
# Getting started with TopSolid'Wood 2008

TopSolid'Wood 2008


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# Contents

# Presentation of TopSolid'Wood

## General issues

***TopSolid'Wood*** is a formidable CAD tool for the furniture's designers and makers. We want to thank you to have chosen it, and to get you familiar with it as soon as possible, we have elaborated some workshops that will allow you to better understand all the richness of the product:

- The power of the modeling functions provided by ***TopSolid.***
- The usefulness of the professional Wood functions (Profiling, Slotting, Edge banding, etc.)
- Productivity in the delivery of your projects (Bill of materials files, Multiple 2D drafting, Exploded views, etc.…).

The philosophy of the ***TopSolid*** environment, leaning towards the performance can be pleasantly surprising for the operators used to more conventional software programs.

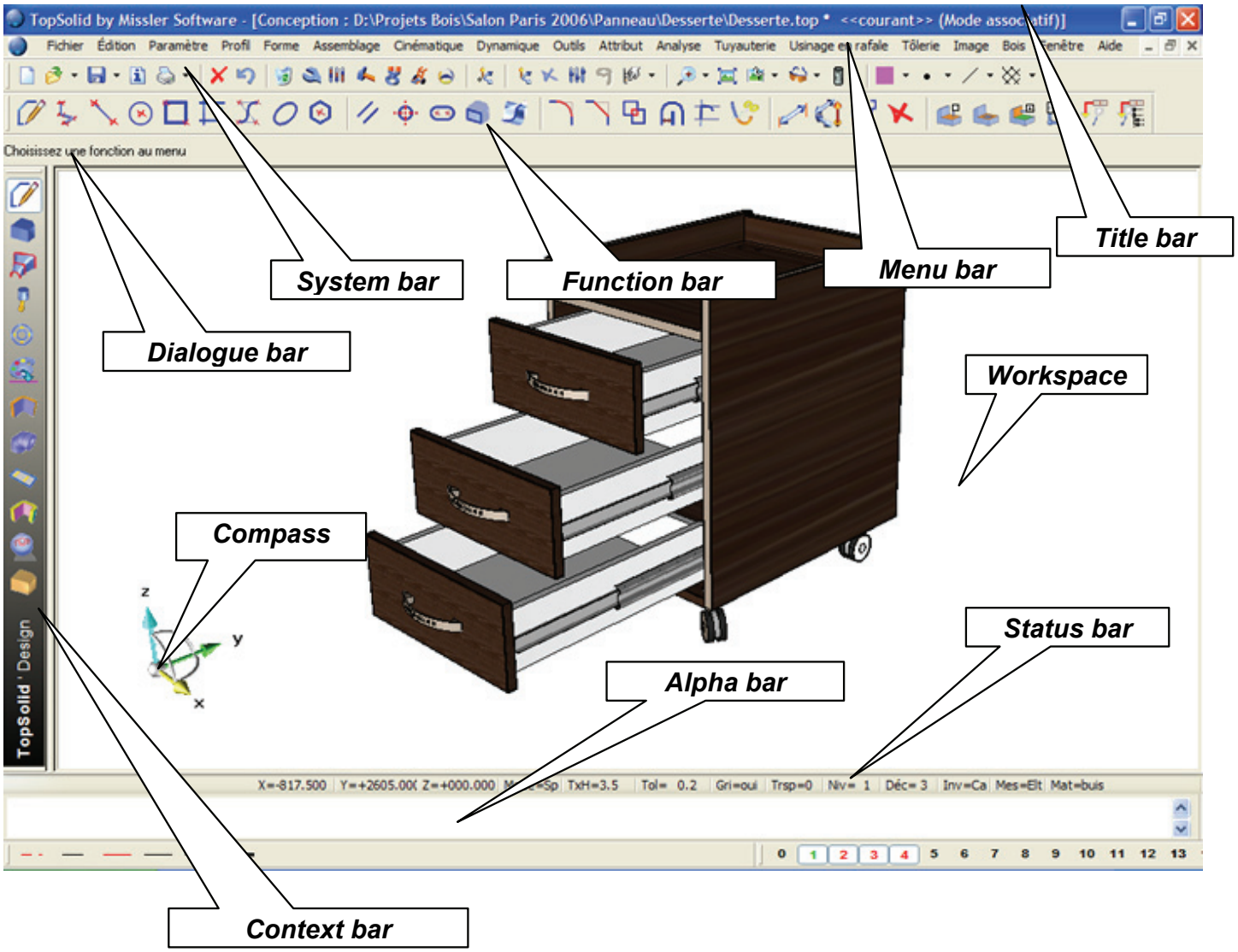
These tutorials have as target to get you familiar with this parametric and associative environment, and we are very sure that you will enjoy modeling your projects as much as we do providing the most innovative solutions adapted to your needs.

Very truly yours, the ***TopSolid'Wood*** team

# User Interface Overview

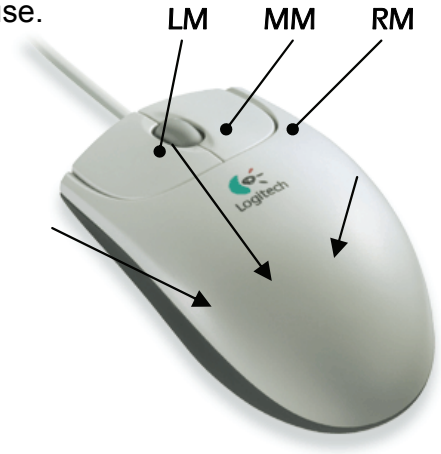
## General environnement

This is the main working interface of TopSolid. You will find the same for all modules of TopSolid (Wood, Mold, Progress, …)..



# Mouse functions

Different functions are associated with the three buttons of the mouse.




**Left Mouse Button (LM):**
Selection of any function from the menus
Selection of an element (dynamic selection) or creation of a point

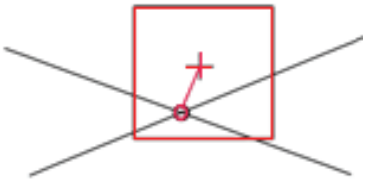
**Middle Mouse Button (MM):**
Creation of points on the current plane when clicked (advanced)
Dynamic Zoom using Scroll
Dynamic Pan when held down

**Right Mouse Button (RM):**
The first option of the current command is accepted when the right mouse button is clicked or the context menu of the current command is displayed when held down.
Display a context menu when used on menu bar, alpha bar.

## Further important uses.

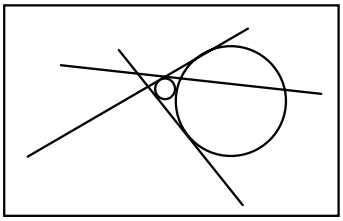
**Intersection of 2 Items:**

To obtain the intersection of two items left click and hold **LM** in the graphics area away from the intersection then move the mouse over the intersection then release the mouse key.
The size of the square can be changed using the + and – buttons on the keyboard.



**Rotative picking of items**:

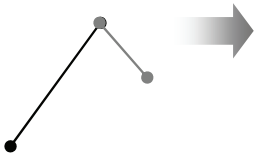
When the mouse is moved over an item, the nearest item is automatically highlighted. If this is not the required item, to select it (without zooming) press and hold down the left mouse button near the item you want to select (the small circle for example) and use the right mouse **RM** click (or middle mouse **MM**) to allow "Rotative picking" through the items at the current position. When the correct item is highlighted release the left mouse **LM**.



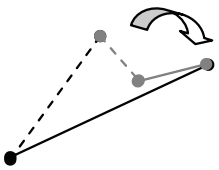
**Middle button property**:

The middle button has one more distinct property in that when drawing lines for instance it will always create a NEW point even if you click onto an existing one.

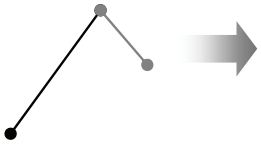
1 – Here we draw 2 separate lines that join at a point. All done with the left button (**LM**).



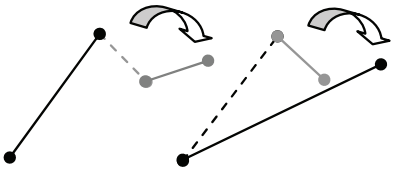
If we move the common point, then we see that both lines alter to remain joined.



2 - Here we draw 2 separate lines that join at a point however the second line was drawn with the middle button (**MM**).



If we move the common point we see that the two lines are in fact separate and can move independently.



Functionality

## The icons

There are two main types of icons in TopSolid: simple icons and icons with options.

The simple icons execute the function with a single left mouse click **LM**.

The icons with options using the left mouse **LM** on the Icon select the command as above.

In the icon menus :

- If you use a left click **LM** the option selected becomes the default option for the next time you use this function
- If you use a right click **RM** the default option does not change

The buttons

### *Buttons without input* :

The first type of button allows the user to change between several options by simply clicking on the button.

***For example, to draw a circle, by default the option Radius is selected. A click on the button switches the command to Diameter.***


The second type of button allows the user to confirm an option.
For example when creating a draft view, the user can use the option **EXPLORE** to select a file using the windows browser.

***A button with lower-case text corresponds to a button with more than one option while a button with all capital text corresponds to a button that will open a further option menu.***

## *Buttons with input :*

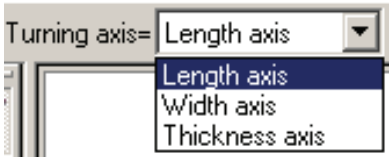
For certain options, TopSolid needs an input from the user, for example :

A numerical input value
***Ex : Diameter value of a circle***


An action to perform
***Ex : Select a piece to slot***


If the choice is restricted, TopSolid proposes a list :

***Ex : This type of list is available during the use of the turning function,***




***Or during the modification of the type of part…..***


## Keyboard actions

- The ***up*** and ***down*** arrows allow the user to cycle through any previously used values.
- "***Control***" enters dynamic rotation,
- "***Shift***" enters dynamic pan,
- and the ***combination of both keys together*** enters dynamic zoom.

The function keys in TopSolid have the following uses, as well as the normal windows functions :

| Key | Function |
|---|---|
| **F1** | Online Help |
| **F2** | Information on points and elements |
| **F3** | Dynamic Zoom |
| **F4** | Dynamic Pan |
| **F5** | Dynamic rotation around X |
| **F6** | Dynamic rotation around Y |
| **F7** | Dynamic rotation around Z |
| **F8** | Cancel Dynamic rotation |
| **F9** | Dynamic Rotation |
| **F10** | |
| **F11** | Reorganization of the toolbars |
| **F12** | Floating windows On/Off |

User defined shortcuts can be created using the **Tools, Options** menu and then **Shortcut key** context

## Entering in coordinates

**Cartesian coordinates** : defines coordinates whose values are absolute from the current coordinate system origin (X, Y, Z). Commas separate the values, the Z value is optional.
***Ex: 12,45,21***

**Polar coordinates** : defines polar coordinates length in XY plan, angle and a height in Z (length; angle, z). The Z height is optional.
***Ex: 20;45,5***

**Spherical coordinates** : defines spherical coordinates length in XY plan, angle in XY then angle in YZ view of the current direction (Length;angle1;angle2).
***Ex: 5;45;30***

**Relative coordinates**: defines coordinates relative to the previous point specified the coordinates are preceded by the symbol &.
***Ex: &10,10,10***

***Tips: to create a point on 0,0,0 you can directly use "Enter" key instead of indicate the coordinates.***

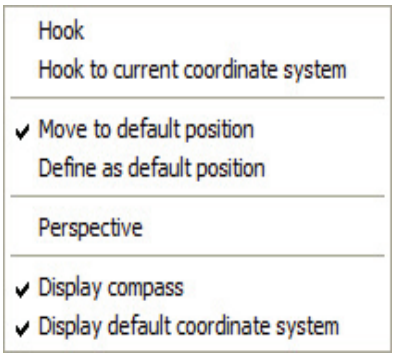
## The compass

The compass may be positioned anywhere in the view or hooked to an element of the document by sliding-moving its centre point.


Once hooked to an element, the compass spherical centre changes to a cube, and this new centre corresponds to the centre of dynamic rotation of the document.

