TopSolid Wood

Training Guide

TopSolid'Wood Basics

MASTER YOUR MANUFACTURING PROCESS

7 Rue du Bois Sauvage
F-91055 Evry, France
Web: www.topsolid.com
E-mail: info@topsolid.com

Version 6.14 Rev. 01

Technical Support:
E-mail: sup.wood@topsolid.com

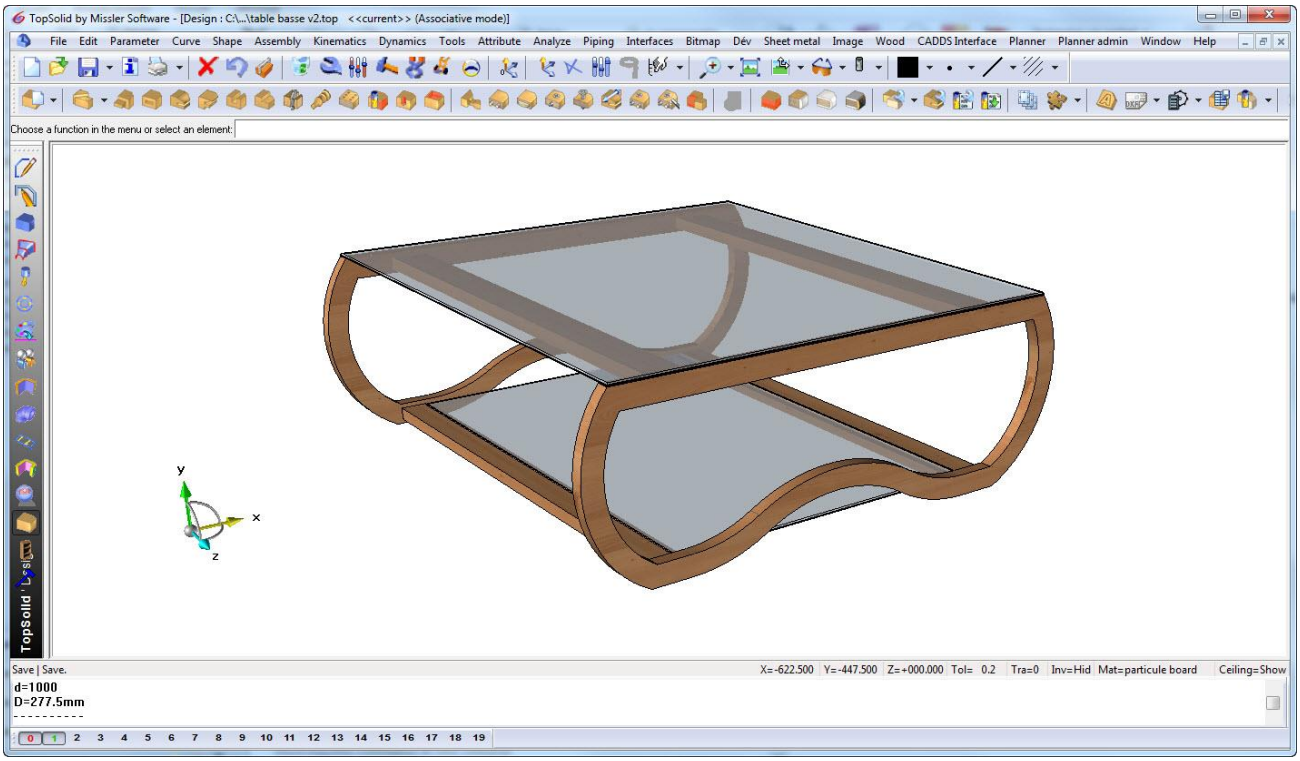
TopSolid'Wood Hotline:
05.61.39.95.95

Contents

Introduction to TopSolid

The purpose of this first section is to understand how TopSolid works and to discover the main methods for using the software.

General workspace



- **The menu bar**

Use the menu bar to navigate between the various menus in TopSolid.

- **The system bar**

The system bar contains the basic functions of TopSolid:

-  **New document**: create a new design or draft document.
-  **Open document** : open TopSolid files and formats compatible with TopSolid.
-  **Save/Save as**: save the current file.
-  **Cancel**: cancel the current function.
-  **Undo**: undo the last action of the current function.
-  **Delete**: delete an item.
-  **Edit**: make changes to an item.

- **The function bar**

The function bar contains the icons of the context selected in the context bar.

- **The context bar**

The context bar (opposite) is used to navigate between the different contexts in order to display the icons in the function bar.

- **Icons**

The system and function bars used two types of icons:

- **Simple icons**: execute a function when they are clicked.

 New File simple icon

- **Drop-down icons**: execute a function when they are clicked and also propose other associated functions when the black icon is clicked or by right-clicking on the icon.

 List icon for printing

Click on the black icon to open a list of associated functions.

Note: Once the function has been selected in the list, it becomes the main function.

- **The dialog bar**

The dialog bar is activated when the user starts a function.
It is used for communications from the user to the software and is read from left to right.

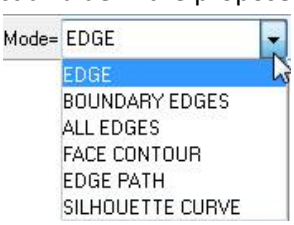
There are four types of dialog buttons:

- **Rotary buttons**: use to switch from one status to another, while staying in the same dialog.

- **Confirmation buttons**: use to confirm the dialog and to progress to the next dialog or open a subdialog.

- **Data entry buttons**: use to enter a value or to select a graphical item.

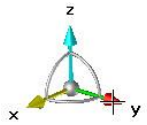
- **Drop down buttons**: use to select a value in the proposed list.



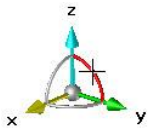
- **Compass**

The different parts of the compass are used to navigate in the graphical zone.

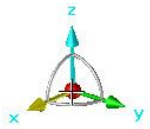
- Rotation around a point: rotate around a point by clicking and dragging one of the arrows.



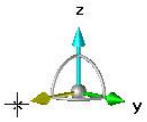
- Rotation around an axis: rotate around an axis by clicking and dragging one of the quarter circles.



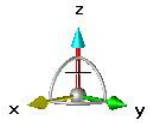
- Moving the compass: click and drag on the center of the compass to move it.



- Set the orientation of the view along an axis: left-click on an axis to orientate the view according to this axis.



- Move the view: click and drag on an axis of the compass to move the view.



- **The alpha bar**

Unlike the dialog bar, the alpha bar is used to provide information to the user.
It sends explanations to the user or errors in the current function.

Right-click inside the alpha bar to change the number of lines displayed.

Using the mouse

- **Left-click**: select an item.
- **Thumbwheel**: zoom in and out in the graphical zone.
- **Click-and-drag plus thumbwheel**: move the view.
- **Right-click**: validate the first button on the left of the dialog bar.

Keyboard shortcuts

- **Ctrl + left-click and drag**: rotate around a point (center of the screen).
- **Shift + left-click and drag**: move the view.
- **Esc**: exit the current function.
- **²** : undo the last action of the current function.
- **F1** : open the online help for the current function.
- **F2** : start the item analysis function in order to obtain information about an item in the graphical zone.

Exercise 1: 2D sketch

The goal of this exercise is to make the component parts of the lamp.

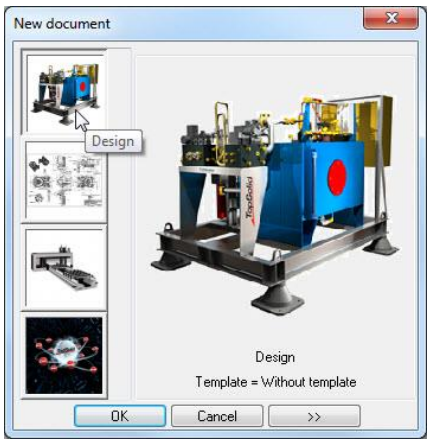
Concepts addressed:

- Creating a sketch
- Dimensioning a sketch
- Constraining a sketch
- Extruding a part
- Turning a part

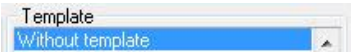
Lamp support

The 2D sketch is used to draw a part in 2D in order to then produce the 3D.

Create a new document



- Create a **new document**. 
- Select a **Design** document type.
  In the **Advanced parameters**, select **Without template**.



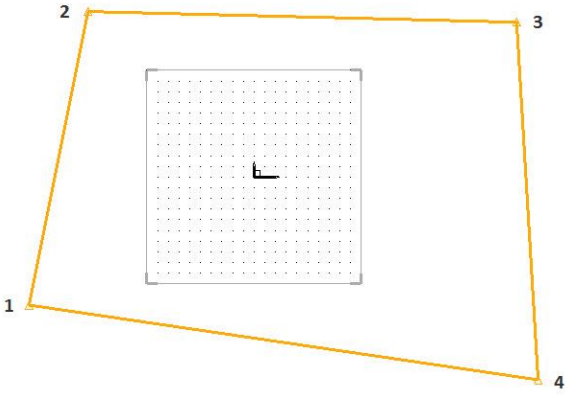
- Select the **Associative** design mode and then **Millimeters**.

- Click on **OK** to confirm.

- In the context bar, activate the **Sketch** context.
- Start a **new sketch** in the function bar. 

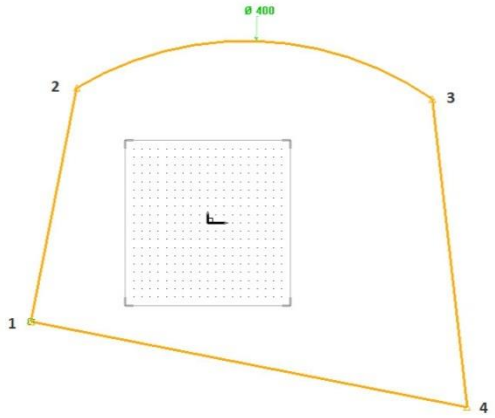
Note: A green frame appears around the work area when the sketch mode is active.

- Create a four-segment **contour** around the absolute coordinate system. 
- Select one of the segments to close the contour.



The sketch turns orange once the contour is finished.
This means that no constraints have been applied yet.

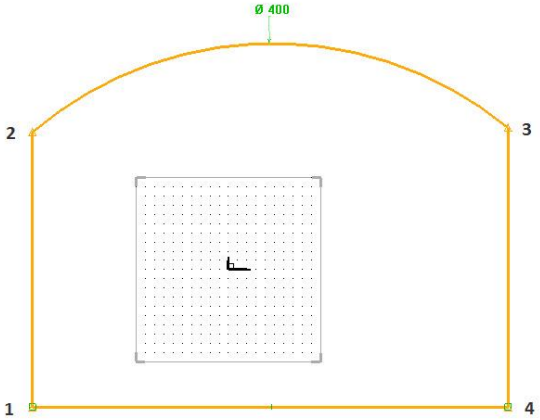
Create a sketch



- **Delete** the segment of the sketch [2;3].
- Start the **Circle** function.
- Set a 400mm **diameter** in the dialog bar.

- Then select points (2) and (3) in **Passing point** mode.
- If the arc of the circle is positioned on the wrong side, select **Invert** in the dialog bar to invert it.

Constrain the sketch



- Start the **Constraint** function.
- Use the **Perpendicularity** constraint.

Apply the constraint between the segments [1;2] and [1;4], and then between the segments [1;4] and [3;4].

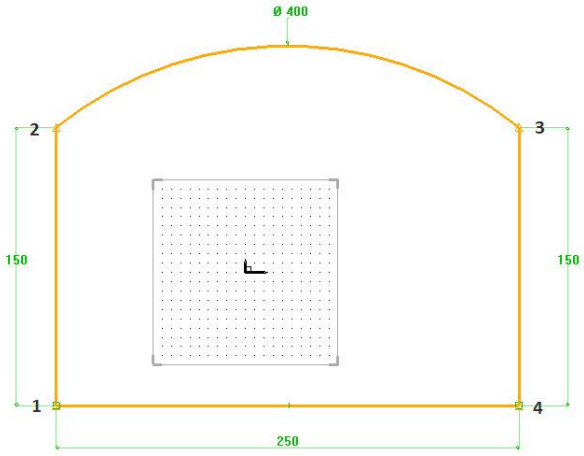
- Use the **Orientation** constraint.

Apply the constraint to segment [1;4] and select **Along X** in the dialog bar.

Note: The constraints are represented by green symbols on the sketch. Delete this symbol in order to remove the constraint.

- Perpendicularity constraint:
- Orientation constraint on the X axis:

Dimension the sketch



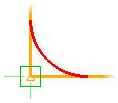
- Start the **Dimension** function.
- Select segment [1;2], position the dimension and set the nominal value to 150mm in the dialog bar.

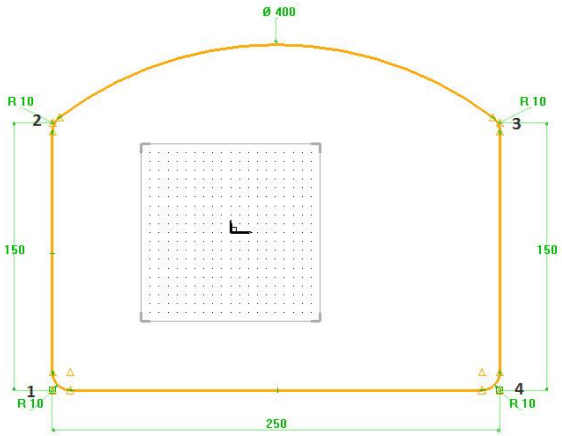
- Repeat the same operation for segment [3;4] with 150mm, and for segment [4;1] with 200mm.
- Use **Modify parameter** to change the value of a dimension.
- Select the dimension positioned on segment [4;1] and change its nominal value to 250mm.

Make Fillets

- Start the **Fillet** function. 
- Enter a **fillet radius** of 10mm, then select the four corners of the contour in **Curve to modify**.

Note: The fillet previews are shown in red.



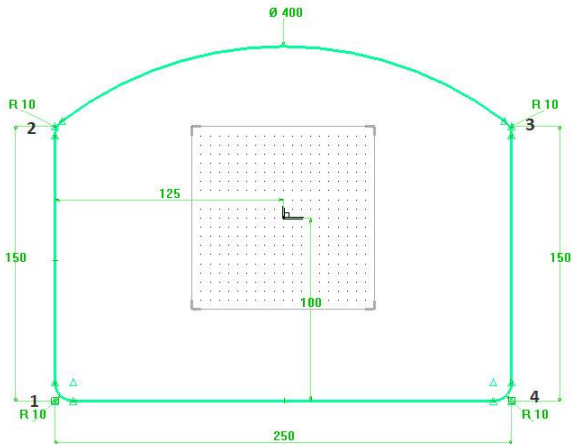


- Create the fillets by selecting **Compute fillet(s)** in the dialog bar.

Dimension the sketch on the absolute coordinate system

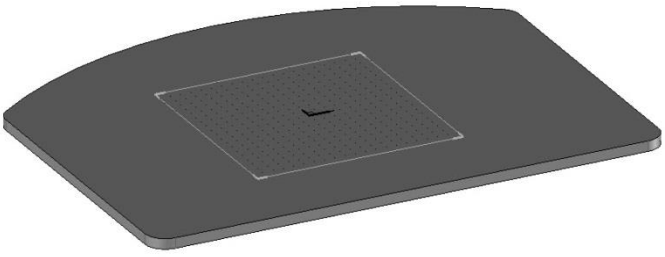
- Start the **Dimension** function. 
- Select segment [1;4], then the X axis of the absolute coordinate system. Enter a nominal value of 100mm.
- Then dimension the segment [1;2] with the Y axis of the absolute coordinate system with a value of 125mm.

Note: Once one or more segments are totally constrained, they turn green.



- Finish the sketch with the **End sketch** function. 

- In the context bar, activate the **Shapes** context.
- Start the **Create extruded shape** function. 
- Select the 2D sketch created previously, enter a height of 5mm in the dialog bar, then press **Enter** to confirm.



- **Save** the file by clicking on the disk icon. 
- Answer **No** to the request for a part definition. This point will be covered later on.
- Create a new folder called *Lamp* and rename the file *Base*.

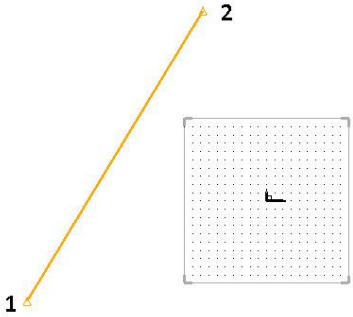
Light shade

Create new document

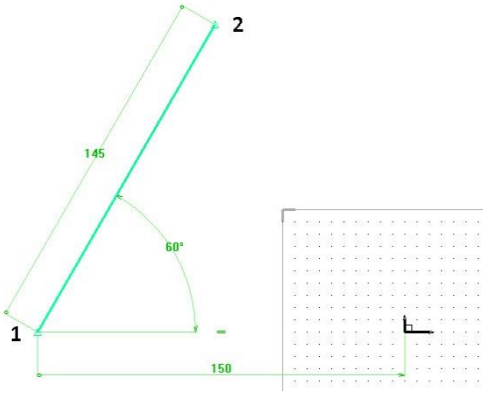
- Create a **new document**.
- Select a **Design** type document.
  In the **Advanced parameters**, select **Without template**.
- Click on **OK** to confirm.
- In the context bar, activate the **Sketch** context. 
- Start a **new sketch** in the function bar. 

Build a line

- Start the **Line** function. 
- Draw a line, as shown opposite.
- Start the **Constraint** function. 
- Use the **Alignment** constraint. 
  Click on the point (1), then the X axis of the absolute coordinate system.
  Click on **Stop** in the dialog bar.



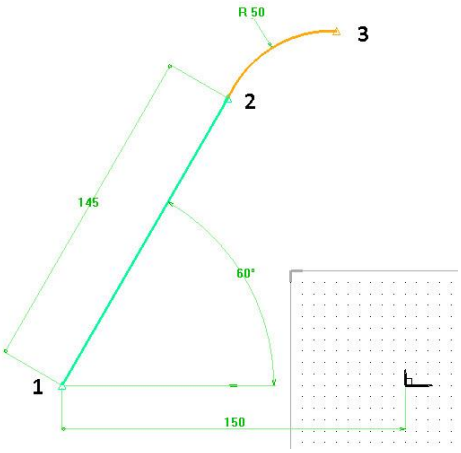
- Start the **Dimension** function. 
- Click on the point (1), then the Y axis of the absolute coordinate system. Position the dimension and enter a nominal value of 150mm.
- Then click on the segment [1;2], position the dimension and enter a nominal value of 145mm.
- Click on the segment [1;2], then on the X axis of the absolute coordinate system to create an angle dimension. Position the dimension and enter a value of 60°.



Create a circle arc

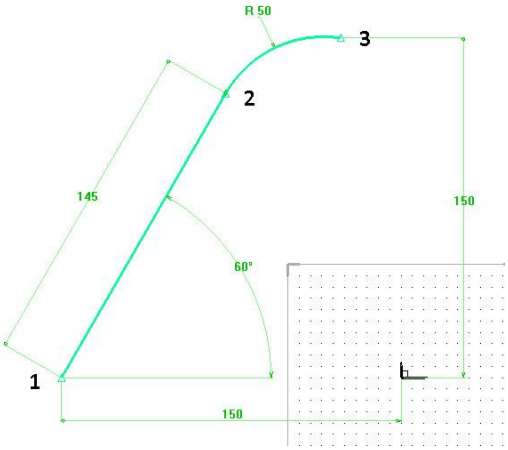
- Start the **Circle** function.
- Enter a **radius** of 50mm.
- In the **Passing point** mode, select the point (2), then a second point on its right.

- If the arc of the circle is positioned on the wrong side, click on **Invert** in the dialog bar.



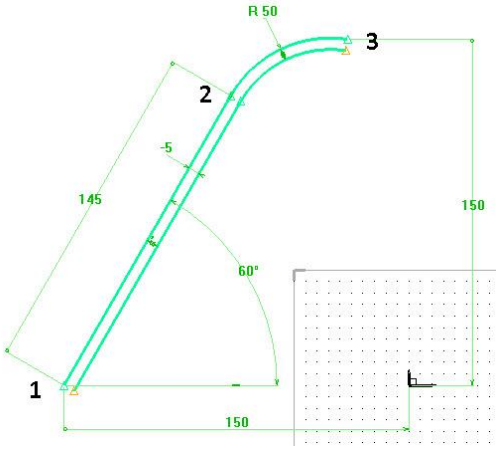
Note: If the segments are drawn using the **Contour** function, it is possible to directly draw a **tangent arc** with the **Link = Tangent** function, using the keyboard shortcut **A**.

Constrain the circle arc



- Start the **Constraint** function. 
- Use the **Tangency** constraint. 
  Select the segment [1;2], then the circle arc 2;3.
  These two entities are now tangential.
- Start the **Dimension**  function and dimension the point (3) relative to the X axis at a distance of 150mm.

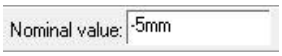
Create an offset



- Start the **Offset** function.
- Set the **Offset type = Profile** option, then select the sketch already drawn.

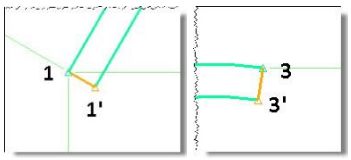
- Change to **Mode = One side** and enter a value of 5mm.

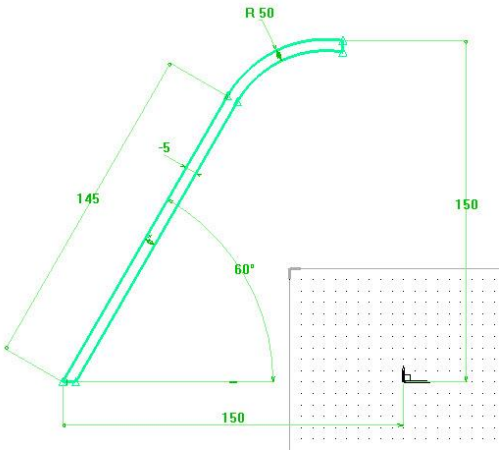
- Press **Enter** to confirm.
- Position the distance dimension of the offset.
- If the offset is on the outside, use **Modify parameter**  to change the **nominal value** of the dimension to -5mm.



Close the contour

- To close the contour, create two lines to close the sketch with the **Line**  function between points 1 and 1', then between points 3 and 3'.





- Start the **Constraint** function. 
- Apply an **orientation constraint**  to the segment [1;1'] along X.
- Apply an **orientation constraint** to the segment [3;3'] along Y.

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

- Finish the sketch with the **End sketch** function. 

Turn the part

- In the context bar, activate the **Shapes** context. 
- Start the **Create turned shape** function. 
- In the dialog bar, set **Type = Solid** and **Generatrix sketch = Global**.
- Select the previously created sketch in the **Section curves or texts** field.

- In the dialog bar, select the **Y+** axis as the **axis of revolution** to generate the part around this axis.

- In the dialog bar, set **Alignment = Normal**, **Generatrix = Hidden** and **Angle = 360°**. Finish by clicking on **OK** to confirm.

- **Save** the file by clicking on the disk icon.
- Answer **No** to the part definition window.
- Save the file in the *Lamp* folder and rename it *Light shade*.

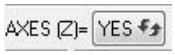
Body of the lamp

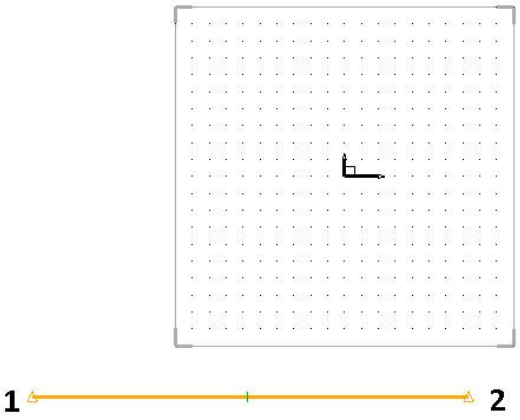
Create a new document

- Create a **new document** of the **Design** type.
  In the **Advanced parameters**, select **Without template**.
- Click on **OK**.
- In the context bar, activate the **Sketch** context. 
- Start a **new sketch** in the function bar. 

Draw a line

- Start the **Line** function. 
- Click on the first point in the line under the absolute coordinate system.
- In the dialog bar, set **Axes (Z) = YES**.



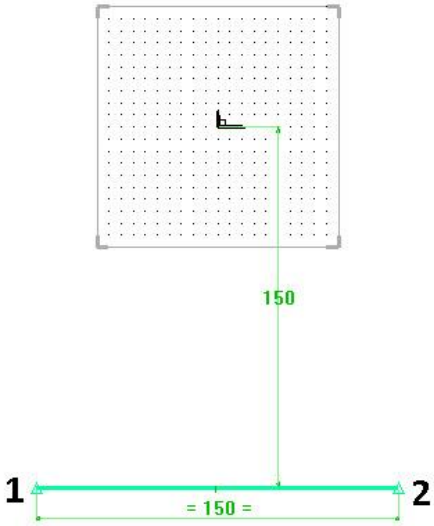


Note: This function is used to automatically draw lines parallel to the X and Y axes.
Consequently, the lines are automatically **constrained** by **orientation according to X** or **according to Y**.

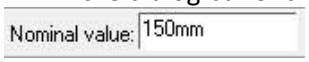
Note: This function can be quickly switched on and off by pressing **Z** on the keyboard.

- Click on a second point to the right of the first one.

Dimension the line



- Start the **Dimension** function and dimension the segment [1;2].
- In the dialog bar of the dimension, enter a nominal value of 150mm.



- Then activate the **Symmetry constraint** option and select **Y**.

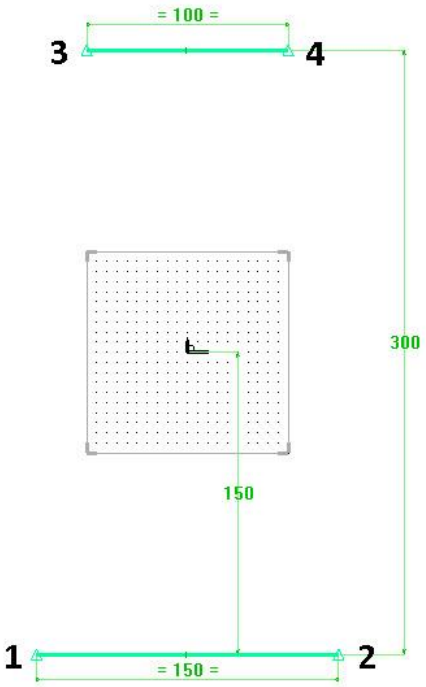
Note: Inserting a **symmetry constraint** on the **Y** axis centers the dimension in relation to the Y axis, irrespective of its length.

Note: When the dimension is constrained on an axis, it is displayed between two '=' signs.

- **Dimension** the segment [1;2] in relation to the X axis, with a value of 150mm.

Build a second line

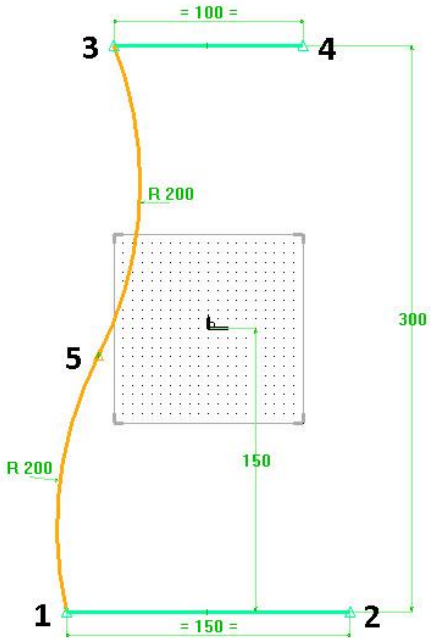
Repeat the same operations to produce the segment [3;4] shown opposite.



- Draw the line [3;4] with **Axes (Z) = YES**. 
- **Dimension** this segment with a **nominal value** of 100mm. 
- Apply a **symmetry constraint** on **Y** to the dimension.

- **Dimension** the segment [3;4] relative to the segment [1;2] with a **nominal value** of 300mm. 

Draw the circle arcs

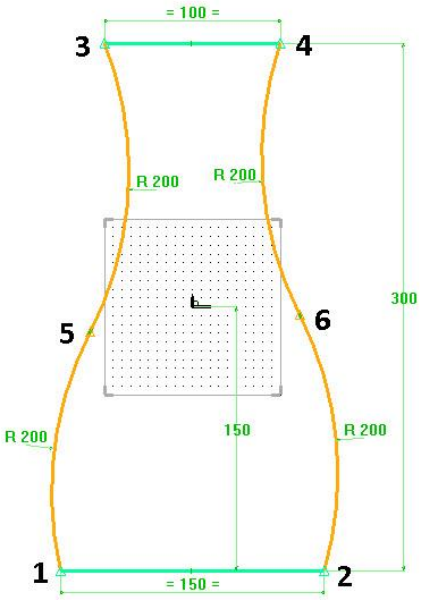


- Start the **Circle** function.
- Enter a **radius** of 200mm and select point (1) in **Passing point** mode, then any point between point (1) and point (3).

- Click on **Invert** to invert the side of the arc.

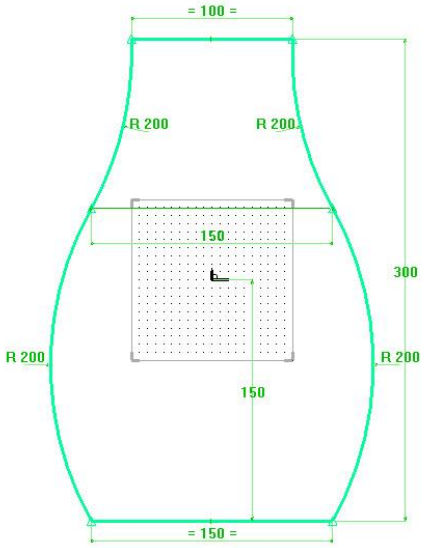
- Without leaving the function, select point (5) created previously, then point (3).
- Then apply a **tangency** **constraint** between the arcs (1;5) and (5;3).

Repeat the same operations to produce the drawing shown opposite.



- Start the **Circle** function.
- Enter a **radius** of 200mm and select point (2) in **Passing point** mode, then any point between point (2) and point (4).
- Without leaving the function, select point (6) created previously, then point (4).
- Click on **Invert** to invert the side of the arc.
- Then apply a **tangency** **constraint** between the arcs (2;6) and (6;4).

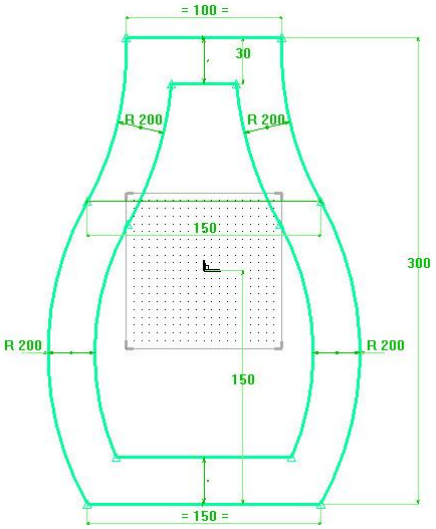
Constrain and dimension the points



- Start the **Constraint** function.
- Use the **alignment** constraint, then select the points (5) and (6).
- Then click on **Stop** and select **Alignment along X**.

- Use the **Dimension** function to dimension the distance between the points (5) and (6) at a nominal value of 150mm.

Create an offset profile

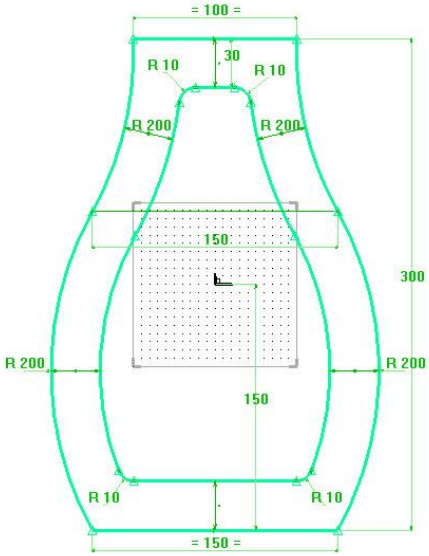


- Start the **Create offset profile** function. 
- Set the **Offset type = Profile**, then select the **Reference curve** option and select the sketch.

- Set **Mode = One side**.
- Place the offset <u>inside</u> the sketch, then enter a value of 30mm in **Through point** and press **Enter**

- Finish positioning the dimension.

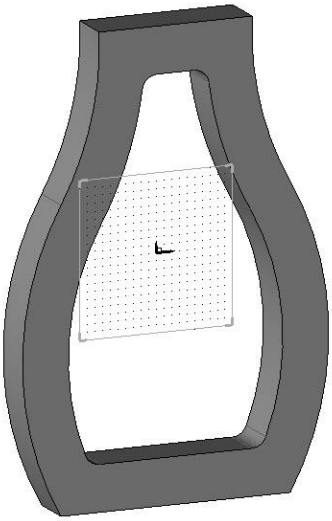
Create fillets



- Start the **Fillet** function. 
- Enter a **fillet radius** of 10mm, then select the four corners of the offset created previously.

- Create the fillets with the **Compute fillet(s)** option.

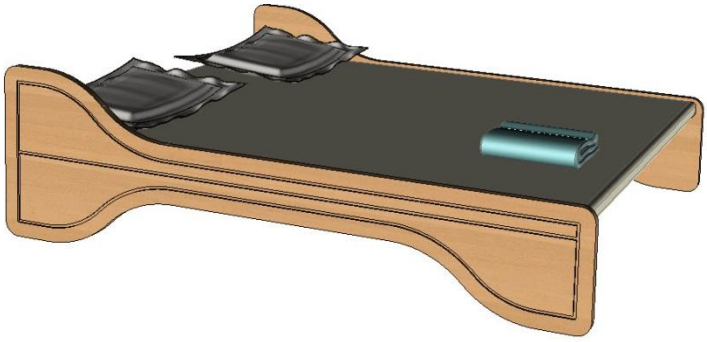
Extrude the sketch



- Finish the sketch with the **End sketch** function. 
- In the context bar, activate the **Shapes** context. 
- Start the **Create extruded shape** function. 
- Select the 2D sketch created previously, enter a height of 30mm in the dialog bar, then press **Enter** to confirm.

- **Save** the file by clicking on the disk icon. 
- Answer **No** to the part definition window.
- Save the file in the *Lamp* folder and rename it *Body*.

Exercise 2: Bed side panel



In this exercise, we are going to make the left-hand side panel of the bed.

