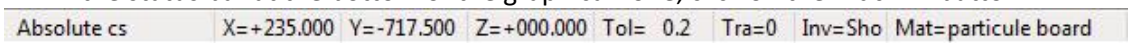
- Make **layer 4** current.
- Start a **new sketch** and draw a **rectangular contour**.
- **Dimension** the height of the rectangle to 19mm**.**
- Place an **alignment constraint** of the right-hand side of the rectangle on the left-hand side of the bent front.
- **Dimension** the distance between the bottom of the shelf and the top of the base. Set **Nominal value** = sh.
- Place a **coincidence constraint** of the bottom left point of the rectangle on the left-hand segment of the side.
- **End the sketch**.

Making the parts

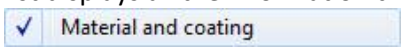
Configure the current material and coating

Note: It is possible to define a current **material** and a current **coating** which will be used by the new extruded shapes.
When creating a number of parts with the same material and coating, this avoids you to set the material and the coating for each part.

- In the status bar at the bottom of the graphical zone, click on the **Mat = …** button.



Note: This box shows the default material.
If the **Mat = …** button is not displayed, right-click on the status bar and select **Material and coating** in the list. This list displays all the information that can be shown in the status bar.



- Once you clicked on the button, set the current material and coating:
  - **Current material**: **wood** > **particule board**.
  - **Current coating**: **paint** > **glossy white paint**.
- Click on **OK** to confirm.

Curving the low parts

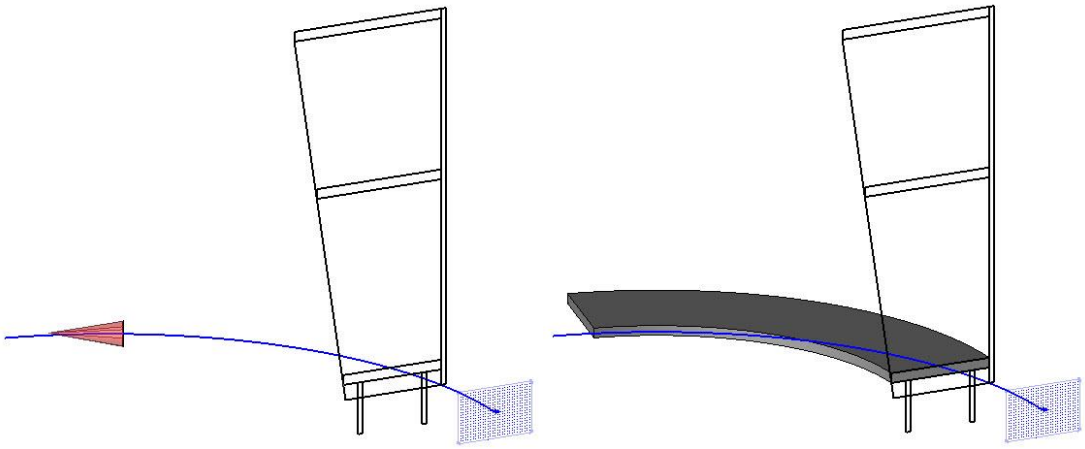
Note: The **pipe shape** can be used to extrude a 2D (**Section curve**) along a path (**Guide curve**).

- Start the **Shape** > **Pipe** function. 
- Set **Pipe = On curves** and **Follow = Subsequent operations**.

- Switch on **layer 0**, and then select the blue driving path as the **guide curve**.
- Adjust the red arrow so that it goes from the sketch to the path. This arrow is used to set the pipe shape direction.
- Set **Corners type = Rounded**, **Curves = Visible**, **Generatrix sketch = Local** and **Type = Solid**.

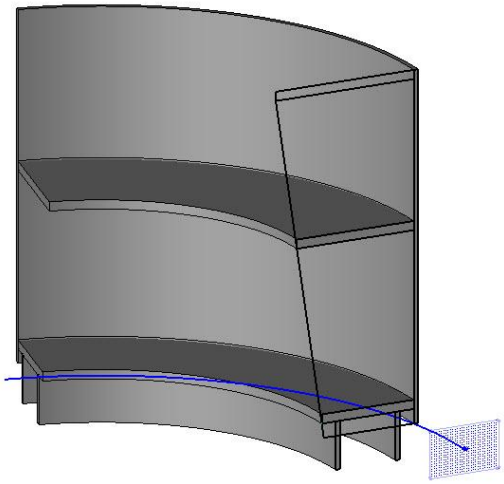
Note: **Corners type = Rounded** is used to round the pipe shape when the guide curve has a sharp corner.
Curves = Visible keeps the curves visible on the screen after creating the pipe shape.
Generatrix sketch = Local is used to select only one contour when a sketch contains several contours.

- In **section curves or texts**, select the sketch of the base.

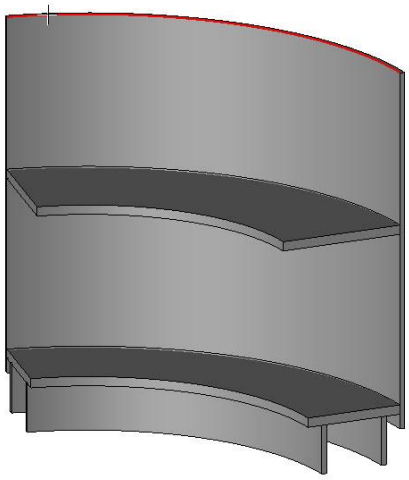


- Repeat the operation to create pipe shapes for the two baseboards, the shelf and the bent front.

Note: The **Curves = Invisible** mode is always used by default. For this exercise, set the mode to **Curves = Visible** at each use.



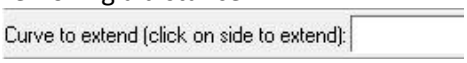
Curving the counter top



Note: As the top is longer than the rest of the counter, its guide curve must be extended to the desired length.

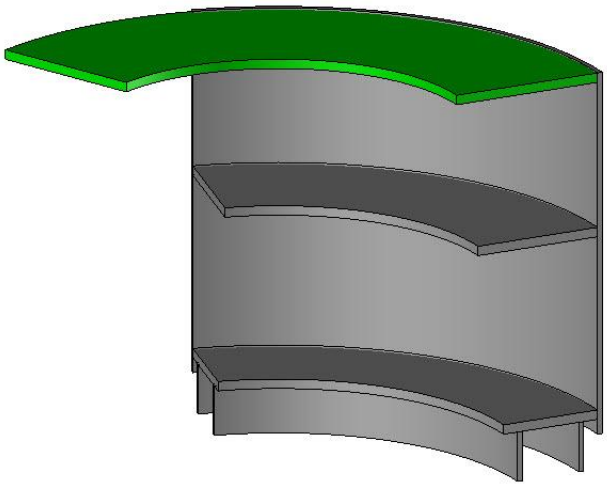
- Make **layer 5** current and switch off **layers 0** to **3**.
- Change the **design color** to **green (10)**.
- In the **Curves** context, start the **Copy edge** function.
- Set **Mode = Edge**.
- Select the upper inner edge of the bent front as the **edge to copy**.
- Start the **Curves** > **Extend** function, and then select the copied green curve of the left-hand side.

Note: The **Extend** function modifies the length of a curve by adding or removing a distance.



- Set **Type = Curvature**, **Mode = Extend length** and **Length** = *p* to set the length of the extension.

- Click on **OK** to confirm.
- Switch on **layer 2**, and then create the **pipe shape** of the top by selecting the extended green curve as the **guide curve**.

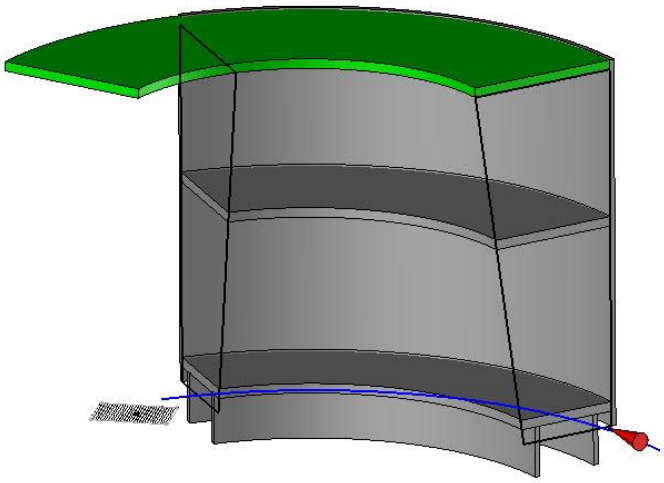


Repeat a sketch and extrude the sides

- Make **layer 6** current and switch on **layer 3** containing the sketch of the side.
- Start the **Current coordinate system** function  in order to change the current coordinate system, and then select **Absolute coordinate system** as the **named coordinate system**. 

Note: Since the **absolute coordinate system** is on **layer 0**, this layer is automatically turned on.

- Start the **Edit** > **Repeat** function.
- In **Template elements to repeat**, select the sketch of the side.
- Set **Propagation = On curve**.
- Select the blue driving path shown opposite as the **curve to propagate from the start**.



- Set:
  - **Distribution mode** = **Distribute**
  - **Distance computing mode** = **Arc length**
  - **Transformation mode** = **Constraint coordinate system**

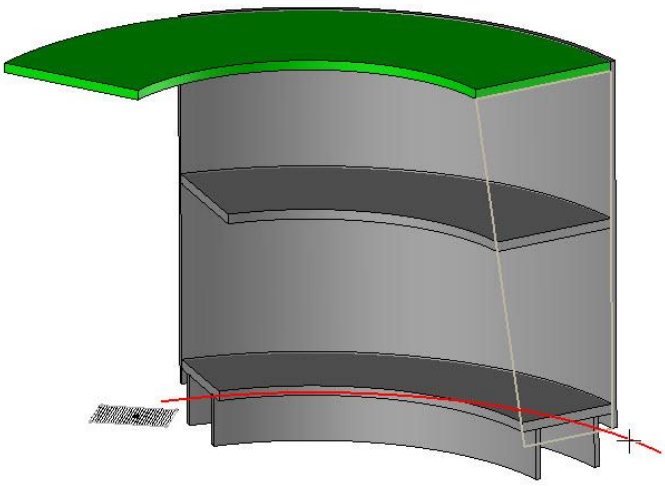
- Validate these parameters, and then enter **Number of instances**: **2**.

- Start the **Extruded** function and set **Extruded shape on = Curves, Generatrix sketch = Local** and **Result = One shape per profile**.
- In **Section curves or texts**, select the **Element detection** function. 

Note: The element detection is used to select only one of the elements belonging to an assembly or a repetition. In this case, in order to extrude only one sketch of the repetition, the detection should be used to specify the sketch to be extruded.

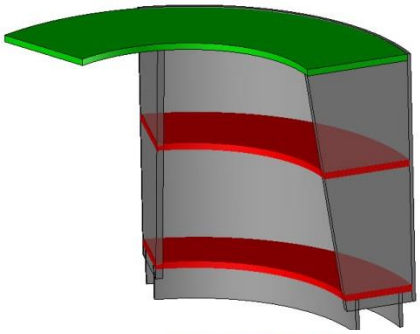
- Select the right-hand sketch as the **element for detection**, and then click on **OK** to confirm.

- **Extrude** this sketch towards the inside of the counter to a **height** of 19mm.
- Repeat the operation to extrude the second side.

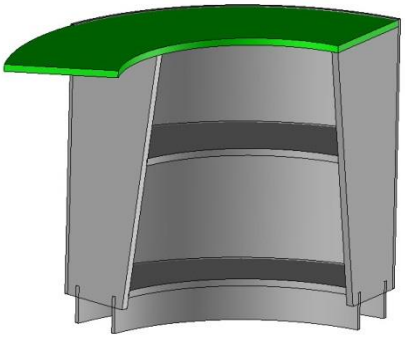


Trim the parts

Now let's trim the shelf, as well as the base in relation to the sides.

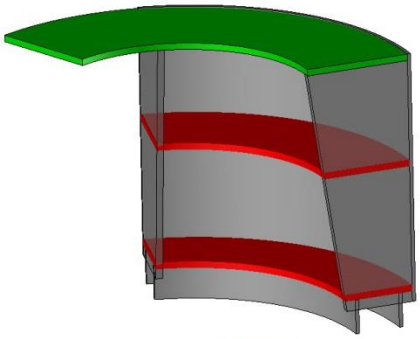


- Start the **Shapes** > **Trim** function. 
- Set **Trim = By plane**.
- In **Shape(s) to trim**, use the **selection** in order to select several shapes. 
- Select the shelf and the base, and then click on **OK** to confirm.
- Select the inner face of the side as the **trimming plane**.
- The red arrow represents the side of the material to be removed. Adjust the arrow so that it shows the outside of the counter, and then click on **OK** to confirm.
- Set **Hide tools = NO** and click on **OK** to confirm.
- Repeat the operation to **trim** the parts in relation to the inner face of the second side.



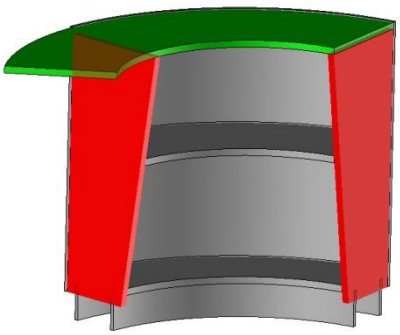
Subtract the parts

Subtracting the sides from the baseboards allows the notches to be made.



- Use **Shapes** > **Subtract**. 
- In **Shape(s) to modify**, use the **selection**  in order to select several shapes.
- Select the two baseboards, and click on **OK** to confirm.
- Set **Hide tools = No**.
- In **Tool shape(s) to use**, use the **selection**,  select the two sides and click on **OK** to confirm.

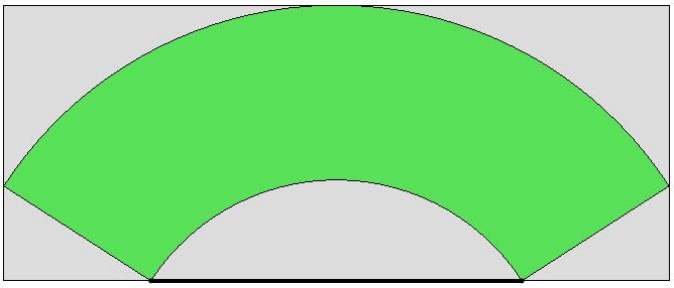
The sides are subtracted from the baseboards.

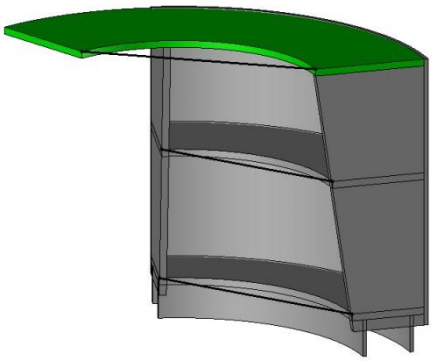


Defining the parts

Define the planar parts

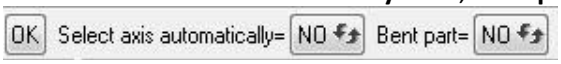
Note: Since the counter parts are curved parts, you must first draw the lines representing the lengths of the parts. This will allow the precise cut of the parts to be calculated.





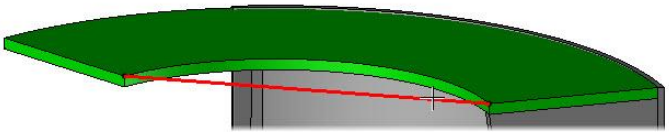
- Make **layer 7** current.
- Change the color to **black**.
- Use the **Curves** > **Line** function to draw the length axes of the top, the base and the shelf as shown opposite.

- Start the **Wood** > **Define** > **Define part** function. 
- Select the counter top as the **part to define**.
- Set **Select axis automatically = No**, **Bent part = No**, and then click on **OK** to confirm.

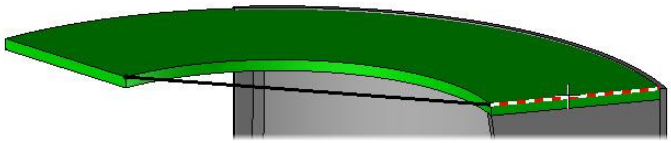


Note: Since the part is a curved part, the axes must be selected manually in order to control the cutting-up dimensions.

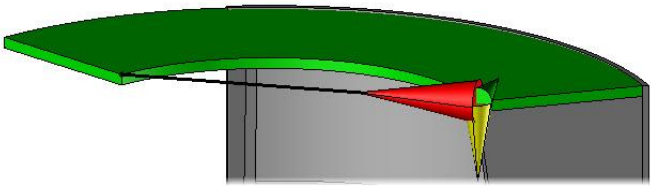
- Select the previously drawn line as the **length axis**, and click on **OK** to confirm.



- Then select the edge of the counter top width as the **width axis**, and click on **OK** to confirm.

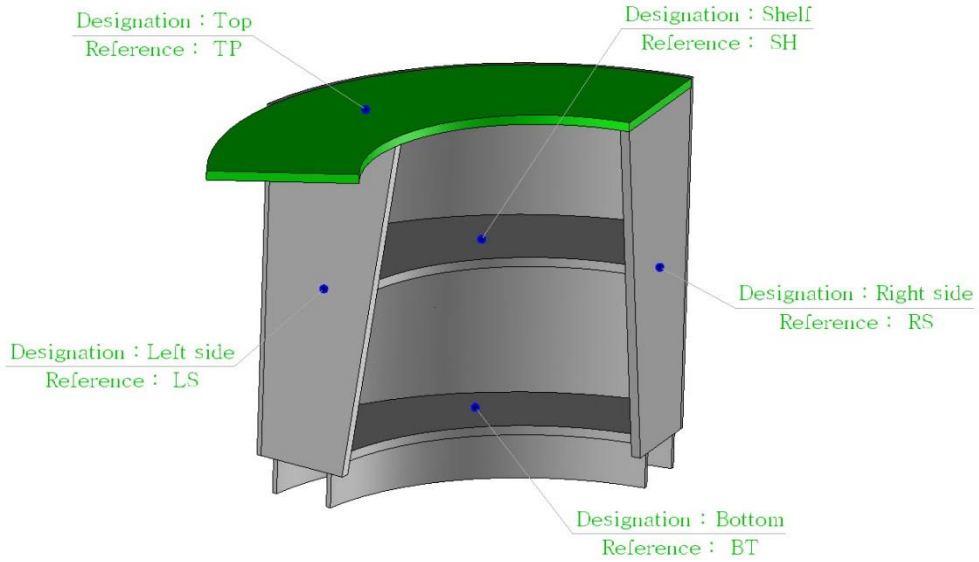


The cutting-up axes are placed on the part and the rectangular cut is calculated.



Sizes	Values	Modes	Over dimensions
Length	1140.0mm	additional	0mm
Width	473.2mm	additional	0mm
Thickness	19.0mm	additional	0mm

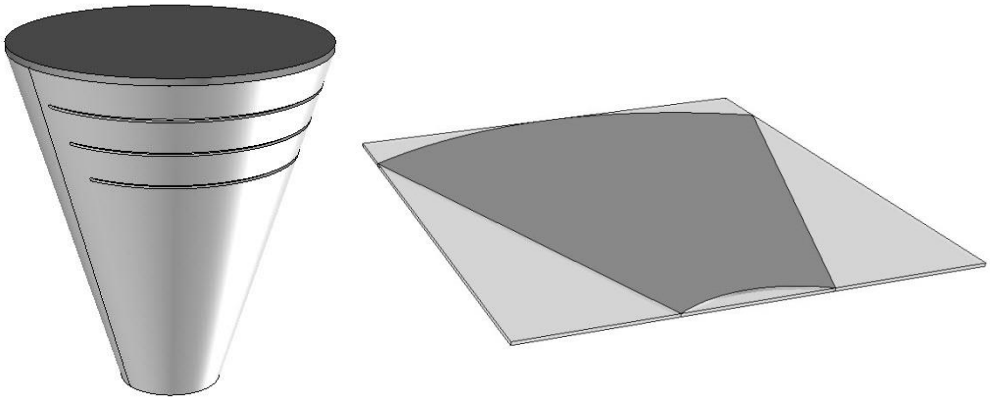
- Enter the following designations and references.



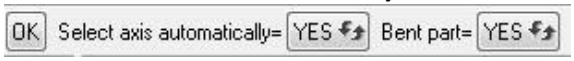
- For the top only, modify the **coating** to **paint** > **mat grey paint**.

Define the bent parts

Note: The baseboards and the bent front are **bent** parts. To calculate their precise cut, the parts must be unfolded.
Unfolding a part also allows this part to be machined before folding.



- Start the **Wood** > **Define** > **Define part** function, and then select the bent front of the counter.
- Set **Select axis automatically = YES** and **Bent part = YES**, and then validate with **OK**.



The settings for the unfolding calculation then appear.

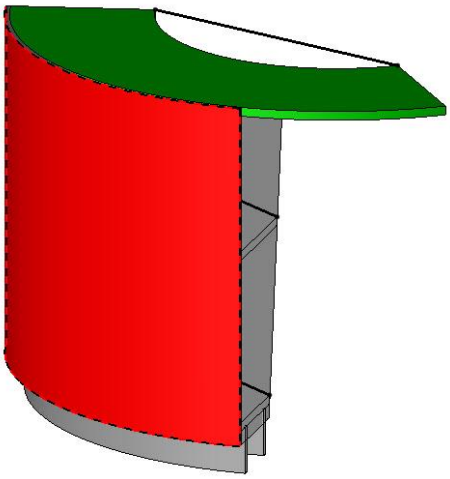
- Enter:
  - **Thickness** = 10mm
  - **Neutral fiber coefficient** = 0.5
  - **Level** = 8

Note: The **thickness** corresponds to the thickness of the part to be unfolded.
The **neutral fiber coefficient** corresponds to the position of the neutral fiber on the panel thickness. A value of 0.5 means that the neutral fiber is positioned in the middle of the part to unfold.
The **layer** corresponds to the layer on which you want to place the result of the unfolding.

- **Follow tangent faces = YES**
- **Drills on reference faces = NO**

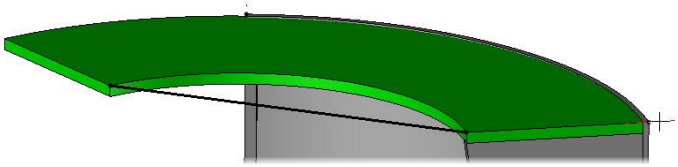
Note: It is possible to unfold several faces of a part in one go. The **Follow tangent faces** option is used to automatically select the faces that are tangent to the selected face.
The **Drills on reference faces** option applies the drillings on the reference face to the unfolding. Only the drilling operations will be applied to the unfolding.

- Select the outer face of the bent front as the **face to unwind**.
- Click on **OK** to confirm.



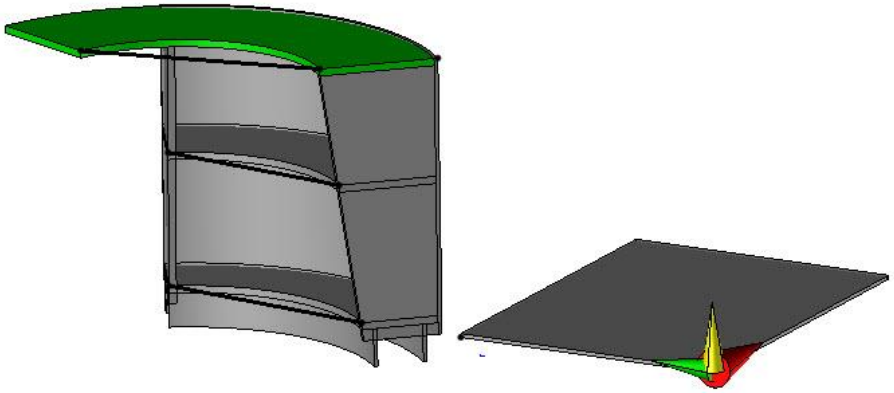
Note: To position the unfolding, an origin coordinate system, as well as a destination coordinate system must be selected.
It is possible to select one of the axes automatically placed on the part corners as the origin coordinate system.

- Select the axis at the top right of the bent front as the **origin coordinate system**.



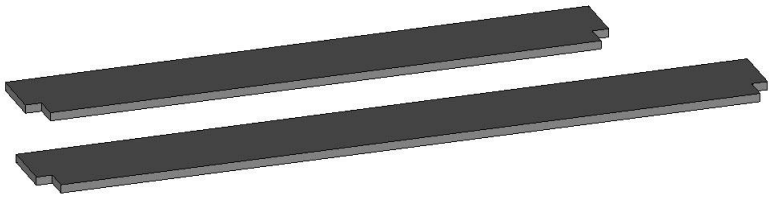
- Select any point to the right of the counter as the **destination coordinate system** to position the unfolding.
- Switch on **layer 8.**

Note: The unfolding can be moved on the point of the destination coordinate system later using the **Move parents** function. 