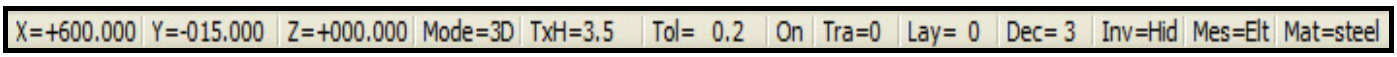
Hooking the compass to an element allows the user:

- to manipulate the view according to the new orientation of the compass: (Rotations along the hook axes…)
- to create a coordinate system on the hook (accessed via the context-sensitive menu, right button)



## Status bar

Provides feedback and allows the user to quickly set layers, colors etc. and set display tolerances and invisible parts.
Information shown can change depending of the modules (TopSolid'Design, TopSolid'Draft …).




Click directly onto the value to change/manage it.

Here is a description of some modes in the bar. (See on line help for more explanation).

**Mode :** Change picking mode : 3D = Spatial, Pr = Projected, Pl = Planar
Allows you to change the selection method in the document.

**Tolerance :** Change tolerance
Determine the precision of the model
(Related to both performance and file size)

**Magnetic snap :** Magnetic snap activation
Allows the use of the grid points of the current coordinate system.

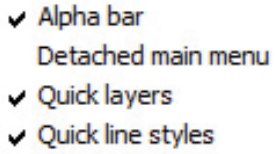
**Visibility :** Show invisible elements : Hid = hidden, Sho = Shown
Controls the visualization of the parts.

**Material :** Change material and coating
Allows you to change the default material of the document.

## Added bars

In order to reveal options for Quick layers and line styles, right-click in the menu bar.

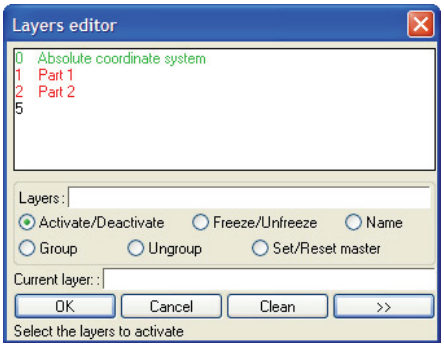
In the menu that appears select **Quick line styles** and **Quick layers**.



## Layers management

The dialog box appears when you click on "Layers" in the status bar.
You have 3 different levels of layers :

- The current layer is in **green**.
- Active layers are in **red** (visible).
- Inactive layers are in **black** (invisible).




The lower part of the dialogue box gives access to several options.
It is possible for example to name a layer, to freeze it (i.e. make it inaccessible), or to group or explode several layers.
The groups only appear in the list and must be given a name.

## Main functions


**New document** : 

There are two main types of documents :

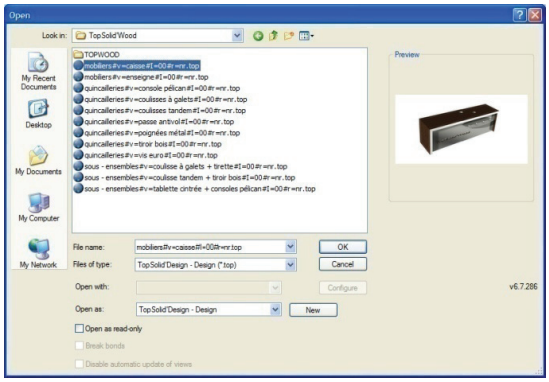
Design documents, **.TOP**
Draft documents, **.DFT**

For each type you can find standard templates (1 coordinate system, 3 coordinate systems, A4, A3…).

**You can create your own templates and save them in your "Config | Template" folder.**

**Open an existing document** : 

TopSolid shows a list of files in the current folder with TopSolid extensions and also files supported by direct interfaces like IGES, STP, DXF, DWG, Parasolid, ACIS …
Some direct interfaces are separately purchasable.



**New** creates a new document.

The **Configure** button is active depending of the type of direct interface file used.

**Save**  or **Save as** 

3D design files are saved with the extension .top and 2D files are saved with the extension .DFT.
In the title bar, if the name of the file is followed by a *, this means that there are changes to the file that have not been saved. If there is an exclamation mark it means there are some invalid elements.

**Types of files** allows the use to save in other formats such as STEP, IGES, DWG, DXF, etc.

**Print**. 

This function allows you to print what is currently on the screen.
Depending of the application used you have will have different printing options.

**Global Undo** : 

Cancel all the actions carried out within the current function but do not exit it, to quit the function press the **Escape** key.

**Undo** : 

Undo the previous action within the current command.

**Delete element** : 

Delete the selected elements. The option **ALL THE ELEMENTS** allows, after confirmation, to clear the current document.

**Modify element** : 

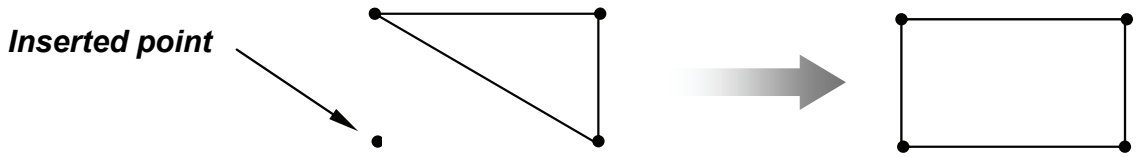
Modify an element or operation e.g. contour, radius boss, transformation…

**Modify parameter** : 

Modify the value of a parameter of an element or operation, e.g. diameter, length, extrusion height …

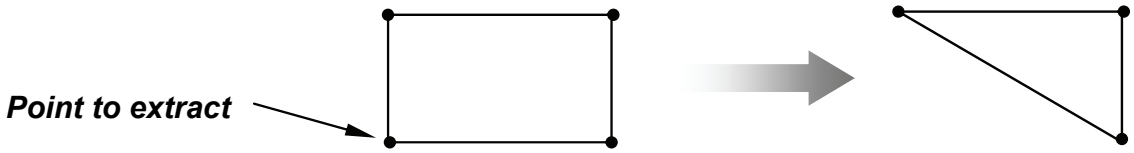
**Insert element** :

Insert an element e.g. point, line, circle ...




**Extract element** :

Extract a portion or feature of an element.(e.g.: point of a contour, drill or fillet on a shape, union, boss, title block element,...).




If there is an ambiguity, TopSolid will ask you to choose between them. The element or the operation is destroyed but the elements that were used to create it are preserved.

Example: the extraction of a boss eliminates the boss but not the profile from which it was generated (the profile remains invisible).

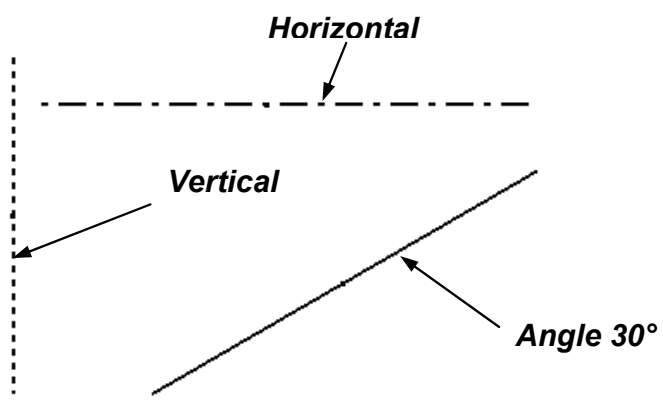
**Move parents** :

Move an element and its construction elements if the element is not fully constrained. TopSolid will show dynamically the possible positions.

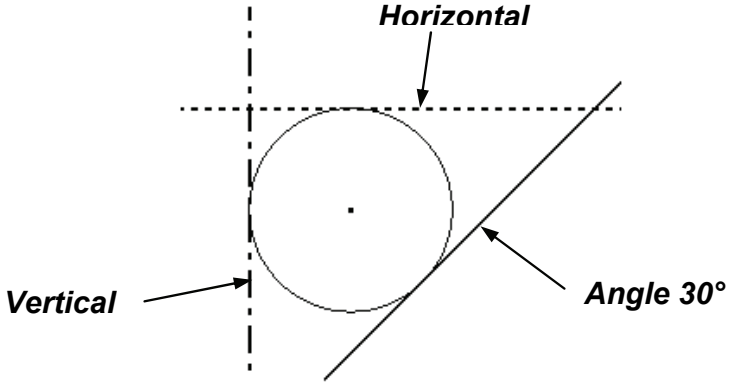
# 2D Functions

## Sketch line

Icon : 
Menu : *Curve* | *Sketch line*

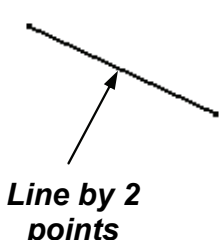

***On point***



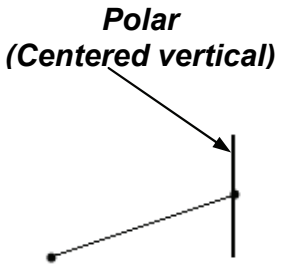

***On tangent***

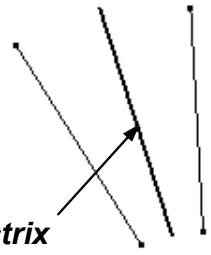
## Line

Icon : 
Menu : *Curve* | *Line*

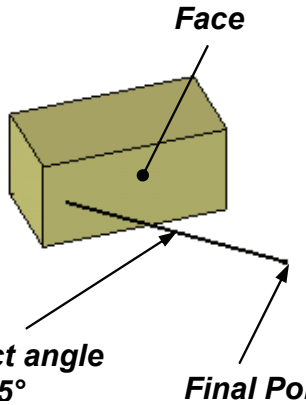



***Line by 2 points***



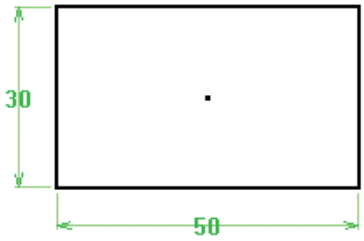



***Bisectrix***

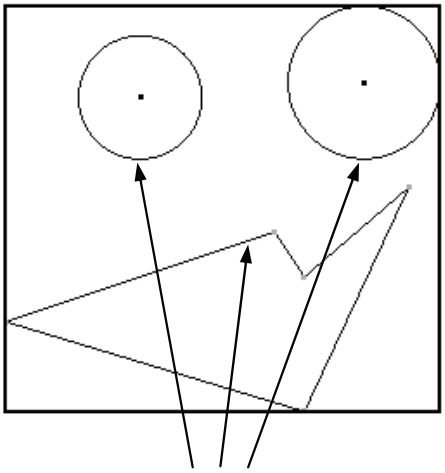


## Rectangle

Icon : 
Menu : *Curve* | *Rectangle*


***Rectangle created with X and Y dimensions***



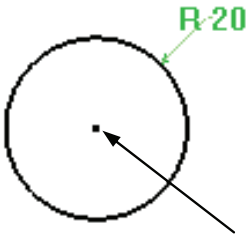
***Elements to enclose***

***Rectangle created with option "DIAGONAL"***

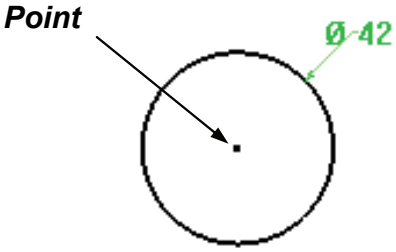
***Rectangle created with option "ENCLOSING" and margin at 0***