Concepts addressed:

- Creating doucines in a sketch

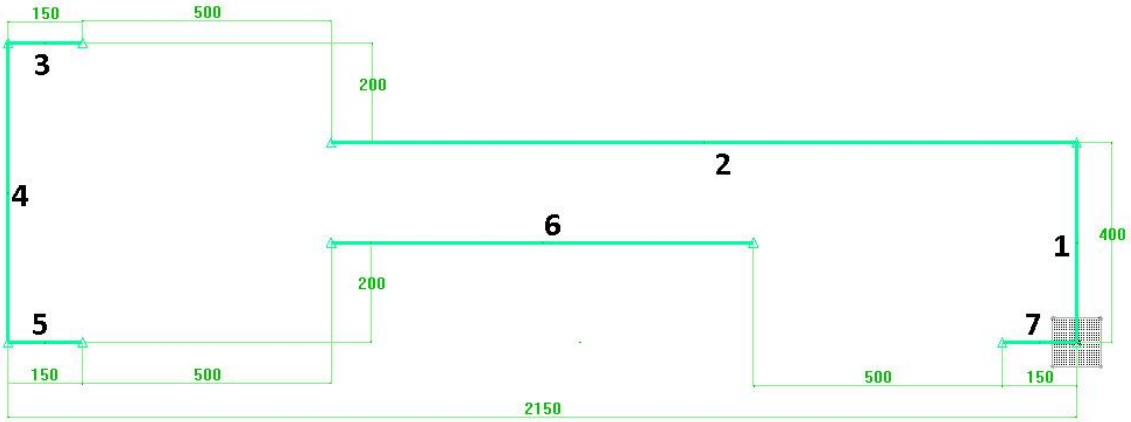
Design of the side panel

Create a new document

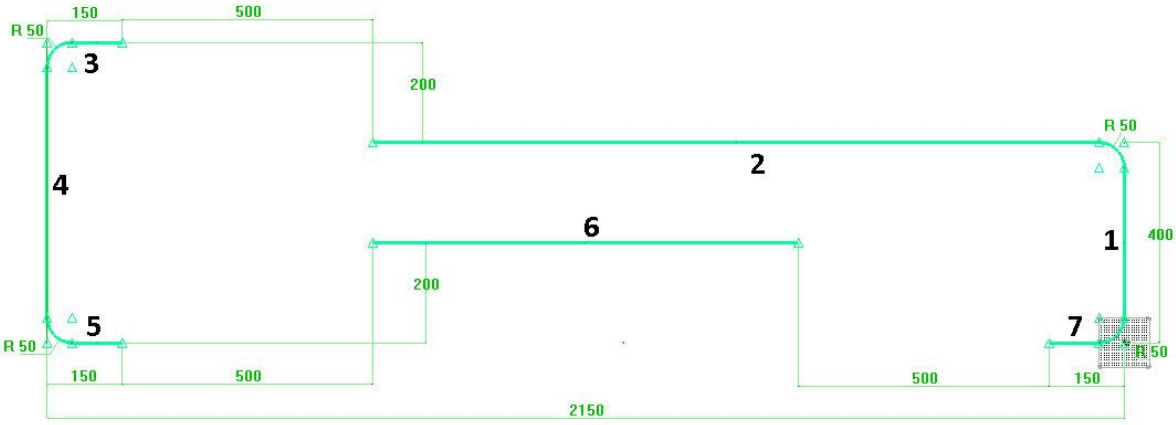
- Create a **new document** of the **Design** type.
  In the **Advanced parameters**, select **Without template**.
- Click on **OK** to confirm.
- In the context bar, activate the **Sketch** context.
- Start a **new sketch**.

Draw the lines

- Draw the seven lines below with the dimensions shown. 
  - All the lines are **oriented** along the **X** or **Y** axes.
  - Lines 7 and 1 start from the absolute coordinate system.
  - Line 5 is **aligned** with the **X** axis of the absolute coordinate system.



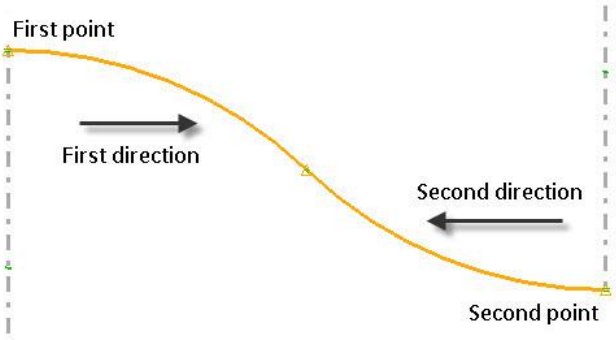
- Create **fillets** with a radius of 50mm between segments 1/2, 3/4, 4/5 and 7/1.



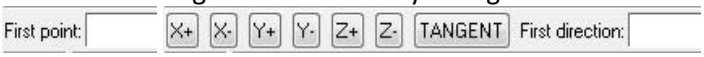
Create the doucines

- Start the **Doucine** function. 

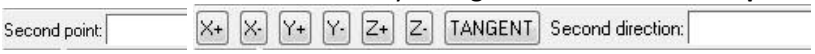
Note: The **Doucine** function is used to automatically create two tangent arcs between two points and in two directions.

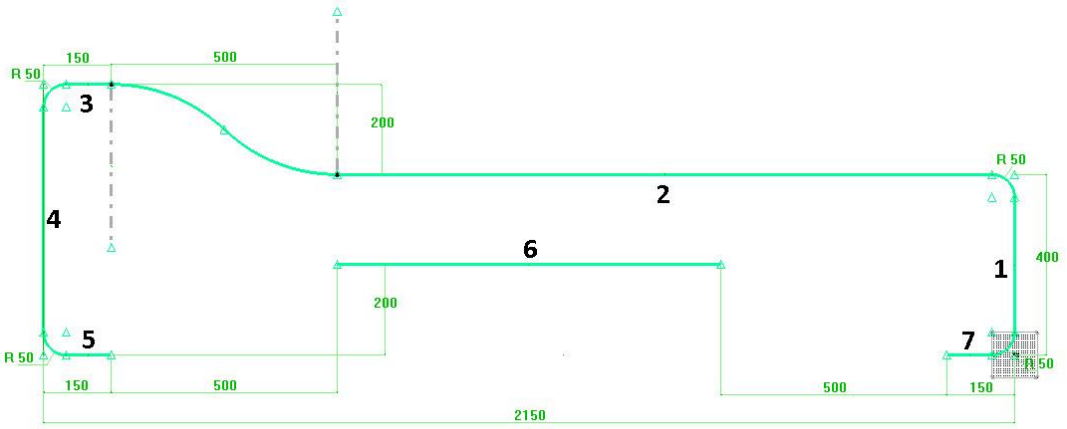


- Select the right-hand extremity of segment 3 as the **first point** and **X+** as the **first direction**.

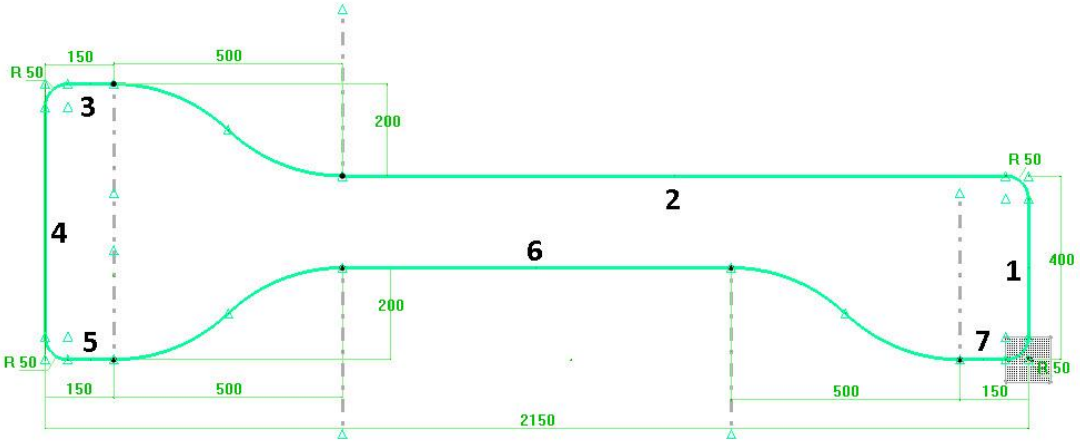


- Select the left-hand extremity of segment 2 as the **second point** and **X-** as the **second direction**.



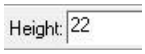


- Repeat the operation to create the **doucines** between segments 5 and 6 and between segments 6 and 7.

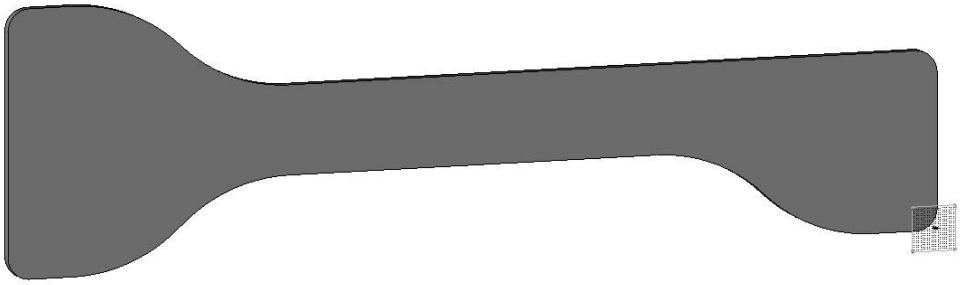


Extrude the part

- Finish the sketch with the **End sketch** function. 
- In the context bar, activate the **Shapes** context. 
- Start the **Create extruded shape** function. 
- Select the 2D sketch created previously and enter a **height** of 22mm in the dialog bar.



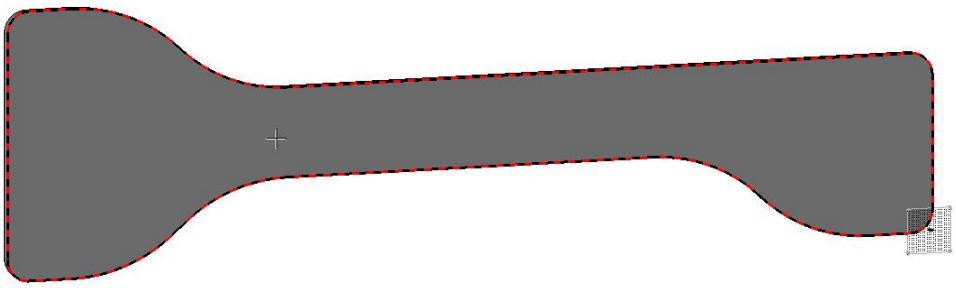
- **Save** the file by clicking on the disk icon. 
- Rename the file *Bed side panel*.



Supplement: Machining the decorative grooves

Create a groove

- In the context bar, activate the **Wood** context. 
- Start the **Groove** function. 
- Select **Sweep = Planar face** and select the main face of the side panel as the **reference face**.

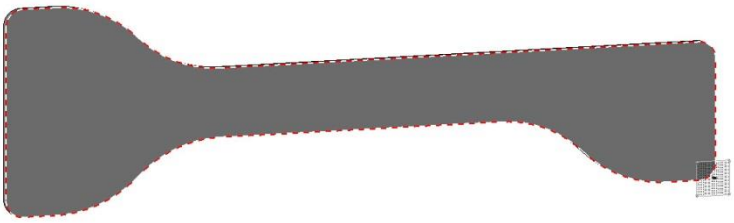


- In the dialog bar, set **Join edges = YES** and **Follow tangent edges = YES**.

Note : The **Join edges = YES** option is used to create a single machining operation for all the selected edges.
The **Follow tangent edges = YES** option is used to automatically select all the tangent edges of the selected edge.

- In the **Reference edge or curve for tool path** option, select one of the edges of the selected face.

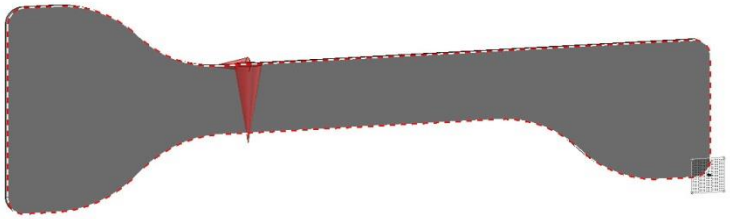
Since all the edges of the face are tangential, they are all selected in one go.



- Click **Stop** to validate the edges.

Two red arrows appear. They represent the direction of the machining and the gap side of the groove.

- The gap must be towards the inside of the part. If this is not the case, click on the arrow to invert.



- Once set, click on **OK** to confirm.

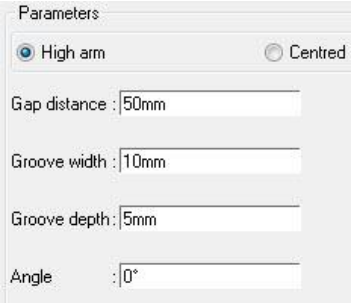
Configure the groove

The groove settings window opens.
Configure the various points mentioned below, from the top to the bottom of the window.

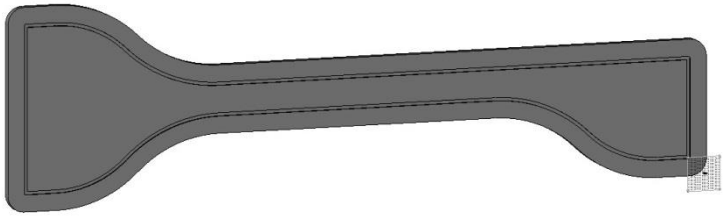
- Select a **Routers** type tool.

- In the list, select **Simple mill**.

- Then go to the **High arm** settings.
- Enter the following parameters:
  - **Gap distance** = 50mm
  - **Groove width** = 10mm
  - **Groove depth** = 5mm
  - **Angle** = 0°



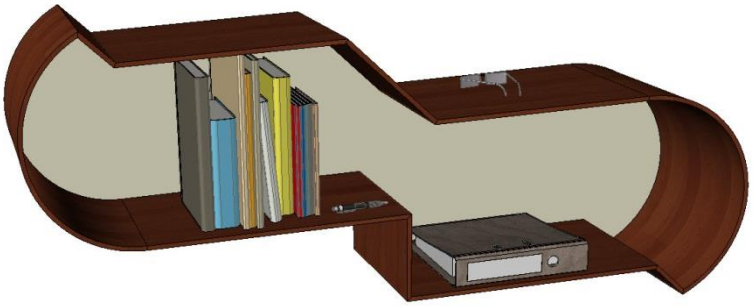
- Click **OK** to validate the settings of the groove.



- **Save** and **close** the file.

Exercise 3: Making a shelf

The goal of this exercise is to make the parts of the shelf.



Making the base

Concepts addressed:

- Creating points
- Dimensioning points
- Extrusion in one direction

Create a new document

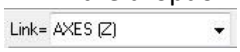
- Create a **new document** of the **Design** type.
- In the context bar, activate the **Sketch** context.
- Start a **new sketch** in the function bar.

Draw the top lines

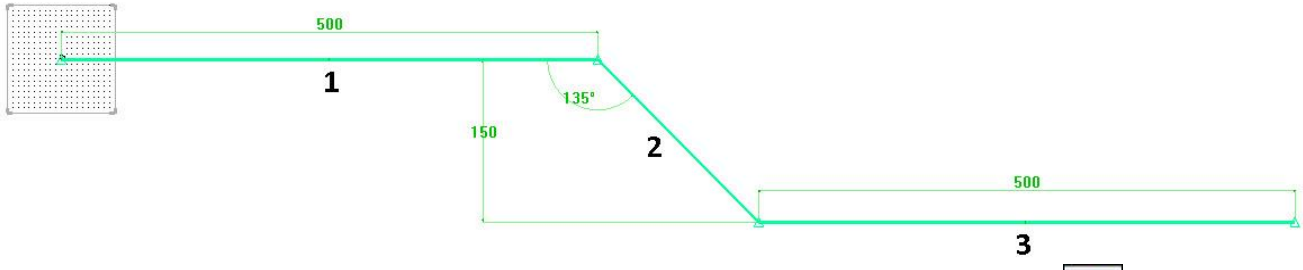
- Start the **Contour** function. 

Note: The **Contour** function is used to draw a series of segments or arcs.
The **Link** dropdown list can be used to choose the type of segment when drawing: **Line**, **Intersection**, **Arc**, **Tangent arc**,etc.

- Select the origin of the absolute coordinate system as the **profile or starting point**.
- In the dropdown list, select **Link = AXES (Z)** to automatically orientate the segment along X or Y.



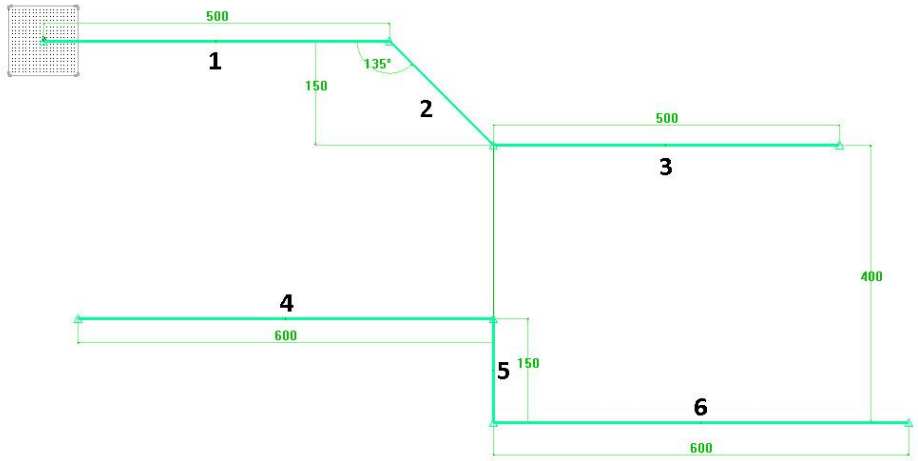
- Draw the three segments as shown below.
- For segment [2], set **Link = Line** in the dropdown list. 



- Use the **Dimension** function to apply the length and angle dimensions as shown above. 

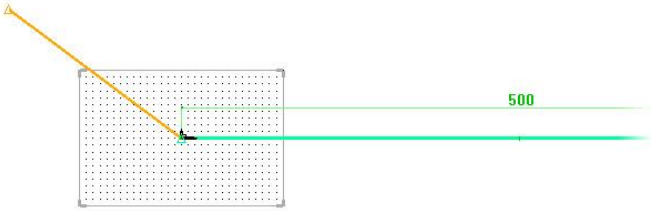
Draw the low lines

- Using the **Contour** function, draw the three other segments [4], [5] and [6] as shown below, with **Link = Axes** (Z).
- **Dimension** the three lengths of these segments.
- **Dimension** the distance between segment [3] and segment [6].
- Use the **Constraint** function to apply an **alignment** constraint between the right-hand point of the segment [4] and the left-hand point of the segment [3].
- Click **Stop** and align **along Y**.



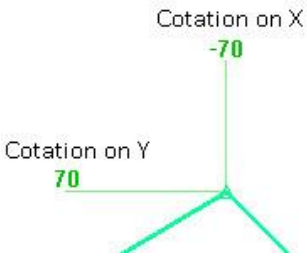
Draw the left-hand part

- Draw a line from the origin of the absolute coordinate system as shown below.

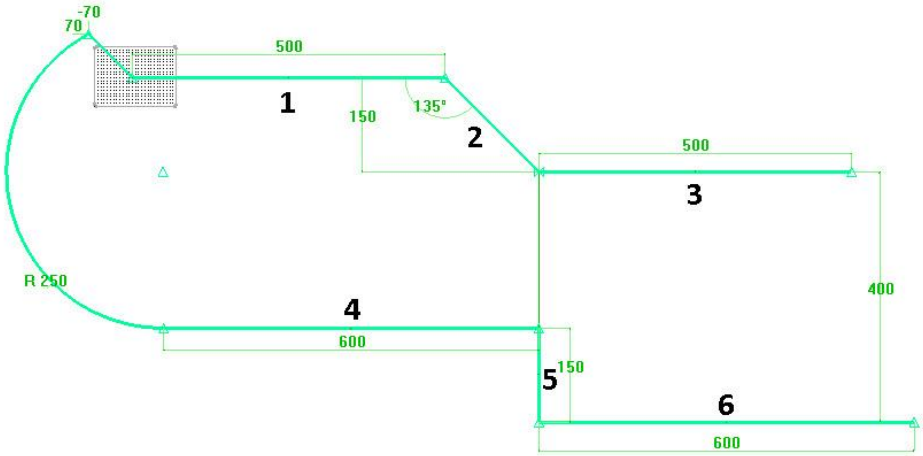


- **Dimension** the left-hand point of the line.

Note: The dimension of a point is made up of two dimensions on **X** and on **Y** from the current coordinate system. The horizontal dimension is the dimension along **Y** and the vertical dimension is along **X**. It is only possible to delete one of them.

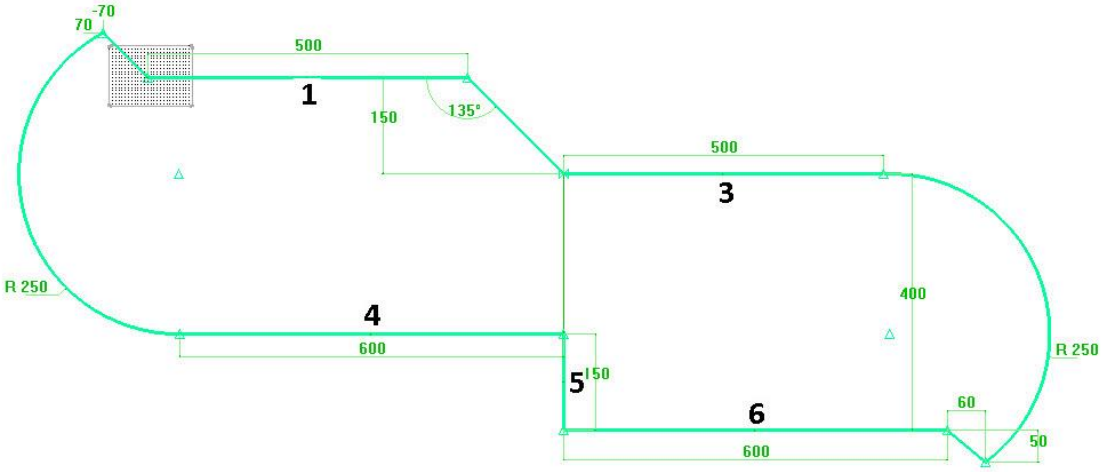


- Use the **Modify parameter** function to change the value of this point to -70mm along **X** and 70mm along **Y**.
- Finish by drawing a **circle** with a **radius = 250mm** from the point previously created to the left-hand extremity of segment [4].



Draw the right-hand part

- Then draw a **line** between the right-hand point of the segment [6] as shown below.
- **Dimension** the right-hand point of this line relative to the right-hand point of the segment [6].
- Apply a value of 60mm along the **X** axis and 50mm along the **Y** axis.
- Draw a **circle** with a **radius=250mm** passing through the point created previously and through the right-hand point of the segment [3].



Extrude the part

- Finish the sketch.
- Activate the **Shapes** context in the context bar and start the **Create extruded shape** function.
- Select the 2D sketch created previously, click on **Direction** and select **Z+**.

This setting is used to adjust the direction of extrusion of the part.

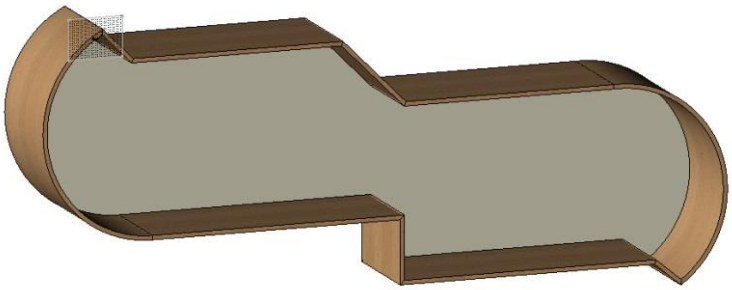
- Enter a **height** of 30mm, then press **Enter** to confirm.

- **Save** the file by clicking on the disk icon.
- Create a new folder called *Shelf* and rename the file *Shelf*.

Making the sides

Concepts addressed:

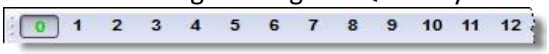
- Using layers
- Copying edges
- Using profiles



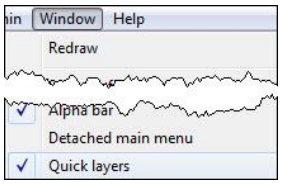
Use the layers

Note: Layers are used to position items on them and to display and conceal them in the course of the design operations.
This reduces the number of items in the display zone and improves the organization of the design.

Layers can be managed using the Quick layers bar.



Note: If the Quick layers bar does not appear on the screen, use **Window** > **Quick layers** to display it.



- Set layer 1 as current by clicking on **1** with the thumbwheel. All newly created items will now belong to layer 1.

Note: The current layer is shown in green in the Quick layers bar. 

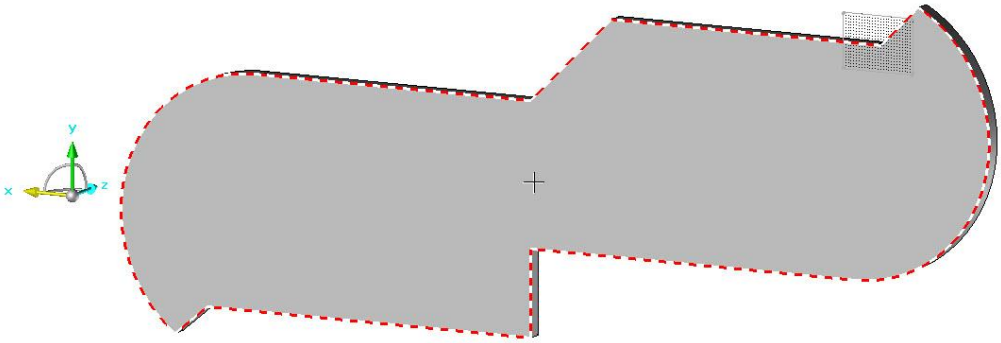
Copy the edge of the base

- Start the **Curves** context. 
- Use the **Edge** function. 

Note: The **Edge** function is used to automatically create several curves from parts such as faces, shapes or edge contours.

- Use **Mode = Face contour** to copy all the edges of a face.

- In the **Face to get the boundary contour from** field, enter the rear face (Z-) of the base.



A contour made up of several curves is automatically created around the face.

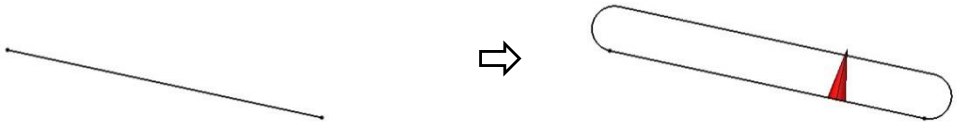
Create thickened curves

- Left-click **0** in the layers bar to switch off layer **0**.

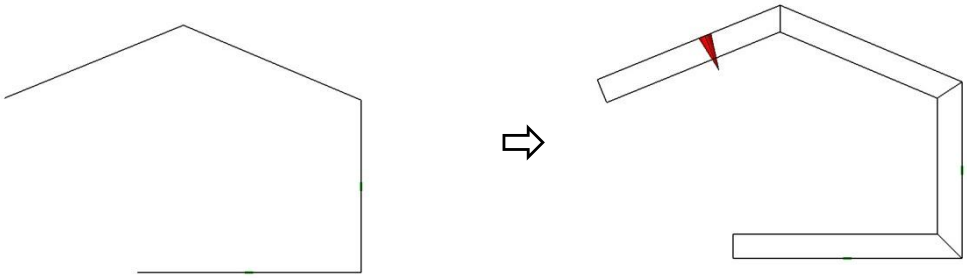
Note: Switching off a layer conceals all the items that belong to that layer. In this case, it is the base part and the absolute coordinate system.

- Use the **Thickened curves** function in the **Curves** > **Thickened curves** menu.

Note: Thickened curves are used to automatically create closed contours from a segment or a contour.



Thickened curve *of a segment at the end* ***External arcs***



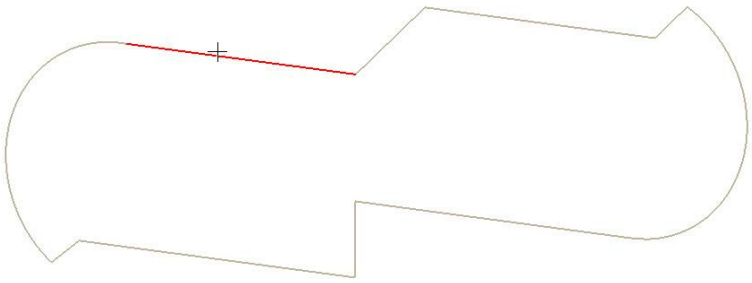
Thickened curve *of a segment at the end* ***Mitre cut***

- Enter **Thickness = 10mm**, **Symmetric = No**, **Second thickness = 0mm**, then **End type = Mitre cut**.

In the **Reference curve** option, select a segment of the contour and not the whole contour. To do this, use the **rotary selection**.

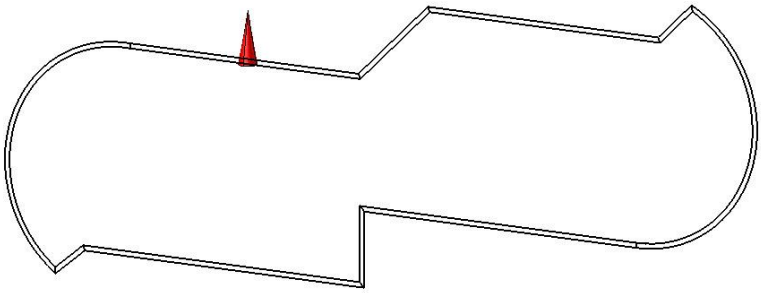
Note: **Rotary selection** is used to select several elements that are superimposed. This is the case here, where the contour and the different segments are superimposed.

- Place the mouse cursor on a segment of the contour, then left-click and hold. Then right-click to go from one part to another.



- Release the left-hand mouse button when the cursor passes over a segment of the contour, as shown above.
- Use the **All segments** option to automatically create thickened curves of the entire contour.

- Place the thickened curves outside the contour with the red arrow, as shown below.



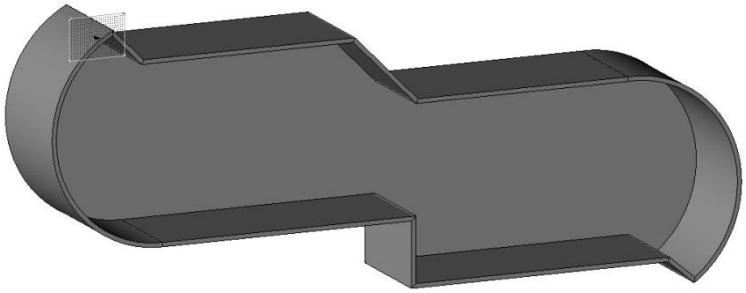
- Click **OK** to validate the thickened curves.

Extrude the sides

- Start the **Shapes** context. 
- Start the **Create extruded shape** function. 
- Select a first thickened curve.
- **Extrude** in the **Z+** direction 

to a **height** of 250mm. 

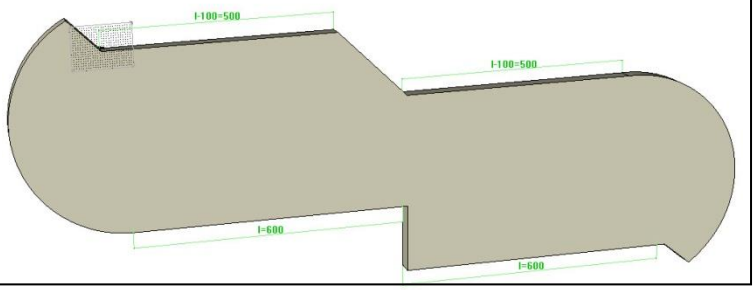
- Repeat the operation to extrude the 10 sides.
- Left-click **0** in the layers bar to switch on layer **0**.



Supplement: Configuring the length of the shelf

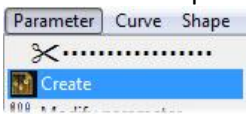
Note

A parameter is used to check and quickly modify the value of one or more design values. This parameter can be a length or angle parameter, or a parameter without units used to configure a quantity.



Create the parameter

- To create a parameter, start the **Create** function in the **Parameter** menu.



- Select **Unit type = Length** and enter a **value** of 600mm. Press **Enter** to confirm.

- Enter **Name=l** and **Designation=Length**. Press **Enter** to confirm.

Note: The **name** is the system name of the parameter that will be used in the various dimensions. The **designation** is what the user sees when using the parameter.

- Click on the **No text** option.

Note : This dialog is used to graphically display the parameter on the screen.

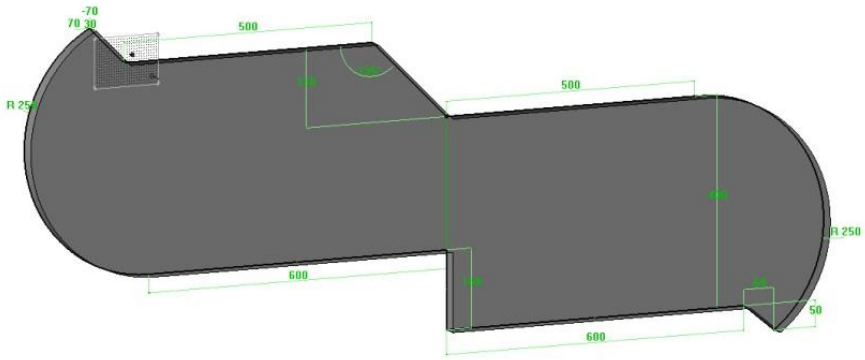
- Press **Esc** to exit the function.

Display the sketch

- Start the **Control elements** function. 

Note: Once a drawing element has been used (e.g., when a sketch is used to extrude a part), it is automatically hidden. Use the **Control elements** function to display the elements used by another element.

- Select the base of the shelf that was extruded earlier. The sketch used to build the base is displayed automatically.



Include the parameter in the sketch

- Start the **Parameter** > **Modify parameter** function.
- Then click on the 600mm length in the bottom left-hand corner.
- In the dialog bar, select **Parameter** PARAMETER, then **Replace**. REPLACE
- Enter **Replacement = Local** and **Replacement parameter = l**.

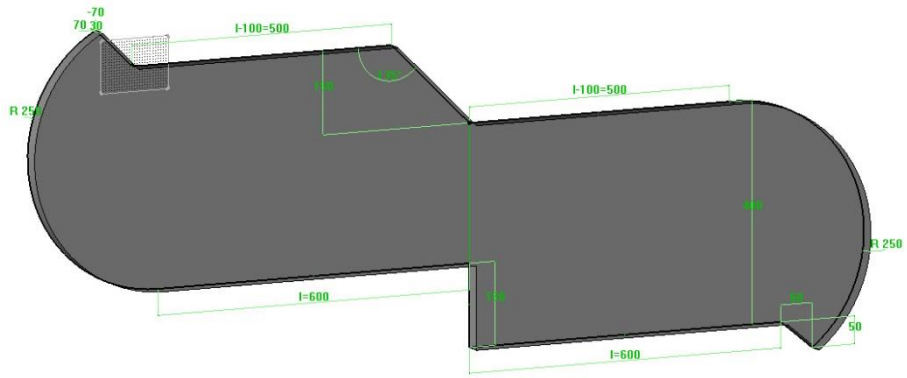
- Press **Enter** to confirm.

