Training Guide
TopSolid'Wood Advanced

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

Contents

Creation of a guitar body

The purpose of this first section is to create 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, dimensions and constraints
- **Extrusion**
- **Wood machinings**: Pockets, drillings and mouldings
- **Data structure**: Part definition

Creation of the body

Start the sketch

- Create a **new Design document**.
  In the advanced parameters, select **Without template**. Select the **Associative** design mode and then the **Millimeters** unit.
- Click on **OK** to confirm.

- Activate the **Sketch** context , and then select **New sketch**.
- Select **Current coordinate system**. CURRENT COORDINATE SYSTEM
- Draw a first **line 1** as shown opposite using the **Axes (Z) = YES** mode.

  AXES (Z)= YES

- Apply a **coincidence constraint** of this line to the **Y axis**. >

- **Dimension** this line with a **nominal value** of *80mm.*
- After applying the value, select **Symmetry constraint** in the dialog bar, and then select **X** as the **symmetry axis**.

  SYMMETRY CONSTRAINT [X] Symmetry axis:
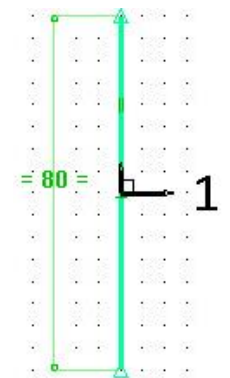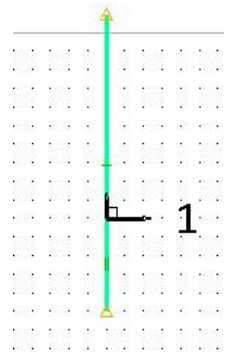
Note: Applying a **symmetry constraint** to a dimension allows the dimension to be automatically centered on the selected axis, regardless of its length.

- Use the **Circle** function in **Passing point** mode to draw the two circle arcs **2** and **3** as shown opposite. Passing point
- Draw the line **4** after the circle arc **3** as shown opposite.
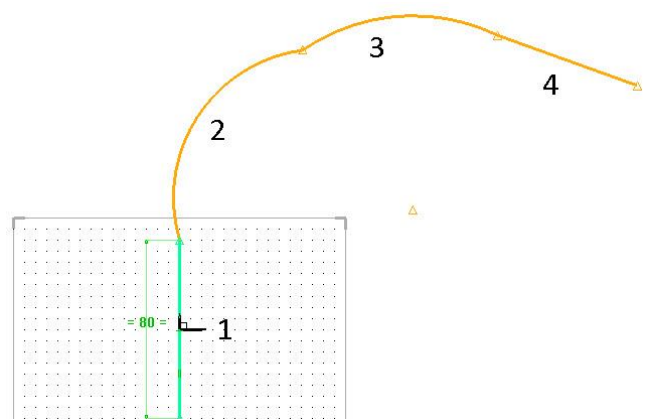
- **Dimension** the point between the segments **2** and **3** and the two points of line **4**.
- Modify the dimension values and apply the values shown opposite.
- Apply a **tangency constraint** between **1** and **2** and between **2** and **3**

Note: All the sketch elements are green, which means they are totally constrained.
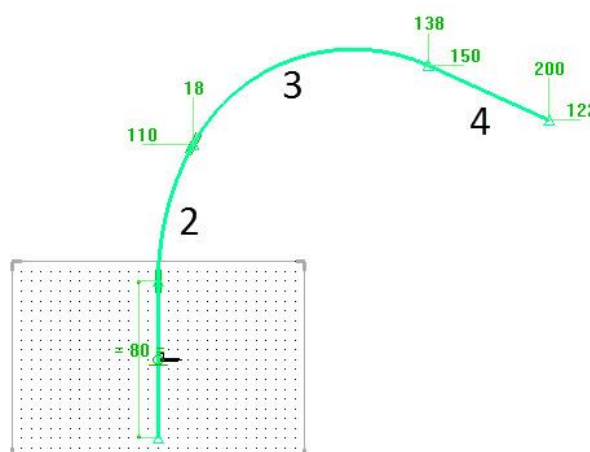
- Apply a **tangency constraint** between the circle arc **3** and the line **4**.

Note: The sketch was already fully constrained. Adding a constraint makes the design overconstrained and the document is invalid (a question mark icon appears in the title bar of the software). Extra constraints and dimensions must be deleted.
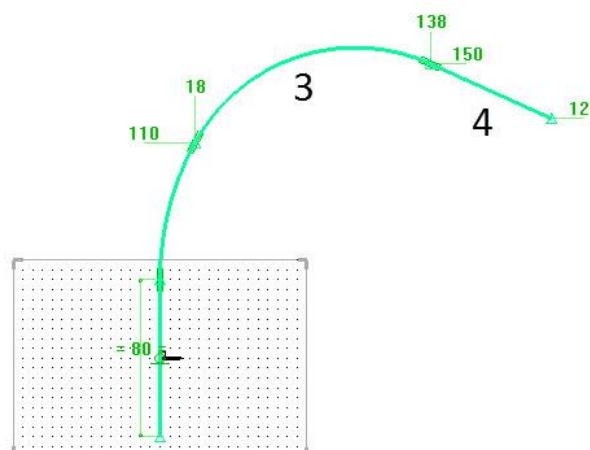
- Remove the dimension on **X** (vertical dimension) of 200mm from the right point of the line.

- Select **Show extremities**.

Note: The **Show extremities** function displays:

   - The two ends of an open contour using dotted circles.
   - The area of a closed contour using hatch lines. If the sketch contains several closed profiles, each of them will have a different hatching pattern with a different color.

This design mode makes it easy to see whether the contour being created is closed or open and, if so, quickly see where it is open.
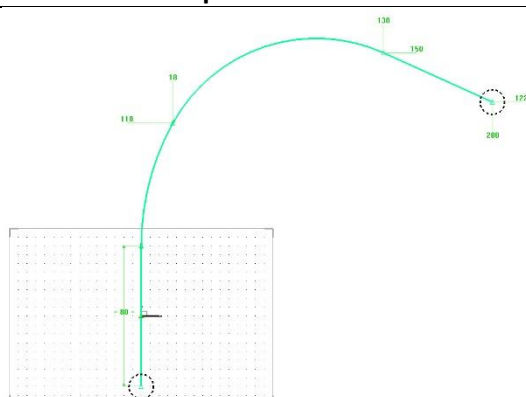
Open contour	Closed contour
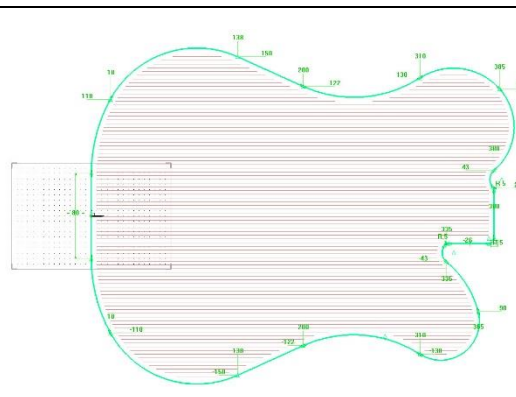
- Select **Modify height**.

Note: The **Modify height** function allows you, in the current sketch, to adjust:

- the height for dimensions;
- the height for constraints;
- The circle radii at the ends of open profiles.

These values are saved per sketch. You can modify these default values in **Tools** > **Options** > **Sketch**.

- Adjust:

 - **Constraints height**: *12*
 - **Circle radius for profile extremities**: *8*

| OK | Dimensions height= 3.175 | Constraints height= 12 | Circle radius for profile extremities= 8 |

- Validate these settings with **OK**.

- Draw 4 **circle** arcs (5, 6, 7 and 8) in **Passing point** mode as shown below.
- Then **dimension** the points with the values specified below.
- Apply **tangency constraints** between the arcs **5** and **6**, **6** and **7**, **6** and **7** and **7** and **8**.
- Also apply a **tangency constraint** between the line **4** and the arc **5**.
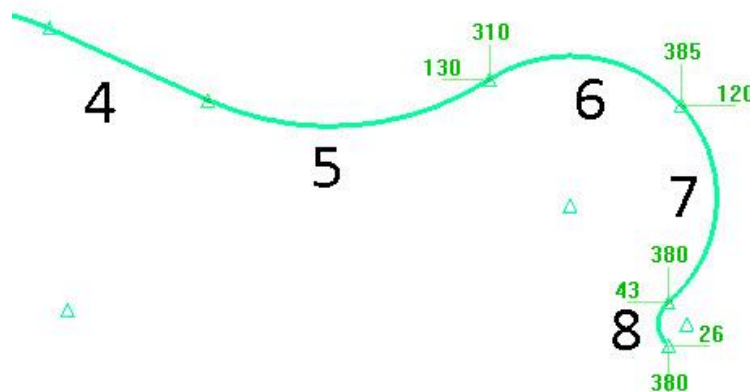
Copy the sketch

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

Template elements to repeat:

- Click on **OK** to confirm.
- Select **Simple mirror** as the **propagation**.

SIMPLE MIRROR

- Select the **ZX** plane as the **symmetry plane**.
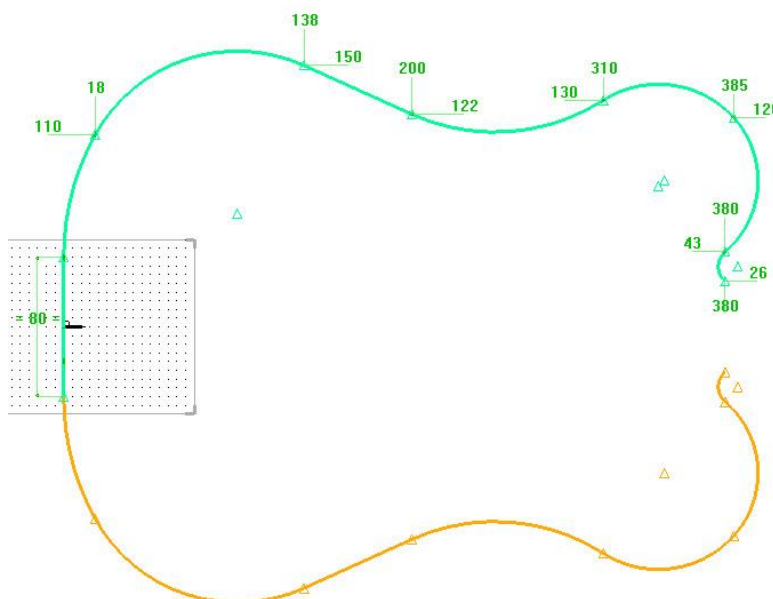
- **Dimension** the points of the copied elements.
  Do not change the values of these dimensions.
- In order not to overconstrain the sketch, remove the dimension on **X** (vertical dimension) of 200mm from the right point of line **4'**.

- Then apply a **tangency constraint** between **1** and **2'**.
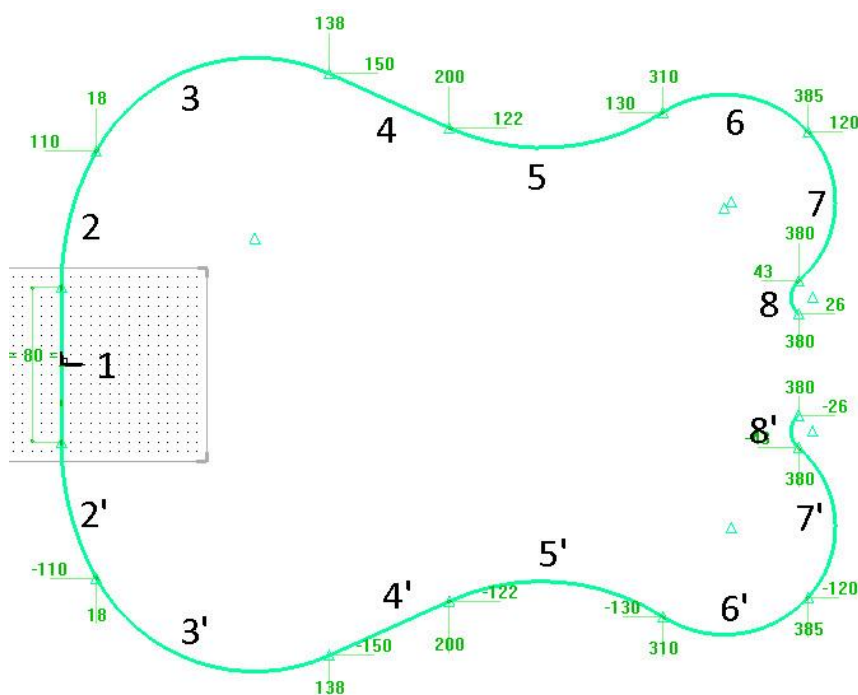
- Use the **Modify parameter** function to change the dimension values of arcs **6', 7'** and **8'** as shown opposite.
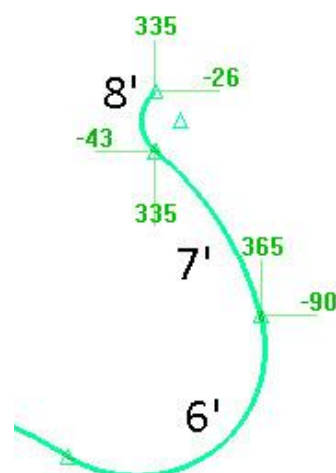
Close the sketch

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

Note: Once the contour is closed, the hatch lines are displayed.

- Apply:

  - an **orientation constraint along X** to the line **9**;
  - a **perpendicularity** constraint between lines **9** and **10**.
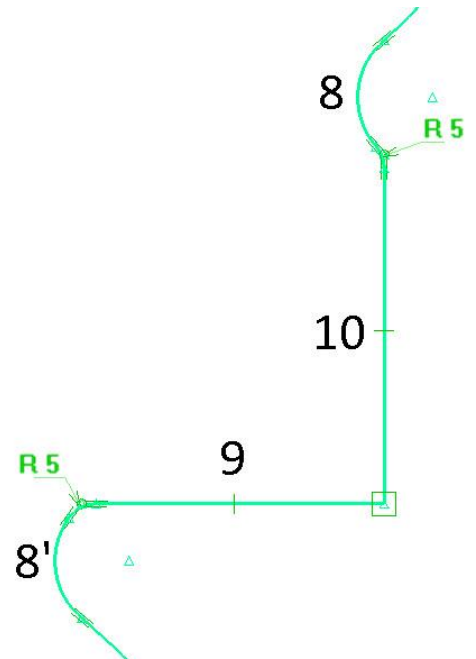
- Select the **Fillet** function.
- Adjust:

  - **Mode = Global**
  - **Fillet radius** = *5mm*.

Mode: GLOBAL ⨍± | RADIUS INTER/EXTER | Fillet radius= 5mm | Curve to modify:

Note: The **Global** mode is used to apply the fillet onto all the sketch's vertices, except onto tangent vertices and vertices where the fillet cannot be performed.

- Select the contour as the **curve to modify**.

Note: The fillets that will be created are previewed in red. Yellow circles are shown on vertices where the fillets cannot be performed.

- Validate the fillets using the **Compute fillet(s)** button. COMPUTE FILLET(S)
- **End the sketch**.

Extrude the body

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

Height: 35
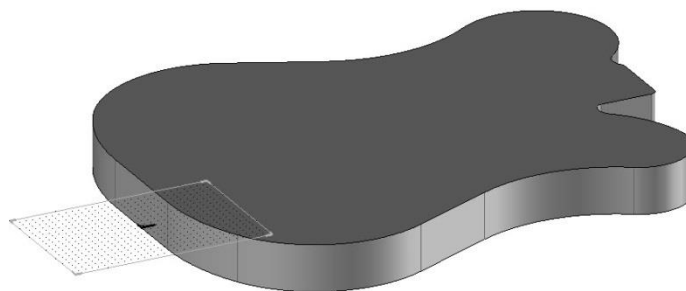
- Save this document:

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

Creation of operations

Create the pockets

- Make **layer 1** current.
- From the **Sketch** context, select **Contour** > **Rectangular**.
- Draw a rectangle as shown opposite and select

    Auto dimension AUTO DIMENSION to automatically place the dimensions of the rectangle.
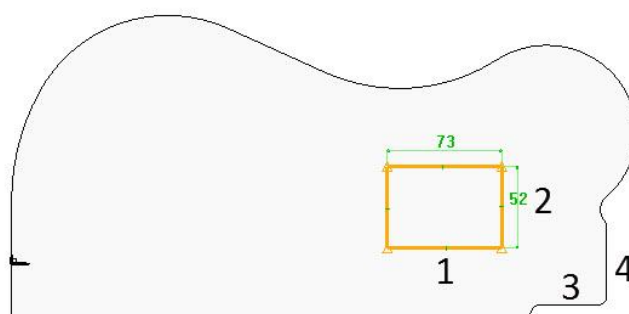
- Use the **Modify parameter** function to change the values to *73mm* on the X axis and to *52mm* on the Y axis.
- Apply two **coincidence constraints**:

    - 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* in **Local** mode onto the rectangle's left angles.
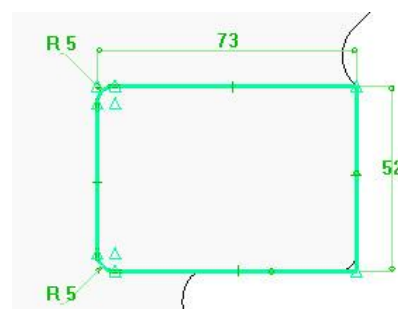
Mode: LOCAL ⨎⯅ | RADIUS INTER/EXTER | Fillet radius= 5mm | Curve to modify:

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

Constraints= PERPENDICULARITY ⨎⯅ | First diagonal point:

- Select **Auto dimension** to place the dimensions of rectangles.

- Use **Modify parameter** to change the dimensions of rectangles:

    - *15mm* on X and *60mm* on Y for rectangle 2;
    - *18mm* on X and *70mm* on Y for rectangle 3.

- Start the **Modify element** function.
- Select the vertical dimension of rectangle 3 (70mm) and select **Constraint** on **X**.

CONSTRAINT

- Repeat the procedure on the rectangle's 2 vertical dimension (60mm).
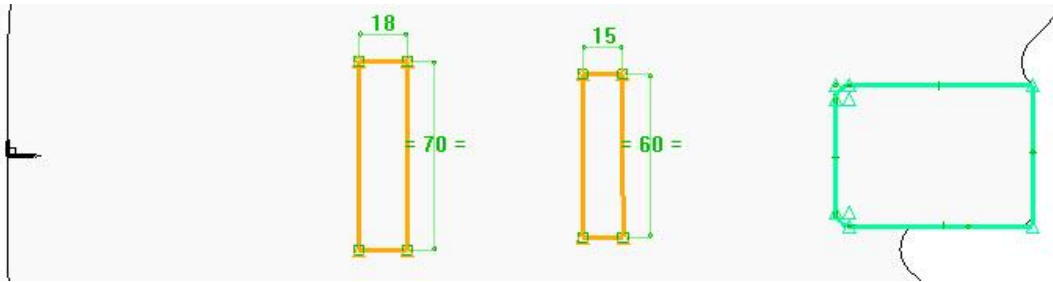
- **Dimension**:

    - the distance between rectangles 1 and 2 to *20mm*;
    - the angle between the left side of rectangle 3 and the X axis of the coordinate system to *105°*;
    - the distance between the left side of rectangle 2 and the bottom right point of rectangle 3 to *100mm.*
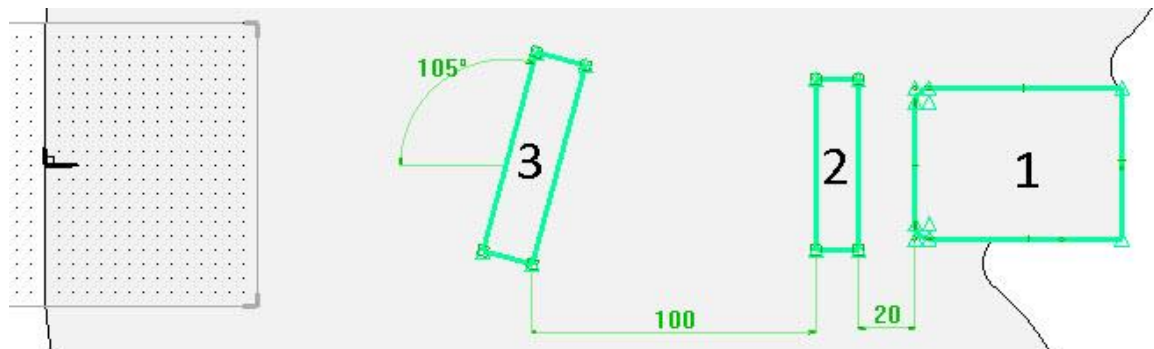
- Apply **fillets** with a **radius** = *7mm* onto the angles of rectangles **2** and **3**.
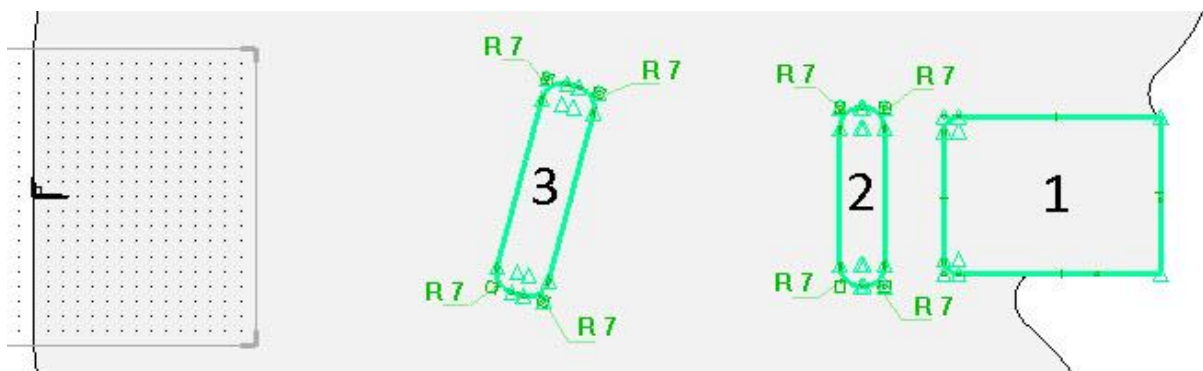
- **End the sketch**.

- From the **Wood** context, select **Pocket**.
- Select the top of the guitar body as the **reference face**.
- Set **Generatrix = Hidden** and **Generatrix sketch = Global**.

Generatrix= HIDDEN ↯ EXTRUSION DIRECTION Generatrix sketch= GLOBAL ↯ Curve(s):

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

- Select the drawn sketch in **Curve(s)**.

The **Pocket parameters** window opens.

- Adjust:

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

Pocket parameters			
Depth			
Through			
● No	○ 1 time	○ Through all	
Depth : 15mm			

- Validate the window with **OK** to create the pockets.
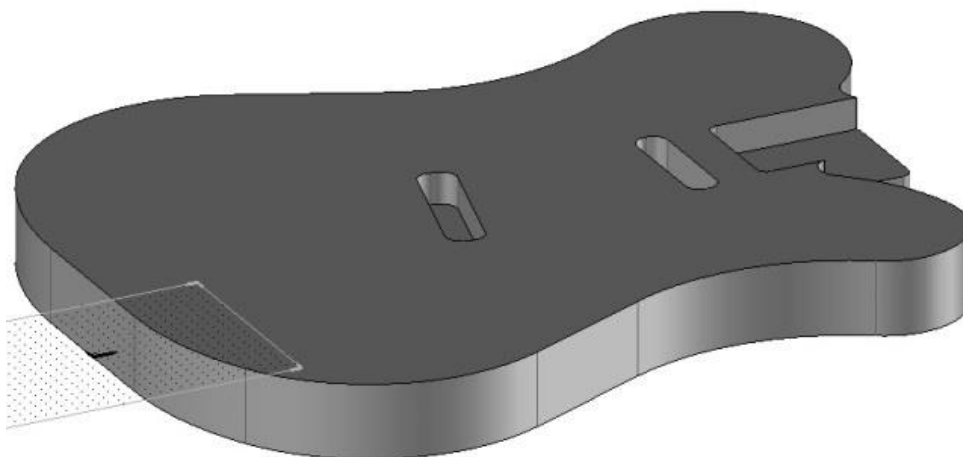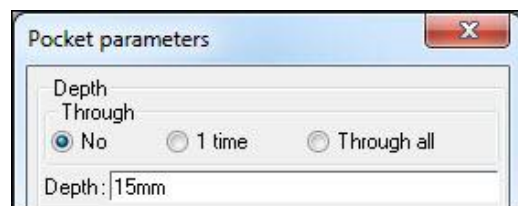
Create the drillings

- Start the **Drilling** function.

- Set **Coordinate system = Constraint**, **Mode = Dynamic** and **Hook = Current coordinate system**

Coordinate system= CONSTRAINT ▾ Mode= DYNAMIC ⚡ Hook= CURRENT COORDINATE SYSTEM ⚡ Face to drill:

Note: The **Dynamic mode** allows a drilling to be positioned and dimensioned at the same time.
The **Hook = Current coordinate system** mode 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 the **Modify parameter** function.
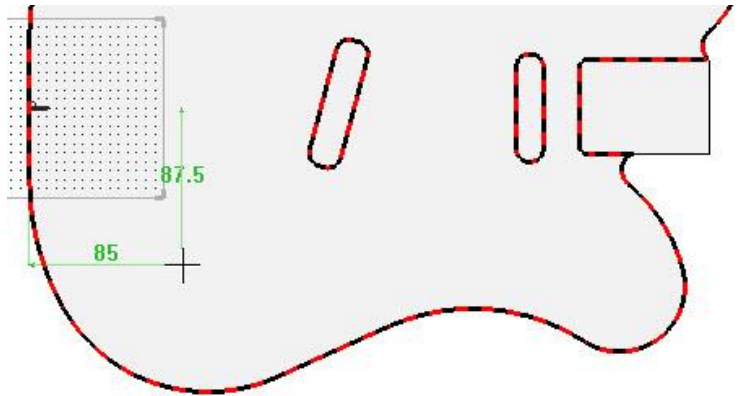
- In **Drilling models**, select **Hole** and adjust the following parameters:

    - **Hole**: **Through one**
    - **Diameter**: *15mm*

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

- Select **Linear propagation** in the **X+ direction**, and then enter **Distance per instance** = *50mm* and **Total number** = *2.*

Distance per instance ⚡ = 50   Total number: 2

- Then make a third drilling:

    - *185mm* in the X+ direction and *87mm* in the Y- direction
    - **Hole**
    - **Through**, **Diameter**: *7mm*
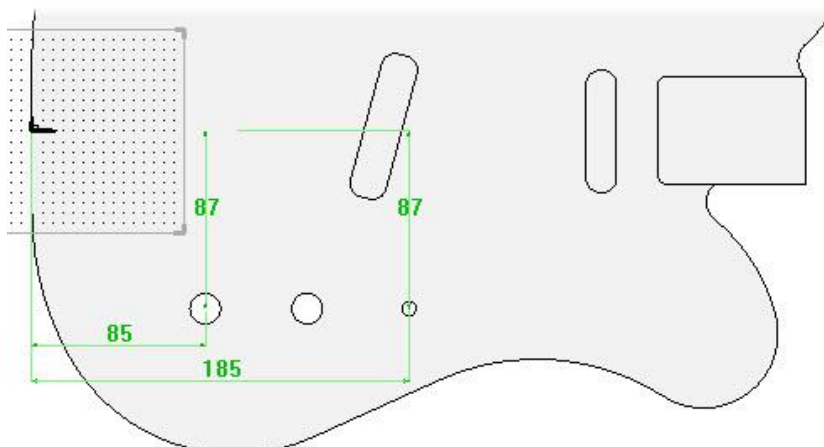
Create the moulding

- 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**.

Sweep= PLANAR FACE ▼ Reference face:

- Set **Join edges = YES** and **Follow tangent edges = YES**
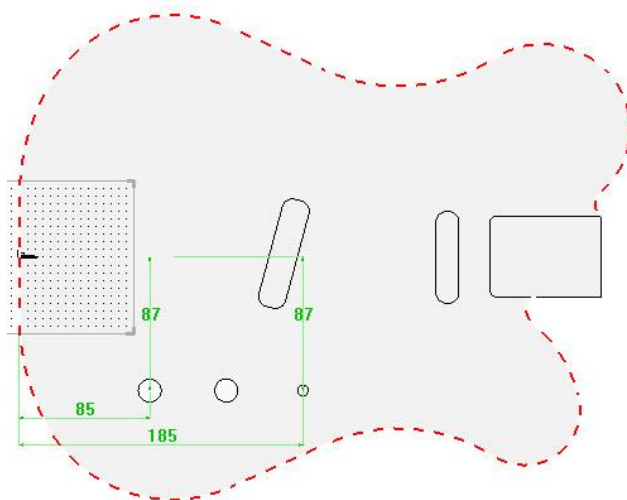
- Select the guitar contour's edge as the **reference edge or curve for tool path**. Validate the selection with **Stop**. STOP

- Validate the default direction of arrows with **OK**. OK

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

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

- Validate with **OK** to create the moulding.
- Use **Copy operation** to create the same moulding on the base of the guitar.
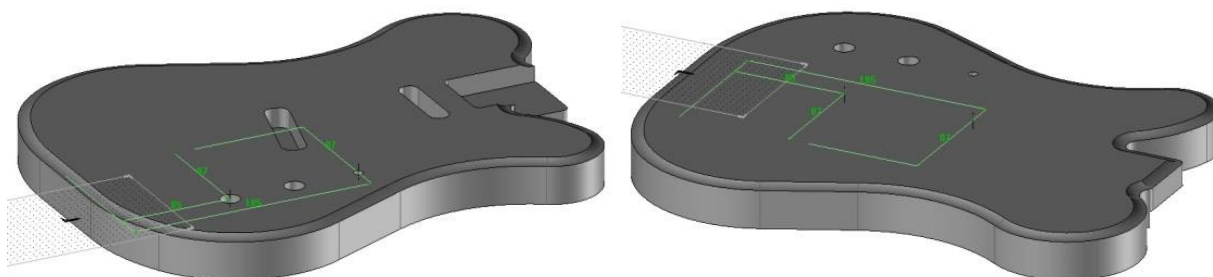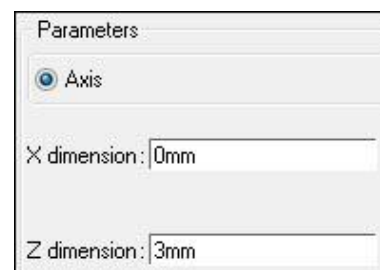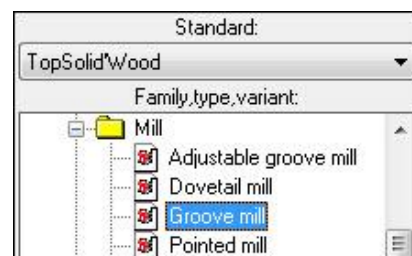
Create the groove

- Make **layer 2** current.

`0` `1` `2`

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

AXES (Z)= YES ⚡
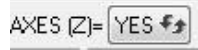
- Apply an **alignment constraint** to this line with the **X** axis.

- Apply two **coincidence constraints**:

    - of the left point of the line on the left pocket;
    - of the right point of the line on the middle pocket.
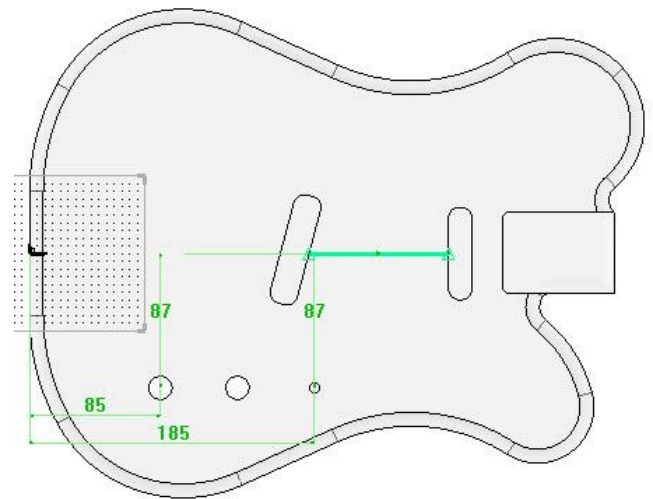
- **End the sketch**.

- Create a **groove** on the top face with the drawn line as the **tool path** and the guitar body as the **shape to profile**.

- From the **Parameters** window of the groove, adjust:

    - **Tool type**: **Routers**
    - **Standard**: **TopSolid'Wood**
    - **Mill** > **Simple mill**
    - **Parameters**: **Centered**
    - **Gap distance**: *0mm*
    - **Groove width**: *5mm*
    - **Groove depth**: *5mm*

Parameters

○ High arm   ⦿ Centred

Gap distance : 0mm

Groove width : 5mm

Groove depth : 5mm
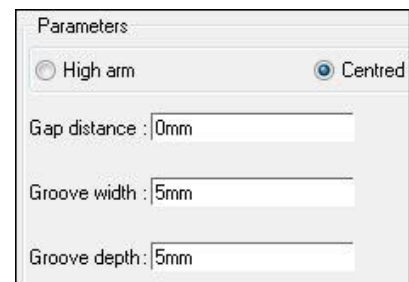
- Validate the parameters with **OK**.
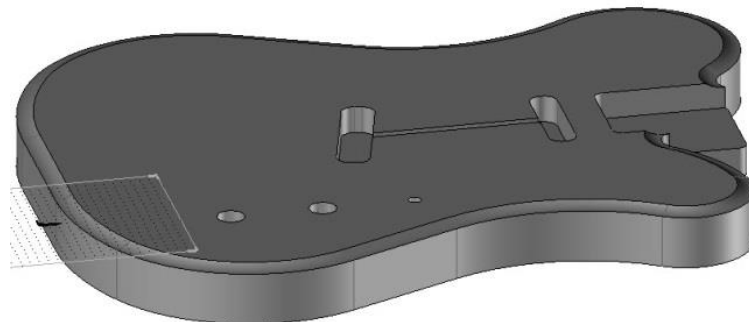
Part definition

- Use the **Wood** > **Define** > **Define part** function to define the guitar body:

    - **Designation**: *Guitar body*
    - **Material**: **TopSolid'Wood** > **Hardwoods** > **Ash european**.

- From the **Cutting-up** tab, add *30mm* **over dimensions** in **length** and **width** in **Additional** mode.

Sizes	Values	Modes	Over dimensions
Length	398.9mm	additional	30mm
Width	316.5mm	additional	30mm
Thickness	35.0mm	additional	0mm

Overvaluations
☐ Machining

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

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

☑ Make stock

- Start the **Define part** function.
- Select the guitar as the **part to define**.
- Select **Characteristics**. CHARACTERISTICS
- From the **Stock** tab, uncheck **Make stock**.
- Save the document.

Introduction to components

When creating 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.

Creation of a library

- Create a **new Design document**.
- Open the options using **Tools** > **Options**.
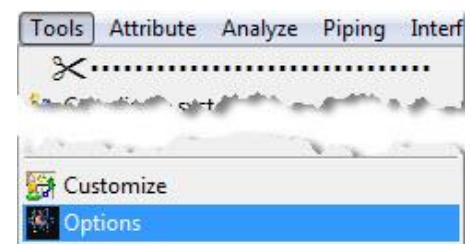
- From the **Component** > **User library** category, add a user library using the **Add** button.
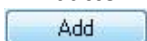
- Enter **Name**: *Training library.*
- Select **Path**: *C:\Projet* and create a new folder named *Training library.*

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

- Click on **OK** to validate 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: Creation of a shelf

The goal of this exercise is to create 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 set
- – Assembling the parts
- – Saving in the library

Designing the construction volume

Create the parameters

- Create a **new Design document** . From 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*

| Unit type= LENGTH | ▼ | TABULATED VALUES | Value: 1200 | > | OK | Name: l | Designation: Shelf length |

Create the construction volume

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

| X length= l=1200mm | Y length= d=400mm | Z length= h=300mm |

- 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.
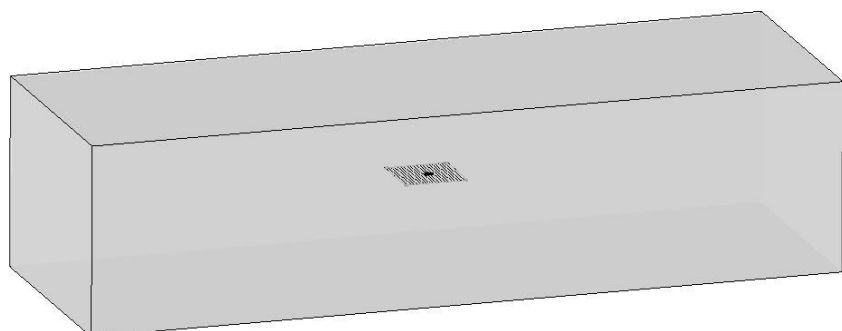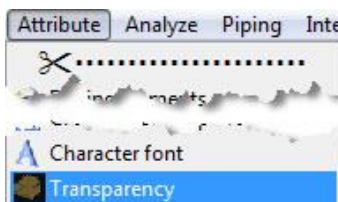
Creation of parts

Bottom part

- Make **layer 1** current.