- In the **part definition**, enter:
  - **Designation**: *Bent front.*
  - **Reference**: *BF.*

- Click on **OK** to confirm.
- Repeat the operations to unfold the two other baseboards:
  - **Designations**: *Baseboard 1 / 2*
  - **References**: *BA 1 / 2*
  - **Coating**: **paint** > **mat grey paint**

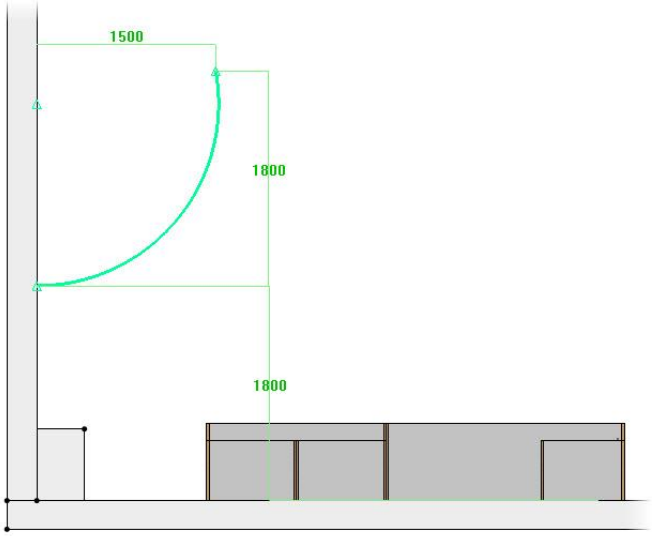


Define the set

- Define the set using **Wood** > **Define** > **Define set** > **Characteristics**:
  - **Designation**: *Interior counter*
  - **Reference**: *IC*
  - **Main set**: **Sub-assembly**
- Save and close this file.

Using the component

Create the destination path

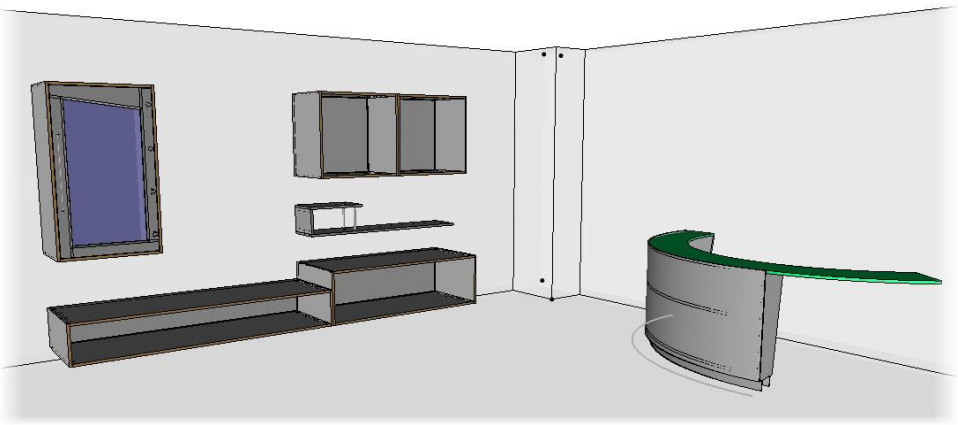


- Open the file *Interior design*.
- Make **layer 13** current.
- Set the **layer name =** *Counter.*
- Draw the sketch shown opposite.
  - The left-hand point of the arc is **coincident** with the left-hand wall.
  - The circle arc is **perpendicular** to the left-hand wall.
- Then place the **dimensions** as shown opposite.
- Finish the sketch.

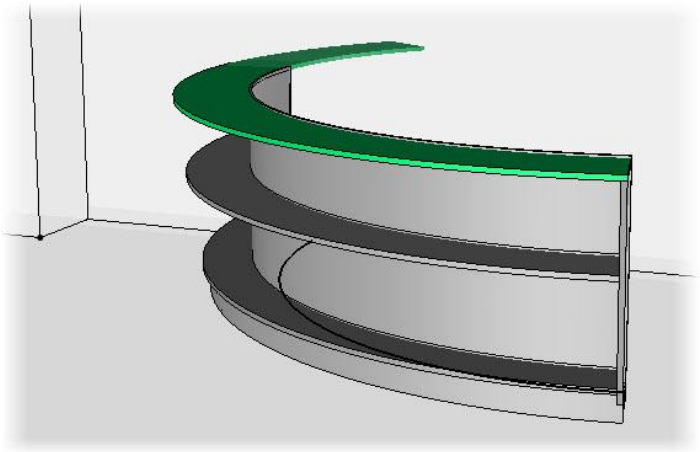
Insert the component

- Use the **Assembly** > **Include standard** function. 
- Select the component **Furniture** > **Counter** > **Interior counter**, then click on **OK** to confirm.
- Set as drivers:
  - **Trajectory shift** = 0mm
  - **Top height** = 800mm
  - **Shelf height** = 350mm
  - **Top extension** = 1000mm
  - **Trajectory**: Select the sketch created previously

The component is automatically calculated according to the parameters specified and the selected path.

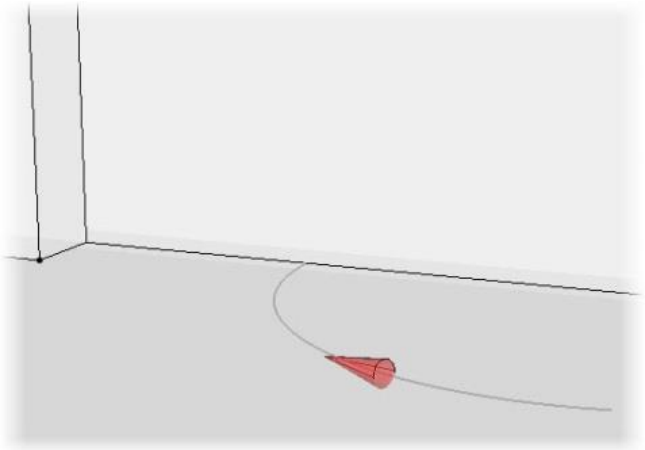


Note: Depending on the direction the circle arc is drawn, the component may be calculated on the other side of the circle arc.



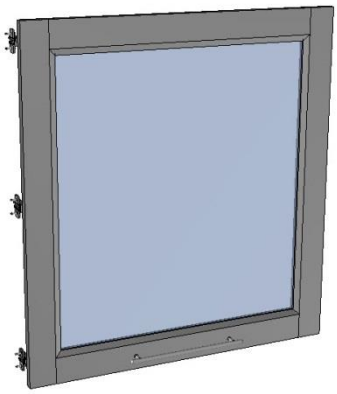
- To invert the path direction:
  - **Delete** the component. 
  - Start the **Curves** > **Origin** function, and then select the previously drawn sketch as the **curve to modify**.

The arrow that appears represents the curve direction. This direction is used to calculate the component.



- Invert the direction of the circle arc by clicking on the red arrow. Click on **OK** to confirm.
- Include the component again as shown above.

Exercise 6: Creating a door



The goal of this exercise is to create a "Door" component as **driver block**. To make this component, you must first create the moulding tool of the door, the hinges, as well as the handle.

Concepts addressed:

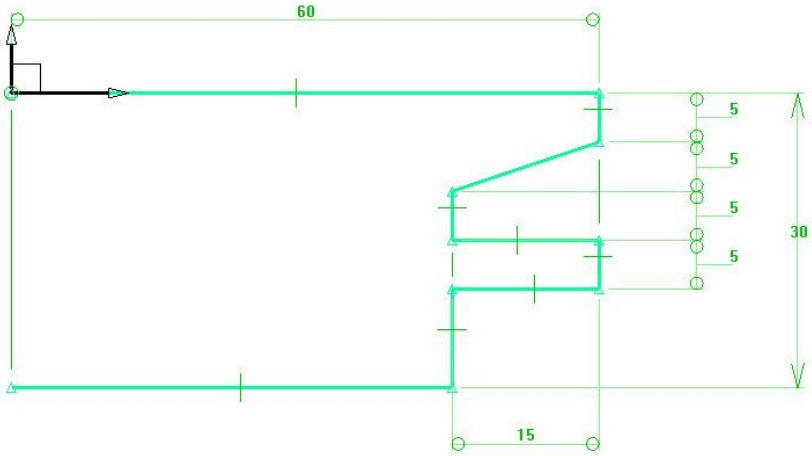
- Moulding/Counter moulding tool
- Door as driver block
- Use of 3D components (hinge and handle)

Creating a moulding tool

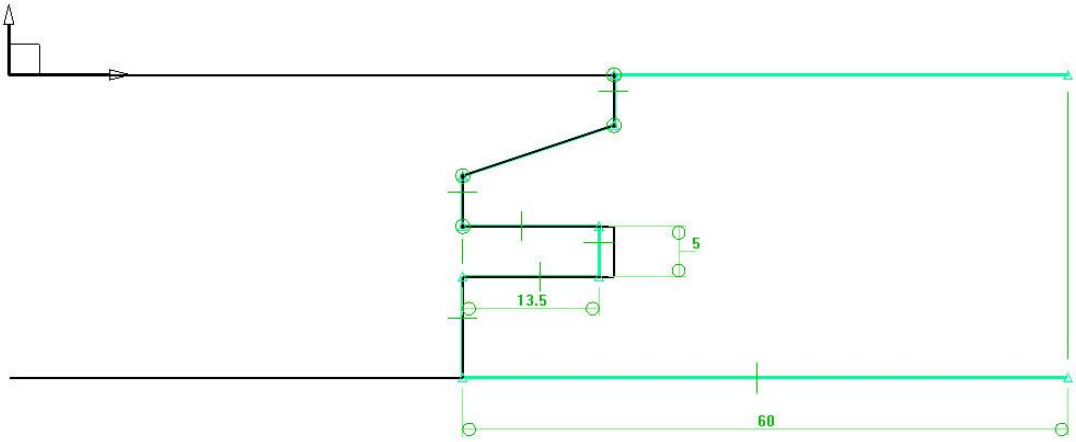
Note: Creating a moulding tool allows you to create your own tools available in production.

Create the profiles

- Create a **new Design document**.
- On **layer 1**, draw the sketch shown below:

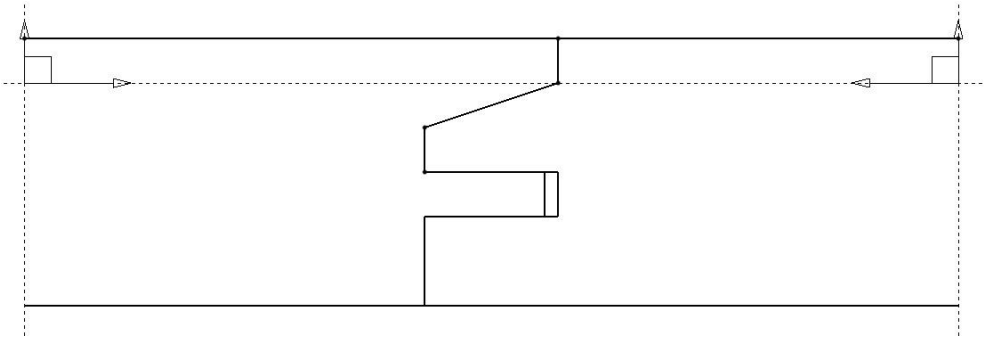


- End the sketch, and then draw the **new sketch** shown below.



Note: In order to obtain a correct result during the machining (**TopSolidWood'Cam** or **machining interfaces**), it is necessary to draw the precise profile of the tool (radius, teeth, height…). However it is recommended to simplify the tool path to optimize the software performance.

- Switch off **layer 0**, and then create the two **coordinate systems 2 axes** shown below using **offset points**.



- Save this file in the standard library using **Assembly** > **Define component** > **Edit/save template** > **Save standard template**:
  - **Standard**: **Training library**
  - **New family**: *Tools*
  - **New type**: *Moulding tools*
  - **New variant**: *Chamfer Panel moulding*

Define the tool

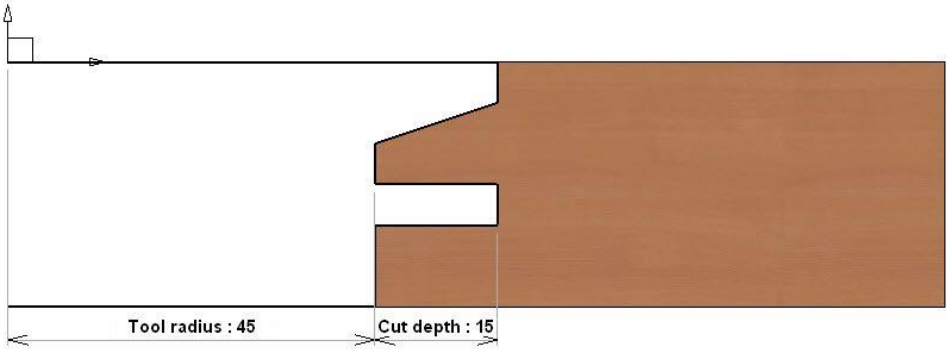
- To define the tool, start the **Wood** > **Define** > **Define tool** function. 
- Set **Mode = Moulding** and select the left-hand sketch as the **tool curve**.

- Enter:
  - **Tool name**: *tool1*
  - **Tool designation**: *Chamfer Panel moulding*
  - **Tool number**: *105*

Note: The **tool number** is the tool number which will be used during exports for machining (TopSolid'WoodCam or machining exports). This field can be left blank; the number used will be the one set by default.

- Click on **OK** to confirm the parameters, and then select the left-hand coordinate system as the **position coordinate system**.
- Enter:
  - **Position name**: *origin*
  - **Position designation**: *Tool origin*

- Set **Tool radius**: **45** and **Cut depth**: **15**.



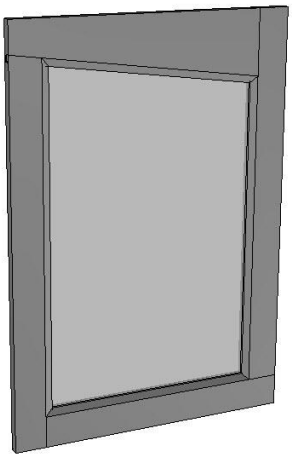
The tool is now defined and can be used to make the mouldings.

Define the counter moulding tool

- Start the **Wood** > **Define** > **Define tool** function. 
- Set **Mode = Counter moulding**, and then select the right-hand sketch as the **tool curve**.

- Enter:
  - **Tool name**: *tool2*
  - **Tool designation**: *Chamfer panel counter moulding*
  - **Tool number**: *106*
- Select the right-hand coordinate system as the **position coordinate system**.
- Enter:
  - **Position name**: *origin2*
  - **Position designation**: *Tool origin 2*
- Set **Tool radius**: **45** and **Cut depth**: **15**.
- Save and close this file.

Creating a door component



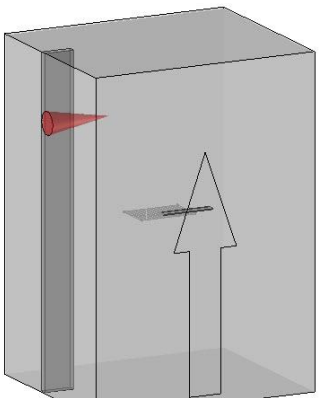
Create the parameters

- Create a **new Design document**.
- Create a **driver block** using **Assembly** > **Define component** > **Define drivers** > **Driver block**.
- Set:
  - **Name of driving element**: *db*
  - **Default housing mode = Housing**
  - Select **Current coordinate system**
  - **Designation of the driving element**: *Cabinet block*
- Create the 3 parameters below in **Unit type = Length** mode:
  - **Value**: 20; **Name**: *cth*; **Designation**: *Crosspiece thickness*
  - **Value**: 80; **Name**: *cwi*; **Designation**: *Crosspiece width*
  - **Value**: 2; **Name**: *dg* **Designation**: *Door gap*
- Using the **Parameter** > **Edit list** function, define these parameters as **drivers**.

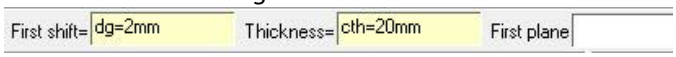
Name	Designation	Display unit	Expression	Value	Effective value	Type	Use	Driver
cth	Crosspiece thickness	mm		20mm	NOMINAL	parameter	6	Yes
cwi	Crosspiece width	mm		80mm	NOMINAL	parameter	10	Yes
dg	Door gap	mm		2mm	NOMINAL	parameter	17	Yes

- Save this file in the library using **Assembly** > **Define component** > **Edit/save template** > **Save standard template**.
  - **Standard**: **Training library**
  - **Family**: **Cabinets**
  - **New type**: *Doors*
  - **New variant**: *Glass panel door*

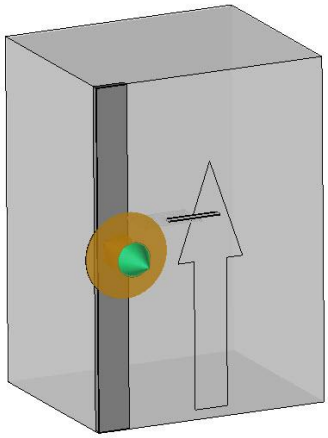
Create the parts



- Make **layer 1** current.
- Start the **Constrained block** function. 
- Set **First shift** = *dg* and **Thickness** = *cth.*



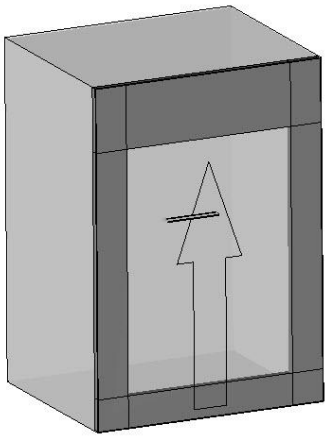
- Select the top face of the **driver block** as the **first plane**.
- Set **Mode = Faces** and enter **second shift** = *dg.*
- Select the lower face of the **driver block** as the **second plane**.



- Enter **First shift** = *dg* and select the left-hand face of the **driver block** as the **first plane**.
- Set **Mode = Length** and enter **dimension** = *cwi.*
- Adjust the red arrow so that the length of the **constrained block** is towards the inside of the **driver block** as shown opposite.

- Select the front face of the **driver block** as the **positioning plane**.
- Adjust the positioning yellow arrow so that the **constrained block** is placed inside the **driver block**.

- Make the three other parts as **constrained blocks** as shown opposite.
  - For the top crosspiece: **Length** = *2*cwi*



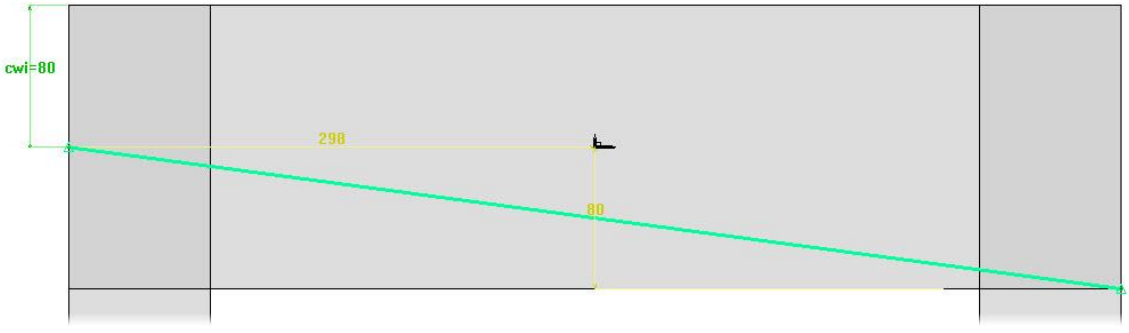
Note: At this time, it is normal that the parts overlap. They will subsequently be set to dimensions during the **counter moulding** operation.

Saw the top crosspiece

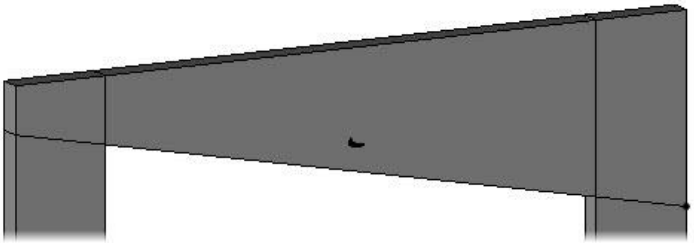
- Switch off **layer 0**.
- Create a **constrained coordinate system** on a face: **Tools** > **Coordinate system** > **Constrained coordinate system on face**. 
- Set **Mode = Non dynamic**, then select the front face of the top crosspiece as the **support face**.
- Then select:
  - **First edge**: The lower edge of the crosspiece
  - **Parallel face or edge**: The upper edge of the crosspiece
  - **Second edge**: The left-hand edge of the crosspiece
  - **Parallel face or edge**: The right-hand edge of the crosspiece

A coordinate system centered on the face is then automatically created.

- Use the **Current coordinate system** function and select the constrained coordinate system on face to make it current. 
- In a **new sketch**, draw the line shown below:
  - The left-hand point is **coincident** with the left-hand edge of the crosspiece.
  - The left-hand point is dimensioned to a distance of *cwi* from the upper edge.
  - The right-hand point is **coincident** on the lower right-hand corner of the crosspiece.

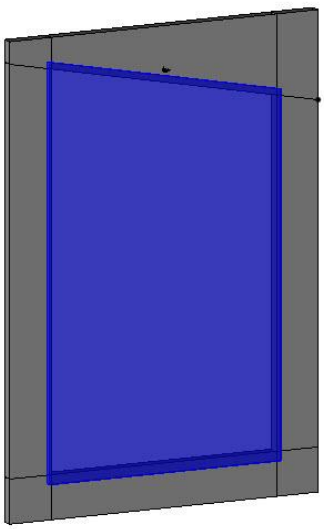


- In the **Wood** context, use the **Sawing** function to saw the top crosspiece in relation to this line. 



Create the glass panel

- Make **layer 2** current and change the **design color** to **blue**.
- Start the **Constrained block** function. 
  - Enter **First shift** = -13.5mm and **Thickness** = 5mm.
  - Select the lower edge of the sawn top crosspiece as the **first plane**.
  - Enter **Second shift** = -13.5mm and set **Allow non parallel faces = YES**.

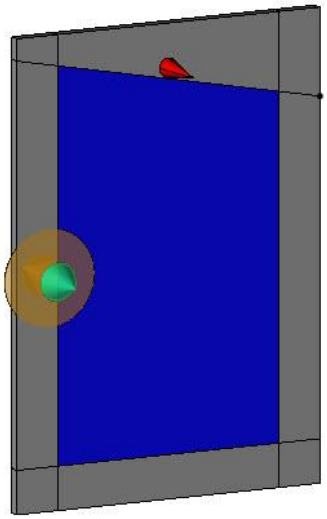


Note: Making a constrained block by allowing non parallel faces helps you create triangle and trapezoidal parts as **constrained blocks**.

  - Select the upper edge of the bottom crosspiece as the **second plane**.
  - Enter **First shift** = -13.5 and select the left-hand edge of the right-hand jamb as the **first plane or point**.
  - Set **Mode = Face** and **Second shift** = -13.5, then select the right-hand edge of the left-hand jamb as the **second plane or point**.

The trapezoidal constrained block is automatically created.

- Enter **Positioning shift** = 10mm, then select the front face of the top crosspiece as the **positioning plane**.
- Adjust the red arrow outwards and the yellow arrow as shown opposite.

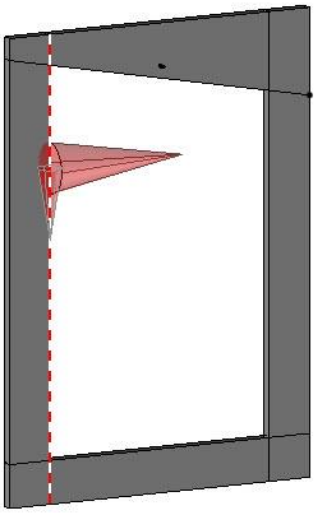


Note: The red arrow corresponds to the **positioning shift** direction.

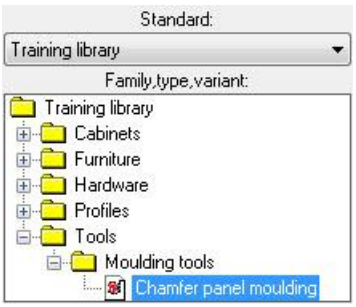
- Click on **OK** to validate the constrained block.

Perform the mouldings

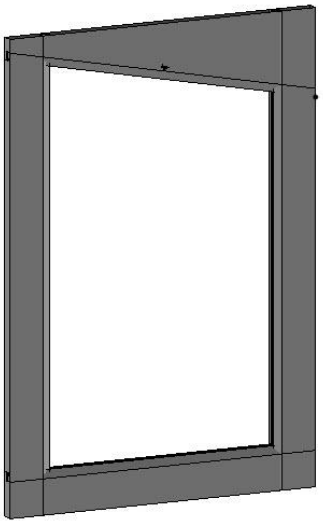
- Make **layer 1** current and switch off **layer 2**.
- Start the **Moulding** function from the **Wood** context. 
- Set **Sweep = Planar face**, then select the front face of the left-hand jamb as the **reference face**.
- Select the inside front edge of the jamb as the **reference edge or curve for tool path**.
- The side of the tool (large red arrow) must be towards the outside of the part.
- The machining direction (small red arrow) can be left by default.
- Click on **Stop** to validate the path.



The **Parameters** window of the moulding opens.



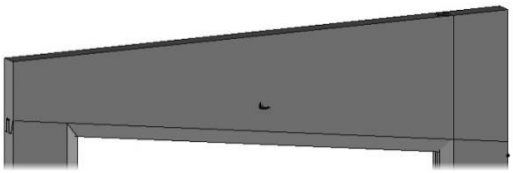
- Select **Training library** in the **Standard** dropdown list.
- Select the standard **Tools** > **Moulding tools** > **Chamfer panel moulding**.
- Set:
  - **Entry/Exit**: **Radius**
  - **Distance to start point/from terminate point**: 0mm
  - **Parameters**: **Tangent** and **Rounded**
  - **X dimension**: 0mm
  - **Z dimension**: 0mm
  - **Angle**: 0°
- Validate the window to perform the moulding.
- Then use the **Copy operation** function to perform the same operation on the three other parts. COPY OPERATION



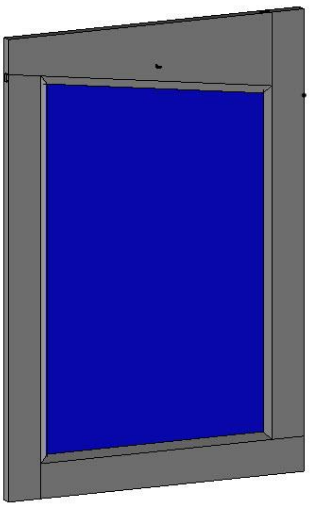
Perform the counter mouldings

- Start the **Counter moulding** function. 