The value is now controlled by the parameter **l**. 

- Repeat the same operations for the 600mm value in the bottom right-hand corner.

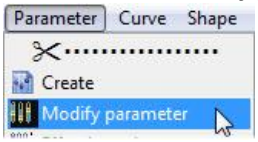
- Then replace the two upper dimensions of 500mm with the **Replacement parameter = l-100**. The dimension then has the same value as the parameter l-100mm.

The four length values of the shelf are now configured.



Use the parameter

- Start the **Control elements** function. 
  Select the base to conceal the sketch used to build it.
- Start the **Modify parameter** function in the **Parameter** menu.



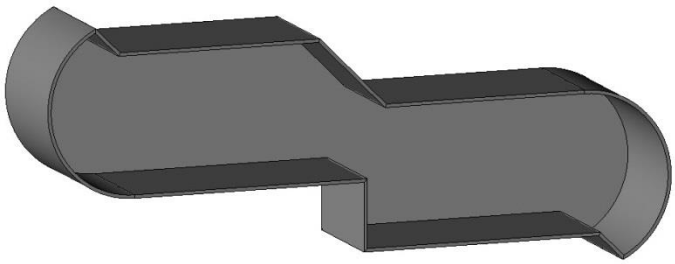
- Set **Parameter to modify = l**, and press **Enter** to confirm.

Note: When entering a parameter in a field, the field turns yellow if the parameter entered already exists.

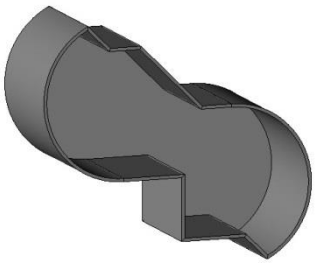
- Change the **nominal value** to 700mm, and press **Enter** to confirm.

The length of the shelf is automatically adjusted according to the parameter.
Any lengths greater than 200mm can be entered.

- For l=700mm:

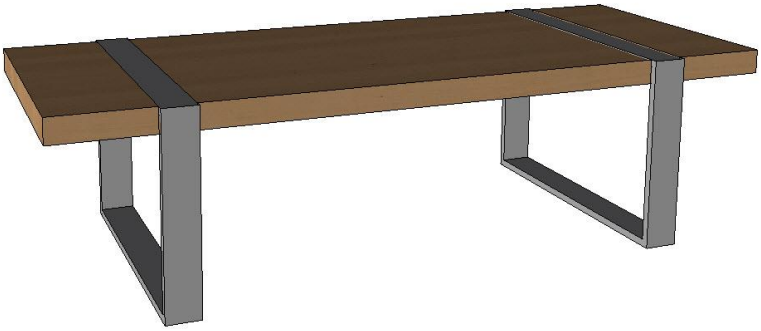


- For l=200mm:



Exercise 4: Making an indoor bench seat

The goal of this exercise is to create the parts and then understand how definition of parts and whole assembly works.



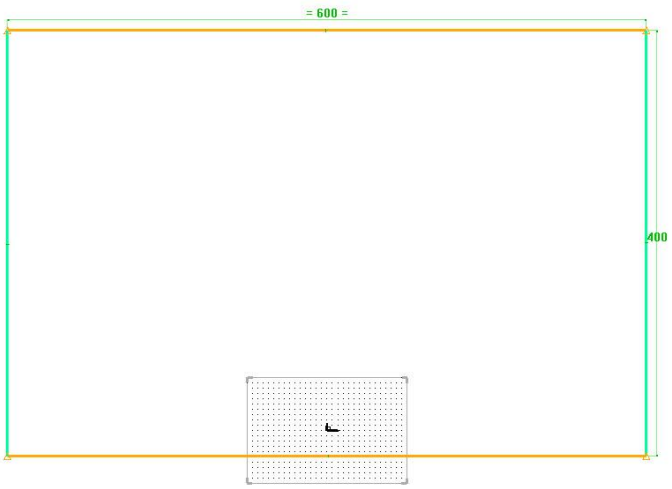
Making the base

Draw the base

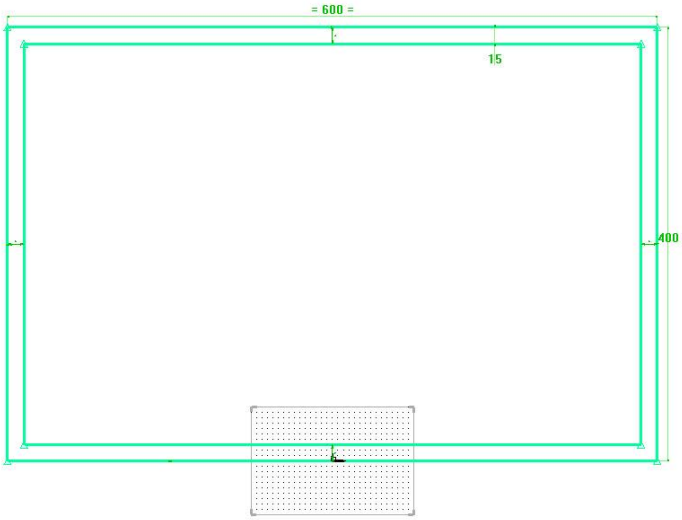
- Create a **new document** of the **Design** type. 
  In the **Advanced parameters**, select **Without template**, then click **OK** to confirm.
- In the context bar, activate the **Sketch** context, then start a **new sketch**. 
- Use the **Create contour** function. 
- Select the **Rectangular** option in the dialog bar. 

- Set **constraints = orientation** and select any two points as the **first** and **second points on the diagonal**.
- To finish, click on **Auto dimension** 

to automatically apply the dimensions. Press **Esc** to exit the function.
- Use **Modify parameter**  to change the **nominal value** of the dimension on **X** to 600mm, then the dimension on **Y** to 400mm.
- Use **Modify element**  , then select the dimension of 600mm. Select **Constraint**, then **Y** to symmetrically constrain the dimension along the **Y** axis.



Position the base

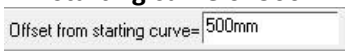


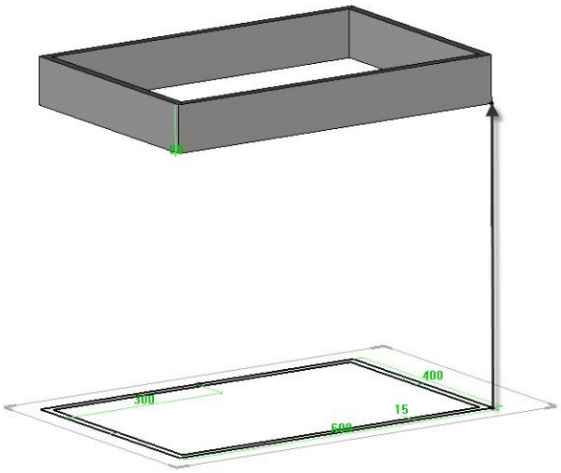
- Use the **Constraint** function and **align** the lower segment along the **X** axis of the absolute coordinate system.
- Set the **Offset** **Offset type = Profile** to draw the offset inside the rectangle at a **distance** of 15mm.
- Finish the sketch with the **End sketch** function.

Extrude the base

- Start the **Shapes** context, then use the **Create extruded shape** function.
- Select the sketch created earlier in the modes **Generatrix sketch = Global** and **Result = One shape**.

- Open the **advanced parameters** and enter an **offset from starting curve** of 500mm, then click **OK** to confirm.

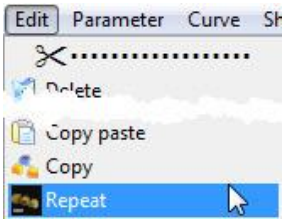




Note: The offset distance offsets the starting point of the extrusion relative to the drawing plan of the sketch.

- Use the **Direction** option to impose an extrusion direction towards **Z+**.
- Enter a **height** of 80mm, then press **Enter** to confirm.
- **Save** the document and rename it *Indoor bench seat*.

Repeat the base



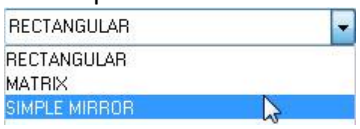
- Start the **Edit** > **Repeat** function.

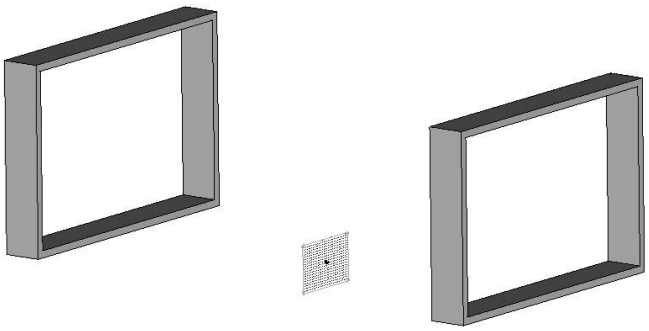
Note: The **Repeat** function is used to produce the same part several times from a template. The parts created are identical.

- Select the previously extruded part in **Hide template = Yes** mode.

Note: The selected part(s) to be repeated turn red.

- Select the **Simple mirror** type of repetition in the dropdown list.

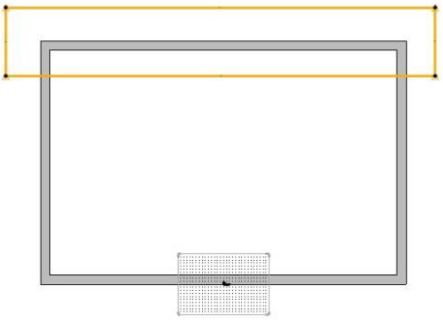




- Select the plane **XY** as the plane of symmetry. 

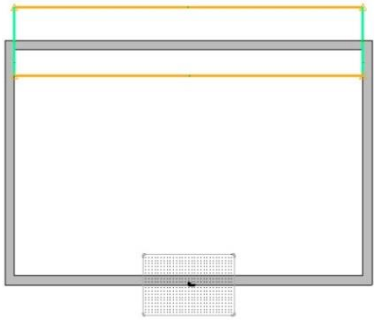
Creating the seat

Draw the seat



- Make **level 1** current.
- Start a **new sketch.** 
- Use the **Create contour** function. 
- Select the **Rectangular** 

option, then draw a rectangle.



- Use the **Constraint**  function to put a **coincidence**  constraint between the left-hand side of the drawn rectangle and the interior left-hand segment of the base.
- Repeat the operation between the right-hand side of the drawn rectangle and the right-hand interior segment of the base.

- Then apply a **coincidence constraint** between the top segment of the drawn rectangle and the top exterior segment of the base.

- Use **Dimension** to set the height of the rectangle to a **nominal value** of 50mm.

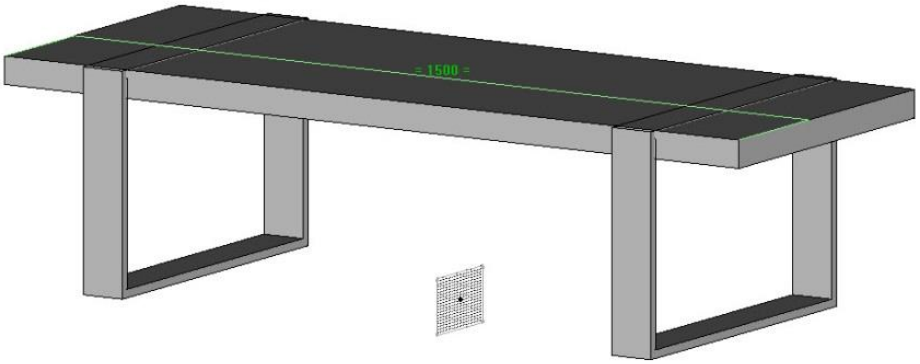
Extrude the seat

- Finish the sketch with the **End sketch** function. 
- Start the **Shapes**  context, then use the **Create extruded shape** function. 
- Select the sketch drawn earlier.
- Use **Alignment = Centered**. 

. Enter a **height** of 1500mm. 

Note: Extruding a shape in **Alignment = Centered** mode automatically centers the height of the extrusion with its sketch.

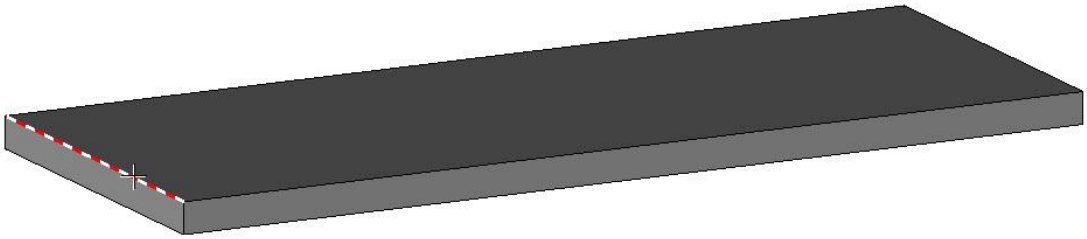
- Press **Enter** to confirm.



Create the grooves

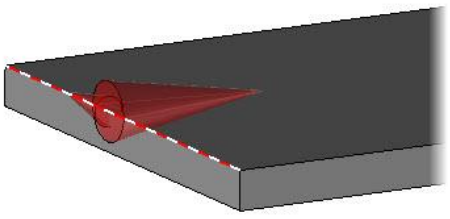
- Left-click on layer **1** to hide the bases.
- Start the **Wood** context. 
- Start the **Groove** function. 
- Select **Sweep = Planar face** and select the top face of the seat as the **reference face**.

- Select the left-hand edge of the seat in the **Reference edge or curve for tool path** field.



- Click on **Stop** to confirm.

Two red arrows appear. They represent the direction of the machining and the gap side of the groove.



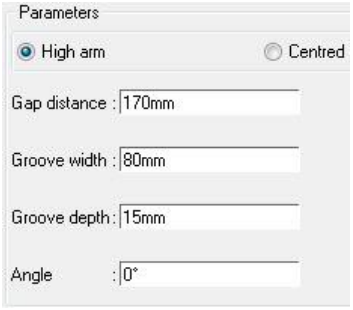
- The gap must be towards the inside of the part. If this is not the case, click on the arrow to invert.
- Once set, click **OK** to confirm.

The groove settings window opens.
Configure the various points mentioned below, from the top to the bottom of the window.

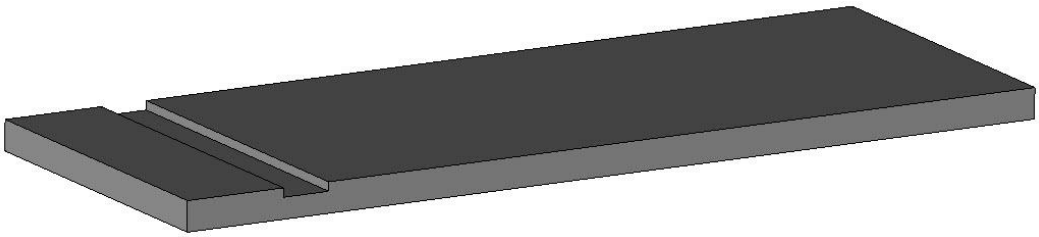
- Select a **Routers** type tool. 

- In the list, select **Simple mill**.

- Then go to the **High arm** settings.
- Enter the following parameters:
  - **Gap distance** = 170mm
  - **Groove width** = 80mm
  - **Groove depth** = 15mm
  - **Angle** = 0°



- Click **OK** to confirm the parameters. The groove is produced automatically.



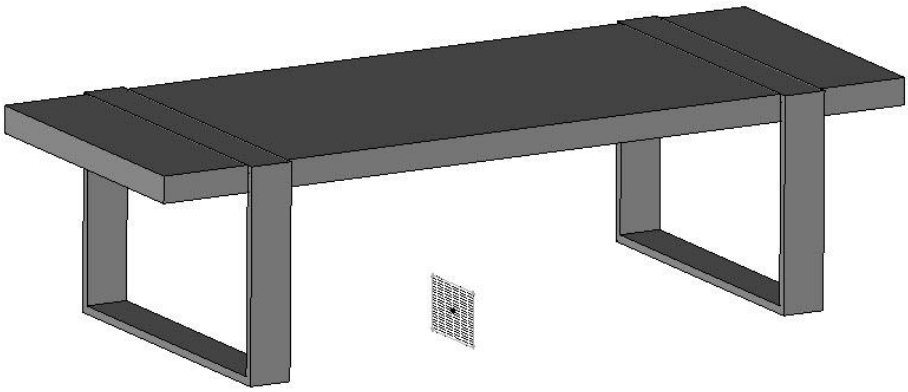
- Without quitting the function, click on **Copy operation**. 

Note: **Copy operation** repeats the last operation with the same parameters.
Warning: When using **Copy operation**, if one of the copied operations is modified, all the copied operations are modified.

- Then again select the top of the seat as the **reference face**.
- In the **Reference edge or curve for tool path** field, select the edge opposite the previously selected edge.
- Click on **Stop** to confirm.

The second groove is now created.

- Press **Esc** to exit the function.
- Left-click on layer **1** to display the bases.

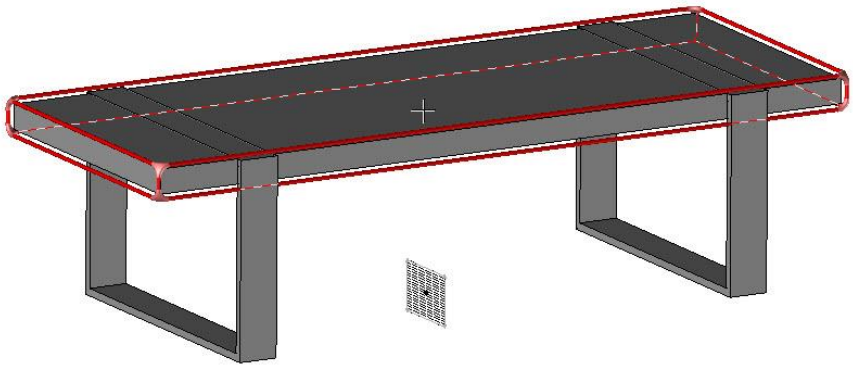


Data definition

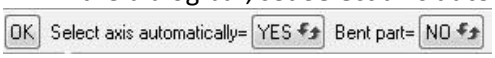
Define the seat

Note: Defining the parts allows different properties to be assigned to them so that they can then be identified and processed, for example in the BOM, drafting, export for sawing, etc.
The part definition is used to assign a **designation,** a **reference**, a **cut**, a **material**, etc.

- In the **Wood** context,  use the **Define** > **Define part** function. 
- In **Assembly = Main assembly**, select the previously extruded seat.



- In the dialog bar, set **Select axis automatically = YES** and **Bent part = No**.



Note: **Select axis automatically = YES** is used to automatically determine the length and width axes on the part. On complex shaped parts, the length and width axes can be selected manually.
Bent part = NO is used to unfold bent parts in order to calculate its precise cut when flat.

- Click **OK** to confirm the dialog.

The **Definition of a part** window opens automatically.

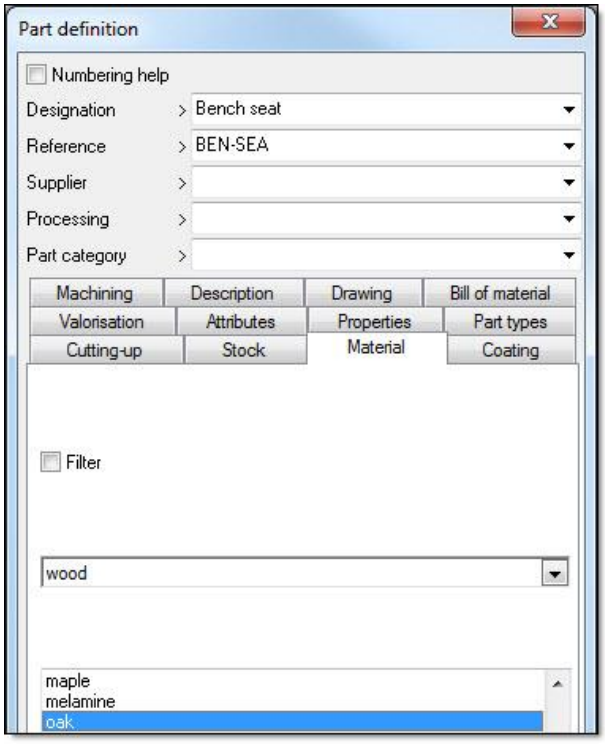
Note: The general properties of the part, such as its **Designation** and **Reference** are shown in the top part of the window.

- In the top part of the window, enter the **designation** *Bench seat* and the **reference** *BEN-SEA.*

Note: The different properties specific to the part are then classified in the various tabs, such as **Cutting-up**, **Material** and **Description**.

- In the **Material** tab, select **wood** in the material category dropdown list, then the **oak** material.

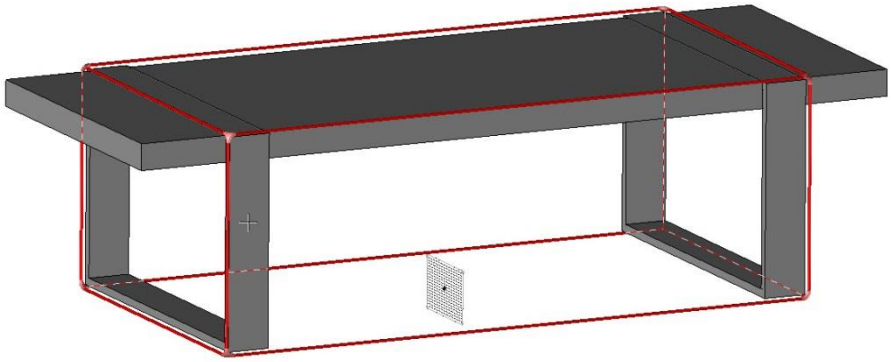
- Click **OK** to confirm.



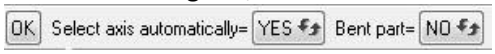
Define the bases

- In the **Wood** context,  use the **Define** > **Define part** function. 
- In **Assembly = Main assembly**, select the bases.

Note: The bases are repeated parts, so they have the same definition. Therefore, all the parts of the repetition are selected.

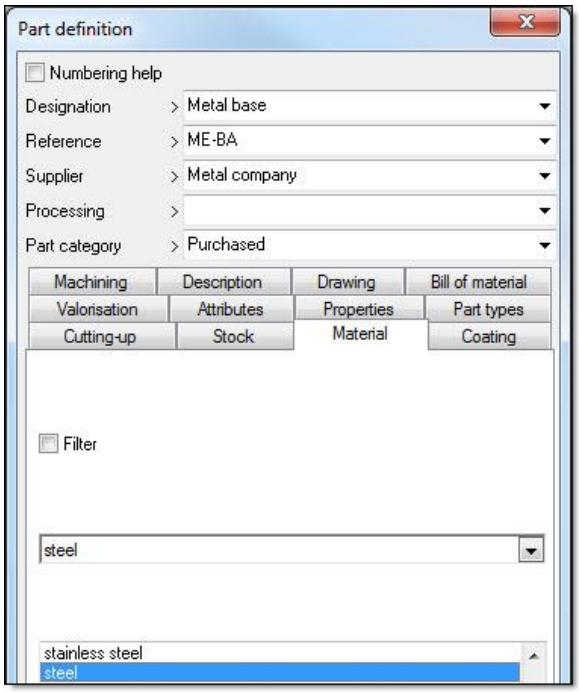


- In the dialog bar, set **Select axis automatically = YES** and **Bent part = No**.

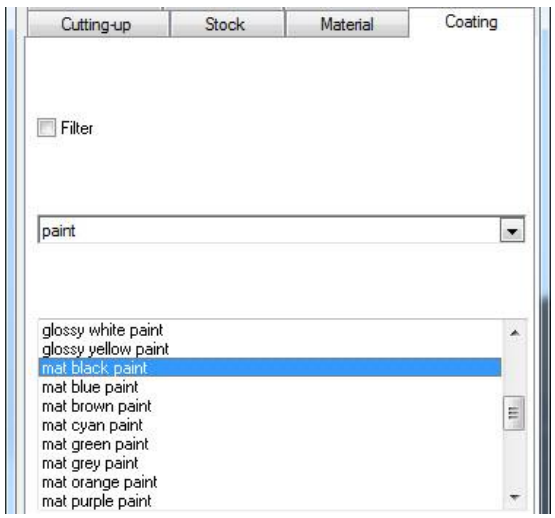


- Click **OK** to confirm the dialog.

- In the top part of the window, enter:
  - **Designation**: *Metal base*
  - **Reference**: *ME-BA*
  - **Supplier**: *Metal company*
  - Part category: *Purchased*

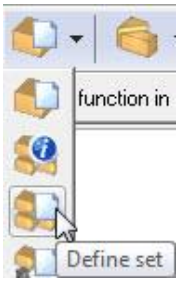


- In the **Material** tab, select **steel** in the **steel** category.
- In the **Coating** tab, select **mat black paint** in the **paint** category.
- Click **OK** to confirm.



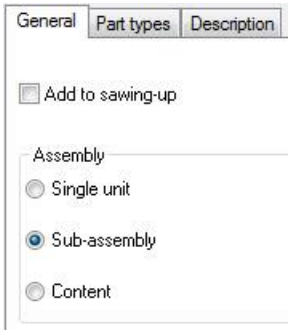
Define the set

Note: Defining the set allows different properties to be attributed to the newly created set. This allows the assembly to be identified and processed, for example in BOMs, drafts, etc.
The definition of a set is used to assign a **designation**, a **reference**, etc.



- In the **Wood** context  , use the **Define** > **Define set** function.
- Click on **Characteristics**. 

- In the **Designation** field, enter *Indoor bench*



- In the **General** tab, deactivate the **Add to sawing-up** field.

Note: Adding the set to sawing-up allows the enclosing dimensions of the set to be calculated. But if this information is not useful, then deactivating the tick box will speed up the calculations.

- In the **Assembly** section, tick the **Sub-assembly** option.

Note: At this point, three assembly modes are possible:

- The **Single Unit** mode: displays the BOM of the assembly without showing the component parts.

1	Indoor bench	-	-	-
NB	DESIGNATION	FINAL_LENGTH	FINAL_WIDTH	FINAL_THICKNESS

- The **Sub-assembly** mode: displays the BOM of the assembly and the component parts.

1	Indoor bench	-	-	-
2	Metal base	600.00	400.00	80.00
1	Bench seat	1500.00	570.00	50.00
NB	DESIGNATION	FINAL_LENGTH	FINAL_WIDTH	FINAL_THICKNESS

- The **Content** mode: displays only the parts contained in the assembly, without showing the assembly.

2	Metal base	600.00	400.00	80.00
1	Bench seat	1500.00	570.00	50.00
NB	DESIGNATION	FINAL_LENGTH	FINAL_WIDTH	FINAL_THICKNESS

- Click **OK** to confirm.
- **Save** the file. 
- Configure the **rendering** by opening the **Rendering**  icon and selecting the **realistic rendering + edges** mode. 

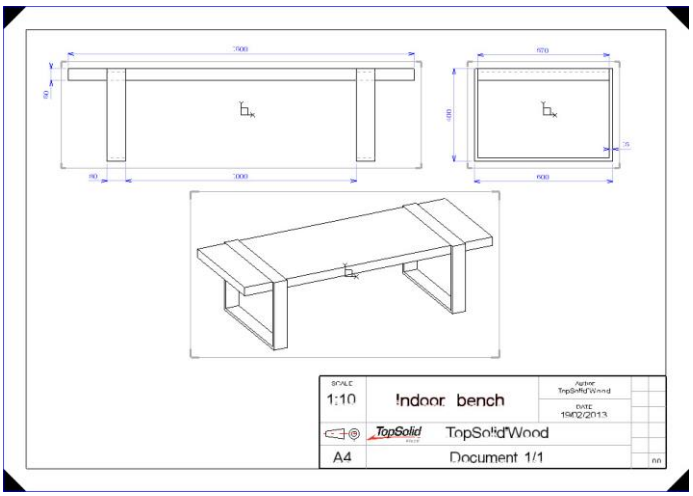
Note: The **realistic rendering** mode displays the textures of the parts directly.

Creating a draft

The goal of this exercise is to create a draft of the complete indoor bench.

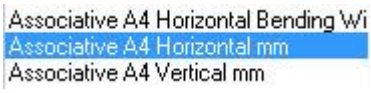
Concepts addressed:

- Creating a draft document
- Positioning a main view
- Editing the draft
- Positioning auxiliary views
- Applying the dimensions

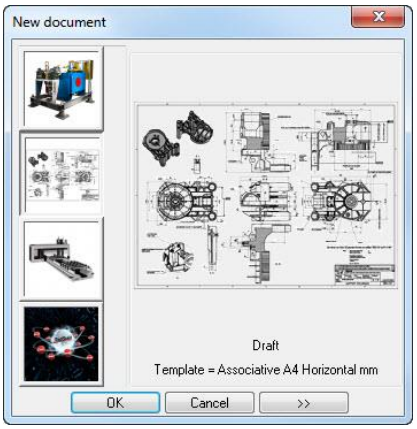


Create a new draft document

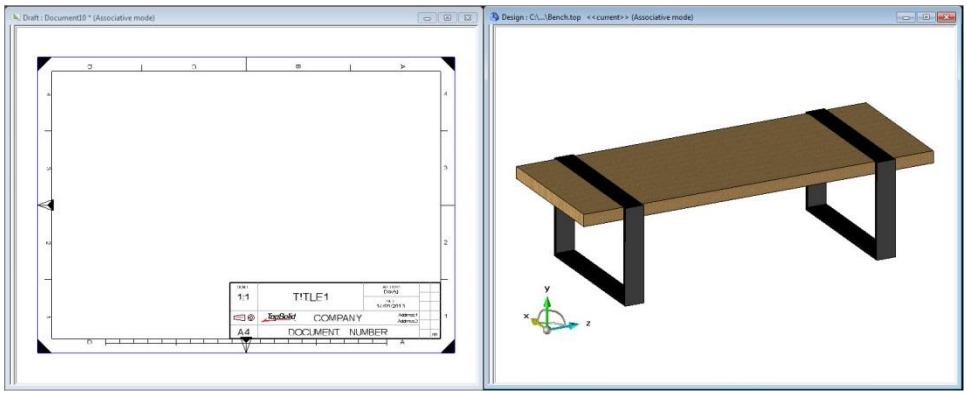
- Create a **new document.** 
- Select a **Draft** type document.
- In the **Advanced parameters**,  select a standard **Associative A4 Horizontal** template.



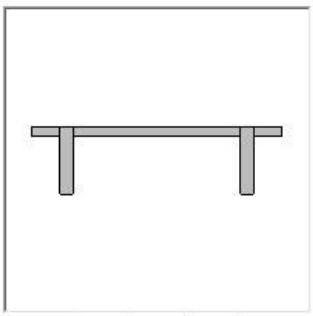
- Click **OK** to confirm.



The draft and design windows are positioned next to one another.

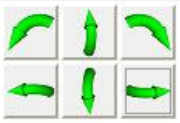


Position the main view



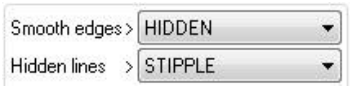
- In the **View**  context, select the **Main view** function. 
- Select **Assembly** to draft the entire document. 
- In **Pick on the document containing the assembly**, select the 3D document by clicking on it.

The **View creation** window opens.



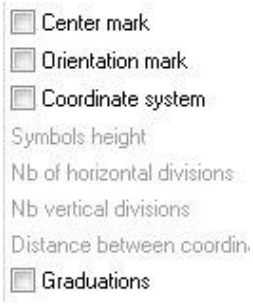
- Configure the main view by positioning the green arrows as shown opposite.

- Set the **smooth edges** to **Hidden**, and the **hidden lines** to **Stipple**.
- Click **OK** to confirm.
- Left-click to position the view.

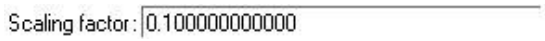


Note: By default the scale of the draft is 1. Scaling is then applied to the entire draft in order to be applied to all the views.
If the scale of the view is OK, auxiliary views can be created on the fly using the **Auxiliary view** button. 

Edit the draft



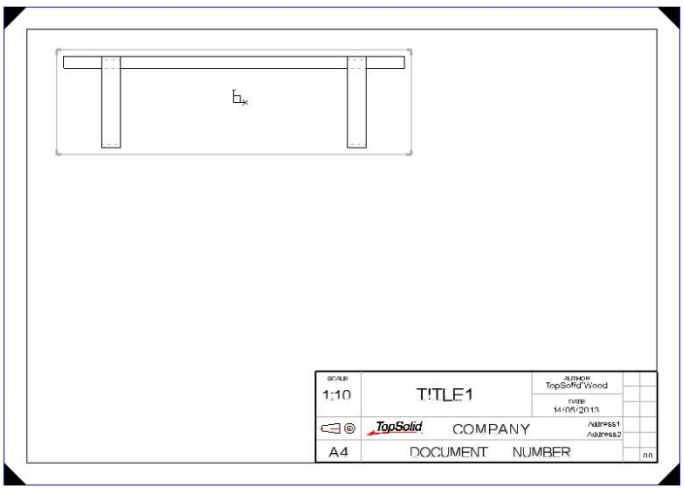
- Use **Modify element**  and select the frame of the sheet. The Edit window opens.
- Deactivate the **Center mark**, **Orientation mark**, **Coordinate system** and **Graduations** tick boxes.
- Set the **scaling factor** to 0.1.



- Click **OK** to confirm.

Note: As soon as a view is modified, it becomes not up to date and must be regenerated. In this way, all the views in a draft are only recalculated once after making several modifications.
However, it is possible to automatically recalculate the views, but this can become complex when modifying large projects.

- Use **Regenerate**  , then select the view to recalculate it.
- Use **Move parents**  to move the view to the top left-hand corner of the draft.



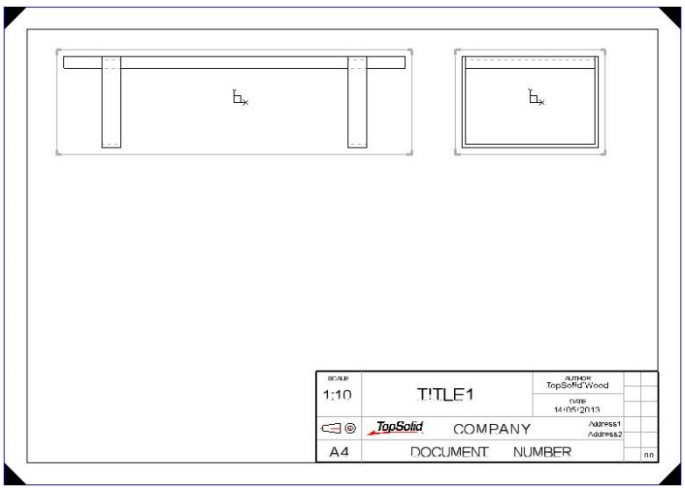
- **Save** the draft using the disk icon.  . Keep the default name.

Note: If the draft only contains one assembly or one part, the document is automatically renamed as this assembly or part.