- From the **Wood** context , start the **Constrained block** function.

- Define the *t* parameter as the **thickness**, and 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 back 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 to place the part inside the construction volume.
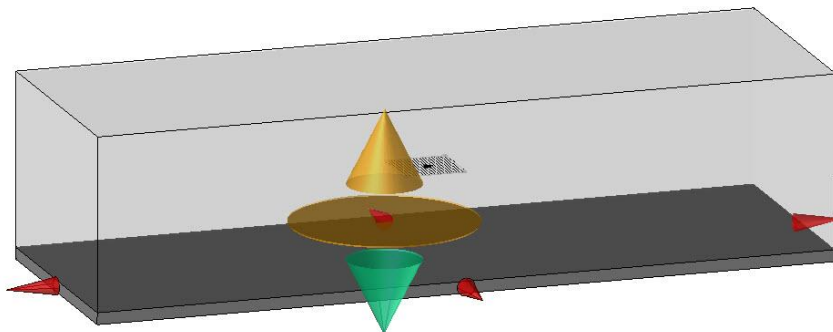
Top part

- Create a new 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*, and then select the right-hand face of the construction volume

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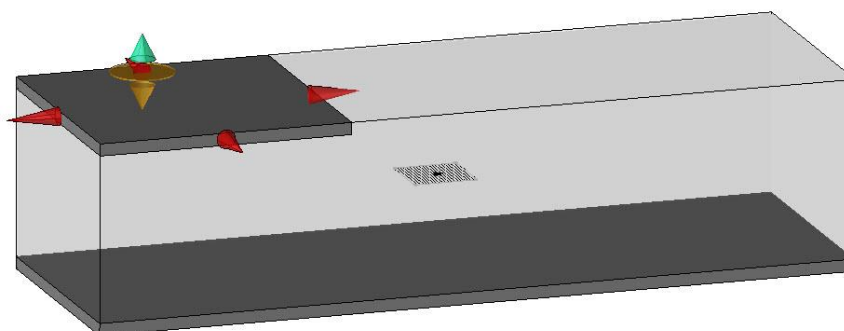
Side part

- Turn off **layer 0**.
- Using the **constrained block**, create the following part:

    - The **constrained block** has a **thickness** = *t*.
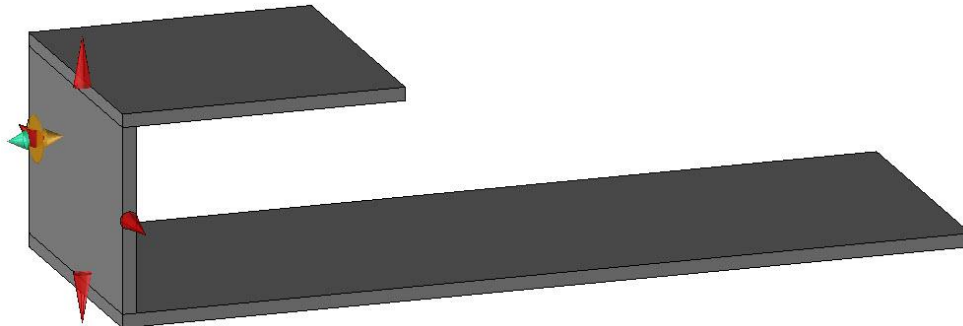
Definition of parts and set

Define the parts

- Define the parts using **Wood** > **Define** > **Define part**:

    - Bottom part designation: *Bottom shelf*
    - Top part designation: *Top shelf*
    - Side part designation: *Shelf side*

- For the three parts, select the material **TopSolid'Wood** > **Hardwoods** > **Birch apple**.
- 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 part's cutting-up axes.
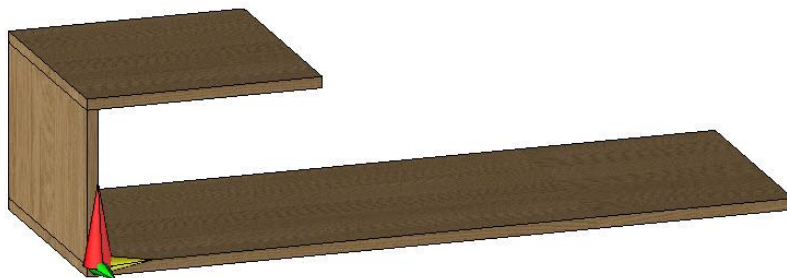
Note: In **TopSolid'Wood**, the length of a part always matches its grain orientation.

Define the set

Note: Once at least two parts have been defined, you are asked whether you want to define the assembly automatically in order to structure data.

- Define the set by validating the dialog with **Yes**.

    - **Designation**: *Wall shelf*
    - **Assembly nature**: **Sub-assembly**
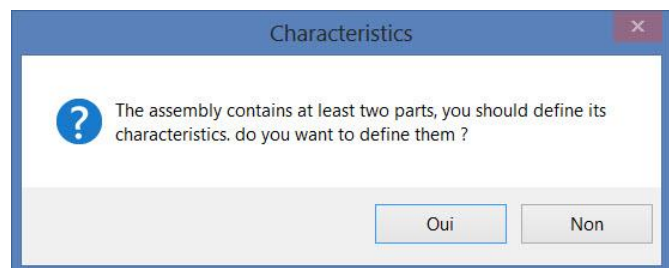
Assembling the shelf

- From the **Wood** context, start the **Dowel assembly** function.
- Select **Wood dowel** and select the **D12 L35 code**.

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

> FILTER ⨏ Propagation = YES ⨏ Support face:

- Select:

  - the contact face between the side et the top as the **support face**. Use the rotating selection to select this face.
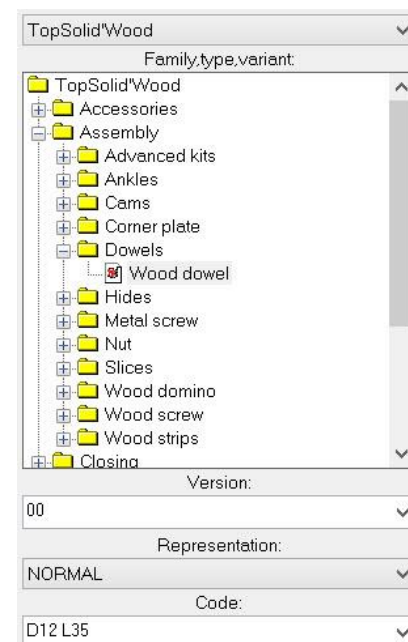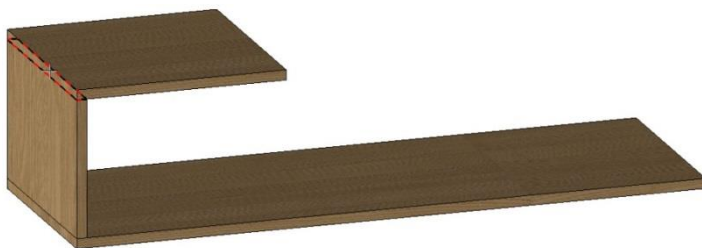
- the front edge of the top as the **start face or edge**.
  - **Centred automatically**. CENTRED AUTOMATICALLY

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**.

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

> Distribution mode
> ○ Step   ○ Step centered   ○ Distance   ◉ Advanced

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 adjust the following parameters:

  - **Step**: *96mm*
  - **Minimum distance to start**: *50mm*
  - **Minimum distance to terminate**: *50mm*

- Validate with **OK** to position the dowels.
- Select **Copy propagation**, and then repeat the operation to assemble the side with the bottom shelf. COPY PROPAGATION

> ☐ Unitary step
> Predefined values   >   96mm ▾
> Step (p)   :   96mm
> Minimum distance to start (d0min): 50mm
> Minimum distance to terminate (d1min): 50mm
> Element number: 3
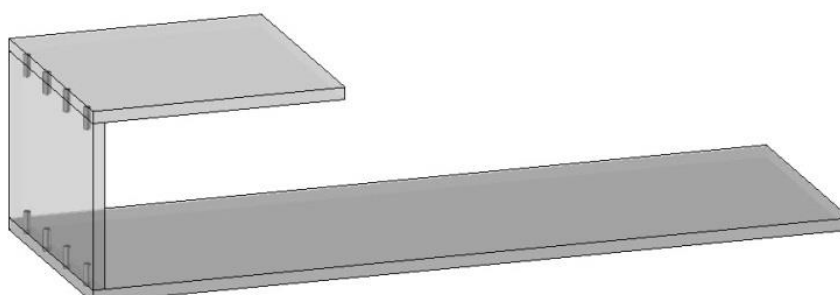
Saving work in library

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.

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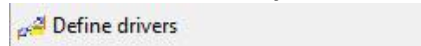
- Enter the **l** parameter as the **driving element to insert**.

Driving element to insert: |

DRIVERS SET : (4)
l : Shelf length = 1200mm
d : Shelf depth = 400mm
h : Shelf height = 300mm
t : Panel thickness = 19mm
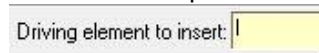
- Press **Enter** to validate.
- Press **Enter** again to validate the parameter designation.
- Define the parameters **d**, **h** and **t** in the same way.
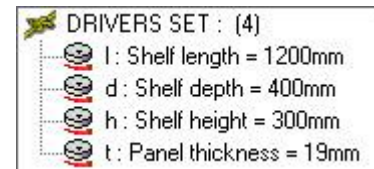
Save the component in the library

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

Edit/save template

- Select **Save standard template**.
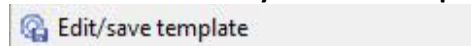
SAVE STANDARD TEMPLATE

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

- From the **Standard** drop-down list, select the library you created: **Training Library**.
- Enter:

    - **New family**: *Furniture*
    - **New type**: *Shelves*
    - **New variant**: *Wall shelf*

Note: The **families**, **types** and **variants** correspond to the different levels of the component library.

Family,type,variant:
Training library
Furniture
Shelves
Wall shelf
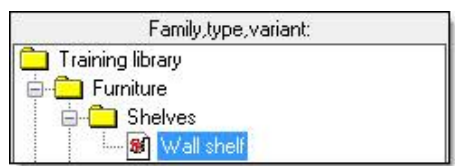
- Enter **New version**: *00*.

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

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

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

Note: The **Purge operations geometry** option improves the calculation performance of components.
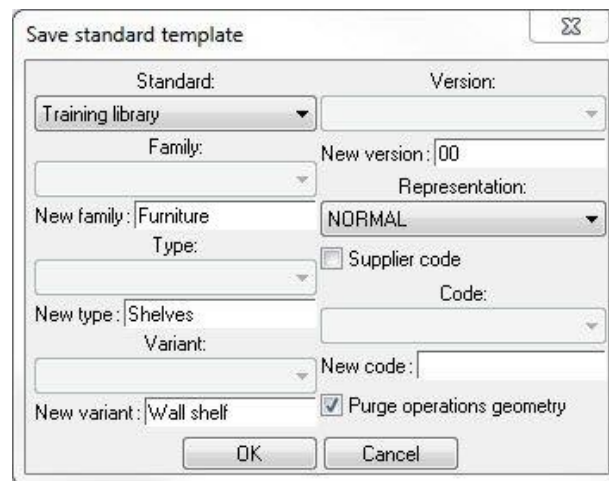
Note: Once the component has been saved in the library, the text $STD=**LibraryName** is displayed before the file name in the software title bar.

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

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

Supplement: Creation of 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's edges to *50mm* as shown opposite.
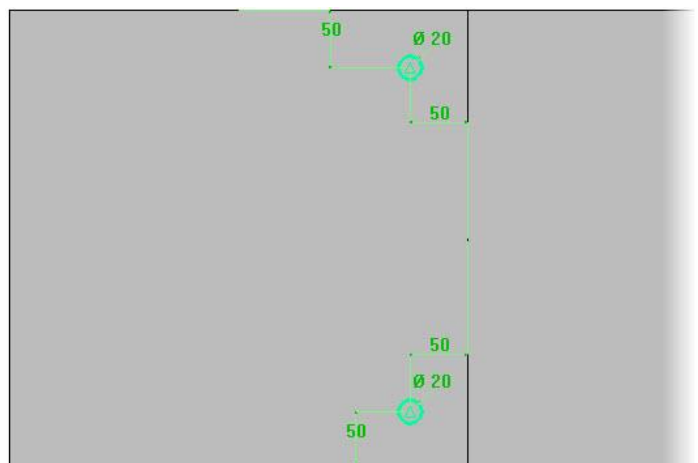
- **End the sketch**, and then extrude it in **Generatrix sketch = Global** and **Result = One shape per profile** modes.

Extruded shape on= CURVES Generatrix sketch= GLOBAL Result= ONE SHAPE PER PROFILE

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

Alignment= CENTERED Type= SOLID Generatrix= HIDDEN DIRECTION >> AUTO DIMENSION Height: (h-2*t)+20
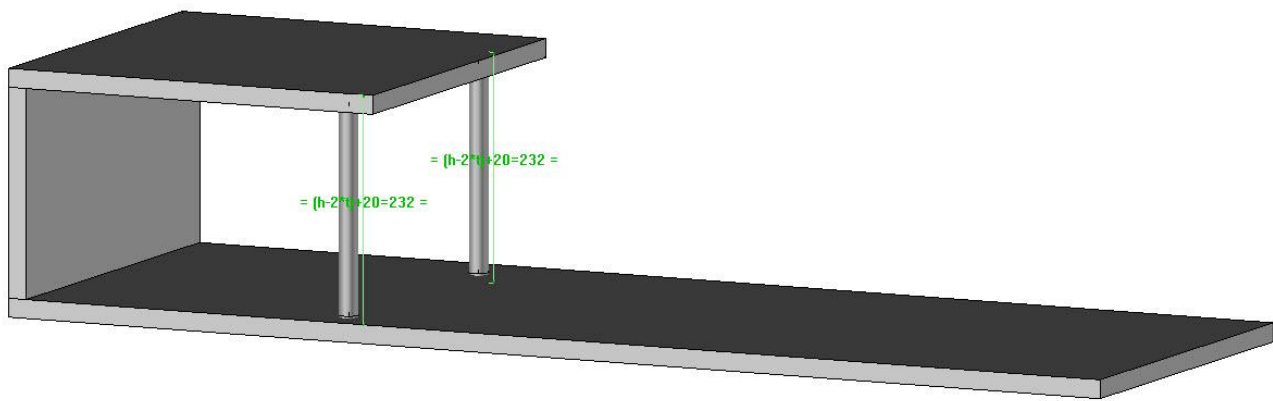
Define the supports

- Using the **Define part** function, define the two supports that were created.
- Select the cylinder of the support as the **length axis** and **Y+** as the **width axis**.
- Adjust the following parameters:

    - **Designation**: *Left support/Right support*
    - **Material**: **TopSolid'Wood** > **Metals** > **Aluminum**

Make the drillings

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

Mode= NON DYNAMIC ⚡ Face to drill:

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

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

- From the **Drilling models** window, select **Hole**.
- Check the **Save as default** option and validate with **OK**.

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

☑ Save as default
OK   Cancel
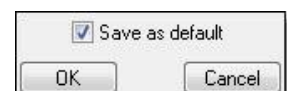
- Adjust the drilling's parameters:

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

Diameter   : 20mm
Depth   : 11mm
Bottom angle : 0°
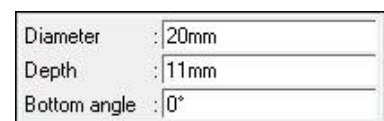
- Save the file using the disk icon.

Note: Once the file is saved in the library, just click the disk icon to save the modifications.

Exercise 2: Creation of 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
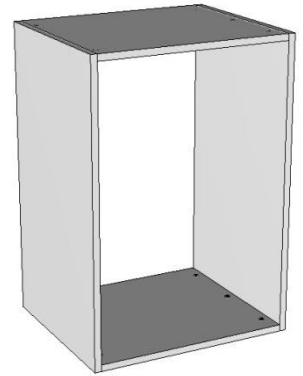
Creation of a driver block component

> **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, shelf…
>
> | **Cabinet component in driver block** | **Insertion of the cabinet in driver block** | **Assembly result** |
> |:---:|:---:|:---:|
> | | | |
> | **Shelf component in driver block** | **Component insertion** | **Result** |
> | | | |
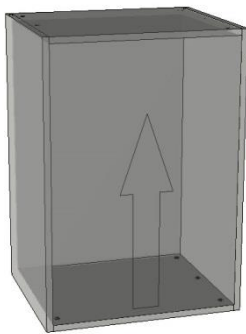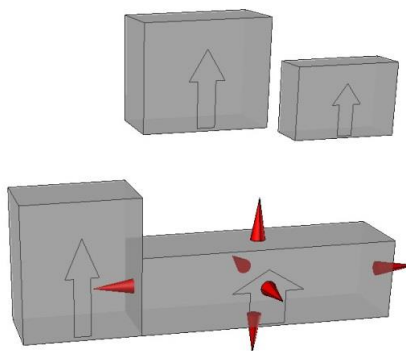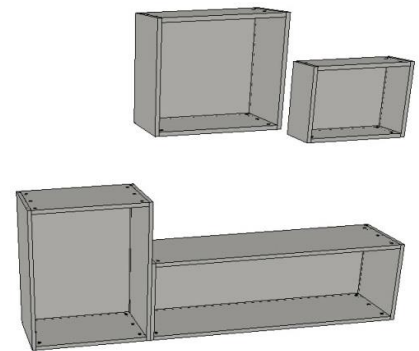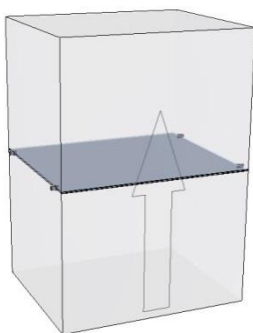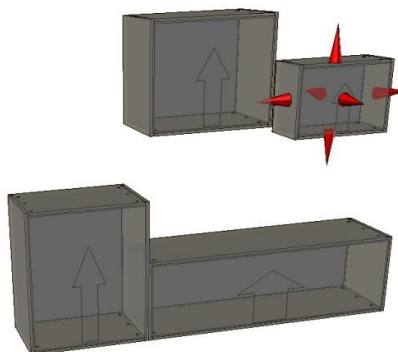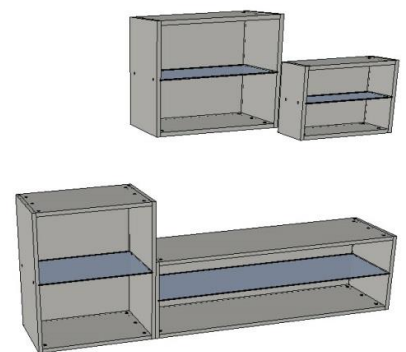
Create the driver block

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

DRIVER BLOCK

- Enter **Name of driving element**: *db.*

Note: The **name** and **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. Leave the default dimensions.
- Choose **Default housing mode = Inside a block**.
- Select **Current coordinate system** to position the **driver block**.   CURRENT COORDINATE SYSTEM
- Enter **Designation of the driving element**: *Cabinet block* and validate with **OK**.

OK   Designation of the driving element: Cabinet block

Note: The arrow on the driver block's face represents the front of the driver block and shows the top of the driver block.
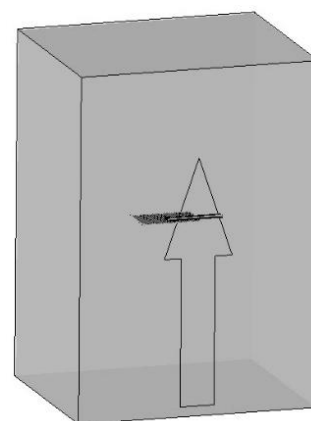
Note: A driver block component can be inserted in six different ways:

- **Housing**:
The component is included in a cabinet housing.
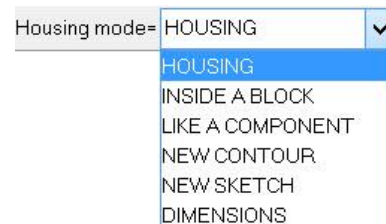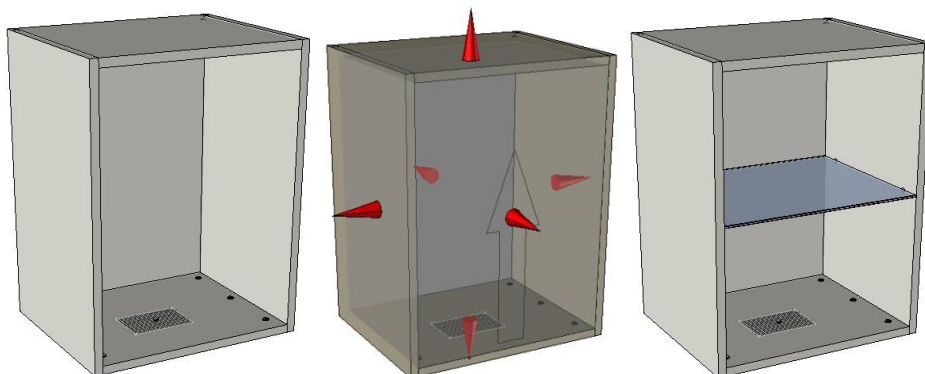
- **Inside a block**:
The component is included inside a block.
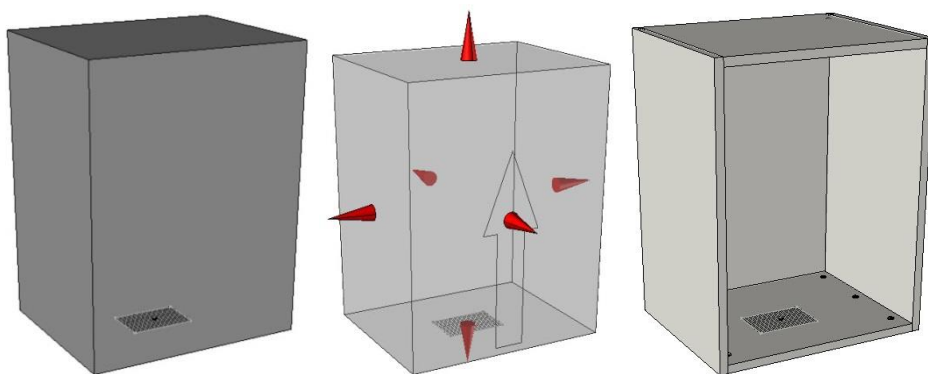
- **Like a component**:
The component is included from a previously inserted driver block component. The **Merge driver blocks = No** option allows you to modify the driver block of the second component. If the option is set to **Yes**, the driver block of the second component will strictly correspond to the first one and cannot be modified independently.

- **New contour / New sketch**:
This mode allows you to draw a new contour or a new sketch, extrude it, and then include the component inside using the **Inside a block** mode.

- **Dimensions**: The **Dimensions** mode allows you to insert the component in a more manual way by specifying the three dimensions on X, Y and Z, and then positioning it with constraints.
If a catalog has been defined with the component, it is then possible to use it with the **Dimensions** mode.
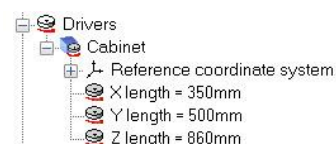
$code	db.x	db.y	db.z
20x60	200	600	600
40x37	400	600	370
40x60	400	600	600
60x37	600	600	370
60x60	600	600	600
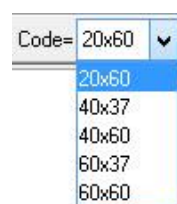
Create the cabinet parts

- 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 four 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.
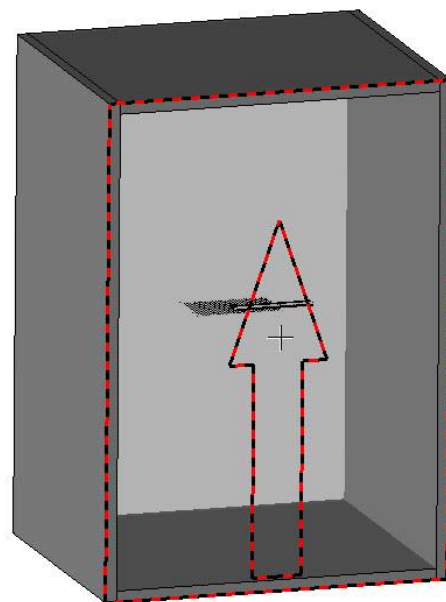
Note: You can use the **Automatic** option to create the **constrained blocks** more quickly. [AUTOMATIC]

- Define the parts for the cabinet:

    - **Designation**: *Top / Base / Left side / Right side*
    - **Material**: **TopSolid'Wood** > **Panels** > **Colors** > **White**

- Define the cabinet set:

    - **Designation**: *Free-running sided cabinet*
    - **Assembly nature**: **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**.
- From the **Save standard template** window, adjust the following parameters:

    - **Standard**: **Training library**
    - **New family**: *Cabinets*
    - **New type**: *Cabinets*
    - **New variant**: *Free-running sides*
    - Uncheck **Supplier code**

- Validate the window with **OK**.
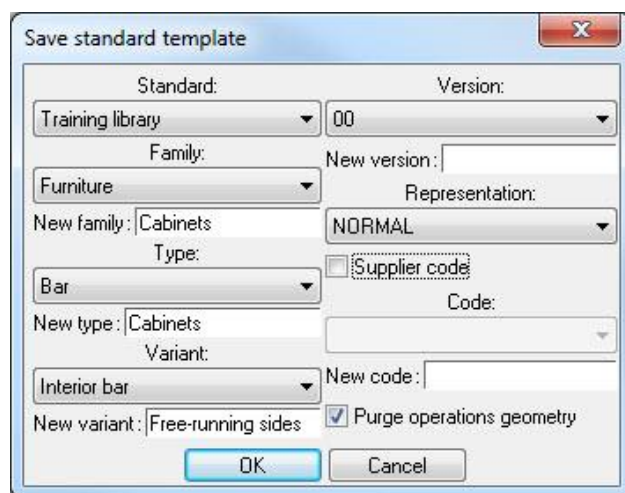
Assembling the cabinet

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 do not have to be filled for each assembly.
>
> These predefined propagations also define different propagation rules according to the length to assemble.
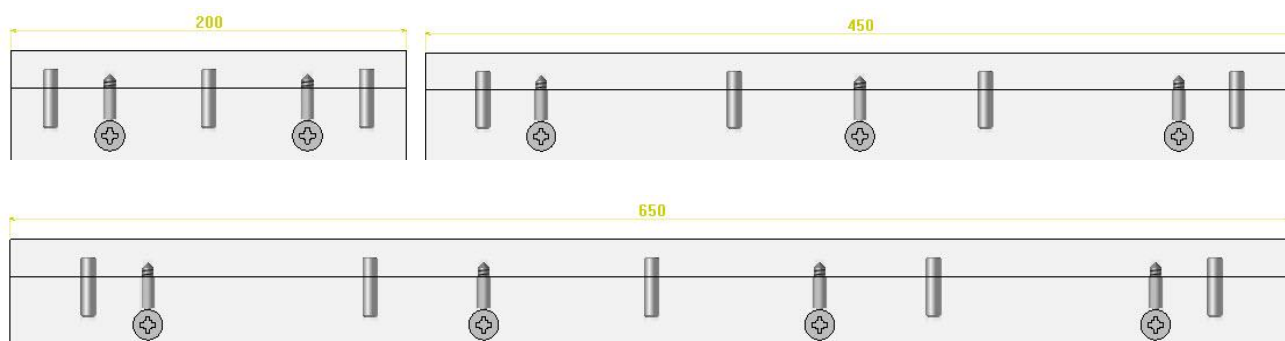
- In a new **Design document**, open **Tools** > **Options** > **TopSolid'Wood configuration** > **Propagations configuration**.

- Select **Add propagation** to create a new propagation. ▭ Add propagation

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

  Propagations
  Dowels
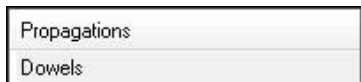
- From the list of propagations, modify the **Right bound** value on the first line to *80mm*.

- Use the **Add range** button to create the different ranges below. ▭ Add range

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

- Then adjust the following propagations according to the ranges.

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 the **Add propagation** button.
- Rename this propagation to *Cams.*
- Recreate the same ranges as for the *Dowels* propagation.
- Adjust the 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	-

- Validate the window with **OK**.

Assemble the cabinet

- In the free-running sided cabinet template, start the **Dowel assembly** function.
- Select **Wood dowel** and select the **D8 L30 code**. Validate with **OK**.

Code:

D8 L30

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

FILTER Propagation = YES Support face:
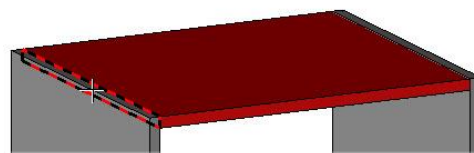
Note: When the support face of the parts to assemble is rectangular, the **Automatic** option can be used to identify the start, terminate and centered faces automatically.

- Select **Automatic**. AUTOMATIC
- The **Distribution definition** window opens. To use here the previously configured predefined propagations, select the **Dowels** propagation from the **Propagation name** drop-down list.

Predefined propagations
Propagation name > Dowels

- Validate the window with **OK** to assemble the parts.
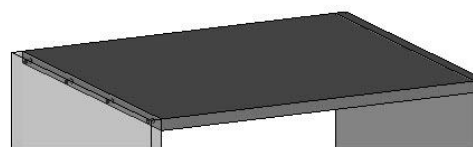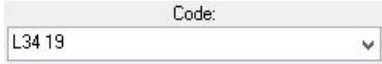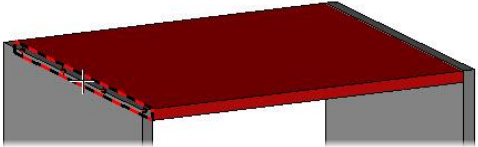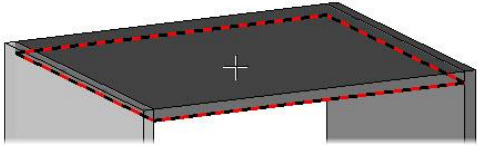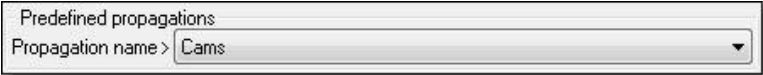
- Use the **Copy propagation** option to assemble other parts with the same propagation.
  COPY PROPAGATION

- Repeat the previous steps to assemble the four parts with dowels
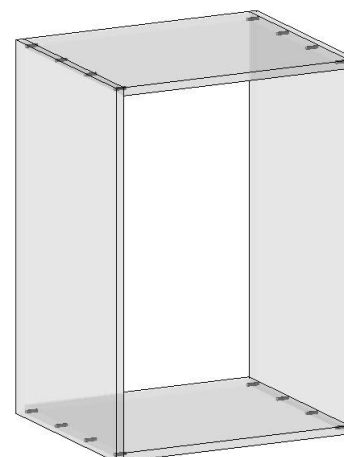
- Start the **Cam** function.
- Select **Simple cam** and the **L34 19 code**.

Code:
L34 19

- Select the **Filter** mode and set **Propagation = YES**.
- Select the contact face between the **top** and the **left 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.
- From the **Distribution definition** window, choose the **Cams** propagation as the **predefined propagation**, and then validate with **OK**.

Predefined propagations
Propagation name > Cams

- Then use the **Copy propagation** option to position the other three assemblies.
- Save the cabinet.
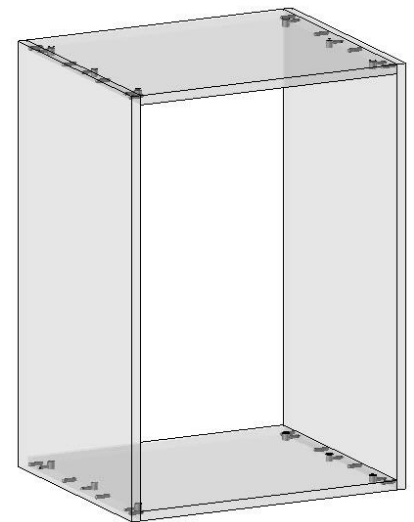
Creation of panels

┌─ **Note** ───

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

- A **panel entity** which includes the part, edges and laminates.
- A **support** that 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**

⊟ Panel entity (SIMPLIFIED)
    Support (SIMPLIFIED)
    flat edge (SIMPLIFIED)
    flat edge (SIMPLIFIED)
    flat edge (SIMPLIFIED)
    flat edge (SIMPLIFIED)
  ⊞ laminate shape (SIMPLIFIED)
  ⊞ laminate shape (SIMPLIFIED)
  ⊞ Model
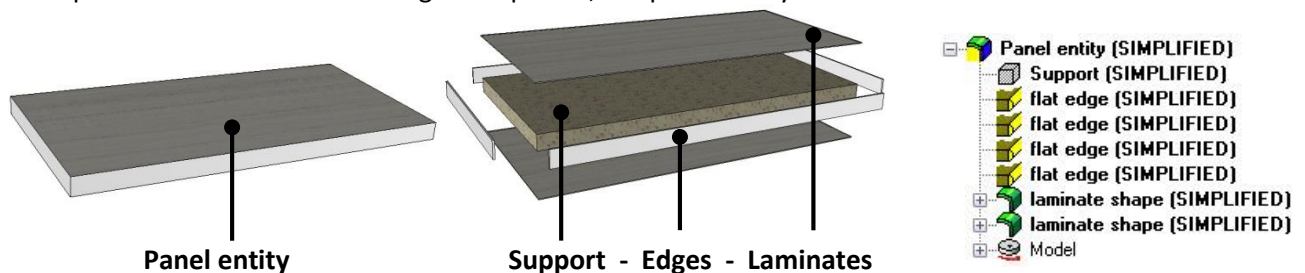
- From the **Wood** context, start the **Panel** function.

- Open the panel's **advanced options**.

- Adjust the following parameters:

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

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

    - **Simplified representation**.

The simplified representation is used to only view the edge textures and 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 design.

The **detailed representation** mode is used for example in a draft to view the real edges.
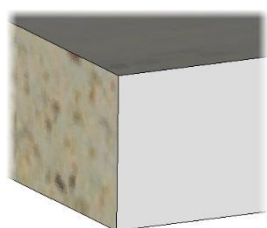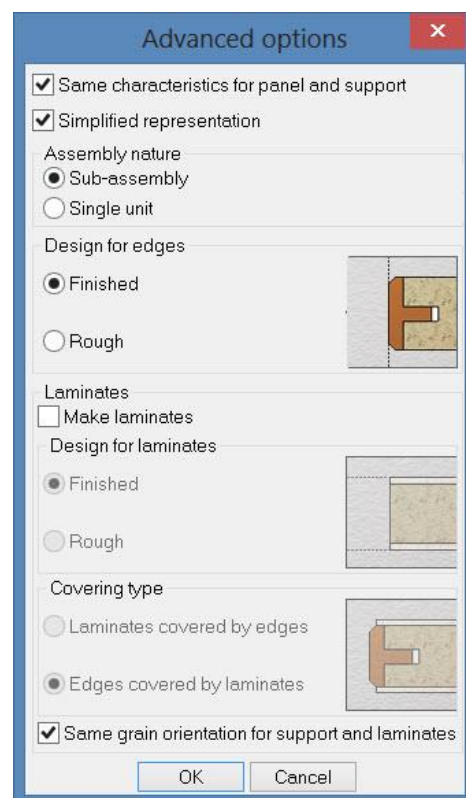
panel process (SIMPLIFIED)
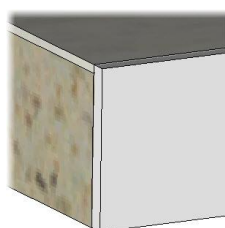
panel process (DETAILED)

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

The **Sub-assembly** mode for panels displays the edges and laminates on additional lines in a bill of material.