Note: The **Counter moulding** function is used to automatically perform the counter moulding on parts, based on a moulding already done. However, the counter moulding tool should first be created in the standard of the tool.



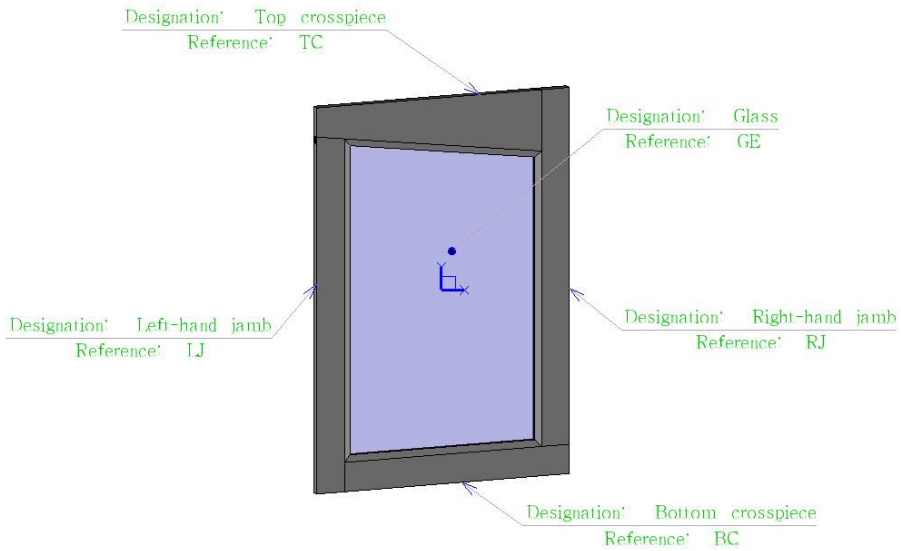
- Select the left-hand jamb as the **shape to modify**.
- Select the moulding of the top crosspiece as the **reference moulding**.
- Confirm the default parameters with **OK** to perform the counter moulding.



- Then perform the following **counter mouldings**:
  - **Shape to modify**: Bottom crosspiece; **Reference moulding**: Left-hand jamb.
  - **Shape to modify**: Right-hand jamb; **Reference moulding**: Bottom crosspiece.
  - **Shape to modify**: Top crosspiece; **Reference moulding**: Right-hand jamb.
- Display **layer 2**.

Define the parts

- Using the **Define part** function, define the different parts of the document.
- For the **jambs** and **crosspieces**:
  - **Material**: **wood** > **beech**
  - **Coating**: **paint** > **mat grey paint**
- For the **glass**:
  - **Material**: **glass** > **glass**
  - **Coating**: **No coating**



Define the set

- Use the **Define set** function to define the set:
  - **Designation**: *Glass panel door*
  - **Reference**: *GPD*
  - **Main set**: **Sub-assembly**
- Save and close the component **Glass panel door**.

Creating a handle component

- Open the provided file *Line handle*.

Note: To design this component, refer to the *TopSolid'Wood Basics Training Guide*, supplement of Exercise 6: *Configured line handle*.

Save the component

- Save this component in the **training library**:
  - **Standard**: **Training library**
  - **New family**: *Hardware*
  - **New type**: *Handles*
  - **New variant**: *Line handle*

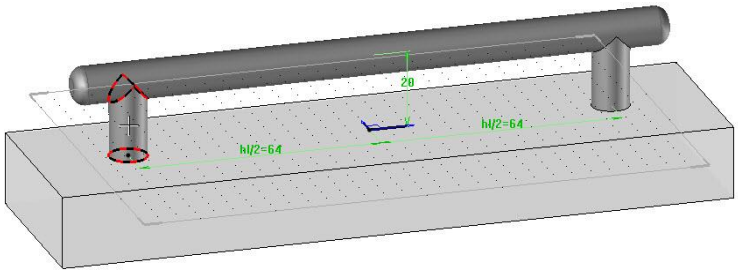
Create the martyr part

- Make **layer 1** current.
- In the **Shapes** context, start the **Block** function, and then set:
  - **X position = Centered**; **X length =** *hl+50.*
  - **Y position = Centered**; **Y length =** *50.*
  - **Z position = Below**; **Z length =** *19.*
- Then position the **block** by selecting the coordinate system origin as the **alignment point**. Press **Esc** on the keyboard to exit the function.

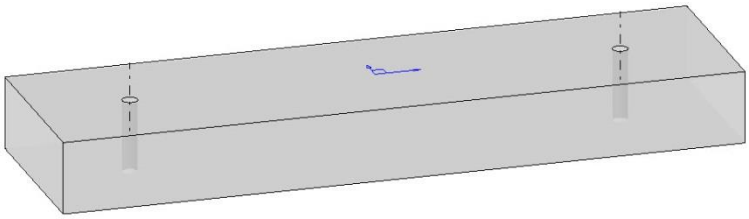
Make the drillings

- In the **Wood** context, start the **Drilling** function. 
- Set **Mode = Non dynamic** and select the top face of the martyr part as the **face to drill**.

- Select the left-hand vertical cylinder of the handle as the **first alignment face or edge**.
- Then set:
  - Hole
  - **Through one**
  - **Diameter**: 4mm



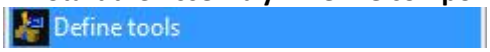
- Perform the same drilling on the top face of the martyr part in relation to the right-hand vertical cylinder of the handle.



Define the drillings as component processes

Note: Defining the drillings as component processes allows the drillings to be performed when inserting the component.

- Switch off **layer 0**.
- Start the **Assembly** > **Define component** > **Define tools** function.



- Set **Operation type = Local operation on shapes**.
- Enter **Name of tool element**: *tool1* and press **Enter** to confirm.

- Modify the designation to *Handle drillings* and press **Enter** to confirm.

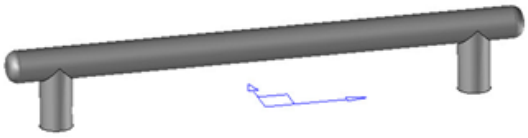
- In **Local operation to insert in tool**, select one of the drillings previously performed.
- Confirm the default **local operation name**.
- Select the second drilling, confirm its default **name**, and then validate the operation selection with **Stop**. 
- Confirm the **Operation as tool definition** window without changing anything.

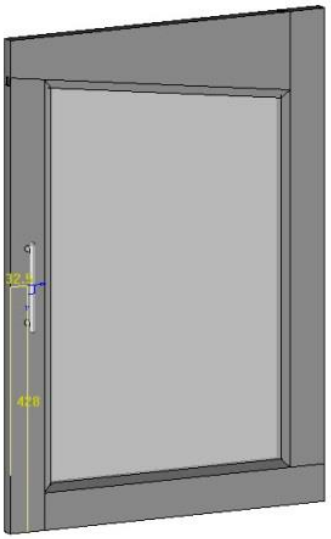
Create a key point

Note

A key point is used to quickly position a component in relation to a coordinate system.
This type of component positioning is often used for components such as hardware, which are positioned identically.

The key point is defined in the component by a coordinate system. If the component can be positioned in many different ways, it is possible to define several key points. The key point to be used is then selected when positioning the component.





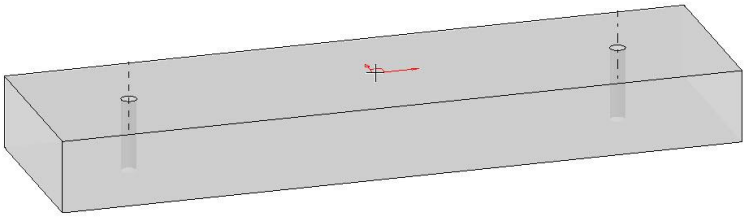
- Change the **design color** to **blue**.
- Create a coordinate system on a point using **Tools** > **Coordinate system** > **Coordinate system on point**. 
- Select the origin of the current coordinate system as the **origin point** in order to create a new coordinate system like the current coordinate system.

Note: In order to distinguish the key points from the absolute coordinate system, the absolute coordinate system should not be defined as a key point.

That's why a new coordinate system identical to the absolute coordinate system is created here.

- Start the **Assembly** > **Define component** > **Define key points** function.

- Select the coordinate system created before as the **key point or key coordinate system to insert**.



- Enter **Name of key coordinate system**: *cs1.*

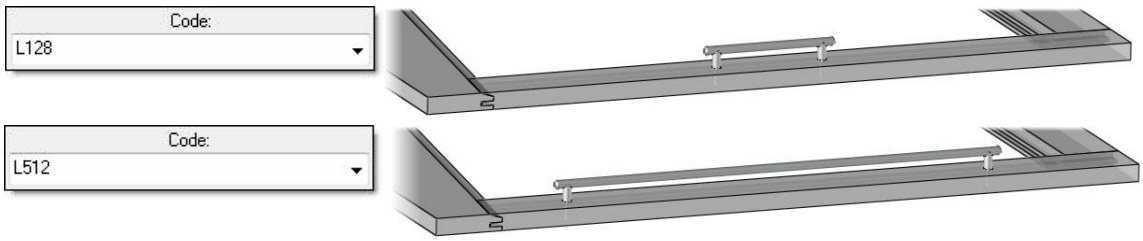
Note: To be able to interchange two components positioned by a key point, the **key points** must have identical names.
Here, the **key point** is named *cs1* pour Coordinate system 1.

- Enter **Designation of key coordinate system**: *Handle middle.*

Create a catalog

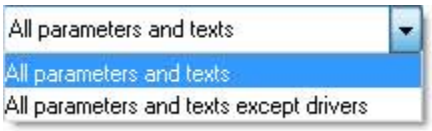
Note

If a component (furniture or hardware for example) is only available in certain dimensions, it is possible to create a **catalog**.
This catalog will be used to automatically configure one or more component parameters in relation to a chosen code.



- Create the catalog using **Assembly** > **Define component** > **Edit catalog header**.

- Then select **All parameters and texts** in the dropdown list to drive all the document parameters in the catalog.



Note: The **catalog** of the component is automatically generated in **Excel** format. If Excel is installed on your computer, the **catalog** will be opened in Excel; if not, the catalog will be opened with the **Notepad**.

The first column of the **$code catalog** is used to enter the component code which will be displayed in TopSolid.
The different parameters to configure according to the codes are on the next columns.

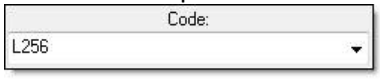
	A	B
1	$code	lp
2	L128	128
3	L256	256
4	L512	512
5	L1024	1024

- Fill in the columns as shown opposite.
- Save and close the Excel file.
- Save the **Line handle** component, and then close it.

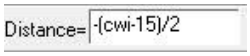
Include the handle

- From a **Design document**, **edit** a standard component using **Assembly** > **Define component** > **Edit/save template** > **Edit standard template**.

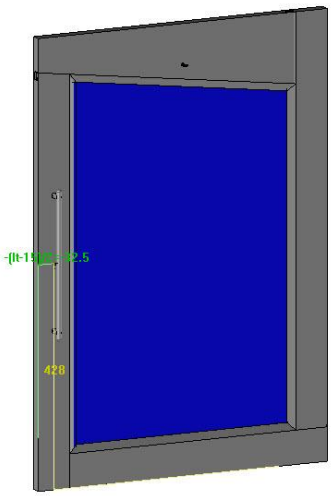
- Select the component **Cabinets** > **Door** > **Glass panel door** in the **training library**, and then validate with **OK** to open this document.
- Make **layer 3** current.
- Include the **line handle** using the **Assembly** > **Include standard** function.
- In the component inclusion window, select **Code**: **L256**.



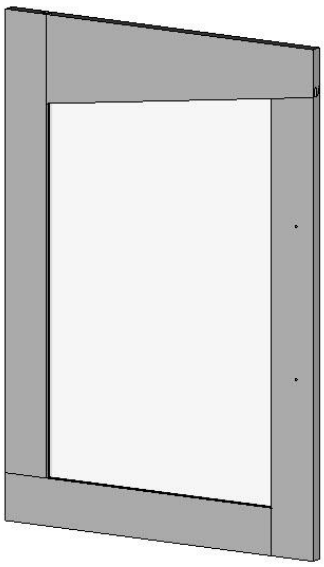
- Position the handle on the **left-hand jamb**.
- Select the left-hand edge of the jamb as the **first alignment edge**.
- Enter **distance** = *-(cwi-15)/2.*



- Select the lower edge of the bottom crosspiece as the **second alignment edge**, then the upper edge of the top crosspiece as the **parallel face or edge** in order to center the handle on the door height.



- Click on **Stop** so that the **handle** is not repeated.
- Select **Automatic** to automatically create the tools of the handle and drill the **left-hand jamb**.

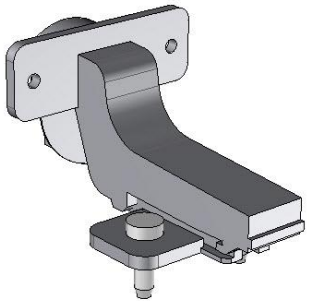


Creating a hinge component

Note

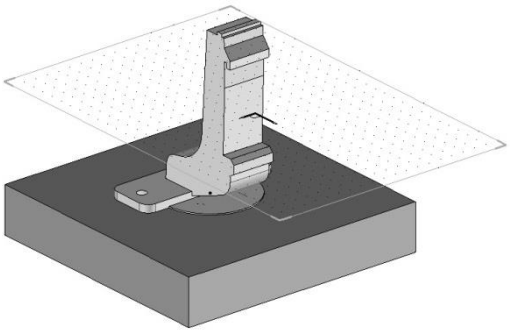
Hardware suppliers propose to provide the 3D geometries of their hardware parts.
These 3D files can be opened and used in TopSolid'Wood, but to be used optimally these components should be made "intelligent" by assigning **processes** and **key points** to them.

For information, a 3D component library from the hardware manufacturer **BLUM** is available on the Installation DVD (Disc 2).
This exercise is based on a hinge base and a hinge body from this library.

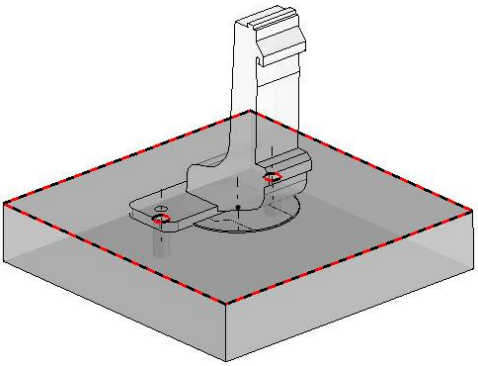


Use the body of the hinge

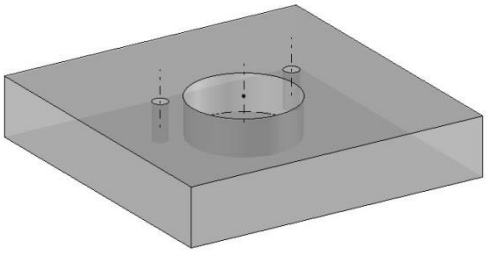
- Open the provided file *Body CLIP top 75t1750.top.*
- Save this file in the **training library**:
  - **Family**: **Hardware**
  - **New type**: *Hinges*
  - **New variant**: *Inserted hinge body 107°*
- Define the set:
  - **Designation**: *Inserted hinge body 107°*
  - **Reference**: *75t1750*
  - **Supplier**: *Blum*
  - **Main set**: **Single unit**



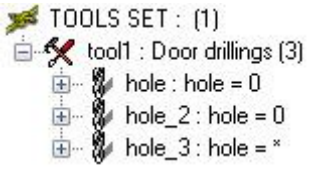
- Make **layer 1** current.
- Create the martyr part on the support plane of the hinge.
- On the martyr part, create two **blind holes** aligned on the drillings of the hinge's screws:
  - **Diameter**: 5mm
  - **Depth**: 12mm
  - **Bottom angle**: 0°.



- On the martyr part, create a **blind hole** aligned on the hinge:
  - **Diameter**: 35mm
  - **Depth**: 12mm
  - **Bottom angle**: 0°.



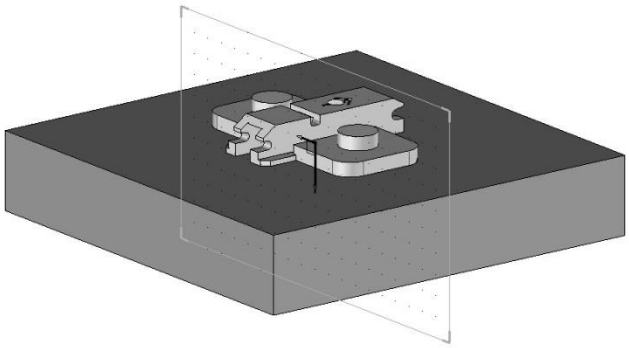
- Define these drillings as component **tools**:
  - **Name of tool element**: *tool1*
  - **Designation of the tool element**: *Door drillings*



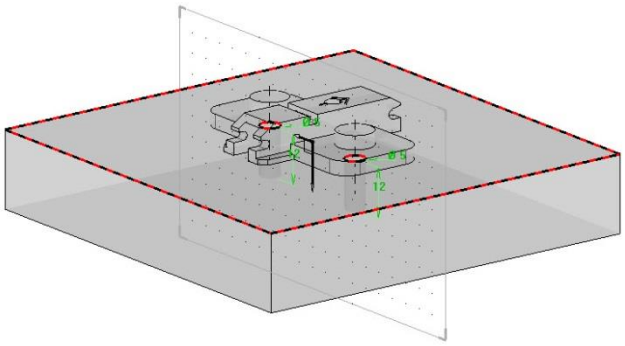
- Insert the 3 created drillings in this tool.
- Save and close the file.

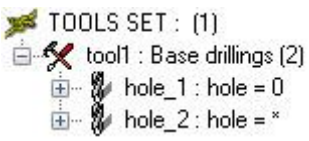
Use the base of the hinge

- Open the provided file *Base CLIP top 174e6100_01.top.*
- Save this file in the **training library**:
  - **Family**: **Hardware**
  - **New type**: *Hinges*
  - **New variant**: *Base CLIP TOP*
- Define the set:
  - **Designation**: *Base CLIP TOP*
  - **Reference**: *174e6100_01*
  - **Supplier**: *Blum*
  - **Main set**: **Single unit**

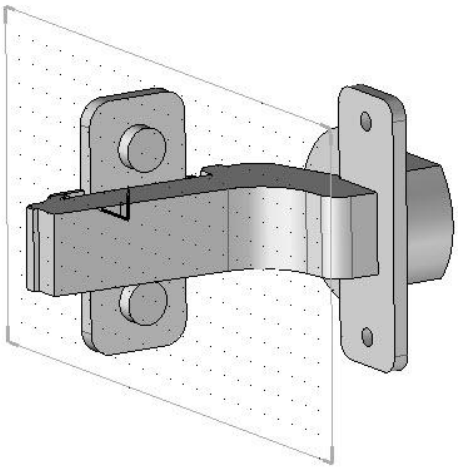


- Make **layer 1** current.
- Create the martyr part on the support plane of the base.
- On the martyr part, create two **blind holes** aligned on the drillings of the base's screws:
  - **Diameter**: 5mm
  - **Depth**: 12mm
  - **Bottom angle**: 0°
- Define these drillings as component **tools**:
  - **Name of tool element**: *tool1*
  - **Designation of the tool element**: *Base drillings*
- Insert the 2 created drillings in this tool.





Create the mounted hinge component

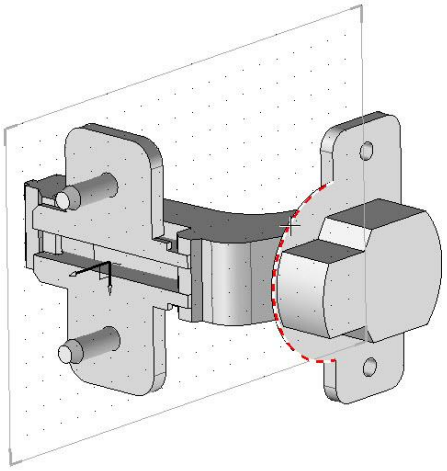


- Create a **new Design document**.
- Include the standard component **Inserted hinge body 107°**.
- Select **Other positioning**. 
- Select the absolute coordinate system of the body in **Positioning coordinate system**, then select the absolute coordinate system of the new document in **Destination coordinate system**.
- Repeat the same operation to assemble the base of the hinge.

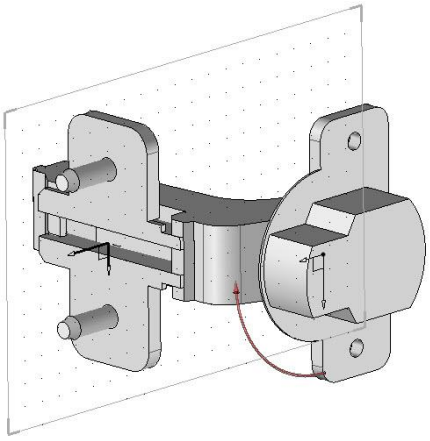
Note: In general, when assembling hardware parts provided by the manufacturers, positioning the elements by absolute coordinate system on absolute coordinate system allows them to be exactly positioned relative to each other.

- Save the mounted hinge in the **training library**:
  - **Family**: **Hardware**
  - **New type**: *Hinges*
  - **New variant**: *Inserted hinge*
- Define the set:
  - **Designation**: *Inserted hinge*
  - **Supplier**: *Blum*
  - **Main set**: **Sub-assembly**

Create the key point of the hinge



- Create a center point using **Tools** > **Points** > **Center key point**. 
- Select the circle arc of the hinge body as the **reference element** as shown opposite.



- Create a coordinate system on two axes: **Tools** > **Coordinate system** > **Coordinate system 2 axes**. 
  - For the **X axis**, use the **Through point** option, select the center point created, and then **Y-**.
  - For the **Y axis**, use the **Through point** option, select the center point created, and then **X-**.
- Define the coordinate system you just created as a **key point**: **Assembly** > **Define component** > **Define key points**.
  - **Name of key coordinate system**: *cs1*
  - **Designation of key coordinate system**: *Door hinge reference*
- Save and close the file.

Insert the hinge on the door

- Using **Assembly** > **Define component** > **Edit/save template** > **Edit standard template**, edit the standard component **Glass panel door**.
- Create a new parameter using the **Parameter** > **Create** function.
- Set **Unit type = No unit**, and then select **Tabulated values**.

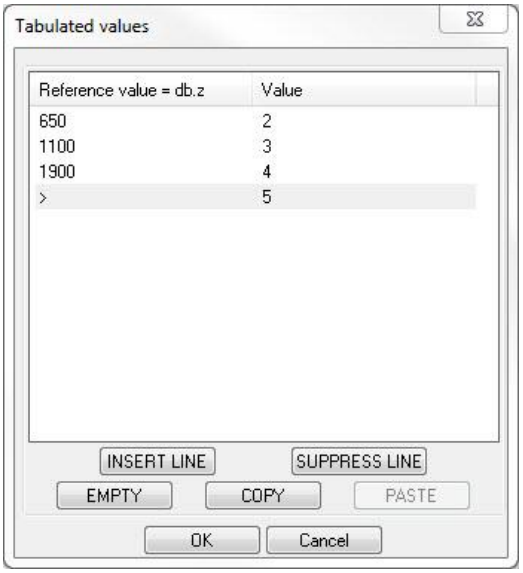
Note: **Unit type = No unit** is used to create a parameter that allows a quantity to be varied. Here, this parameter will be used to vary the quantity of hinges on the door.
A parameter with **tabulated values** modifies the value of a parameter according to the value of another parameter. Here, the number of hinges will vary depending on the height of the door.

- Enter **Reference parameter**: *db.z*

Note: As a reminder, the door has been designed on the basis of a **driver block**.
This **driver block** has been named **db**, so the **db.z** parameter is the dimension of the **driver block** on the Z axis, i.e. the height of the component.

The **Tabulated values** table opens.

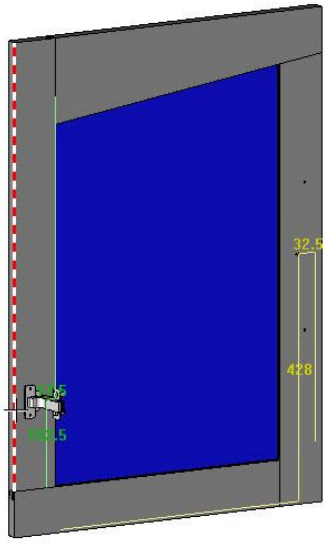
- Create the different lines by entering the **reference values** of **db.z** and the **values** as shown opposite.