## Circle

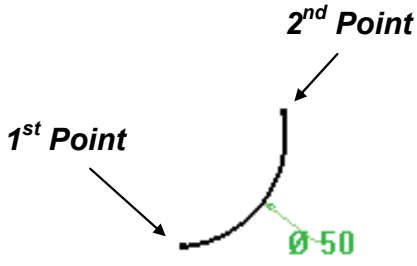
Icon : 
Menu : *Curve* | *Circle*

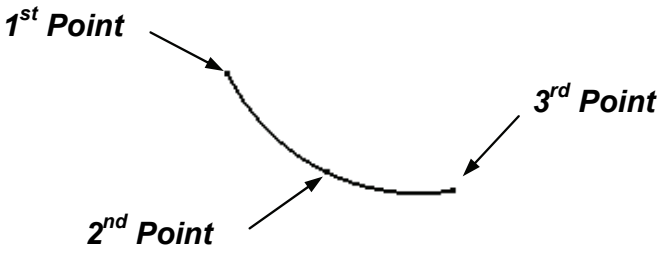

***Radius and Center***



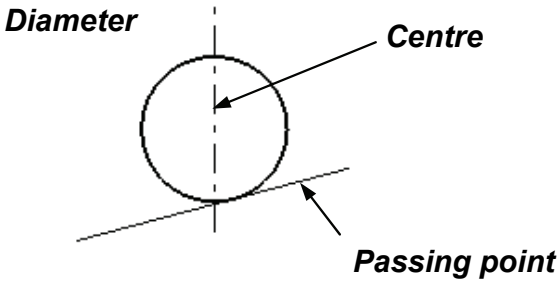

***Diameter and Center***




***Diameter and Passing points***

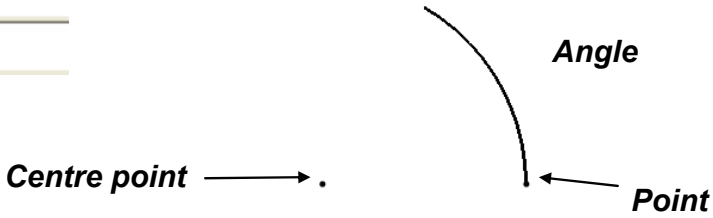



***Passing point only***




***Center, Passing point and Diameter***


***Use the option POINT***

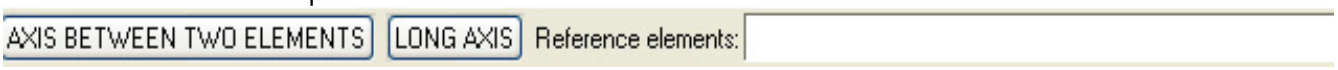



***Center, Point and Angle***

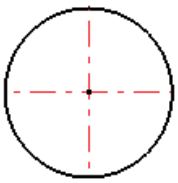
## Axes

Icon : 

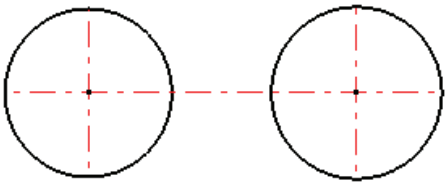
Menu : *Curve* | *Axes*




***Axis between two***



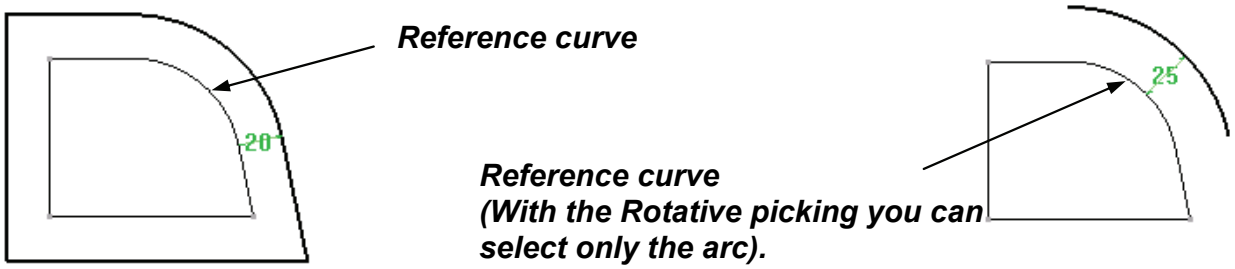
***Axes on elements***



***Long axis ("LONG AXIS" allows you to replace many axes in one).***

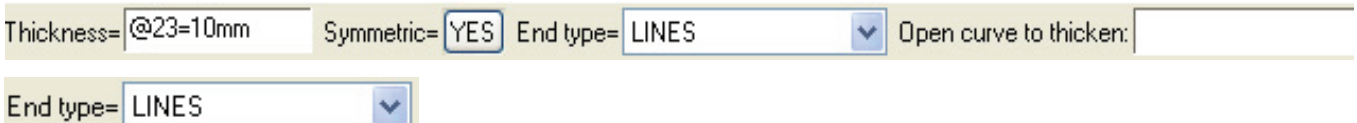
## Offset curve

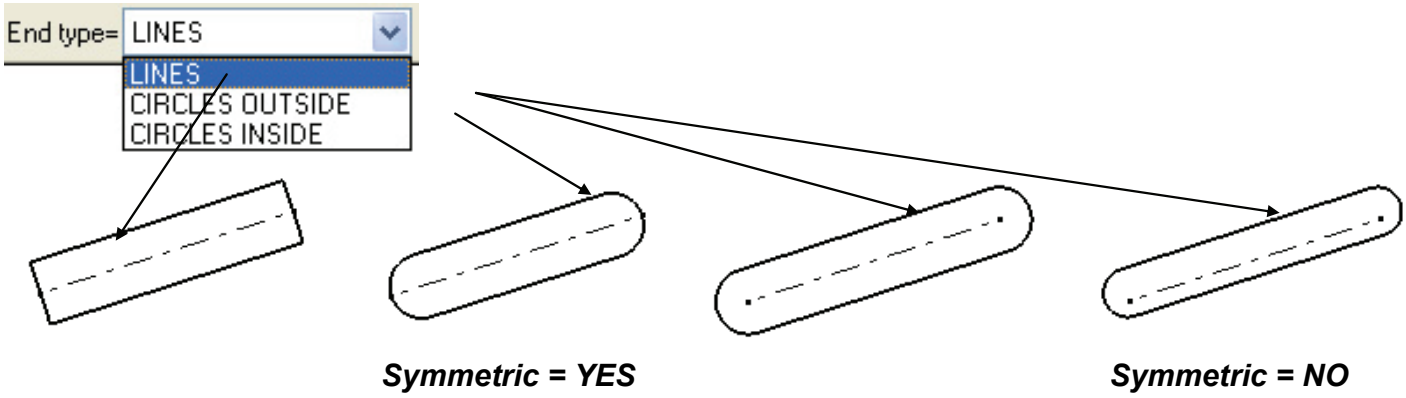
Icon :
Menu : *Curve* | *Offset curve*


## Thickened curve

Icon :
Menu : *Curve* | *Thicken*

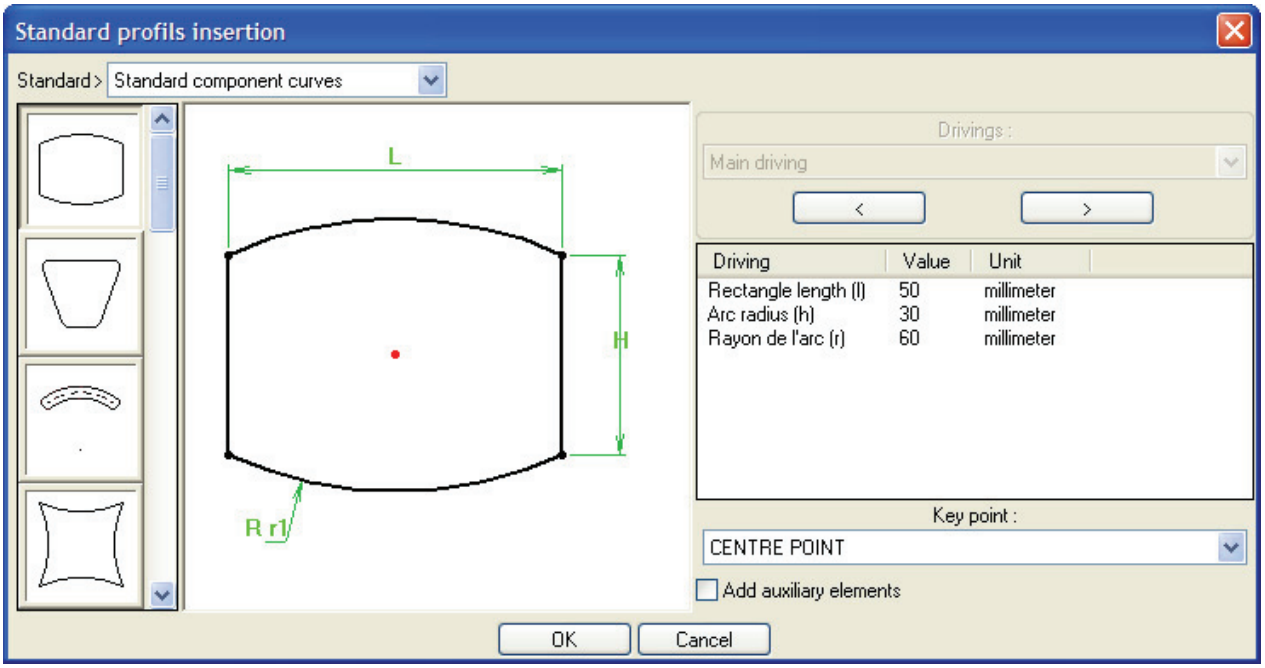


## Standard curve

Icon : 
Menu : *Curve* | *Other curves*|*Standard curve*

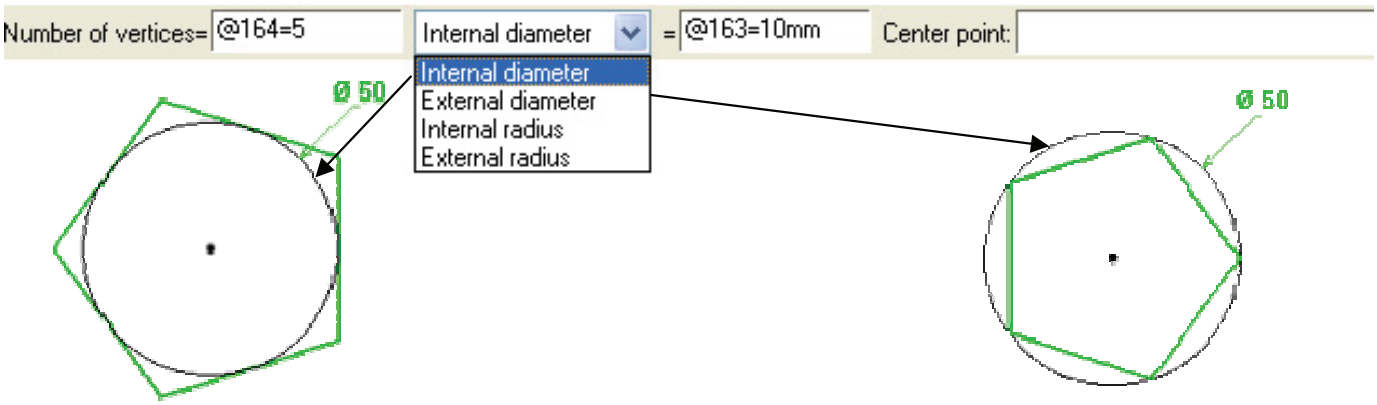
Create profiles from standard curves.




***Note: you can create your own standard curves.***

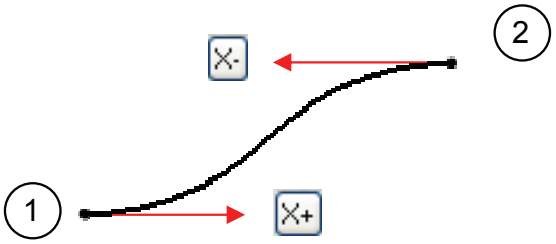
## Regular polygon

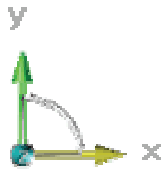
Icon : 
Menu : *Curve* | *Other curves* | *Regular polygon*

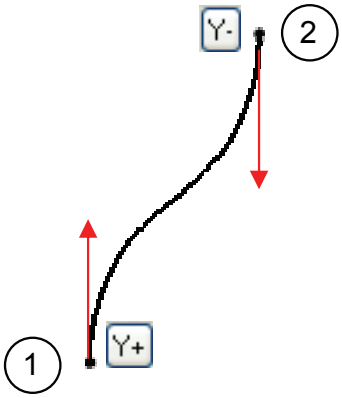


## Arc bend

Icon :
*Curve* | *Other curves* | *Arc bend*

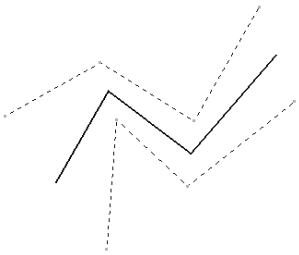




## Middle curve

Icon :
Menu : *Curve* | *Other curves* | *Middle curve*

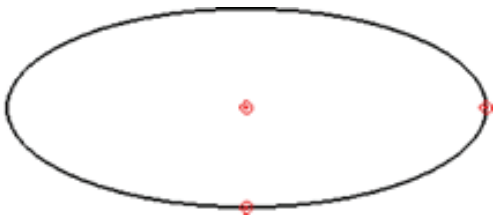
Create a middle curve between two curves (made by lines and arcs).