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

- Validate the parameters with **OK**.

- Select the main face of the **left side** as the **reference face**.

The panel configuration window opens.
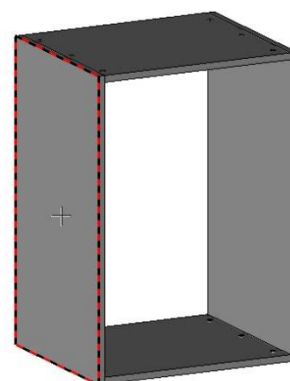
- Double-click in the **Edge type - code** box on the first line to select an edge:

    - **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, and 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 in order not to place an edge on the panel's rear edge.

Its arrow then turns transparent.

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

|	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
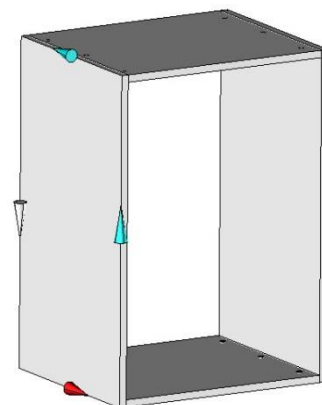
- Validate the window with **OK** to create the panel.
- Repeat these operations to apply edges to the other three parts:

    - On the right-hand side, the rear edge is not placed.
    - On the top and the base, only the front edge is placed.

- Save this file.
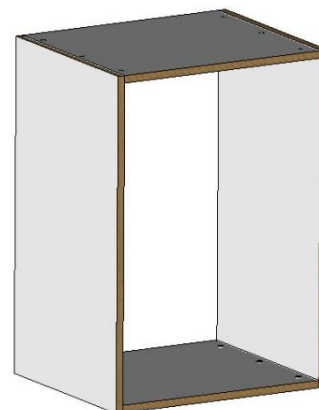
Supplement: Predefined values
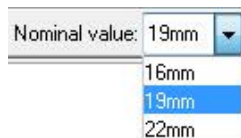
- Start the **Parameter** > **Modify parameter** function.
- Enter **Parameter to modify**: *t* and press **Enter** to validate.
- Open **Advanced parameters** using the ⟫ button.
- Select **Predefined values=0**. PREDEFINED VALUES=0

The **Predefined values** window opens.

- Enter the three **values 16**, **19** and **22**.
- Check **Only those values**.
- Validate the window with **OK**.
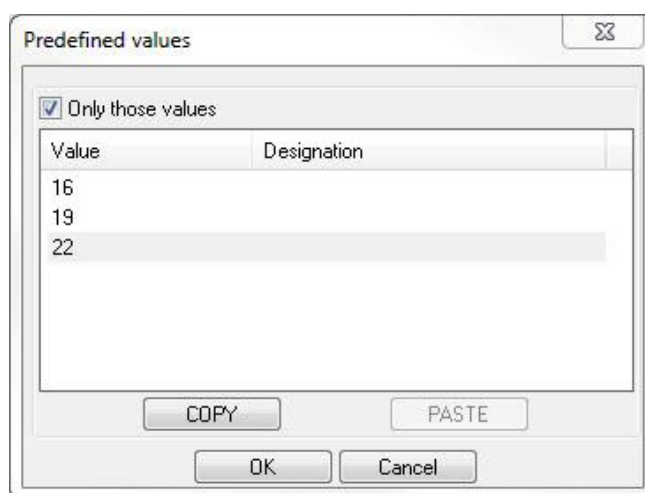
Note: The **Only those values** option allows you to enter for this parameter only the predefined values specified here.
The **Designation** field associates a designation to the predefined values.

- Save the file.

Exercise 3: Creation of a grooved back component

In a library component, it is possible to define machinings that will be performed when the component is inserted.

For example, this is the case for cams and dowels which perform drillings after their insertion.

This process will be used in the grooved back cabinet component which will automatically perform the grooves once inserted.

Concepts addressed:

- **-**   Negative shifts on constrained blocks
- -   Martyr parts
- -   Component processes
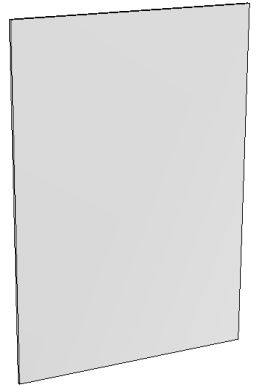
Creation of the component

Create the driver block

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

`DRIVER BLOCK`

- - **Name of driving element**: *db*
- - Leave the default dimensions
- - **Default housing mode = Housing**
- - Select **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`
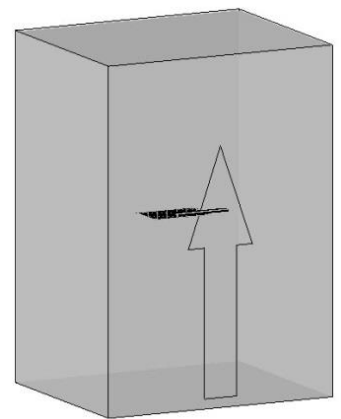
Create the back

- Make **layer 1** current.

- Use the **Constrained block** function to create the back:

- - **Thickness** = *8mm*
- - **Shift** = *-5mm* on the four selected faces
- - **Selected planes**: Four sides of the driver block
- - **Positioning shift**: *-10mm*
- - **Positioning plane**: Rear face of the driver block
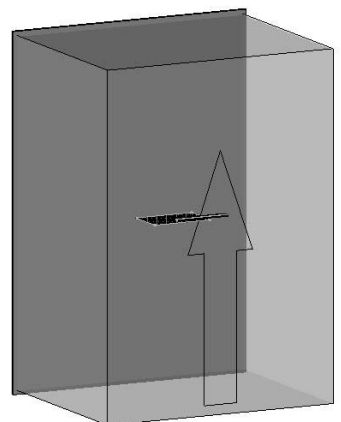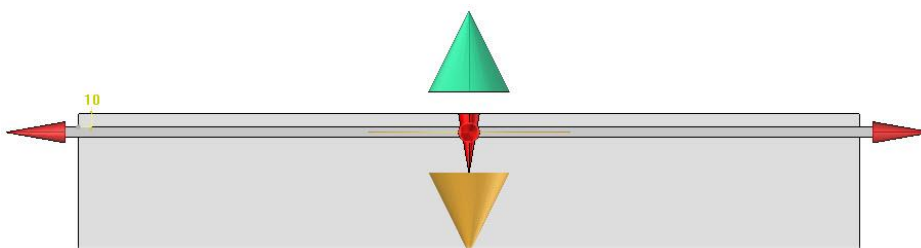
Define the part

- Define the back using the **Define part** function:

    - **Designation**: *Grooved back*
    - **Material**: **TopSolid'Wood** > **Panels** > **Colors** > **White**

Note: An assembly containing one part is automatically defined in **Single unit** mode. The part information is automatically included in the assembly in a bill of material.
All information must then be specified on the part.

- 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**: *Grooved back*

Creation of processes

Create the martyr parts

Note: The goal of creating the **Grooved back** component is that it can perform its grooves automatically when inserted in a cabinet. To do this, the grooves must be created 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 between martyr parts.

- Use the **Constrained block** function to automatically create the four martyr parts on the driver block's sides:

    - **Thickness** = *19mm*
    - **Selected planes**: Four sides of the driver block
    - **Positioning**: Outside the driver block
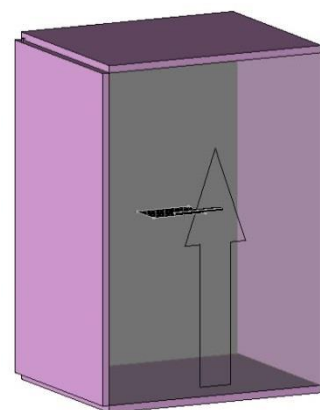
Note: The martyr parts of a component must not be panel entities.

Create the grooves

- Turn 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 groove offset shown by the red arrow must point towards the inside of the part. If needed, click the arrow to invert it.

- Validate the groove path with **Stop**.
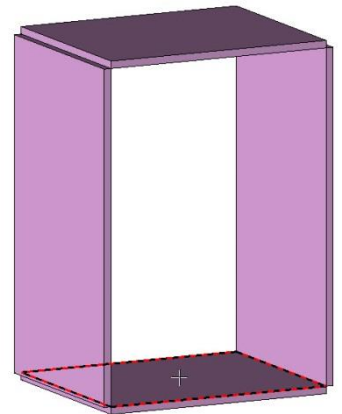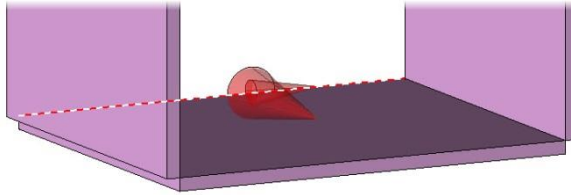
- Adjust the groove's parameters:

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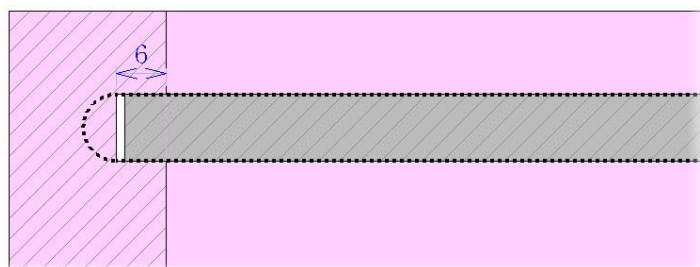
- Validate the window with **OK** to create the groove.
- Use **Copy operation** to perform the same groove on the other three martyr parts.

Define 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.
A tool must be defined for each machined martyr part. Here, four tools must be created.

- 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.

- Enter **Name of tool element**: *tool1* and validate by pressing **Enter**.

Operation type= LOCAL OPERATION ON SHAPES ▼ Name of tool element: tool1

- Change the designation to *Bottom groove* and validate by pressing **Enter**.

OK Designation of the tool element: Bottom groove

- In **Local operation to insert in tool**, select the groove that was created on the bottom martyr part.

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.

- Validate the default **local operation name**.

OK Name of local operation: shaping_operation_3

Note: The operations selected during the creation of a tool are shown in red.

- Validate the tool with **Stop**. STOP
- Validate the **Operation as tool definition** window without changing anything.

- Repeat the procedure to create the other three tools:

  - **Name of tool element**: *tool2* - **Designation of the tool element**: *Right groove*
  - **Name of tool element**: *tool3* - **Designation of the tool element**: *Top groove*
  - **Name of tool element**: *tool4* - **Designation of the tool element**: *Bottom groove*

- Make **layer 1** current and turn off **layers 0** and **2**.
- **Save** and **close** the document.
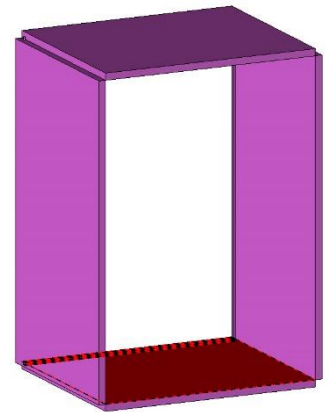
Exercise 4: Use of components in an interior design

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

Concepts addressed:

- Importing a DWG file
- Use of layers
- Creation of the room to fit out in 3D from 2D
- Insertion of standard components
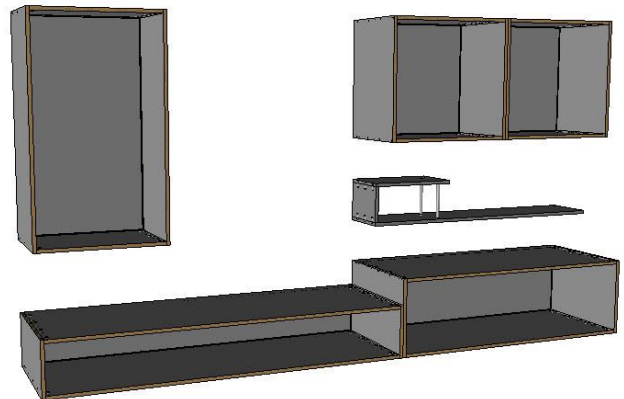
Importing and using a DWG file
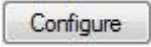
Import the file

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

- Start **File** > **Open** and click the *Missler.DWG* file.
- Still from the **Open File** window, set **Open as**: **TopSolid'Design - Design**.

Open as: TopSolid'Design - Design
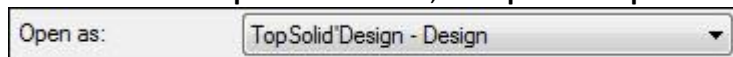
Note: It is possible to configure how to open a DMG file using the **Configure** button. ⟨Configure⟩

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

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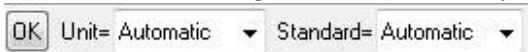
- **Unit**: Used to set the unit to use for this document (m, mm, km…).
  Select **Unit = Automatic** to automatically detect the unit of the document.
- **Standard**: Used to set the type of drawing standard to use for this file (Iso, Ansi…).
  Select **Standard = Automatic** to automatically detect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

- **Validate** the parameters and then the Import Options window.
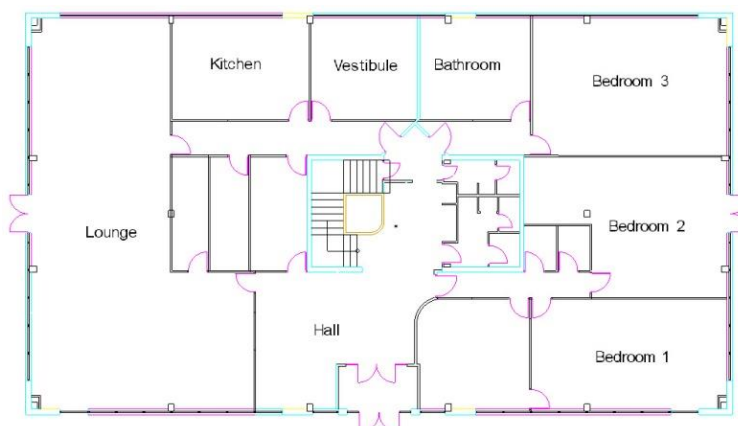
Note: When imported, a file is automatically converted to the **TopSolid** format (.top) in a new document.

- Save this document in a folder named *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.
In addition, 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**.
- Click the **Layers** tab.

Note: The **Layers** tab of the construction tree displays the different layers of the document.
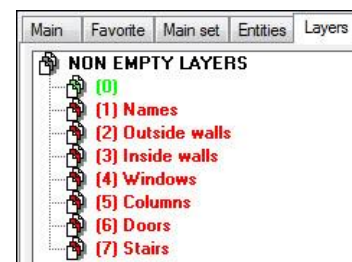
Right-clicking on a layer allows you to make it **current** or **active**, give it a **name** or **add elements**.
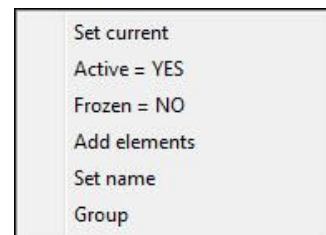
By default, only the **non-empty** layers are displayed.

- **Right-click** > **Definition** on the first **Non empty layers** line and adjust the following parameters:

    - **Display = All layers**
    - **Sort = Numerical** and validate with **OK**
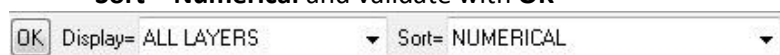
Create the walls

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

- From the **Shape** context, select **Extruded**.
- 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**.
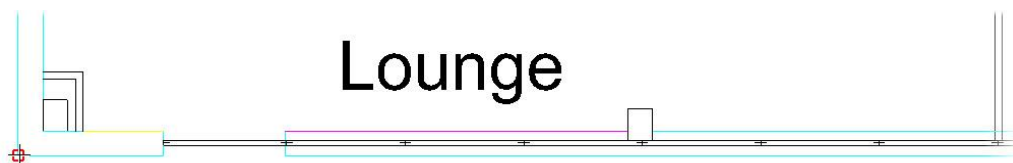
- Select the intersection point between the exterior wall (light blue) and the interior wall (black) as the **second diagonal point**.

- Extrude this wall in the **Z+** direction to a **height**: *2500mm*.
- Repeat this operation to extrude a second wall perpendicular to the first one to the double door (purple).
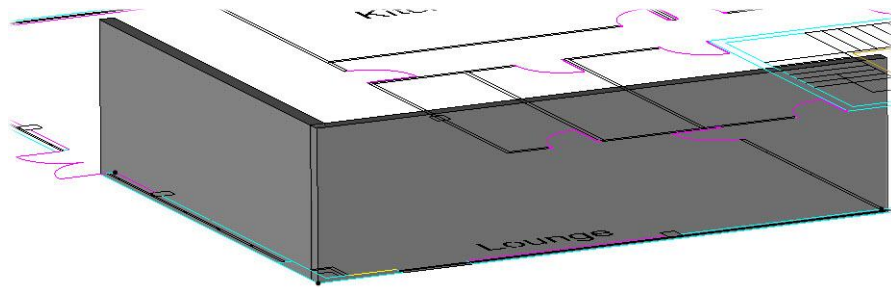
- Create several walls of the house in the same way.
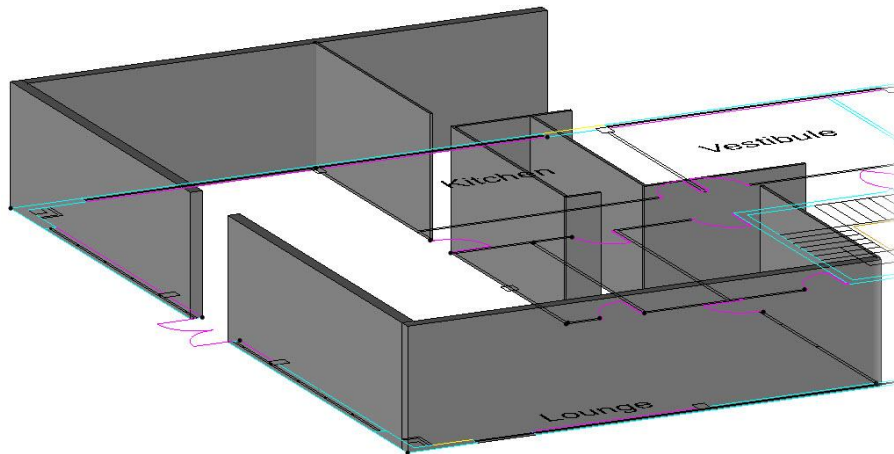
Create the columns

- Follow the same method to extrude the three columns of the lounge.
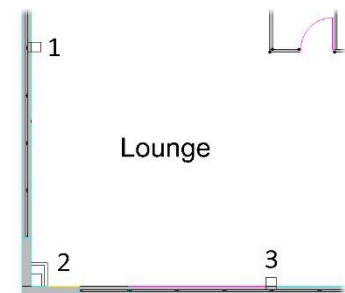
Create the floor

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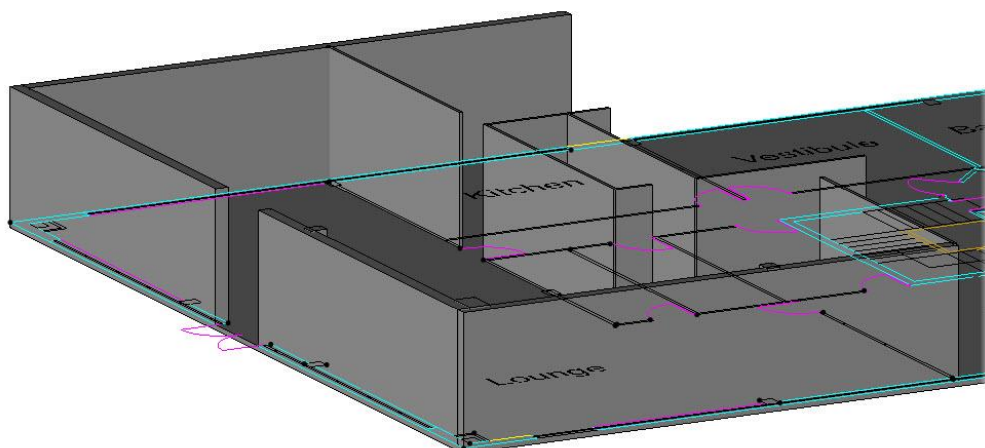
Use of standard components

Draw the construction volumes

- Make **layer 11** current and name it *Construction volumes*.
- Turn off **layers 0** to **8**.
- Create a **new sketch**.

- Use the **Contour** > **Rectangular** function to draw any rectangle.
RECTANGULAR

- Next click the **Auto dimension** button to automatically place the rectangle's dimensions.
AUTO DIMENSION

- The next dialog allows you to place symmetry constraints on dimensions. Click on **OK**.
- Repeat the operation to draw four other rectangles as shown below.
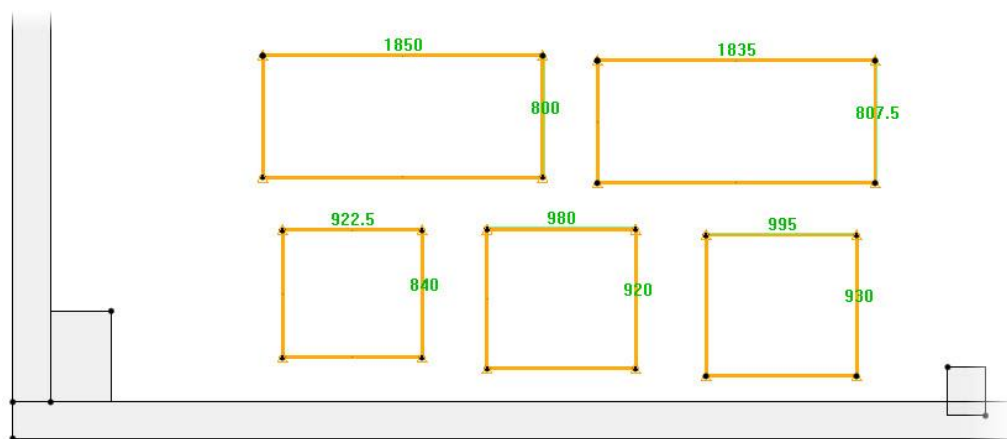
- Use **Modify parameter** to adjust the rectangle dimensions to the following values:

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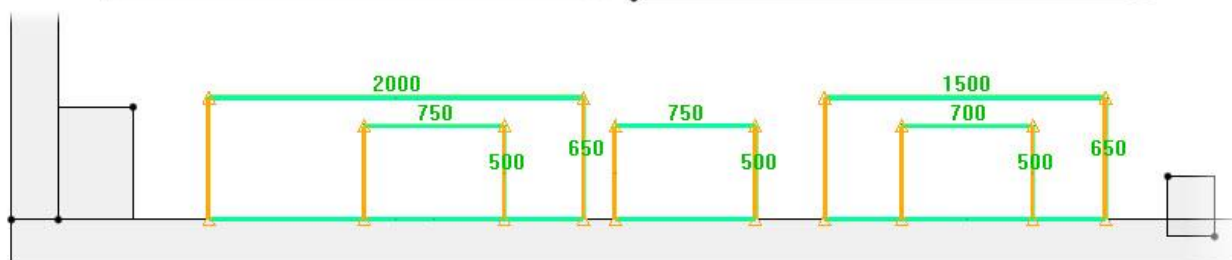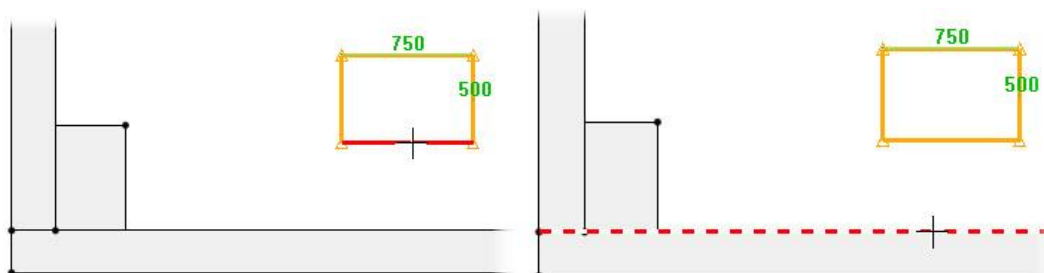
- Place a dimension between the rectangle of 2000mm and the right-hand column to a **value**: *1000mm.*
- Place **alignment constraints** between the rectangles to obtain the following result.
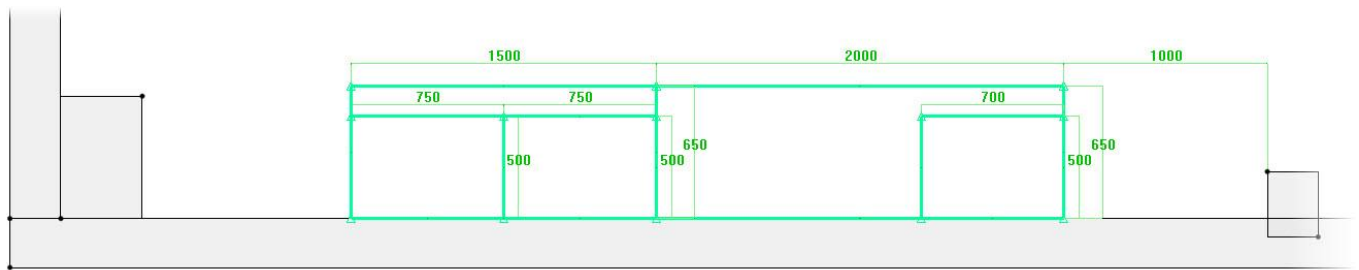
- **End the sketch**.

Extrude the construction volumes

- Change the **design color** to **cyan**.
- From the **Shape** context, select **Extruded**.
- Set **Generatrix sketch = Global** and **Result = One shape per profile**.

Note: The **Generatrix sketch = Global** mode extrudes the whole sketch in one go.
The **Result = One shape per profile** mode allows you to obtain a distinct shape per drawn contour.

- Next select the sketch.
- From the advanced parameters **>>**, set **Offset from starting curve** = *100mm*, and then validate with **OK**.

- Extrude the blocks in the **Z+** direction to a **height**: *450mm*.
- Using the **Attribute** > **Transparency** function, apply a transparency of **7** to the extruded blocks.
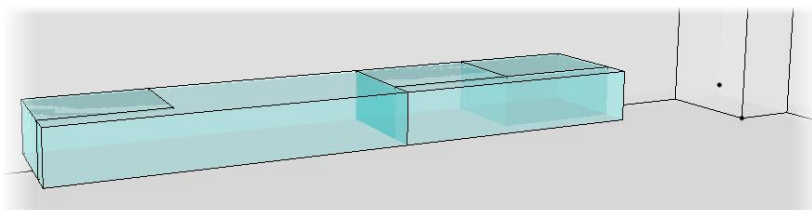
Modify the construction volumes

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

Note: The **Edit** function allows you 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**.
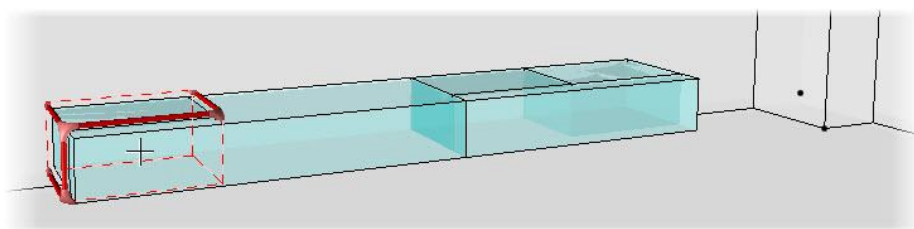
The extruded block is now open in the construction tree.

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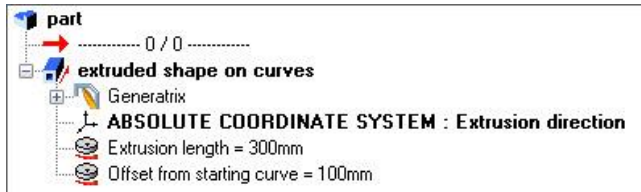
Note: All the construction volumes have been extruded in a single operation. This means that the **extrusion lengths** and the **offsets from starting curves** are merged between the blocks.
To modify a volume without changing the others, the parameter must be **replaced**.

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

- **Right-click** on **Offset from starting curve** and select **Replace**.
- Set **Replacement = Local** and **Replacement parameter** = *800*.
- Also replace the **extrusion length** by *1200*.
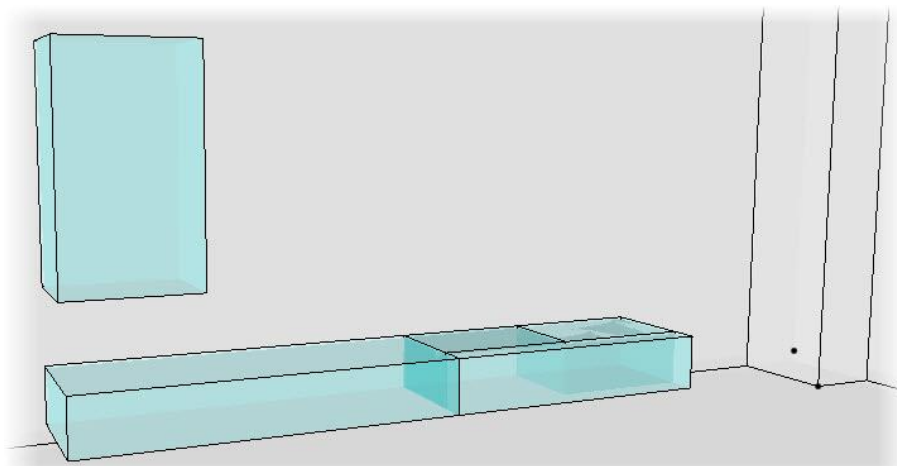
- **Edit** the two identical right-hand blocks:

- - **Replace** their **extrusion length** by *700mm.*
  - **Replace** their **offset from starting curve** by *1300mm.*
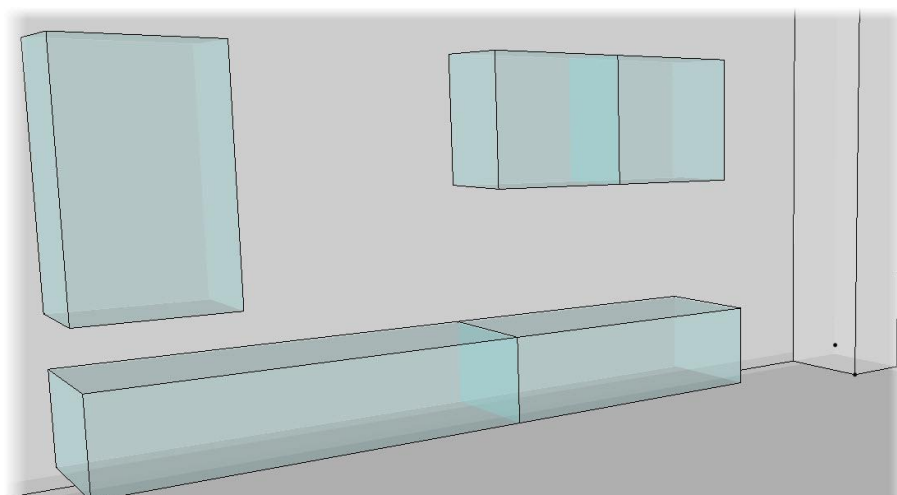
Include standard components

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

- To include standard library components, use the **Include standard** function from the **Assembly** context.
- From the **Standard component inclusion** window:

    - Select **Training library** in the drop-down list.
    - Select **Cabinets** > **Cabinets** > **Free-running sides**.
    - Validate with **OK**.
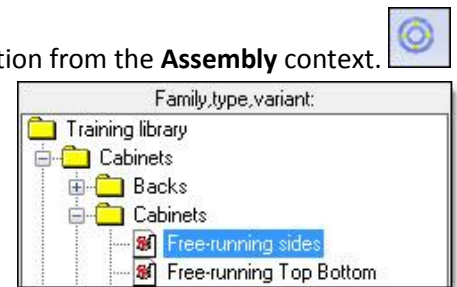
The **Cabinet** component is a **Driver block** component. When including this component, you will be asked to select the destination volume of the **Driver block**.

- 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 on the six faces of the block allow you, for each support plane, to:

    - modify the plane;
    - add an offset or a length to the plane;
    - define a passing point for the plane.

In addition, two curved arrows on the front face allow you to rotate the driver block around the Z axis.
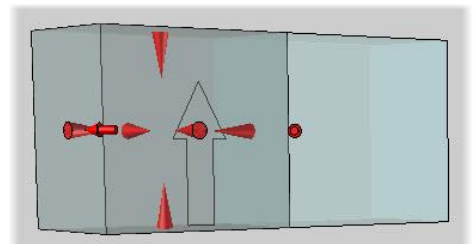
- Set **Multiple inclusion = YES** to include the same **Driver block** component several times.
- Select the front faces of the other four construction volumes, and then validate with **OK**.
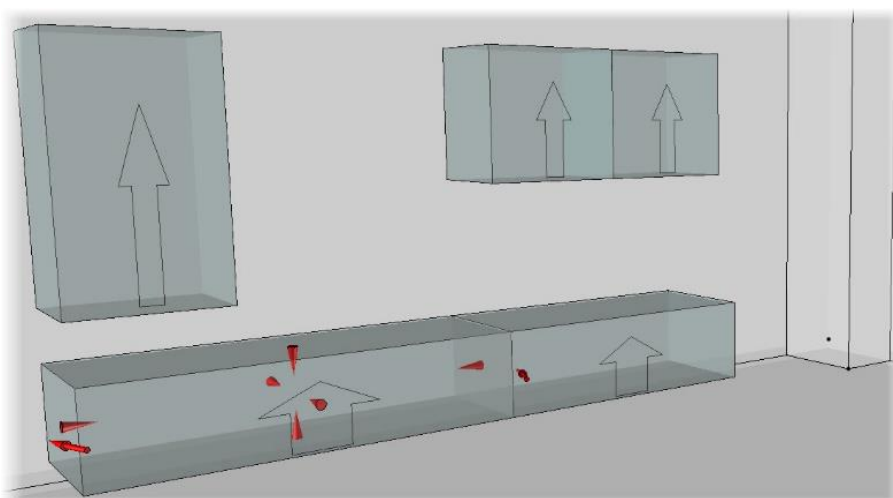
- Set **Thickness of panels** = *19mm* and validate with **OK**.
- Turn off **layer 11** to hide the construction volumes.
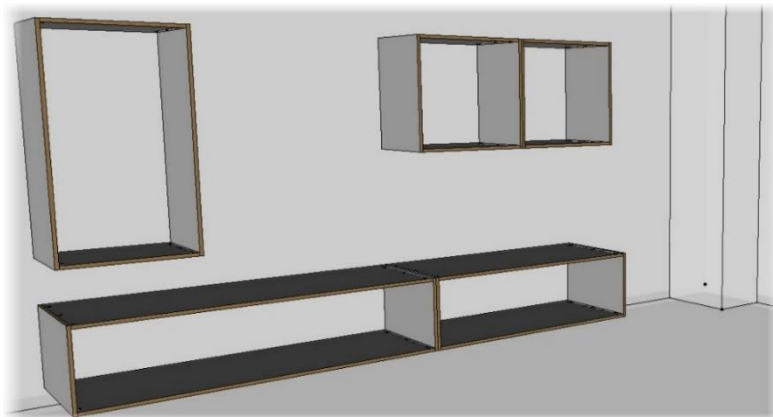
Modify a driver block component

- Start the **Modify element** function and select the bottom left cabinet.
- Select **Driver block = Cabinet block** in order to modify the driver block of the cabinet.