Position auxiliary views

- Use the **Auxiliary view function.** 
- Validate the default parameters and position the auxiliary view using **Auxiliary view.** 

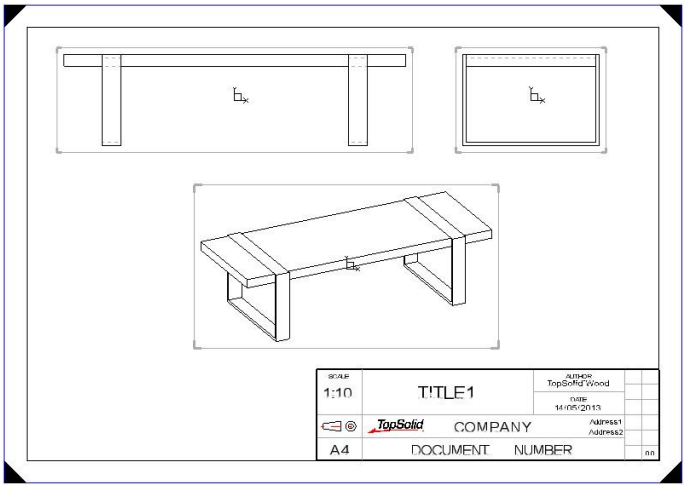
- Place the cursor to the right of the main view, then click to position the view.



- Select the first positioned view as the **reference view**.

- Configure the **smooth edges** and **hidden lines** as **Hidden**.

- Position the view with **Auxiliary view.** Position this view beneath and to the right of the main view. A perspective view is automatically generated.
- Use the **Move parents** function. 
Center the perspective view on the width of the draft.



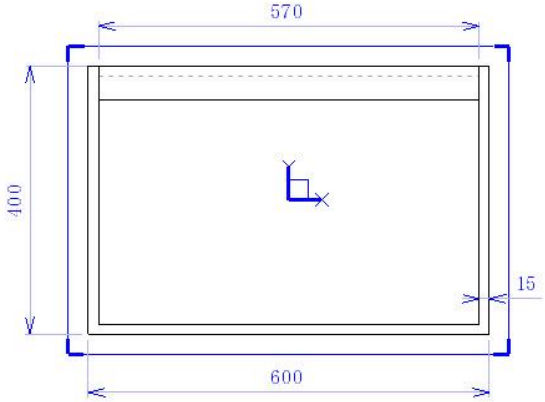
Dimension the views

- Start the **Dimension** context. 
- Use the **Fast dimension** function. 

Note: The **Fast dimension** function is used to quickly size a drawing element:

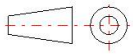
- click on a segment to dimension the length;
- click on an element, then a second element to dimension the distance between them;
- click on an segment, then a second, non-parallel segment to dimension the angle between them;
- click on a circle to dimension its diameter/radius.

- Place the different dimensions, such as the length/width/height of the bench, the dimensions of the base and the dimensions of the seat, on the drawing.



Fill in the title block

- Use **Modify element**  and change **Title 1** to *Indoor bench*.
  Replace **Company** by *TopSolid'Wood* and **Document number** by *Document 1/1.*
- Use **Delete element**  to delete **Address 1** and **Address 2.**
- **Save** the document.

SCALE 1:10	Indoor bench	Author TopSolid'Wood		
		DATE 19/02/2013		
	TopSolid Wood TopSolid'Wood			
A4	Document 1/1			00

Exercise 5: Making a shelf by bottom up assembly

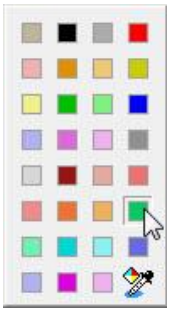
The goal of this exercise is to make three standard production parts, then assemble them to make the shelf.

Concepts addressed:

- Using design colors
- Including files
- Assembling by constraint

Making the parts

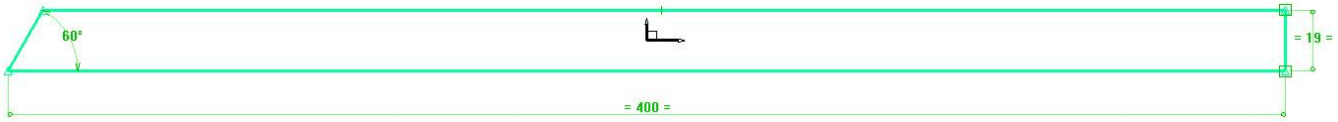
Make the first part



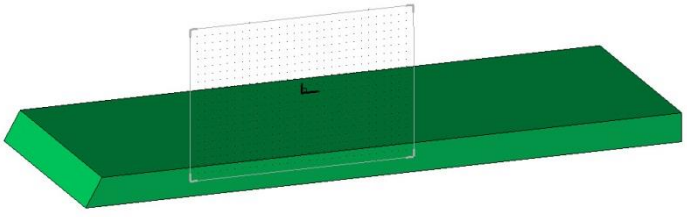
- Create a **new document** of the **Design** type.
  In the **Advanced parameters**, select **Without template**.
- Change the **design color** by opening the **Color** tab. 
  Select **Spring green (24)**.

Note: The design color is used to draw different design elements in a specific color.

- Start a **new sketch**.
- Draw the sketch shown below.
  The 400mm and 19mm dimensions are constrained along the X and Y axes.

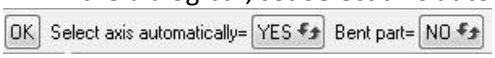


- **Finish the sketch**, then **extrude** the part by a **height** of 150mm.



Define the part

- In the **Wood** context,  use the **Define** > **Define part** function. 
- In **Assembly = Main assembly**, select the created part.
- In the dialog bar, set **Select axis automatically = YES** and **Bent part = No**.



- Click **OK** to confirm the dialog.

- In the top part of the window, enter:
  - **Designation**: *Part*

- In the **Material** tab, select **lime** in the **wood** category.
- Click **OK** to confirm.

Define the set

- In the **Wood** context,  use the **Define** > **Define set** function. 
- Click on **Characteristics**. 

- Then select the **Modify sawing-up** button.

Note: In this case, the file only contains one ready-defined part.
Therefore, the definition of the assembly is the same as the definition of the part. Therefore, a modification of the sawing-up of the assembly is proposed.

- Enter:
  - **Designation**: *Shelf part 1*
  - **Reference**: *SP-1*
- In the **Assembly** section, tick the **Single Unit** option.

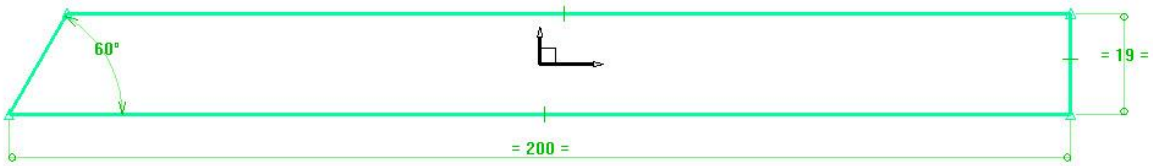
Note: The **Single Unit** mode is used mainly in two cases:

- either when the assembly only contains one part;
- or when the assembly is purchased already finished, in which case it is not necessary to know and process the various parts making up the assembly.

- Click **OK** to confirm.
- **Save** the file an keep its default name.

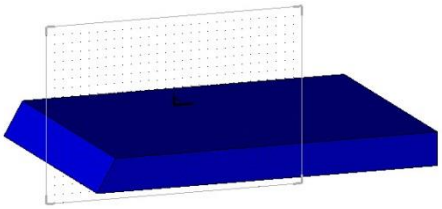
Make the second part

- Draw the sketch shown below in a **new Design document**.
- Use the design color **blue (12)**.



- **Extrude** the sketch to a **height** of 150mm.

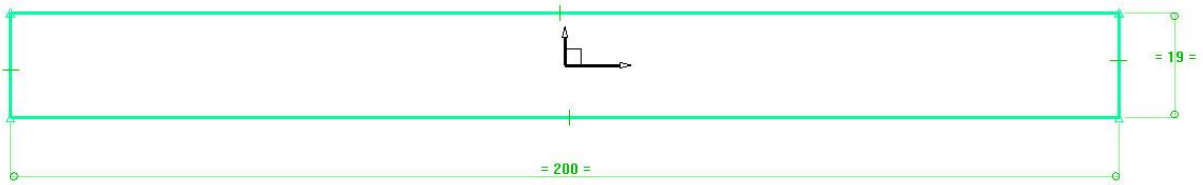
- **Define the part**:
  - **Designation**: *Part*
  - **Material**: **wood** > **lime**



- **Define the set**> **Characteristic**:
  - **Designation**: *Shelf part 2*
  - **Reference**: *SP-2*
  - **Mode**: **Single Unit**
- **Save** the file in the *Shelf by bottom up assembly* folder.

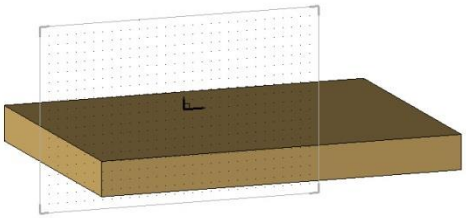
Make the third part

- Draw the sketch shown below in a **new Design document**.
- Use the design color **light orange (7)**.
- To draw this rectangle, use the command **Sketch**  > **Contour**  > **Rectangular** RECTANGULAR to directly draw a rectangle.



- Extrude the sketch to a **height** of 150mm

- **Define the part**:
  - **Designation**: *Part*
  - **Material**: **wood > lime**



- **Define the set**> **Characteristic**:
  - **Designation**: *Shelf part 3*
  - **Reference**: *SP-3*
  - **Mode**: **Single Unit**
- **Save** the file in the *Shelf by bottom up assembly* folder.

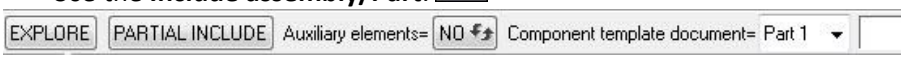
Assembling the shelf

Note: Bottom up assembly allows all or some of the parts contained in another file to be assembled in a file.
For example, this allows the same standard component to be inserted several times in one or more files.

- **Open** the three files you have just created. 

Create the bottom up assembly file

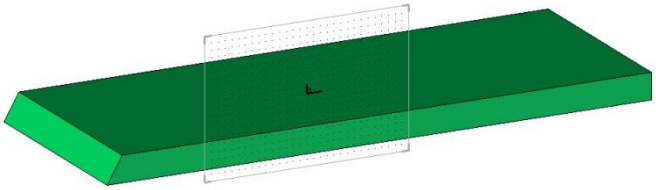
- Create a **new document** of the **Design** type.
  In the **Advanced parameters**, select **Without template**.
- **Save** this file in the *Shelf by bottom up assembly* folder and rename it *Shelf assembly.*
- Start the **Assembly** context. 
- Use the **Include assembly/Part**. 



  - **Explore**: opens a browser to include a file stored in a folder.
  - **Partial include**: includes only one or a few parts from a file.
  - **Auxiliary elements = No**: only includes the auxiliary elements of the selected document.
  - **Component template document =** is used to select the document to be inserted in the current document, either using the dropdown list that contains the open files, or by directly clicking in the document to be included.

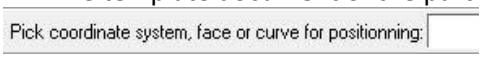
- Select **Part 1** in the dropdown list.

The document **Part 1** is included in the current document.

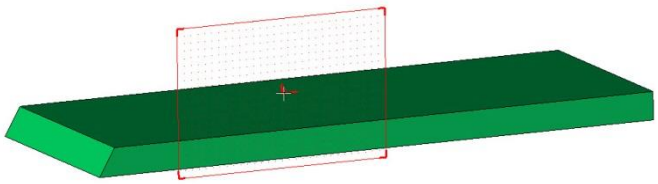


- Select **Other positioning**.

- The template document of the part opens. Select the **absolute coordinates**.



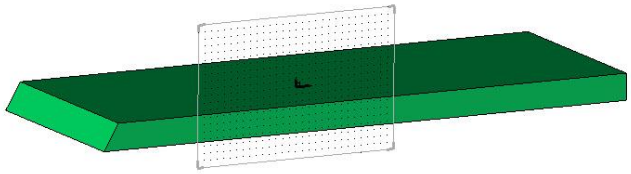
- Then select the **absolute coordinates** of the assembly document as the **destination coordinates**.
- Press **Esc** to exit the function.



Place and constrain the second part

- Insert **Part 2** using **Include Assembly/Part**. 

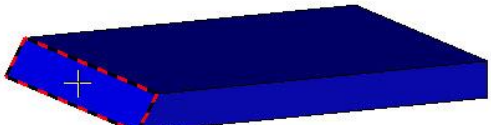
- Select **Part 2** in the dropdown list.
- Left-click in the space to release the part.



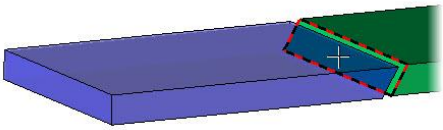
Positioning constraints can now be applied to the inserted part.

- In the dialog, set **Type = Auto** to automatically choose the type of constraint to be used and **Magnetic = Yes** to dynamically display the result of the constraint.

- Select the chamfer of the blue part as the **origin geometry**.



- Select the chamfer of the green part as the **destination geometry**. The blue part is positioned accordingly.



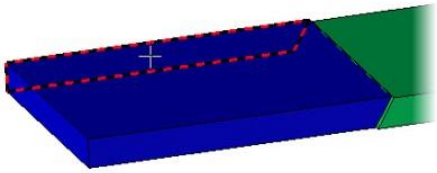
The following dialog is used to configure the newly created constraint.

- **Type =** Since several constraints are possible, it is possible to select the type of constraint used.

- Select the **Mate** constraint.

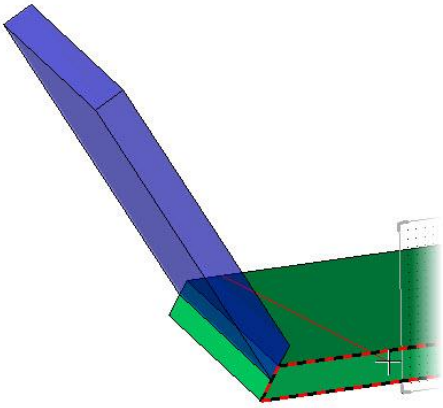
- **Distance =** A gap can be included in the constraint.

- Enter 0mm.
- Click **OK**.



- More constraints can be placed in the dialog. Select the rear edge of the blue part as the **origin geometry**.

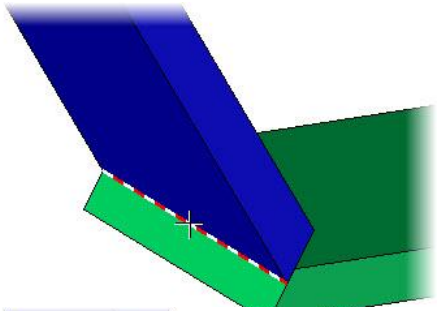
Note: If you have difficulty selecting an element (rear face or edge on a face), then you can use the **rotary selection**. Left-click and hold, then right-click to navigate between the various superimposed elements.



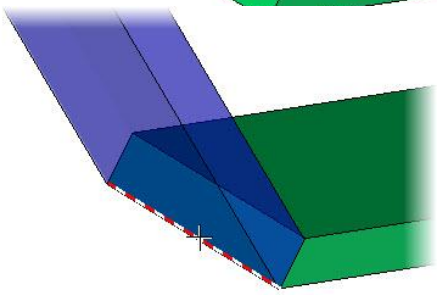
- Select the front edge of the green part as the **destination geometry**.

- Click **OK** to validate the next dialog, with **Type = Alignment** and **Distance = 0mm**.

- Place one last constraint, so that the green part has no more degrees of freedom.
- Select the lower edge of the blue part as the **origin geometry**.

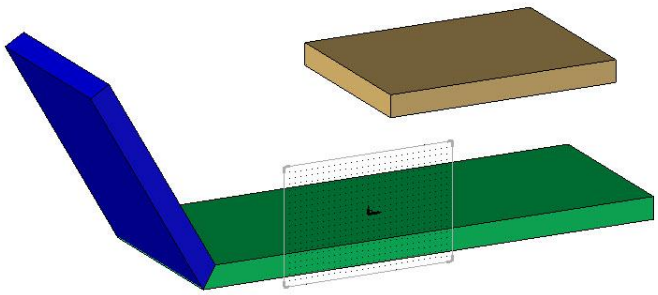


- Select the lower edge of the chamfer of the green part as the **destination geometry**.
- Press **Esc** to exit the function.

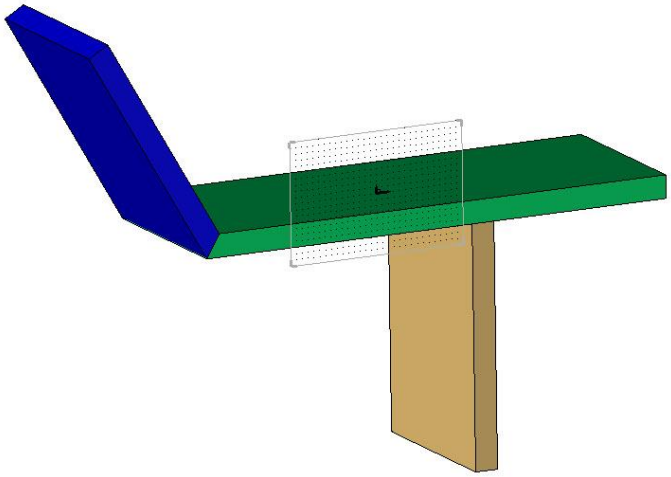


Include and constrain the third part

- Insert **Part 3** using **Include Assembly/Part**. 
- Select **Part 3** in the dropdown list, then place it in the document.



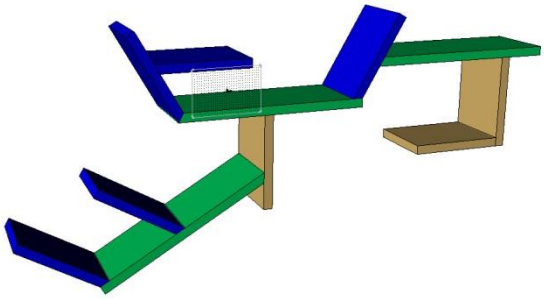
- Apply **constraints** to part 3 to position it as shown opposite:
  - Part 3 edge / Lower face of Part 1.
  - Part 3 front edge / Front edge of Part 1.
  - Part 3 right face / Part 1 right edge. For this constraint, enter a **distance** of 150mm.



- Finish positioning by clicking **Stop**. 

- The next dialog can be used to directly repeat the inserted part. Do not repeat the part with **No propagation**. 

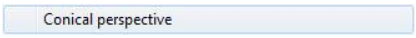
Place other parts



- The next dialog can be used either to place another **part 3**, or to select another part to be inserted using **Other component**.

- Continue to place the parts until you reach the final result.
- Use the **realistic rendering** in the **Rendering** tab.

- Open the **View**  tab and select the **Configure view** function. 
- In the list, select **Conical perspective**.

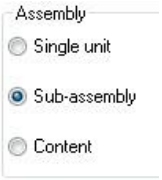


Note: **Conical perspective** allows for perspectives with a vanishing point. It offers a more realistic view, but is more difficult to use.

- **Save** the document.

Definition of the assembly

- **Define the assembly** using the command in the **Wood** context, **Define** > **Define set**.
- Select **Characteristics**. CHARACTERISTICS
- As the **designation,** enter *Bottom up assembly shelf*, then as **reference** *SHE.*

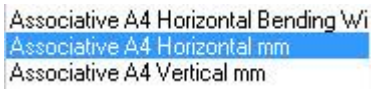


- In the **Assembly** section, tick the **Sub-assembly** option.
- Click **OK** to confirm.
- **Save** the file.

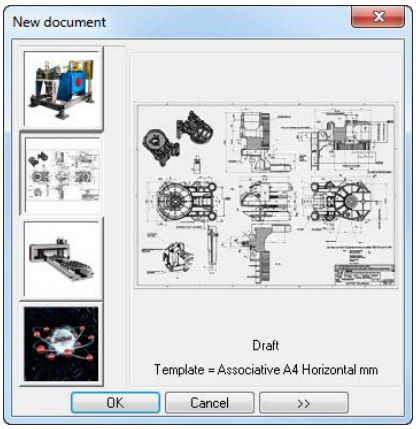
Creating a draft

Create a new draft document

- Create a **new document**.
- Select a **Draft** type document.
- In the **Advanced parameters**, >> select a standard **Associative A4 Horizontal** template.

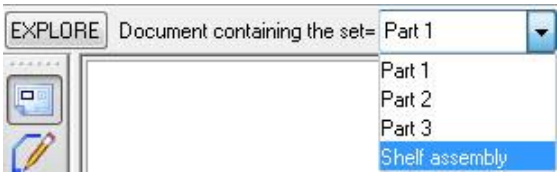


- Click **OK** to confirm.



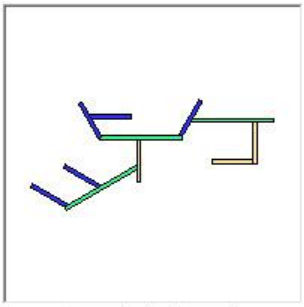
Position the main view

- Start the **View** context and select the **Main view** function.
- Select **Assembly** to draft the entire document. ASSEMBLY
- In **Pick on the document containing the assembly**, select **Shelf assembly** in the dropdown list.



- Configure the main view by positioning the green arrows as shown opposite.
- Set the **smooth edges** to **Hidden**, and the **hidden lines** to **Stipple** and click **OK** to confirm.

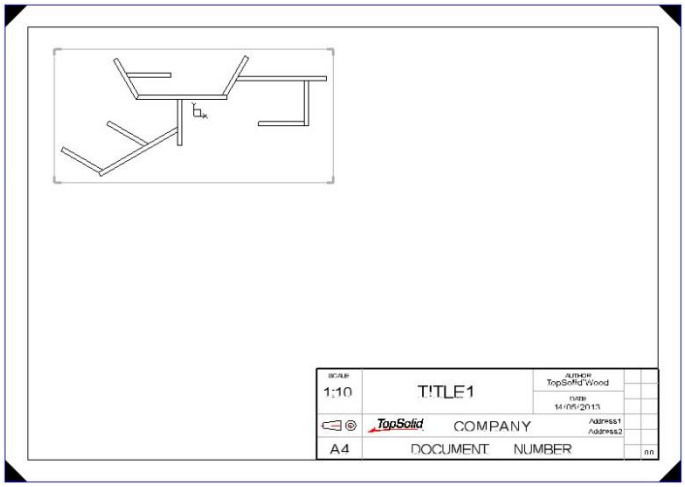
- Position the view.



Edit the draft

- Use **Modify element** and select the frame of the sheet. Deactivate the **Center mark**, **Orientation mark**, **Coordinate system** and **Graduations** tick boxes.
- Set the **scaling factor** to 0.1.

- Click **OK** to confirm.
- Use **Regenerate**, then select the view to update it.
- Use **Move parents** to move the view to the top left-hand corner of the draft.
- **Save** the draft using the disk icon.

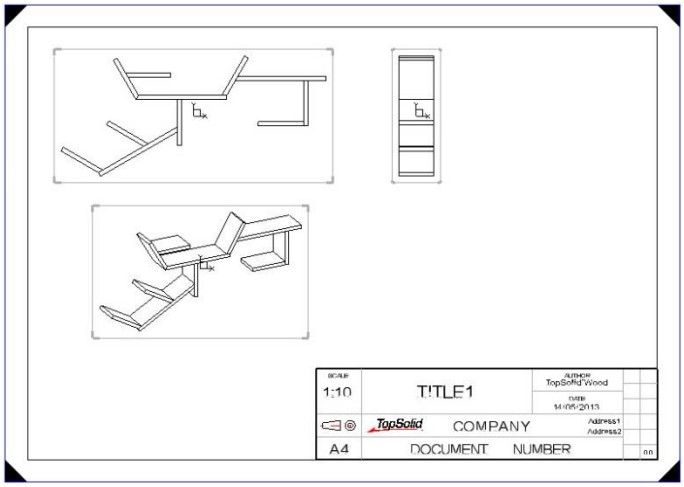


Position auxiliary views

- Use the **Auxiliary view** function.
- Configure the **smooth edges** and **hidden lines** as **Hidden**.

- Position the view with **Auxiliary view.**
- Position this view to the right of the main view.
- Select the main view as the **reference view**.

- Set the **smooth edges** and **hidden lines** to **Hidden**, then place the view using **Auxiliary view.** Position this view beneath and to the right of the main view to generate a perspective view.
- Use the **Move parents** function.
  Move the perspective view under the main view.



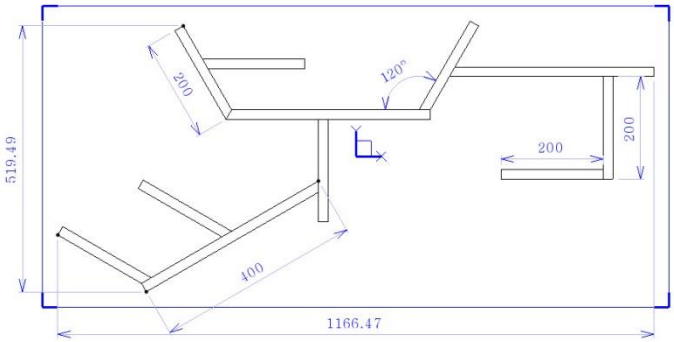
Dimension the views

- Start the **Dimension** context. 
- Use **Fast dimension**  to apply the various dimensions to the drawing:
  - depth of the shelf;
  - length and total height of the shelf;
  - dimensions of the parts.

Note: If a dimension must rest on a point, use the **Point** option in the dialog bar to select a point. POINT

If a dimension is positioned between two points, the **orientation** can be changed in the dialog bar.

Orientation= VERTICAL



Fill in the title block

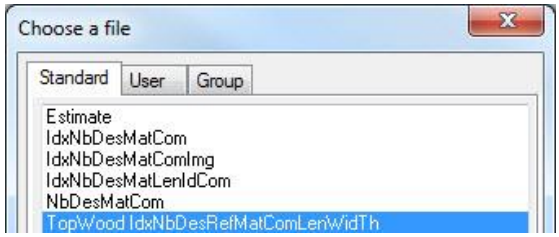
- Use **Modify element**  to change the title block text **Title 1** to *Shelf.*
  Replace **Company** by *TopSolid'Wood* and **Document number** by *Document 1/1.*
- Use **Delete element**  to delete **Address 1** and **Address 2.**
- **Save** the document.

SCALE 1:10	Shelf	AUTHOR TopSolid'Wood DATE 14/05/2013	
	TopSolid TopSolid'Wood		
A4	Document 1/1		00

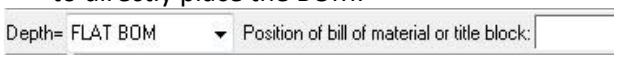
Supplement: Adding elements to the draft

Add a BOM

- Start the **Bill of materials**  context, then execute **Bill of materials**. 
- In the **Standard** tab, select the BOM **TopWood IdxNbDesRefMatComLenWidTh**.



- In **Designate a 2D view**, select the main view of the document.
- Set **Depth = Flat BOM** and in the **Position of bill of material or title block** field, select the document title block to directly place the BOM.



3	3	Shelf part 1	SP-1	Pear	-	400.0	150.0	19.0	400.0	150.0	19.0
2	3	Shelf part 3	SP-3	Pear	-	200.0	150.0	19.0	200.0	150.0	19.0
1	5	Shelf part 2	SP-2	Pear	-	200.0	150.0	19.0	200.0	150.0	19.0
!NDEX	NB.	DESIGNATION	REFERENCE	MATTER	COMMENT	LENGTH(FP)	WIDTH(FP)	THICKNESS(FP)	OVERLENGTH(CU)	OVERWIDTH(CU)	OVERTHICKNESS(CU)

Add coordinates

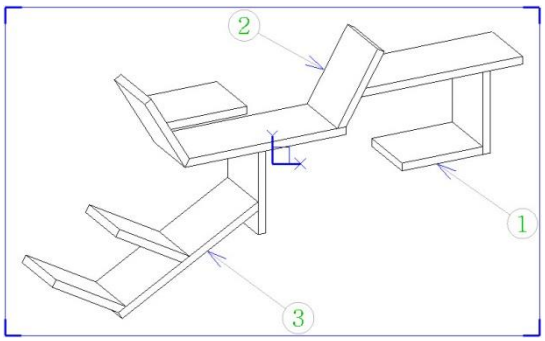
- Use **BOM index**  in the **Bill of materials** context.
- In the **Element to index** field 

, select a part to be indexed in the views. Place the index.

The BOM index is generated automatically and the BOM is completed accordingly.

- Position indexes to index the three parts.

Note: BOM indexes can be placed on perspective views.



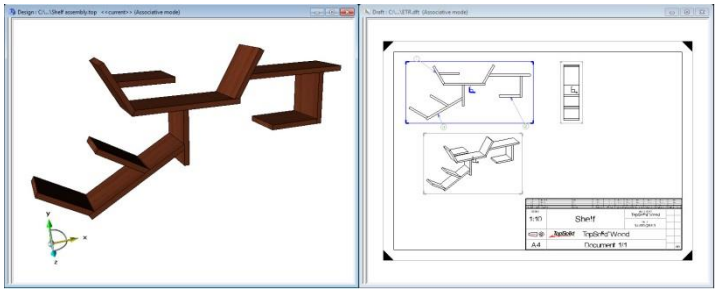
2	3	Shelf part 3
1	5	Shelf part 2
3	3	Shelf part 1
INDEX	NB.	DESIGNATION

Add a perspective view as the 3D

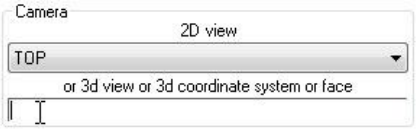
The steps described below place a realistic view in a draft as a 3D design view.

- With the draft file open, open the 3D design document.
- Open the **View**  tab and select **Configure view**  to configure the 3D view with conical perspective.

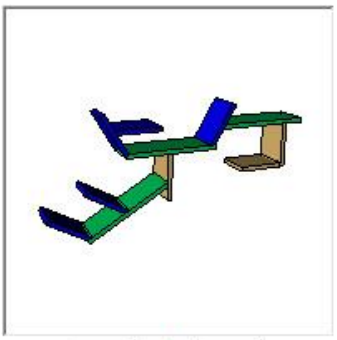
- In the list, select **Conical perspective**.
- Orientate the view, then **save** the file.
- Switch to **Tile vertically** in the **Window** menu.



- Click in the draft document to make it current.
- Activate the **View** context, then use the **Main view** function.
- Select **Assembly**, then click directly in the 3D design document to select the shelf assembly. ASSEMBLY
- In the view creation window that opens, click in the **or 3D view or 3D coordinate system or face** field.

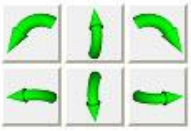


- The click in the 3D document space to configure the draft view as the 3D view.



Note: The draft view is a conical perspective view positioned as the 3D document.

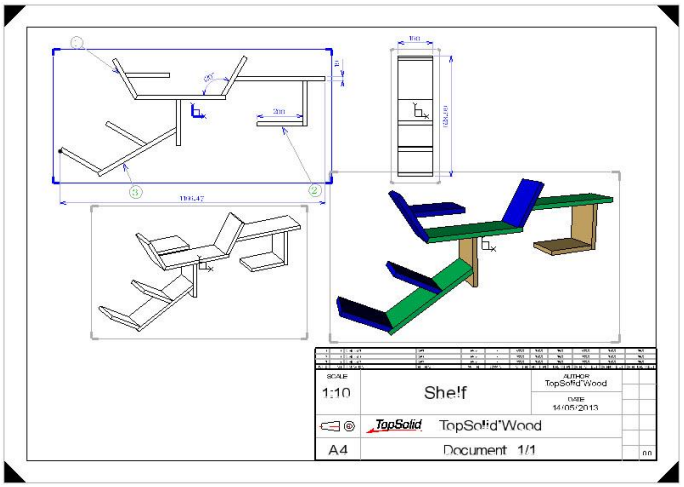
- Set the **smooth edges** and **hidden lines** to **Hidden**, then tick the **Shading view** box.



Note: The **Shading view** option displays the view in the design colors. Another setting in the document displays the shading view with realistic rendering.

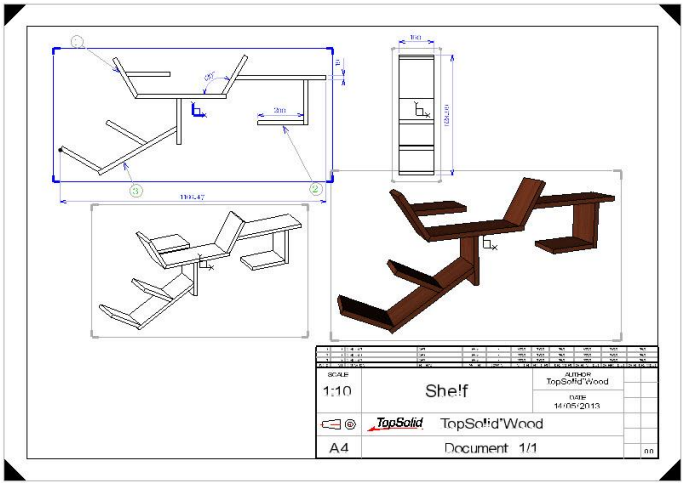
- Validate the window with **OK**, then place the view.
- Open the **document properties** of the draft.
- In the **Projection parameters** section, tick **Use realistic rendering**.

- Click **OK** to confirm.



The shaded view automatically changes to **realistic rendering.**

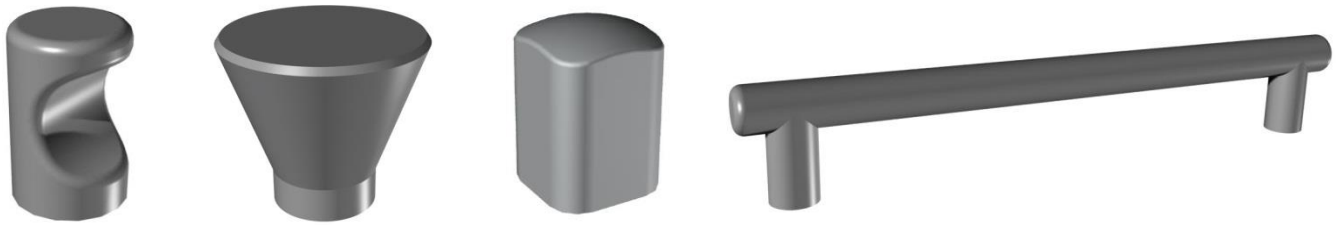
- **Save** the draft.



Note: This setting can be applied to a **Draft** document template so that it does not have to be applied to every draft.

Exercise 6: Basic shapes

Shapes are used to quickly design complex shaped parts. However, the shaping operations performed are not recognized as machining.
For example, shapes can be used to easily create hardware parts.