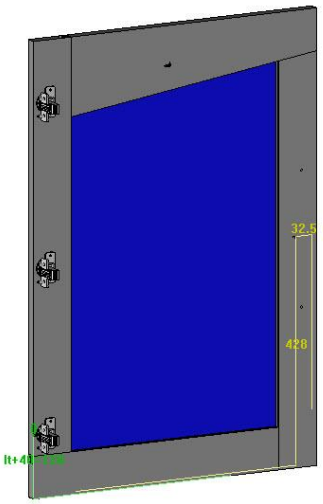
The result is the following, with the door height as reference:

- from 0 to 650mm => 2 hinges
- from 650 to 1100mm => 3 hinges
- from 1100 to 1900mm => 4 hinges
- beyond 1900mm => 5 hinges

- Click on **OK** to confirm the window.
- Enter:
  - **Name**: *hn*
  - **Designation**: *Number of hinges.*

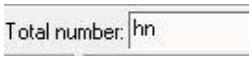


- Insert the hinge: **Assembly** > **Include standard**.
- Select the rear face of the right-hand jamb as the **destination coordinate system**.
- Select the left-hand edge of the right-hand jamb as the **first alignment face or edge**, as shown opposite.
- Enter **distance** = *24.5-dg.*
- Select the lower edge of the bottom crosspiece as the **second alignment face or edge**.
- Enter **Distance** = *cwi + 40.*



- Select the **Repeat** option to directly repeat the hinge. 
- Select **Propagation**: **Linear**, **Propagation direction: Y+**.
- Enter **Total distance** = *db.z-(2*cwi)-80.*

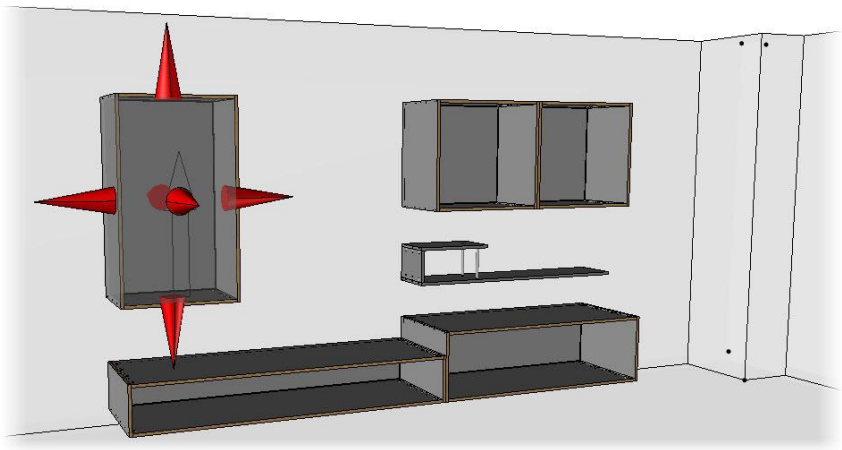
- Enter **Total number**: *hn*



- Select **Automatic** to perform the drillings of the hinges on the door.
- Save and close this component.

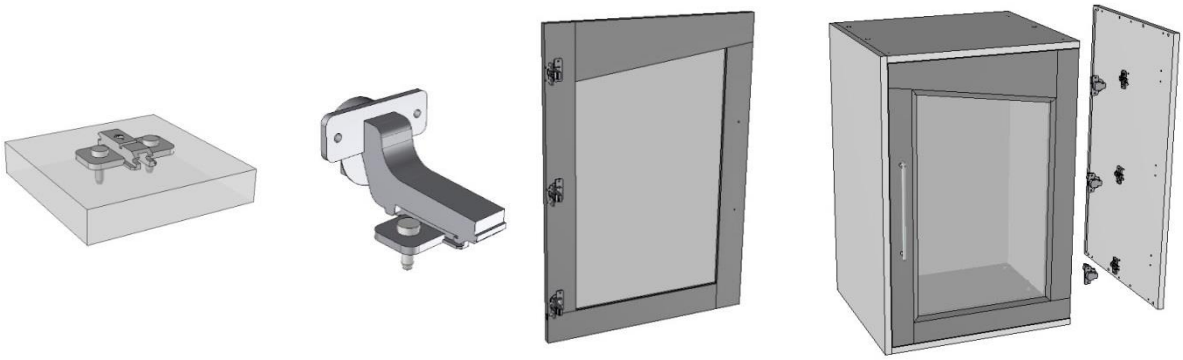
Using the door

- Open the file *Interior design.top.*
- Make **level 14** current and set its name to *Cabinets elements.*
- In the **Assembly** context, use the **Include standard** function to include the door component. 
- Then include the door as **driver block** in the left-hand cabinet.



- Validate the default values of the driving parameters, and then perform the machinings using the **Automatic** option.

Note: The **tool processes** defined in a component are automatically inherited at each assembly level .
Here, the **tools** of the base's drillings are automatically inherited, then performed on the cabinets.

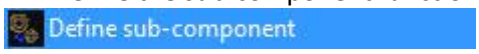


Supplement: Declaring the handle as a sub-component

Note

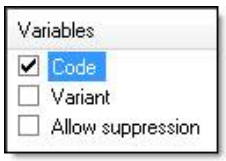
It is possible to declare several elements of a component as sub-components. This allows you to make changes to the sub-component once the component has been included.
In the case of the door with the handle, declaring the handle as a sub-component will allow the handle to be modified or interchanged with another one, once the door has been included.

- Edit the template document of the door: **Assembly** > **Define component** > **Edit/save template** > **Edit standard template**.
- Define the sub-component function: **Assembly** > **Define component** > **Define sub-component**.



- Select the **line handle** as the **component**.
- Enter **Name** = *Handle*, and then press **Enter** to confirm.

Note: Three options are available to configure the sub-component:



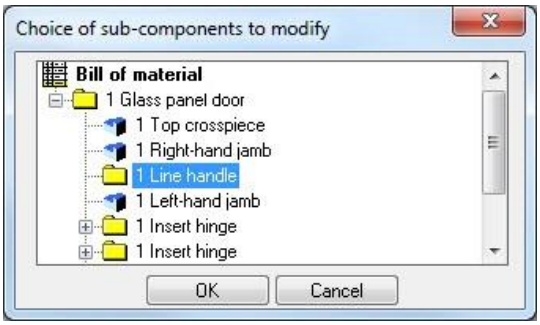
- **Code**: Used to modify the catalog code of the sub-component.
- **Variant**: Used to interchange the sub-component with another one.
- **Allow suppression**: Used to delete the sub-component.

- Check the **Code** option and click on **OK** to validate.
- Then click on **Stop**.

The handle is now defined as a **sub-component**. It will then be possible to modify its catalog code after the door insertion.

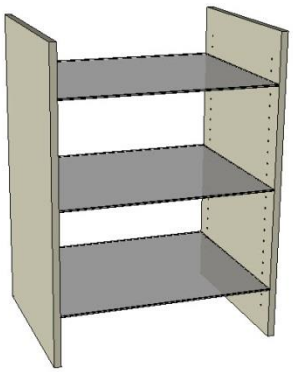
- Save and close the door component.
- Open the file *Interior design.top*.
- Start the **Modify element** function, and then select the door. 
- Click on the **Sub-component** button to modify a sub-component of the door. 

- In the window that has just opened, unfold the node of the door, click on **Line handle**, and then click on **OK** to confirm.
- Modify the **code** of the handle and click on **OK** to confirm.



The handle is then modified in the assembly.

Exercise 7: Creating glass shelves



The goal of this exercise is to make a "Glass shelves" component in a **driver block**.

Concepts addressed:

- The **linear constraint** distribution of a constrained block
- The measured parameter
- The multi-drilling
- The driver propagation

Creating the component

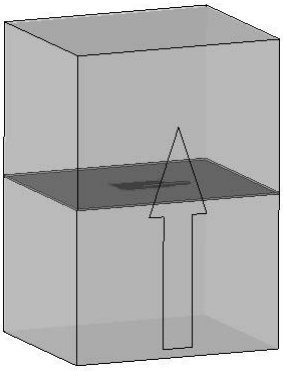
Create the parameters

- Create a **new Design document**.
- Create a **driver block**:
  - **Name**: *db*
  - **Default housing mode**: **Housing**
  - **Designation**: *Cabinet block*
- Create a parameter, then define it as a **driver**:
  - **Unit type** = **No unit**
  - **Value**: 3
  - **Name**: *ns*
  - **Designation**: *Number of shelves*

Save the component

- Save this component in the **training library**:
  - **Family**: ***Cabinets***
  - **New type**: *Separation panels*
  - **New variant**: *Glass shelves*

Create the shelves

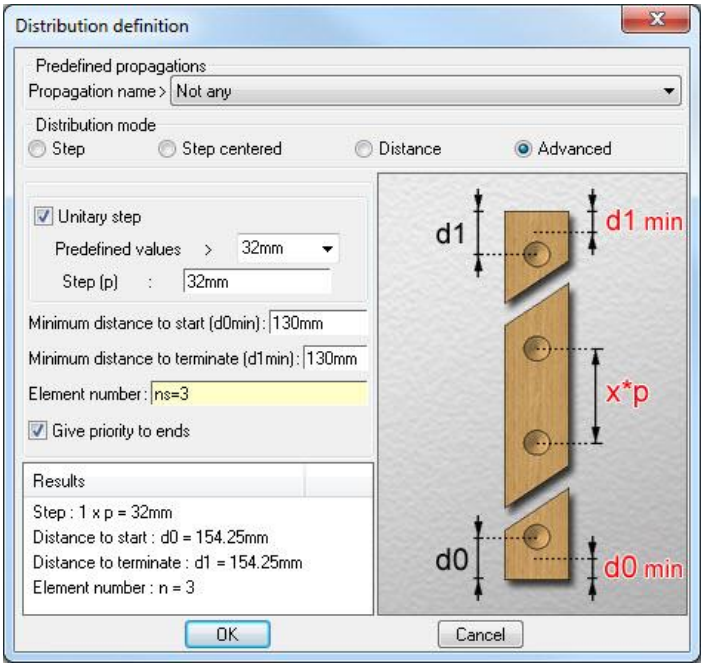


- On the **layer 1**, create the shelves as **constrained blocks**:
  - **Thickness** = 5mm
  - **Planes**: 4 faces of the **driver block**
- When selecting the **positioning plane**, use the **Linear constraint** option. 

Note: The **Linear constraint** option is used to repeat the **constrained block**.

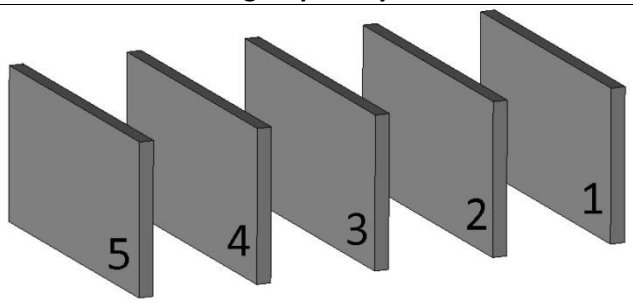
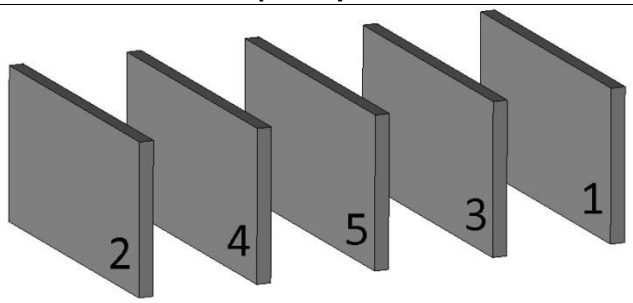
- Select the low face of the **driver block** as the **start face or edge**.
- Select the top face of the **driver block** as the **terminate face or edge**.

- In the distribution definition window, select the **Advanced** distribution mode.
- Set:
  - Check **Unitary step**
  - **Step (p)**: 32mm
  - **Minimum distance to start**: 130mm
  - **Minimum distance to terminate:** 130mm
  - **Element number**: ns=3
  - Check **Give priority to ends**



Note: The **Advanced** mode in **Unitary step** is used to distribute a given number of elements with minimum start and terminate distances.
With the 32mm **unitary step**, the gap between the elements will always be a multiple of 32mm.

Checking the **Give priority to ends** option is used to change the distribution mode of the repetition instances. This is necessary for the rest of the exercise (measured distance parameter).

Do not give priority to ends	Give priority to ends
	

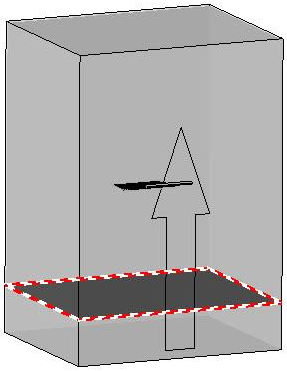
- Click on **OK** to confirm the **distribution definition** window.
- Center the **constrained blocks** in relation to the calculated distribution.

Modify the repetition template

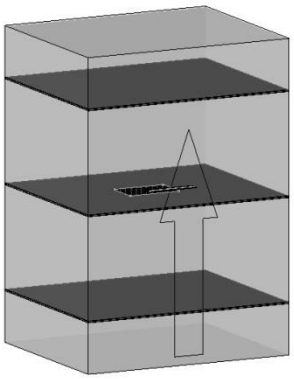
Note: The **Linear constraint** option repeats the **constrained block**. This means that a template of the repetition is hidden and some instances are displayed.

- Use the **Modify element** function and select one of the repeated **constrained blocks**.
- Select the **Edit template** option to hide the repetition instances and display the template. 

- From the **Shapes** context, use the **Chamfer** function to apply a **chamfer** with a **length** = 1mm to the bottom and top faces of the shelf. 
- Use the **Edit** > **Repeat** > **Show repetition** function, and then select the template of the shelf to display the repetition instances again and hide the template. 

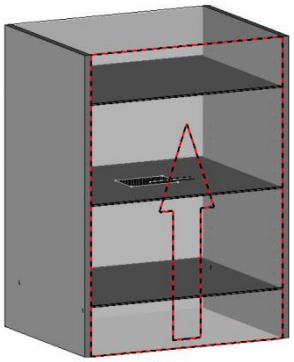


Define the shelves



- Define the shelves:
  - **Designation**: *Glass shelf*
  - **Reference**: *GL-SH*
  - **Material**: **glass** > **glass**
  - **Coating: No coating**
- Define the set:
  - **Designation**: *Glass shelves component*
  - **Reference**: *GL-SH-CP*
  - **Main set**: **Sub-assembly**

Creating the processes



Create the martyr parts

- Set the **level 2** current.
- Create the martyr parts of the component by creating two **automatic constrained blocks** with a **thickness** = 19mm on the right-hand and left-hand sides of the **driver block**. Position the constrained blocks outside the driver block.
- Switch off **layer 0**.

Create the measured parameter

Note: As the shelves are repeated in **Advanced** mode, the distance of the first shelf from the bottom automatically adjusts to the height and the number of elements.
In order for the multi-drillings to automatically start from the first shelf, a measured parameter from the bottom of the component to the first shelf will be created.

- Create a measured **parameter**. 
- Select **Distance parameter**. 
- Select the lower face of the martyr part as the **first element**. Use the **Plane** option to select only faces. 

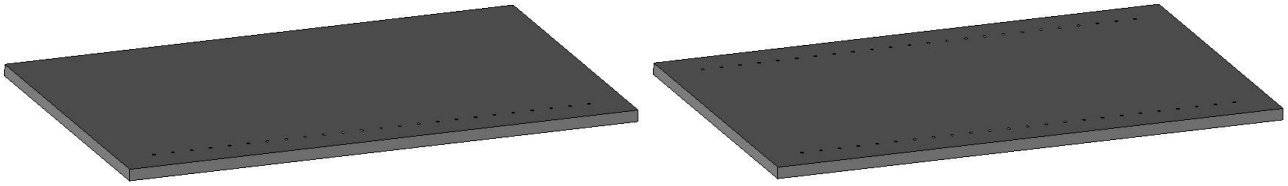
- Then select the bottom face of the low shelf as the **second element**.
- **Name** this parameter *dps.*

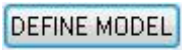
Make the multi-drillings

Note

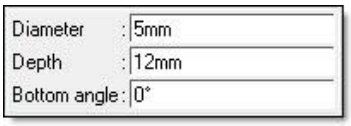
The **Multi-drillings** function is used to propagate a drilling directly on a face in one or two directions.
Before performing the multi-drillings, a drilling model should be created.

Simple propagation ***Double propagation***

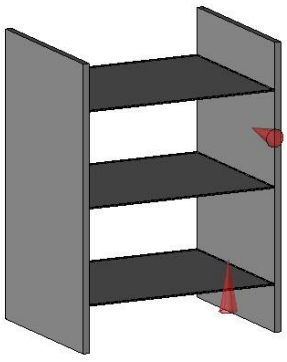


- In the **Wood** context, start the **Drilling** function. 
- Select **Define model** to create a new drilling. 
- Create this drilling in **Model = User** mode, and then name this model *Shelf support 5x12*.

- In the drilling definition window, select **Hole** in the left-hand list, and then move it to the right-hand list. Click on **OK** to confirm.
- Set:
  - **Hole**: **Blind**
  - **Diameter**: 5mm
  - **Depth**: 12mm
  - **Bottom angle**: 0°



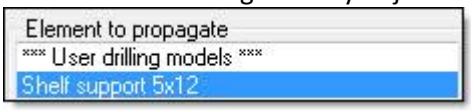
- Confirm with **OK** to create the new **drilling model**.
- Start the **Multi-drillings** function. 
- Set **Propagation = Double** and select the inner face of one of the martyr parts as the **support face**.



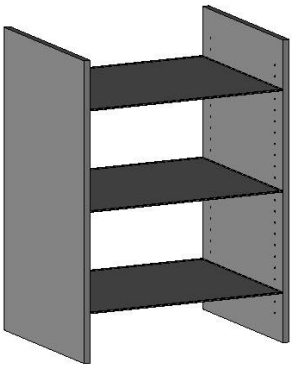
- Select:
  - the bottom edge of the martyr part as the **starting face or edge for the first propagation**.
  - the top edge of the martyr part as the **terminate face or edge for the first propagation**.
  - the front edge of the martyr part as the **starting face or edge for the second propagation**.
  - the rear edge of the martyr part as the **terminate face or edge for the second propagation**.

The distribution definition window opens.

- Select the drilling model you just created as the **element to propagate**.



- Set for the **first propagation** (on the height of the component):
  - **Distribution mode**: **Step**
  - **Step (p)**: 32mm
  - **Distance to start**: dps – 64
  - **Optimize the number of elements**



- Set for the **second propagation** (in the depth of the component):
  - **Distribution mode**: **Distance**
  - **Distance to start (d0)**: 50mm
  - **Distance to terminate (d1)**: 50mm
  - **Element number**: 2
- Validate these parameters with **OK** to perform the multi-drillings.
- Similarly, perform the **multi-drillings** on the second martyr part.

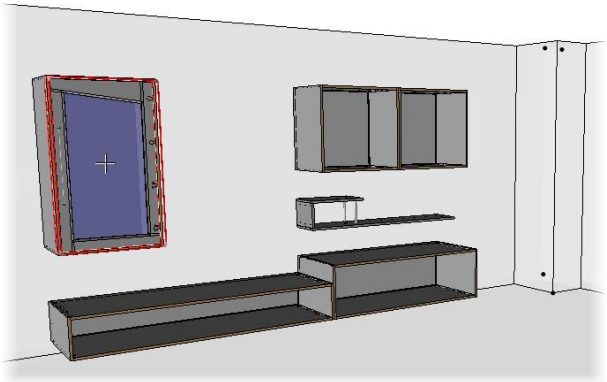
Define the multi-drillings as processes

- Define the multi-drillings as component processes using **Assembly** > **Define component** > **Define tools**.
  - **Name of tool element**: *tool1*
  - **Designation of the tool element**: *Multi-drillings*
- Save and close this component.

Include the component

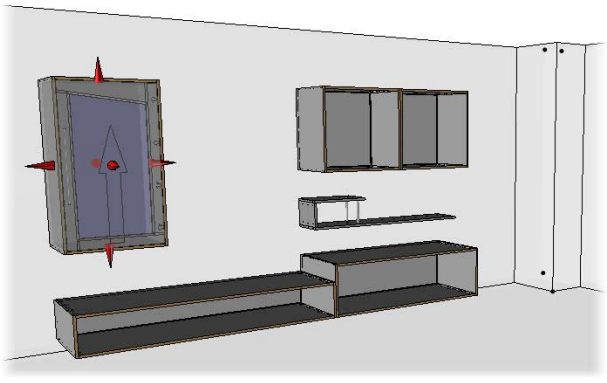
- Open the file *Interior design.top*.
- Include the **Glass shelves** component using **Assembly** > **Include standard**. 
- To insert a **driver block** component in the same cabinet as an already inserted **driver block** component, select **Housing mode = Like a component**.
- Set **Merge drivers = NO**.

Note: **Merge drivers = YES** is used to merge the **driver blocks** of the two components. It is then not possible to modify the second **driver block** without modifying the first one, and vice versa.



- Select the door inserted in the left-hand cabinet as the **reference component**.
- Select the front arrow of the generated **driver block**, and use the **shift**. 

- Set **Shift** = 50mm, and select **OK** to confirm.



- Validate the **driver block** with **OK** and set the **Number of shelves** parameter.
- Perform the component processes automatically using the **Automatic** option.
- Save the document.

Supplement: Shelf propagation as driver

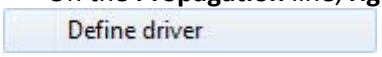
Note

The **linear** and **circular** propagations, as well as the **TopSolid'Wood distributions** can be defined as drivers. This allows the position of a repetition instance to be modified after the component inclusion.
For example, this helps avoid a collision between a hinge and another element, or adjust the position of a shelf.

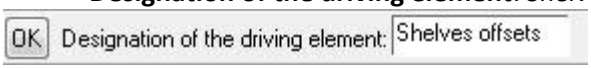
- Edit the **Glass shelves** component.
- Open the construction tree using **Ctrl + ²**.
- In the **Main** tab, **right-click** > **Edit** and select the repetition of the shelf.

Note: Editing a repetition in the construction tree makes it easy to find and modify the repetition instances, the propagation, as well as the repetition template.

- On the **Propagation** line, **right-click** > **Define driver**.



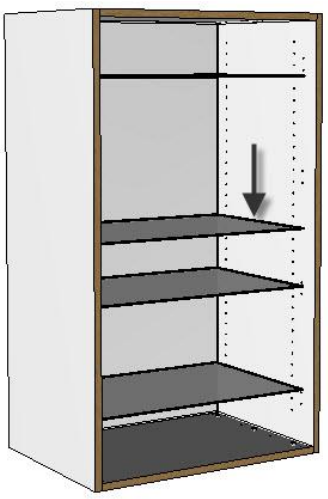
- Set:
  - **Name of driving element**: *so*
  - **Designation of the driving element**: *Shelves offsets*



- Open the interior design document. Delete the **Glass shelves** component, and then reinsert it.

Note: The driver propagations are taken into account when inserting the component. If the **driver propagation** is declared after being inserted, the component must be deleted, and then repositioned to be taken into account.

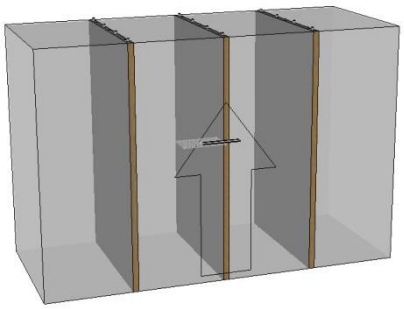
- Once the component has been reinserted, use **Modify element** and select one of the glass shelves.
- The **Offset instance** option is used to offset the selected shelf.



Note: It is possible to set a positive or negative value as **offset**. The positive offset will be done in the propagation direction and the negative offset in the opposite direction.

- Save the document.

Exercise 8: Creating separation panel components



The goal of this exercise is to make two components as **driver blocks** of vertical and horizontal separation panels assembled by dowels and cams.
To create this type of component, a shelf component assembled with martyr parts should first be created. This component will then be included, and repeated in a **driver block** component.

Concepts addressed:

- The automatic assembly

Creating the shelf component

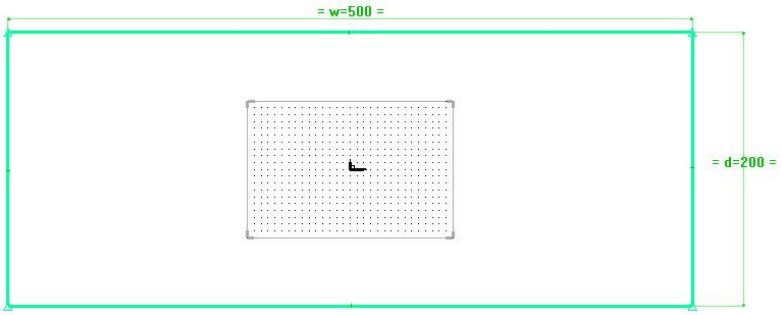
Create the parts

- Create a **new Design document**.
- Create the following three parameters in **Unit type = Length**:
  - **Value** = 500mm; **Name**: *w*; **Designation**: *Width.*
  - **Value** = 200mm; **Name**: *d*; **Designation**: *Depth.*
  - **Value** = 19mm; **Name**: *th*; **Designation**: *Thickness.*

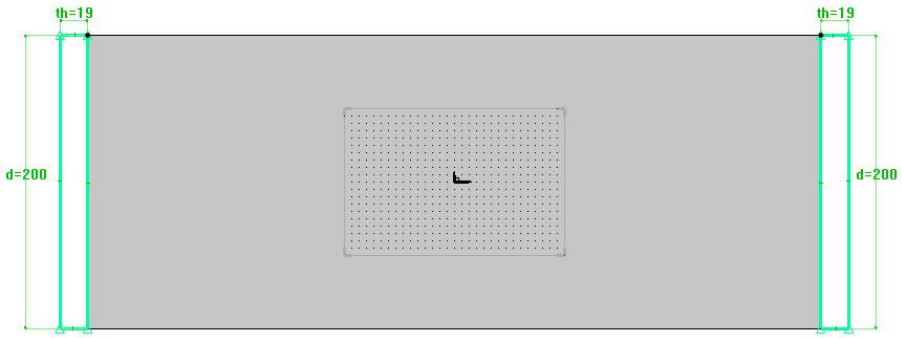
- Define these parameters as **drivers**.
- Create the sketch shown opposite.

The dimensions are constrained on X and Y.