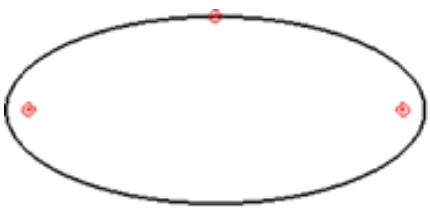
## Ellipse

Icon :
Menu : *Curve* | *Other curves* | *Ellipse*

Allows the user to create an ellipse using a centre point and two passing points, or by using two focus points and a passing point.
It is possible to dimension these points.



***Center point and Passing points***

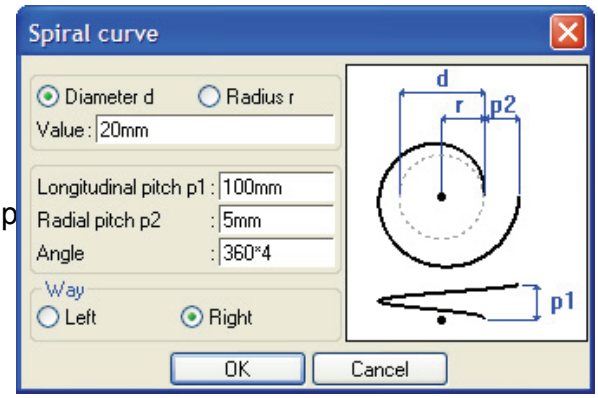


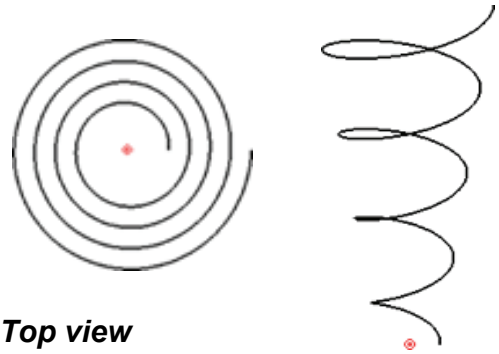
***2 Focus points and Passing point***

## Spiral

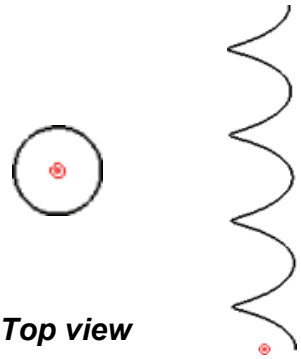
Icon :
Menu : *Curve* | *Other curves* | *Spiral* | *helix*

Create spiral manage by diameter/radius, two p
See the two examples below :



***Top view***

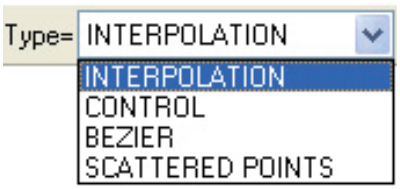


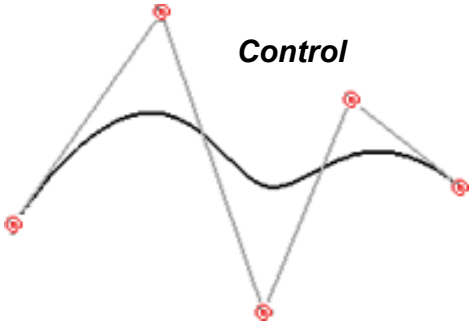
***Top view***

## Splines

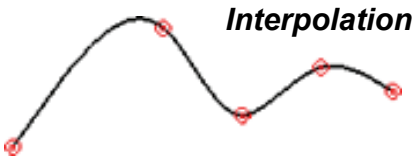
Icon :
Menu : *Curve* | *Splines*

Create splines by different types of point creation.



***Control***

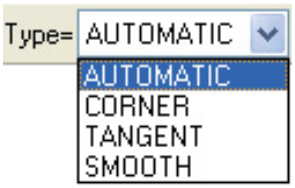


***Interpolation***

## C-Spline

Icon : 
Menu : *Curve* | *Other curves* | *C-spline*

It is possible to choose between various types of connections between the arcs :




A C-spline is only made up of lines and arcs

## Contour

Icon : 
Menu : *Curve* | *Contour*


The contour, or sketcher, function is one of the most important sketching functions in the woodworking application.

It can be used to create different line types :

- **Contours by points** => The path-points define the shape of the contour.
- **Contours by tracing** => The contour is made up 2D elements (lines, arcs, arc bends, etc.)
- **Mixed contours** (combination of the elements above)

# The contour function

## The contour function

The contour function is used to draw shapes composed mainly of lines and circles. This line is a single entity of segments that are connected one another and is not a juxtaposition of 2D entities (lines, circles, etc.).

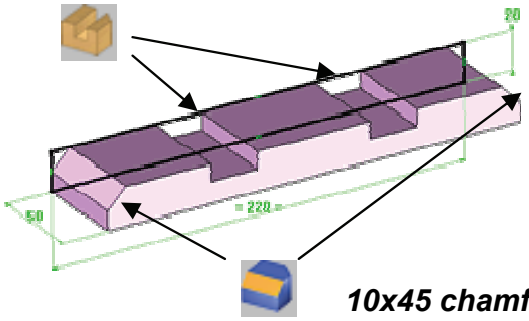
In the Wood module application, creating contours (the geometry of the part to be created) is an essential step in the 3D modeling of the object.

This versatile function can be used to draw most shapes. Therefore, it is important to have a strong command of this geometric tool. This function can be supplemented with the Wood functions in order to substantially simplify the drawing.

There are several ways of modeling parts:

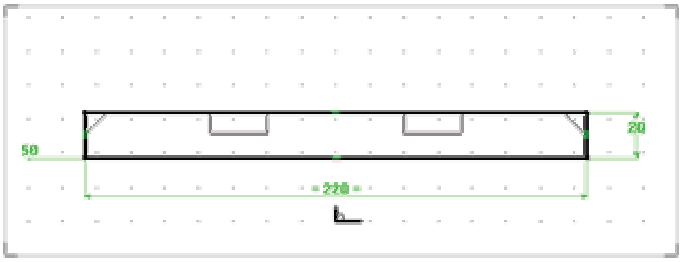
- **First method** : Once a simplified contour of the geometry of the required volume has been drawn, the Wood or CAD tools (drilling, grooving, folding, forming, etc.) can be used to work the tool by removing or adding materials to obtain the required result.

***Groove with 55 offset, 25x9***

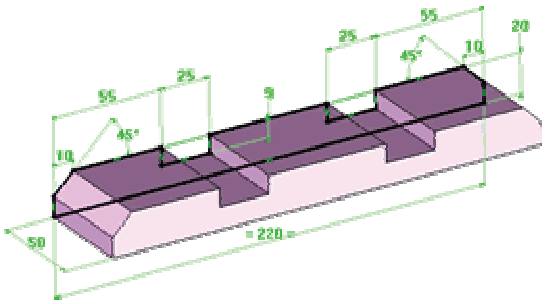



***10x45 chamfer***

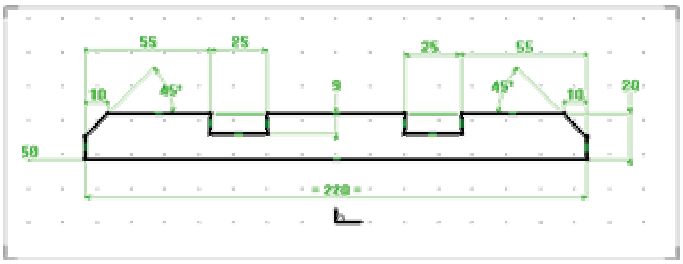
***Drawing of a contour encompassing the part to be created***




- **Second method** : The part is modeled by drawing a precise contour of the geometry of the part and then using the *Extrude* function.




***Drawing of the final contour of the part***




Both solutions are effective, offering some divergences and differing advantages.

The purpose of the operations is to significantly simplify the initial drawing and to automate the machining of the part in TopSolid'WoodCAM.

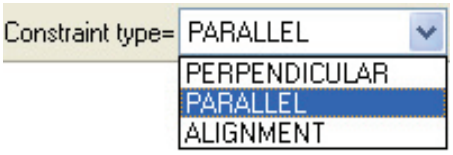
For example, making a fold requires a path and a tool, while drilling requires a diameter, depth and drill axis.

Creating the lines of the geometry means that complex parts can be drawn more quickly and that constraints can be applied to the dimensions in accordance with the user's requirements.

## Constraints

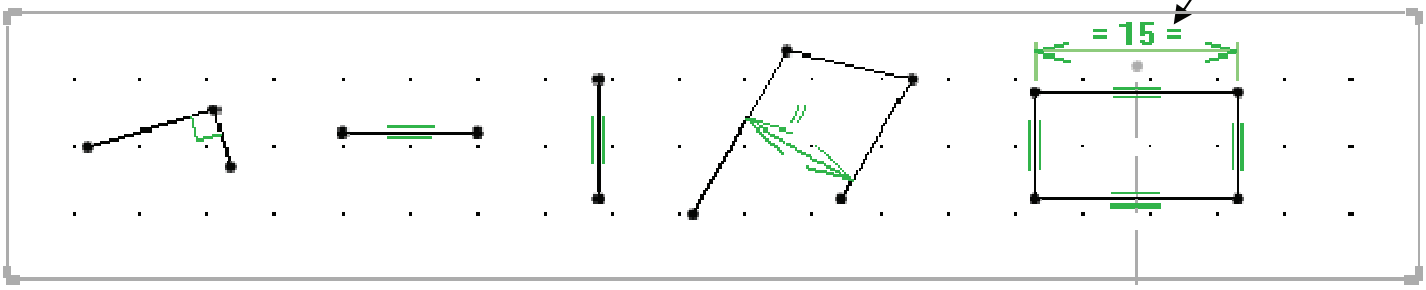
The contour is a function used to draw a contour point by point. TopSolid'Design automatically applies the geometric constraints in order to freeze the geometry of the drawing.

There are various types of constraint. The software only creates the alignment (horizontal or vertical) and the perpendicularity constraints automatically. However, constraints can be deleted or added manually using the **Tools | Constrain** menu.




***The symmetry constraint is applied using the wrench function on the dimension.***

***Note: Constraints can be identified by the following symbols***.




When creating a contour, the numerical values at the extremities determine:

- The relative coordinates in relation to the preceding point
- The angle with the horizontal axis
- The length of the segment.

When the cursor is horizontally or vertically aligned with a point, it changes shape and proposes to apply a constraint. (fig. 1)

If the waypoint is vertically or horizontally aligned with another point in the same contour, a constraint is automatically, symbolized by a grey line. (fig2)

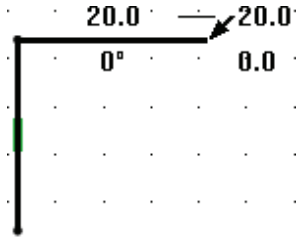



***fig1***

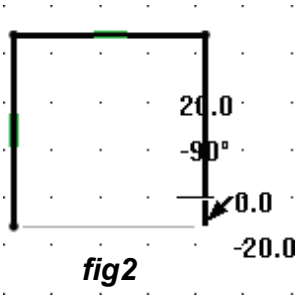
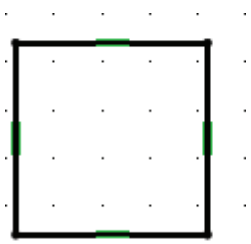
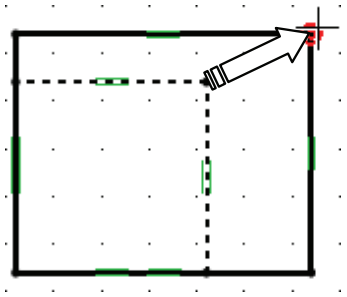


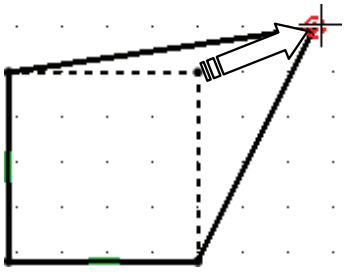

***fig2***





***Moving the upper right-hand point***

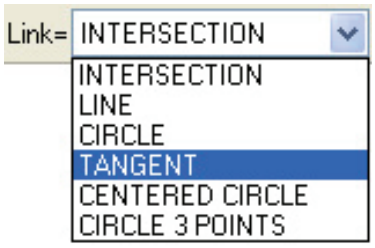
***Destruction of the two alignment***

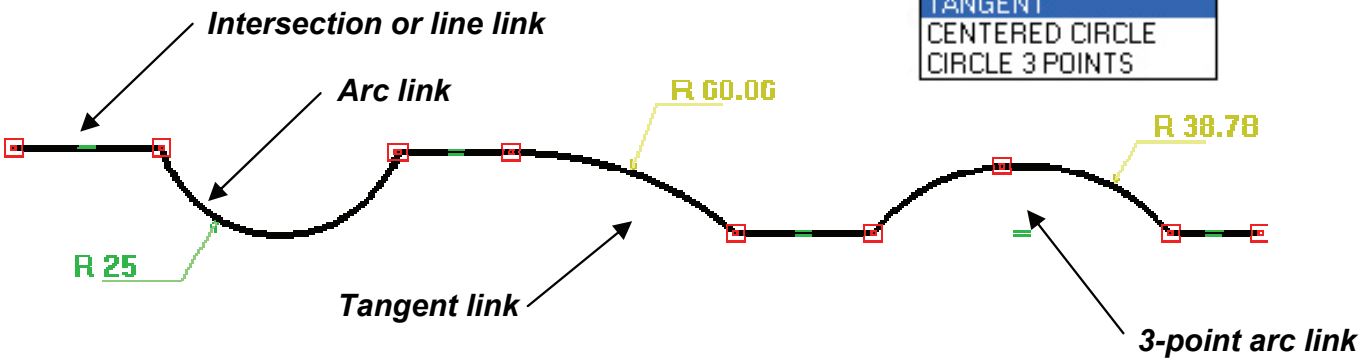


## Link and join

When creating the contour, the link and join options can be used if the shape to be drawn is made up of different entities (lines and arcs).