Cylindrical button

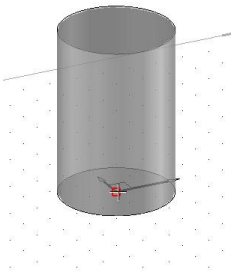
In this exercise, you will learn about:

- Creating cylinders
- Using design colors
- Creating Cartesian points
- Subtracting shapes
- Fillets on shapes

Create the first cylinder

- Create a **new document** of the **Design** type and select **Without template**.
- In the context bar, activate the **Shapes** context. 
- Create a **cylinder**. 
- Enter a **diameter** of 20mm.

- Select the **Z+** direction.

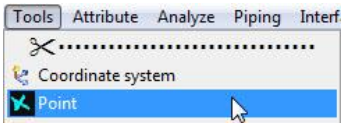


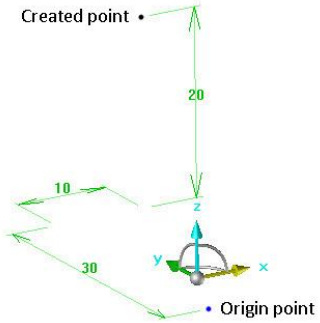
- Select **Alignment = Normal** and enter a **height** of 30mm. Press **Enter** to confirm.
- Select the point of origin of the absolute coordinate system to place the cylinder.

Create the second cylinder

- Change the **design color** using the black tab of the **Color** icon.
  Select **blue (12)**.

- Use the **Tools** > **Point** function.





- In the function bar, select **Cartesian point**.

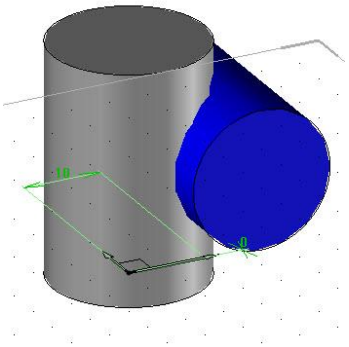
Note: The Cartesian point is used to create a point from another point by offsetting it by a distance in the X, Y and Z axes.

- Select the origin point of the absolute coordinate system as the **origin point**.

Then enter 10, 0 and 16mm as the **X, Y and Z coordinates**.
Place the dimensions of the point to X and Y and press **Esc** to exit the function.

- Start the **Cylinder** function.
  Create a new cylinder with a **diameter** of 18mm and in the **Y+ direction**.
  Select **Alignment = Centered** and enter a **height** of 30mm.

- Select the **Cartesian point** as the **alignment point**.



Subtract the cylinders

- Start the **Subtract** function.

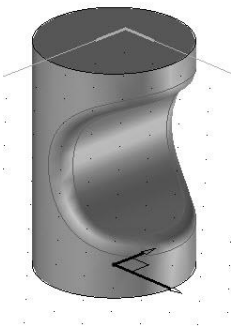
Note: The **Subtract** function is used to subtract one shape (**Tool**) from another shape (**Shape to be modified**).

- In **Shape(s) to modify**, select the first grey cylinder.

- Set **Hide tools = Yes** and **Fillet radius = 2**.

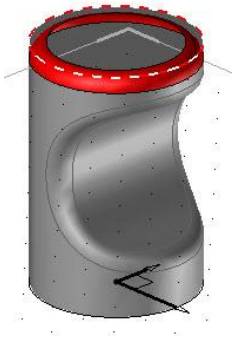
Note: **Hide tools** is used to automatically hide the parts used as tools for the subtraction.
Fillet radius automatically generates fillets on the edges created by the subtraction.

- In **Tool shape(s) to use**, select the blue cylinder to be subtracted from the grey cylinder.



Fillets

- Start the **Fillet** function. 
- Select **Fillet = One radius**, **Follow tangent edges = YES** and **Radius = 2mm**.
- In **Edge or face**, select the top face of the grey cylinder.



The fillets are shown in red before they are made.

- Create the fillets with the **Compute fillet(s)** option.

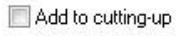
Define the part and the set

- Start the **Wood** context. 
- Start **Define** > **Define part** and select the newly created part. 

Note: Since this part is cylindrical, TopSolid'Wood does not automatically find any sawing axes.

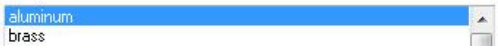
- Select **Z+** as the lengthways axis and **X+** as the width.
- In the **Part definition** window, enter the **designation** *Part*.

- Untick the **Add to cutting-up** box.

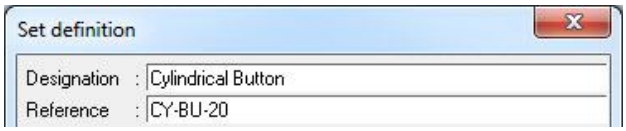


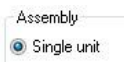
Note: This box is used to calculate the cutting-up dimensions of the parts and the material overvaluations. In the case of this hardware part, deactivating this box achieves better performance in the calculation of the BOMs.

- In the **Material** tab, select **aluminum** in the **metal** category.
- Click **OK** to confirm.



- Start the **Define** > **Define set**  function, then select **Characteristics**. 
- Enter the **designation** *Cylindrical button* and the **reference** *CY-BU-20*.
- In the **Assembly** section, tick the **Single unit** option.
- Click **OK** to confirm.





- **Save** this file in a new folder called *Handle shapes.*
- **Close** this file.

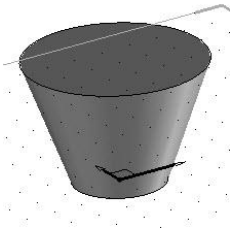
Conical button

In this exercise, you will learn about:

- Creating a cone
- Extruding on a face
- Uniting shapes
- Chamfers

Create the cone

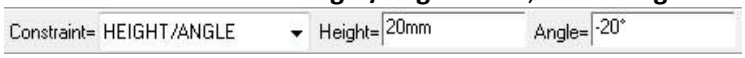
- Create a **new document** of the **Design** type and select **Without template**.
- In the context bar, activate the **Shapes** context. 
- Create a **cone**. 
- Select **Cone = Truncated** and enter a **diameter** of 15mm.



- Select the **Z+** direction.

- Select **Alignment = Normal**

- And in **Constraint = Height/Angle** mode, enter **Height = 20mm** and **Angle = -20°**

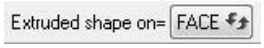


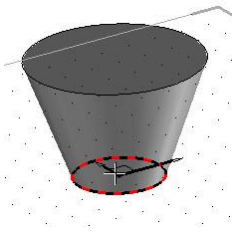
- Place the shape by selecting the absolute coordinate system origin point as the **alignment point**.

Note: When selecting a point, the **Enter** key can be used to automatically select the origin point of the current coordinate system.

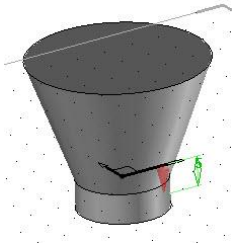
Extrude the base

- Start the **Create extruded shape** function. 
- Set **Extruded shape on = Face**.





- Select the base face of the cone as the **reference face**.



- Extrude this face downwards to a **height** of 5mm**.**

Unite the parts

- Use the **Unite** function. 
- Select the cone in **Shape(s) to modify**.

- In **Hide tools = YES** mode, select the previously extruded base in **Tool shape(s) to use**.

Note: The **Unite** function is used to merge two 3D shapes. One shape is created from the two selected shapes.

Make the chamfer

- Start the **Chamfer** function. 
- Set **Chamfer = Length/Length** and **First length = 2mm.**

- In **Edge or face**, select the top face of the cone.

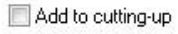
The chamfer is shown in red before it is created.

- Create the chamfer with the **Compute chamfer(s)** button.

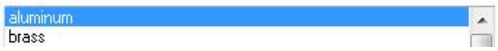
Define the part and the set

- In the **Wood**  context, start **Define** > **Define part** and select the newly created part. 
- Select **Z+** as the lengthways axis and **X+** as the width.
- In the **Part definition** window, enter the **designation** *Part*

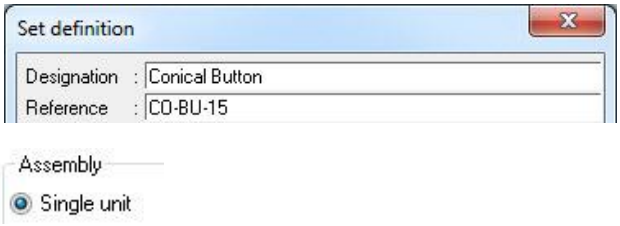
- Untick the **Add to cutting-up** box.



- In the **Material** tab, select **aluminum** in the **metal** category.
- Click **OK** to confirm.



- Start the **Define** > **Define set**  function, then select **Characteristics**. 
- Enter the **designation** *Conical button* and the **reference** *CO-BU-15*.
- Set the assembly to **Single unit** mode.
- Click **OK** to confirm.
- **Save** this file by renaming it *Conical button*, then **close** the file.



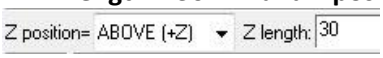
Square button

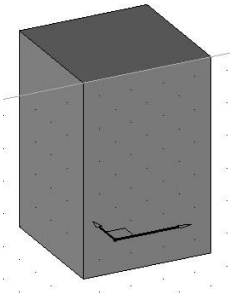
In this exercise, you will learn about:

- Creating a block
- Drawing an offset point
- Creating spheres
- Limiting by shape

Create the block

- Create a **new document** of the **Design** type and select **Without template**.
- In the context bar, activate the **Shapes** context. 
- Create a **block**. 
- Enter:
  - **X length = 20mm** and **X position = Centered**.
  - **Y length = 20mm** and **Y position = Centered.**
  - **Z length = 30mm** and **Z position = Above (+Z)**.

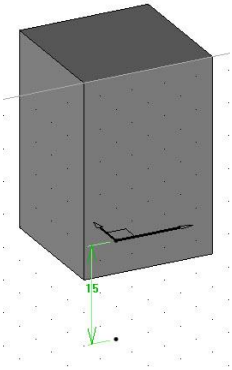




- Select the origin of the absolute coordinates in **alignment point**.

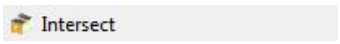
Create the sphere

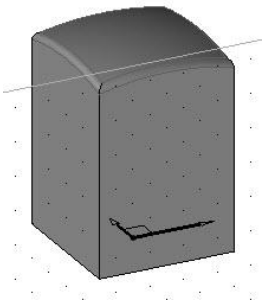
- Create a **point** that is **offset** from the origin of the absolute coordinates in the **Z-direction** at a **distance** of 15mm using **Tools** > **Points** > **Offset point**. 
- Create a sphere using the **Sphere** function in the **Shapes** context. 
- Enter a **diameter** of 90mm and select the newly created point in **Center point.**



Limit the block

- Execute the command **Shape** > **Surfacic operations / Booleans** > **Intersect**.



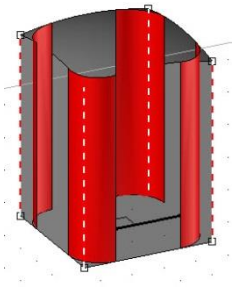


- Select the initial block in **Shape(s) to be limited**.
- Set **Hide tools = YES** and **Fillet radius = 1**.

- Select the sphere as the **tool shape**.

Make the fillets

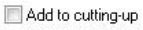
- Create **fillets** with a 5mm **radius** on the four vertical edges of the block. 



Define the part and the set

- In the **Wood**  context, start **Define** > **Define part** and select the newly created part. 
- In the **Part definition** window, enter the **designation** *Part*.

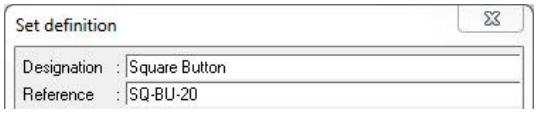
- Untick the **Add to cutting-up** box.



- In the **Material** tab, select **aluminum** in the **metal** category.
- Click **OK** to confirm.

- Start the **Define** > **Define set**  function, then select **Characteristics**. 

- Enter the **designation** *Square button* and the **reference** *SQ-BU-20*.
- Set the assembly to **Single unit** mode.
- Click **OK** to confirm.
- **Save** this file by renaming it *Square button*, then **close** the file.



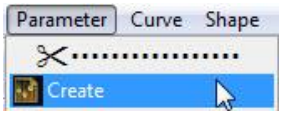
Supplement: Configured line handle

In this exercise, you will learn about:

- Creating/using parameters
- The notion of shapes again: cylinder, union, fillet

Create the parameter

- Create a **new document** of the **Design** type and select **Without template**.
- Start the **Create** function in the **Parameter** menu.



- Select **Unit type = Length** and enter a **value** of 128mm**.**Press **Enter** to confirm.

- In the **Name** field, enter *hl.* In the **Designation** field, enter *Handle length.*

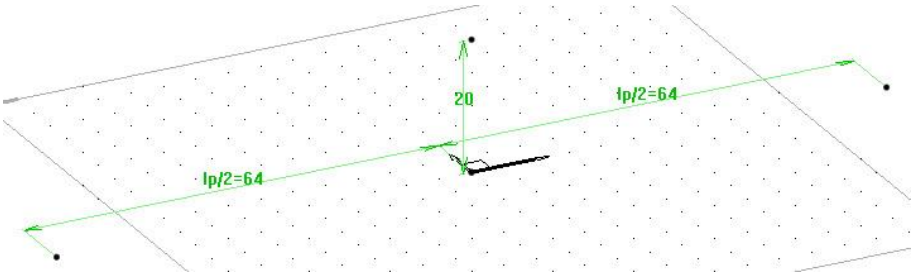
Note: The **name** is the system name of the parameter. This name will be used in the value fields or in expressions. The **name** must be simple and cannot contain any spaces.
The **designation** is what the user sees when using the parameter. Therefore, it must be explicit in order to use the parameter and can contain spaces.

- Click **OK**  to confirm, then click **No text**. 
- Press **Esc** to exit the function.

Create the offset points

- To create an **offset point,** open the **Tools** > **Points** > **Offset point** menu. 
- Select the origin of the absolute coordinate system as the **origin point**, **X+** as the **direction**, and the expression *hl/2* as the **distance**.

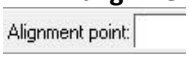
- Apply the dimension, then repeat the operation to create an **offset point** from the origin of the absolute coordinates in the **X-** direction by a **distance** of *hl/2*.
- Finish by creating an **offset point** from the origin of the absolute coordinate system in the **Z+** direction by a **distance** of 20mm**.**

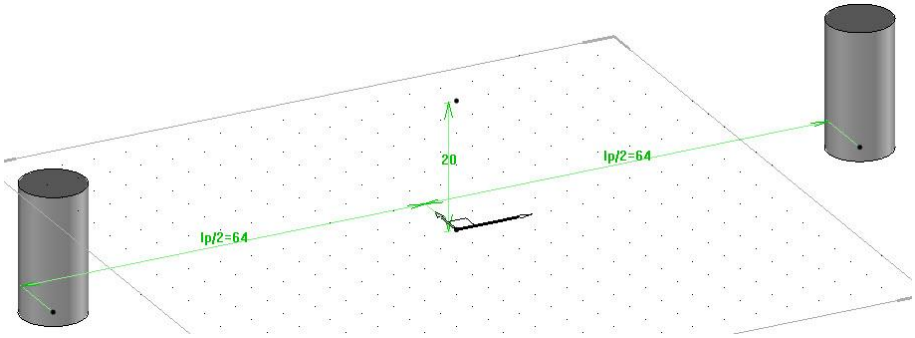


Create the cylinders

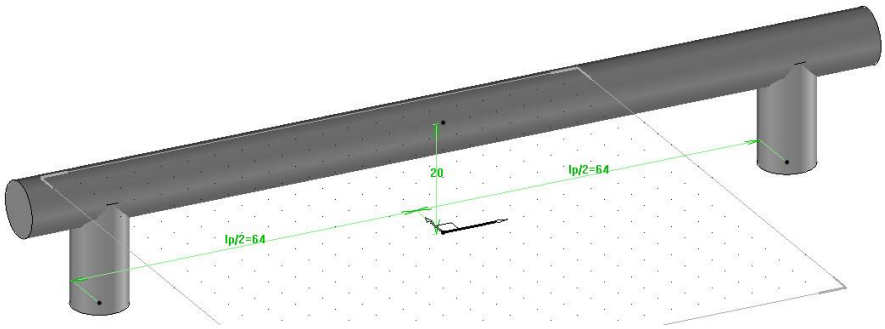
- In the **Shapes**  context, create a **cylinder**  with a **diameter** of 10mm, in the **Z+ direction** and a **height** of 0mm in the **Alignment = Normal** mode.

- In **alignment point**, select the two **offset points** in the X axis created previously.





- Create another **cylinder** with a **diameter** = 10mm, in the X+ **direction** and a **height** = hl+30 in **Alignment = Centered** mode.
- In **alignment point**, select the **offset point** towards **Z+**.

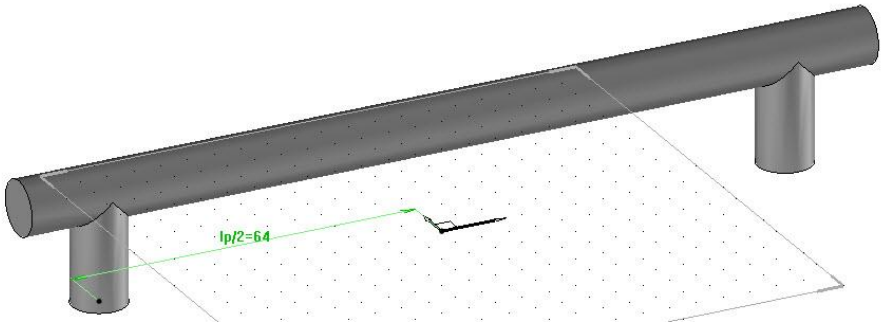


Unite the cylinders

- Use **Unite**  to unite the left-hand cylinder (**Shape to modify**) and the upper cylinder (**Tool to be used**). Use the **Hide tools = YES** mode.

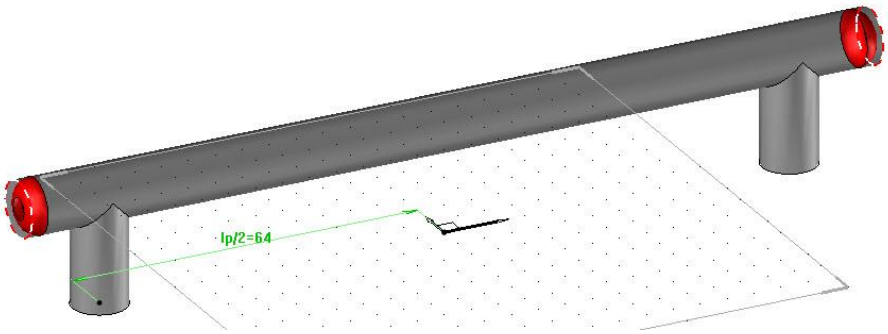
- **Unite** the two previously united elements (**Shape to modify**) with the right-hand cylinder (**Tool to be used**).

The three cylinders now form a single part.



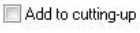
Make the fillets

- Create the **fillets** with a **radius** = 3mm on the two end faces of the upper cylinder.

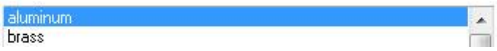


- Validate the fillets with the **Compute fillet(s)** button.

Define the part and the set

- Use **Define part**  in the **Wood**  context to define the part created.
- Select **X+** as the **lengthways axis**  and **Y+** as the **width axis**. 
- In the **Part definition** window, enter the **designation** *Part*
- Untick the **Add to cutting-up** 

- In the **Material** tab, select **aluminum** in the **metal** category.
- Click **OK** to confirm.



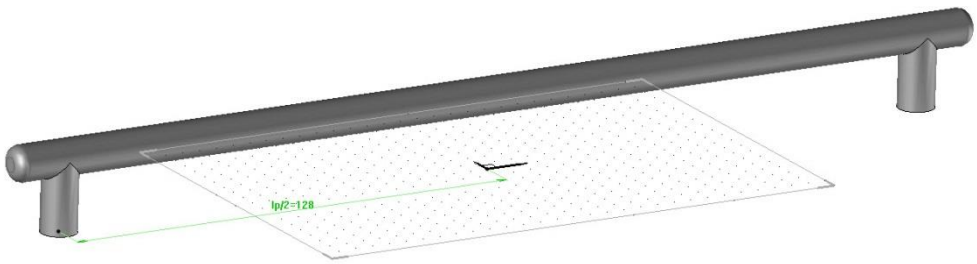
- Start the **Define** > **Define set**  function, then select **Characteristics**. 

- Enter the **designation** *Line handle* and the **reference** *HA-LI*.
- Set the assembly as **Single unit**
- Click **OK** to confirm.
- **Save** this file by renaming it *Line handle*, then **close** the file.

Vary the length of the handle

- Start the **Modify parameter** function. 
- In **Parameter to modify**, enter *hl,* then press **Enter** to validate.

- Change the **nominal value** of the parameter *hl,* then press **Enter** to validate.



Exercise 7: Building a bottle rack

In this exercise, we are going to make the bottle rack.

Concepts addressed:

- Same length constraint
- Sketch copy
- Part duplication
- Pocket and drilling operations

Making the supports

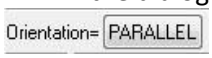
Draw the sketch

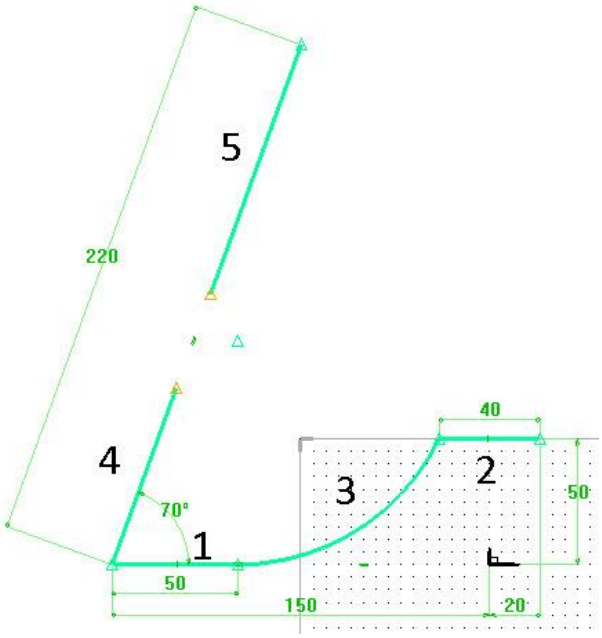
- Create a **new document** of the **Design** type and select **Without template**.
- Start a **new sketch**.

- Draw and dimension lines 1 and 2 as shown opposite.
  - Lines 1 and 2 are oriented along the X axis.
  - Line 1 is aligned with the X axis.
- Draw the circle arc 3.

Apply a **tangency** constraint with line 1.

- Draw lines 4 and 5.
  - Line 4 is **dimensioned** at a 70° angle relative to line 1.
  - Line 5 is **aligned** with line 4.
  - **Dimension** the distance between the two points to 220mm. To set the dimension parallel to the lines, in the dialog bar set **Orientation = Parallel**.





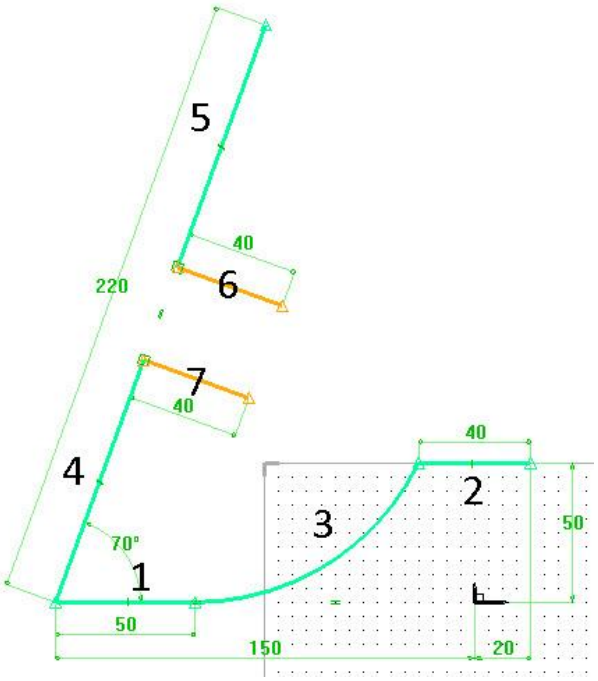
- Open the **Constraint** function, then select the **Same length** constraint.
- Select line 4, then line 5.

Note: It is possible to select several segments to be constrained to the **same length**.

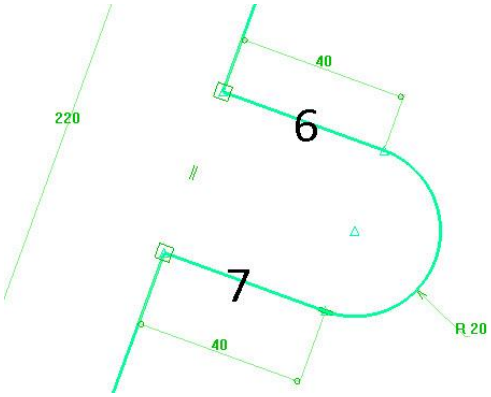
- Click on **Stop** to finish.

Draw the additional lines

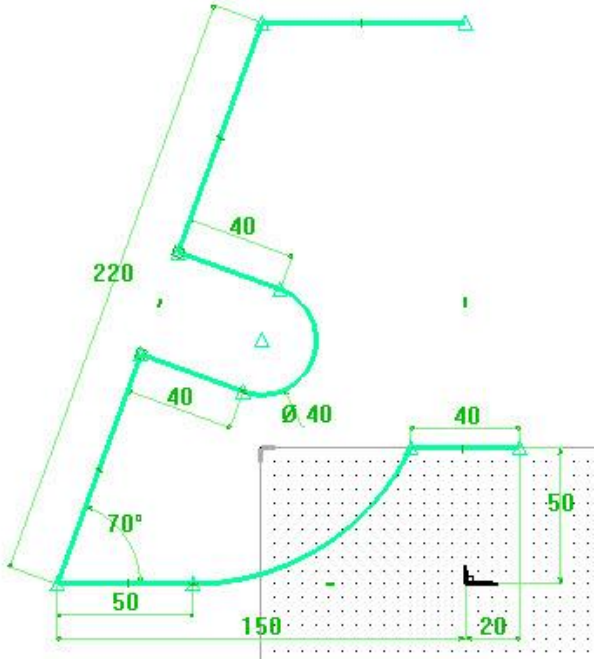
- Draw lines 6 and 7.
- Apply a **perpendicularity** constraint to lines 6 and 7, relative to lines 5 and 4.
- **Dimension** lines 6 and 7 to a length of 40mm.



- Then draw a **circle arc** between lines 6 and 7.
- **Dimension** this circle arc to a **radius** of 20mm.
- Then apply a **tangency constraint** between this arc and line 7.



- Finish by drawing the upper line, as shown opposite.
  This line is **oriented along the X axis** and the right-hand point of the line is **aligned** with the **Y** axis.



Copy the elements of the sketch

Note: The **Copy sketch** function is used to copy segments of a sketch with a transformation.
Copy sketch can copy the **orientation constraint** applied the the copied segment at the same time.
Copy sketch also copies the dimension constraint of the segment (length, radius, etc.).
The geometric constraints are automatically copied only if all of the constraint segments are selected.

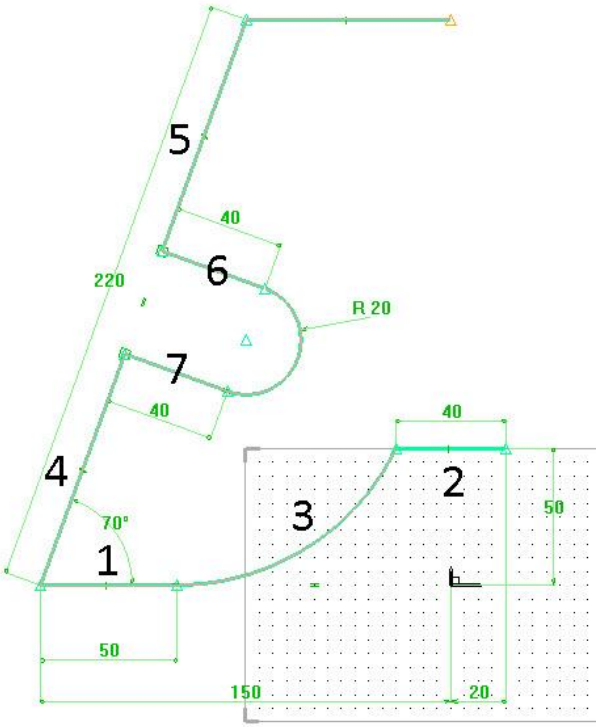
- Start the **Copy** function. 
- Set **Duplicate orientation constraints = YES**.

Note: In certain functions, it is possible to select several graphical elements and apply the function to them. The **selection** lasso is active in these cases. 

- Start the **selection.** 
- In **Template elements to repeat**, select segments 1, 3, 4, 5, 6 and 7.

Note: The selected elements are highlighted in grey.

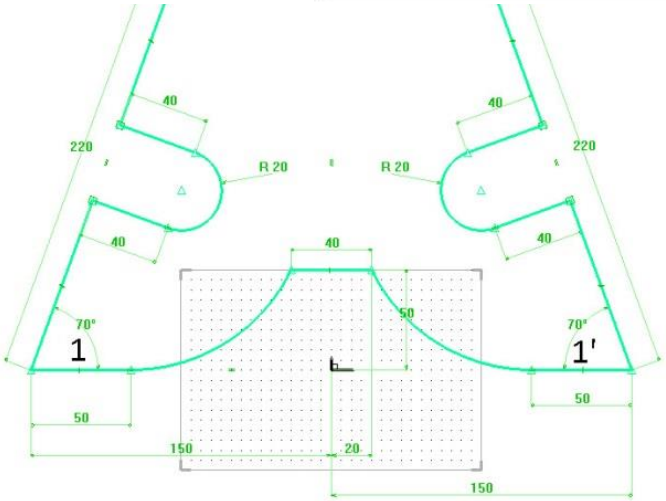
- Click on **OK** to confirm the selection. 
- Select **simple mirror** in the dropdown list in **propagation**.
- Select the **YZ** plane as the **plane of symmetry**. 



All the selected segments are copied by **simple mirror**. The dimensions associated with the segments and the dimensions and constraints between the segments are also copied.

- Finish by **dimensioning** the distance between the right-hand point of segment 1' and the **Y** axis of the absolute coordinates at 150mm.

Note: The **Move parents** function can be used to reposition the dimensions on the plane. 



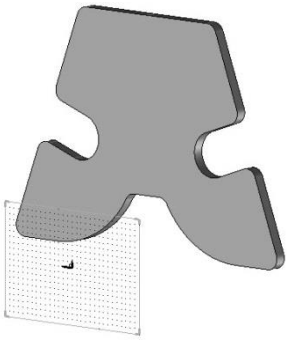
Make the fillets

- Use **Fillet** to make fillets with a **radius** = 10mm on the 10 angles in the sketch.

Fillet radius= 10mm Curve to modify: > COMPUTE FILLET(S)

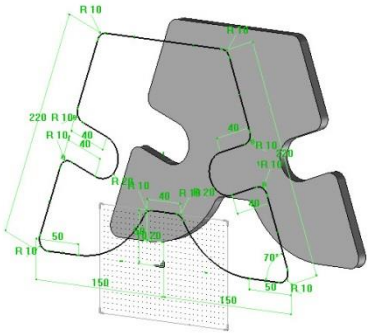
- **Finish the sketch**.
- **Save** this document in a new folder *Bottle rack,* then rename the file *Bottle rack.*

Extrude the sketch



- Extrude the sketch in the **Z+ direction**. DIRECTION > Z+
  Enter an **offset from starting curve** of 100mm in the advanced parameters.
  >> > Offset from starting curve= 100mm
  Enter a **height** =10, then press **Enter** to confirm.
  Height: 10

Modify the sketch

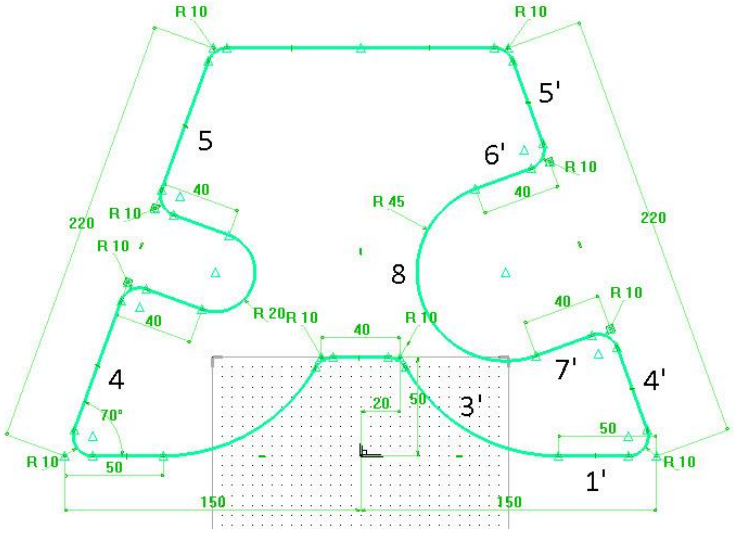


Note: Once a drawing element has been used (e.g., when a sketch is used to extrude a part), it is automatically hidden. Use the **Driving elements** function to display the elements used by another element.

- Open **Driving elements**, then select the extruded part.
- Use **Modify element** to modify the displayed sketch.
- Use **Modify parameter** to modify the **radius** of the circle arc 8 to the **nominal value** = 45mm.

Nominal value: 45mm

- **Finish the sketch**. The extrusion of the part is automatically recalculated.
- Use **Driving elements** again on the extruded part to hide the sketch.



Duplicate the support

Note: **Duplicate** creates a copy of an existing part. It can be used to create two parts with an identical basis, which are then distinguished as a left-hand side and a right-hand side with different machining operations.
Two duplicated parts can be defined differently.

- Start the **Edit** > **Duplicate** function.
- Select **Rotation** ROTATION, then the **Y+** axis as the **axis of rotation**. Y+
- Enter a **rotation angle** of 180°, then press **Enter** to confirm.

Rotation angle or first point: 180

- Set **Follow = Existing operations**.

Follow= EXISTING OPERATIONS

Note: **Follow existing operations** applies only the existing operations to the duplicated part.
Follow subsequent operations is used to apply subsequent operations to the duplicated part.

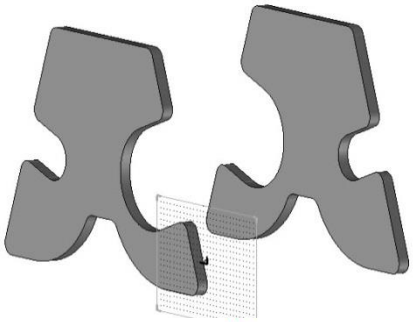
- Leave the **Layer number or name** field empty.

Layer number or name=

Note: **Layer number or name** is used to change the destination layer of the duplicated part.

- Select the previously extruded part in **Elements to duplicate**.