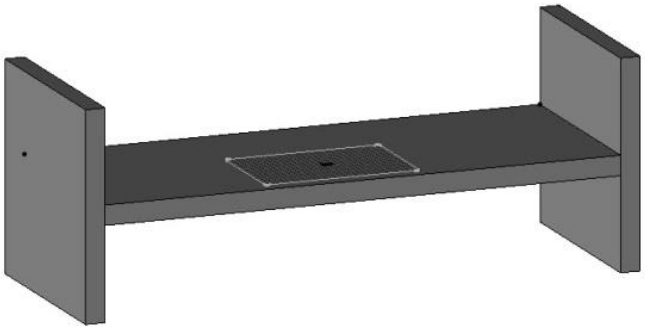
- Extrude this sketch to a **height**: *th.*



- Then draw the sketch of the martyr parts as shown opposite.

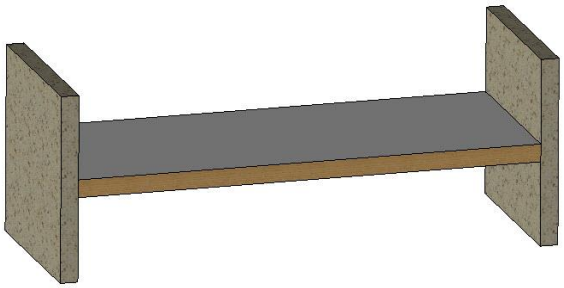


- Make **layer 1** current, then extrude the martyr parts in **Alignment = Centered**, **Generatrix sketch = Global** and **Result = One shape per profile** to a **height** of 200mm.



- Create a panel on the shelf.

Note: A panel can only be made on a defined part. So when a panel is made on a non-defined part, the part definition is automatically started.



- Define the shelf:
  - **Designation**: *Fixed shelf*
  - **Reference**: *FX-SH*
  - **Material**: **wood** > **particule board**
  - **Coating**: **paint** > **mat white paint**
- Validate the definition with **OK** and configure the panel:
  - Place an edge on the front face of the panel.
  - Use the codified edge **EDG-TH-2-PVC-OAK**.

Save and define the component

- Save the component in the **training library**:
  - **Family**: **Cabinets**
  - **Type**: **Separation panel**
  - **New variant**: *Fixed unitary shelf*
- Define the set
  - **Designation**: *Fixed shelf component*
  - **Reference**: *FX-SH-CP*
  - **Main set**: **Content**

Assembling the parts automatically

Create the automatic assembly

Note

The **Automatic assembly** function is used to assemble several parts together automatically. This function associates one or more standard components with a predefined propagation per component.

- From a **Design document**, open the options: **Tools** > **Options**.
- Go to **TopSolid'Wood Configuration** > **Automatic assembly**.
- Select **Add rule** to create a new automatic assembly.
- Double-click on the **Untitled** line and name this automatic assembly: *Dowels + Cams.*

- Add a **Pin** component to the **automatic assembly**.
- Select **Smooth pin** and **Code**: **30 x 8**, then validate with **OK**.

The **Smooth pin** component is added to the automatic assembly.

- Select the **Dowels predefined propagation** for this component.
- Set **Centered thickness**.

- Then add an **Eccentric** component to the **automatic assembly**.
- Select **Wood eccentric assembly hinge** and **Code**: **th 19**, then validate with **OK**.
- Set the **Cams predefined propagation** to **Centered thickness**.

Dowels + Cams

Component-code	Matter-coating	Predefined propagation	Centring thickness (Z)	Centered thickness
Smooth pin - 30x8	-	Dowels	-	X
Wood eccentric assembly hinge - ep 19	-	Cams	-	X

- Validate the Options window.

Assemble the parts

- Start the **Automatic assembly** function. 

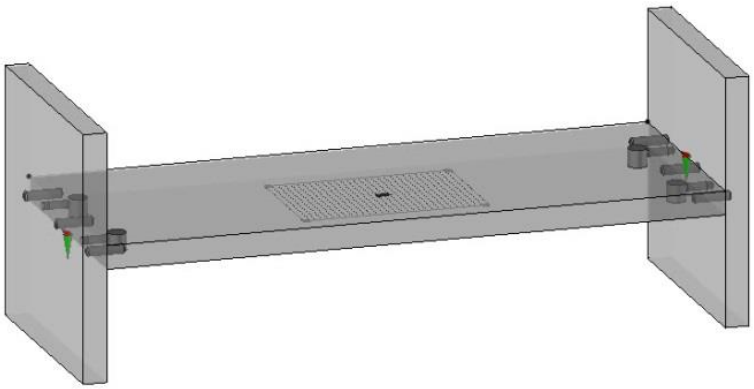
The martyr parts are not defined, so they must be selected manually.

- Select the two martyr parts, as well as the shelf. Click on **Stop** to confirm the selection.
- In the **Automatic assembly** window, select **Rules** > **Dowels + Cams**.

- Reselect the parts manually in the graphical zone, and validate the window.
- Select the front face of the shelf as the **start face**.
- Click on **Centred automatically** to automatically center the assembly in the thickness.
- Click on **OK** to confirm.

Note: In the case of a closed assembly (a cabinet for example), selecting **Outside face** or **Inside face** allows you to set the position of the cam cases automatically.

The assemblies are generated.
The green arrow represents the positioning direction of the assemblies in the thickness.
The red arrow represents the positioning direction of the assemblies in the depth.

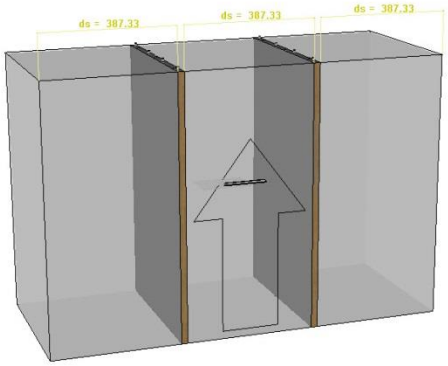


- Place the cam cases downwards.
- Validate the positionings with **OK** to perform the machinings.
- Save and close the file.

Creating the distributed separation panel components

Create the vertical separation panel component

- Create a **new Design document**.
- Create a **driver block**:
  - **Name**: *db*
  - **X length** = 1200; **Y length** = 500; **Z length** = 800
  - **Current coordinate system**
  - **Designation**: *Cabinet block*
- Create the following two parameters:
  - **Unit type = No unit**; **Value** = 2; **Name**: *ns* and **Designation**: *Number of separation panels*
  - **Unit type = Length**; **Value** = 19; **Name**: *th* and **Designation**: *Panel thickness*
- Define these parameters as drivers.

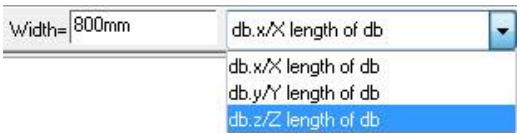


To position and distribute the separation panels, the distance between the separation panels must be calculated beforehand.

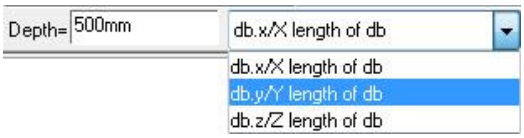
- Create a new parameter:
  - **Unit type**: **Length**
  - **Value** = (db.x-(ns*th))/(ns+1)
  - **Name**: *ds*
  - **Designation**: *Distance between separation panels*
- Save the component in the **training library**:
  - **Family**: **Cabinets**
  - **Type**: **Separation panel**
  - **New variant**: *Distributed vertical separation panels*

Include the component

- Include the **Fixed unitary shelf** component.
- Select for the parameter **Width = db.z/Z length of db**.



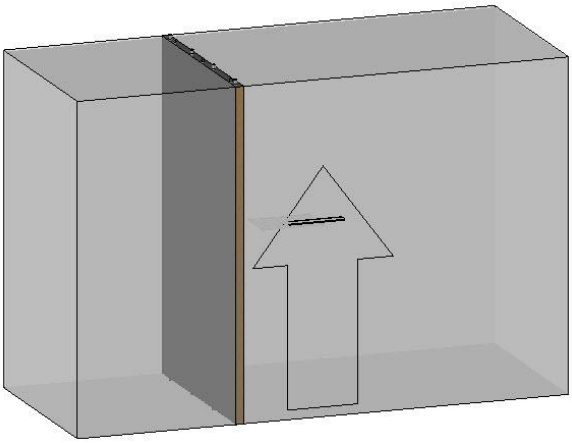
- Select for the parameter **Depth = db.y/Y length of db**.



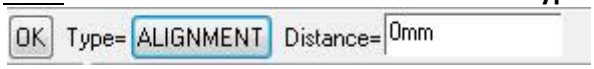
- Select for the parameter **Thickness = th/Panel thickness**.

Note: The two parameters of the components have the same name; it is then possible to connect them automatically using the **Automatic -> Panel thickness = 19.000mm** option.

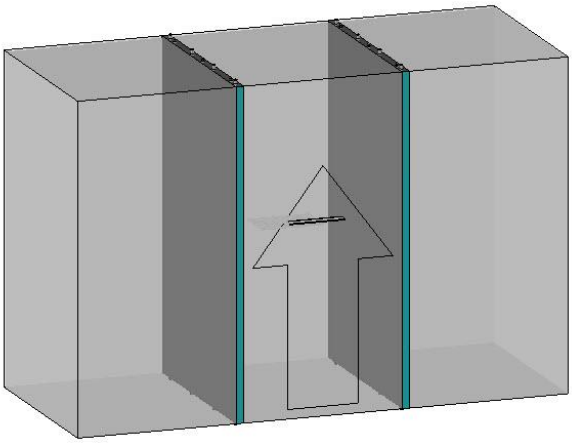
- Click in the document to release the component.
- Apply two constraints of the shelf on the driver block:
  - **Origin**: Front edge of the shelf; **Destination**: Front face of the driver block
  - **Origin**: Lower edge of the shelf; **Destination**: Lower face of the driver block



Note: These two constraints must be set to **Type = Alignment**.



- Then apply a third constraint:
  - **Origin**: Left-hand face of the shelf
  - **Destination**: Left-hand face of the driver block
  - **Type = Alignment**
  - **Distance** = *ds*



- Select **Stop** to stop placing constraints.
- Select **Propagation**: **Linear**, **Propagation direction**: **X+**.
- Then set **Distance per instance** = *ds+th.*

- Finally, enter **Total number** = *ns.*
- Click on **Stop**.

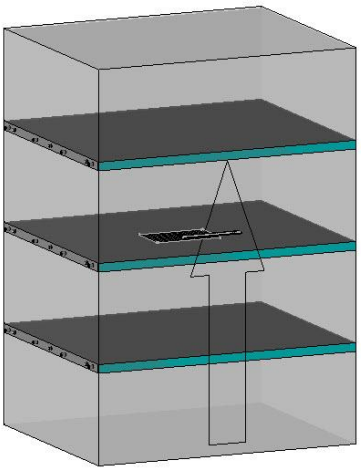
- Define the set:
  - **Designation**: *Vertical separation panel component*
  - **Reference**: *VERT-CP*
  - **Main set**: **Sub-assembly**

- Save and close the document.

Create the horizontal separation panel component

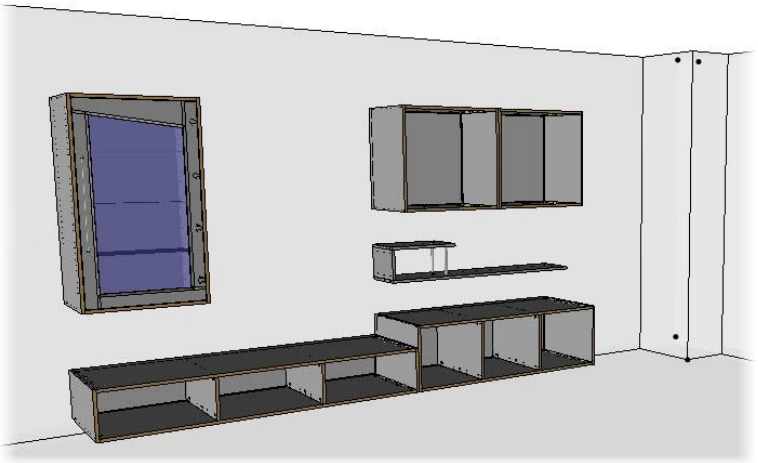
Using the same method as for the vertical separation panels, create a **Horizontal separation panels** component.

- In a **new document**, create a **driver block**.
- Create the two driving parameters: *Number of separation panels* and *Panel thickness.*
- Create the distance parameter between the separation panels: **Value** = (db.z-(ns*th))/(ns+1).
- Include the **Fixed unitary shelf** component and set the parameters.
- Apply the constraints of the shelf to the driver block.
- Propagate the separation panel linearly.
- Define the set:
  - **Designation**: *Horizontal separation panel component*
  - **Reference**: *HORIZ-CP*
  - **Main set**: **Sub-assembly**
- Save this component in the **training library**:
  - **Family**: **Cabinets**
  - **Type**: **Separation panel**
  - **New variant**: *Distributed horizontal separation panels*



Using the separation panels

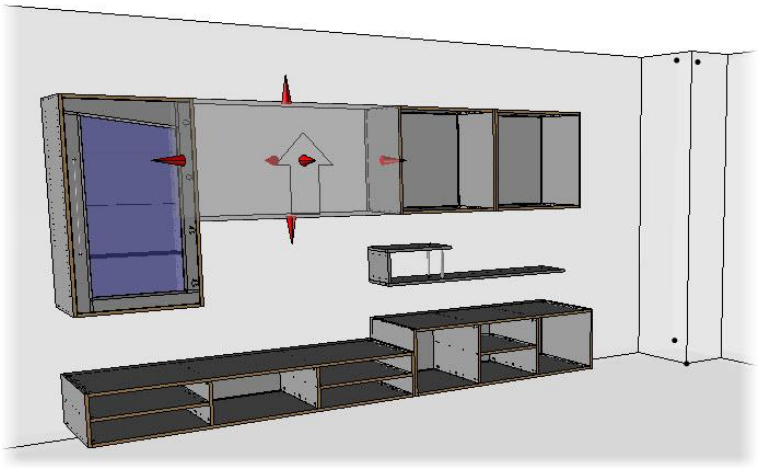
- Open the interior design file.
- Insert the **Distributed vertical separation panels** standard component in the two bottom cabinets.
- Set the **number of separation panels** to 2.
- Perform the machinings using **Automatic**.



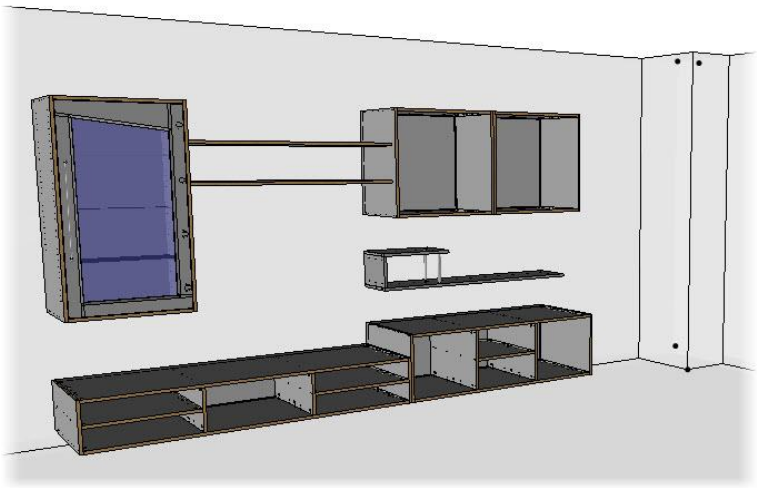
- Then insert the **Distributed horizontal separation panels** component in the middle compartment of the right-hand cabinet, as well as in the left-hand and right-hand compartments of the left-hand cabinet.
- Set the **number of separation panels** to 1.
- Perform the machinings using the **Automatic** option.

Note: A **Driver block** component can also be inserted outside a cabinet or a closed housing. If an open volume forms a rectangular parallelepiped, you can insert a **driver block** component in it.

- Insert the **Distributed horizontal separation panels** component.
- Select the outer right-hand face of the upper-mid cabinet as the **cabinet block** to place the separation panels between the upper-mid and the upper left cabinets.

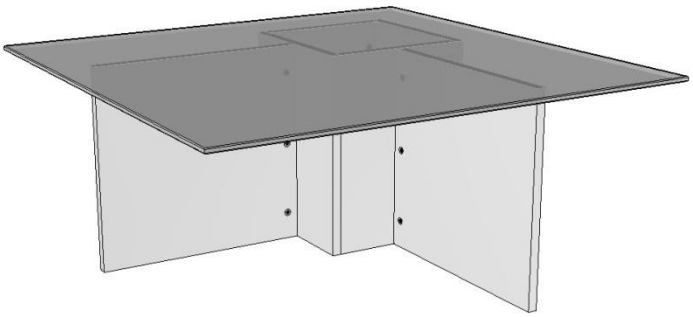


- Set an **offset** = 50mm on the **front plane** arrow.
- Set 2 separation panels, and then click **OK** to confirm.
- Perform the machinings using the **Automatic** option.
- Save the document.



Exercise 9: Making a coffee table

The goal of this exercise is to make a coffee table using the **constrained block** and the **automatic assembly**.

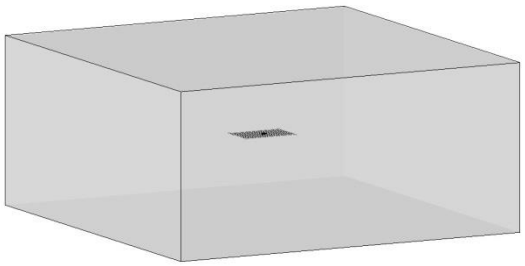


Creating the table

Create the construction volume

- Create the following three length parameters:
  - **Value**: 1000; **Name**: *w*; **Designation**: *Table width*
  - **Value**: 450; **Name**: *h*; **Designation**: *Table height*
  - **Value**: 19; **Name**: *th*; **Designation**: *Panel thickness*

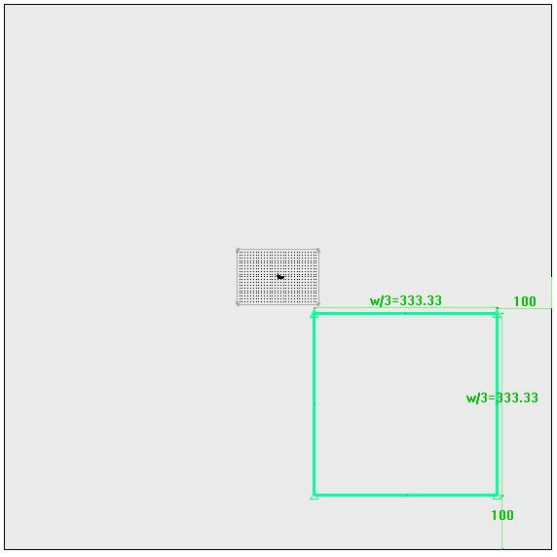
- Define these parameters as **drivers**.
- Create a construction volume using the **Block** function:
  - **X length** = *w*
  - **Y length** = *w*
  - **Z length** = *h*
  - **Alignment point**: Origin of the absolute coordinate system
- Apply a **transparency** of **7** to this **block**.



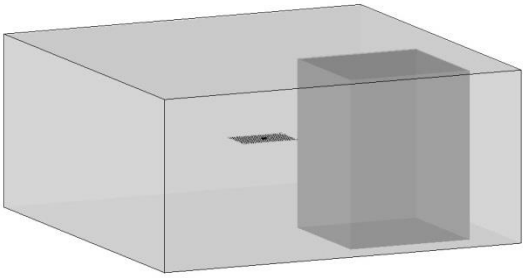
Create a second construction volume

Note: The goal is to create a second construction volume in order to make the base of the table.

- Start a **new sketch**.
- Create a **rectangular contour**:
  - Dimension the width and height of the contour to a **nominal value** = w/3.
  - Dimension this contour to a **nominal value** = 100mm from the bottom and the right-hand side of the construction volume.
- End the sketch.

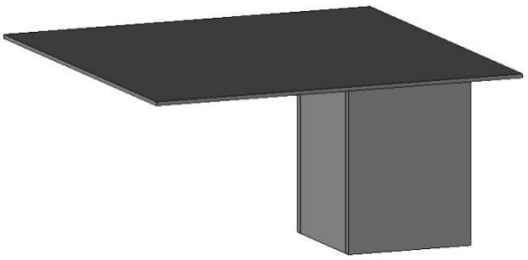


- Extrude this sketch:
  - Set **Offset from starting curve** = -h/2.
  - Select the **Z+ direction**.
  - Set **Height** = h-10.
- Apply a **transparency** of **4** to this block.

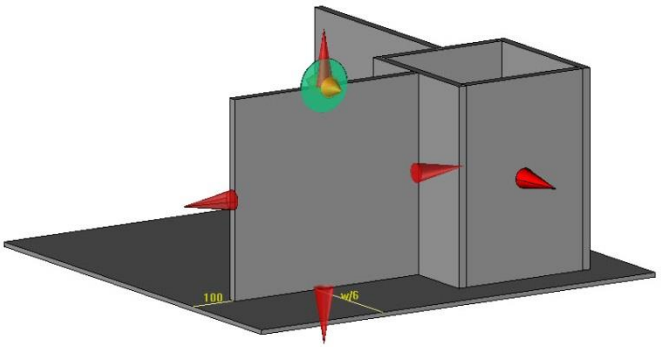


Create the parts

- Set the material to **wood**> **particule board** and the coating to **paint** > **mat white paint**.
- Make **layer 1** current.
- Create an **automatic constrained block** on the top face with a **thickness** = 10mm.
- Position this **constrained block** inside the construction volume.
- Create the four parts of the base as **constrained blocks** from the second construction volume as shown opposite:
  - **Thickness** = th
  - There are two covering sides and two covered sides.



- Switch off **layer 0**.
- Create the two other parts making up the base as **constrained blocks**:
  - **Thickness** = th
  - **First shift** = 100mm
  - **Positioning shift** = w/6



- Make a **chamfer** with a **length** = 2mm on the top and bottom faces of the table top.

Saving and defining the table

Define the parts

- Define the parts of the base:
  - **Designation**: *Base 1-6*
  - **Reference**: *B1-6*
- Define the table top:
  - **Designation**: *Glass table top*
  - **Reference**: *GT*
  - **Material**: **glass** > **glass**
  - **Coating**: **No coating**
- Define the set:
  - **Designation**: *Rectangular coffee table*
  - **Reference**: *REC-CT*
  - **Main set**: **Sub-assembly**

Save in the library

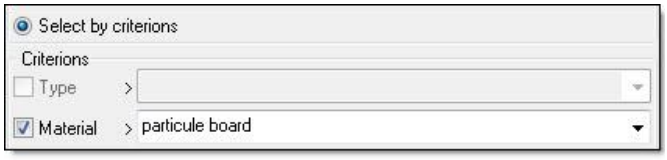
- Save this component in the library using **Assembly** > **Define component** > **Edit/save template** > **Save standard template**:
  - **Standard**: **Training library**
  - **Family**: **Furniture**
  - **New type**: *Tables*
  - **New variant**: *Rectangular coffee table*

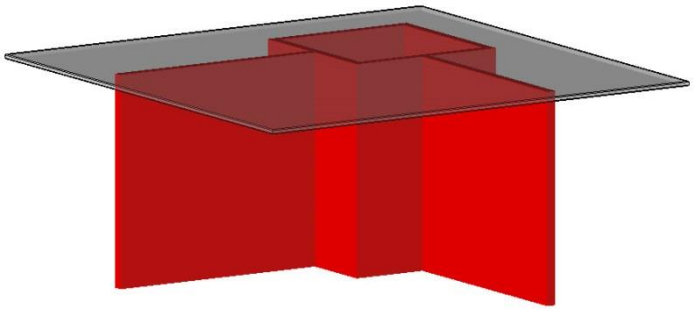
Assembling the table

- Start the **Automatic assembly** function. 
- Select **Main assembly**.

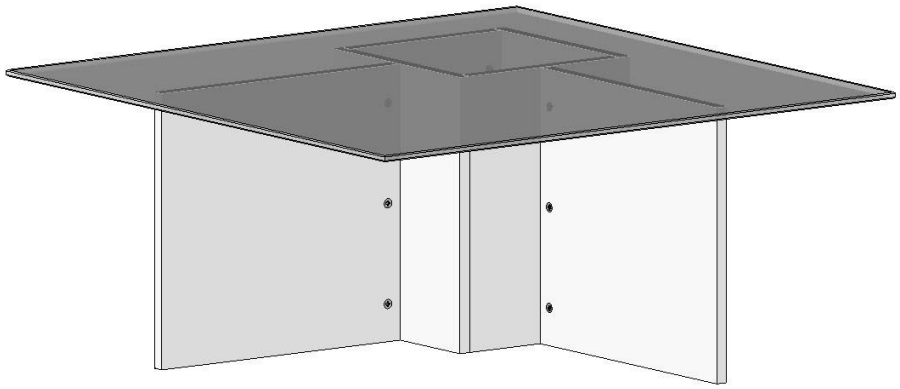
Note: In the TopSolid'Wood selection windows, including the one for the **automatic assembly**, parts can be selected following different criteria.

- Activate **Select by criterions**.
- Check **Material** and select **particule board** in the dropdown list. Thus, all the parts that are made out of **particule board** are selected.





- Click on **OK** to confirm the selection.
- Select:
  - The lower edge of one of the selected parts as the **start face**
  - **Centred automatically**
  - **Inside face**
- Using the green arrows, adjust the position of the cam cases as shown below, then validate with **OK** to place the hardware parts and perform the machining operations.