  DRIVER BLOCK=Cabinet block

- Select the up arrow of the component to modify the top support plane.
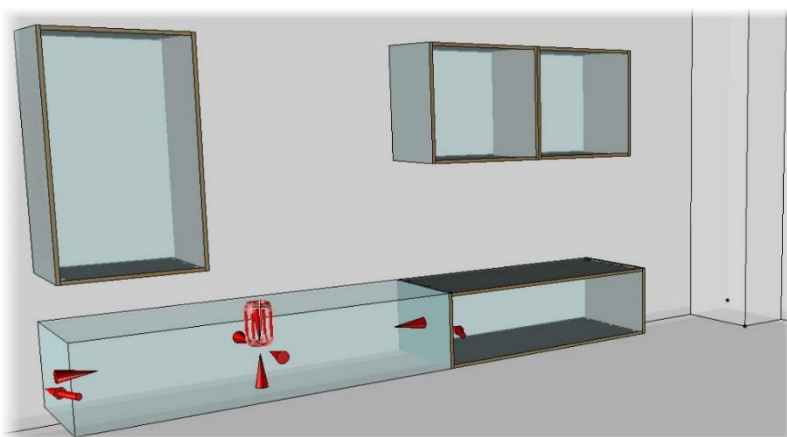
- Select the **Shift** option. SHIFT
- Enter **Shift** = *150mm* and validate with **OK**.

  OK | SUPPRESS SHIFT | Shift= 150mm | THROUGH POINT

- Validate the changes made to the driver block with **OK**.
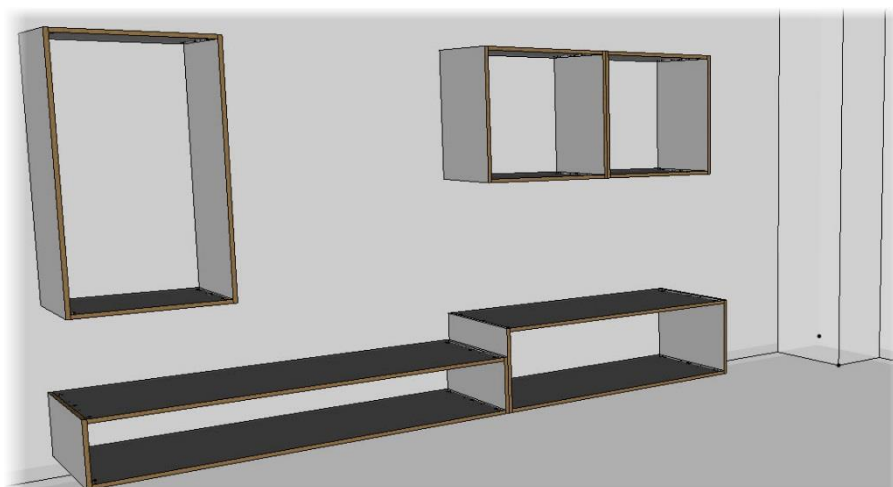
Adjust the display mode

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

Note: To insert the standard components in **simplified representation**, select **Tools** > **Options** > **Components** > **Components management** > **Representation**: **Simplified**.
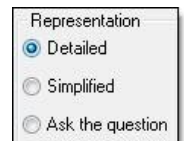
- Open the construction tree using **Ctrl + ²**.
- From the **Main set** tab, right-click the **ASSEMBLY** line and select **Representation**.
- Set **Representation** = **Simplified**.

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

Include the grooved backs

- Use the **Include standard** function to include the standard component **Grooved back**.
- Set **Housing mode = Housing**.

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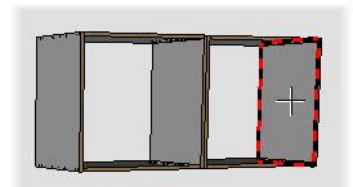
- Select one of the inner faces of a cabinet as the **cabinet block**.

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

- Set **Multiple inclusion = YES** and position the **driver block** in the other four cabinets.
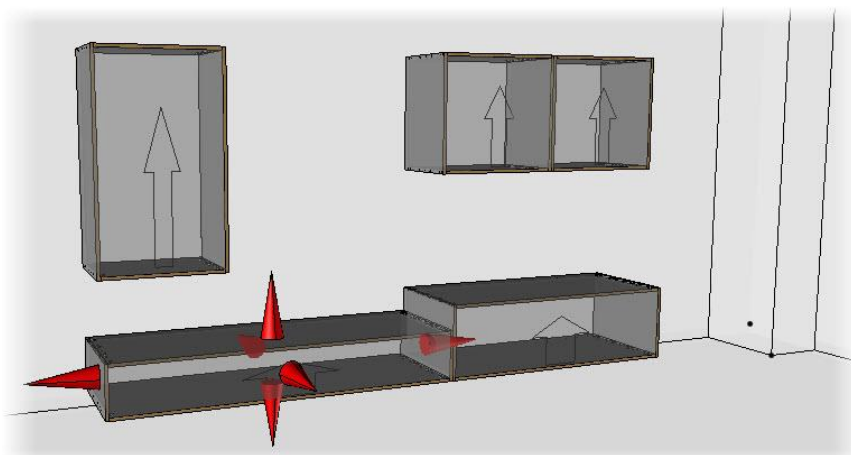
- Validate the **driver blocks** with **OK**.
- Then create the processes of the **Back** component using the **Automatic** function.
  AUTOMATIC
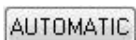
Note: To create the processes of a component after including it, use the **Assembly** > **Use process** function and select the component with the processes.
The **automatic** processes are only created on defined parts.

Include the shelf

- Use the **Include standard** function to include the **Wall shelf** component.
- Adjust the parameters for the shelf:

    - **Shelf length** = *1500mm*
    - **Shelf depth** = *300mm*
    - **Shelf height** = *250mm*
    - **Panel thickness** = *19mm*

The **Wall shelf** component is calculated, and then inserted based on the parameter values 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 bottom right-hand cabinet, **Distance** = *250mm*
    - Left-hand side of the shelf/Left-hand side of the bottom right-hand cabinet, **Distance** = *0mm*
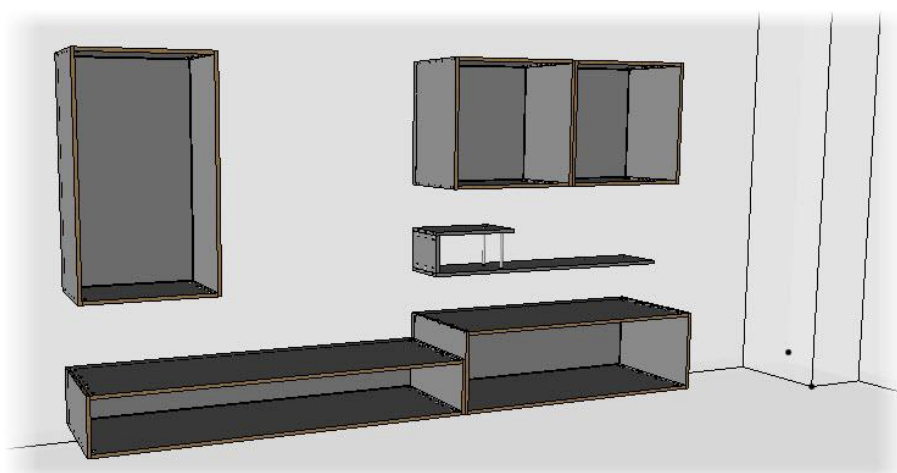
Define the set

- Define the set using **Wood** > **Define** > **Define set** > **Characteristics**:

    - **Designation**: *Lounge fitting.*

Exercise 5: Creation of 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:

- Component driven by a path or ***component with geometric drivers***
- Pipe shape
- Trim by planes
- Unbent parts
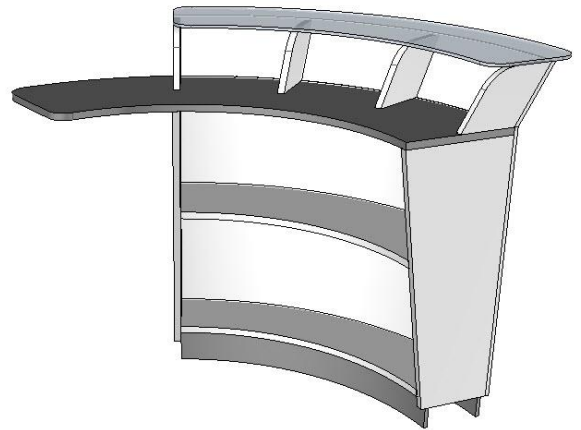
Note

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.
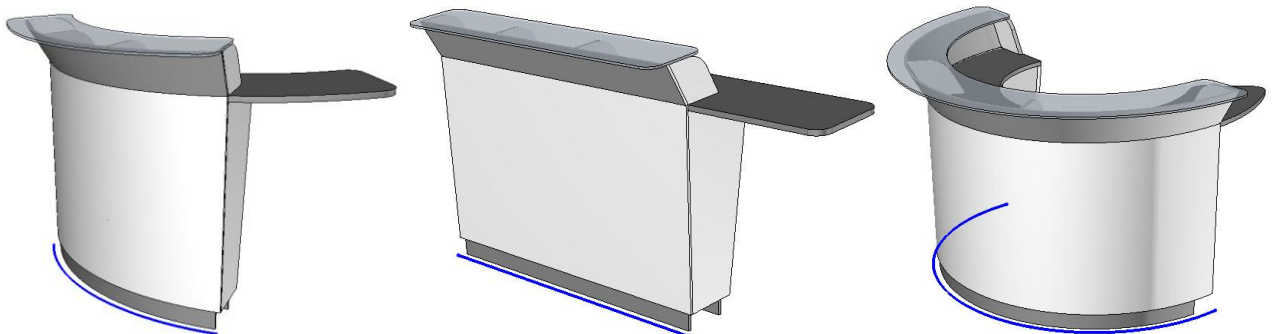
Creation of a component with geometric drivers

Create the driving path

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

Passing point

Note: The circle arc must be rather large to facilitate the design.
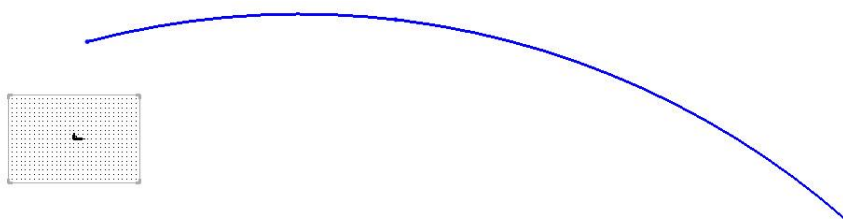
- Start the **Edit** > **Break associativity** function.

Note: **Breaking the associativity** of an element breaks the links it has with its construction elements. It therefore becomes completely independent.
This operation must be performed on the component's driving path to avoid any errors when assembling.
Moreover, a sketch cannot be basified. That is the reason why the driving path was created from a curve.

- Select the circle arc in **Elements to break their associativity**.

The circle arc then becomes completely independent.

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

| OK | Name: |t | Designation: |Trajectory |

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

- Make **layer 1** current.

- Start the **Tools** > **Coordinate system** function.
- Set **Wizard = YES**.

Wizard= YES⏁ Point, face, edge, curve or coord system:

Note: Several types of coordinate systems are available in TopSolid in order for you to create the most appropriate one for the current design. These coordinate systems can be found in **Tools** > **Coordinate system**.

The coordinate system wizard allows you, by selecting several elements, to automatically select the appropriate type of coordinate system, and then create it.

Here, a coordinate system on curve and point will be created in order to be positioned perpendicular to the curve path and at the end of this path.

- First select the circle arc, and then select the right end of the circle arc as the **point**.
- Select **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

- Open the list of parameters using **Parameter** > **Edit list**.
- Define the parameters as drivers by changing the **Driver** column's value from **No** to **Yes**.

Driver
Yes
Yes
Yes
Yes

Define the path as driver

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

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

Main	Favorite	Main set	Entities	Layers

DRIVERS SET : (5)
- td : Trajectory shift = 100mm
- h : Top height = 800mm
- sh : Shelf height = 350mm
- p : Top extension = 500mm
- t : Trajectory
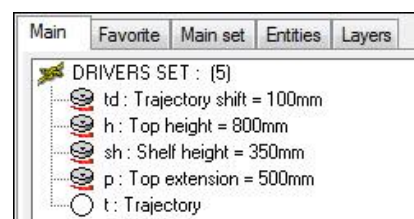
Save the component

- Save the component in the library: **Assembly** > **Define component** > **Edit/save template** > **Save standard template**.
- Adjust the following parameters:

  - **Standard**: **Training library**
  - **Family**: **Furniture**
  - **New type**: *Counter*
  - **New variant**: *Interior counter*

- Validate the window with **OK**.

Drawing the parts

Draw the bent front

- Turn off **layer 0** and set the **design color** to **black**.
- Start a **new sketch** on the current coordinate system.
- Draw a **rectangular contour** on the left of the coordinate system, and then dimension it automatically using the **Auto dimension** button.

- 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**.
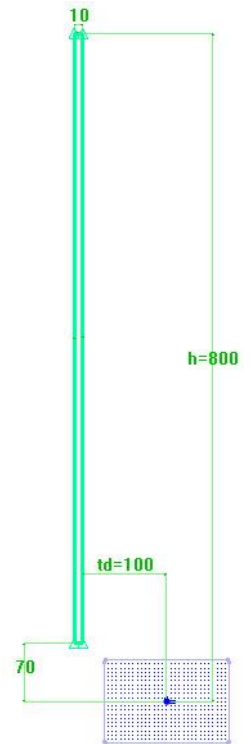
Draw the tops and baseboards

- Make **layer 2** current.
- Draw four **rectangular contours** as shown opposite. Use **Auto dimension** to place the dimensions automatically.
- Use **Modify parameter** to modify the rectangle dimensions to the dimensions shown opposite.
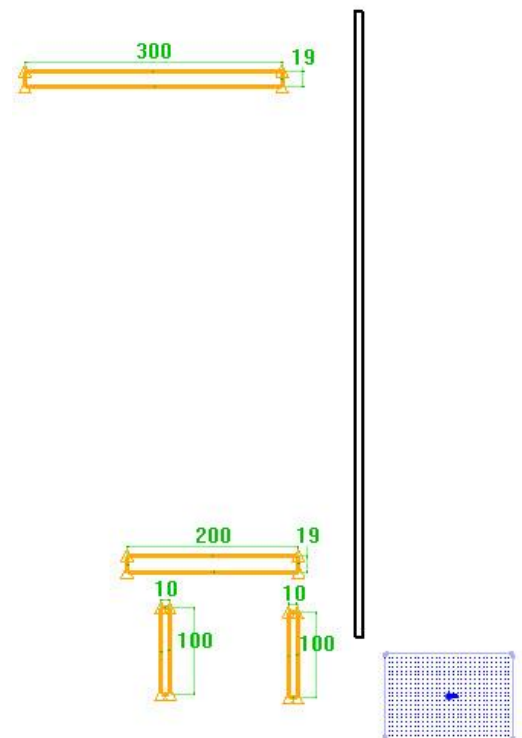
- Next apply:

  - 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*.
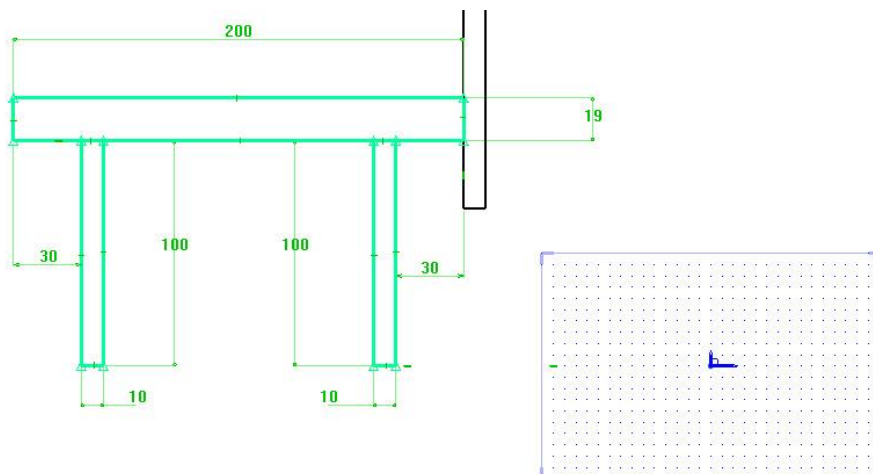
Draw the side

- 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 point and the bottom left point of the bent front;
  - an **orientation constraint** of the lower segment **along X**;
  - a **coincidence constraint** between the left segment and the bottom left point of the base.

- **End the sketch**.
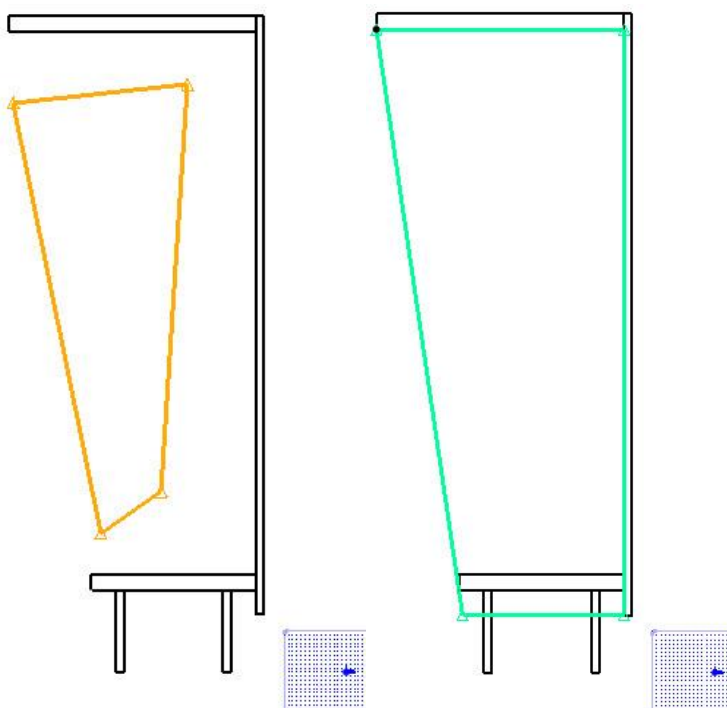
Draw the shelf

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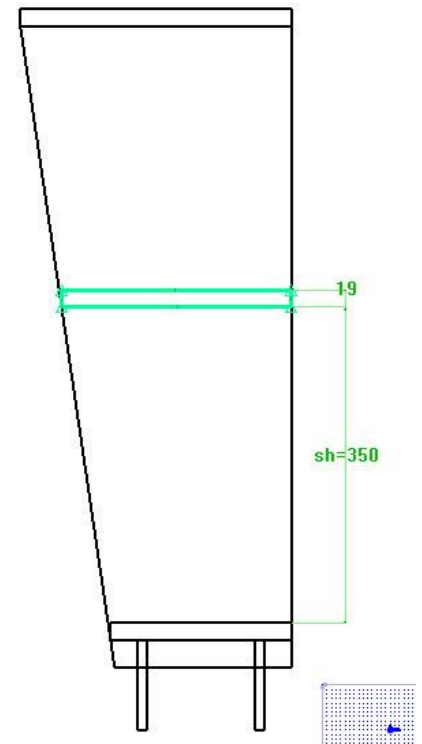
Creation of parts

Configure the current material and coating

Note: It is possible to define a current **material**, a current **coating** and a current finishing 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 coating for each part.

- In the status bar at the bottom of the graphics area, click the **Mat = …** button.

| Absolute cs | X=+235.000 | Y=-717.500 | Z=+000.000 | Tol= 0.2 | Tra=0 | Inv=Sho | Mat=particule board |

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.

| ✓ | Material and coating |

- Once you clicked on the button, set the current material and coating:

  - **Current material**: **TopSolid'Wood** > **Panels** > **Colors** > **White**
  - **Current coating**: **No coating**

- Validate with **OK**.

Curve 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**.

Pipe= ON CURVES   ▼ Follow= SUBSEQUENT OPERATIONS ⏮ Guide curve:

- Turn 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 adjust the pipe shape direction.
- Set **Corners type = Rounded**, **Curves = Visible**, **Generatrix sketch = Local** and **Type = Solid**.

Corners type= ROUNDED ⏮ Curves= VISIBLE ⏮ Generatrix sketch= LOCAL ⏮ Type= SOLID ⏮ Section curves or texts:

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

- In **Section curves or texts**, select the sketch of the base.
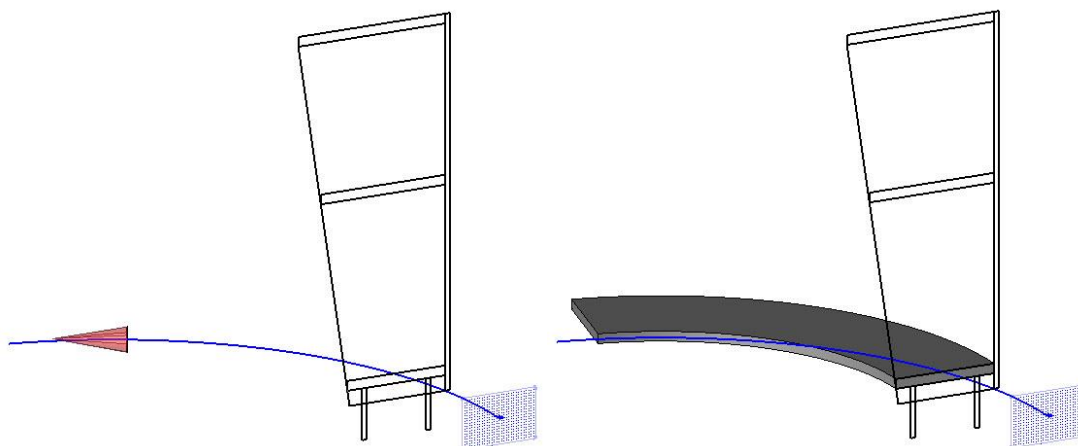
- Repeat the procedure to create pipe shapes for the two baseboards, the shelf and the bent front.

Note: The **Curves = Invisible** mode is always used by default. For this exercise, set the mode to **Curves = Visible** for every application.
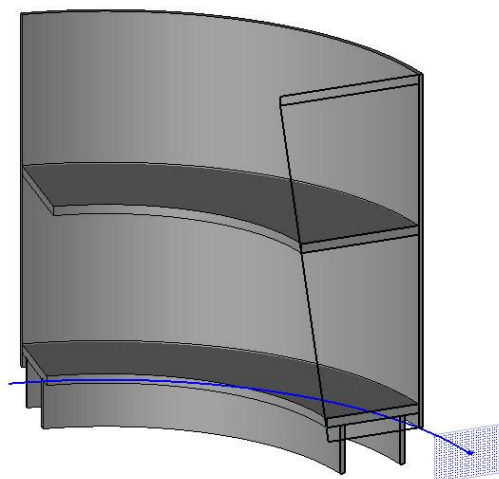
Curve the counter top

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 turn off **layers 0** to **3**.
- Change the **design color** to **green (10)**.
- From the **Curve** 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 **Curve** > **Extend** function, and then select the copied green curve of the left-hand side.
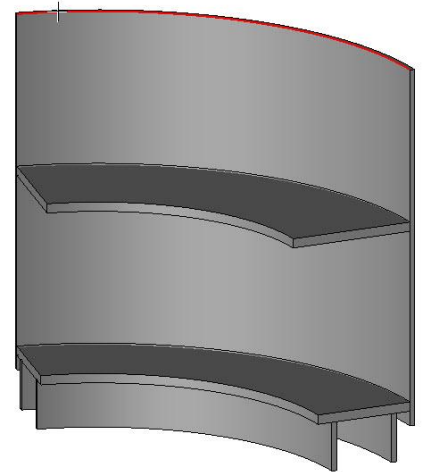
Note: The **Extend** function modifies the length of a curve by adding or removing a distance.

Curve to extend (click on side to extend): |

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

OK Type= CURVATURE ▼ Mode= EXTEND LENGTH Length= P

- Validate with **OK**.
- Turn on **layer 2**, and then create the **pipe shape** of the top by selecting the extended green curve as the **guide curve**.
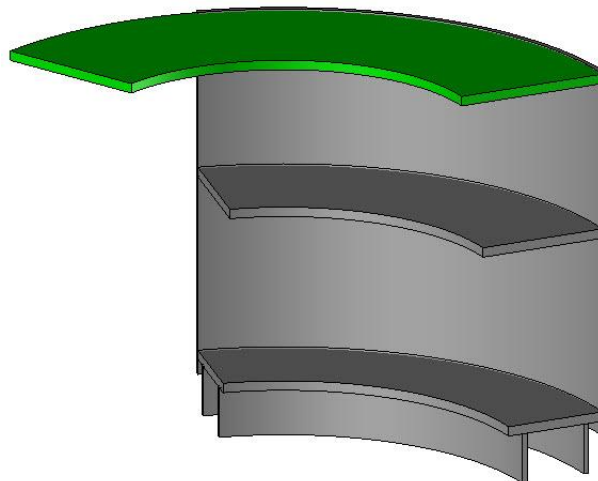
Repeat a sketch and extrude the sides

- Make **layer 6** current and turn 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**. ABSOLUTE COORDINATE SYSTEM

Note: As 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**.
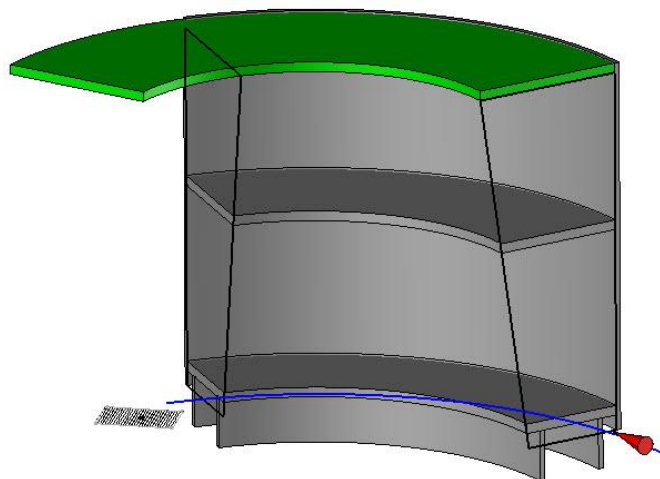
- Adjust the following parameters:

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

Distribution mode= DISTRIBUTE ⚡ Distance computing mode= ARC LENGTH ▼ Transformation mode= CONSTRAINT COORDINATE SYSTEM ▼

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

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 validate with **OK**.

EXIT Element for detection:

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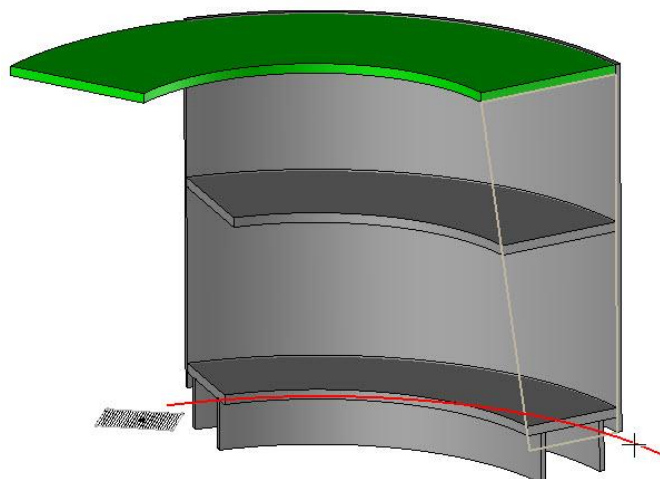
Trim the parts

Now let's trim the shelf and the base in relation to the sides.

- Start the **Shape** > **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 validate with **OK**.
- 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 validate with **OK**.
- Set **Hide tools = No** and click on **OK** to validate.
- Repeat the procedure to **trim** the parts in relation to the inner face of the second side.
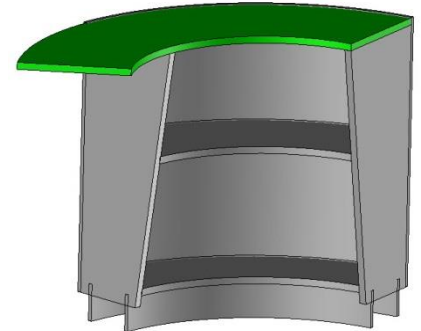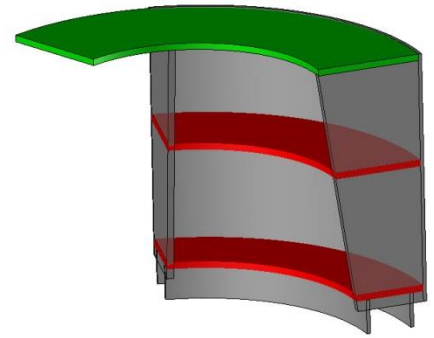
Subtract the parts

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

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

 The sides are then subtracted from the baseboards.
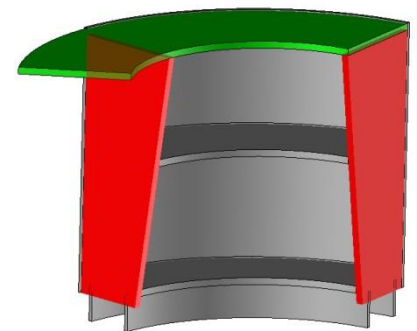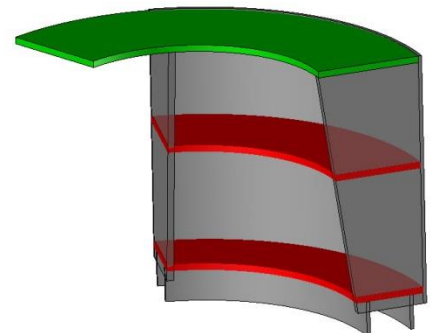
Definition of parts

Define the planar parts

Note: Since the counter parts are curved parts, you need to define the length and width axes manually in order to control and calculate the precise cutting-up of parts.
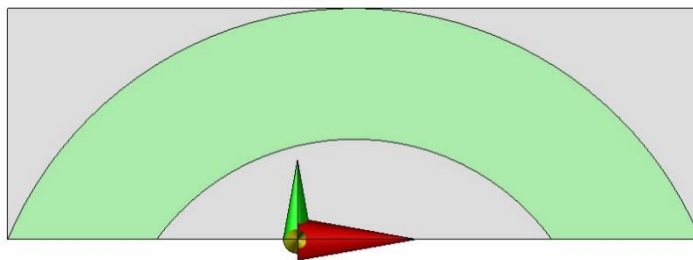
- Make **layer 7** current.

- 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 validate with **OK**.

Note: Not selecting the axes automatically will allow you to define them manually.

- For **Length axis**, select **Through point**. THROUGH POINT
- Select the front left point of the counter top as the **through point** like in the next image.
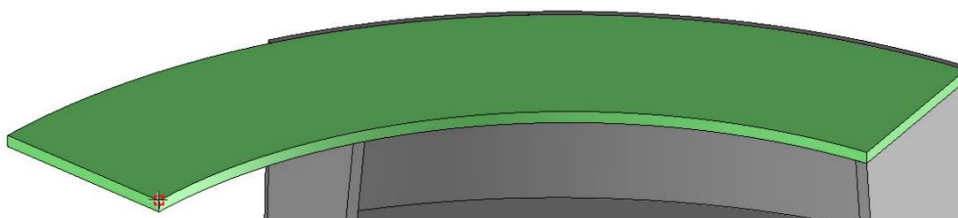
- Then select the front right point of the counter top as the **second through point**.
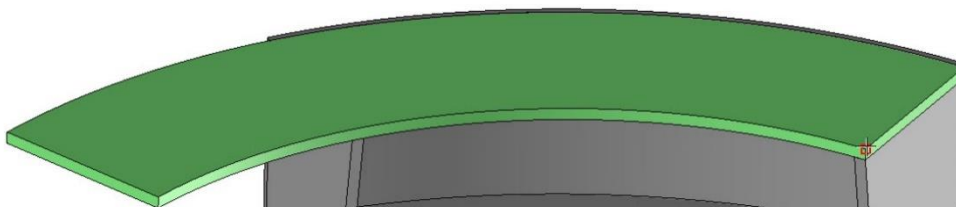
- Validate the direction of the default axis with **OK**.
- For **Width axis**, select the edge of the counter top width as shown below, and then validate its default direction with **OK**.
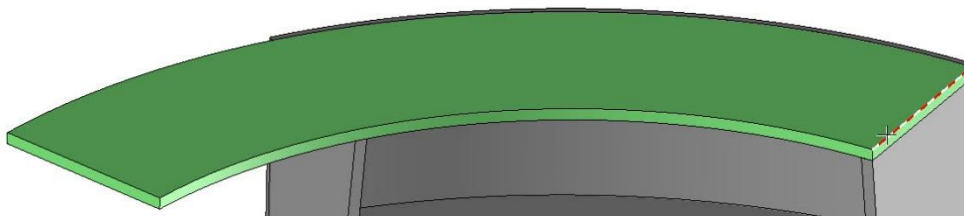
The cutting-up axes are placed on the part and the rectangular cut is calculated.
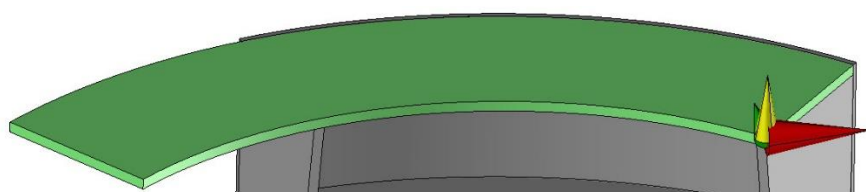
Sizes	Values	Modes	Over dimensions
Length	1140.0mm	additional	0mm
Width	473.2mm	additional	0mm
Thickness	19.0mm	additional	0mm

- Repeat the steps for the other two arc parts.
- Also define the two sides by specifying the following designations.
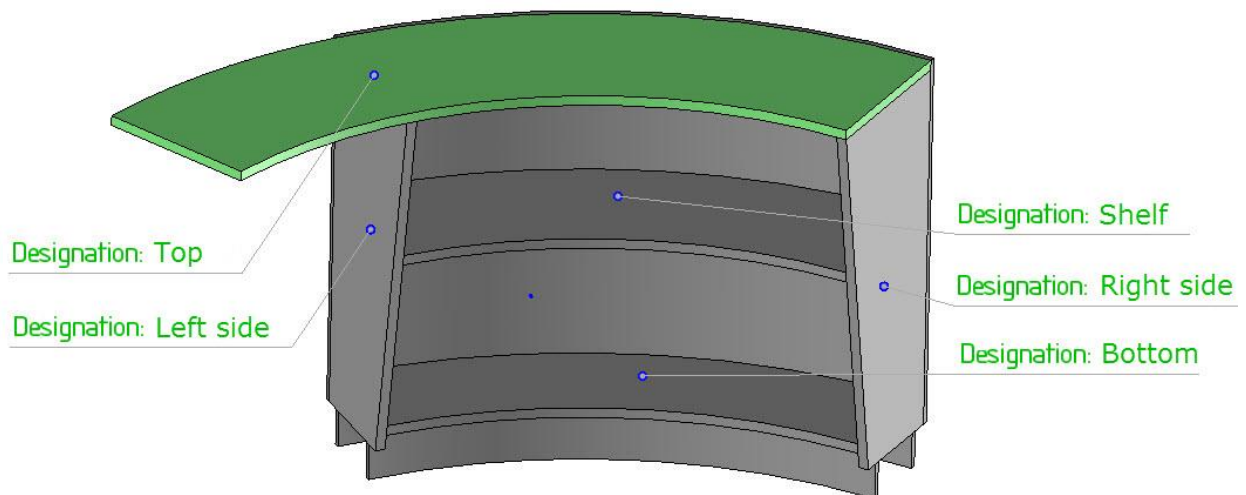
- For the top only, change the material to **TopSolid'Wood** > **Hardwoods** > **Beech hearted**.