Elements to duplicate:

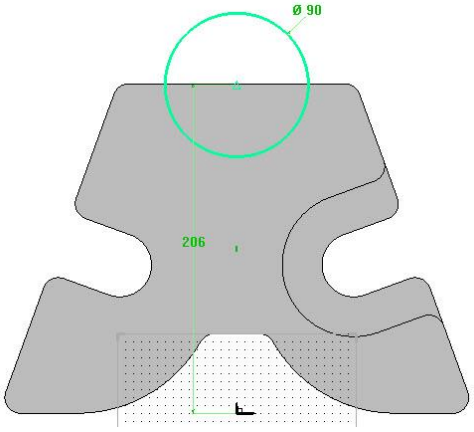


Machining the supports

Make a sawing

- Start a **new sketch**.
- Position a circle with a **diameter** of 90mm.

Diameter = 90mm

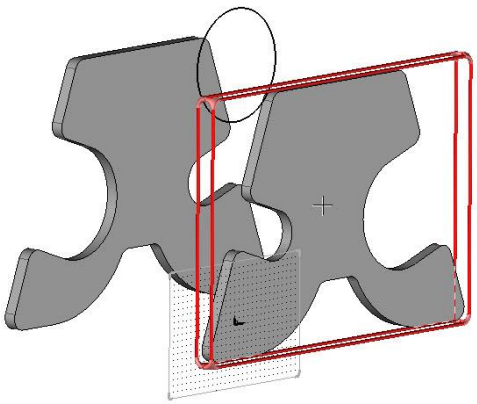


- Apply an **alignment constraint** between the center of the circle and the **Y** axis.
- **Dimension** the center of the circle with the **X** axis at an offset of 206mm.
- **Finish the sketch.**
- Start the **Wood** context, then select **Sawing**.
- Select one of the two supports in **Shape(s) to saw**.

Shape(s) to saw:

- Select the drawn circle in **Sawing path curve**.

Sawing path curve:



The red arrow represents the offcut side.

- Position the offcut inside the circle by clicking on it and using the **Invert direction**

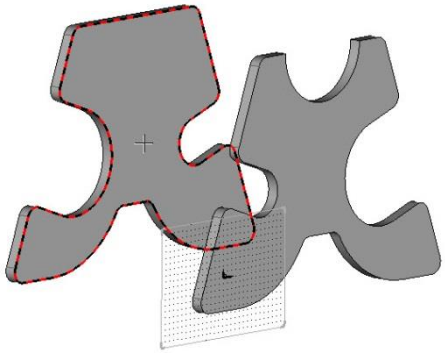
The arrow points to offcut: INVERT DIRECTION

- Click on **OK** to cut.

Make a drilling

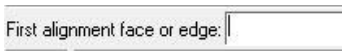
- Start the **Drilling** function. 
- Set **Coordinate system = Constraint** and **Mode = Non dynamic**.

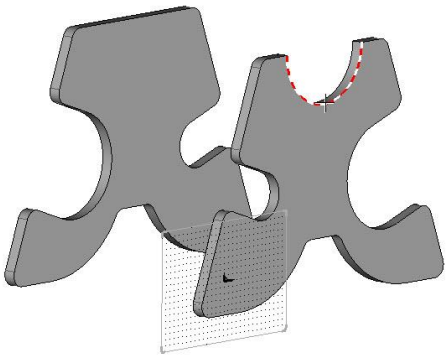
Note: **Coordinate system = Constraint** places the drill hole in relation to **dimensions** or **constraints**.
With **Mode = Non dynamic** it is not necessary to apply dimensions directly to the drill hole. **Mode = Dynamic** automatically places the drill hole with the dimensions on the nearest edges.



- In **Face to drill**, select the inner face of the unsawn part.

- Then, in **First alignment face or edge** select the edge of the sawing circle arc.





Note: When selecting the edges for a drilling operation, selecting a circle allows the drill hole to be positioned automatically in the axis of the selected circle.

- In the **Drilling models** window, select **hole** in **Standard models**, then click **OK** to confirm.

- Select **Hole - Through one**, then enter a **diameter** of 40mm.

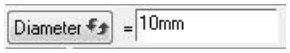
- Click on **OK** to confirm.

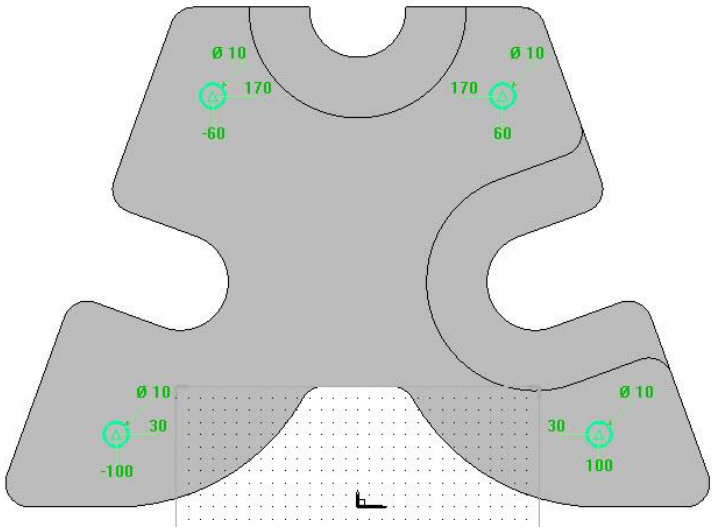
Making the support rods

Draw the sketch

- Make **layer 1** current.

- Start a **new sketch**, then position four circles with a **diameter** = 10mm as shown opposite.





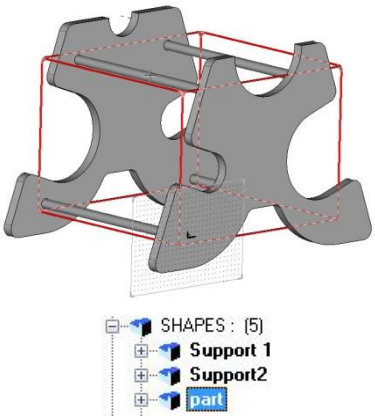
- Use **Dimension** to dimension the positions of the four circle centers:
  - Left bottom: ***X=-100; Y=30***
  - Right bottom: ***X=100; Y=30***
  - Right top: ***X=60; Y=170***
  - Left top: ***X=-60; Y=170***

Extrude the rods

- **Finish the sketch.**
- Start the **Create extruded shape** function.
- Use the **Sketch = Global** mode and **Result = One shape per curve**.

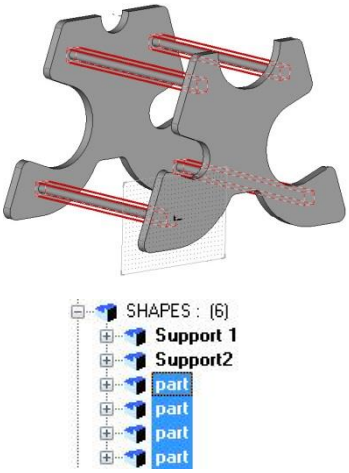
Note: **Sketch = Global** is used to extrude any sketch in one go.

In this case, **Result = One shape per curve** is used to generate four distinct shapes.
In **Result = One shape** mode, only one shape is produced.



<= **Result = One shape**

Result = One shape per profile =>



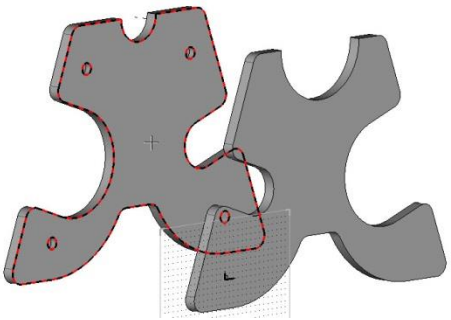
- Select one of the circles in **Section curves or texts**.
- In the dialog bar, set **Alignment = centered**, enter a **height** of 210mm, and press **Enter** to confirm.

Drilling the faces

- Start the **Drilling** function. 
- In **Face to drill**, select the lower face of a support.
- In **First alignment face or edge**, select one of the four extruded cylinders.
- In the drilling model window, select **Hole** in **Standard models**.
- Tick **Save as default**. This saves the drilling values for the following drilling operations.
- Click on **OK** to confirm.

- Configure a **blind hole** with a **diameter** = 10mm, a **depth** = 6mm and a **bottom angle** = 0°.

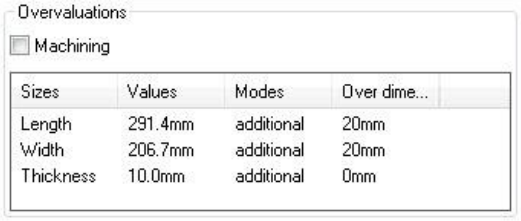
- Click on **OK** to confirm.
- Repeat the operation to make four drill holes for the rods on this surface, then four drill holes for the rods on the opposite face.

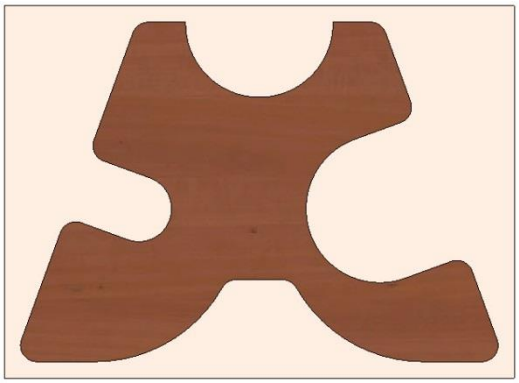


Definition of the parts and the assembly

Define the supports

- Start **Wood** > **Define** > **Define part**, then define the two supports.
  - **Designation**: *Support 1 / Support 2*
  - **Reference**: *SUP1 / SUP2*
  - **Material**: **wood** > **lime**
- In the **Sawing-up** tab, tick the **Add to sawing-up** box.
- In the **Over dimensions** fields, add the **length** and **width** 20mm lines by double-clicking in the field, then press **Enter** to validate.



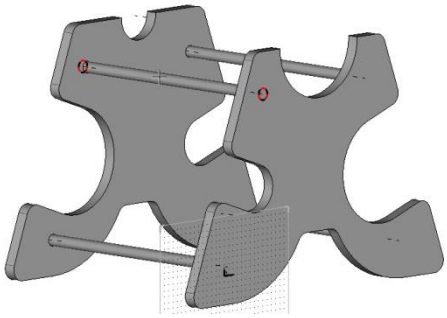


Note: The sawing over dimensions can be used to add more dimensions to the part's enclosing dimensions for the sawing-up of the material.
The rough can be configured in the **Rough** tab in **Define part**.

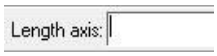
Define the rods

- **Define** the four supporting rods.

Note: Since these parts are cylindrical, users must select their axes manually.



- In **Length axis**, select the cylinder of the rod currently being defined. This means that the length of the rod will be the axis of the cylinder.



The axis is shown by a **red arrow.** Click on **OK** to confirm.

- Select **X+** as the **width axis**. 
- Then set:
  - **Designation**: *Support rod*
  - **Reference**: *ROD*
  - **Material**: **metal** > **aluminum**
- Click **OK** to confirm, then repeat the operation for all four rods.

Define the set

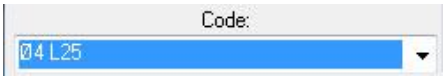
- Start the **Wood** >**Define** > **Define set** function, then select **Characteristics**.
- Enter:
  - **Designation**: *Bottle rack*
  - **Main set**: **Single Unit**
- Click on **OK** to confirm.

Supplement: Adding attaching screws

- Make **layer 2** current.

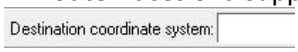
- Start the **Screw**  function in the **Wood** context.
- Select **metal screw** as the **standard component**.

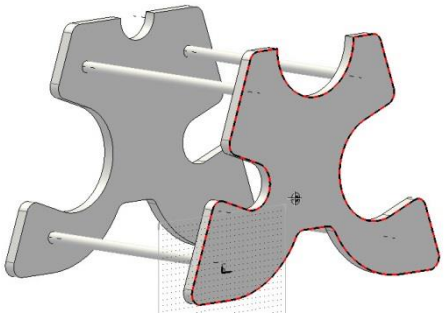
- Select the **code**: **Ø4 L25**.
- Click **OK** to confirm.



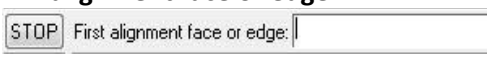
Note: The assembly function is used to place and automatically propagate screws between two parts.
In this case, the screws are placed individually.

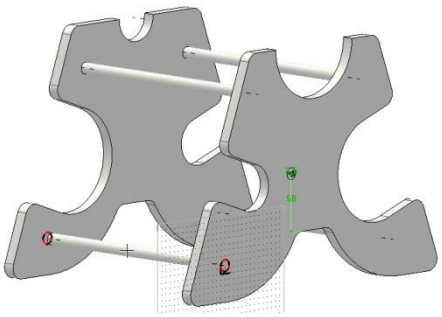
- In the dialog bar, select **Standard positionning**. STANDARD POSITIONNING
- In the **destination coordinate system** for the screw, select one of the outer faces of a support.





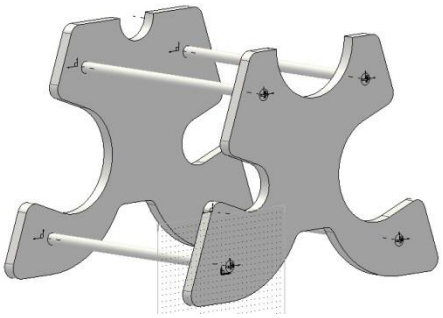
- As when positioning the drill holes, select the cylinder of a rod in **First alignment face or edge**.





- Click **Stop** to confirm the position of the screw. STOP
- Select **Automatic** to automatically machine the drill holes for the screw. AUTOMATIC
- Repeat the operations to place the eight attaching screws on the two outer faces of the supports.

- **Save** the file.



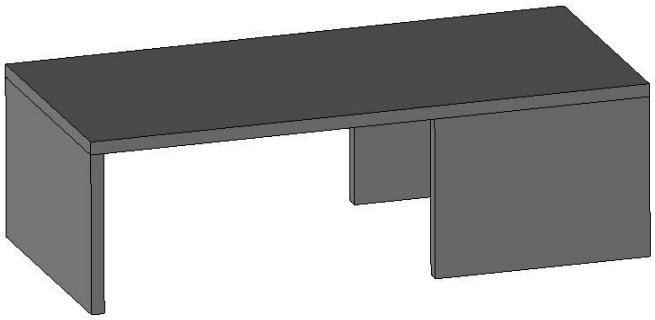
Exercise 8: Making a rectangular coffee table

The goal of this exercise is to make the rectangular parts without using a 2D drawing, and then to perform the wood machining operations.

Designing the table

Concepts addressed:

- Creation of a construction volume
- Using transparency
- Using constrained blocks



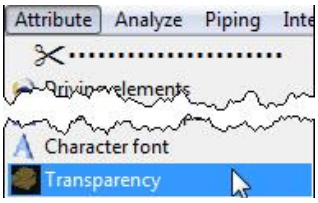
Create the construction volume

Starting with a construction volume for the design of the table will make it possible to design the parts quickly and easily.

- Create a **new Design document** and select **Without template**.
- Activate the **Shapes** context, then create a **block** with the dimensions **X = 1300, Y=600** and **Z=400.** Apply **Alignment = Centered** to all three dimensions.

- Select the origin of the absolute coordinate system in **alignment point** to place the block.

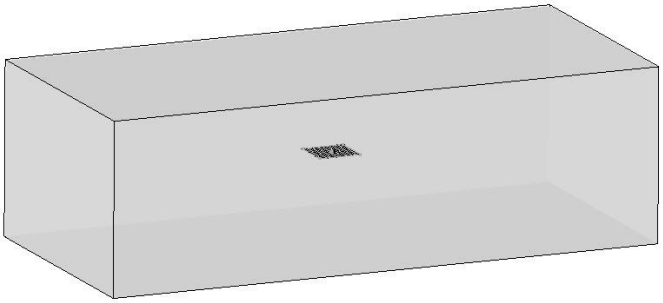
- Start the **Transparency** function in **Attribute** > **Transparency.**



- Select a transparency of **7**. 

Note: **Transparency** is defined on a scale of **1** to **10**. **10** corresponds to maximum transparency of the shape (only the edges are visible).

- Select the construction volume in order to apply the transparency.

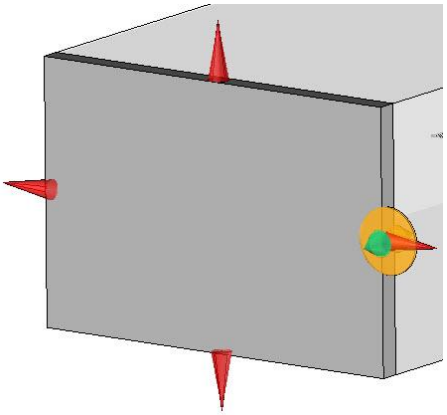


Create a part as a constrained block

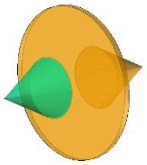
Note

Constrained blocks are used to design parts directly in 3D, without using a 2D drawing of the parts.
This allows to design parts simply, with a rapid and easy modification.

Constrained blocks are created using a **thickness**, four dimensioning **planes** and a **positioning plane**.



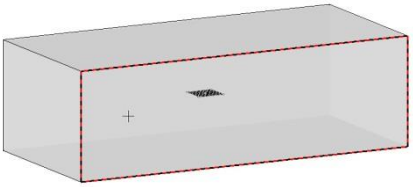
Arrows for the dimensioning **planes**. There are two for each dimension.



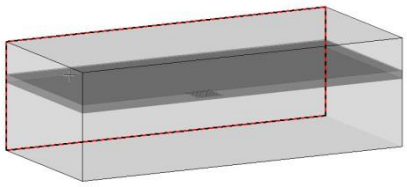
One arrow for the **positioning plane.** This is used to place the block on the right, in the center or on the left of the selected positioning plane.

- Make **layer 1** the current layer by clicking the thumbwheel on **1**. 
- Start the **Wood**  context, then select **Constrained block**. 
- Enter a **thickness** of 30mm.

- In **First plane**, select one side of the length of the construction volume.

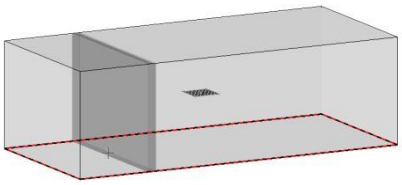


- Set **Mode = Faces**, then select the second lengthways side of the block in **second plane or point**.

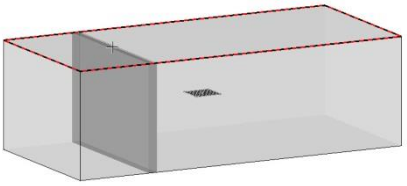


- If the block is placed horizontally, click **Switch direction** to place it vertically.

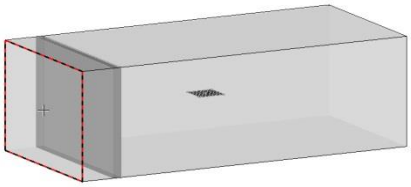
- Then select the lower face of the block in **First plane**.



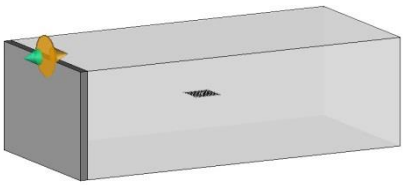
- In **second plane or point**, select the top face of the block in **Mode = Faces**.



- Then select the end face of the block in **positioning plane**.

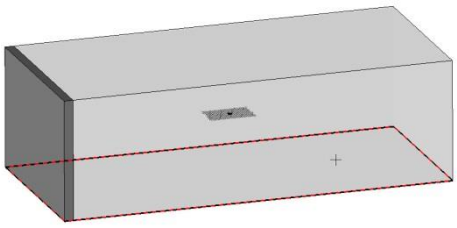


- Click on the yellow arrow on the left to place the **constrained block** on the right of the **positioning plane**.
- Validate the **constrained block** by clicking **OK**.

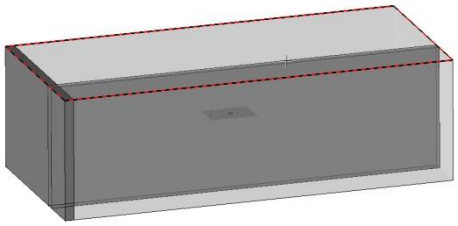


Create the other two bases

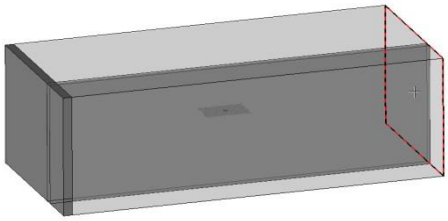
- For the next constrained block, enter a **thickness** of 30mm.
- In **First plane**, select the bottom face of the construction volume.



- In **Second plane or point**, select the top face of the construction volume.

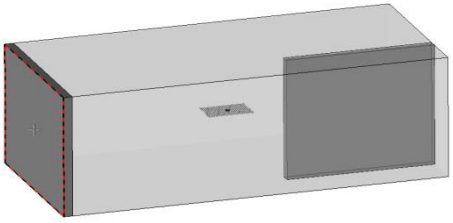


- The part must be placed lengthways in the block. If it is placed width ways, use **Switch direction** to change the direction of the part.
- In **First plane**, select the right-hand face of the construction volume.

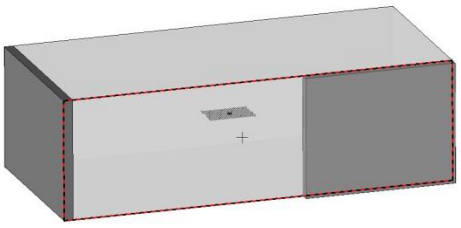


- In **Mode = Faces** , enter a **second shift** of 800mm.

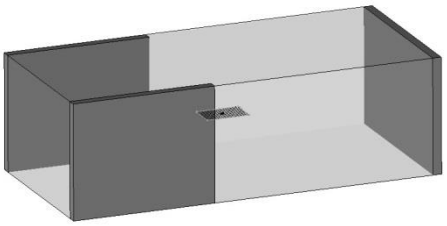
- In **Second plane or point**, select the left-hand face of the construction block.



- In **positioning plane**, select the front face of the construction block. Place this **constrained block** inside the construction block.



- Repeat this operation to create the second leg in the lengthways axis of the table.
- **Save** this file in a new folder called *Rectangular coffee table*, then rename the file *Rectangular coffee table.*



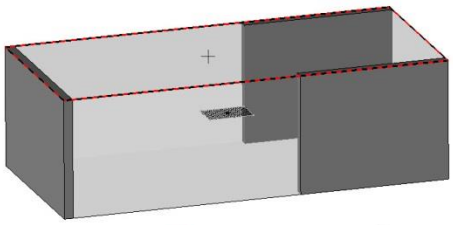
Create a constrained block automatically

- Make **level 2** current.

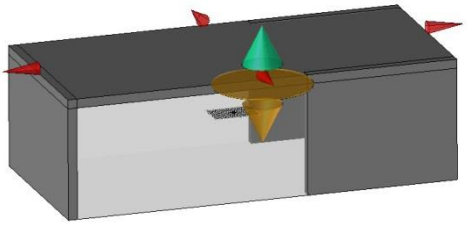
- In **Constrained block**, select **Automatic** mode in the dialog bar.

Note: **Automatic** mode automatically generates a rectangular constrained block on one face.

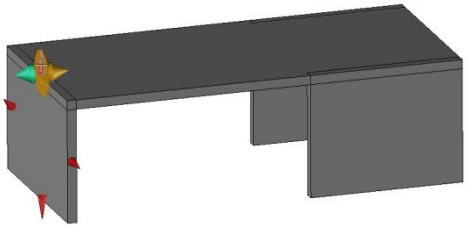
- In **Positioning plane**, select the top face of the construction volume.



- Select the top yellow arrow to place the constrained block inside the construction volume.
- Validate the constrained block by clicking **OK**.



Modify a constrained block

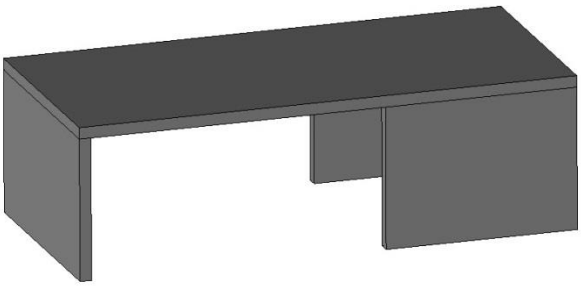


- Switch off layer **0**.
- Open **Modify element**, then select one of the faces of the left-hand base.
- Then select the upper red arrow.
- Enter a **second shift** of 30mm, then press **Enter** to confirm.

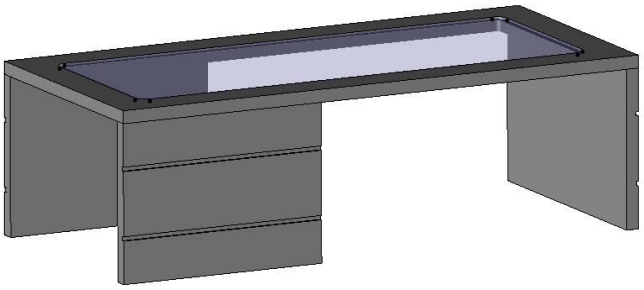
This allows the top face of the **constrained block** to be offset by 30mm.

- Validate the constrained block by clicking **OK**.

- Repeat this operation for the other two bases.



Performing the operations



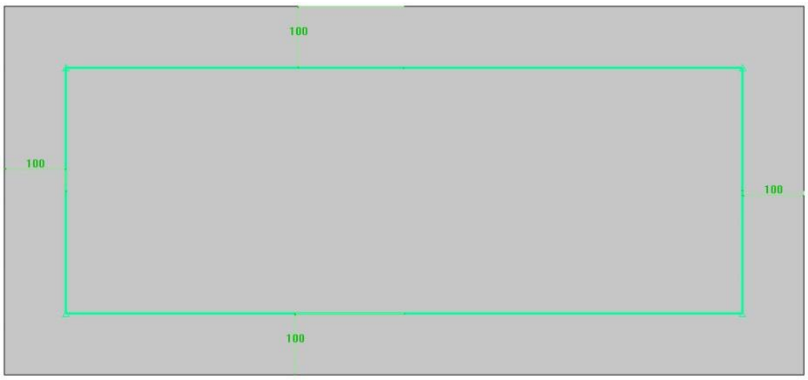
Concepts addressed:

- Sawing
- Rabbets
- Grooves

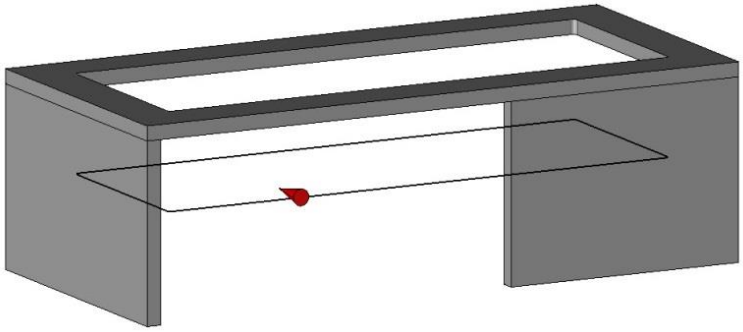
Draw the sketch and saw the top

- Activate the **Sketch** context, then start a **new sketch**.
- Create a **contour**, then select the **Rectangular** button. RECTANGULAR
- Draw a rectangle, as shown opposite.

- Then use **Dimension** to dimension the distances between the four sides of the rectangle and the four sides of the top. Set the **nominal value** to 100mm for each of the sides.



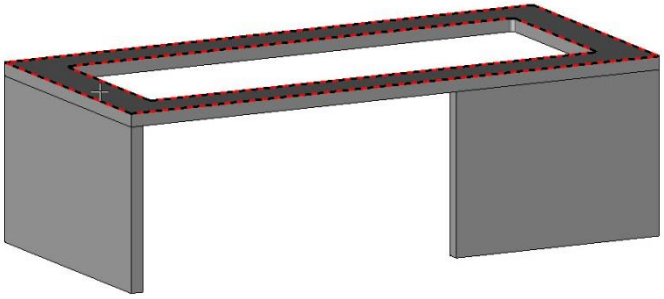
- Use **Fillet** to apply fillets with a **fillet radius** = 10mm to the four corners of the rectangle.
- **Finish the sketch.**
- In the **Wood** context, use the **Saw** function.
- Select the top of the table as the **shape(s) to saw**.
- Select the sketch drawn previously as the **sawing path curve**.
- Click on the red arrow so that it points towards the offcut.
- Click on **OK** to confirm.



Make the rabbets in the table top

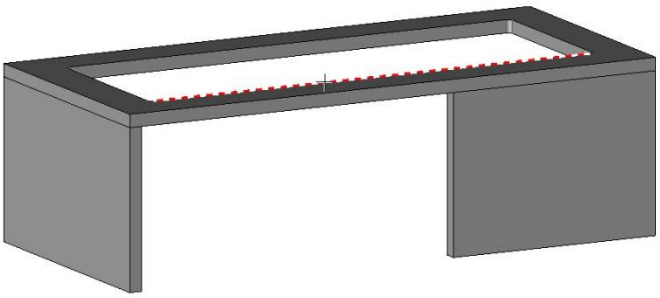
- In the **Wood** context, start the **Rabbet** function. 
- In **Sweep = Planar face** select the top face of the table top as the **reference face**.

Note: **Sweep = planar face** is the most commonly used mode. It allows the operation to be performed on a flat face.



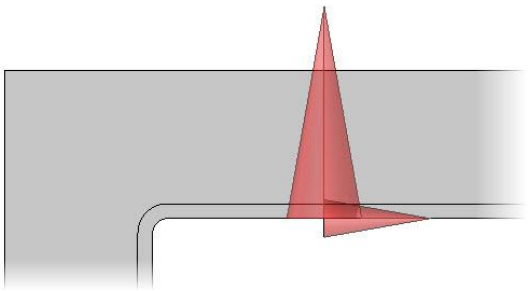
- Then set **Join edges = YES**, and **Follow tangent edges = YES**.

- Select one of the top edges of the saw cut as the **reference edge or curve for tool path**.



Note: Since all the edges of the face are tangential, they are all selected in one go.

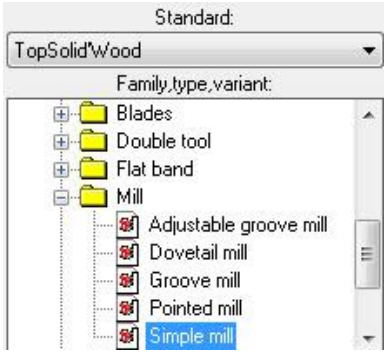
- Click **Stop** to confirm the path. 

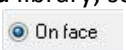


Note: The two red arrows represent the direction of machining and the machining side of the rabbet.

- Set the machining side of the rabbet towards the material of the part, as shown opposite.
- Click on **OK** to confirm.
- In the **Parameter** window, set:
  - **Tool type**: **Routers** 

  - In the **TopSolid'Wood** library, select **Simple mill**



  - **Parameters**: **On face** 

  - **Rabbet width**: 10mm

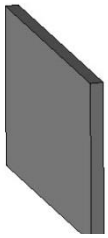
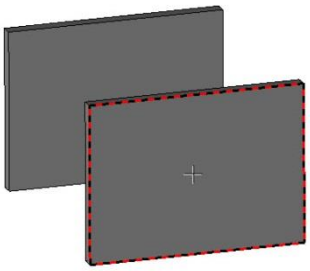
  - **Rabbet depth**: 5mm.

- Click on **OK** to confirm.

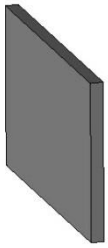
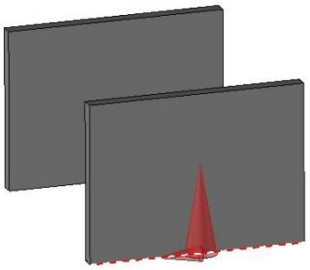
Create the grooves

- Make **layer 1** current, then switch off **layer 2**.

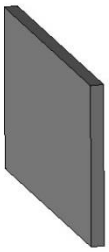
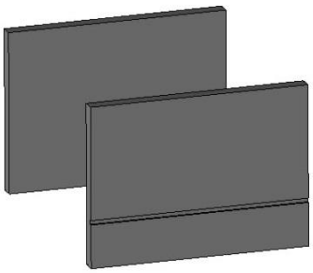
- Start the **Groove** function. 
- Select **Sweep = Planar face** and select the outer face of a leg as the **reference face**.

- Select the lower edge of the leg in the **Reference edge or curve for tool path** field.
- Click **Stop** to validate.
- Set the upward offset of the groove.
- Click on **OK** to confirm.

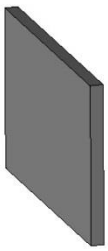
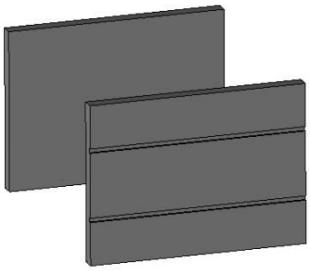
- Set:
  - **Tool type**: **Routers**
  - **Simple mill**
  - **Parameters**: **High arm**
  - **Gap distance** = 100mm
  - **Groove width** = 10mm
  - **Groove depth** = 5mm
  - **Angle** = 0°

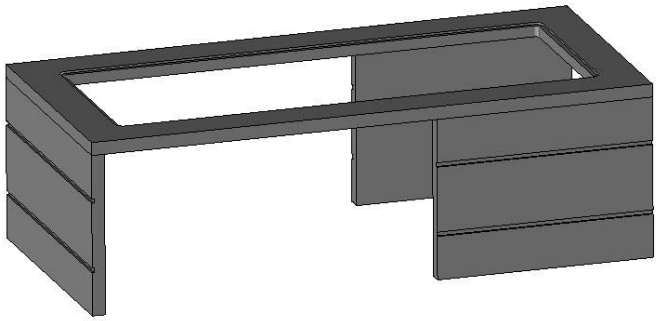
- Click **OK** to confirm the parameters.

- **Copy the groove operation**. 

- Select the outer face of the leg as the **reference face** again.
- In **Reference edge or curve for tool path**, select the top edge of the leg.
- Click **Stop** to validate.

- Continue copying the groove until you have made the six grooves on the three legs.
- Switch on **layer 2**.

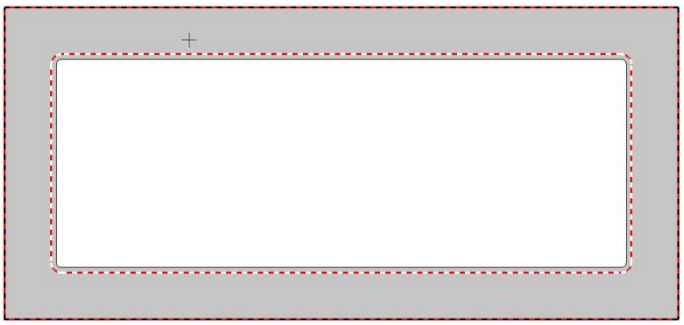


Finishing the table

Concepts addressed:

- Copying an edge contour
- Extrusion between two faces

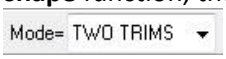
Making the table top



- Make **layer 3** the current layer.
- Select the **design color cyan (26)**.
- Start a **new sketch**.
- Use the **Edge** function. 
- Use **Mode = Contour** and select the top face of the table top.