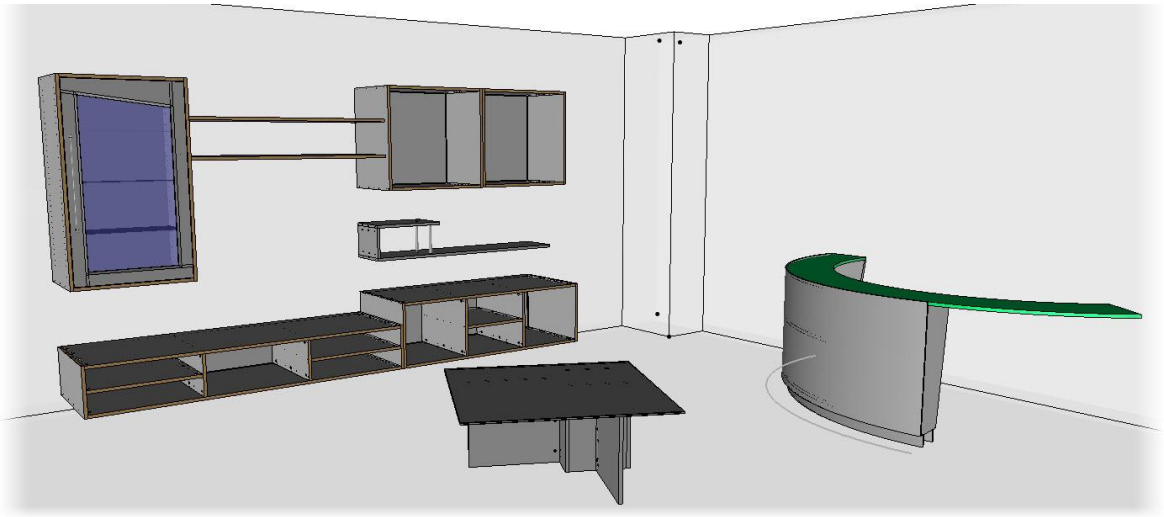
- Save and close this file.

Inserting the table

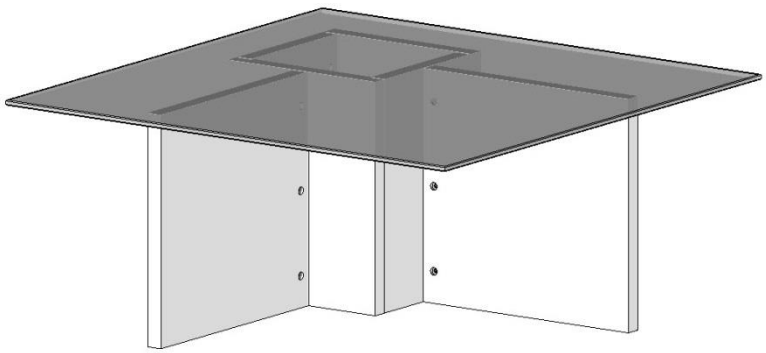
- Open the interior design file and insert this table on **layer 15**.
- Place a constraint between the bottom of the table and the ground.
- Press the **Esc** key to exit the function.

Note: As long as a component is not fully constrained, it can be moved and/or rotated using the **Edit** > **Move parents** and **Rotate parents** functions.

- Start the **Edit** > **Move parents** function.
- Select the inserted table as the **element to move**.
- Position the table in the interior design, then left-click to validate the positioning.
- Start the **Edit** > **Rotate parents** function.
- Select **Mode = Auto**.
- Select the top face of the table as the **element to rotate**.
- Rotate the table, then left-click to validate the positioning.



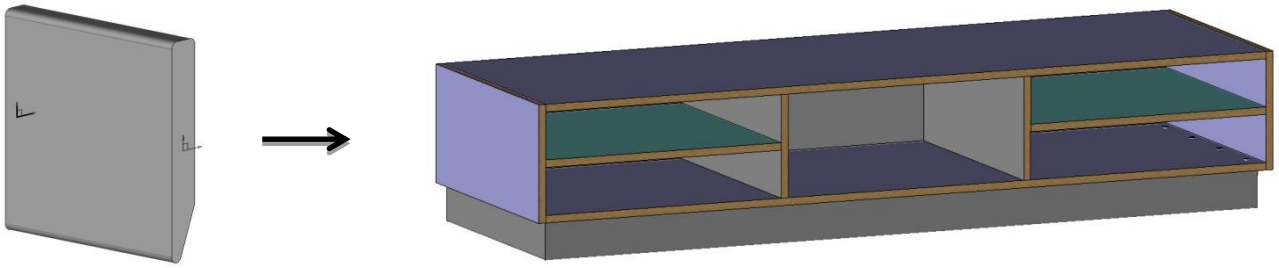
- Start the **Modify element** function.
- Select the **rectangular coffee table**.
- Select **Parameter** and select the parameter **Panel thickness** in the dropdown list.
- Set this parameter to the value: *30mm* and confirm with **OK.**



Exercise 10: Creating an extruded component

Note

Creating an **extruded** component enables you to easily use linear components such as baseboards, angles or cover profiles.
It will then be possible to place this component directly between two points or on a whole contour, and perform the cuts between the different extruded components automatically (mitre cuts, covering/covered cuts).



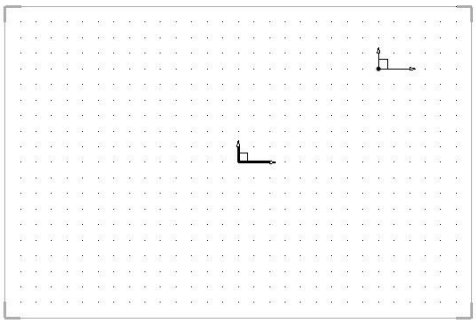
An **extruded** component must be created according to the procedure described below.

Creating the extruded component

Create the coordinate systems

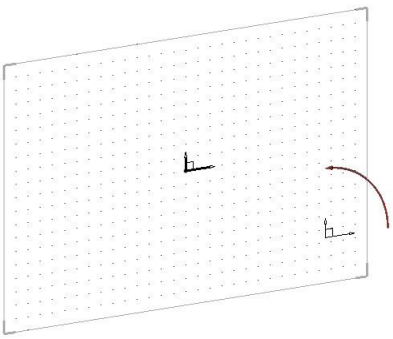
- Create a **new Design document** by selecting **Without template**.
- Make **layer 1** current.
- Create a new coordinate system using **Tools** > **Coordinate system**, and click any point to the right of the absolute coordinate system.
- Then select **Set as current**.

Note: To create an extruded component, it is necessary not to work on the **absolute coordinate system**, that's why a new coordinate system is created and set as current.

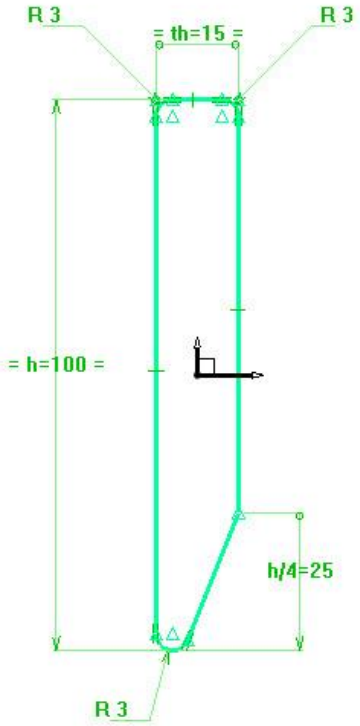


- Start the **Edit** > **Name** function and name this new coordinate system *fr1*.
- Leave the **Designation** field blank, and click on **OK** to confirm.

- Switch off **layer 0**.
- Create a length **parameter** with a **value** = 100mm, a **name**: *l* and a **designation**: *Length.*
- Create a **duplicated coordinate system** using **Tools** > **Coordinate system** > **Duplicate coordinate system**.
- Select **fr1** as the **coordinate system**, and set a **translation** in the **Z+ direction** and a **translation distance**: l.
- **Name** this coordinate system *fr2*.

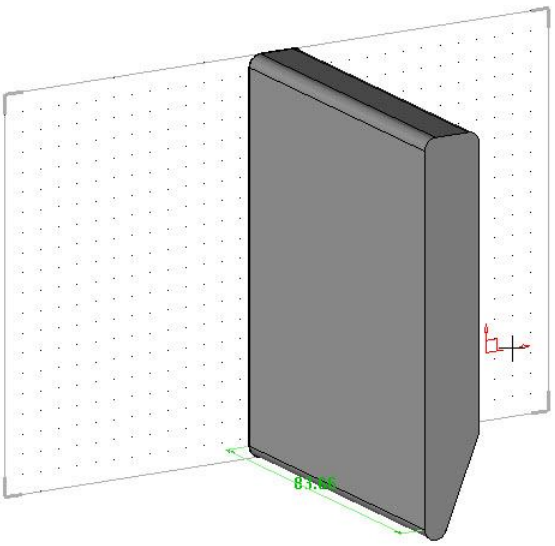


Create the profile



- Create two length parameters:
  - **Value**: 100; **Name**: *h;* **Designation**: *Baseboard height*
  - **Value**: 15; **Nom**: *th;* **Designation**: *Baseboard thickness*
- Create the sketch for the baseboard as shown opposite:
  - The thickness and height dimensions are constrained in symmetry on the X and Y axes.
  - The three **fillets** have a **radius** of 3mm.
- End the sketch.

- Extrude the sketch, but instead of entering a **height**, select the **duplicated coordinate system**.

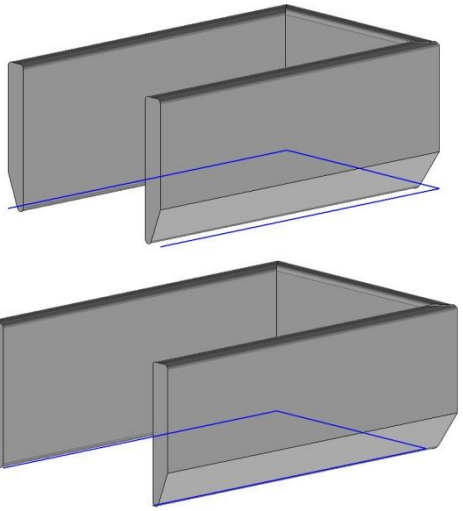


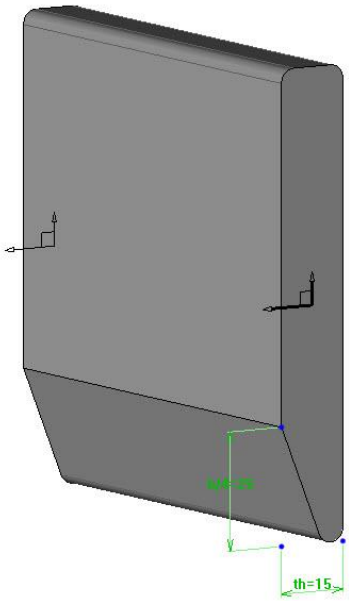
Note: Here the **fr2** coordinate system must be selected and not the origin point of the coordinate system.

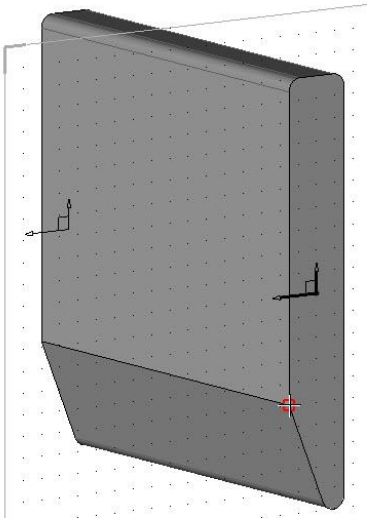
Create the key points

Note

Creating several **key points** (shown in blue opposite) allows the **extruded** component to be positioned in different ways in relation to the selected path (shown in blue below).







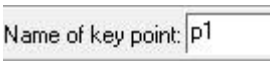
- Start the **Tools** > **Points** > **Offset point** function. 
- Select the point shown opposite as the **origin point**.

Note: The key points must be on the plane of the **fr1** coordinate system.

- Select the **Y- direction** and enter a **distance**: h/4.
- Position the dimension to finish creating the **offset point**.

The creation function automatically returns to the beginning.

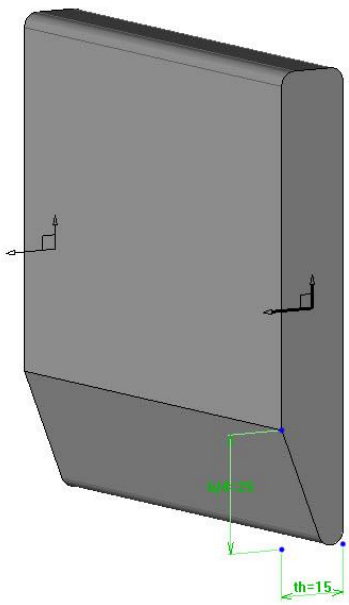
- Select the offset point created previously as the **origin point**.
- Select the **X- direction** and enter a **distance**: th.
- Position the dimension.

- Use **Assembly** > **Define component** > **Define key points**.
- Select the left-hand **offset point** as the **key point or key coordinate system to insert**.
- Enter:
  - **Name**: *p1*. 

  - **Designation**: *Bottom left point*. 

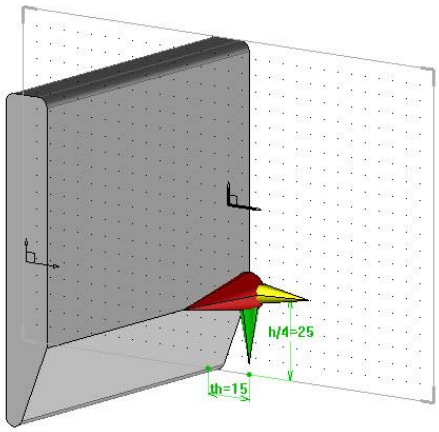
Note: To be able to interchange two extruded components, the **key points** must have identical names.
Here, the **key point** is named *p1* for Point 1.

- Then select the right-hand **offset point**.
- Enter the **name**: *p2* and the **designation**: *Bottom right point.*



Define the component

- Define the part using the **Wood** > **Define** > **Define part** function:
  - **Designation**: *Part*
  - **Material**: **wood** > **beech**
  - **Coating**: **paint** > **mat grey paint**
- In the **Cutting-up** tab, click on the **Invert axis** button to obtain the length axis on the extruded component length. INVERT AXIS



- Define the set using the **Wood** > **Define** > **Define set** function:
  - **Designation**: *Baseboard type 1*
  - **Reference**: *BS-1*
  - **Main set**: **Single unit**
- Save the component in the library using **Assembly** > **Define component** > **Edit/save template** > **Save standard template**:
  - **Standard**: **Training library**
  - **New family**: *Profiles*
  - **New type**: *Baseboards*
  - **New variant**: *Type 1*

Note: In order to be used as an **extruded component** and automatically perform the cuts, this component must be defined as an extruded component.

- Start the **Assembly** > **Define component** > **Define extruded component** function.
- Select **All assembly** to define all the parts of the assembly as **extruded** parts.

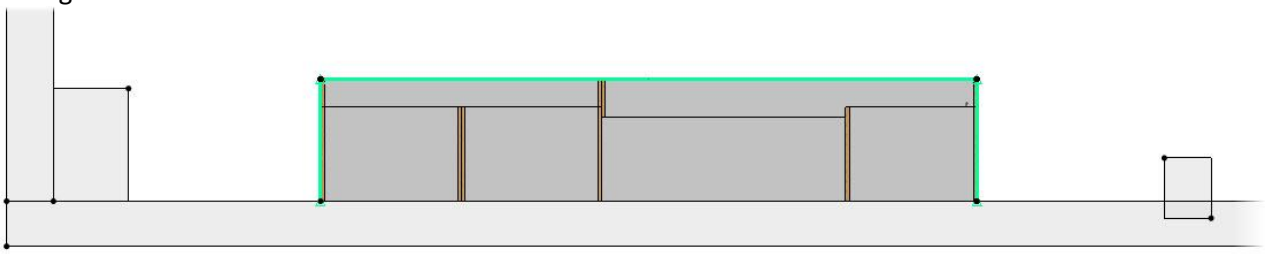
ALL ASSEMBLY

- Save and close the file.

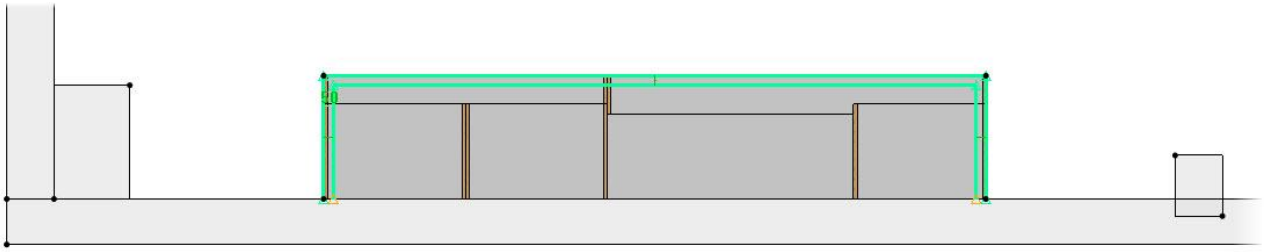
Using the extruded component

Draw the contour

- Open the interior design file.
- Make **layer 16** current.
- Start a **new sketch**.
- Using the **Contour** function, draw a contour enclosing the cabinets on the left side, the front side, as well as the right side as shown below.



- Start the **Create offset profile** function in **Offset type = Profile** mode to draw the offset of the **curve** to a **distance** = 50mm.



Note: The offset curve must be inside the **reference curve**.
To do this, when prompted to enter the **distance**, place the offset inside using the mouse, enter *50* (by default in the **Through point** box), then press **Enter** to confirm.

- End the sketch.

Insert the extruded component

- In the **Wood** context, start the **Extruded component** function. 
- In the **Standard component inclusion** window, select the **Training library** library and select the component **Profiles** > **Baseboards** > **Type 1**.

Note: By default, an extruded component is positioned between two points to generate. Once the extruded components have been generated, cuts can be performed between the extruded components using the **Assembly** > **Use process** function.
The **On curves** option is used to automatically generate all the extruded components on a contour, and then perform the cuts automatically.

- Select the **On curves** option. 

- Select the offset sketch in **Curves**.

The three extruded components are then generated.

Different parameters can then be configured:

- **Key point**: Used to choose the key point to position the extruded components.
- **Rotation angle**: Used to rotate the extruded components.
- **Cut**: Used to choose the type of cut to be applied to the components.

- Set:
  - **Key point**: **Bottom right point**
  - **Rotation angle**: 0°
  - **Cut = Mitre cut**

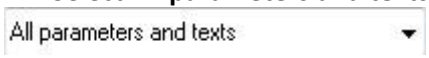
- Click on **OK** to confirm.

Note: As the counter is included on a path, the direction of the extruded components may be reversed. To modify the direction of a sketch, use the **Curves** > **Origin** function, select the sketch, invert the direction of the **red arrow**, then reinsert the extruded components.

Supplement: Creating a catalog and adding baseboards

Create the catalog

- Delete the baseboards inserted in the interior design document, and then close this document.
- Edit the template document of the baseboard: **Assembly** > **Define component** > **Edit/save template** > **Edit standard template**.
- Generate the catalog: **Assembly** > **Define component** > **Edit catalog header**.
- Select **All parameters and texts** in the dropdown list.

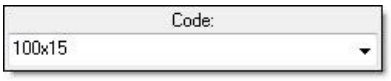


The catalog is then generated in Excel format.

- As the parameter **l** is used to define the extruded component, delete the column **l** in the Excel catalog.
- Fill in the catalog boxes as shown opposite.

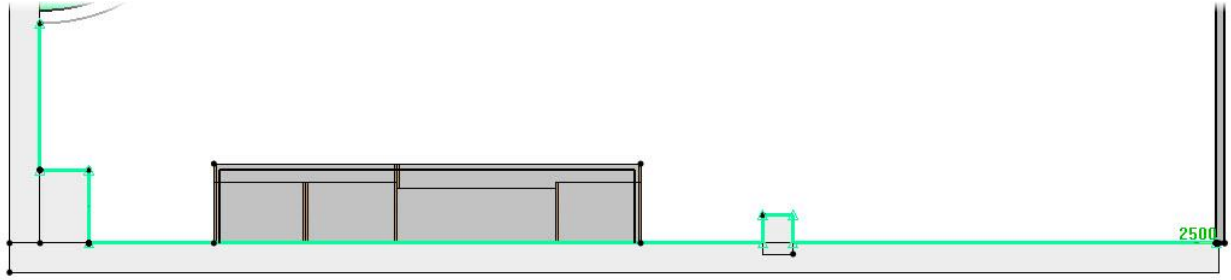
	A	B	C
1	$code	h	th
2	80x10	80	10
3	100x15	100	15
4	120x18	120	18

- Reopen the interior design document.
- Reinsert the extruded components on the sketch by selecting the **code**: **100x15** when selecting the standard extruded component.



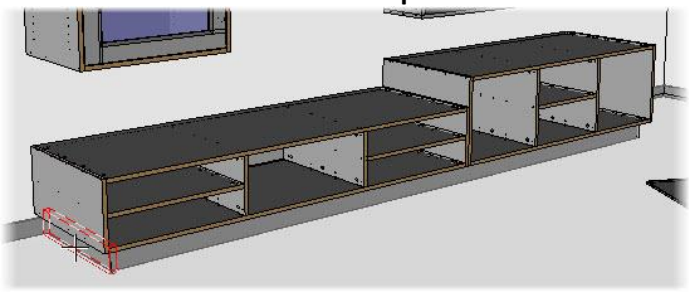
Add detailing baseboards

- Start a **new sketch**.
- Draw a contour on the wall around the part, as shown below.



- End the sketch.
- Using the **Wood** > **Extruded component** function, insert the baseboard with the **code**: **80x10**.
- Select **On curves** and select the previous sketch.

- Set:
  - **Key point**: **Bottom left point**
  - **Rotation angle**: 0°
  - **Cut = Mitre cut**
- Click on **OK** to confirm.
- Start the **Assembly** > **Use process** function.
- Select the left-hand baseboard of the cabinets as the **component to use**.



- Select **Main cut**. 

- Set **Offset**: 0mm, **Hide tools = NO** and select the baseboard on the wall as the **tool shape to use**.

- The arrow must show the side of the baseboard to delete. Click on the arrow to invert.
- Confirm with **OK** to perform the cut.
- Repeat the operation to cut the right-hand baseboard of the cabinets.

- Save the document.

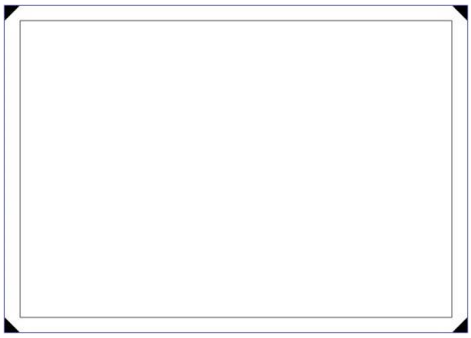
Exercise 11: Creating a draft template

Note

In TopSolid, documents can be saved as **templates**.
A **new document** can be created from this **template**, and then saved, regardless of the original **template**.
Creating draft templates allows you to preconfigure drafts with for example a specific title block, the company logo, display settings…

Creating a draft template

Create the file



- Create a **new Draft document**.
- Select **Without template** and select **Paper format: A4H** in the dropdown list.
- If necessary, delete the document's **title block**. 
- Using the **Modify element** function, modify the draft and uncheck the **Center mark**, **Orientation mark**, **Coordinate system** and **Graduations** boxes. 

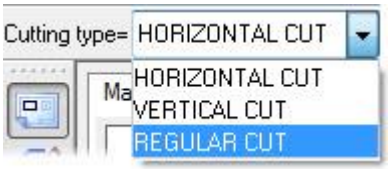
Note: In order to be available when creating a **new document**, the **template** document must be saved in the *template* folder of the *Config* or *Group* folder.
The *Config* folder corresponds to the individual configuration of the user station, while the *Group* folder corresponds to the common configuration of the different stations of the company.

- Save the file in the folder ***C:\Missler\Group\Template***.
- Rename this file *Training draft A4H*.

Create a title block

- Use the **Detailing** > **Title block** function.
- Set **Width**: 110mm and **Height**: 35mm.
- Select **Hook on border**.

It is then possible to create cells for the title block.

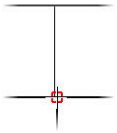


- Select **Cutting type = Regular cut** in the dropdown list.
- Set **Number of rows** = 4 and **Number of columns** = 1.

- Click on **OK** to confirm and click inside the title block to cut it.
- Reselect **Cutting type = Regular cut** in the dropdown list.
- Set **Number of rows** = 1 and **Number of columns** = 3, then click on **OK** to confirm.
- In **Cell to cut**, select the second row from the bottom.

- Select **Cutting type = Vertical cut**, and then cut the first and the third rows from the bottom as shown below.

Note: To align a cutting line with an existing line, simply click the end point of the existing line.



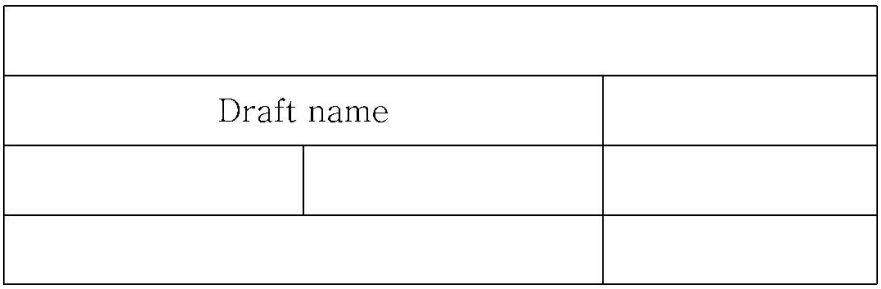
Insert text in the title block

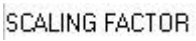
- Start the **Insert** function. 
- Select the title block as the **element near insertion place**.
- Select the left-hand cell of the third row from the bottom to insert an element in.
- In the cell, enter **Element to insert or text to create**: *Draft name.*

- Press **Enter** to validate, then select **Fast choice** > **Center center** to center the text on the width and height of the cell.