Define the bent parts

Note: The baseboards and the bent front are **bent** parts. To calculate their precise cutting-up, parts must be unfolded.
Unfolding a part also allows this part to be machined before folding.
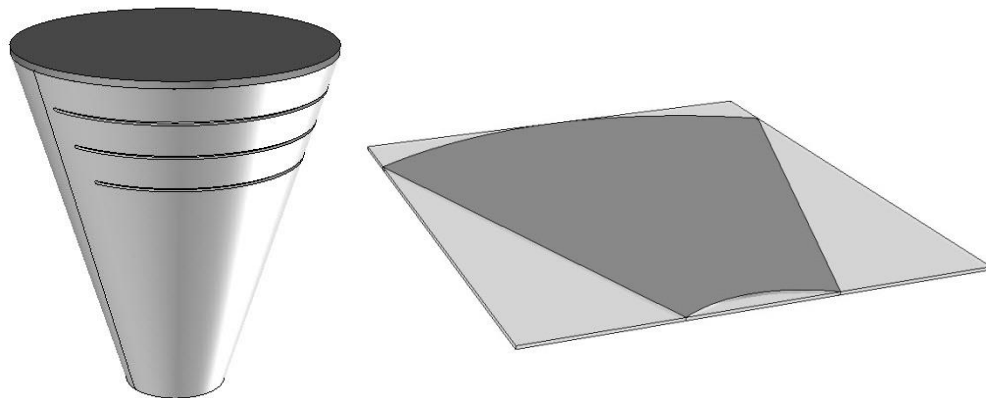
- Start the **Wood** > **Define** > **Define part** function, and then select the bent front of the counter.
- Set **Select axis automatically = YES** and **Bent part = YES**, and then validate with **OK**.
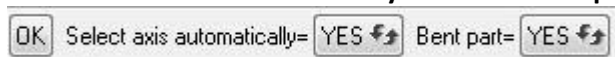
The settings for the unfolding calculation then appear.

- Adjust the following settings:

    - **Thickness** = *10mm*
    - **Neutral fiber coefficient** = *0.5*
    - **Layer** = *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**

Follow tangent faces= YES ⤴ | >> | Drills on reference faces= NO ⤴ Select faces to unwind:

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**.
- Validate with **OK**.
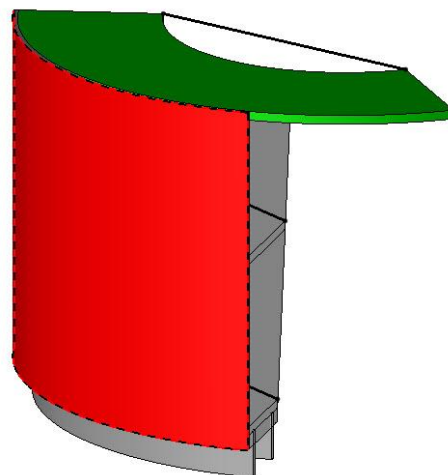
Note: To position the unfolding, an origin coordinate system, as well as a destination coordinate system must be selected.

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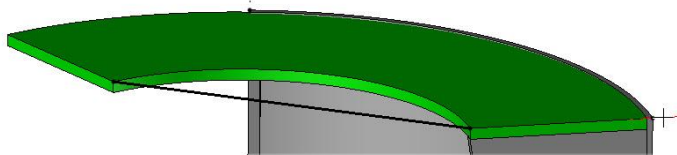
- Select any point to the right of the counter as the **destination coordinate system** to position the unfolding.

Note: The unfolding can be moved on the point of the destination coordinate system later using the **Move parents** function.
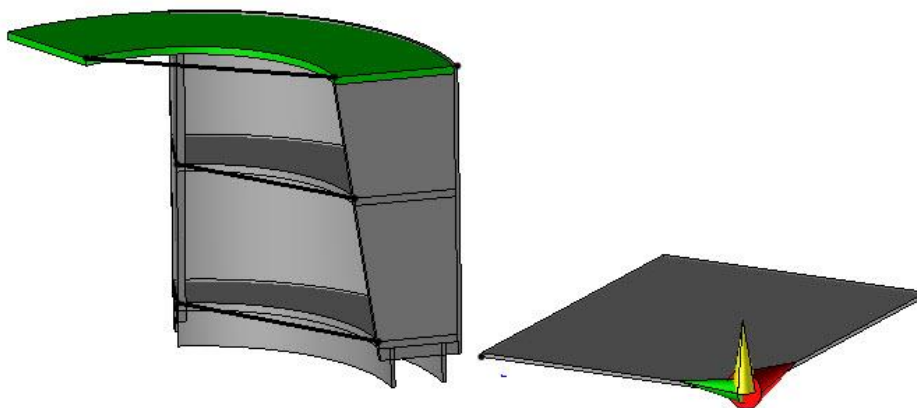
- Enter **Part designation**: *Bent front.*
- Validate with **OK**.

- Repeat the steps to unfold the other two baseboards:

    - **Designations**: *Baseboard 1 / 2*
    - **Material**: **TopSolid'Wood** > **Panels** > **Colors** > **Olive**
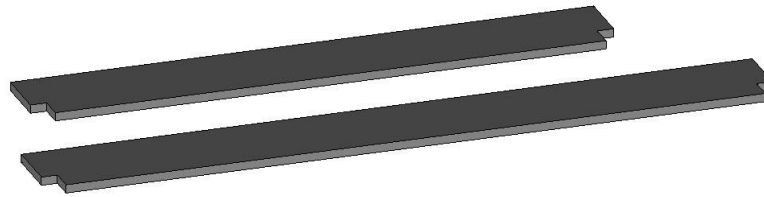
Define the set

- Define the set via **Wood** > **Define** > **Define set** > **Characteristics**:

    - **Designation**: *Interior counter*
    - **Assembly nature**: **Sub-assembly**

- **Save** the component.

Use of the component

Create the destination path

- Open the file *Interior design*.
- Make **layer 13** current.
- 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 add the **dimensions** as shown opposite.
- End the sketch.
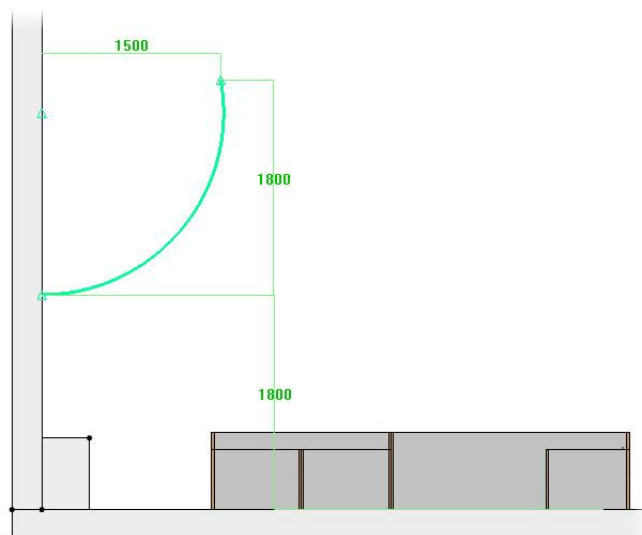
Insert the component

- Use the **Assembly** > **Include standard** function.
- Select the component **Furniture** > **Counter** > **Interior counter** and validate with **OK**.
- Set as drivers:

    - **Trajectory shift** = *0mm*
    - **Top height** = *800mm*
    - **Shelf height** = *350mm*
    - **Top extension** = *1000mm*
    - **Trajectory**: Select the previously created sketch

The component is automatically calculated according to the specified parameters and the selected path.
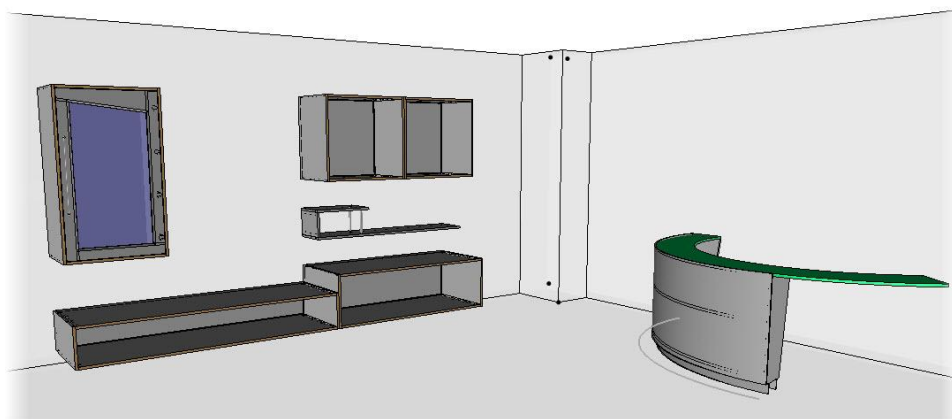
Note: Depending on the direction the circle arc is drawn, the component may be calculated on the other side of the circle arc.
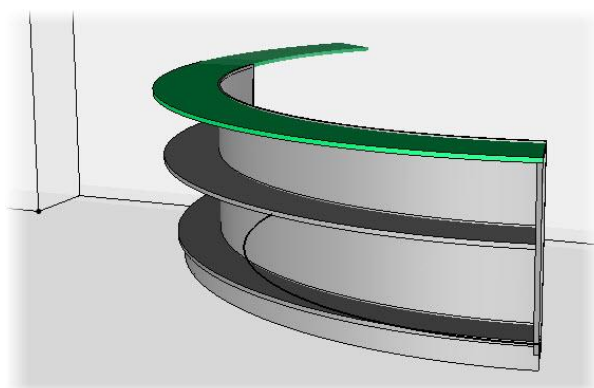
- To invert the path direction:

  - Delete the component.
  - Start the **Curve** > **Origin** function and select the previously created sketch as the **curve to modify**.

The arrow that appears represents the curve direction. This direction is used to calculate the component.
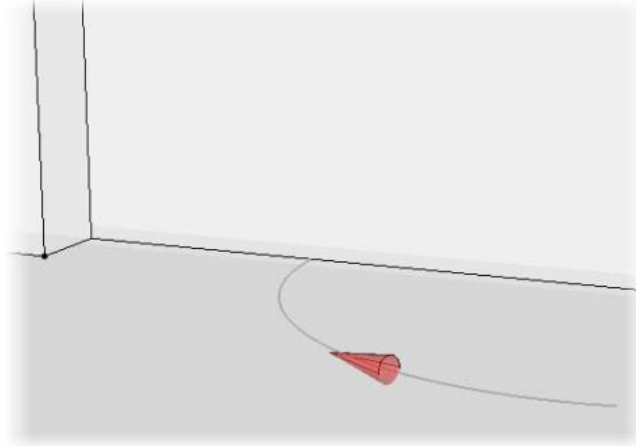
  - Invert the direction of the circle arc by clicking the red arrow. Click on **OK** to confirm.
  - Include the component again as shown above.

- Save the interior design file.

Exercise 6: Creation of a door

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

Concepts addressed:

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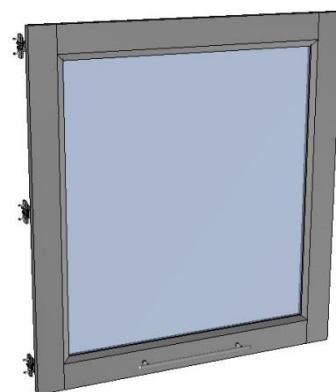
Creation of a moulding tool

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:
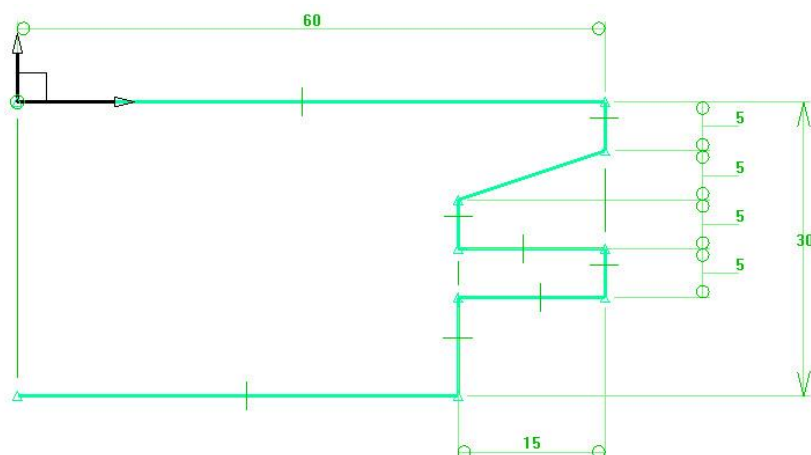
- End the sketch, and then draw the **new sketch** shown below.
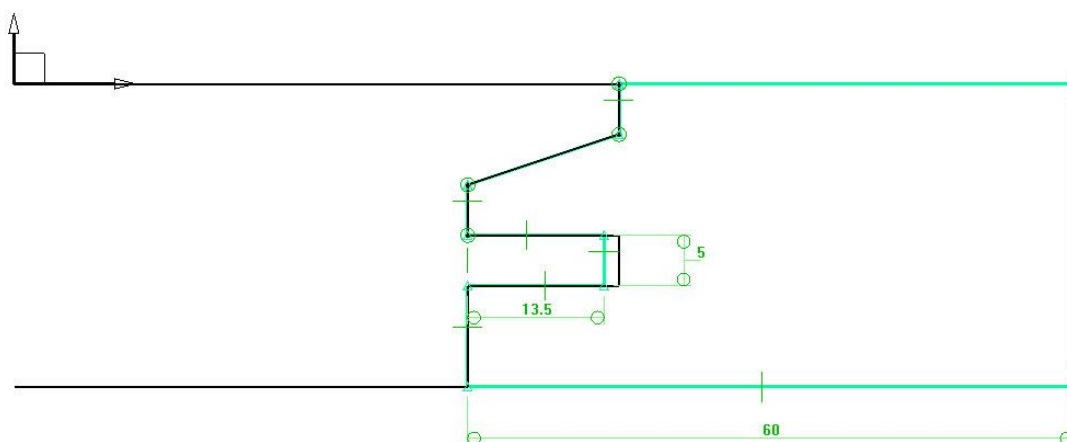
Note: In order to obtain a correct result during the machining (**TopSolid'WoodCam** or **machining interfaces**), it is necessary to draw the precise profile of the tool (radius, teeth, height…). However it is recommended to simplify the tool path to optimize the software performance.

- Turn off **layer 0**, and then create the two offset points shown below using the **Tools** > **Point** > **Offset point** function.
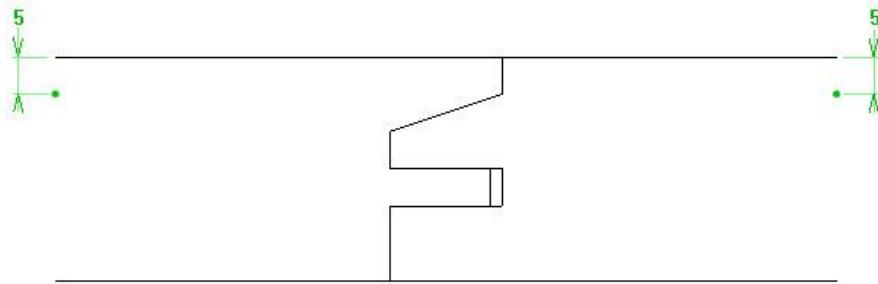
- Use the coordinate system wizard to create the coordinate system on the left by selecting the three points **1**, **2** and **3** shown below.
- Repeat the procedure to create the same coordinate system on the right.
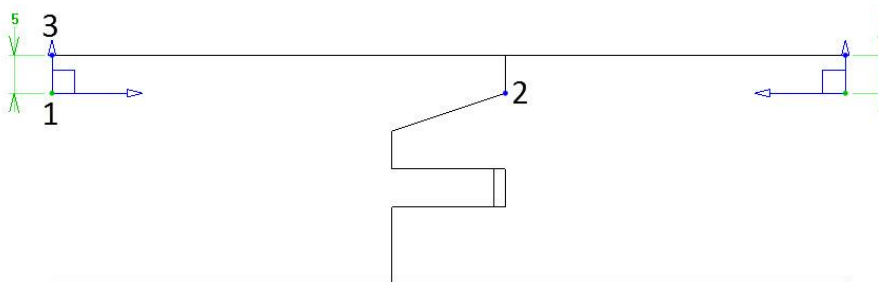
- Save this file in the standard library using **Assembly** > **Define component** > **Edit/save template** > **Save standard template**:

  - **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**.

Mode= | MOULDING ⇟ | Tool curve: |

- Adjust the following parameters:

  - **Tool name**: *tool1*
  - **Tool designation**: *Chamfer panel moulding*
  - **Tool number**: *105*

OK | Tool name: |tool1 | Tool designation: |Chamfer panel mo | 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.

- Validate the parameters with **OK**, and then select the left-hand coordinate system as the **position coordinate system**.
- Enter:

  - **Position name**: *origin*
  - **Position designation**: *Tool origin*

OK | Position name: |origin | Position designation: |Tool origin |

- Adjust **Tool radius**: *45* and **Cut depth**: *15*.
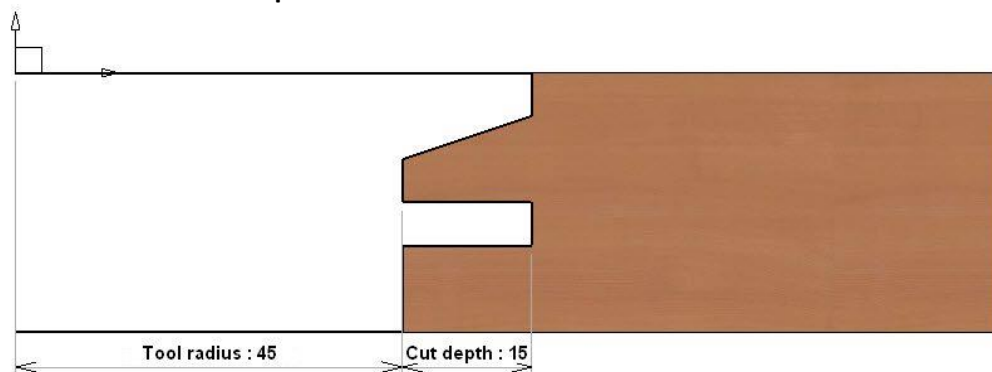
The tool is now defined and can be used to create the mouldings.

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**.

Mode= COUNTER MOULDING ⤴ Tool curve:

- Adjust the following parameters:

  - **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*

- Adjust **Tool radius**: *45* and **Cut depth**: *15*.
- **Save** and **close** this file.

Creation of a door component

Create the parameters

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

    - **Name of driving element**: *db*
    - **Default housing mode = Housing**
    - Select **Current coordinate system**
    - **Designation of the driving element**: *Cabinet block*

- Create the following three parameters 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.

- Enter **Thickness** = *cth* and **First shift** = *dg.*

AUTOMATIC | Section= NO | Thickness= cth=20mm | First shift= dg=2mm | First plane

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

Mode= FACES | Second shift= dg=2mm

- Enter **First shift** = *dg* and select the left 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**.
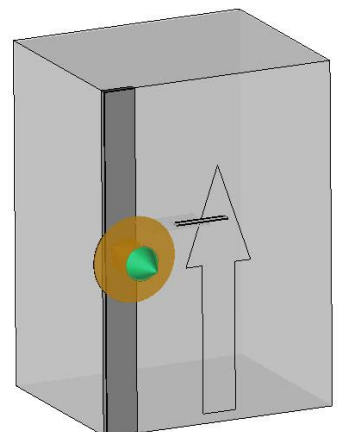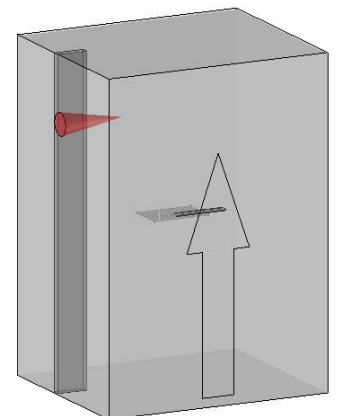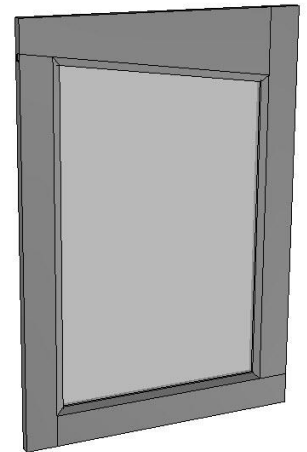
- Create the other three parts as **constrained blocks** as shown opposite.

  - 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.
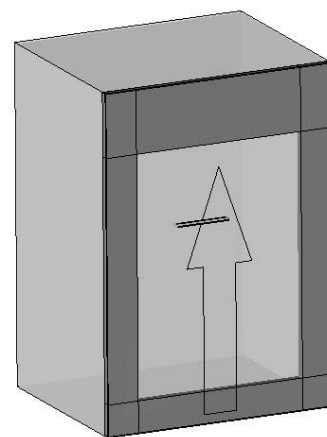
Saw the top crosspiece

- Turn off **layer 0**.
- Create a coordinate system on the top crosspiece using **Tools** > **Coordinate system** > **Wizard = Yes**.
- Place the coordinate system on the front face of the top crosspiece.
- Validate the coordinate system with a left mouse click once both displayed dimensions appear in yellow as shown below.

Note: The yellow dimensions cannot be modified. Here, as the coordinate system is centered on the face, the dimensions cannot be modified.
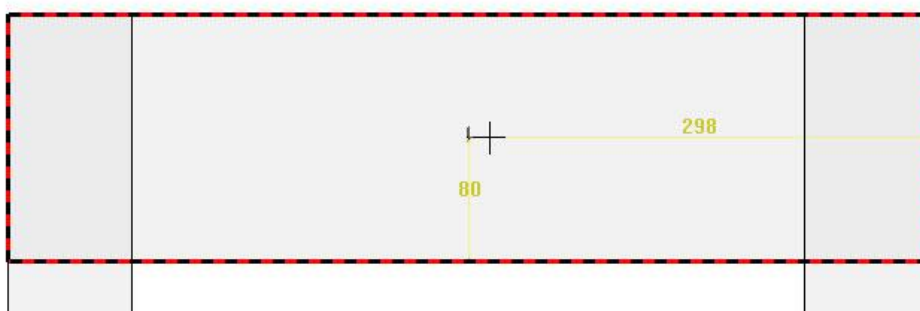
- Validate this coordinate system with **OK**. OK
- Use the red arrow to orient the X axis of the coordinate system on the door width as shown below.
- Set the coordinate system as current. CURRENT COORDINATE SYSTEM
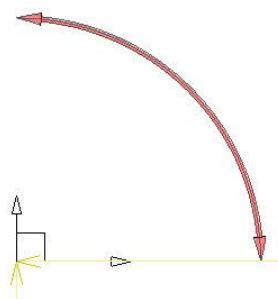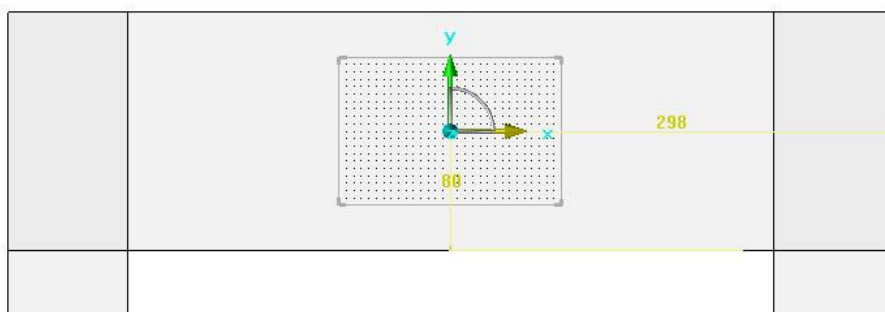
- In a **new sketch**, draw the following line:

  - The left point is **coincident** with the left edge of the crosspiece.
  - The left point is dimensioned to a distance of *cwi* from the upper edge.
  - The right point is **coincident** on the lower right corner of the crosspiece.
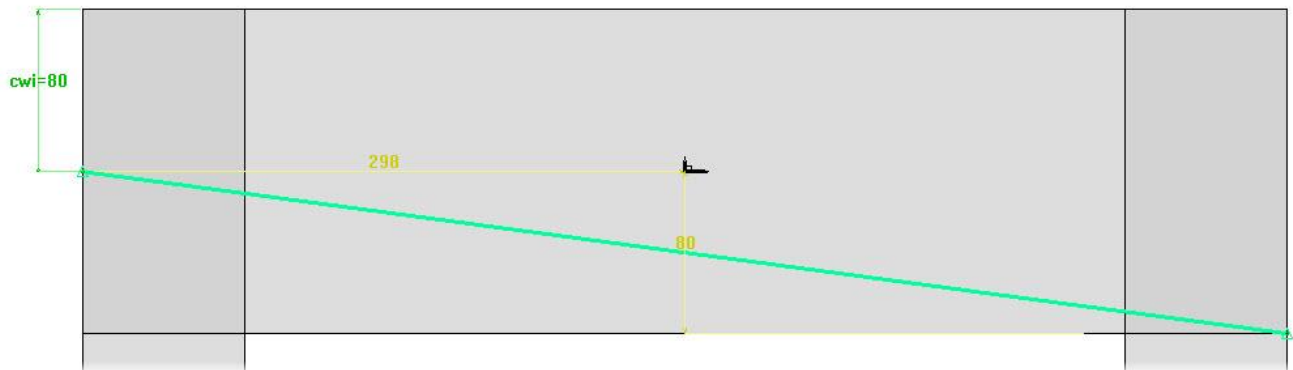
- From the **Wood** context, use the **Sawing** function to saw the top crosspiece in relation to this line.
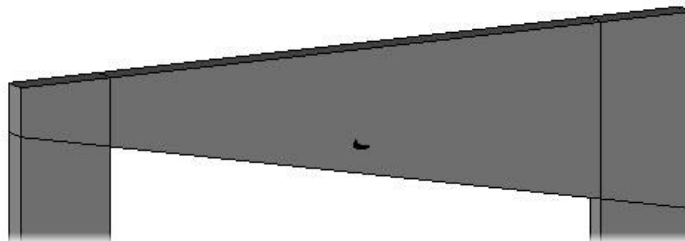
Create the glass panel

- 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**.

Second shift= -13.5mm   Allow non parallel faces= YES   Second plane

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,* and then select the right-hand edge of the left-hand jamb as the **second plane or point**.

The trapezoidal constrained block is automatically created.
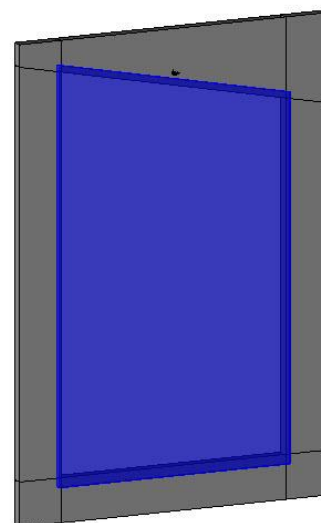
- Enter **Positioning shift** = *10mm* and select the front face of the top crosspiece as the **positioning plane**.
- Adjust the red arrow outwards and the yellow arrow as shown opposite.

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

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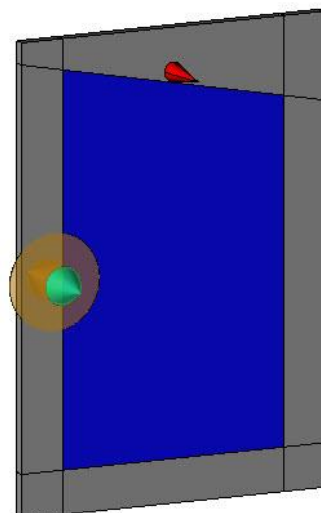
Create the mouldings

- Make **layer 1** current and turn off **layer 2**.

- Start the **Moulding** function from the **Wood** context.
- Set **Sweep = Planar face** and 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.
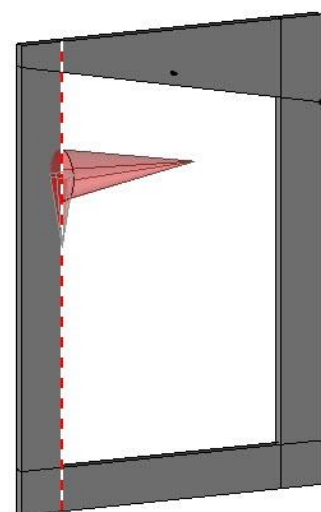
The **Parameters** window of the moulding opens.

- Select **Training library** from the **Standard** drop-down list.
- Select the standard **Tools** > **Moulding tools** > **Chamfer panel moulding**.
- Adjust the following parameters:

    - **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 create the moulding.
- Then use the **Copy operation** function to perform the same operation on the other three parts. COPY OPERATION
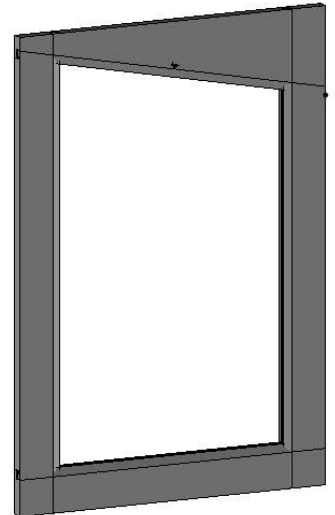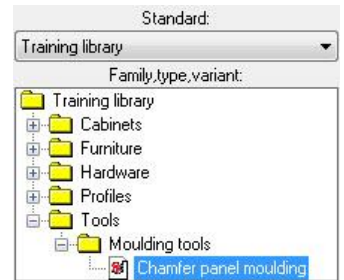
Create the counter-mouldings

- 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**.
- Validate the default parameters with **OK** to create the counter-moulding.
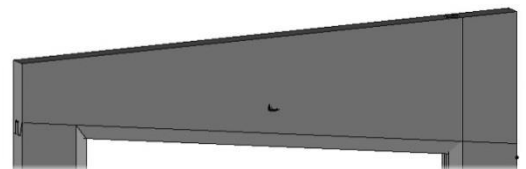
- Then perform the following **counter-mouldings**:

    - **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**.
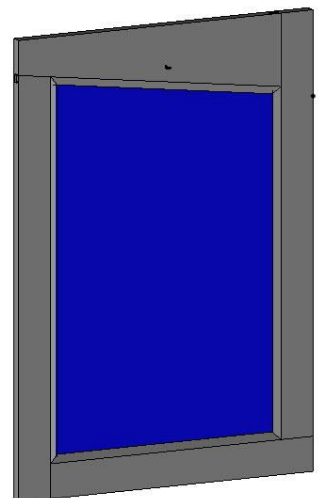
Define the parts

- Using the **Define part** function, define the different parts of the document.
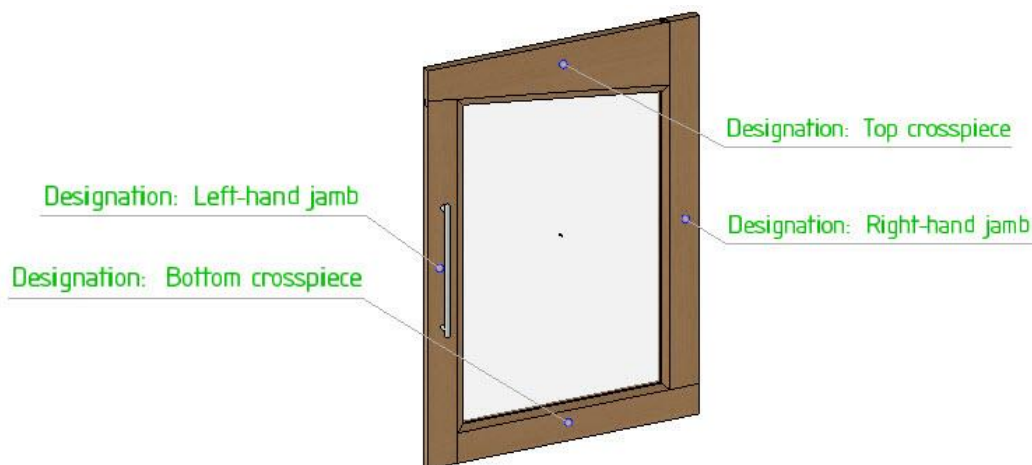
- Define the materials for the jambs and crosspieces: **TopSolid'Wood** > **Hardwoods** > **Beech hearted**.
- Then define the material for the glass: **TopSolid'Wood** > **Glasses** > **Clear window glass**.

Define the set

- Use the **Define set** function to define the set:

  - **Designation**: *Glass panel door*
  - **Assembly nature**: **Sub-assembly**

- Save and close the **Glass panel door** component.

Creation of a handle component

- Open the provided file *Line handle*.

Note: To design this component, refer to the *TopSolid'Wood Basics Training Guide*, Exercise 6: *Supplement - 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.

- From the **Shape** context, start the **Block** function and adjust the following parameters:

  - **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

- From 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**.

Mode= NON DYNAMIC

- Select the left-hand vertical cylinder of the handle as the **first alignment face or edge**.
- Then set:

  - **Hole**
  - **Through one**
  - **Diameter**: *4mm*
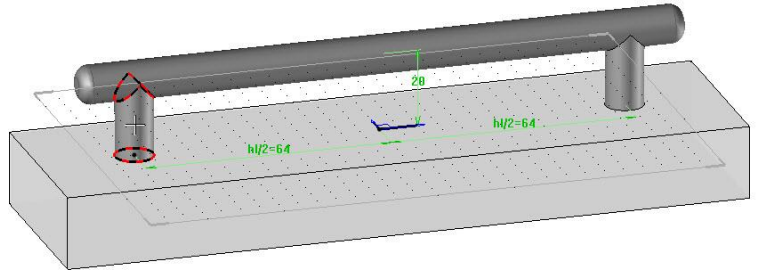
- Perform the same drilling on the top face of the martyr part in relation to the right-hand vertical cylinder of the handle.
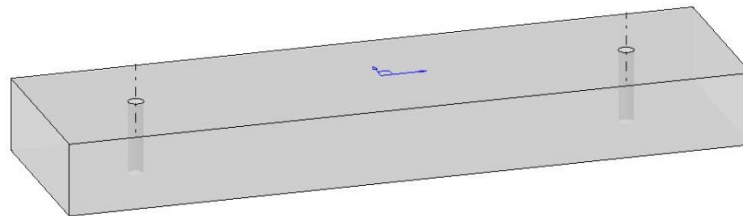
Define the drillings as component processes

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

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

Define tools

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

Operation type= LOCAL OPERATION ON SHAPES   ▾   Name of tool element: tool1

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

OK   Name: tool1   Designation: Handle drillings
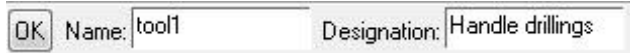
- In **Local operation to insert in tool**, select one of the previously created drillings.
- Validate the default **local operation name**.
- Select the second drilling, validate its default **name**, and then validate the operation selection with **Stop**. STOP
- Validate the **Operation as tool definition** 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.
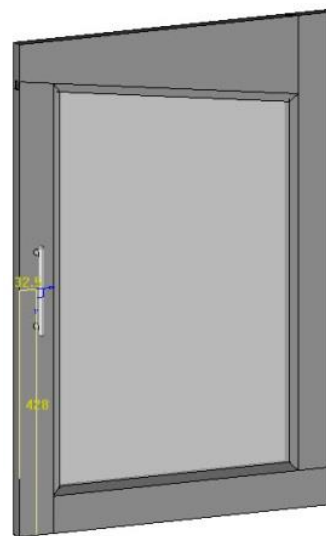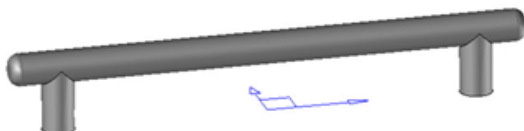
- Change the **design color** to **blue**.
- Create a coordinate system on point via **Tools** > **Coordinate system** > **Wizard = Yes**.
- Select the origin of the current coordinate system as the **origin point** in order to create a new coordinate system like the current coordinate system.
- Validate the coordinate system on point with **OK**.