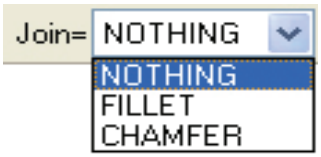
Connections between two points of a drawing are called links.

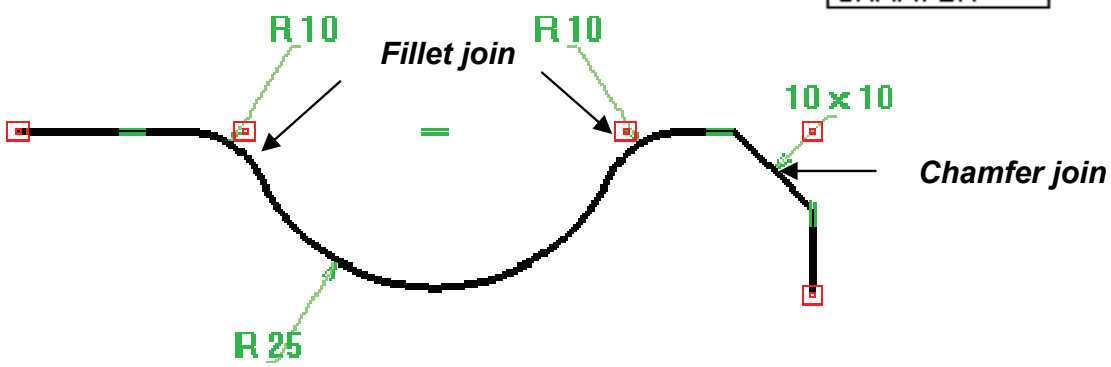



***The distinction is made between lines and intersections when a contour is drawn using a 2D entity.***

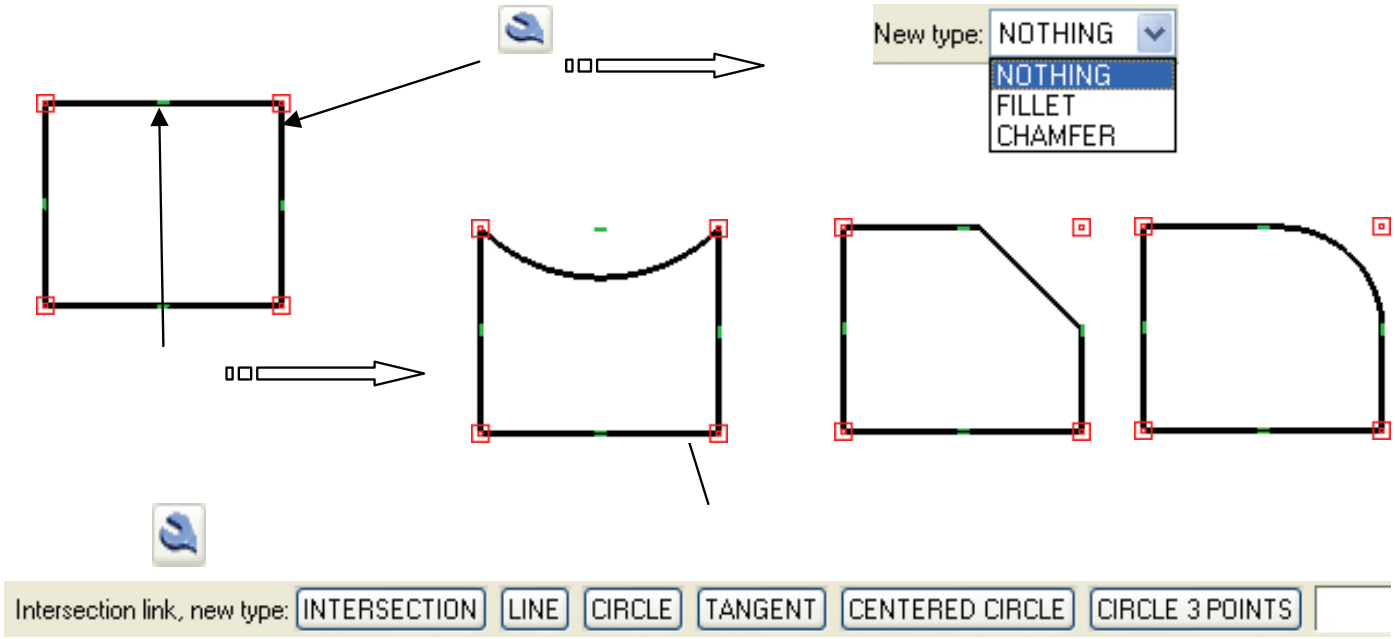
Connections between two links are called joins.



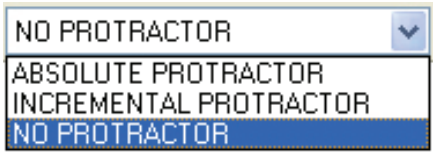
Once the drawing is finished, these options are modified using the function 

You can choose to edit the link or join of the contour depending on the selected zone.




## Protractor

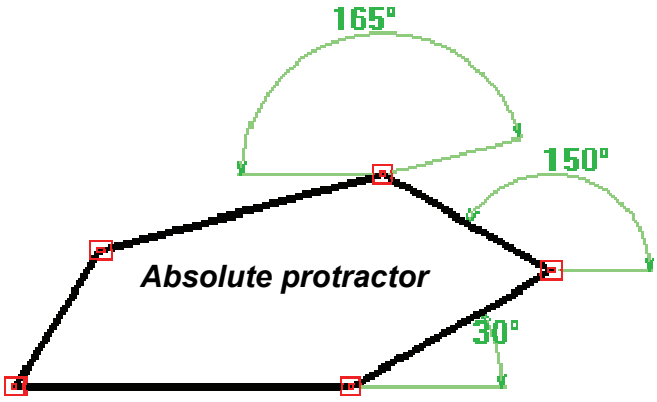
The advanced options,  , include the **PROTRACTOR** option, which is used to automatically create the angular dimensions when creating the contour.

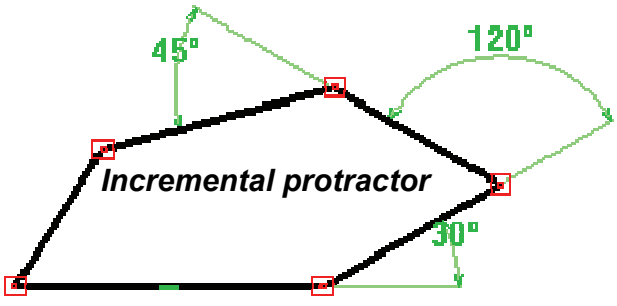



***No protractor:*** *the next line in the contour will not use the protractor and may be at any angle to the preceding line.*

***Incremental protractor:*** *the next line in the contour will be built with an angular pitch relative to the preceding line.*

***Absolute protractor:*** *the next line in the contour will be built with an angular pitch relative to the axes of the current coordinate.*



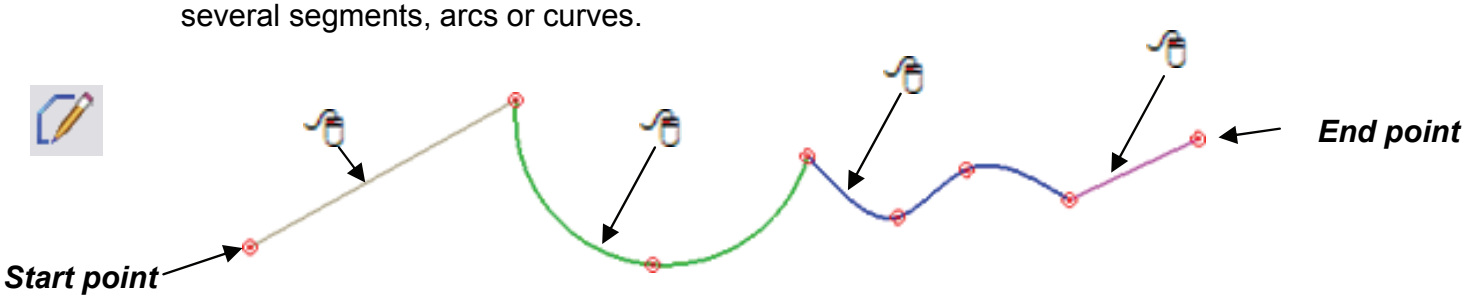



## Trace modes

When creating a contour using 2D entities, different modes can be used to limit the number of mouse clicks.

**ON SEGMENTS**: This is the default mode.

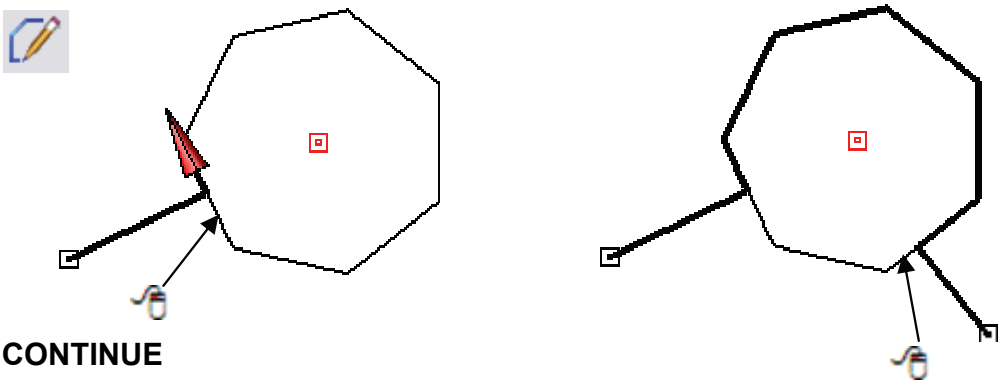
Click on each element that you want to use in the sketch.
All of the intermediate segments must be selected, even for composite lines made up of several segments, arcs or curves.




**ON COMPOSITES**

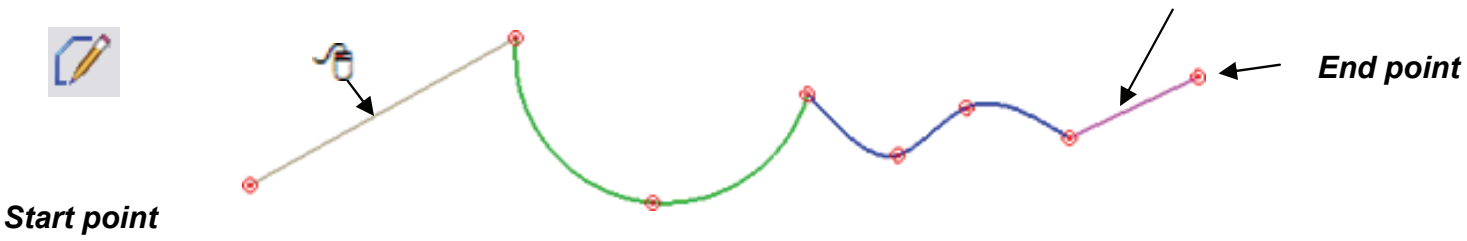
If a composite curve is selected in this mode, the designated segment is used as the point of departure of the composite curve. Enter the segment corresponding to the sketch and the system asks for the end segment.