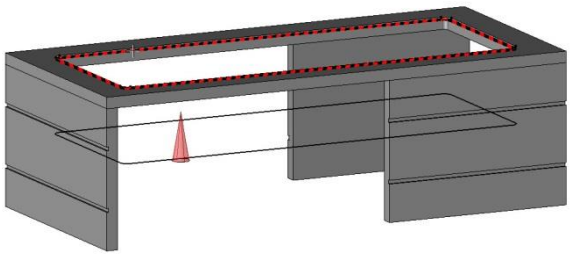
The edge of the rabbet is automatically copied.

Note: When selecting the edge to be copied, the reference face is framed in red and the edge to be copied is framed in red and white.

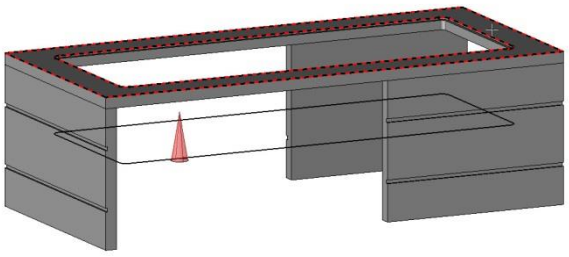
- **Finish the sketch.**
- Start the **Create extruded shape** function, then select the sketch.
- Select **Mode = Two trims**. 

Note: **Mode = Two trims** allows a part to be extruded between two faces or points, without entering the extrusion height.

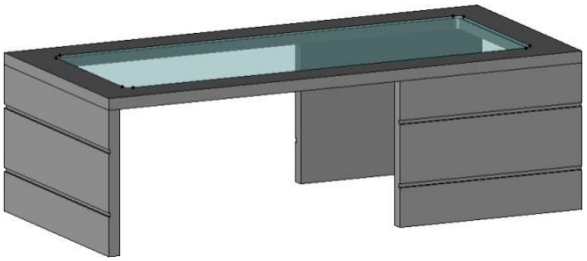
- Select the bottom of the rabbet in **First trimming face or point**.



- Select the top of the table top in **Second trimming face or point**.



- Apply a **design transparency** of **7** to the part using the **Attribute** > **Transparency** function.



Define the parts

- Use **Wood** > **Define** > **Define part** to define the parts.
- For the glass table top, enter:
  - **Designation**: *Glass table top*
  - **Reference**: *GL-TA*
  - **Material**: **glass** > **glass-02**
- For the table top, enter:
  - **Designation**: *Table top*
  - **Reference**: *TO-01*
  - **Material**: **wood** > **oak**
- For the three legs, enter:
  - **Designations**: *Base 1 / 2 / 3*
  - **Reference**: *BA-01 / 02 /03*
  - **Material**: **wood** > **oak**

Define the set

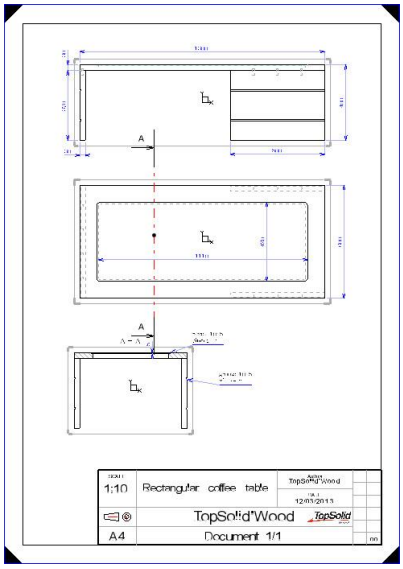
- Use **Define** > **Define set** > **Characteristics** to define the set.
- Enter:
  - **Designation**: *Rectangular coffee table.*
  - **Reference**: *RECTAB.*
  - **Main set**: **Sub-assembly**.
- **Save** the file.

Creating a draft

The goal of this exercise is to create a draft of the complete table.

Concepts addressed:

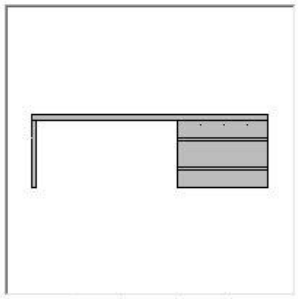
- Section view
- Dimensioning of wood operations: rabbets and groove



Create a new draft document

- Create a **new Draft document**  and select the standard template **Associative A4 Vertical**.
- Click on **OK** to confirm.

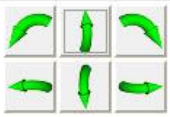
Position the main view



- Start the **View** context in the draft document. 
- Select **Main view** in the function bar. 
- Select **Assembly** to draft the entire document. 

- Select the file **Rectangular coffee table** in the dialog bar.

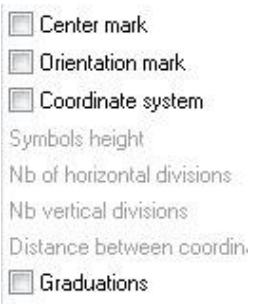
The **View creation** window opens.



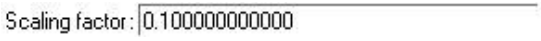
- Configure the main view by positioning the green arrows as shown opposite.

- Set the **smooth edges** to **Hidden**, and the **hidden lines** to **Stipple**.
- Click on **OK** to confirm.
- Position the view.

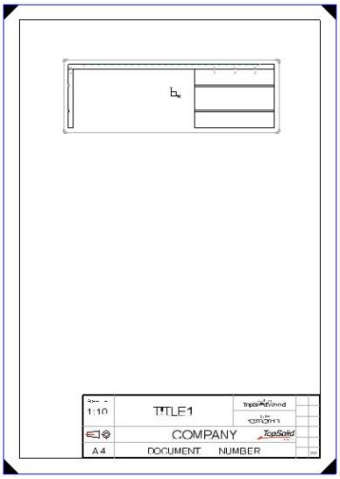
Edit the draft



- Modify the **draft frame** using **Modify element**. 
- Deactivate the **Center mark**, **Orientation mark**, **Coordinate system** and **Graduations** tick boxes.
- Set the **scaling factor** to 0.1.

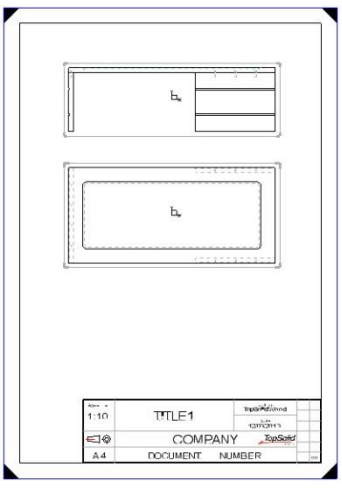


- Click on **OK** to confirm.



- **Regenerate** the invalid view to recompute it. 
- **Move parents** to place the view at the top of the page in the middle. 
- **Save** the draft using the disk button. 
Keep the default name (reference of the drafted assembly).

Position an auxiliary view



- Use the **Auxiliary view** function. 
- Validate the default parameters and position the auxiliary view using **Auxiliary view**. 

- Place the cursor beneath the main view, then click to position the view.

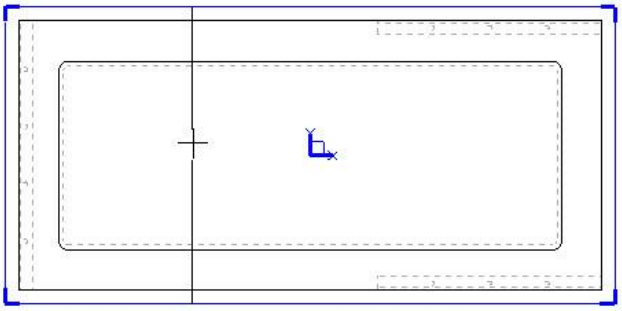
Create a section view

- Start the **Section** function in the **View** context. 
- Select the newly created auxiliary view as the **reference view**.
- Select **Horizontal or vertical cutting curve**.

HORIZONTAL OR VERTICAL CUTTING CURVE

- Place the cutting curve as shown opposite.
- If the line is horizontal, select **Change to vertical** in the dialog bar.

CHANGE TO VERTICAL

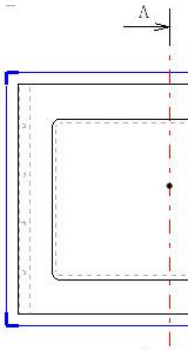


- The cutting direction must be to the right. If it is to the left, click **Invert** in the dialog bar.

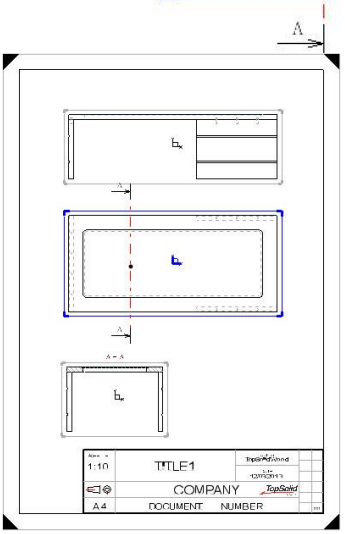
INVERT

- Confirm the cutting line by clicking **OK**.

OK



- Set the following in the dialog bar:
  - **Alignment = NO**
  - **Set the section view upright = YES**
  - **Hidden lines = Hidden**



Note: If the section view is not aligned, then it does not have to be aligned in the plane with the reference view. This means that the view can be placed wherever you like.
Set the section view upright is used to set the view upright in the draft.

- Click **OK** to confirm the section view parameters.
- Place the section view under the auxiliary view.

Dimension the views

- Start the **Dimension** context. 
- Use the **Fast dimension** function. 
- Place the various dimensions on the main view, the auxiliary view and the section view:
  - General dimensions
  - Dimensions of the bases and the top of the table.
  - Dimensions of the glass table top.

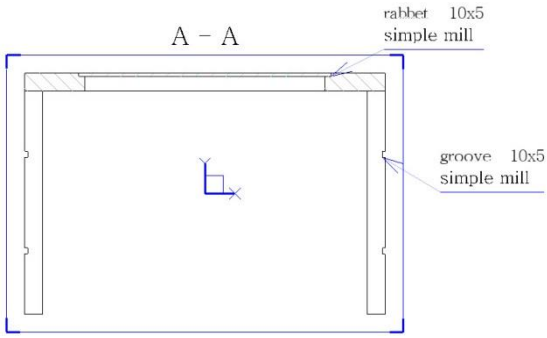
Dimension the wood operations

- Start the **Wood** context. 
- Start the **Dimension groove**  function, then select a groove of a base in the section view.

An information note about the groove is generated automatically.

- Place this note in the draft.
- Then use the **Dimension rabbet**  function and select the rabbet in the glass table top in the section view.

An information note about this rabbet is generated automatically.



- Place this note in the draft.

Note: The information that appears when dimensioning the wood operations is configured in **Document properties** > **TopSolid'Wood properties** > **Draft**. 

Fill in the title block

- Use **Modify element**  to modify the text in the title block:
  - **Title 1**: *Rectangular coffee table*.
  - **Company**: *Name of your company.*
  - **Document number**: *Document 1/1.*

- Use **Delete element**  to delete **Address 1** and **Address 2.**
- **Save** the document.

SCALE 1:10	Rectangular coffee table	Author TopSolid'Wood DATE 12/03/2013	
	TopSolid'Wood TopSolid Wood		
A4	Document 1/1		00

Supplement: Assembly and configuration

Concepts addressed:

- Assembly with pins

Assembly with pins

- Open the 3D design document.
- Make **layer 4** current. 
- In the **Wood** context, start the **Dowel assembly** function. 

Note: The **Dowel assembly** function is used to automatically place pins between two parts according to a given type of propagation.

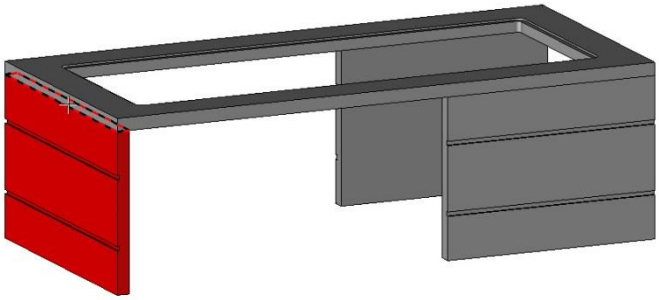
The **Standard positioning** window is used to select the pin.

- Select **Striated pin**, then select **code 35x10**.

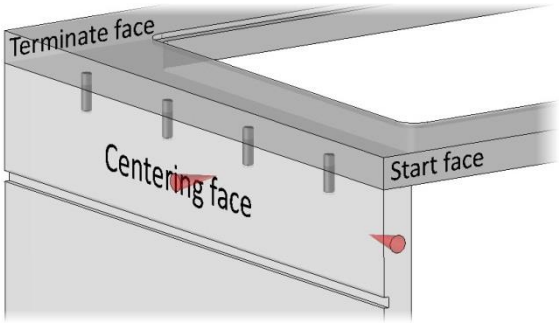
- Click **OK** to confirm the pin.
- In the dialog bar, switch to **Filter** mode and **Propagation = YES**.

Note: **Filter** mode is used to select only the contact faces between two parts.
Propagation = YES propagates the pins on the assembly face. In **Propagation = NO** mode, only one pin is placed in the center of the width and at a configured distance (in **Tools** > **Options)** on the length.

- In **Support face** select the contact face between a base and the top.

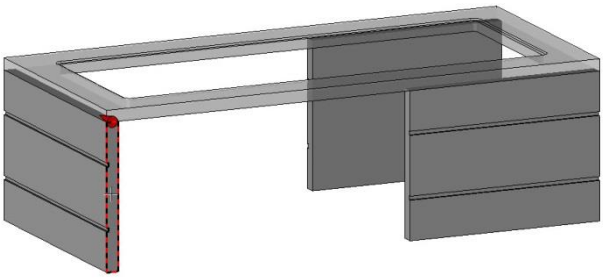


Note: The start and terminate faces correspond to the start and the end of the propagation of the pins (four pins in this case).

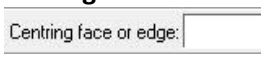


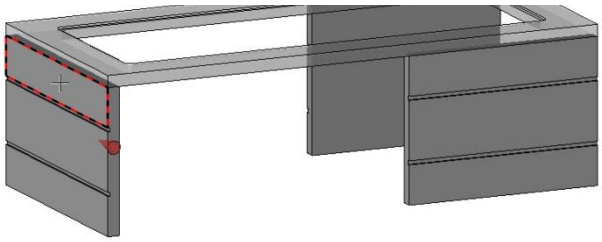
The centering face corresponds to the positioning of the propagation across the width of the assembly.
Once you have selected the centering face, you can center the pins across the width by selecting the opposite face or by entering a distance to offset them.

- Select the front face of the base in **Start face or edge**.

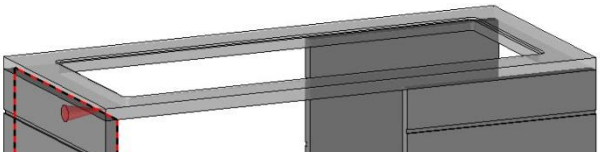


- Select the outer face of the base in **Centering face or edge**.

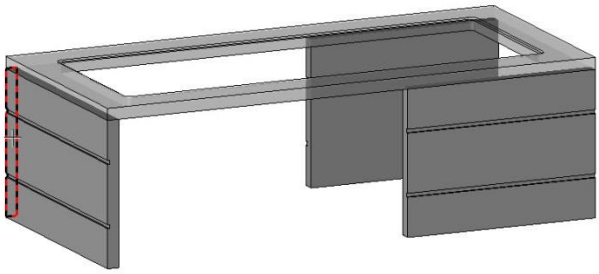




- To position the pins in the center of the width, select the inner face of the base in **Parallel face or edge**.



- Select the rear face of the base in **Terminate face or edge**.



The **distribution definition** window opens.

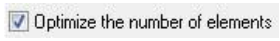
Note: In this window, you can use the propagations already defined in **Tools** > **Options** by selecting them in the **Propagation name** dropdown list.
Four types of distribution are then available: **Step distribution mode**, **Centered step**, **Distance** and **Advanced**. Each mode is used for a specific distribution that best meets the need.

- Select the **Centered step** distribution mode.

Note: In the **Centered step** mode, the distance between each pin can be configured, as well as the quantity of pins. The start and terminate distances of the propagation are equal.

- Set the **Step** to 128mm by selecting **128mm** in the dropdown list of **predefined values**.

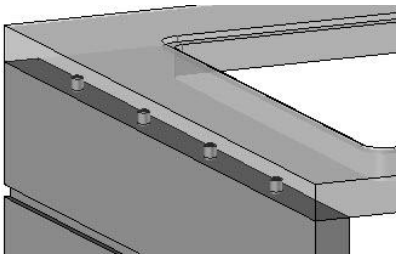
- Check **Optimize the number of elements**.



Note: The **Optimize the number of elements** option is used to automatically calculate the highest quantity of pins that can be placed with the selected step.
This quantity is recalculated if the assembly is subsequently changed.

- Click on **OK** to confirm.

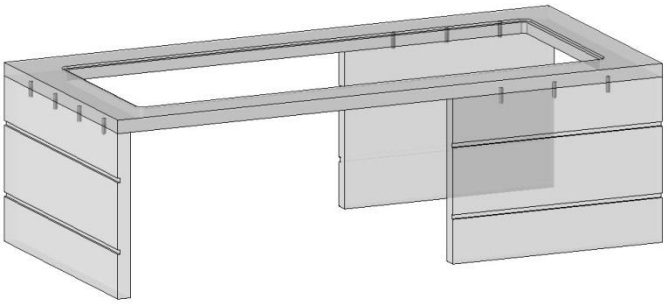
The pins are automatically placed and the parts are drilled.



- Use **Copy propagation** to copy the assembly propagation.

COPY PROPAGATION

- Repeat this operation for the other two bases.



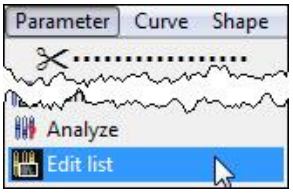
Configure the table

- Create three **length** parameters using **Parameter** > **Create** :
  - **Value = 1300mm**, **Name = L**, **Designation = Table length**
  - **Value = 600mm**, **Name = w**, **Designation = Table width**
  - **Value = 400mm**, **Name = h**, **Designation = Table height**

L	Table length	mm	1300mm
w	Table width	mm	600mm
h	Table height	mm	400mm

- Switch on **layer 0** to display the construction block. 
- Use **Modify element** to modify the construction volume. 
- Replace the lengths of the block on **X**, **Y** and **Z** with the parameters **L, w** and **h**.

- Press **Enter** to confirm.
- Switch off **layer 0**.
- Change the values of the parameters using **Parameter** > **Edit list**.



Note: **Edit list** is used to make several changes to several parameters at the same time. It can also be used to create new parameters.

- Select **Document** to modify all the parameters created in the document.

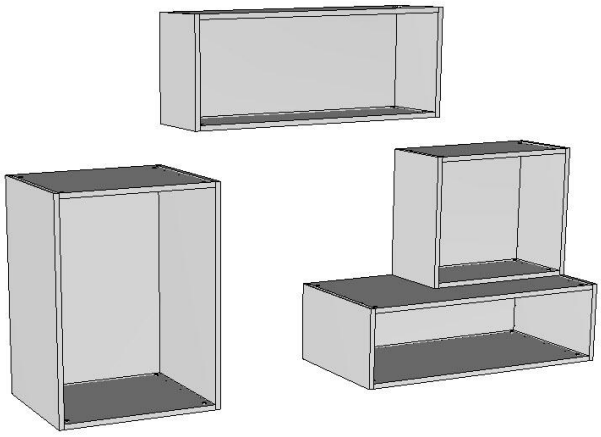
DOCUMENT

- Double-click on the **Value** field of a parameter to change the value. Change the value of the three parameters.
- Click **OK** to confirm the changes.

The table is automatically updated with the new dimensions.

Exercise 9: Making a storage cabinet

The goal of this exercise is to build a cabinet configured with the constrained block in order to reuse it with different dimensions in an assembly.



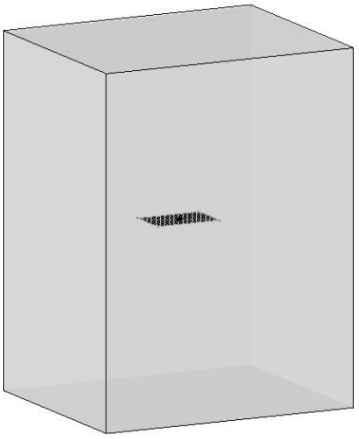
Design of the cabinet

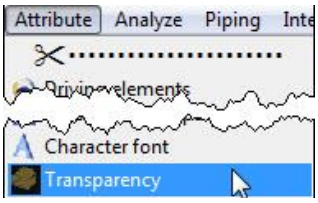
Create the construction volume

- In a **new Design document**, create four **length** parameters with the **Parameter** > **Create** function.

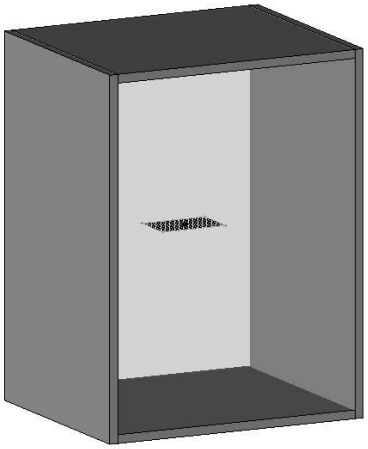
Name	Designation	Display unit	Value
w	Cabinet width	mm	600mm
d	Cabinet depth	mm	500mm
h	Cabinet height	mm	800mm
th	Thickness	mm	19mm

- Create a **block** with the dimensions **X = w, Y= d** and **Z= h** in **Alignment = Centered** mode.
- Select the origin of the absolute coordinate system in **alignment point** to place the block.
- Apply a transparency of **7** to the block using the **Attribute** > **Transparency** function.

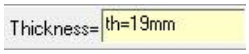




Create the parts



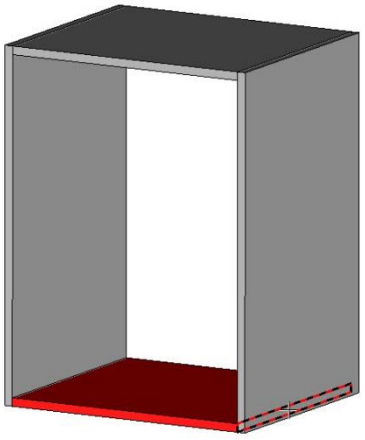
- Make **level 1** current.
- Start the **Wood**  context, then select **Constrained block**. 
- Enter the **thickness = th**.



- Create the two sides, the top and the base of the cabinet.

Note: The sides are free-running at the top and the base.

Pin assemble the parts

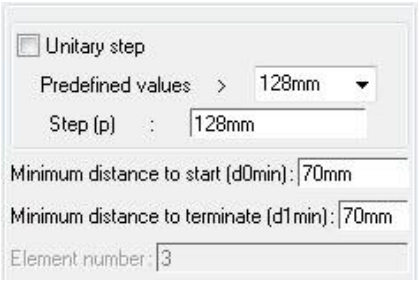


- Hide **layer 0**.
- Start the **Wood** > **Pin assembly** 
- Select the **smooth pin** with the **code 30x8**.
- Set **Propagation = YES**, then select the contact face between the base and the right-hand side as the **support face**.

Note: Rotary selection can be used to select the contact face between two parts more easily.

- Select **Automatic** to automatically detect the **start**, **terminate** and **centering** faces. 

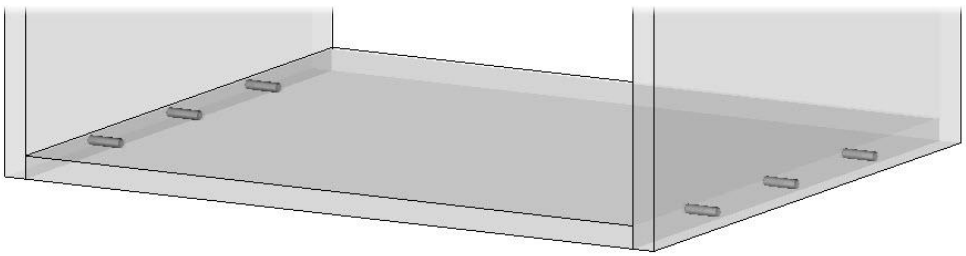
- In the **distribution definition** window, set:
  - **Distribution mode**: **Advanced**
  - **Step (p)**: 128mm
  - **Minimum distance to start**: 70mm
  - **Minimum distance to terminate**: 70mm



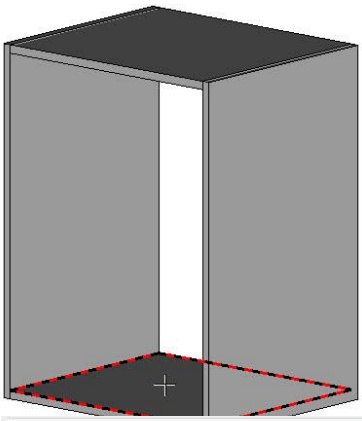
Note: The **Advanced** mode is used to automatically calculate the number of units to be placed, on the basis of a step between the units and minimum start and terminate distances.

- Confirm with **OK** to position the pins.

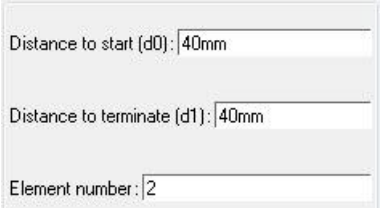
- Use **Copy propagation**, then **Automatic** to pin assemble the other parts. 



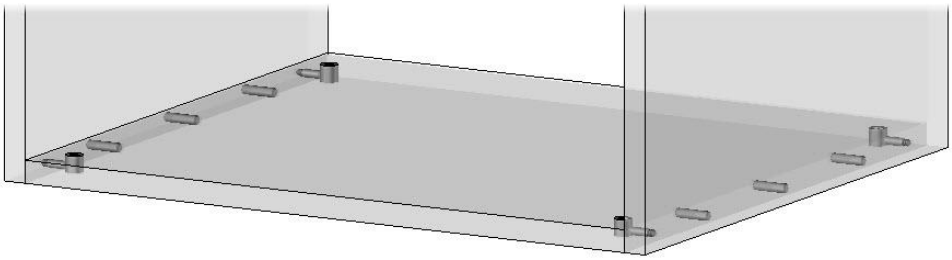
Place the cams



- Start the **Wood** > **Cams and dowels** function. 
- Select **Wood eccentric assembly hinge** with the **code: ep 19**.
- Set **Propagation = YES**, then select the contact face between the base and the right-hand side as the **support face**.
- Select the lower face of the base as the **face to drill for the case**.
- Select **Automatic**.
- In the **distribution definition** window, set:
  - **Distribution mode**: **Distance**
  - **Distance to start**: 40mm
  - **Distance to terminate**: 40mm
  - **Number of elements**: 2
- Confirm with **OK** to position the eccentrics.



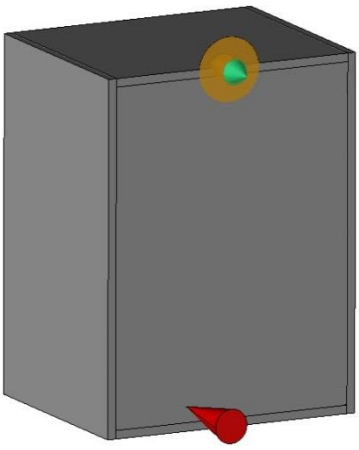
- Use **Copy propagation** to place the eccentrics on the other parts. COPY PROPAGATION



Create the back

- Make **level 2** current.
- Create the base of the cabinet as a **constrained block**.
  - Enter a **thickness** of 10mm.
  - Apply a **shift** of -8mm to the four planes.
  - Select the inner faces of the sides, the top and the base as **planes**.

Note: Entering a negative shift allows the constrained block to be included in the selected plane.



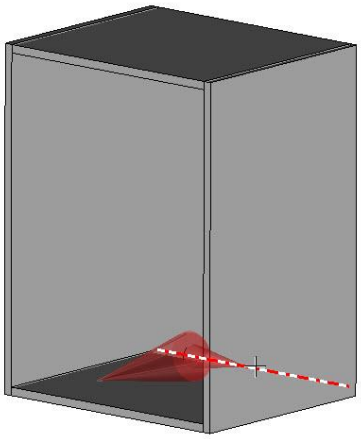
- Enter a **positioning shift** of 10mm, then select one of the rear edges of the cabinet as the **positioning plane**.
- Adjust the red arrow so that the positioning shift is towards the interior of the cabinet, then adjust the green positioning arrow so that there are 10mm between the rear of the base and the rear of the cabinet.

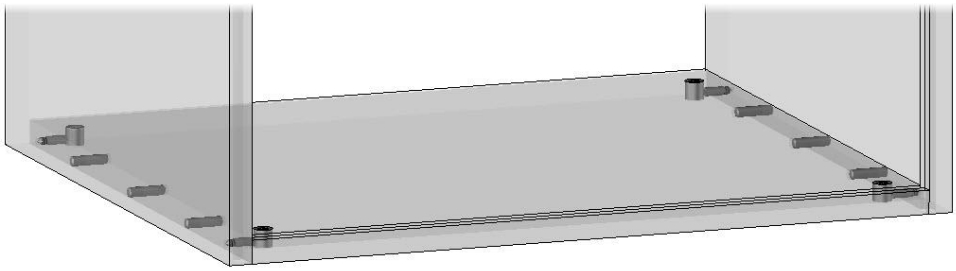
Machining the parts

Make the groove in the base

- Start the **Wood** > **Groove** function. 
- Set **Sweep = Planar face**, then select the interior face of the base as the reference face.

- Then select the rear edge of the selected face as the **reference edge or curve for tool path**.
- Then select **Stop**.
- In the groove parameters window, set:
  - **Tool type**: **Routers**
  - **Standard**: **Simple mill**
  - **Parameters**: **High arm**
  - **Gap distance**: 10mm
  - **Groove width**: 10mm
  - **Groove depth**: 9mm
- Click **OK** to validate and make the groove.
- Use **Copy operation** to make the same groove.



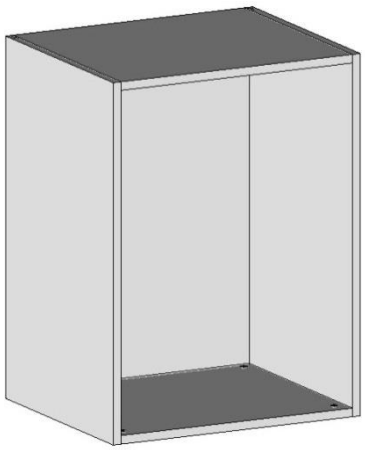


Definition of the parts and the assembly

- **Save** this file in a new folder called *Cabinet* and rename the file *Standard cabinet*.

Define the parts

- Use **Wood** > **Define part** to define the five parts of the cabinet:
  - **Designation**: *Top*, **Reference**: TO
  - *Bottom*/BO
  - *Right-hand side/RS*
  - *Left-hand side/LS*
  - *Back*/BA
- Select the **material**: **wood** > **fiberboard** and the **coating**: **paint** > **mat white**.



Define the set

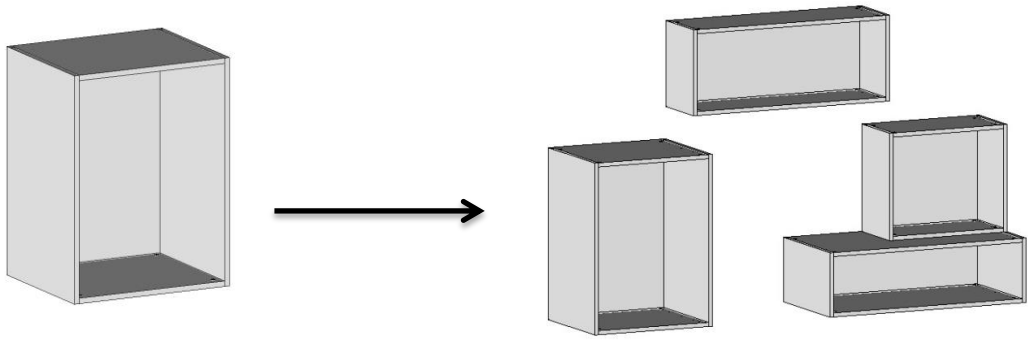
- Use **Wood** > **Define set** to define the assembly:
  - **Designation**: *Standard cabinet*
  - **Reference**: *CAB*
  - **Main set**: **Sub-assembly**

Creating an assembly

Define the drivers

Note

Defining a parameter as a driver allows its value to be changed in an assembly that contains the component.
In the case of this cabinet, defining the parameters as drivers allows several cabinets of different dimensions to be assembled using the same template.



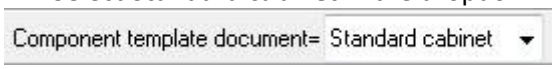
- Open the list of parameters using **Parameter** > **Edit list**.
- The last column **Driver** can be used to define a parameter as a driver. Double-click in this field to change the value of the four parameters to **Yes**.
- Click **OK** to confirm the list of parameters.

Name	Designation	Display unit	Expression	Value	Driver
w	Cabinet width	mm		600mm	Yes
d	Cabinet depth	mm		500mm	Yes
h	Cabinet height	mm		800mm	Yes
th	Thickness	mm		19mm	No
new parameter		length			

- Switch on **layer 0**, then save the file.

Assemble the cabinets

- Create a **new Design document**.
- **Save** this file in the *Cabinet* folder and rename it *Cabinet assembly.*
- Start the **Assembly context.** 
- Use the **Include assembly/Part**. 
- Select **Standard cabinet** in the dropdown list.



Note: Only the files that are open are included in the dropdown list. If the file *Standard cabinet.top* is closed, use **Explore** to select it in Windows Explorer.

Once the cabinet has been selected for inclusion, the system asks for the parameters defined as drivers.

- Enter the values for the four driver parameters.

The first cabinet will be positioned as **absolute coordinate system** on **absolute coordinate system**.

- Select **Other positioning**.

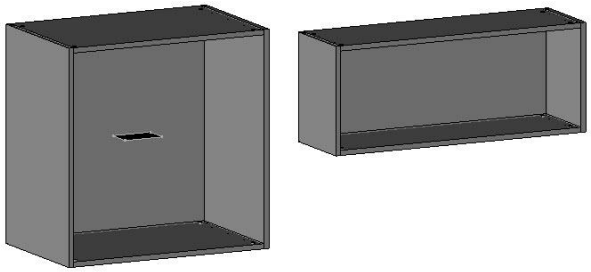
- The cabinet template document opens. Select the **absolute coordinates system**.
- Then select the **absolute coordinates system** of the assembly document as the **destination coordinates system**.