- Start the **Assembly** > **Define component** > **Define key-points** function.
- Select the coordinate system that was created before as the **key point or key coordinate system to insert**.
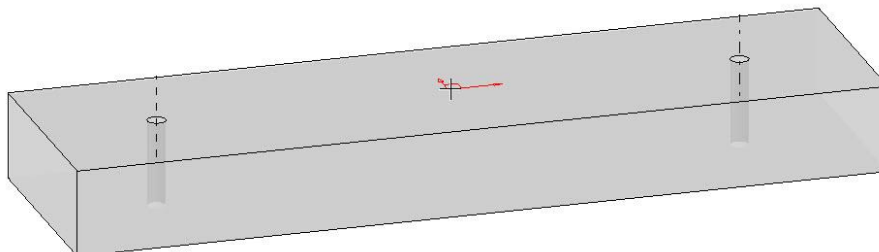
- Enter **Name of key coordinate system**: *cs1.*

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.*

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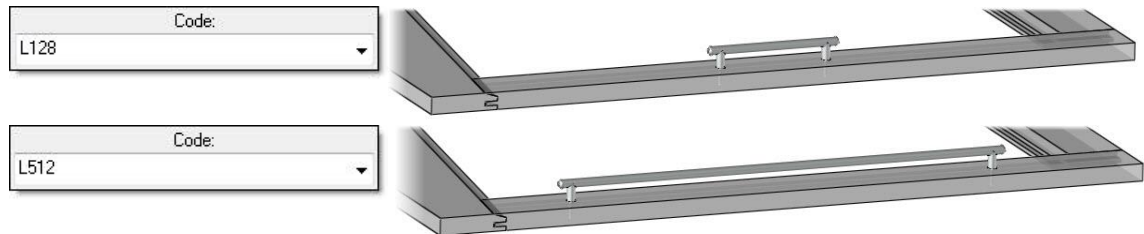
- Create the catalog using **Assembly** > **Define component** > **Edit catalog header**.

  ![Edit catalog header]

- Then select **All parameters and texts** in the drop-down list to manage all the document parameters in the catalog.

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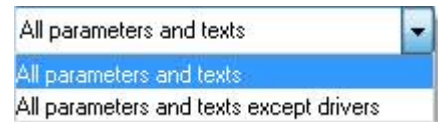
Note: The **catalog** of the component is automatically generated in **Excel** format. If Excel is installed on your computer, the **catalog** will be opened in Excel; if not, the catalog will be opened with **Notepad**.

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	hl
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**. ![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**.

Code:
L256
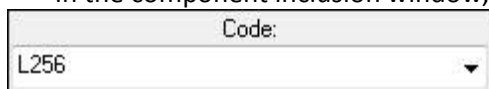
- Position the handle on the **left-hand jamb**.
- Select the left-hand edge of the jamb as the **first alignment edge**.
- Enter **Distance** = -(cwi-15)/2.

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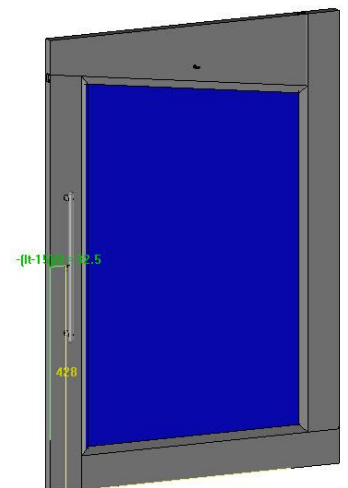
- Click on **Stop** so that the **handle** is not repeated.
- Select **Automatic** to automatically create the tools of the handle and drill the **left-hand jamb**.
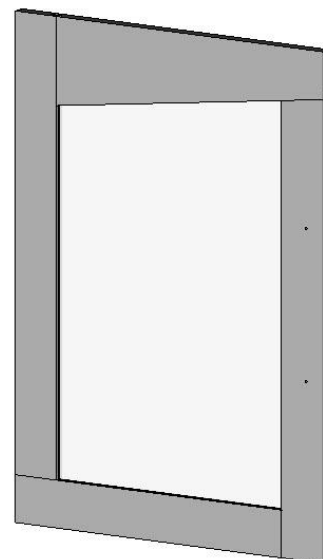
Creation of a hinge component

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.
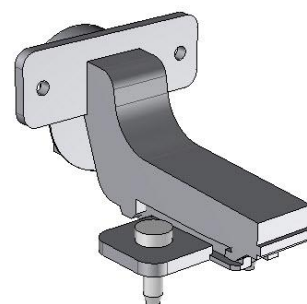
Use the body of the hinge

- 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 part:

    - **Designation**: *Inserted hinge body 107°*
    - **Reference**: *75t1750*
    - **Supplier**: *Blum*

- Make **layer 1** current.

- Define the document's **absolute coordinate system** as a **key point** via **Assembly** > **Define component** >

    Define key-points, and then select the coordinate system.
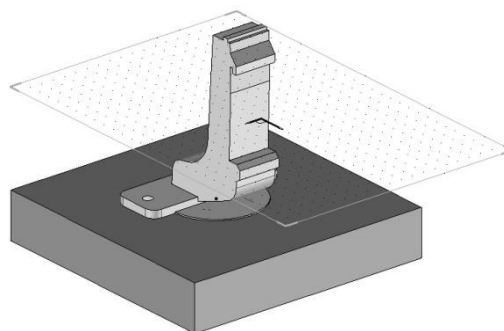
    KEY-POINTS / KEY-COORDINATE SYSTEMS SET : (1)
    └── ABSOLUTE COORDINATE SYSTEM : ABSOLUTE COORDINATE SYSTEM

- Create the rectangular sketch that will allow the extrusion of the martyr part.
- When extruding the sketch, open the advanced parameters. [>>]
- Then select the hinge's bottom support face as the **shift origin face** in order to extrude the part from this face.
- Validate the parameter with **OK** and extrude the part to *19mm*.
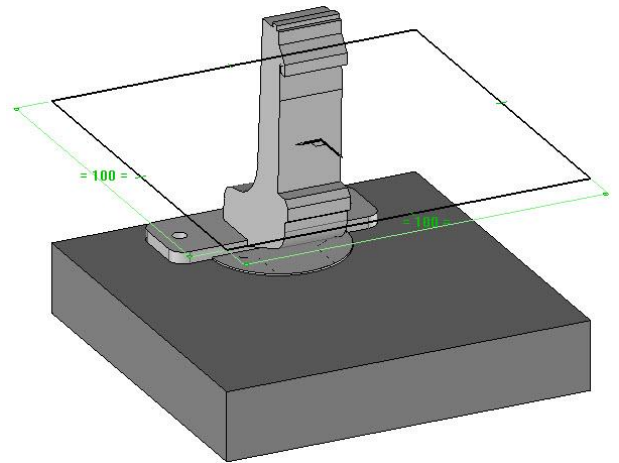
- On the martyr part, create two **blind holes** aligned on the drillings of the hinge's screws:

    - **Diameter**: *5mm*
    - **Depth**: *12mm*
    - **Bottom angle**: *0°*
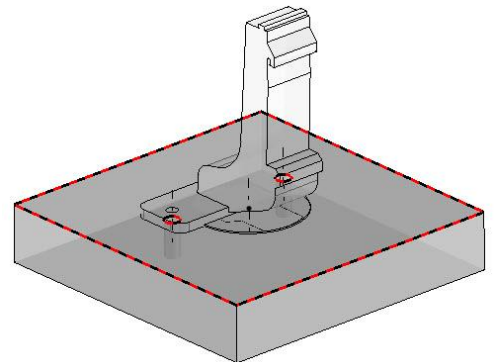
- On the martyr part, create a **blind hole** aligned on the hinge:

    - **Diameter**: *35mm*
    - **Depth**: *12mm*
    - **Bottom angle**: *0°*
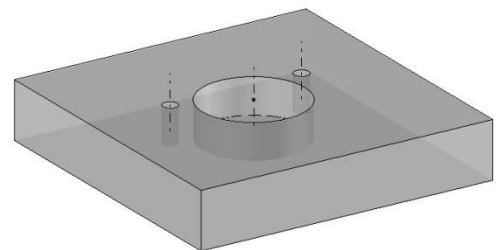
- Define these drillings as component **tools**:

    - **Name of tool element**: *tool1*
    - **Designation of the tool element**: *Door drillings*

- Insert the three created drillings in this tool.
- Save and close the file.
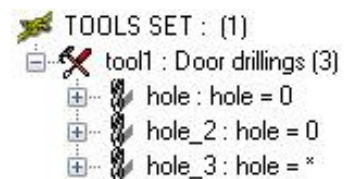
Use the base of the hinge

- 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 part:

  - **Designation**: *Base CLIP TOP*
  - **Reference**: *174e6100_01*
  - **Supplier**: *Blum*

- Make **layer 1** current.

- Define the document's **absolute coordinate system** as a **key point** via **Assembly** > **Define component** > **Define key-points**, and then select the coordinate system.

- Create the martyr part from a sketch positioned on the support plane of the base.
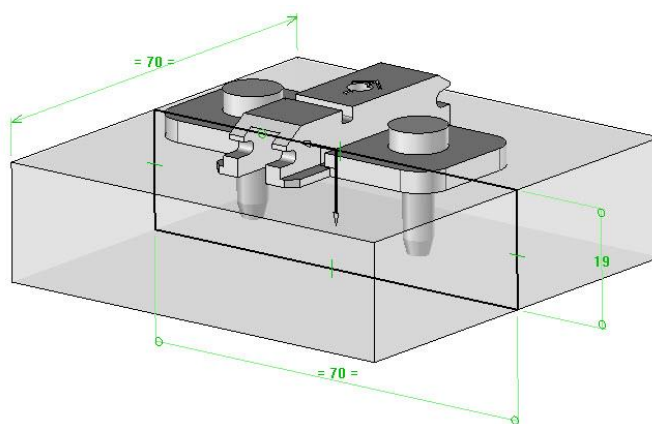
- On the martyr part, create two **blind holes** aligned on the drillings of the base's screws:

  - **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 two created drillings in this tool.
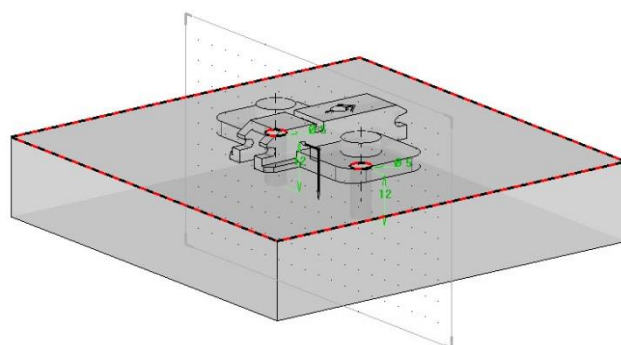

```
TOOLS SET : (1)
  tool1 : Base drillings (2)
      hole_1 : hole = 0
      hole_2 : hole = *
```

Create the mounted hinge component

- Create a **new Design document**.
- Include the standard component **Inserted hinge body 107°**.

- Select the absolute coordinate system of the new document as the **destination coordinate system**.
- Repeat the same operation to assemble the base of the hinge.
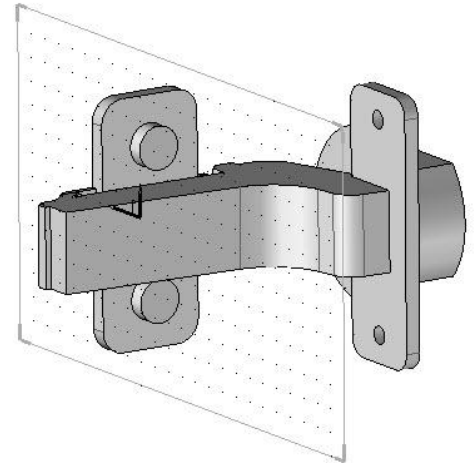
Note: In general, when assembling hardware parts provided by manufacturers, positioning the elements absolute coordinate system on absolute coordinate system allows them to be exactly positioned relative to each other.

- Save the mounted hinge in the **training library**:

  - **Family**: **Hardware**
  - **New type**: *Hinges*
  - **New variant**: *Inserted hinge*

- Define the part:

  - **Designation**: *Inserted hinge*
  - **Supplier**: *Blum*

Create the key point of the hinge

Note: As the hinge is positioned in relation to the cabinet side, the key point will be positioned in relation to the absolute coordinate system of the complete hinge.

- Start the **Tools** > **Coordinate system** > **Wizard = Yes** function.
- Select the **absolute coordinate system**.

Note: When selecting a coordinate system, the wizard will create a duplicate coordinate system.
The six red arrows allow you to create a translated coordinate system, while the **advanced options** allow you to create another transformation.

- Select the **Advanced options** button. ADVANCED OPTIONS
- Adjust the following parameters for the transformation:

  - **Rotation**
  - **Axis**: **Y-**
  - **Rotation angle**: *90°*

- Define the created coordinate system as the **key point**: **Assembly** > **Define component** > **Define key-points**.

  - **Name of key coordinate system**: *cs1*
  - **Designation of key coordinate system**: *Cabinet side*

- Save and close the file.
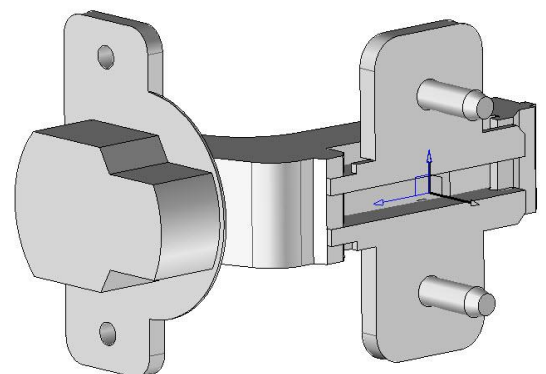
Insert the hinge on the door

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

Note: The **Unit type = No unit** mode will create a parameter that allows a quantity to vary. 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 door height.

- 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

- Validate the window with **OK**.
- Enter:

  - **Name**: *hn*
  - **Designation**: *Number of hinges*

- Display the driver block on the **layer 0**.

Note: As the hinge is positioned on the cabinet, you need to create a martyr part to position the hinge.

- Using a **constrained block**, create the martyr part to the right of the driver block. Position this part outside the block as shown opposite.
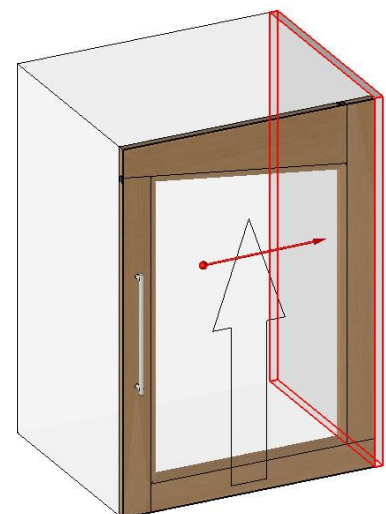
- Turn off **layer 0**.
- Insert the hinge using the **Assembly** > **Include standard** function.
- Select the inner face on the martyr side as the **destination coordinate system**.

- Select the lower edge on the martyr side as the **first alignment face or edge**.
- Enter **Distance** = *cwi + dg + 40.*

- Select the front edge on the martyr side as the **second alignment face or edge**.
- Enter **Distance** = *38.5 + cth.*
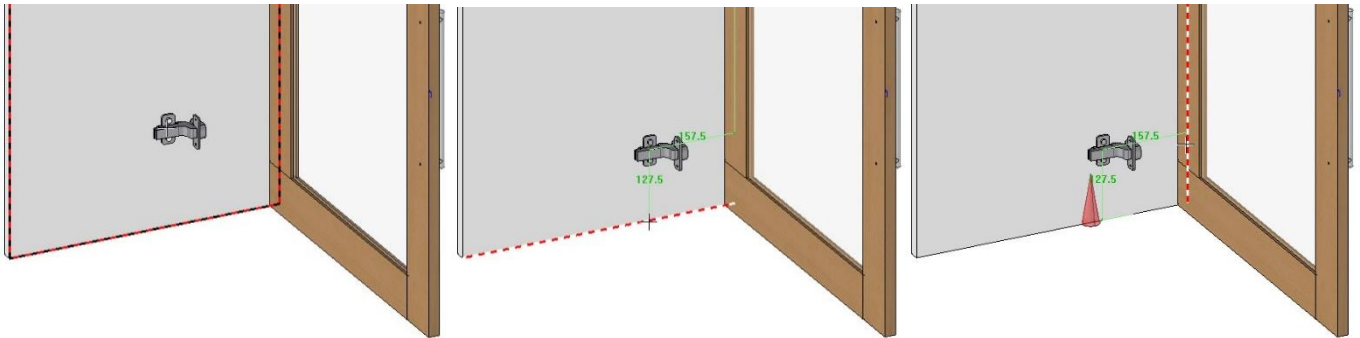
- Select the **Repeat** option to directly repeat the hinge. REPEAT
- Select **Propagation**: **Linear**, **Propagation direction: Y+**.
- Enter **Total distance** = *db.z-(2*cwi)-(2*dg)-80*

Total distance = db.z-(2*cwi)-(2*dg)-80 Give priority to ends= NO

- **Total number**: *hn*

Total number: hn

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

Use of the door

- Open the file *Interior design.top.*
- From 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.
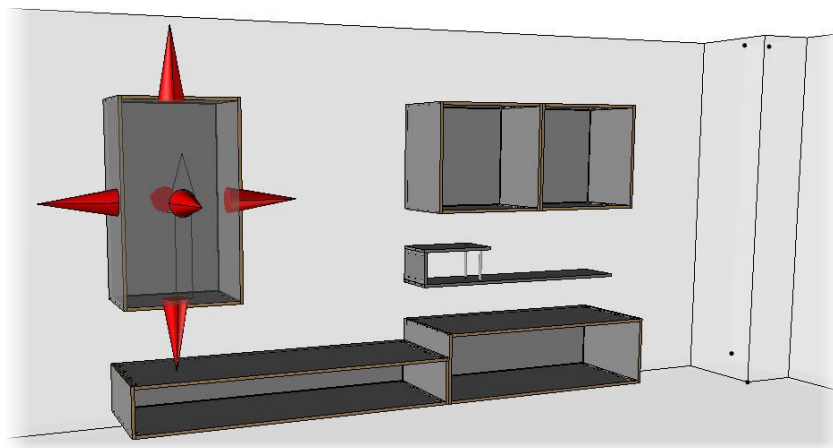
- Validate the default values of the driving parameters, and then perform the machinings using the **Automatic** option.

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.
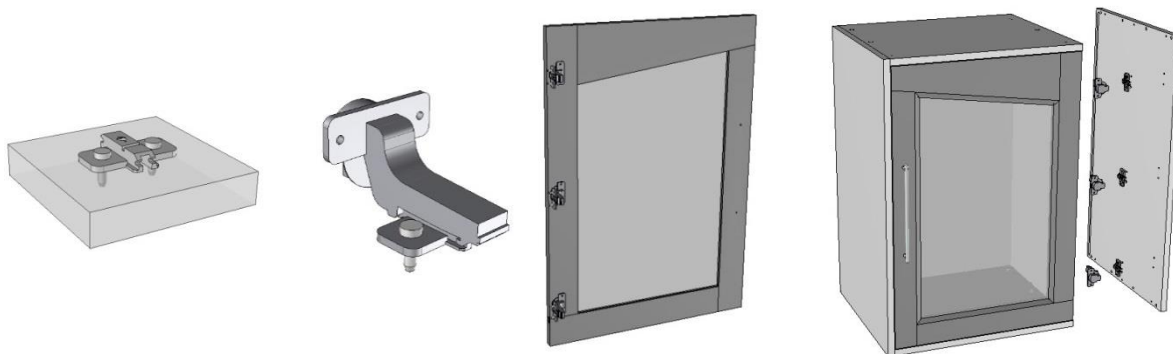
Supplement: Declaration of the handle as a sub-component

> **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**.

- Start the sub-component function: **Assembly** > **Define component** > **Define sub-component**.

- Select the **line handle** as the **component**.

- Enter **Name** = *Handle* and press **Enter** to confirm. 　Name: Handle

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 select the door.

- Click the **Sub-component** button to modify a sub-component of the door. SUB-COMPONENT
- In the window that appears, expand the node of the door, click on **Line handle**, and then click the **Interchange** button.
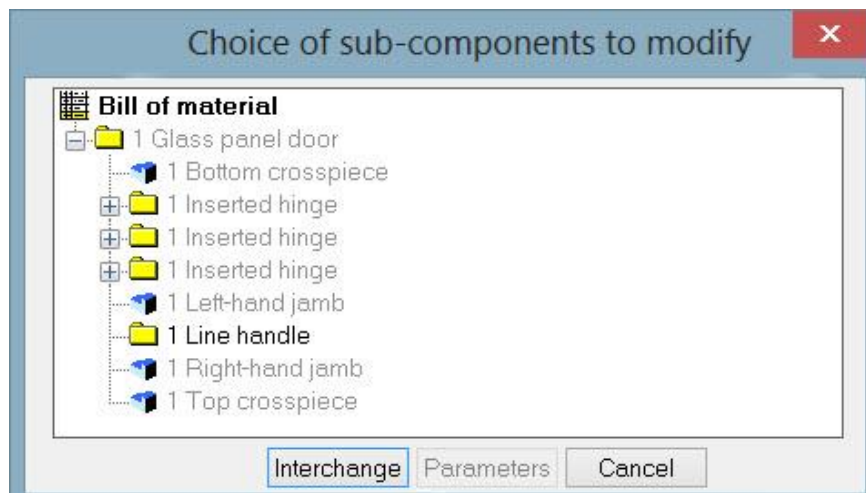
- Change the **code** of the handle and click on **OK** to validate.

The handle is then modified in the assembly.

Exercise 7: Creation of glass shelves

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

Concepts addressed:

- **Linear constraint** distribution of a constrained block
- Measured parameter
- Multi-drilling
- Driver propagation

Creation of 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 and 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 **layer 1**, create the shelves as **constrained blocks**:

    - **Thickness** = *5mm*
    - **Planes**: Four faces of the **driver block**

- When selecting the **positioning plane**, use the **Linear constraint** option.
  `LINEAR CONSTRAINT`

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

- Select the bottom 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**.
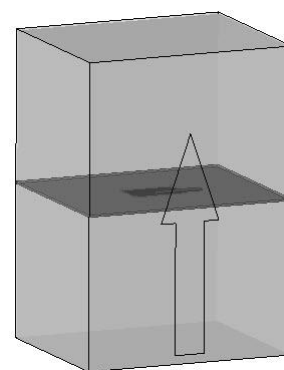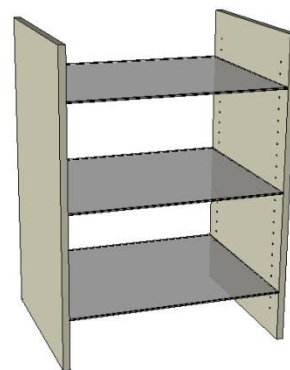
- From the **Distribution definition** window, select the **Advanced** distribution mode.
- Adjust the following parameters:

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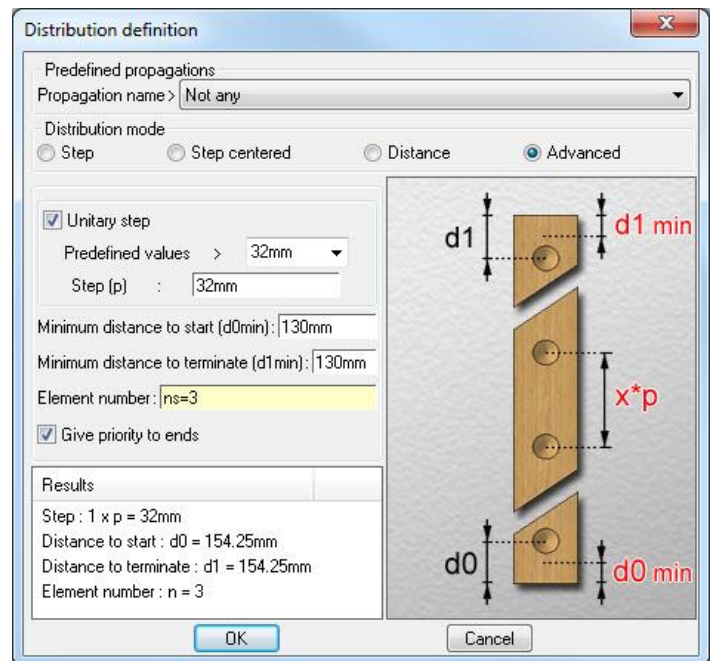
Checking the **Give priority to ends** option changes the distribution mode of the repetition instances. This is necessary for the rest of the exercise (measured distance parameter).

Do not give priority to ends	Give priority to ends
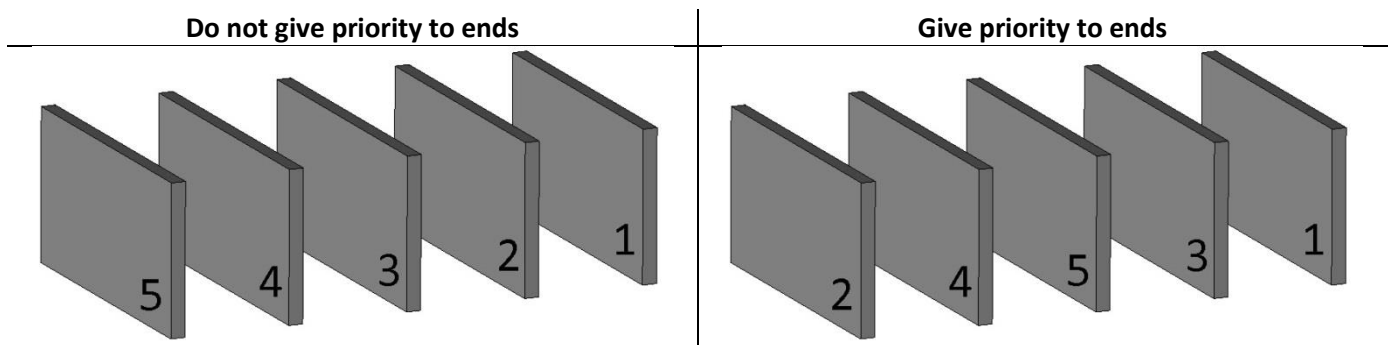
- Click on **OK** to validate the **Distribution definition** window.
- Center the **constrained blocks** in relation to the calculated distribution.
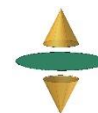
Modify the repetition template

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. `EDIT TEMPLATE`
- From the **Shape** 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. `SHOW REPETITION`
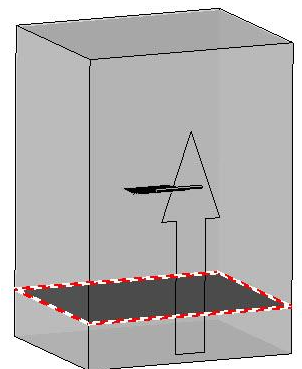
Define the shelves

- Define the shelves:

  - **Designation**: *Glass shelf*
  - **Material**: **TopSolid'Wood** > **Glasses** > **Clear window glass**

- Define the set:

  - **Designation**: *Glass shelves component*
  - **Assembly nature**: **Sub-assembly**
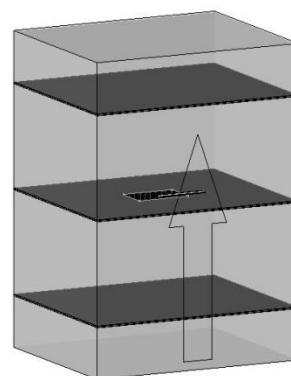
Creation of processes

Create the martyr parts

- Make **layer 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.
- Turn off **layer 0**.
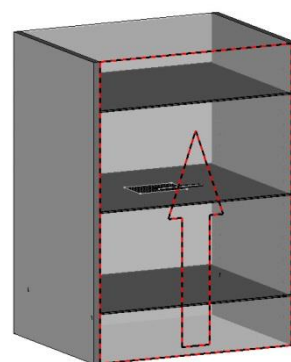
Create the measured parameter

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

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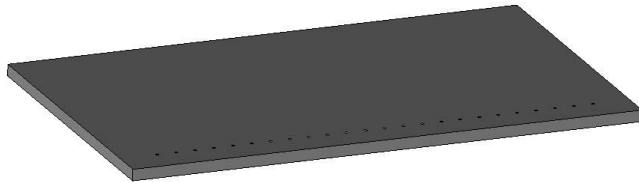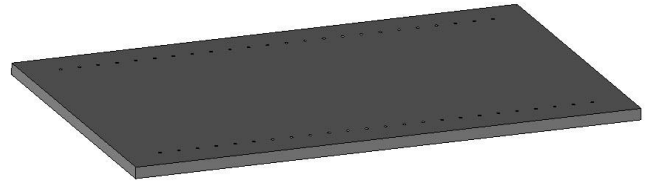
- From the **Wood** context, start the **Drilling** function.
- Select **Define model** to create a new drilling. `DEFINE MODEL`
- Create this drilling in **Model** = **User** mode and name this model *Shelf support 5x12*.

Model= USER ⚡ Drilling new name: Shelf support 5x12

- From the drilling definition window, select **Hole** in the left-hand list, and then move it to the right-hand list. Click on **OK** to validate.
- Adjust the following parameters:

  - **Hole**: **Blind**
  - **Diameter**: *5mm*
  - **Depth**: *12mm*
  - **Bottom angle**: *0°*

Diameter	: 5mm
Depth	: 12mm
Bottom angle	: 0°

- Validate 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**.

Propagation= DOUBLE ⚡ 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**.
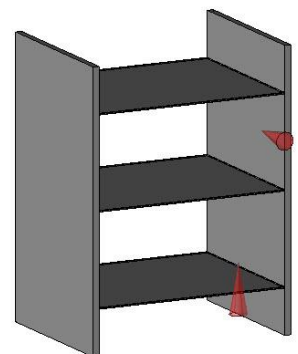
The distribution definition window opens.

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

Element to propagate
*** User drilling models ***
Shelf support 5x12

- Adjust the parameters 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**

- Adjust the parameters 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 **multi-drillings** on the second martyr part.
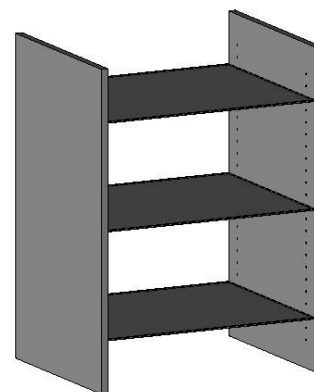
Define the multi-drillings as processes

- Define the multi-drillings as component processes using **Assembly** > **Define component** > **Define tools**.

Machining	Name of tool element	Designation of tool element
Right multi-drillings	*tool1*	Right drillings
Left multi-drillings	*tool2*	Left drillings

- Save and close this component.

Include the component

- Open the file *Interior design.top*.

- Include the **Glass shelves** component via **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**.

Housing mode= LIKE A COMPONENT ▼ Merge drivers= NO ⚡ Reference component

Note: The **Merge drivers = Yes** mode is used to merge the **driver blocks** of both 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**. SHIFT
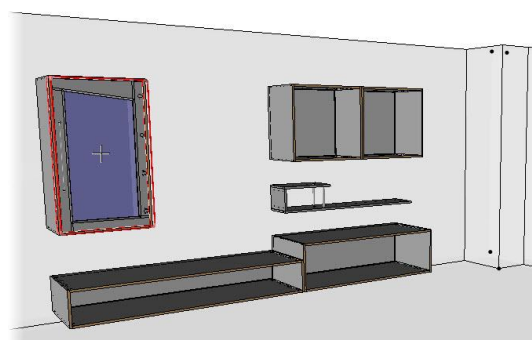
- Enter **Shift** = *50mm* and select **OK** to validate.

OK SUPPRESS SHIFT Shift= 50mm

- Validate the **driver block** with **OK** and adjust the **Number of shelves** parameter.
- Perform the component processes automatically using the **Automatic** option.
- Save the component.
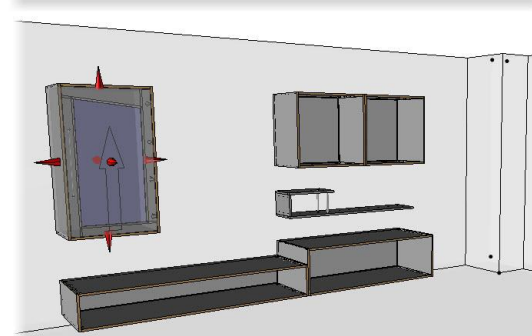
Supplement: Shelf propagation as driver

> **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**.
- From 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.

- **Right-click** on the **Propagation** line and select **Define driver**.

  Define driver

- Adjust the following parameters:

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

  OK   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.

OFFSET INSTANCE

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.
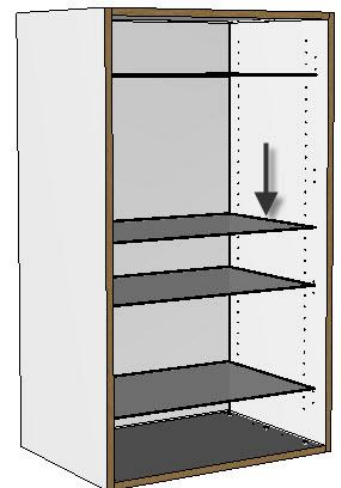
Exercise 8: Creation of separation panel components

The goal of this exercise is to create 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.
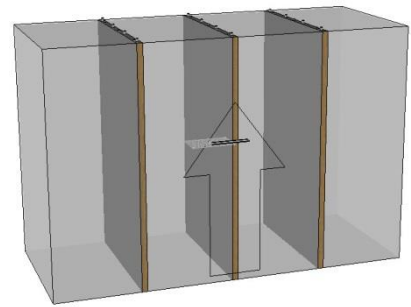
Concepts addressed:

- – Automatic assembly

Creation of the shelf component

Create the parts

- Create a **new Design document**.
- Create the following three parameters in **Unit type = Length** mode:

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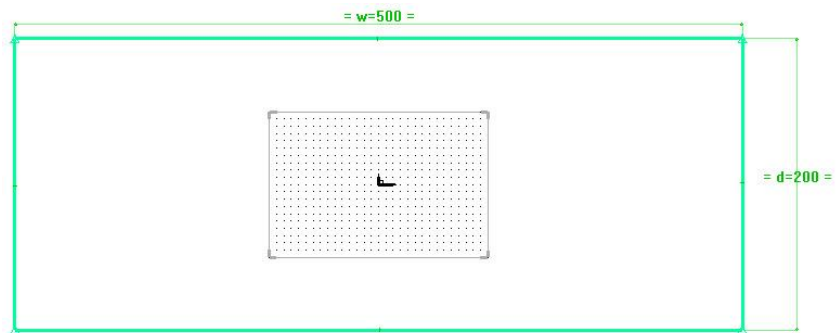
- Extrude this sketch to a **height**: *th.*

- Then draw the sketch of the martyr parts as shown opposite.
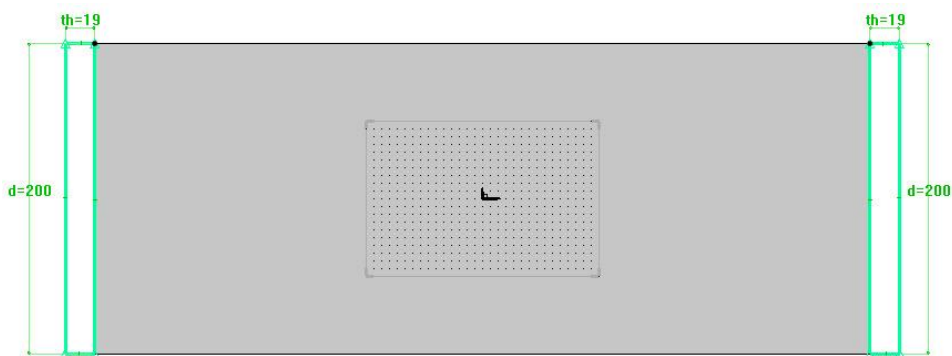
- Make **layer 1** current, and then extrude the martyr parts in **Alignment = Centered**, **Generatrix sketch = Global** and **Result = One shape per profile** modes to a **height** = *200mm.*
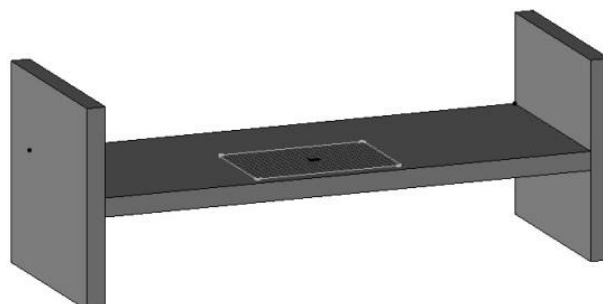
- Create a panel on the shelf using the **Panel** function.

Note: A panel can be created only on a defined part.