You can then continue to draw the contour on other sketches or points.



**AUTO CONTINUE**

Composite curves are not necessary in this mode.
If using a succession of 2D entities, this mode can be used to create a contour passing through the different entities in just a few clicks, providing that the link between the entities is correct. ***(this is useful when importing dxf files)***.



## Dimension

The final step in the construction of a contour consists in defining the dimensions using the dimension and constraint functions.

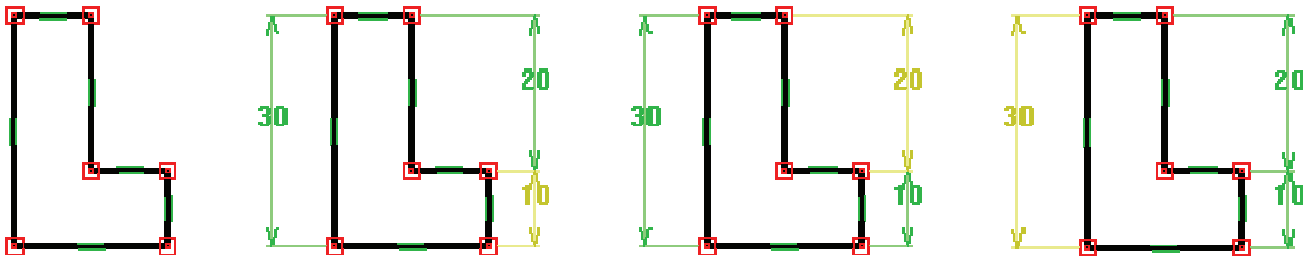
Points that are not dimensioned or constrained are free to move in the coordinate in which they were created. A poorly dimensioned contour can be easily deformed if the user does not check it or define sufficient dimensions.

The problem is to understand how contours are properly dimensioned.
Too many dimensions will apply too many constraints to the drawing, while insufficient dimensions will result in incorrect drawings.

Dimensions can be applied to the curve below in different ways.
The color of the dimensions changes depending on the order in which they are created.

- **Active dimensions** (green) can be modified and change the contour.

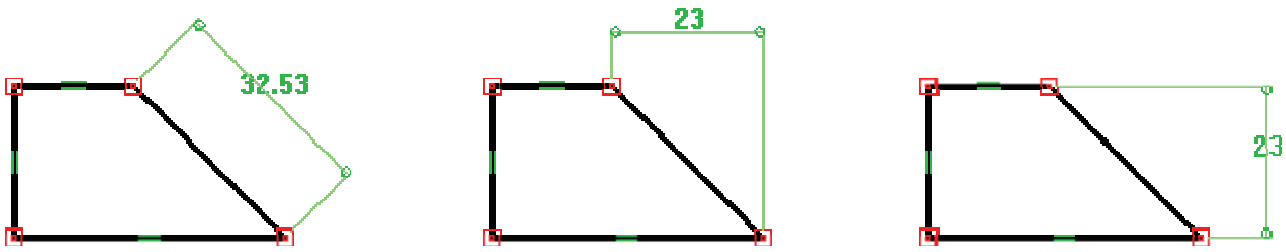
- **Passive dimensions** (yellow) cannot be modified and are the result of additional constraints. They are automatically deactivated by the software to prevent conflicts.




*In simple cases, the software automatically creates passive dimensions. More complex cases many contain an excess of active dimensions and consequently apply too many constraints to the contour.*

***The role of this function is to impose dimensions. Therefore, they must not be deleted, once the dimensions of the contour are correct.***

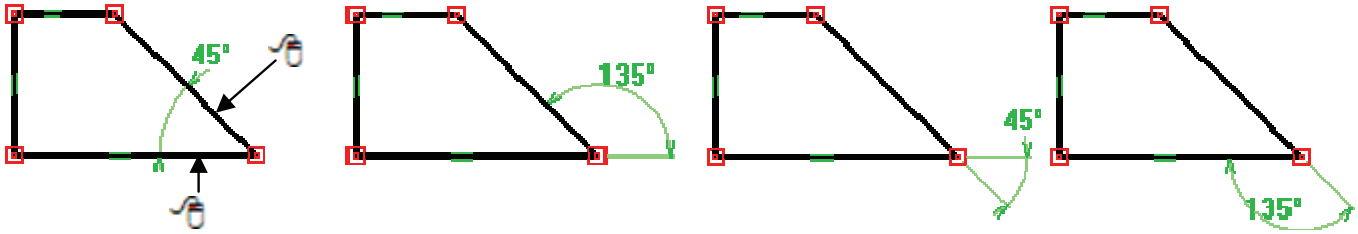
To apply a dimension, simply designate the corresponding points or lines.
The ends of the arrows change, depending on whether the dimensions are applied to lines or points.




**PARALLEL** **HORIZONTAL** **VERTICAL**

In the **FREE** mode, one of the three modes above can be selected, depending on the position of the cursor.

To apply dimensions to an angle, enter two lines. One of the four cases below is obtained, depending on the position of the cursor when validating.




## Modifying dimensions

The software contains two functions for modifications:

- Modification of elements *Edit* | *Modify Element.*
- Modification of parameters *Parameter* | *Modify.*

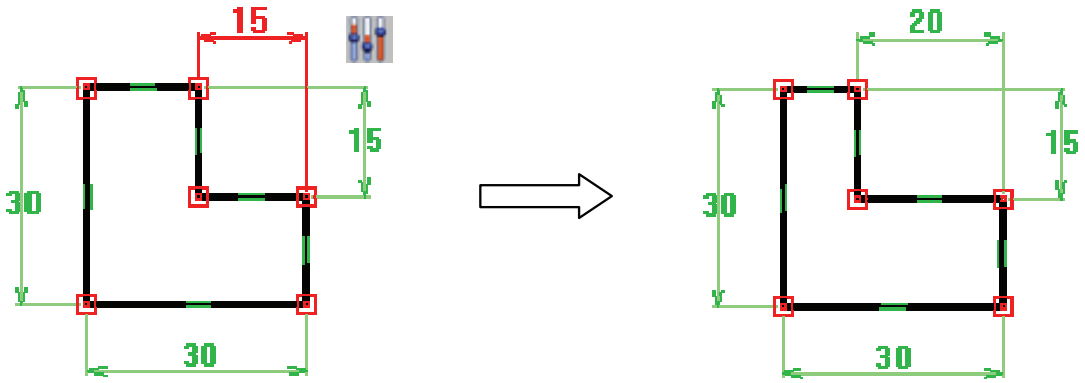
The function must be selected according to the element to be modified, since the functions cannot be used to modify the same items.

Use the function to modify dimensions.
This function modifies numerical values, such as the value of a drill hole diameter.

Use the function to modify an option or a physical characteristic of an element.
This function can be used to change the type of hole, to switch between a line and an arc or to edit an operation.

## Dependencies between dimensions

Each dimension is independent. Only the values of the selected dimension are modified.

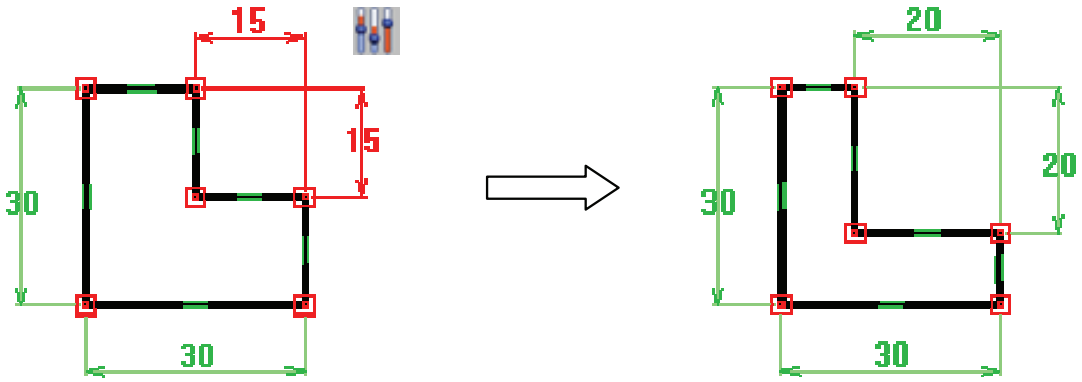


The dimension to be changed is indicated by a red highlight.
There are no links between the dimensions and they can all be modified independently.

Dependencies can be created between dimensions, in which case modifying one dimension will also modify all the linked dimensions.

The *Parameter* | *Merge* function is used to make two dimensions identical.
Enter the reference dimension to be applied and the dimension to be modified.

A red highlight is used to check the dimensions to be modified.




## Parameters

The notions of dependencies above are not enough for the effective design of a project.

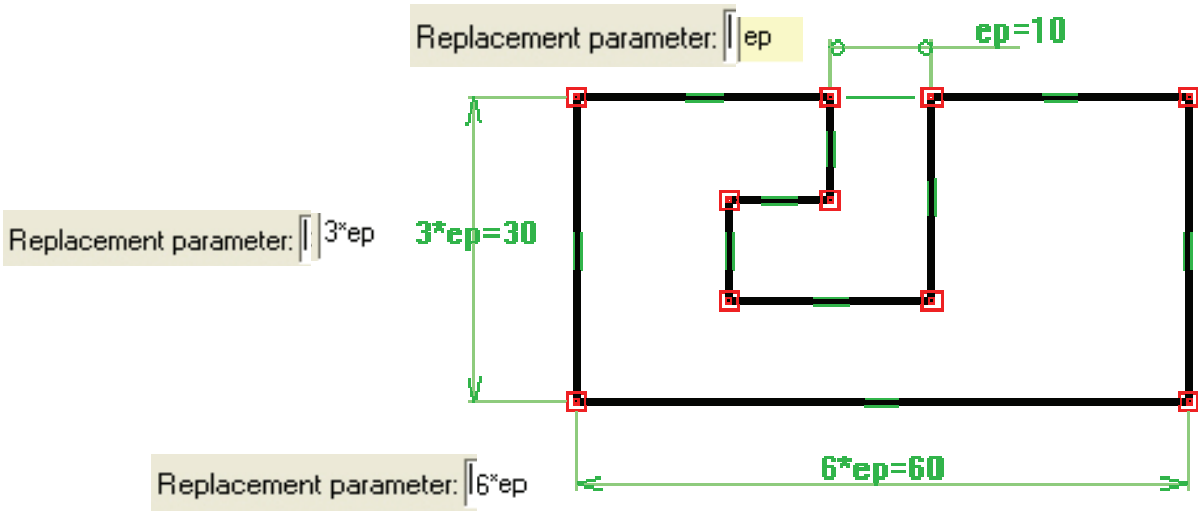
For example, how can the thickness of a base be made half that of a panel?

Parameters are used to answer this question.
If the term "th" is the thickness of the panel, then the term "th/2" is the thickness of the base.

Parameters are created using the *Parameter* | *Create* function.
Select the unit, value and name.
The default unit is length in mm. Parameters can also be created for angles, numbers, volumes, etc.

Use the  function to apply a parameter to a dimension. Then enter the name of the parameter in the **Replacement Parameter** cell using the **REPLACE** option.

Equations can also be entered in this cell.



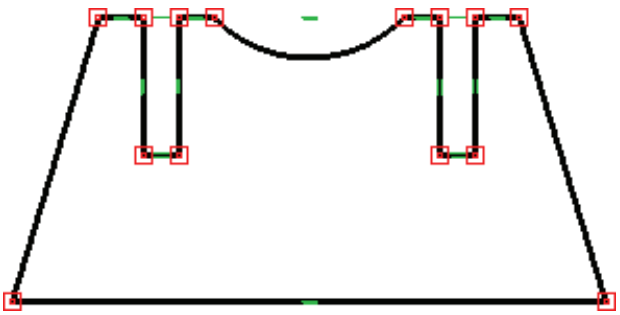

## Types of contours

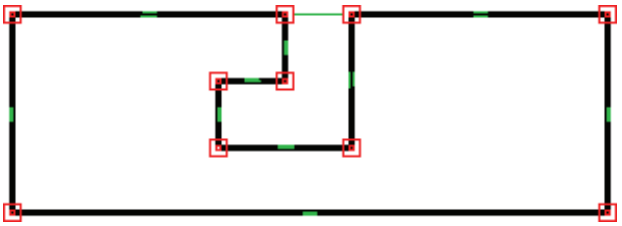
The notions explained above are the options of the contour function.
The following drawing methods can be used, depending on the part to be modeled.