- Select **No propagation**, then **Stop** so that the cabinet is not propagated.

NO PROPAGATION > STOP

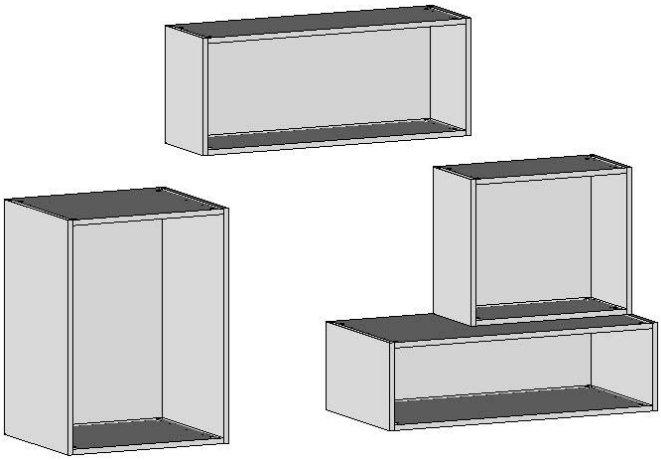
The component insertion function loops at the start of the cabinet inclusion in order to insert another one.

- Enter the new dimensions for the second cabinet.
- Once the second cabinet has been generated, click in the assembly document to release it.



- Select the lower face of the second cabinet as the **source geometry**.
- Select the top face of the first cabinet as the **destination geometry** in order to create a constraint.
- Confirm the distance for the constraint, then place the two other constraints on the second cabinet.

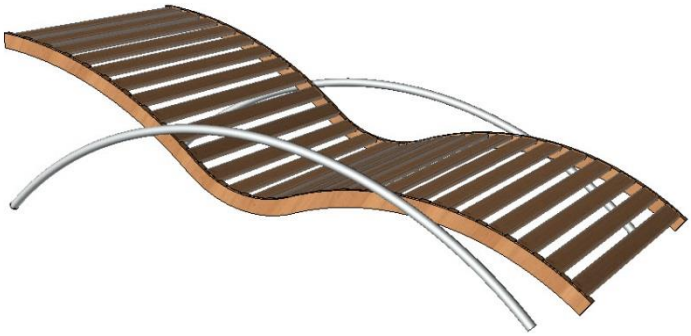
- Once all the constraints have been created, select **Stop** to proceed to the next step.
- Then select **No propagation**, then **Stop**.
- Continue by inserting several cabinets in order to create an assembly.



Exercise 10: Making a deck chair

The goal of this exercise is to build the deck chair using certain functions that have already been covered, plus some new functions:

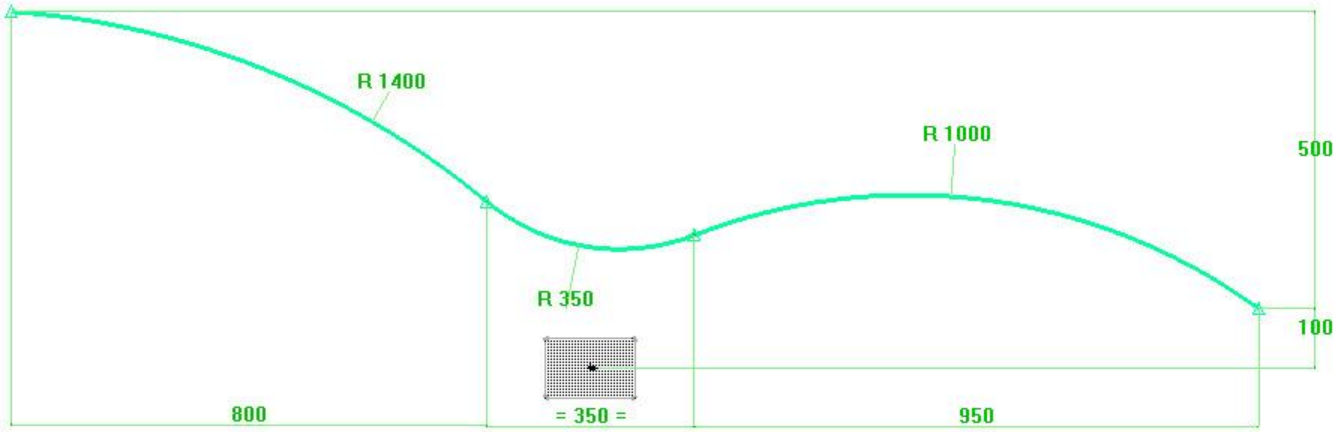
- Creating sketches
- Extrusions
- Repetition on curves
- Pipe shapes



Making the parts

Create the upright

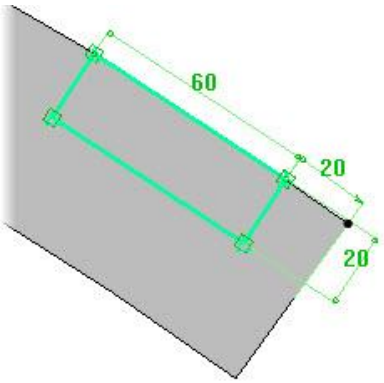
- Draw the sketch shown below. The three circle arcs are tangential.

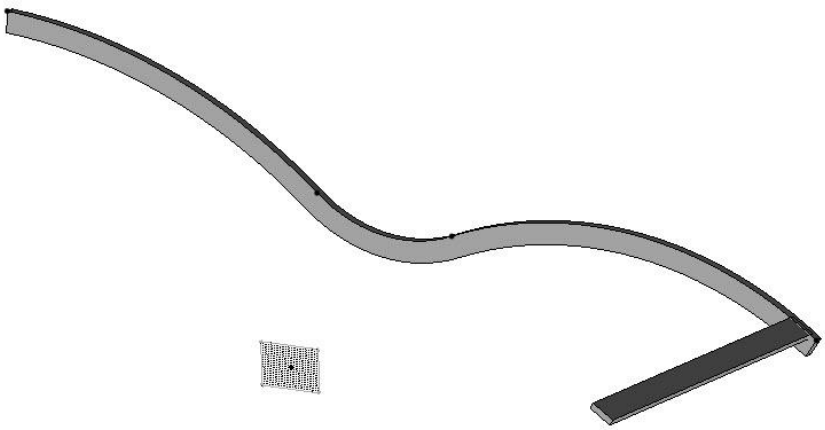


- Draw a parallel of this sketch 50mm lower down, then close the contour. The two lines that close the contour are perpendicular to the circle arc.
- Extrude this sketch in the **Z- direction**, with an **offset** of 400mm and a **height** of 20mm.

Create the blade

- Draw the sketch opposite on the right of the upright.
  - The four segments are perpendicular. Extrude this sketch in **Alignment = Centered** mode by a **height** of 820mm.





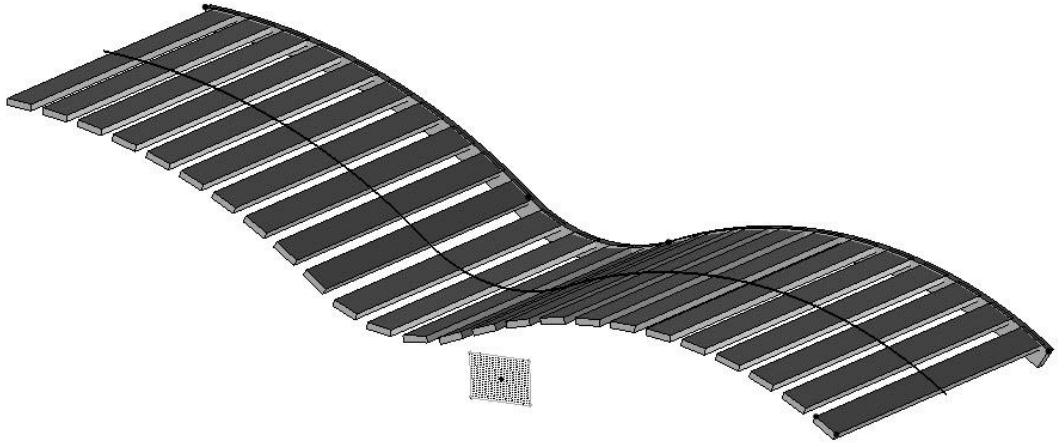
Repeat the blade

- Create a new sketch, copy the three upper edges of the upright and finish the sketch.
- Start the **Curve** > **Extend** function.
- Select the left-hand side of the sketch as the **curve to extend**, then enter a **length** of -40mm.

Note: This operation shortens the sketch by 40mm in order to repeat the blade according to this sketch.

- Start the **Repeat** function, then select the first blade as the **template elements to repeat**.
- Select **Propagation = On curve**.
- Select the right of the sketch as the **curve to propagate from the start**.
- Set:

- In **Number of points**, enter 25.
- Select the right-hand point of the top of the blade as the **first reference point**, and the left-hand point of the top of the blade as the **second reference point**.



Make the pockets

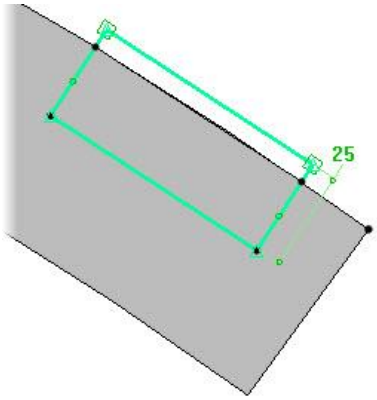
- Start the **Modify element** function. 
- Select one of the blades in the repetition, then select **Edit template**.

Note: A repetition is made up of three components:

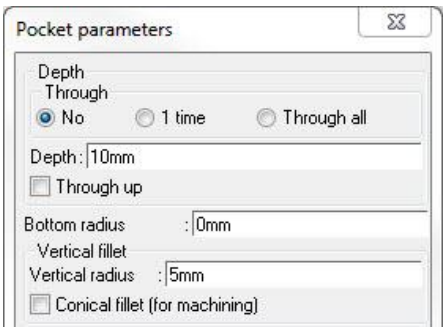
- the repetition template;
- the propagation;
- the copies created by the repetition (or instances).

Editing the model allows changes to be made to the repetition template, which can then be applied to all the instances.
The blades are then hidden and the repetition template is displayed.

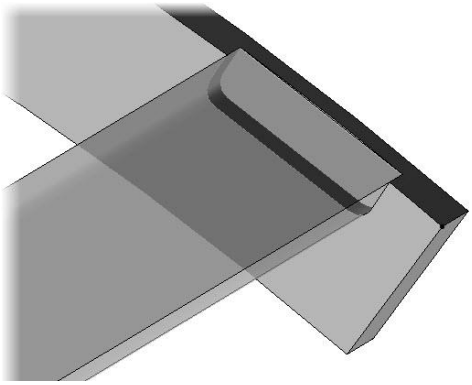
- Draw the sketch shown opposite by passing over the blade.



- Make a pocket on the upright using **Wood** > **Pocket**. 
- Select the top face of the upright as the **reference face**, and the preceding sketches as the **curve(s)**.
- Enter a **depth** = 10mm with a **vertical radius** = 5mm.

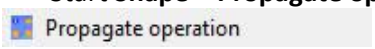


- Use **Shape** > **Fillet** to create a fillet with a **radius** of 5mm on the two lower edges of the blade.



- Use **Edit** > **Repeat** > **Show repetition**, then select the blade template to display the instances of the repetition again.

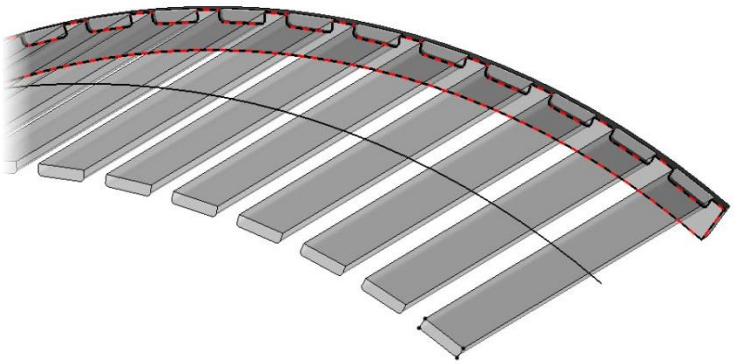
- Start **Shape** > **Propagate operation**.



- Select the pocket on the upright as the **operation to propagate**.

Note: Propagations can be performed as existing propagations to avoid having to enter new parameters. To do this, simply click on one of the instances of the existing repetition.

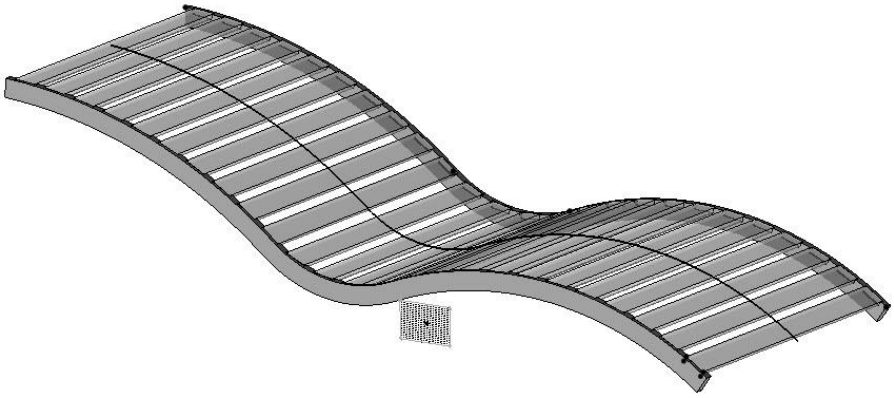
- Select one of the blades in the repetition as the **propagation**. The pocket is then propagated in the same manner.



Duplicate the upright

Note: In this case, the uprights are two different parts (left and right). Therefore, the part must be duplicated in order to have two different part definitions.

- Start the **Edit** > **Duplicate** function.
- Set **Simple mirror** with **XY** as the **plane of symmetry**.
- Then select the upright as the **element to duplicate**.



Making the supports

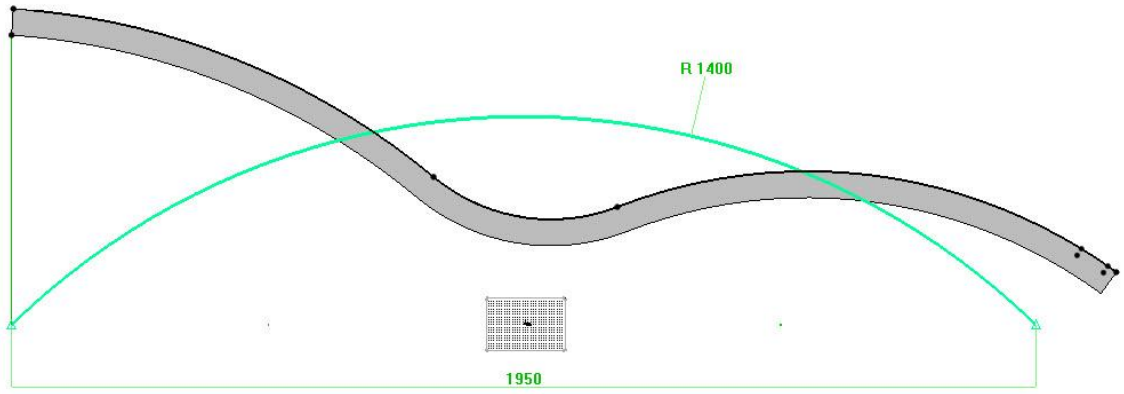
Draw the support

- Create a **duplicate coordinate system** using **Tools** > **Coordinate system** > **Duplicate coordinates system**. 

Note: **Duplicate coordinate systems** are used to create a coordinate system from an existing coordinate system by performing a transformation (translation, rotation, etc.).

- Select the document's absolute coordinate system as the **coordinate system**.
- Set a **transformation** consisting of a **translation** in the **Z-** direction with a **translation distance** of 435mm.
- Then **quit** and select **Set as current.**

- Draw the sketch shown below.
  - The two points are **aligned** with the **X axis**.
  - The left-hand point is **aligned** with the left-hand point of the upright **along Y**.

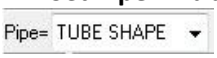


Curving the support

- Start the **Shape** > **Pipe** function. 

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

- Set **Pipe = Tube shape** to directly produce a tube following a given path.



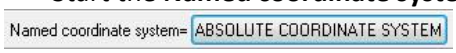
- Select the sketch as the **guide curve**.
- Enter an **external diameter** of 30mm.

- Enter a **thickness** of 4mm.

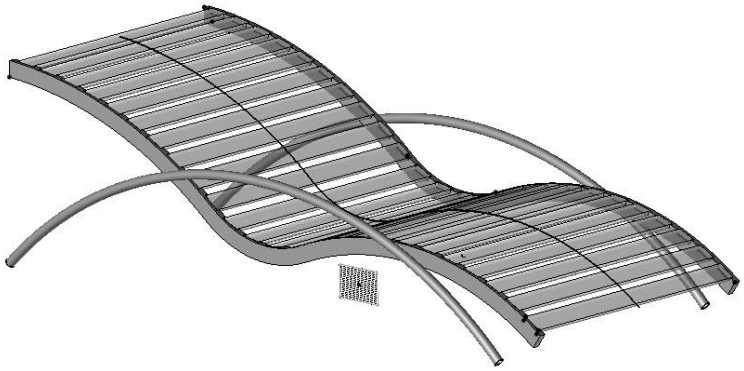
The support tube is now generated.

Repeat the support

- Start the **Named coordinate system**  function, then select **Absolute coordinate system**.

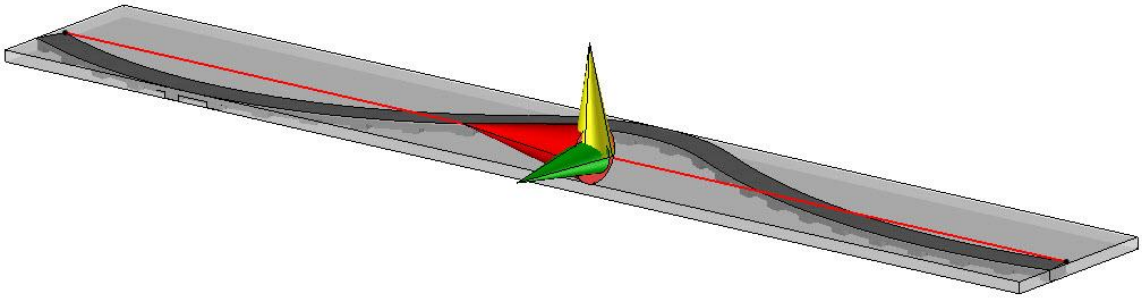


- Then repeat the support tube in **Simple mirror** mode with **XY** as the **plane of symmetry**.

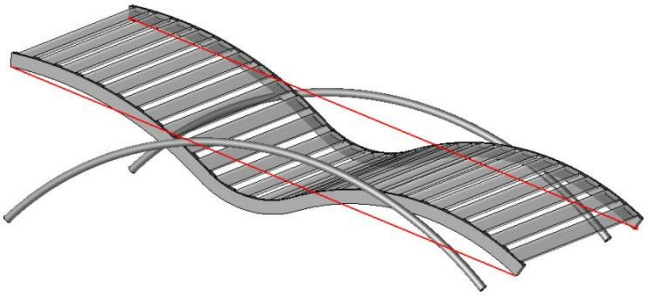


Definition of the parts and the assembly

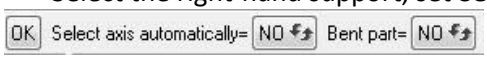
Note: Since the supports are curved parts, their sawing-off axes must be defined manually. To do this, first create a line representing the sawing-off length of the support.



- Use **Curve** > **Line** to draw the sawing-off lengths of the two supports.



- Start the **Wood** > **Define part** function.
- Select the right-hand support, set **Select axis automatically = No**, then click **OK** to confirm.



- Select the previously drawn line as the **lengthways axis** and **Y+** as the **width axis**.
- Then set:
  - **Designation**: *Left support/Right support*
  - **Reference**: *LS/RS*
  - **Material**: **wood** > **teak**
- Define the tubes:
  - **Lengthways axis**: **X+** and **Width axis**: **Y+**
  - Untick **Add to sawing-up**
  - **Material**: **metal** > **aluminum**
- Define the blades:
  - **Designation**: *Blade.*
  - **Reference**: *BL.*
  - **Material**: **wood** > **teak**

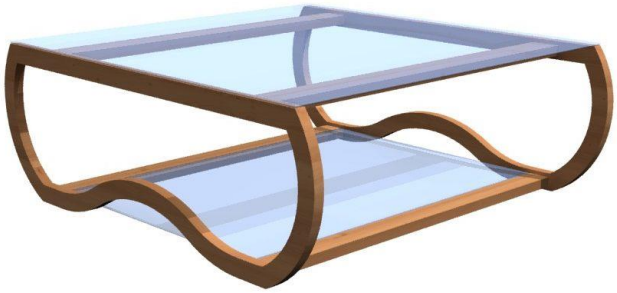
Note: Once the **Add to sawing-up** box has been unticked, remember to tick it again for parts that are sawn up.

- Define the assembly:
  - **Designation**: *Deck chair*
  - **Reference**: *DEC*
  - **Main set**: **Sub-assembly**

Exercise 11: Creating a coffee table

The goal of this exercise is to make a table using the notions covered during the training:

- Creating parameters
- Creating sketches
- Extrusions
- Wood operations
- Constrained blocks
- Drafts



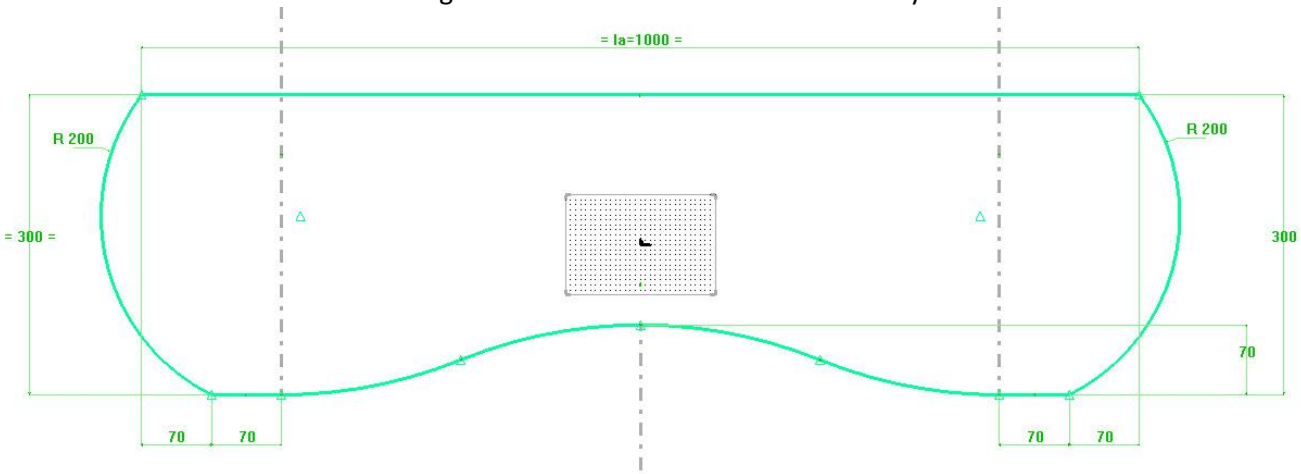
Creating the base

Create the parameters

- Create the length and width parameters of the table with the default values of 1000x1000.

Create the sketch

- Create the sketch below, including the width parameter of the table.
  - Use the **Doucine** function to make the circle arcs at the base.
  - The center of the doucines is aligned with the Y axis of the coordinate system.



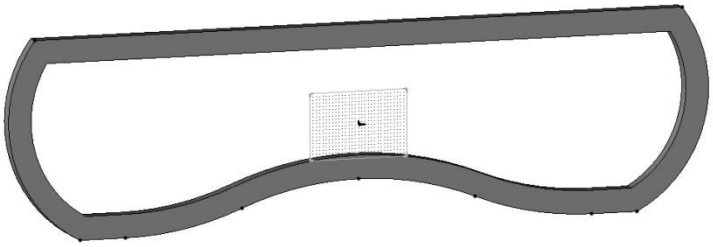
Extrude the base

- Extrude this sketch by a **height** of 20mm.

Saw

- Saw the base at a distance of 35mm.

Note: The tool used for sawing is a parallel sketch of a copy of the side edge.

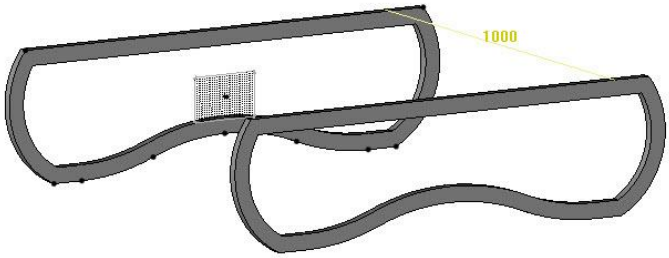


Define the leg

- **Length axes = X+** and **Width axis = Y+**.
- Enter a **designation**, a **reference** and a **material.**

Repeat the leg

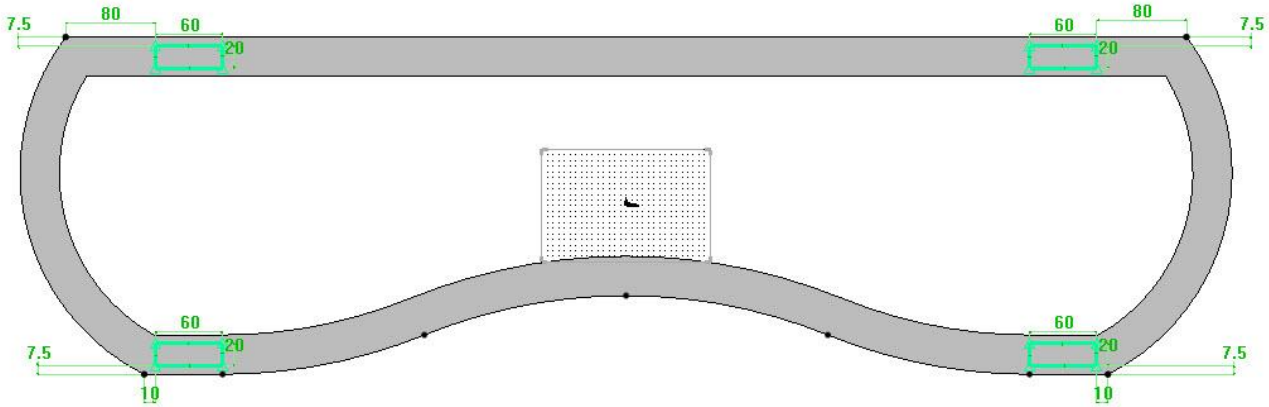
- Repeat the leg in a linear manner so that the total space between the two legs equals the length parameter.



Create the crosspieces

Draw the crosspieces

- Create the sketch of the four crosspieces as shown below.
  - The section of the crosspieces is 60x20mm.



Note: If the point of attachment of the dimension has not yet been created, a point can be created in the dimension function.

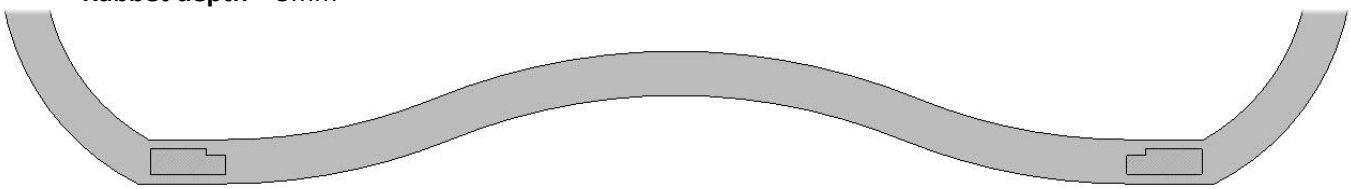
- When selecting the **first element to dimension**, start the **Point** function in the **system bar**. 
- In this case, create a **relative point**. 
- Select the start point of the dimension as the **position** for the point.

Extrude the crosspieces

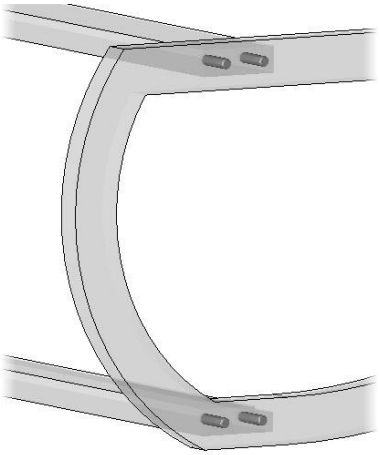
- Extrude the cross members in the mode **Result = One shape per curve** between the two repeated bases.

Make the rabbets on the crosspieces

- Make two rabbets on the top, inside the two lower crosspieces for the glass table top.
  - **Rabbet width =** 15mm
  - **Rabbet depth =** 5mm



Define the crosspieces



- Define the four cross members by entering a **designation,** a **reference** and a **material**.

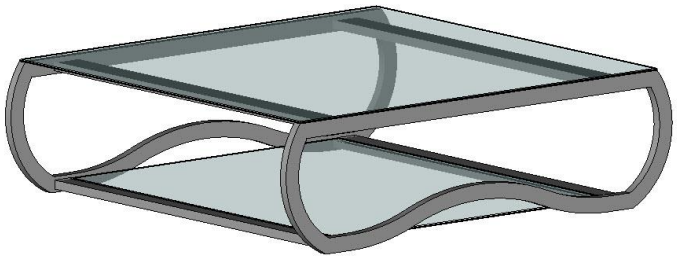
Assemble the crosspieces

- **Pin assemble** the four cross members with the two bases:
  - **Smooth pin 30x8**
  - **Step centered**
  - **Step** = 32mm
  - **Number of elements** = 2

Making the table tops

Create the table tops

- Create the two glass table tops on the top and the bottom as **constrained blocks** with a **thickness** = 5mm.
  - The top glass plate is attached to the upper cross members with a **shift =** -80mm.
  - The bottom plate is attached to the rabbets in the lower cross members.



Chamfer the top table top

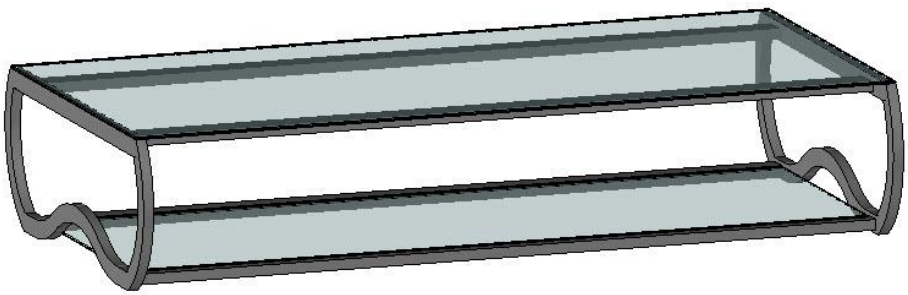
- Make two 2mm chamfers on the top and bottom faces of the table tops.

Define the parts and the set

- Define the two table tops.
- Define the assembly by entering a **designation** and a **reference**. This assembly is in **Sub-assembly** mode.

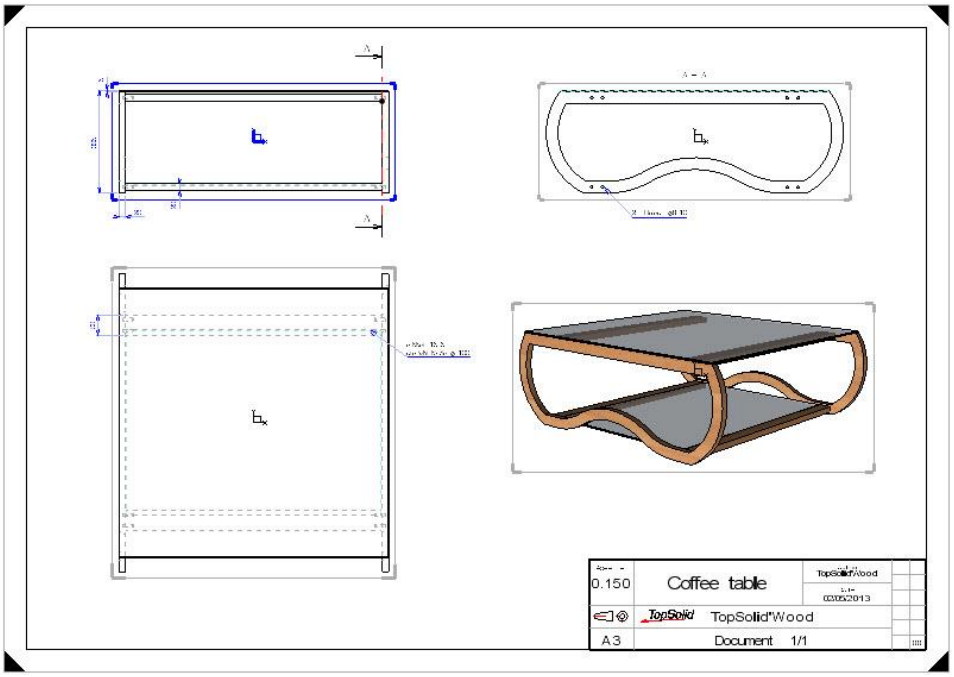
Use the parameters

- Use the **Parameter** > **Edit list** function to vary the dimensions of the table.



Creating a draft

- Create the draft of this table in a new horizontal A3 document.
  - Main view of the assembly
  - Auxiliary view
  - Section view
  - Dimensions
  - Dimensions of the wood operations



Notes

Individual course evaluation form

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

TopSolid'Wood - Basics

Name : .....

Company : .....

Date(s) from ..... to .....

By completing this individual evaluation form, you are helping to improve the quality and usefulness of the training provided in the future. Please complete it carefully.

Number of people during the course: Onsite at your company? YES ☐ NO ☐

GENERAL ASSESSMENT	Poor	Average	Good	Excellent
Overall, this course has been:	☐	☐	☐	☐

What grade would you assign?

0	1	2	3	4	5	6	7	8	9	10

LOGISTIC	Poor	Average	Good	Excellent
Orientation (quality, organization, user-friendliness, etc.)	☐	☐	☐	☐
Physical setup (room, materials, etc.)	☐	☐	☐	☐

TRAINING	Poor	Average	Good	Excellent
Instructor's teaching method	☐	☐	☐	☐
Group relationship (participation, sharing of experiences)	☐	☐	☐	☐
Quality and clarity of educational materials (documentation)	☐	☐	☐	☐
Balance between Theory and Practice	☐	☐	☐	☐
Consistent presentations with what has been announced	☐	☐	☐	☐
Training content	☐	☐	☐	☐

DURATION	No	Not really	Quite	Yes
Does the overall duration of the course seem appropriate?	☐	☐	☐	☐
If no, was it?	Too short ☐		Too long ☐	

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

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

Comments and suggestions: .....

.....

.....

.....

.....

.....