- Validate with **OK** to position the text.

Note: Inserting a plain text will make it possible to quickly modify this text when drafting to enter the draft name.



- Then select the right-hand cell of the same row.
- Select **Scaling factor** in the dropdown list. 

Note: This dropdown list contains variables that are automatically updated when using the draft document.

- Set **Fast choice** > **Center center** and validate with **OK**.
- Similarly, insert the following on the second row from the bottom:
  - Left-hand cell: **Creation date**
  - Middle cell: **Author**
  - Right-hand cell: **Folio number (i/n)**
- Then insert the variable **Property** > **Designation** in the first row.
- Validate with **OK** and select **No element**.

Note: Inserting a variable using the **No element** option allows the variable to be inserted according to the drafted element. The designation of the drafted element will be then automatically displayed here.

–		
Draft name		1:1
12/07/2013	TSW	1/1

- In the left-hand cell of the first row from the bottom, insert the name of your company.
- Then click in the right-hand cell and select the **Bitmap** option. 

Note: The **Bitmap** option is used to insert an image in the title block.

- Select the company logo you saved on your computer or the provided file *TopSolid'Wood.jpg*.
- Uncheck the **Linked to the bitmap file** option and select **Position in cell** > **Center center**.

Note: By unchecking the **Linked to the bitmap file** option, the image of the title block is not associated with the image file in Windows. This option is necessary if the draft template is used on different stations on the network or if the image is then deleted or moved.

- To avoid distorting the logo, uncheck **Resize with distortion**.
- Validate with **OK** to insert the image.

–		
Draft name		1:1
12/07/2013	TSW	1/1
Missler Software		TopSolid Wood

Configure the draft template

- Open the **document properties**. 
- In **Projection parameters**, check **Use realistic rendering**.
- **Save** and **close** this file.

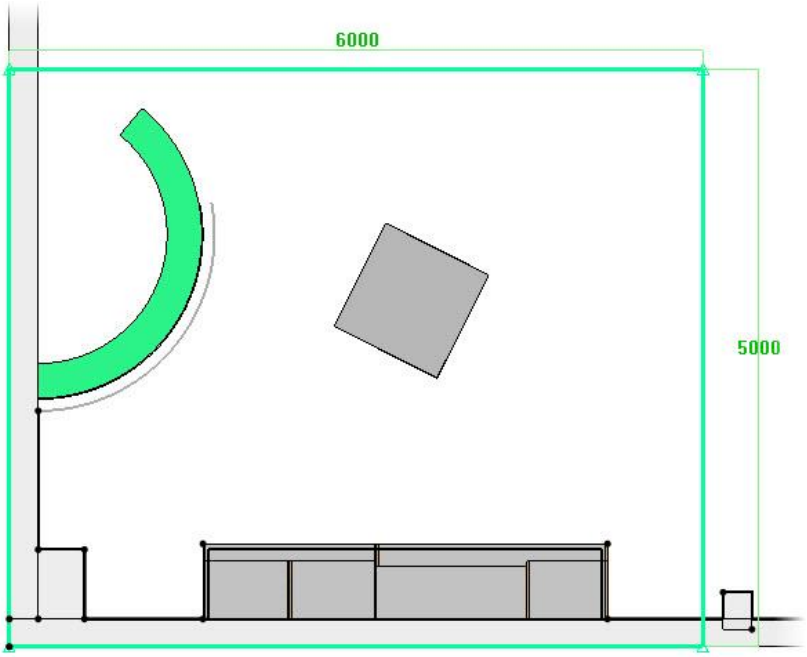
Using the draft template

Create a trimming volume

Note

When making a draft, it is possible to create a **trimming volume**. This allows you, when only a portion of the **Design** file has to be drafted, to trim the view by a block drawn beforehand.

- Open the interior design file.
- Make **layer 17** current and start a new sketch.
- Using the **rectangular contour**, draw a rectangle that encloses the interior design as shown below.



- End the sketch, then extrude it with an **offset from starting curve** = -100mm and to a **height** = 2600mm.
- Apply a **transparency** of **10** to this block.
- Start the **Tools** > **Draft trimming volume** function.

- Select the extruded volume as the **trimming volume for views**.
- **Name** this volume *tv* and click on **OK** to confirm.

- Save the file.

Create the draft and the main views

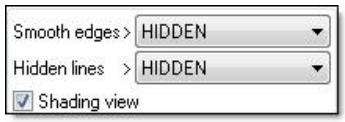
- Create a **new Draft document**.
- In the advanced parameters, select the **Training draft A4H** draft template in the **group templates**.

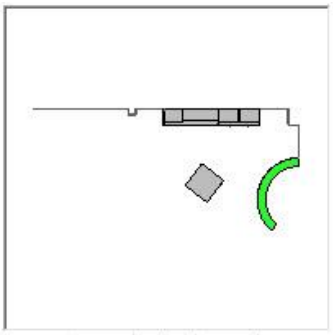
- Click on **OK** to confirm.

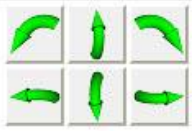
Note: A new draft file is then generated from the template.

- Create a **main view**. 
- Select **Assembly**.
- Select **Document containing the set = Interior design** in the dropdown list.

- Adjust the view as shown opposite using the **green arrows**.
- Check **Shading view**.
- Set the **smooth edges** and the **hidden lines** to **Hidden**.

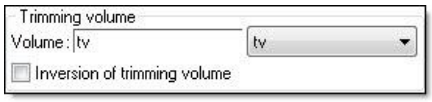






Note: The setting of the **smooth edges** corresponds to the visualization of the edges between the tangent faces.
The **hidden lines** are the non-visible edges in the projected view. Here, the **hidden lines** are **hidden** to avoid overloading the view.

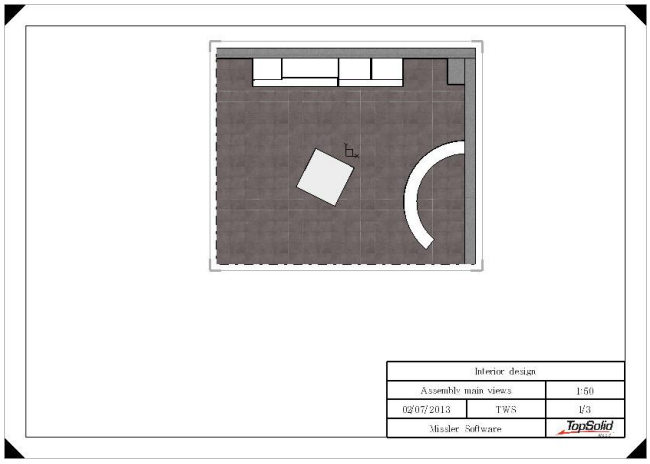
- At the bottom right of the window, select **Trimming volume** > **Volume**: **tv** in the dropdown list
- Confirm the window with **OK** to calculate the view.
- Place the view and set the **scaling factor relative to drawing** to 0.02.



- To regenerate the view, select the **Regenerate** function and select the view. 
- Use **Move parents** to position the view as shown below. 

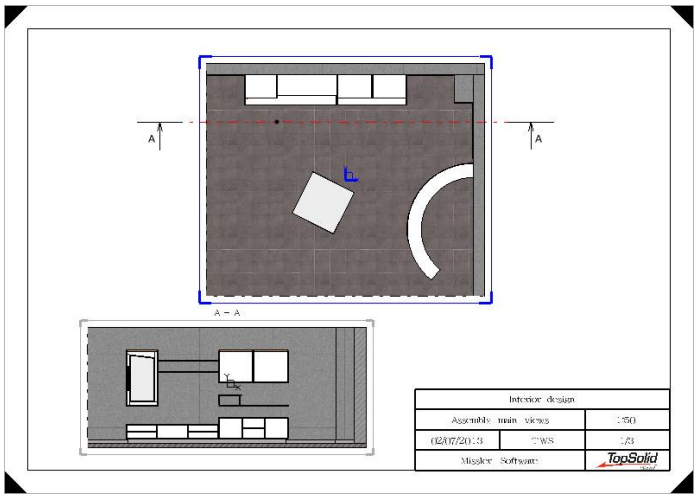
Note: For now, only the elements that make up the assembly have been drafted. Since the walls and the ground are not included in the assembly, they have not been drafted. To view them, add them manually in the view.

- Select **Window** > **Tile vertically**.
- Start the **Insert** function. 
- Select the main view in **Element near insertion place**.
- To select several elements, use the **selection lasso**. 
- In the 3D document select the walls, the ground and the columns, then validate the selection with **OK**.
- **Regenerate** the view.



- Create the **full section view** as follows: 
  - **Alignment = No**
  - **Set the section view upright = No**
  - **Hidden lines = Hidden**

- Modify the section view to check the **Shading view** option.



- **Save** this draft document in the *Interior design* folder and keep the default name.
- Use **Modify element** to modify the text of the title block: **Draft name**.
- In **Text**, enter *Assembly main views*.

Interior design		
Assembly main views		1:50
12/07/2013	TWS	1/3
Missler Software		TopSolid Wood

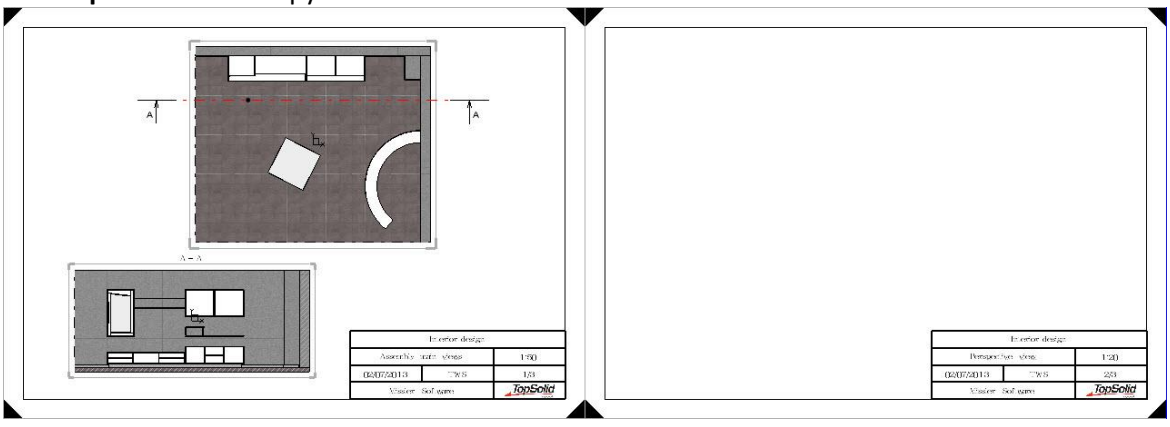
Create a new drawing

Note: It is possible to create several drawings in one document. Each drawing is called **folio**.

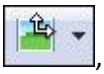
- Use **Tools** > **Drawing**. 

- Select the existing drawing as the **drawing to copy**.
- Click on **OK** to confirm the parameter window of the new drawing without changing anything.
- Set **Hook point = Bottom-Left** and select the bottom right point of the first drawing in **New position for the drawing**.

- Select the title block of the first drawing in **Title blocks and tables to copy**.
- Click on **Stop** to finish the copy. 



Create a perspective view

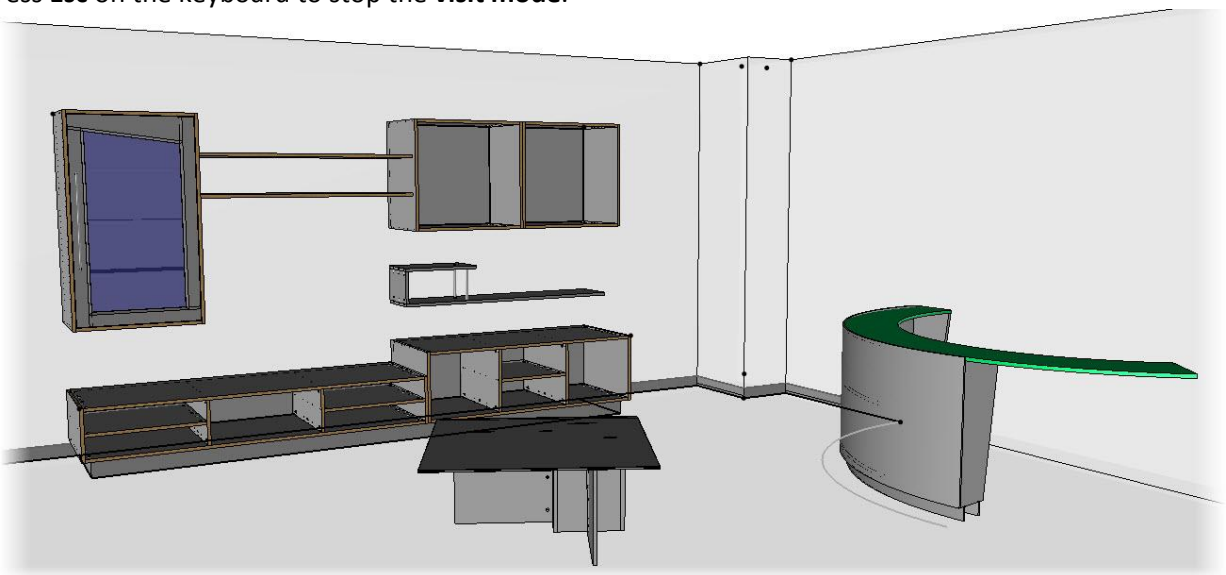
- Open the **Design** file of the interior design.
- Make **layer 15** current and switch off **layer 17**.
- Open the **View** tab  , and then select **Visit mode**. 

Note: In an interior design, the **visit mode** is used to visit the fitted-out room(s). Like a FPS video game, the mouse is used to rotate the view and the keyboard arrows allow you to move in the drawing.

- Select any point in the room in **Position of the user**.
- Select a point on the cabinets as the **target point**.

The **visit mode** starts.

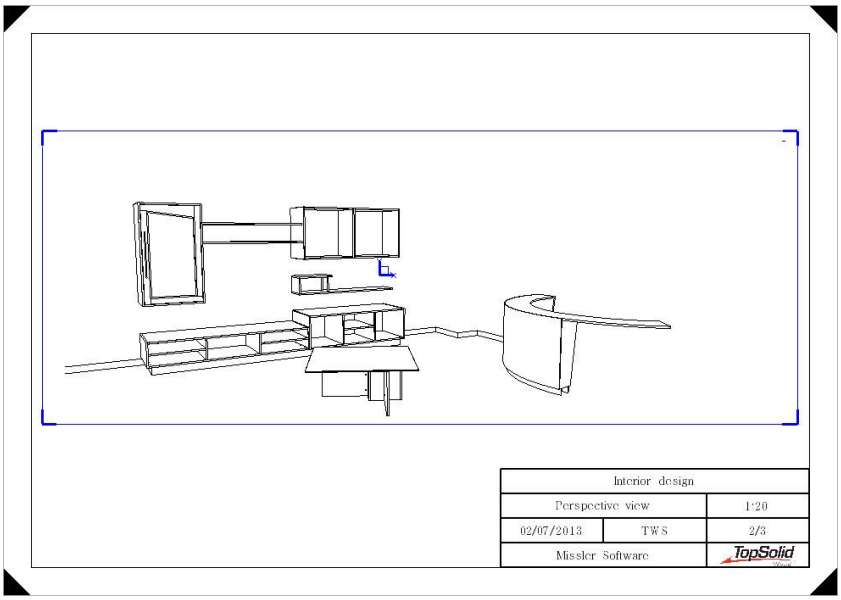
- Using the keyboard arrows and the mouse, configure the same view as shown below.
- Press **Esc** on the keyboard to stop the **visit mode**.



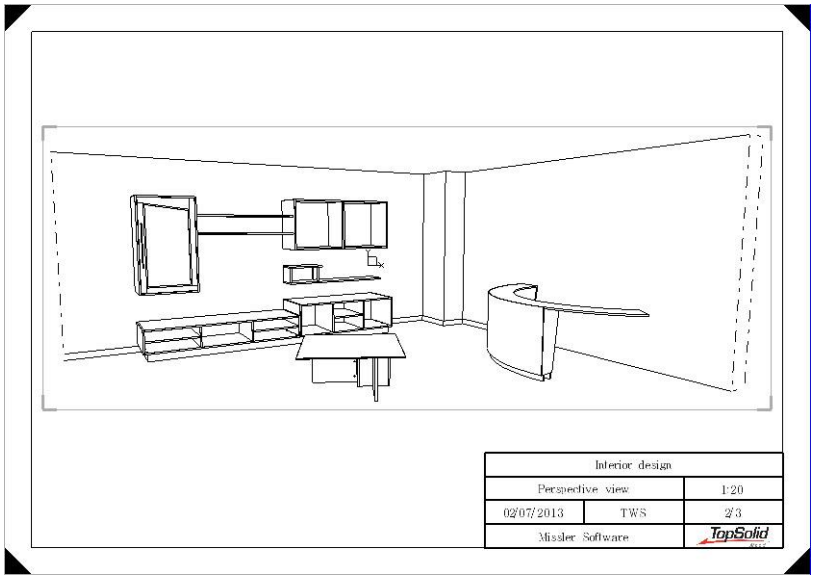
Note: To go back to the perspective view, open the **View** tab  and select **Perspective view**. 

- Select **Window** > **Tile vertically**.
- Click in the draft document to make it current.
- Create a **main view**. 
- Select the copied left-hand drawing as the **drawing in which to work**.
- Select **Assembly**, then select **Document containing the set = Interior design** in the dropdown list.
- Set the **smooth edges** and the **hidden lines** to **Hidden**.
- Select **Trimming volume**: **tv**.
- Click in the **3D or 3D coordinate system or face** field, and then click in the space of the **Design** document to generate the drawing view as the 3D.
- Validate with **OK** and position the view.
- Using **Modify element**, modify the drawing.
- Set the **scaling factor** to 0.03.
- Validate with **OK** and **regenerate** the view.

- Use **Move parents** to position the view as shown below. 
- Modify the text **Assembly main views** to *Perspective view*.



- Using the **Insert** function, it is possible to add the walls and the columns of the design document in the perspective view. 



Exercise 12: Creating a BOM template

Note

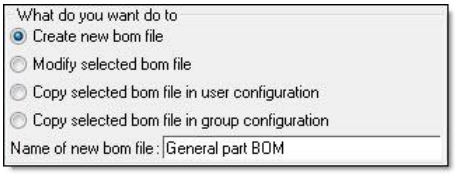
A BOM template is used to create bills of materials in a draft document, but also to export projects to Excel for example.
Creating a template allows all relevant information to be displayed in the BOM.

Creating a BOM template

- In a **Draft** document, start the **Bill of materials** > **Edit bill of material file** function. 

Note: This function can also be found in a **Design** document in **Tools** > **Edit bill of material file**.
Three tabs are available:

- **Standard**: Contains the TopSolid default BOM templates.
- **User** and **Group**: Contain the **User** and **Group** BOM templates.

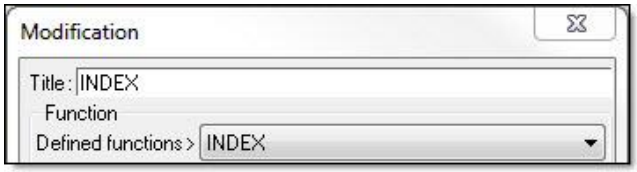


- Click on the **Group** tab.
- Select **Create new bom file**.
- In **Name of new bom file** enter *General part BOM*.
- Click on **OK** to confirm.

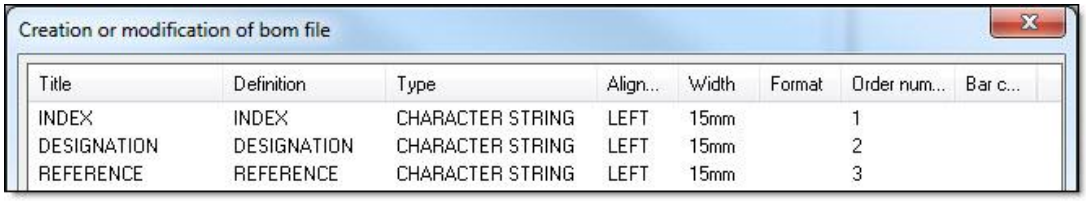
Note: The **Creation or modification of bom file** window opens. Each row corresponds to a different column of BOM information.

- Double-click in the **Definition** box on the first row to insert information.
- In the modification window, select **Defined functions** > **Index** in the dropdown list.

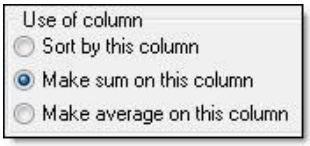
Note: The BOM functions are not alphabetically ordered.
To find the **Index** function more easily, press the '**i**' key several times in order to scroll through the functions starting with '**i**'.



- Click on **OK** to confirm.
- Repeat the operation to add the **Designation** and **Reference** functions.



- Then insert the **Count** function. Before validating the function, change the title to *QUANTITY* and check the box **Make sum on this column** to obtain the right quantity.



- Insert the **Material** and **Coating** functions.

- Insert the **Part length** function.

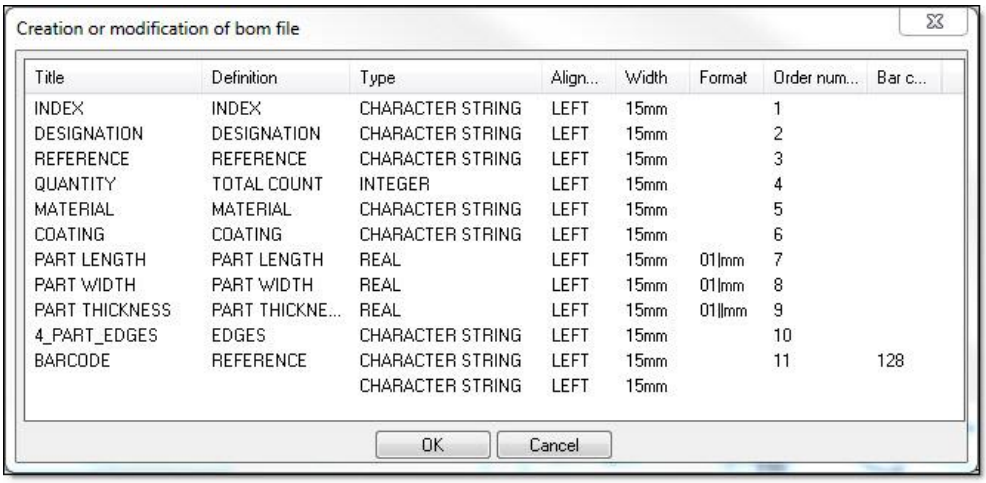
Note: By default, dimensions are in meters. The function format must be changed.

- In the **Format** field, enter *01|mm.*
  - 01 corresponds to the desired number of decimal places.
  - To make the / symbol, press **Alt Gr + 6**.
  - *mm* displays the dimensions in millimeters.

- Validate the **function** and insert the **Part width** and **Part thickness** functions in the same way.
- Then insert the **Four edges or laminates** function.
- Change the title of this function to *4_PART_EDGES*.

Note: The **Four edges or laminates** function allows the four edges placed on a part to be displayed on the same column.

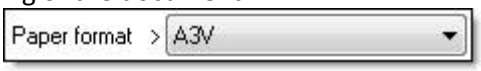
- Finally, insert the **Reference function** again:
  - Modify the title of this **function** to *BAR CODE.*
  - Select **Bar code** > **128** in the list to display this column as follows: Bar code 128.
- Click on **OK** to validate the BOM template.



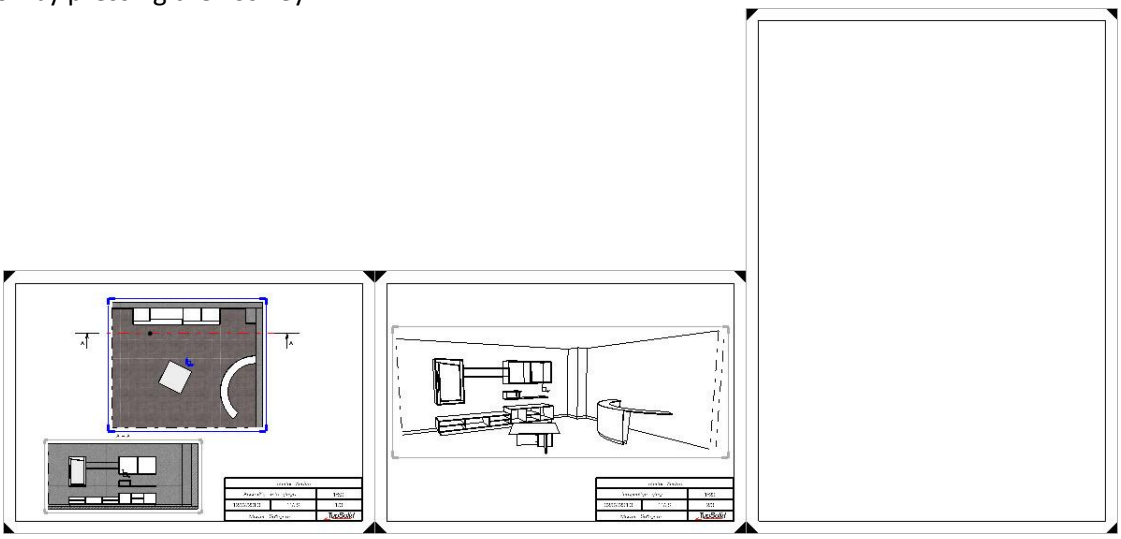
Using the BOM template

Create a new drawing

- Open the Design document *Interior design.top*, then the Draft document *Interior design.dft*.
- In the draft, Start the **Tools** > **Drawing** function and select a drawing of the document.
- In the drawing modification window, select **Paper format** > **A3V**.