- Define the shelf:

  - **Designation**: *Fixed shelf*
  - **Material**: **TopSolid'Wood** > **Panels** > **Colors** > **White**

- Configure the panel:

  - Place an edge on the front face of the panel.
  - Use the codified edge **EDG-TH-2-PVC-OAK**.
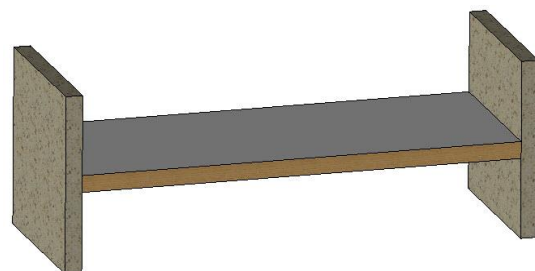
	N°	Codification	Edge type - code	Length	Beginning cut ty...	End cut type
☑	1	EDG-TH-2-PVC-OAK	Flat edge - ep 2	695.0mm	Covering	Covering
☑	2	EDG-TH-2-PVC-OAK	Flat edge - ep 2	232.5mm	Covered	Covered
☑	3	EDG-TH-2-PVC-OAK	Flat edge - ep 2	695.0mm	Covering	Covering
☑	4	EDG-TH-2-PVC-OAK	Flat edge - ep 2	232.5mm	Covered	Covered
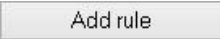
- Save the component in the **training library**:

  - **Family**: **Cabinets**
  - **Type**: **Separation panel**
  - **New variant**: *Fixed unitary shelf*

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.*

Rules
Dowels + Cams

- Add a **Dowel** component to the **automatic assembly**.
- Select **Wood dowel** and the **D8 L30 code**, and then validate with **OK**.

The **Wood dowel** component is added to the automatic assembly.

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

- Then add a **Cam** component to the **automatic assembly**.
- Select **Simple cam** and the **L34 19 code**, and then validate with **OK**.
- Set the **Cams predefined propagation** to **Centered thickness**.

Component-code	Matter-coating	Predefined propagation	Centring thickness (Z)	Centered thickness
Wood dowel - D8 L30	-	Dowels	-	X
Simple cam - L34 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 and the shelf. Click on **Stop** to confirm the selection.
- From the **Automatic assembly** window, select **Rules** > **Dowels + Cams**.

Rules > Dowels + Cams

- Reselect the parts manually in the graphics area 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 validate.

Note: In the case of a closed assembly (a cabinet for example), selecting **Outside face** or **Inside face** allows you to adjust 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.
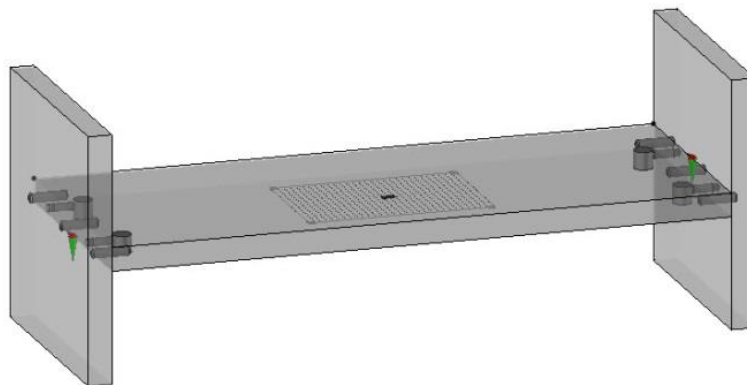
- Place the cam cases downwards.
- Validate the positionings with **OK** to perform the machinings.
- Save and close the file.

- Define the set:

  - **Designation**: *Fixed shelf component*
  - **Assembly nature**: **Content**

Creation of 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*
    - **Default housing mode = Housing**
    - **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.
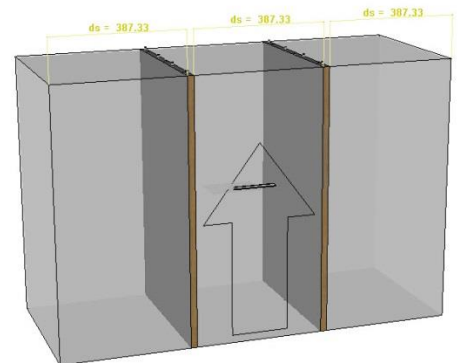
- Create a new parameter:

    - **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**.
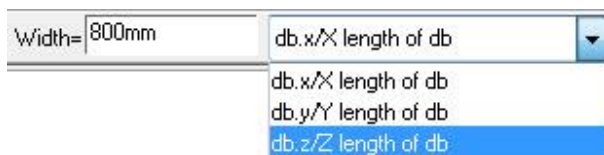
- Select for the parameter **Depth = db.y/Y length of db**.
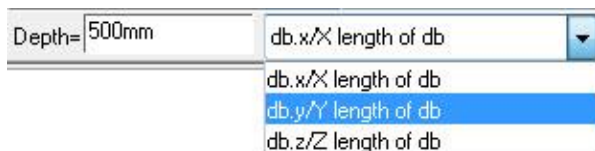
- Select for the parameter **Thickness = th/Panel thickness**.

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**.

OK Type= ALIGNMENT Distance= 0mm

- Then apply a third constraint:

    - **Origin**: Left-hand face of the shelf
    - **Destination**: Left-hand face of the driver block
    - **Type = Alignment**
    - **Distance** = *ds*

OK Type= ALIGNMENT Distance= ds

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

Distance per instance = ds+th

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

- Define the set:

    - **Designation**: *Vertical separation panel component*
    - **Assembly nature**: **Sub-assembly**

- Save and close the document.
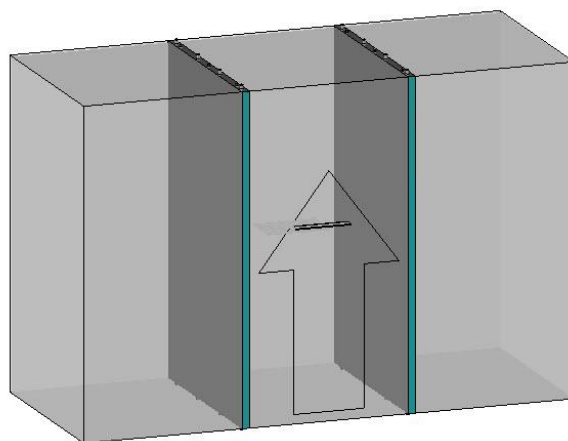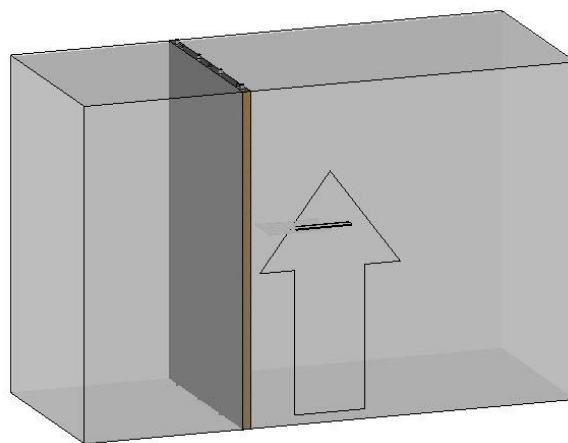
Create the Horizontal separation panel component

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 adjust 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*
    - **Assembly nature**: **Sub-assembly**

- Save this component in the **training library**:

    - **Family**: **Cabinets**
    - **Type**: **Separation panel**
    - **New variant**: *Distributed horizontal separation panels*
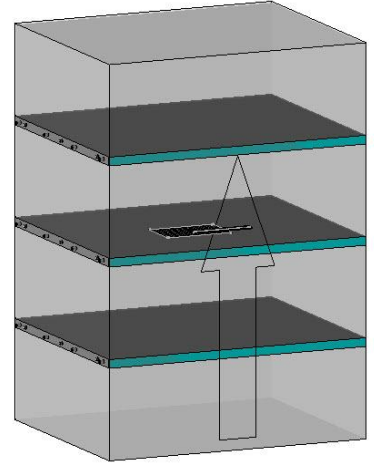
Using the separation panels

- Open the interior design file.
- Insert the **Distributed vertical separation panels** standard component in the two bottom cabinets.
- Enter **Number of separation panels** = *2.*
- Perform the machinings using the **Automatic** option.
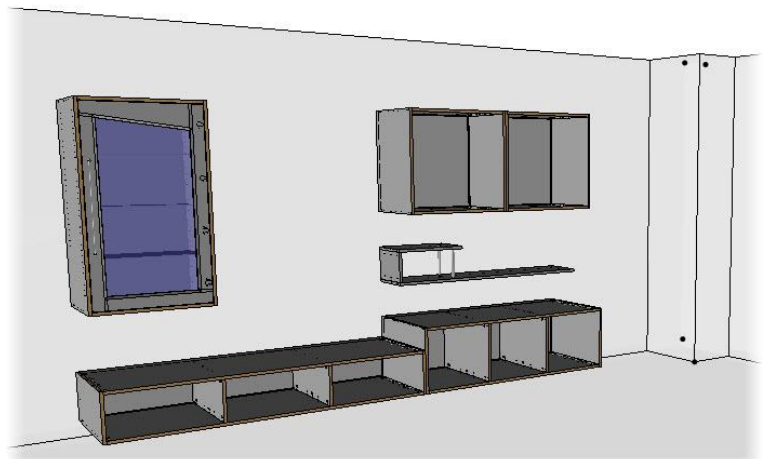
- Then insert the **Distributed horizontal separation panels** component in the middle compartment of the right-hand cabinet, as well as in the left-hand and right-hand compartments of the left-hand cabinet.
- Enter **Number of separation panels** = *1.*
- Perform the machinings using the **Automatic** option.
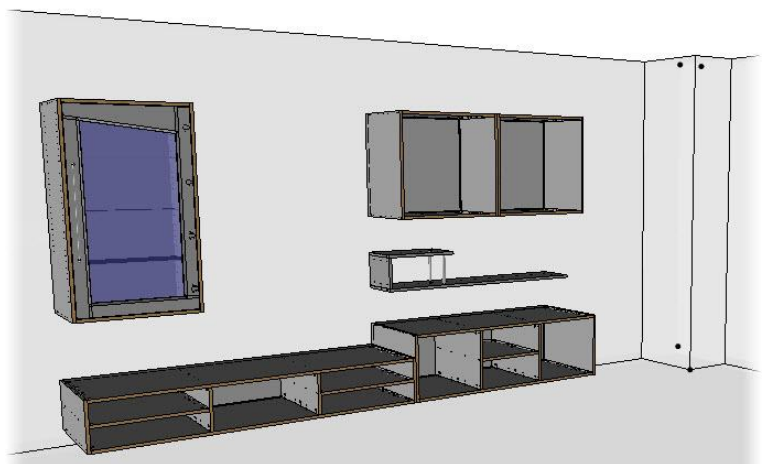
Note: A **Driver block** component can also be inserted outside a cabinet or a closed housing. If an open volume forms a rectangular parallelepiped, you can insert a **driver block** component in it.

- 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.
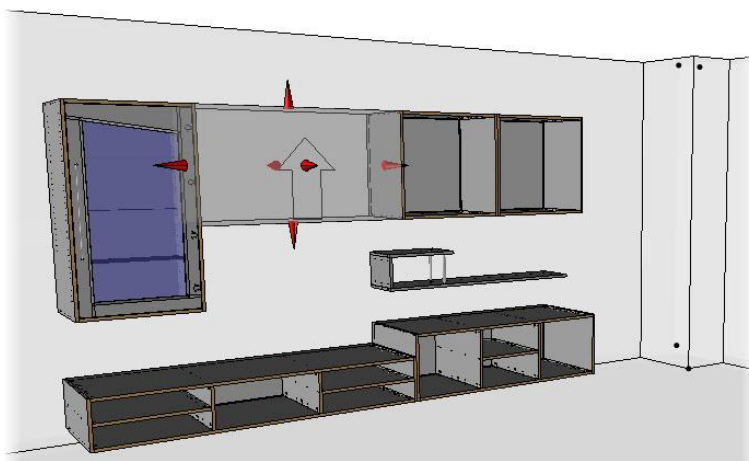
- Set **Offset** = *50mm* on the **front plane** arrow.
- Set 2 separation panels, and then click **OK** to validate.
- Perform the machinings using the **Automatic** option.
- Save the document.
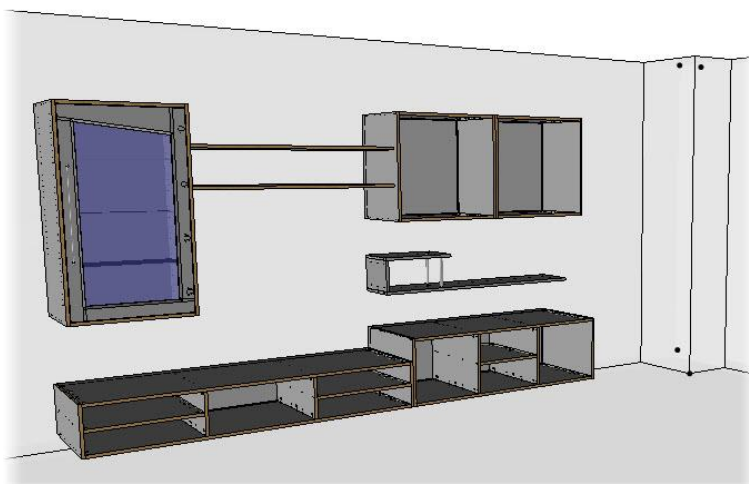
Exercise 9: Creation of a coffee table

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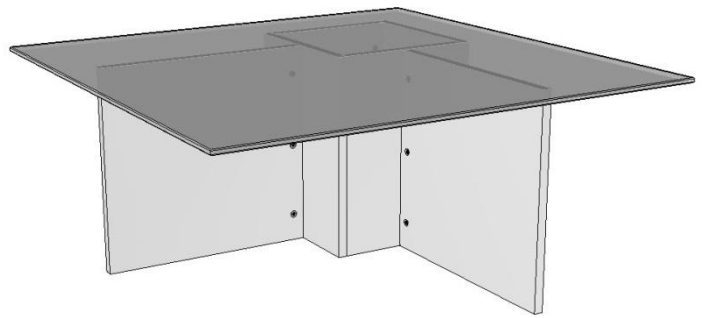
Creation of the table

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**.
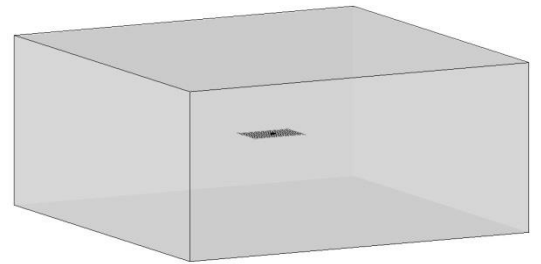
Create a second construction volume

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:

    - Enter **Offset from starting curve** = *-h/2*.
    - Select the **Z+ direction**.
    - Enter **Height** = *h-10*.

- Apply a **transparency** of **4** to this block.
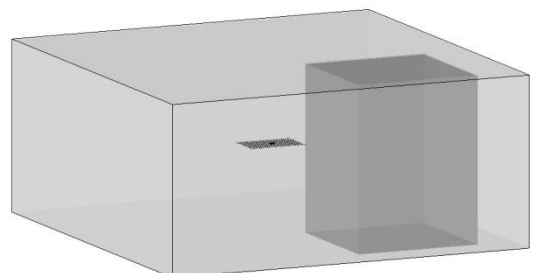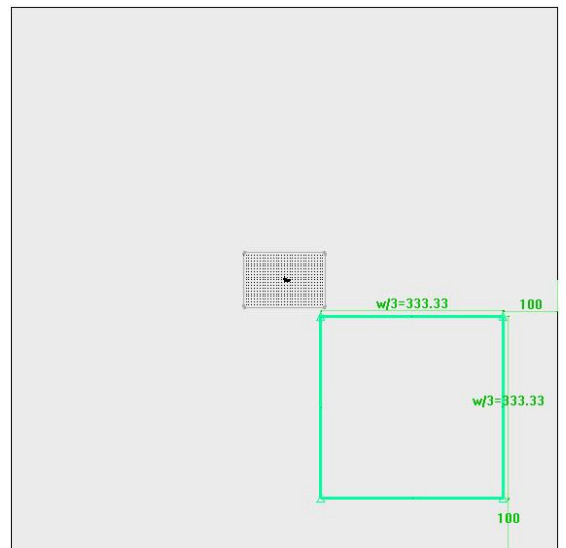
Create the parts

- Adjust:

  - The current material to **TopSolid'Wood** > **Panels** > **Colors** > **White**.
  - The current coating to **No coating**.

- 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*
  - Two covering sides and two covered sides

- Turn off **layer 0**.
- Create the other two parts making up the base as **constrained blocks**:

  - **Thickness** = *th*
  - **First shift** = *100mm*
  - **Positioning shift** = *w/6*

- Start the **Shape** > **Chamfer** function.
- Enter **Length** = *2mm* and select **Shape.**
- Select the glass table top to apply a chamfer onto all the shape's edges.
- Validate the chamfers with **Compute chamfer(s)**.
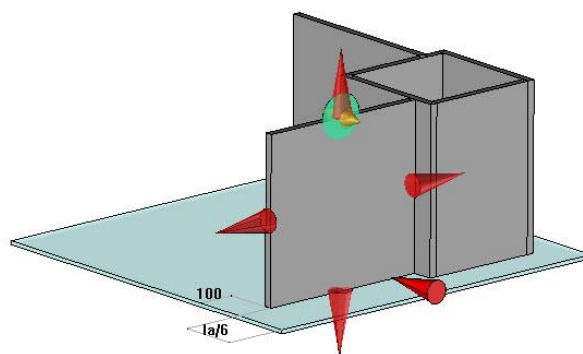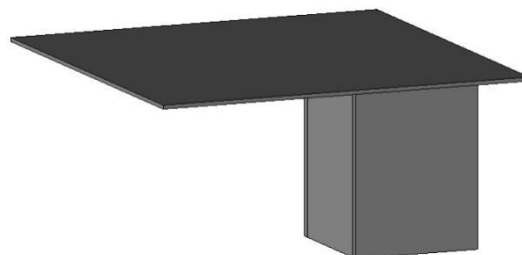
Saving and defining the table

Define the parts

- Define the parts of the base:

  - **Designation**: *Base 1-6*
  - **Material**: **TopSolid'Wood** > **Panels** > **Colors** > **White**

- Define the table top:

  - **Designation**: *Glass table top*
  - **Material**: **TopSolid'Wood** > **Glasses** > **Clear window glass**

- Define the set:

  - **Designation**: *Rectangular coffee table*
  - **Assembly nature**: **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.

- Enable the **Select by criterions** option.
- Then check the **Material** box and select **White**. Thus, all the **white** parts are selected.
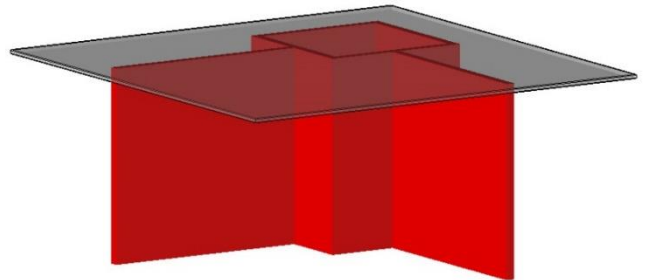
- Validate the selection with **OK**.
- Select:

  - 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.
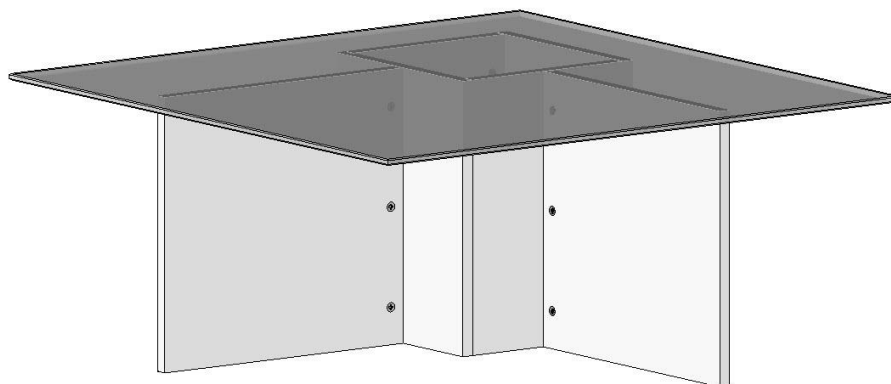
- Save and close this file.

Insertion of 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 floor.
- 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, and 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, and then left-click to validate the positioning.
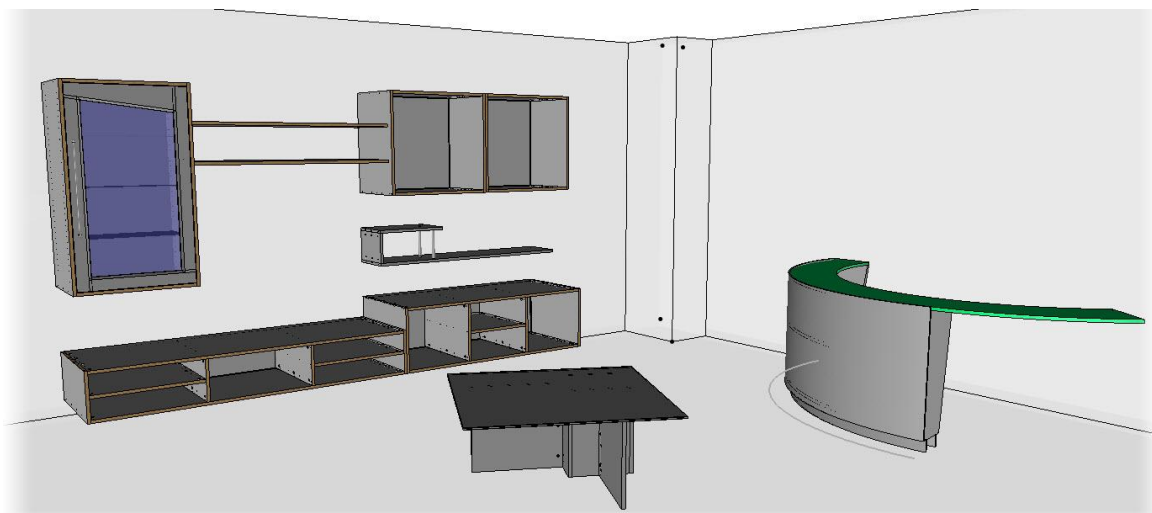
Exercise 10: Creation of an extruded component

> **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.
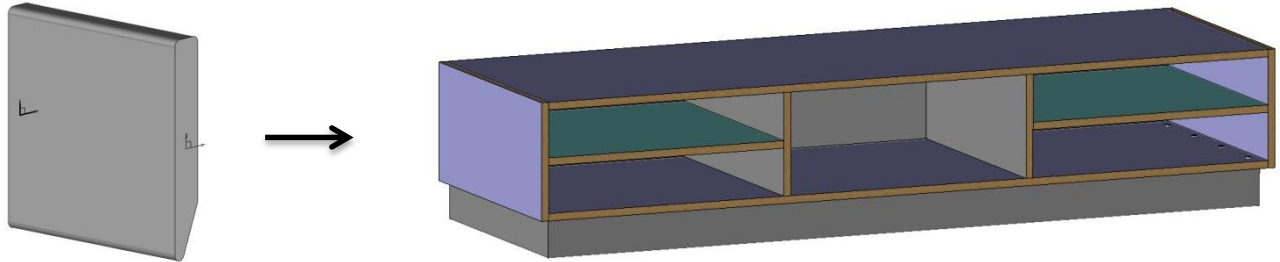
Creation of the extruded component

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** > **Wizard = Yes**, and then click any point to the right of the absolute coordinate system.
- Validate the coordinate system on point with **OK**.
- 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.
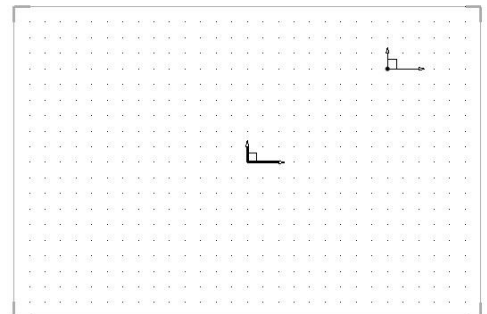
- Start the **Edit** > **Name** function and name this new coordinate system *fr1.*
- Leave the **Designation** field blank, and click on **OK** to confirm.

OK   Name: fr1   Designation:
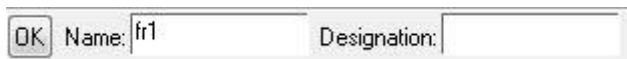
- Turn off **layer 0**.
- Create a length **parameter** with a **value** = *100mm,* a **name**: *l* and a **designation**: *Length.*

- Create a **coordinate system** using **Tools** > **Coordinate system** > **Wizard = Yes**.
- Select the **fr1** coordinate system in order to create a duplicated coordinate system.
- Select the red arrow in the Z+ direction in order to duplicate the coordinate system along a translation in the Z+ direction.
- Enter **Distance**: *l.*
- **Name** this coordinate system *fr2.*

OK   Name: fr2   Designation:
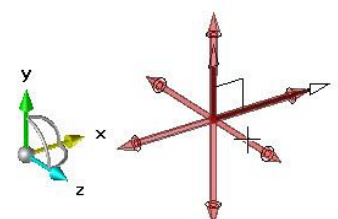
Create the profile

- 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 to a **height** = *l.*
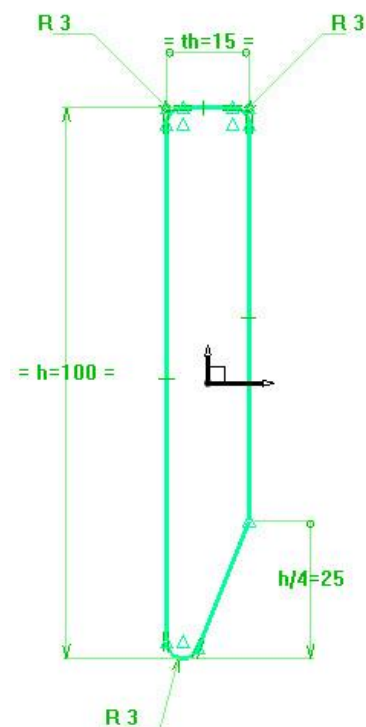
Create the key points

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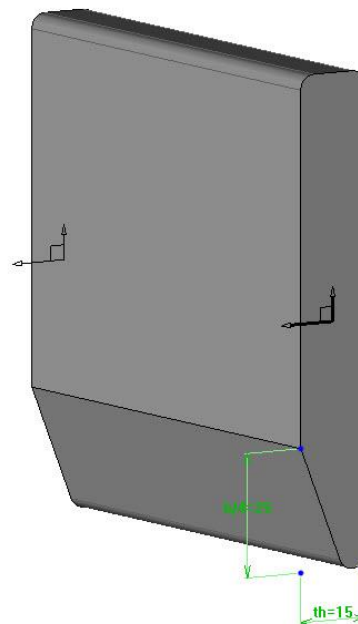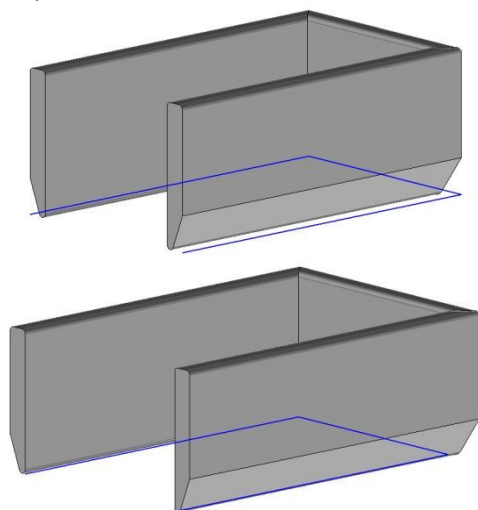
- Start the **Tools** > **Point** > **Offset point** function.
- Select the point shown opposite as the **origin point**.

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

- Select the **Y- direction** and enter **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 **Distance**: *th.*
- Position the dimension.
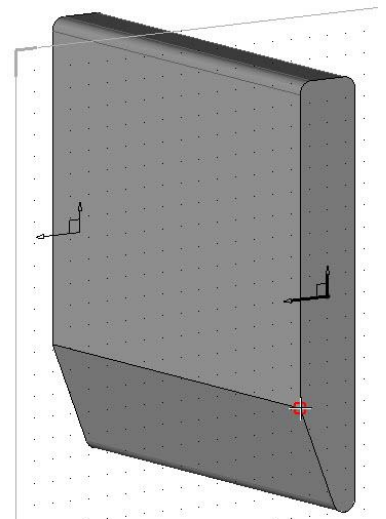
- Use **Assembly** > **Define component** > **Define key-points**.
- Select the left-hand **offset point** as the **key point or key coordinate system to insert**.
- Enter:

  - **Name**: *p1.*   Name of key point: |p1|

  - **Designation**: *Bottom left point*. |OK| Designation of key point: |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 **Name**: *p2* and **Designation**: *Bottom right point.*

> KEY-POINTS / KEY-COORDINATE SYSTEMS SET : (2)
> ⊞ Bottom left point (2)
> ⊞ Bottom right point (2)
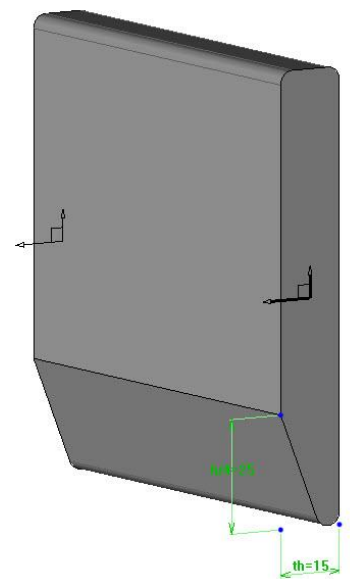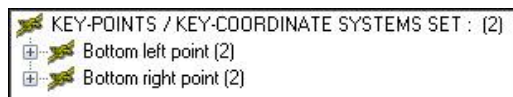
Define the component

- Define the part using the **Wood** > **Define** > **Define part** function:

  - **Designation**: *Baseboard type 1*
  - **Material**: **TopSolid'Wood** > **Panels** > **Colors** > **Olive**

- 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.

Use of the extruded component

Draw the contour

- Open the interior design file.
- Make **layer 14** current.
- Start a **new sketch**.
- Using the **Contour** function, draw a contour enclosing the cabinets on the left side, the front side and the right side as shown below.
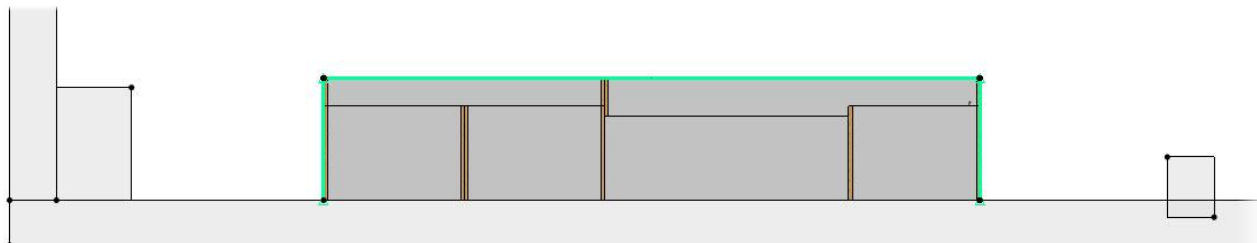
- Start the **Create offset profile** function in **Offset type = Profile** mode to draw the offset of the **curve** to a **distance** = *50mm.*

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 validate.

- End the sketch.

Insert the extruded component

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

Note: By default, an extruded component is positioned between two points. Once the extruded components have been generated, cuts can be performed 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. ON CURVES
- 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.

- Adjust the following parameters:

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

| OK | Key point= BOTTOM RIGHT POINT | Rotation angle: 0° | Cut= Mitre cut | ▼ |

- Click on **OK** to validate.

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 **Curve** > **Origin** function, select the sketch, invert the direction of the **red arrow**, then reinsert the extruded components.

Supplement: Creation of a catalog and addition of 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 drop-down list.

All parameters and texts   ▼

The catalog is then generated in Excel format.

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

- Fill in the catalog boxes as shown opposite.
- Delete the column of the **l** parameter.
- Reopen the interior design document.
- Reinsert the extruded components on the sketch by selecting the **code**: **100x15** when selecting the standard extruded component.

Code:
100x15   ▼

Add detailing baseboards

- Start a **new sketch**.
- Draw a contour on the wall around the part, as shown below.
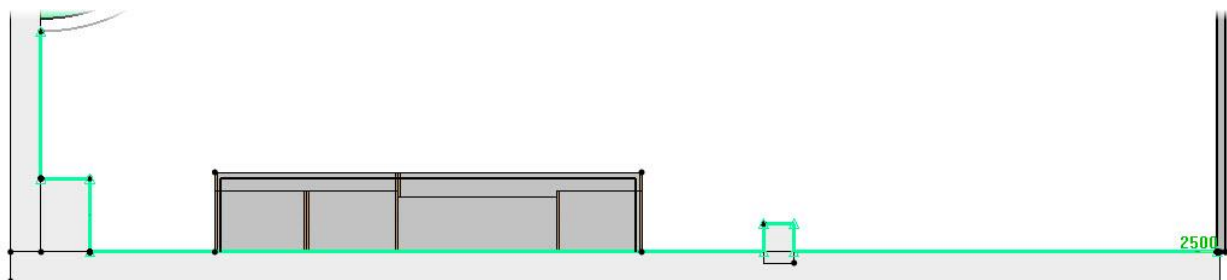
- End the sketch.
- Using the **Wood** > **Extruded component** function, insert the baseboard with the **code**: **80x10**.
- Select **On curves** and select the previous sketch.

- Adjust the following parameters:

  - **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**.
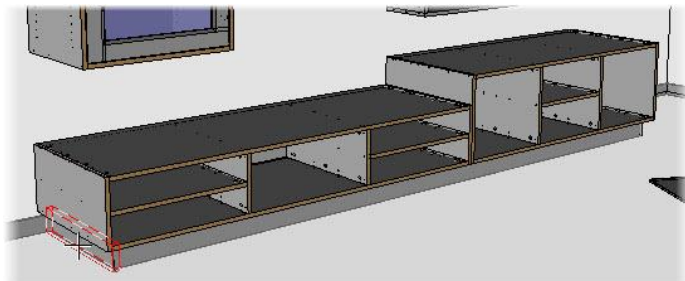
- Select **Main cut**. MAIN CUT
- Set **Offset**: 0mm, **Hide tools = No** and select the baseboard on the wall as the **tool shape to use**.

Offset: 0mm   Hide tools= NO   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.
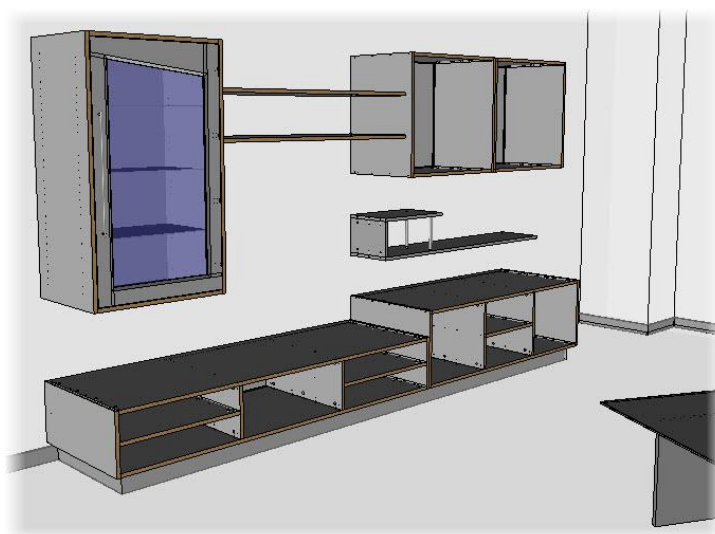
- Save the document.

Exercise 11: Creation of 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...

Creation of a draft template

Create the file

- Create a **new Draft document**.
- Select **Without template** and select **Paper format: A4H** in the drop-down 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 drop-down list.
- Enter **Number of rows** = *4* and **Number of columns** = *1.*
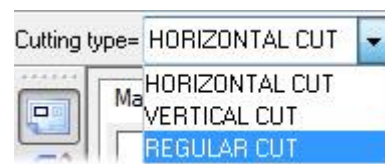
- Click on **OK** to validate and click inside the title block to cut it.
- Reselect **Cutting type = Regular cut** in the drop-down list.
- Enter **Number of rows** = *1* and **Number of columns** = *3,* then click on **OK** to confirm.
- In **Cell to cut**, select the second row from the bottom.