The same method is not used to draw a cupboard or the foot of a unit.

## Point by point contour

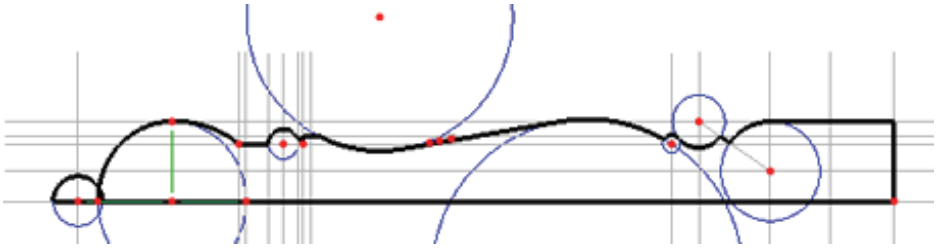
The drawing of the contour is defined by the different waypoints and the dimensions are used to size the part.





## Traced contours

Traced contours are built by taking an existing geometry (lines, arcs, sketches, etc.) and creating the corresponding shape.



## Mixed contours (combination of the elements above)

The two preceding methods can be combined in a single contour.

# Points

Some functions use passing points (center point of circle, starting point …), and you need to create them if they don't exist.

The Tools | Points menu includes a list of geometric points that can be used to create the points.

**You can create points in two ways :**

- Create the point before using a function.

Open the Tools | Points menu, then select a point from the list. You can also click on the  icon and select a point from the list of special points.

This icon bar matches the list of points in the Tools | Points menu below.

- Create the point on the fly while using the function.

When the function asks for a point, you can use the  shortcut to create the point and then resume use of the function.

***You can combine several creations of points.***
***For example, create the middle point of two center points.***

Certain points are used frequently.

You must be able to create the following points in particular :

***Cartesian point***

***Offset point***

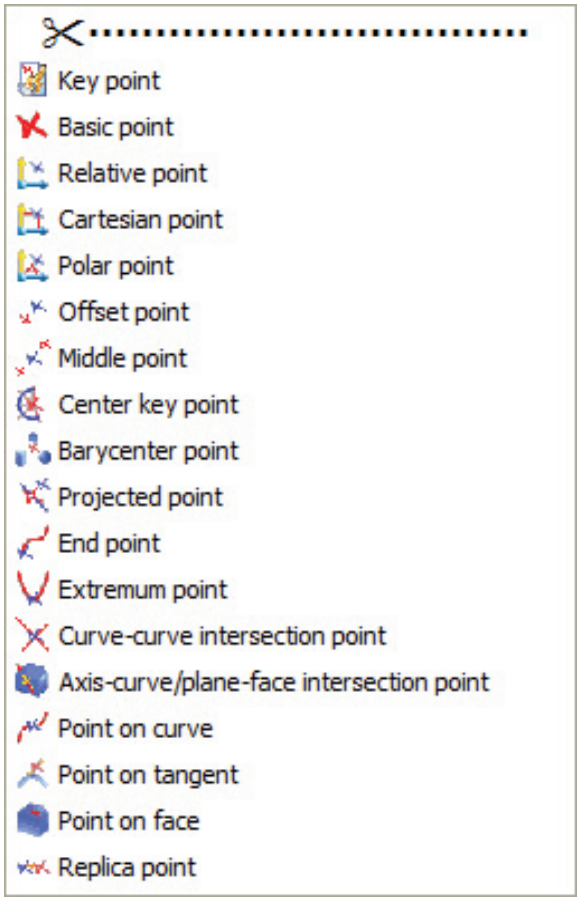
***Middle point***

***Center key point***

***Curve-curve intersection point***

***Point on curve***

**Replica point**

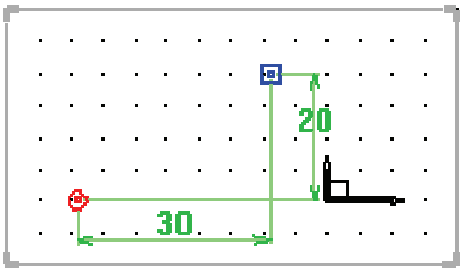


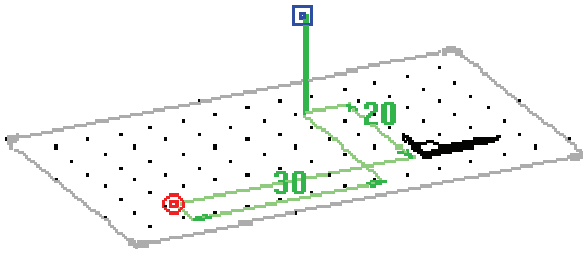
Example of use of points.

**Key :**

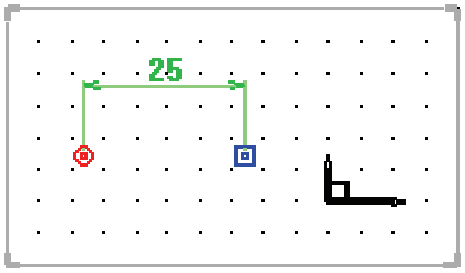
Original point
Point created

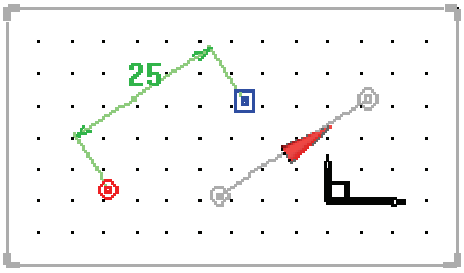
**Cartesian points:** are used to create points using the three X Y Z coordinates of the current reference.
Only the X and Y dimensions appear on the screen.



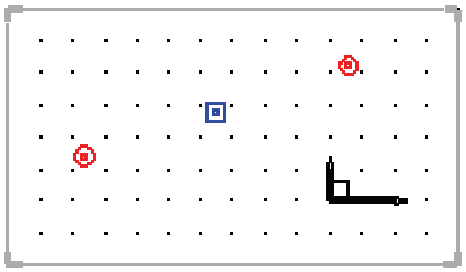



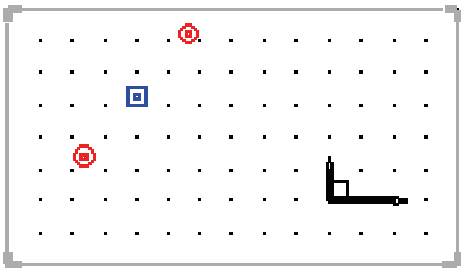

**Offset points:** are used to create points that are offset from a reference point.
Points can be offset in a different direction from the axes of the reference.



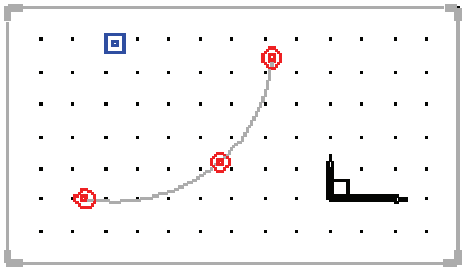



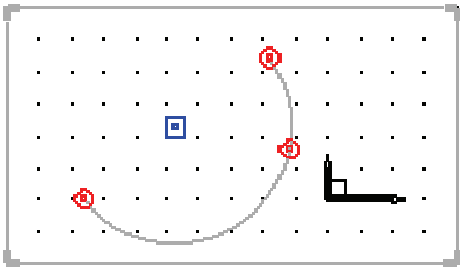

**Middle points:** are used to create a point in the middle of two points.



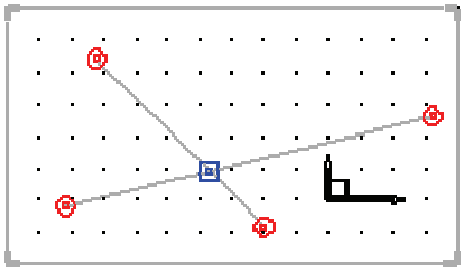


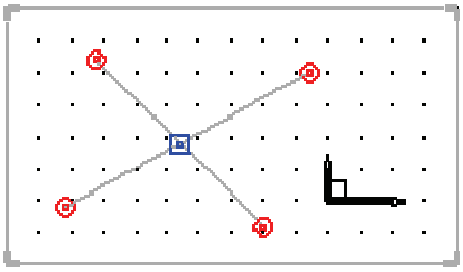
**Center key points:** are used to create a point in the center of an element (curve, edge, etc.). Entering a line when creating a center point selects the middle of the line.



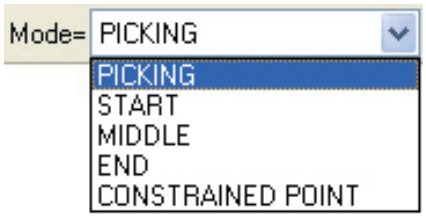


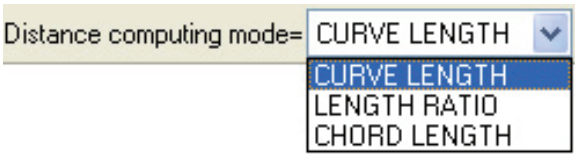
**Curve-curve intersection point:** is used to create an intersection point between two curves or edges. This type of point can be retrieved by pressing and holding the left-hand mouse button.

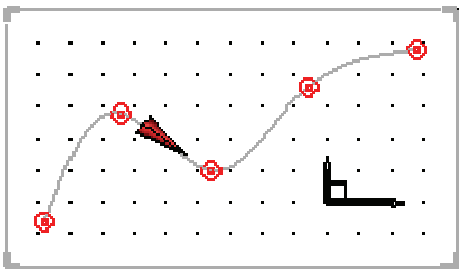


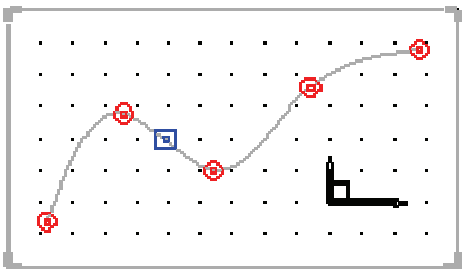


**Point on curve:** is used to create a point on a curve. (5 types of position - Designation, Start, Middle, End, Constraint). Different types of length can be processed in constrained mode.





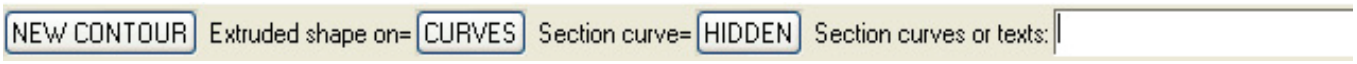
All of the points above remain associated with their point of origin when created.

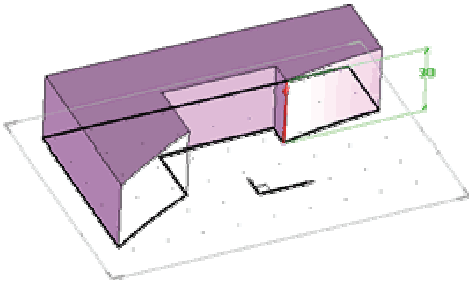
# Simple shapes

# Extruded

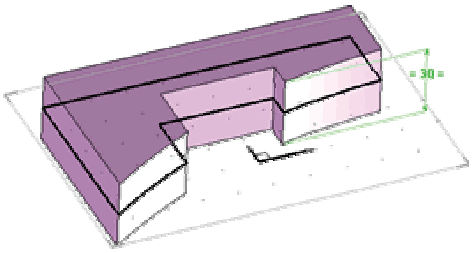
*Icon :* 
*Menu :* Shape | Extruded

This function is used to created surface or volume shaped by extruding a contour or a surface. The contour can be open or closed.
Open contours produce surfaces, while closed contours produce solids.