- Using the **hook point Bottom-Left**, position this drawing to the right of the second drawing, and exit the function by pressing the **Esc** key.



Create the BOM

- Open the **Bill of Material** context.
- Start the **Bill of material** function.

- In the **Group** tab, select the **General parts BOM** template, and then click on **OK** to confirm.

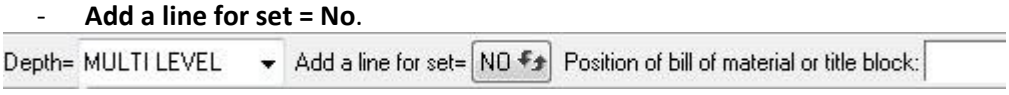
- Select the A3V drawing as the **drawing in which to work**.
- Select **Assembly** to include all the elements of an assembly in the BOM.

- Select **Document containing the set = Interior design** in the dropdown list.

Note: A BOM can be displayed according to three depths:

- **At top level**: The sets of the first level are displayed.
- **Flat BOM**: The elements of the lowest levels are displayed.
- **Multi level**: The sets can be unfolded in order to display the component elements.

- Set:
  - **Depth = Multi level**.
  - **Add a line for set = No**.



Note: A bill of material can be placed over a title block or between two points.

- In **Position of bill of material or title block**, select the bottom left point of the A3V drawing's frame, then the bottom right point as the **second alignment point**.
- Exit the function by pressing the **Esc** key.

Unfold the BOM

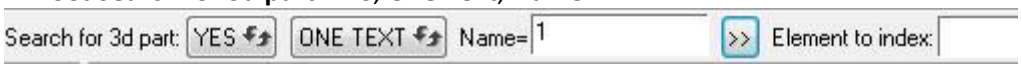
- Start the **Modify element** function. 
- Select the bill of material.
- Go to the **Bill of material level choice** tab.

Note: This tab is used to modify the depth of the bill of material. In **multi-level depth**, the sets can be unfolded to display the component parts.

- Using the **+** icon, unfold the **Vertical separation panel component** set, then the **Glass panel door** and **Wall shelf** sets, and then the **Free-running sided cabinet** set.
- Validate the window with **OK** to calculate the desired BOM.

Index elements

- In the **Bill of material** context, start the **Index** function. 
- Set **Search for 3d part: YES**, **One Text**, **Name** = *1*.



Note: **Search for 3d part: YES** is used to index 3D file elements.
One text displays the part index only.
Name = *1* is used to set the created index number. The next indexes will be incremented, based on this value.
>>: Use the advanced parameters to define the type of positioning of the balloon, as well as its style.

- In **Element to index**, select an element in one of the three drawing views.

The **index** is created.

- Place the index on the drawing, then click to validate.

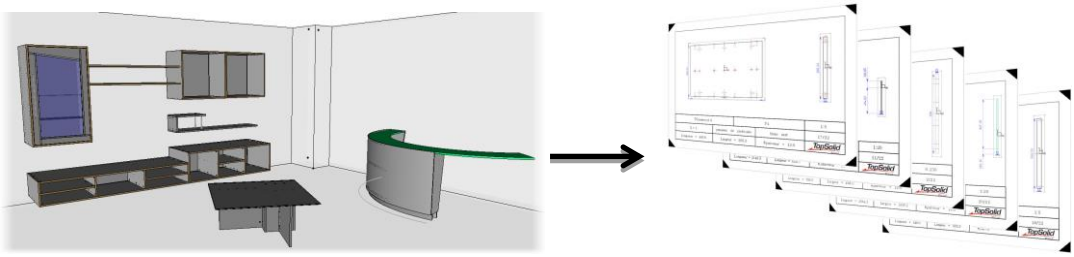
Note: As the bill of material is in **multi-level**, it is possible to index two different BOM elements with the same selection.
For example, in the case of a **free-running sided cabinet**:

- In **Bill of material** the **Free-running sided cabinet** set, as well as the **Top** panel are displayed.
- When selecting the element to index, if selecting the top, it is possible to index either the **free-running sided cabinet**, or the **top**.
- The **Bill of material level choice** opens and the element to be indexed is required.
- Select the element to be indexed and click on **OK** to confirm the window.

Exercise 13: Creating a multi-draft

Note

Creating a multi-draft automatically generates, from a specific template, the draft of several parts and/or assemblies by dimensioning them and including information in the title block automatically.



Creating the multi-draft template

Create the template

Note: To create a multi-draft, a specific draft template should be created. Here, the template will be created from the draft template previously created.

- Open the file *Training draft A4H* from the folder *C:\Missler\Group\Template*.
- Use **File** > **Save as**, and then save this file in the folder *C:\Missler\Group\Template* by renaming it *Multi-draft A5H*.
- Modify the draft frame using **Modify element**. 
- Select **Paper format** > **Not standard** in the dropdown list.
- Then set:
  - **Width**: 210mm
  - **Height**: 148.5mm
  - **Name**: *A5H*

- Click on **OK** to confirm.

Create the views

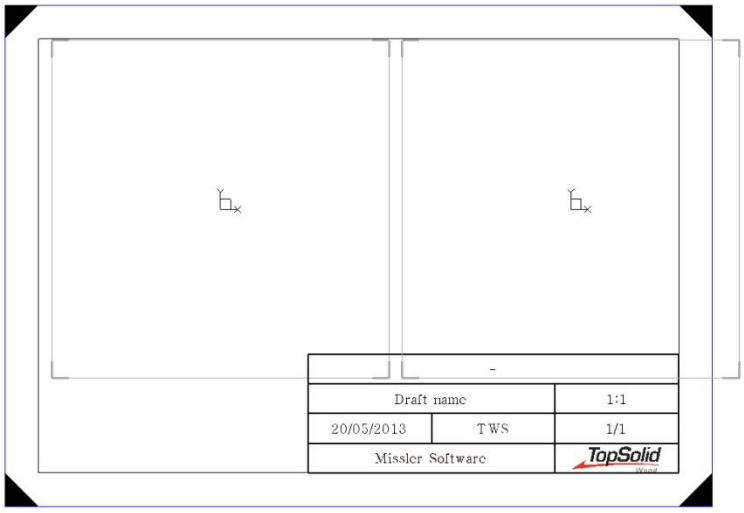
- In the **Wood** context  , start the **Multi-draft** function. 
- Select **Create template**, and then **Create view**.

Note: **Create view** is used to position a view of the part or the assembly which will be drafted.
Three types of views are available:

- **2D view**: It is the view configured from the part definition's **Drawing** tab. By default, it is oriented like the sawing up coordinate system.
- **First machining/second machining**: These views are configured from the **First positioning** and **Second positioning** tabs of the part definition's **Machining** tab. By default these views are identical to the sawing up coordinate system, and can be used to generate draft documents for machining.

- Select **View**: **2D** in the dropdown list and click on **OK** to confirm.
- Keep the default **scaling factor** to 1, and then position the view on the left-hand side of the drawing.
- Then click on **Auxiliary view** and position a side view on the right-hand side of the drawing.

Note: The view frames may go beyond the title block or the draft frame. This does not matter since the views are automatically scaled and recentered on the drawing when generating the multi-draft.



Set the template

- Open the **document properties**. 
- Go to **TopSolid'Wood properties** > **Draft** to set the multi-draft.
- In **Elements to dimension**, tick:
  - **Dimension drawings** to dimension the part views.
  - **Part dimensions** to dimension the part dimensions.
  - **Edge dimensions** to place notes on the part's edges.
  - **Codification** to note the edge codifications on the part.
- Uncheck all the other boxes.

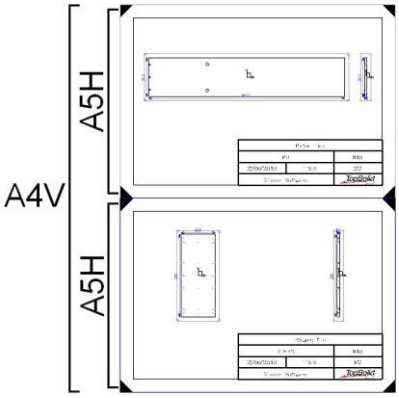
Note: To calculate the scale factor, two options are available:

- **Free scale factor**: The draft scale factor is calculated as accurately as possible, according to the clutter in the view, the title block and the dimensions. However, this mode can generate non-conventional scale factors (for example 0.127).
- **Scale factor chosen in the list**: The scale factor is first calculated as accurately as possible, but the scale factor used is the nearest lower one in the list.

- Select **Scale factor chosen in the list**.
- Click on **Add** to insert the new scale factors of 0.05 and 0.15.
- Validate the window with **OK**.
- Use the **Delete element** function to delete the text **Draft name**.
- Start the **Insert** function. 
- Select the title block, and then the box in which the text has been deleted.
- In the dropdown list, select **Property**, then **Reference**. Click on **OK** to confirm.
- Select **No element**.
- In the positioning window, select **Center center**, then click on **OK** to confirm.
- Save and close this document.

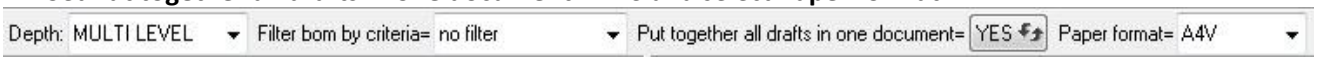
Making the multi-draft

- Open the design document *Interior design*.
- In the **Wood** context, select the **Multi-draft** function. 
- Set **Depth**: **Multi level** in order to view all the parts of the assemblies.



Note: When creating the multi-draft, it is possible to put on a paper format two drafts in lower format.
For example, as the draft template created is a horizontal A5 format, two drafts can be placed on a vertical A4 paper format.

- Set **Put together all drafts in one document = YES** and select **Paper format = A4V**.



- Click in the graphical zone to choose this file.
- Select **User templates** > **Multi-draft A5H** in the list.

All parts made of particule board will be drafted.

- Tick the **Material** box and select **Particule board** in the dropdown list.

- Confirm the selection with **OK**.

All the parts are then automatically drafted.

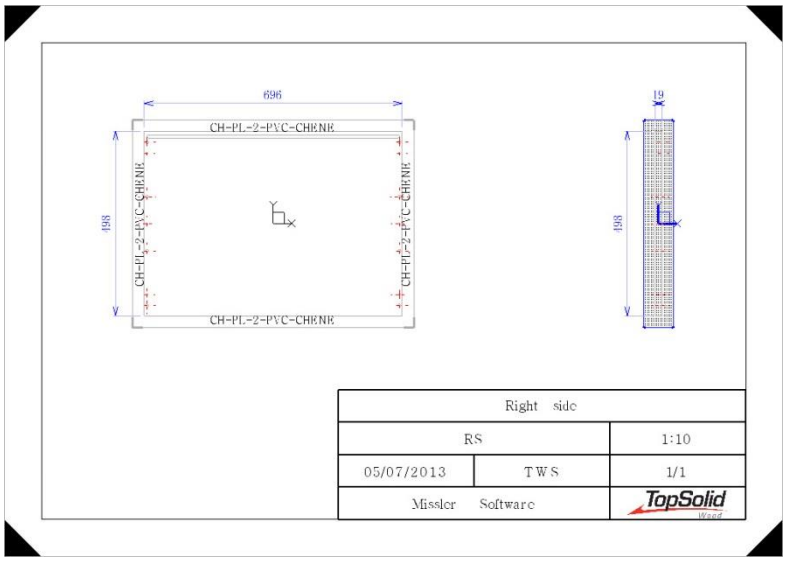
- Save this file in the folder *Interior design* and rename it *Draft of the parts*.

Place the axes automatically

In a **Draft** document, it is possible to place the axes of all the drafted circles automatically.

- In the **View** context, start the **Axes** function. 
- Select **Projected axis**, **All views**, **Automatic** and **OK**.

Note: The **All views** option is used to place the axes on all the document's views and the **Automatic** option is used to place the axes on all the document's cylinders.
It is also possible to define a **minimum radius** and a **maximum radius** so that the axes are not placed on certain cylinders.



Note: The part drafts of a multi-draft behave in the same way as a simple draft. This makes it possible to modify, move and dimension the views, or modify the drawings.

Supplement: Inserting information on the part in the title block

- Open the multi-draft template *Multi-draft A5H* in the folder *C:\Missler\Group\Template*.
- Delete the existing title block.
- Recreate a title block using the **Detailing** > **Title block** function.
- Enter **Width**: 190mm and **Height**: 30mm. Select **Hook on border** to position the title block.
- Perform a vertical cut at about 35mm from the right-hand segment.
- Perform:
  - A regular cut on a column and three rows of the two cells so as to obtain the following result:

  - A regular cut of a row and two columns on the top left row.
  - A regular cut of a row and three columns on the two bottom left rows.

- Using the **Insert** function, insert the following in the three left-hand cells:
  - the logo
  - **the folio number (i/n)**
  - **the scaling factor**

1:1
1/1
TopSolid Wood

Note: To insert additional variables in a title block, first create a bill of material in order to retrieve the information in the title block.

- In the **Wood** context, start the **Multi-draft** function.

- Select **Create template**, and then **Create bill of material**.
- In the **Group** tab, select the **General part BOM** template, and then click on **OK** to confirm.
- Click any two points to the left of the template to place the BOM.
- Click on **OK** to confirm.
- In order for the BOM not to be displayed during the multi-draft, use the **Mode/Visibility** function and select the title block and the two points you just clicked to hide them. 
- Start the **Insert** function. 
- Select the title block, and then the upper left-hand cell.
- Select **BOM property** in the dropdown list. 

The different columns of the BOM are available.

- Select **Designation**. 

- Select **No element**.
- Set **Fast choice** > **Center center** to center the text in the cell, and then click on **OK** to confirm.
- Select the other cells to insert the information below:

Designation		Reference	1:1
Quantity	Matter	Coating	1/1
Length	Width	Thickness	TopSolid Wood

Note: As there is no information to be displayed at the moment, the variables are shown as **–** (dash).

In order to make the reading easier, it is possible to add text with the variables.

- Using the **Modify element** function, modify the **Part_length** variable (lower left-hand cell).

Note: The variable is displayed between the symbols < and > as follows:
<text_displaying_bill_of_material_information_n>

- Insert *Length* = before the variable.

Length = **<text_displaying_bill_of_material_information_6>**

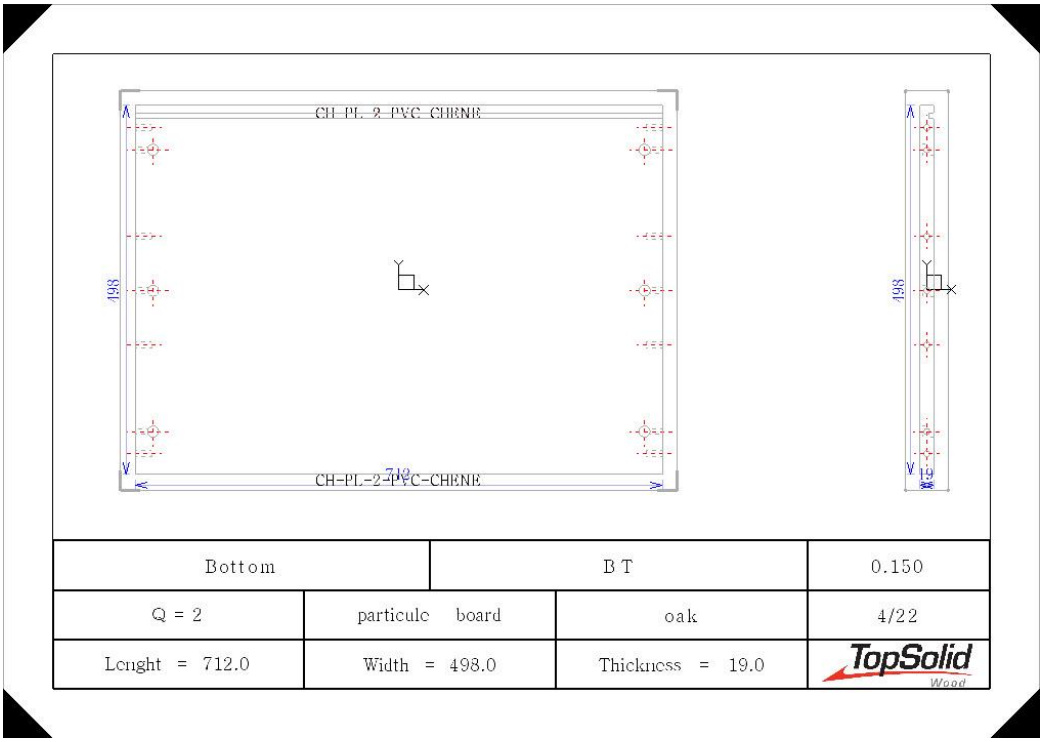
The text *Length* = will be displayed before the variable.

- Repeat the operation in order to obtain the following title block:

-	-	1:1
Q = -	-	1/1
Length = -	Width = -	Thickness = -	TopSolid Wood

- Open the document properties. 
- Open the **Table/Bill of material** > **Bill of material** section.
- Check **Automatic update of bills of material**, and then click on **OK** to confirm.
- Save and close this file.
- Open the file *Interior design.top*.
- Restart the multi-draft by following the same procedure described on page 109.

The information relative to the parts are then automatically included in the title block.

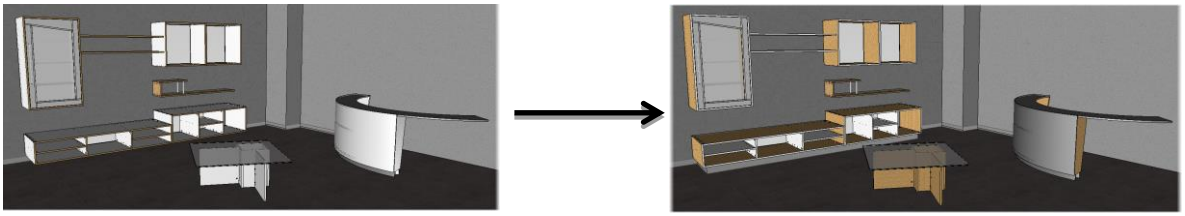


Bottom		B T	0.150
Q = 2	particulc board	oak	4/22
Lenght = 712.0	Width = 498.0	Thickness = 19.0	TopSolid Wood

Exercise 14: Configuring the project

Note

Once a project is complete, some functions allow you to **configure** it.
For example, you can modify the part material or the panel edges without modifying the library template. The configured elements are then specific to that project.



- Open the project *Interior design.top*.
- In the **Wood** context, start the **Matters configurator** function.

Note: There are four types of configurations:

- **Matter configuration**: Used to modify the part matters.
- **Edge configuration**: Used to modify the panel edges.
- **Laminate configuration**: Used to modify the panel laminates.
- **Property configuration**: Used to modify the part properties (**Designation**, **Reference**, **Part type**…).

Edges and laminates can be configured in the assembly only if they have been made **modifiable in the assembly** via this same configurator from the template document.

- Select **Main assembly** to take into account all the parts of the assembly in the configurator.

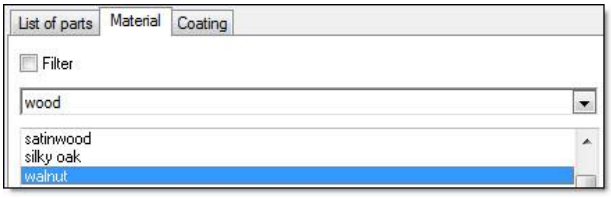
MAIN ASSEMBLY

The configuration window opens. The top part of the window is used to set the type of part selection. Three modes are available:

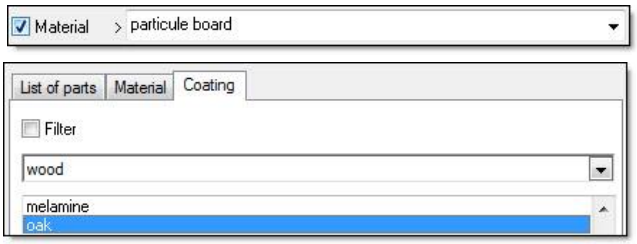
- **Select manually**: The parts to configure can be selected directly from the graphical zone or from the list of parts with a **right-click** > **Select**.
- **Select parts by material**: The selection by material includes all the parts with the same matter and thickness.
- **Select by criterions**: This selection is used to select parts according to one or several criteria, among those proposed by the software (**Type**, **Material**, **Coating**, **Thickness** and **Property**).

- Activate the **Select by criterions** mode.
- Check **Material** and select **oak** in the dropdown list.
- In the **Material** tab, select **wood** > **walnut**.
- Click on **Apply matter** to apply the chosen material to the selected parts.

APPLY MATTER



- Select **Material** > **particule board** in the dropdown list.
- In the **Coating** tab, select **wood** > **oak**.

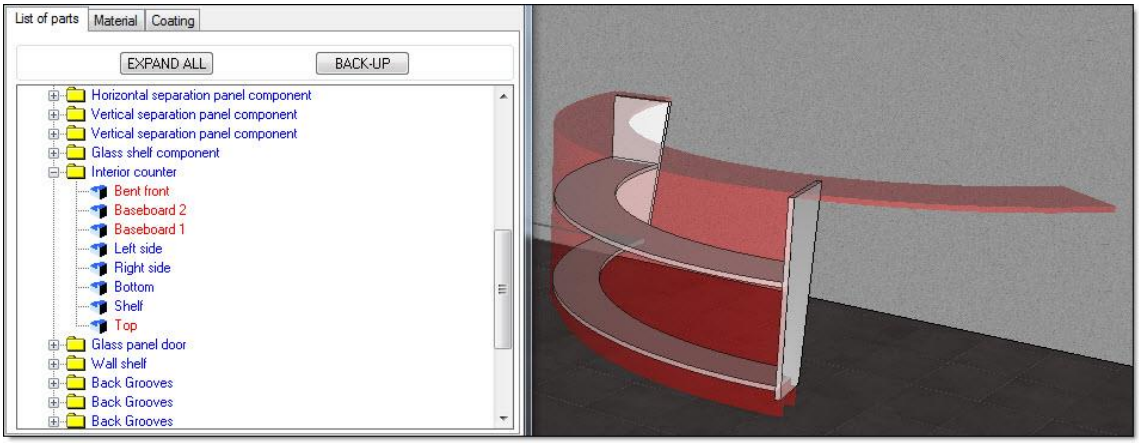


- Click on **Apply coating** to apply the chosen coating to the selected parts.

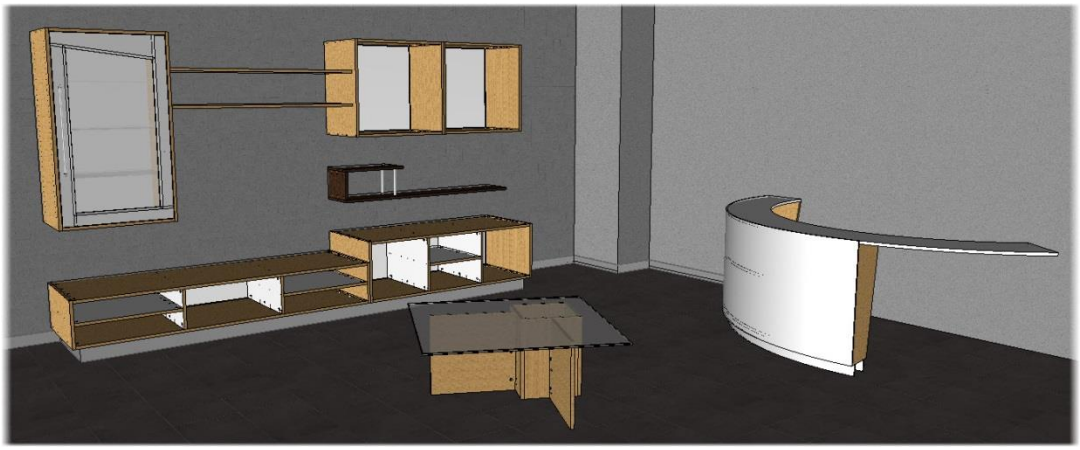
- Keep checked **Material** > **particule board**.
- Check **Thickness** > **8mm**.

- In the **Coating** tab, select **paint** > **mat white paint**.
- Apply the coating using the **Apply coating** button.
- Uncheck **Material** and **Thickness**.
- Check the **Property** option and set **Designation** > **Fixed shelf**.
- Apply the coating: **paint** > **mat white paint**.

- Activate the **Select manually** mode.
- In the graphical zone select the **bent front**, the **top**, as well as the **baseboards** of the counter.



- Apply the coating: **paint** > **mat white paint**.
- Click on **OK** to validate the window.
- Save the file.



Notes

Individual course evaluation form

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

TopSolid'Wood - 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	Not really	Quite	Yes
Does the overall duration of the course seem appropriate?	☐	☐	☐	☐
If no, was it?	Too short ☐		Too long ☐	

PACE	No	Not really	Quite	Yes
Does the overall pace of the course seem appropriate?	☐	☐	☐	☐
If no, was it?	Too slow ☐		Too fast ☐	

USE OF ACQUIRED KNOWLEDGE IN THIS TRAINING	No	Somewhat no	Somewhat yes	Yes
Have you found this training to be useful in your work?	☐	☐	☐	☐
Do you think you can put the acquired knowledge into use quickly?	☐	☐	☐	☐
Do you believe that you have achieved your objectives upon completion of this course?	☐	☐	☐	☐

Comments and suggestions: .....

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