- 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.
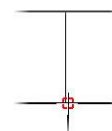
Insert text in the title block

- 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.*

Element to insert or text to create: Draft name

- Press **Enter** to validate and select **Fast choice** > **Center center** to center the   Fast choice > CENTER CENTER ▼
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.

Draft name

- Then select the right-hand cell of the same row.
- Select **Scaling factor** in the drop-down list. SCALING FACTOR

Note: This drop-down 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. BITMAP

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.

Use of 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 16** current and start a new sketch.
- Using the **rectangular contour**, draw a rectangle that encloses the interior design as shown below.
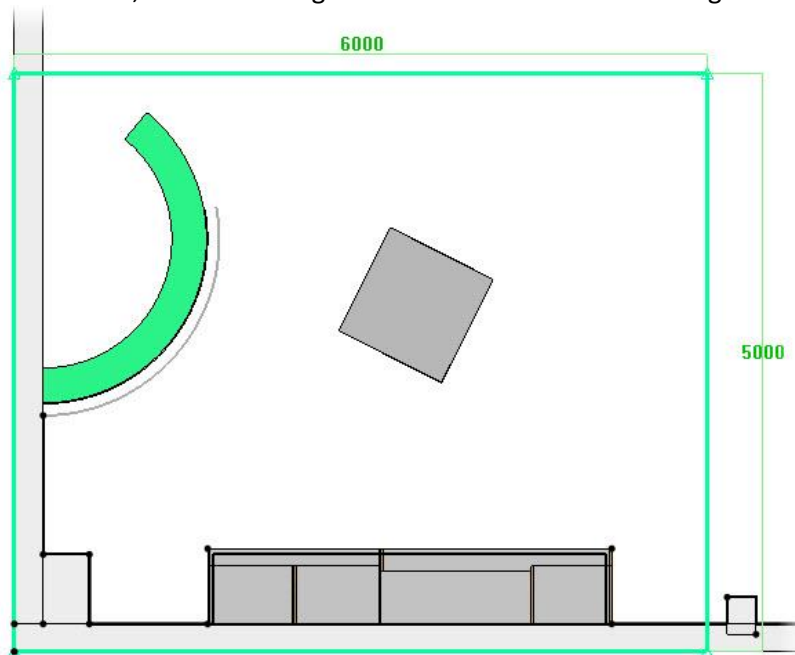
- End the sketch, then extrude it with an **offset from starting curve** = *-100mm* and to a **height** = *2600mm.*
- Apply a **transparency** of **10** to this block.
- Start the **Tools** > **Draft trimming volume** function.
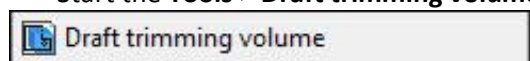
- Select the extruded volume as the **trimming volume for views**.
- **Name** this volume *tv* and click on **OK** to validate.
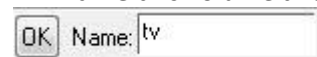
- Save the file.

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 validate.

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 drop-down list.

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

Smooth edges >	HIDDEN	▼
Hidden lines >	HIDDEN	▼
☑ Shading view		

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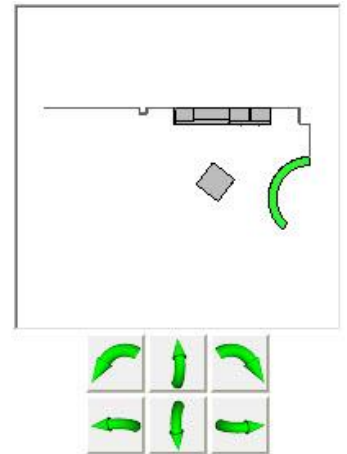
- At the bottom right of the window, select **Trimming volume** > **Volume**: **tv** in the drop-down list.
- Validate the window with **OK** to calculate the view.
- Place the view and set the **scaling factor relative to drawing** to *0.02.*

- 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 floor and the columns, then validate the selection with **OK**.
- **Regenerate** the view.
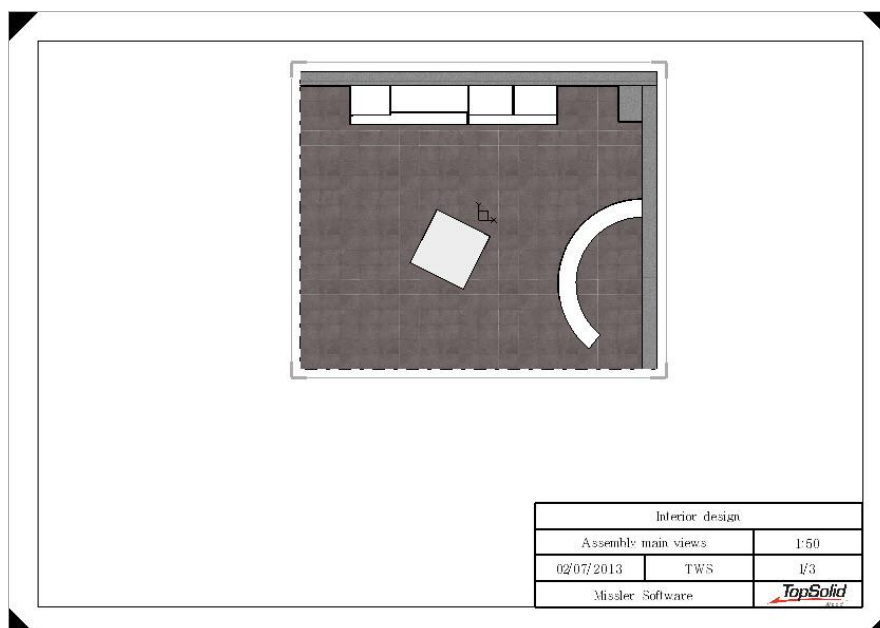
- Create the **full section view** as follows:

  - **Alignment = No**
  - **Set the section view upright = No**
  - **Hidden lines = Hidden**

`OK` Alignment= `NO` Set the section view upright= `NO` Hidden lines= `HIDDEN`   ▼

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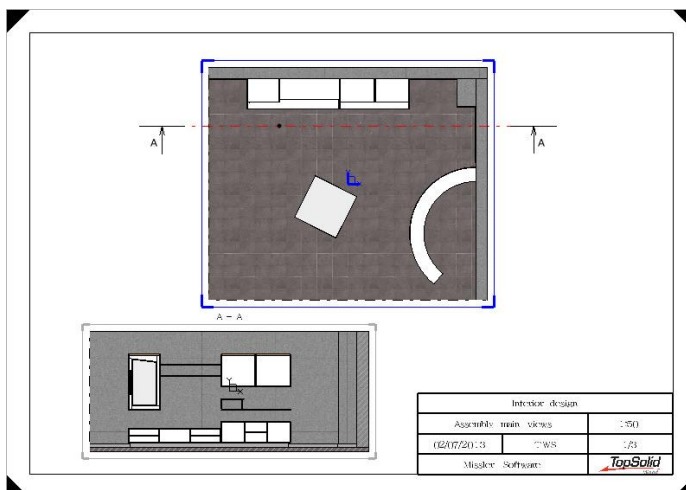
- **Save** this draft document in the *Interior design* folder and keep the default name.
- Use **Modify element** to modify the text of the title block: **Draft name**.
- In **Text**, enter *Assembly main views.*

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**. `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**.

Hook point= `BOTTOM-LEFT` 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. `STOP`
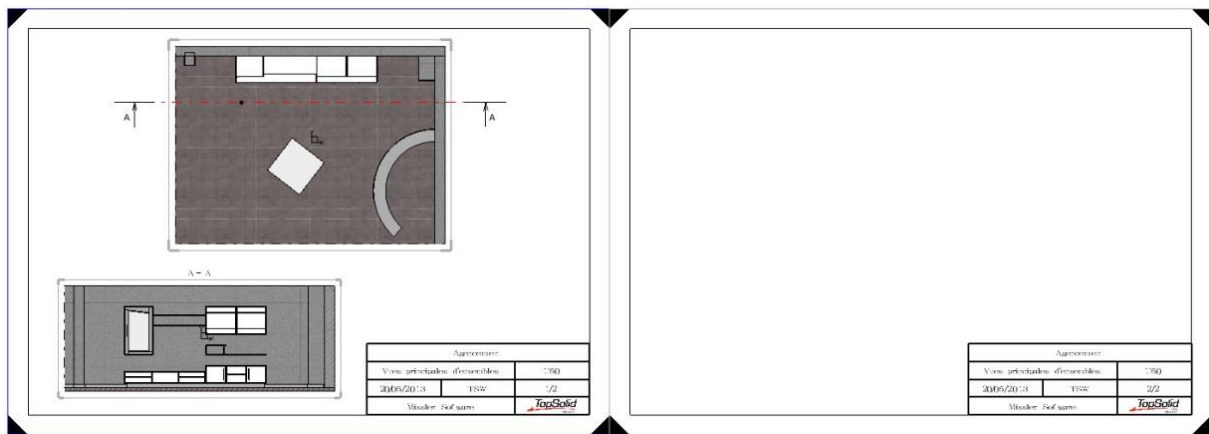
Create a perspective view

- Open the **Design** file of the interior design.
- Make **layer 15** current and switch off **layer 16**.
- Open the **View** tab [icon] and select **Visit mode**. [icon]

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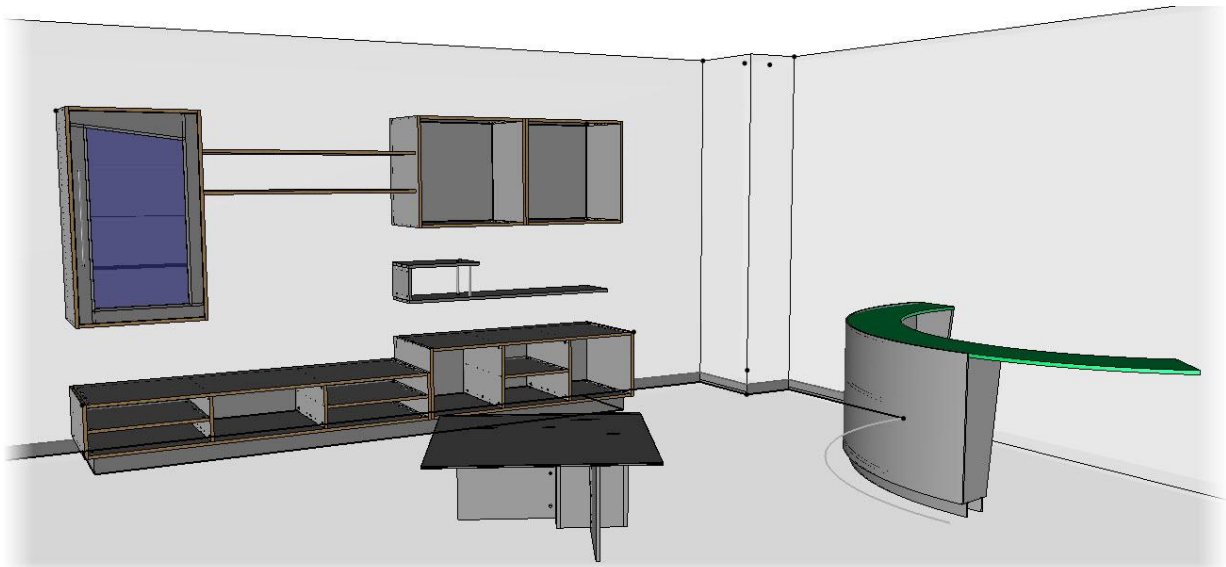
Note: To go back to the perspective view, open the **View** tab [icon] and select **Perspective view**. [icon]

- Select **Window** > **Tile vertically**.
- Click in the draft document to make it current.
- Create a **main view**. [icon]
- Select the copied left-hand drawing as the **drawing in which to work**.
- Select **Like a view** and select one of the two existing views.

Note: The **Like a view** function only allows you to create a main view from elements already projected in the selected view. This function does not copy the properties of the view (view orientation, visibility of edges, trimming volume, exceptions, etc.)

- 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.
- Enter **Scaling factor** = *0.06.*
- 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*.
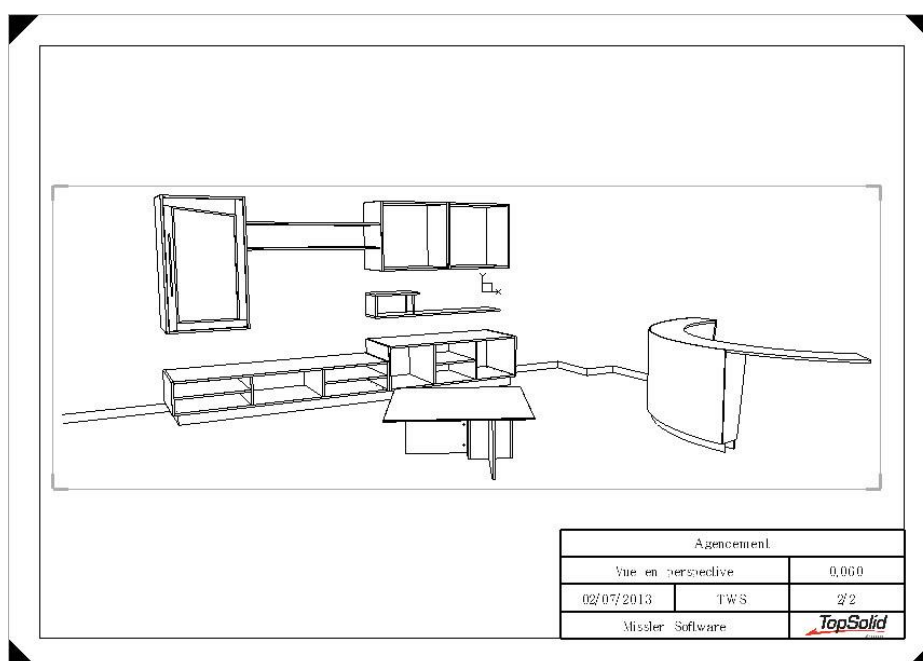
Exercise 12: Creation of a BOM template

> **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.

Creation of 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 the **Group** tab.
- Select **Create new bom file**.
- Enter **Name of new bom file**: *General part BOM.*
- Click on **OK** to validate.

What do you want do to
- Create new bom file
- Modify selected bom file
- Copy selected bom file in user configuration
- Copy selected bom file in group configuration

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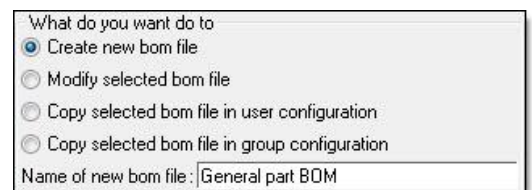
Note: The **Creation or modification of bom file** window opens. Each row corresponds to a different column of BOM information.

- Double-click in the **Definition** box on the first row to insert information.
- From the **Modification** window, select:

- **Defined modules** > **TopSolid'Design**
- **Defined functions** > **Index**

Modification

Title : INDEX
Modules
Defined modules > TopSolid'Design
Function
Defined functions > INDEX

Note: The BOM functions are categorized (**Defined modules**).
In addition, to find a function more easily in the list (**Index** for example), you can press the '**i**' key in order to scroll through the functions starting with '**i**'.

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

Creation or modification of bom file

Title	Definition	Type	Align...	Width	Format	Order num...	Bar c...
INDEX	INDEX	CHARACTER STRING	LEFT	15mm		1	
DESIGNATION	DESIGNATION	CHARACTER STRING	LEFT	15mm		2	
REFERENCE	REFERENCE	CHARACTER STRING	LEFT	15mm		3	

- Then insert the **Total count** function. Before validating the function, change the title to *QUANTITY*.

Modification

Title : QUANTITY
Modules
Defined modules > TopSolid'Design
Function
Defined functions > TOTAL COUNT

- Insert the functions **TopSolid'Design - Material and coating** > **Material** and **Coating**.
- Then insert the function **TopSolid'Design - Physical properties** > **Part length**.

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 enter the / symbol, press **Alt Gr + 6**.
  - *mm* displays the dimensions in millimeters.

Format	: 01|mm

- Validate the **function** and insert the **Part width** and **Part thickness** functions in the same way.
- Then insert the function **TopSolid'Wood** > **Four edges or laminates**.
- 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 **TopSolid'Wood** > **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.

Creation or modification of bom file

Title	Definition	Type	Align...	Width	Format	Order num...	Bar c...
INDEX	INDEX	CHARACTER STRING	LEFT	15mm		1	
DESIGNATION	DESIGNATION	CHARACTER STRING	LEFT	15mm		2	
REFERENCE	REFERENCE	CHARACTER STRING	LEFT	15mm		3	
QUANTITY	TOTAL COUNT	INTEGER	LEFT	15mm		4	
MATERIAL	MATERIAL	CHARACTER STRING	LEFT	15mm		5	
COATING	COATING	CHARACTER STRING	LEFT	15mm		6	
PART LENGTH	PART LENGTH	REAL	LEFT	15mm	01|mm	7	
PART WIDTH	PART WIDTH	REAL	LEFT	15mm	01|mm	8	
PART THICKNESS	PART THICKNE...	REAL	LEFT	15mm	01||mm	9	
4_PART_EDGES	EDGES	CHARACTER STRING	LEFT	15mm		10	
BARCODE	REFERENCE	CHARACTER STRING	LEFT	15mm		11	128
		CHARACTER STRING	LEFT	15mm			

OK   Cancel

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.

Paper format > A3V

- 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.
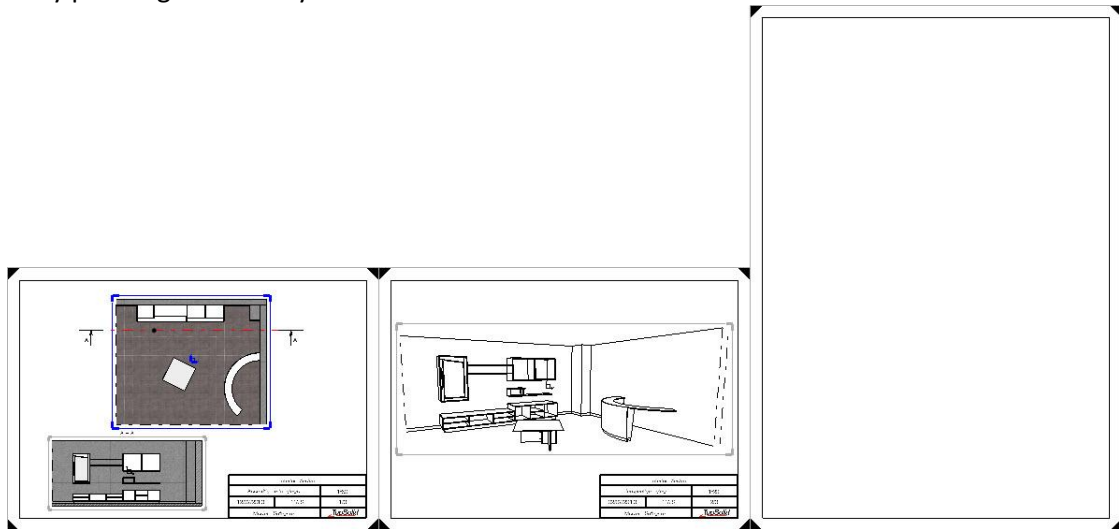
Create the BOM

- 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.

Choose a file

Standard	User	Group

General parts BOM
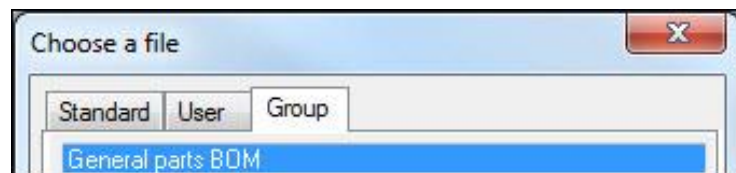
- Select the A3V drawing as the **drawing in which to work**.
- Select **Assembly** to include all the elements of an assembly in the BOM.   ASSEMBLY
- Select **Document containing the set = Interior design** in the drop-down list.

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.

- Adjust the following parameters:

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

Depth= MULTI LEVEL   ▾   Add a line for set= NO 🔩 Position of bill of material or title block:
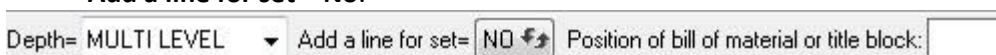
Note: A bill of material can be placed over a title block or between two points.

- In **Position of bill of material or title block**, select the bottom left point of the A3V drawing's frame, and 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

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

Note: The **Search for 3d part**: **YES** option is used to index 3D file elements.
The **One text** option displays the part index only.
The **Name** = *1* option is used to set the created index number. The next indexes will be incremented, based on this value.
The **>>** button allows you to 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 and 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 validate the window.
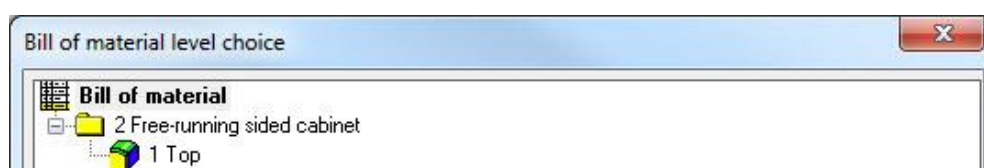
Exercise 13: Creation of a multi-draft

> **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.
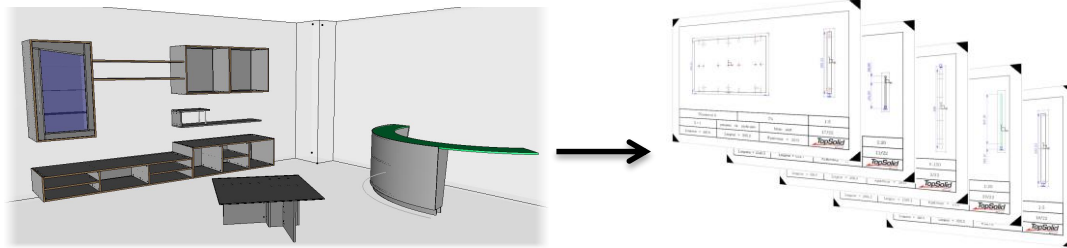
Creation of the multi-draft template

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 drop-down list.
- Then set:

  - **Width**: *210mm*
  - **Height**: *148.5mm*
  - **Name**: *A5H*

- Click on **OK** to validate.

Create the views

- From the **Wood** context , start the **Multi-draft** function.
- Select **Create template**, and then **Create view**.
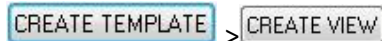
Note: **Create view** is used to position a view of the part or the assembly which will be drafted.
Three types of views are available:

  - **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 drop-down 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: You can adjust the dimensions of the view frames in order to obtain a better result when calculating the part views.

- Start the **Modify element** function.
- Select the angles of the frames corresponding to the two views and position them as shown opposite.
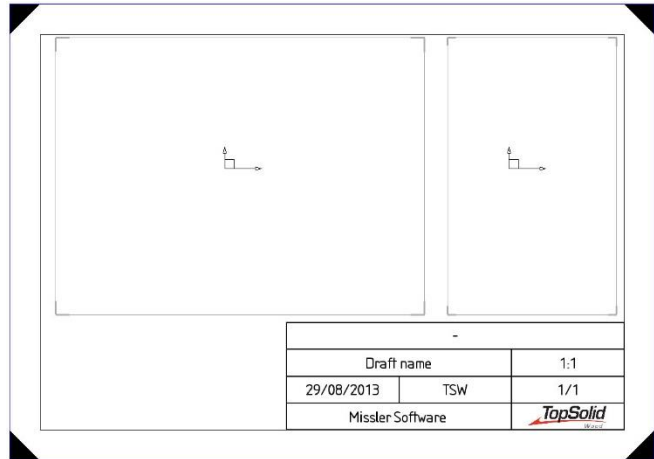
Set the template

- Open the **document properties**.
- Go to **TopSolid'Wood properties** > **Draft** to adjust the multi-draft.
- In **Elements to dimension**, check the following boxes:

  - **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 size of the view, the title block and the dimensions. A rounding value can be used to adjust the resulting scale factor in order not to get inconsistent scale factors (for example, 0.127 rounded to 0.1).
  - **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 defined in the list.

- Enable the **Free scale factor** option.
- Enter **Rounded value**: *0.1.*

- Check the **Scale factor given by extend of views** option to position the views in the previously adjusted frames.
- Validate the window with **OK**.
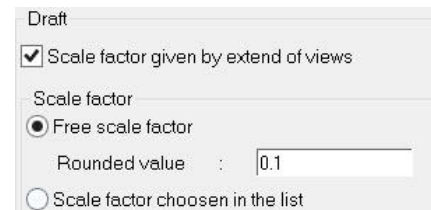
- Use the **Delete element** function to delete the text **Draft name**.

- Start the **Insert** function.
- Select the title block, and then the box in which the text has been deleted.
- From the drop-down list, select **Property** > **Reference**. Click on **OK** to validate.
- Select **No element**.
- In the positioning window, select **Center center** and click on **OK** to validate.
- Save and close this document.

Creation of the multi-draft

- Open the design document *Interior design*.

- From the **Wood** context, select the **Multi-draft** function.
- Set **Depth**: **Multi level** in order to view all parts of 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.
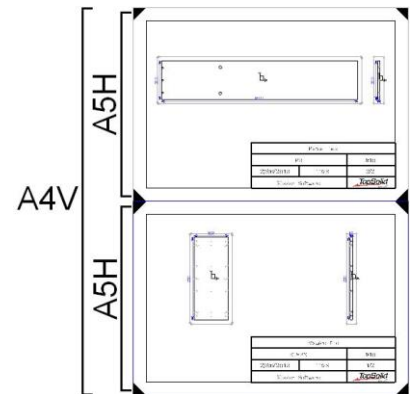
- Set **Put together all drafts in one document = YES** and select **Paper format = A4V**.

| Depth: MULTI LEVEL ▼ | Filter bom by criteria= no filter ▼ | Put together all drafts in one document= YES ⚡ | Paper format= A4V ▼ |

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

All parts made of particle board will be drafted.

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

| ☑ Material > particule board ▼ |

- Validate 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.

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

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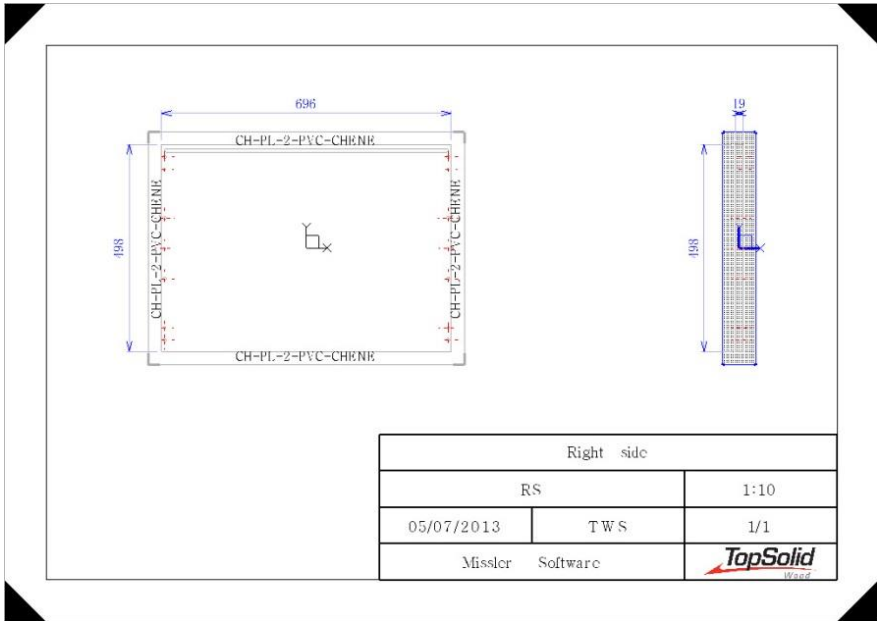
Note: The part drafts of a multi-draft behave in the same way as a simple draft. This makes it possible to modify, move and dimension the views, or modify the drawings.

Supplement: Insertion of 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 right-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.

- From 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 drop-down list. `BOM PROPERTY`

The different columns of the BOM are available.

- Select **Designation**. `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 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.
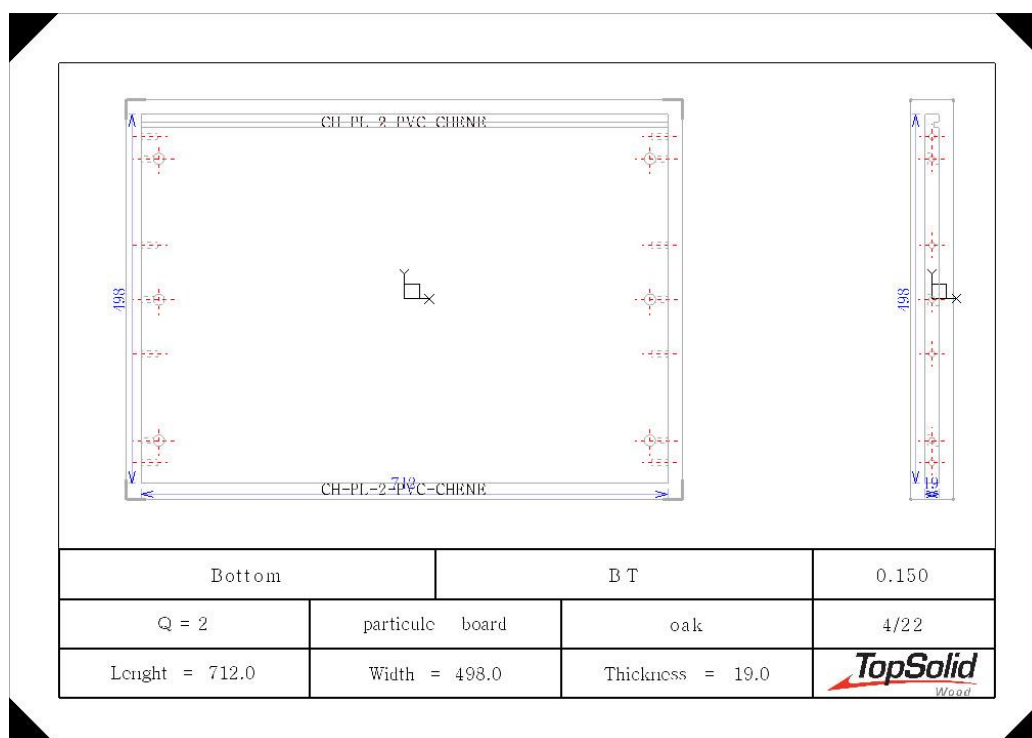
Exercise 14: Project configuration

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.
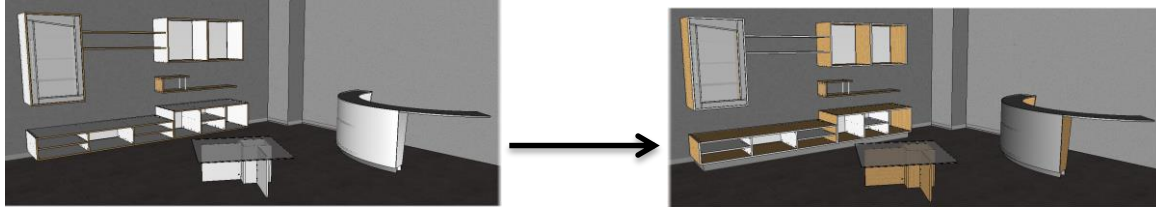
Note: Three different functions allow you to modify the elements inserted in a project:

- **Define multiple parts**: Used to modify all the characteristics of several parts, in place or belonging to

  components, in a single operation.

- **Edge configuration**: Used to modify the panel edges.

- **Laminate configuration**: Used to modify the panel laminates.

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.

- Open the project *Interior design.top*.

Configuration of components

- From the **Wood** context, start the **Define** > **Define multiple parts** function.
- Select **Main assembly** to take into account all the parts of the assembly in the configurator.

  MAIN ASSEMBLY

The multiple parts definition window opens.
The left pane of the window (the right pane is currently hidden) allows you to select the parts to be configured. Three selection modes are available:

- **Select manually**: The parts can be selected directly from the graphics area or from the bill of material with a **right-click** > **Select**.
- **Select by criterions**: This selection allows you to select parts according to one or several criteria, among those available in the software (**Type**, **Material**, **Coating**, **Thickness** and **Property**).
- **Select by properties**: This selection allows you to select parts according to one or several properties.

The bill of material of the multiple parts definition displays the rows for parts of different colors depending on the element:

Color	Type of element	Information
Blue	Part with cutting-up	
Yellow	Part without cutting-up	A part without cutting-up cannot be configured at the same time as a part with cutting-up.
Yellow	Assembly	An assembly cannot be configured at the same time as a part.
Green	Parts/Assemblies currently being configured	
Gray	Single unit assembly of one part	A single unit assembly of one part must be configured from its component part.

- Select the **Select by criterions** mode.
- Check the **Material** box and select **White** in the drop-down list
- Then check the **Property** box and select the **Designation** > **Fixed shelf** property.

- Select the **Configure** button to configure the selected parts. [Configure]

Note: When configuring parts via the **Define multiple parts** function, some properties can display **No modification**. This means that, for this property, the parts being configured do not have the same value.
It is then possible to:

- Leave the **No modification** value: Once changes have been made, each of the parts will keep their original value on this property.
- Replace the **No modification** value with another value: Once changes have been made, each part will take the new value applied to this property.

Once the parts have been configured, the right pane of the window appears.

- Open the **Material** tab.
- Select the material **TopSolid'Wood** > **Panels** > **Colors** > **Olive**.
- Select **Apply** to apply the modifications. [Apply]
- Close the right pane manually. [<<]

- From the graphics area, select the tops of the two lower cabinets manually as shown below.

- Select the **Configure** button. `Configure`
- Open the **Material** tab.
- Select the material **TopSolid'Wood** > **Panels** > **Woods** > **Panel beech hearted**.
- Select the **Apply** button to apply the modifications. `Apply`

Modifying the material of undefined elements

Note: The undefined elements such as the floor and walls cannot be modified via the **Define multiple parts** function.

- Start the **Attribute** > **Material** function.
- Select the **Matter Coating Finishing** button, and then the **Material list** button.

MATTER COATING FINISHING > MATERIAL LIST

- From the **Material** tab, select the material **TopSolid'Wood** > **Building** > **Tiles** > **Tile 30 rectangular**.
- From the graphics area, select the floor to apply the material onto it.

- Then select the material **TopSolid'Wood** > **Building** > **Concretes** > **Concrete 05** and apply it to the poles in the interior design.

- Validate the window with **OK**.
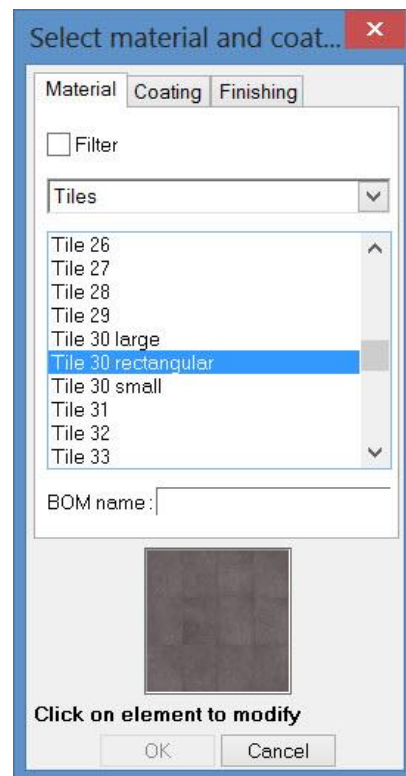
Notes

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	Somewhat no	Somewhat yes	Yes
Does the overall duration of the course seem appropriate?	☐	☐	☐	☐
If no, was it?	Too short ☐		Too long ☐	

PACE	No	Somewhat no	Somewhat yes	Yes
Does the overall pace of the course seem appropriate?	☐	☐	☐	☐
If no, was it?	Too slow ☐		Too fast ☐	

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

Comments and suggestions: ....

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