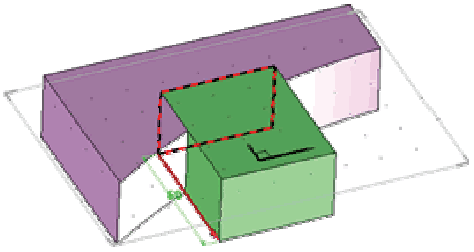


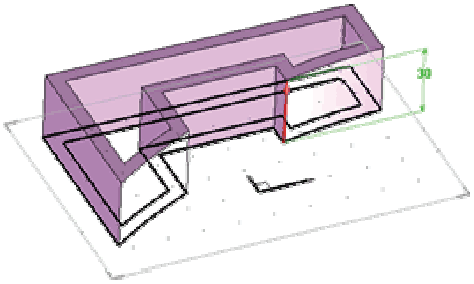
***NORMAL alignment***



***CENTERED alignment***



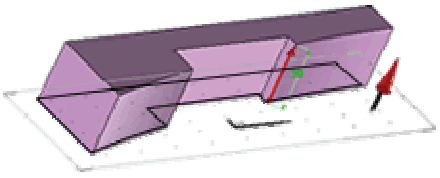
***Extrusion of a surface***



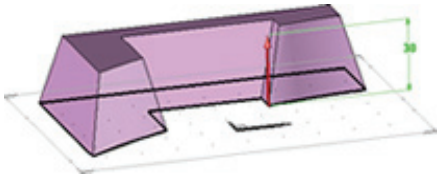
***Selection of several curves*** 

Extruded shapes can be made using the advanced options  :

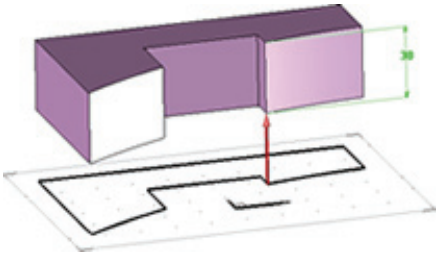
- by imposing a direction of extrusion. (enter an axis)
- with an offset extrusion curve (upwards or downwards).
- by automatically giving a draft angle all the surfaces of the solid.

***Extrusion in a direction***



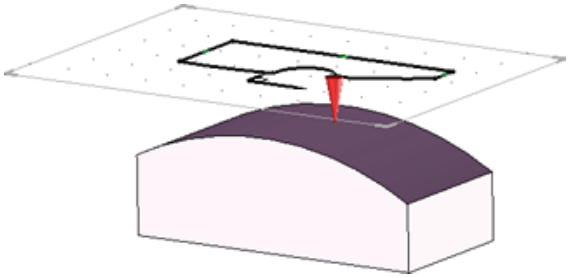
***Extrusion with taper angle***



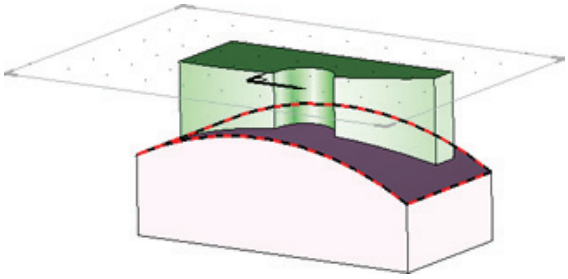
***Extrusion with offset distance***

There are two extrusion modes:

- the **HEIGHT** mode, described above, in which the user enters a value,
- the **TO PLANE** mode, in which the shape is trimmed by a plane or a surface defining the height of extrusion.



***Show the curve***



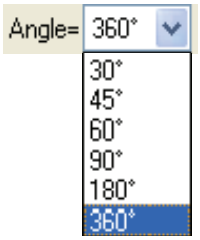
***Show the limitation surface***

## Revolved

Icon : 
Menu : *Shape* | *Revolved*

This function is used to created surface or volume shaped by revolving a contour around an axis.


***Revolved through 85°***




***A list of predefined values can be used***

***Revolved through 180°***
***Selection of several curves***

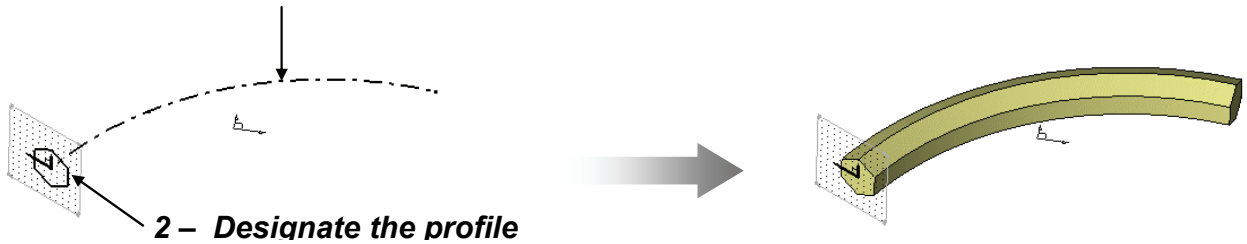
## Pipe

Icon : 
Menu : *Shape* | *Pipe*

Is used to create a shape by running a section along a guide curve.
There are different available modes:

- the **ON CURVES** mode uses any curve drawn by the user,
- the **WIRE SHAPE** and **TUBE SHAPE** modes for circular shapes.


***1 – Designate the guide curve***

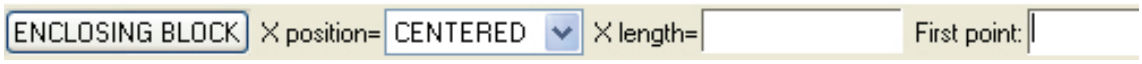


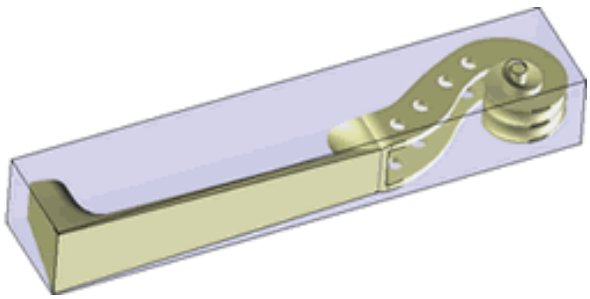
***2 – Designate the profile***

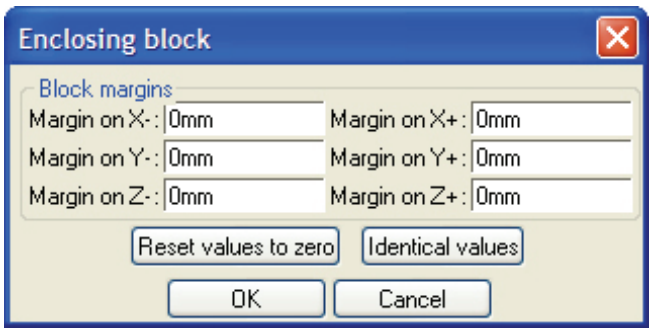
## Block

Icon : 
Menu : *Shape* | *Block*

Is used to create paralleliped blocks using an alignment point and the three lengths. Blocks can be created to encompass a complex shape, with or without margins, in order to create a stock model





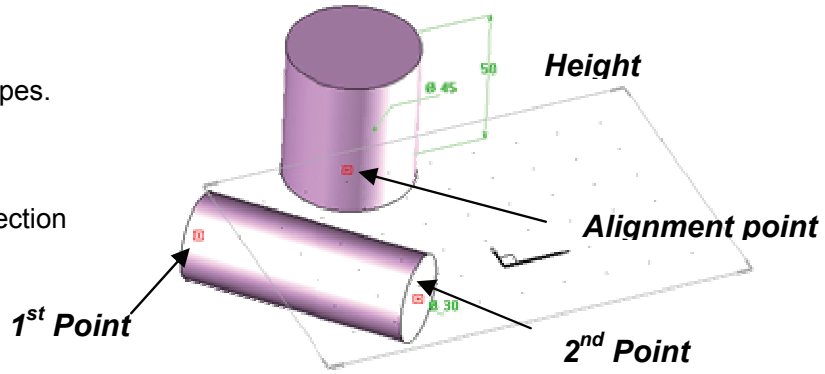

***Different margins can be applied to each surface.***

## Cylinder

Icon : 
Menu : *Shape* | *Cylinder*

Used to create solid, cylindrical shapes.
You can create:

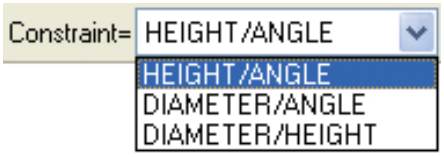
- Using two waypoints
- Or a length and its direction

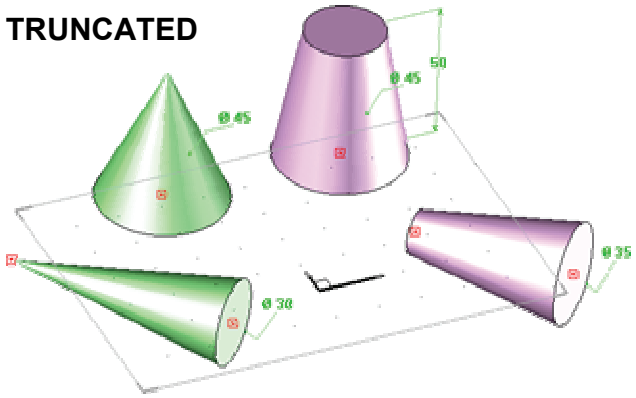



## Cone

Icon : 
Menu : *Shape* | *Other shapes* | *Cone*

Used to create two types of cone: **POINTED** or **TRUNCATED**
Different values can be adjusted for truncated cones.

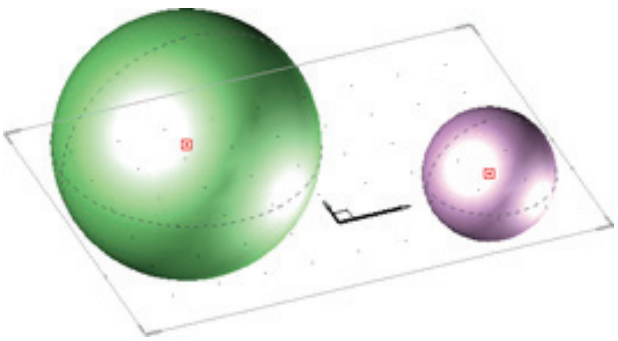


## Sphere

Icon : 
Menu : *Shape* | *Other shapes* | *Sphere*

Used to create spheres. The radius or diameter can be set.

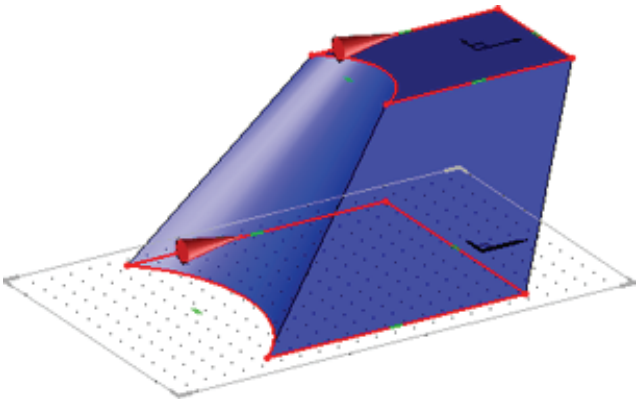


# Complex shapes

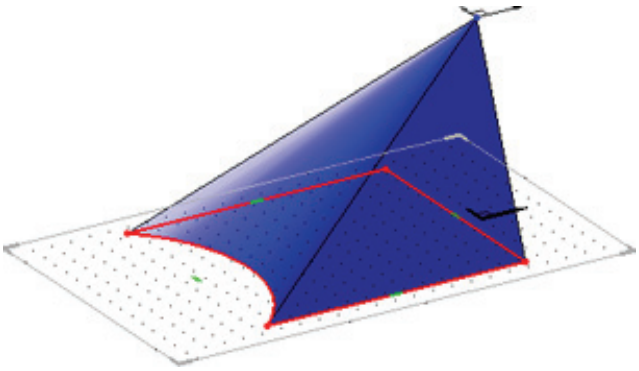
## Ruled

Icon : 
Menu : *Shape* | *Other shapes* | *Ruled*

Used to create a ruled surface or a solid between two curves or a curve and a point.
See notion of the origin of contours.



***Shaped ruled on two contours***



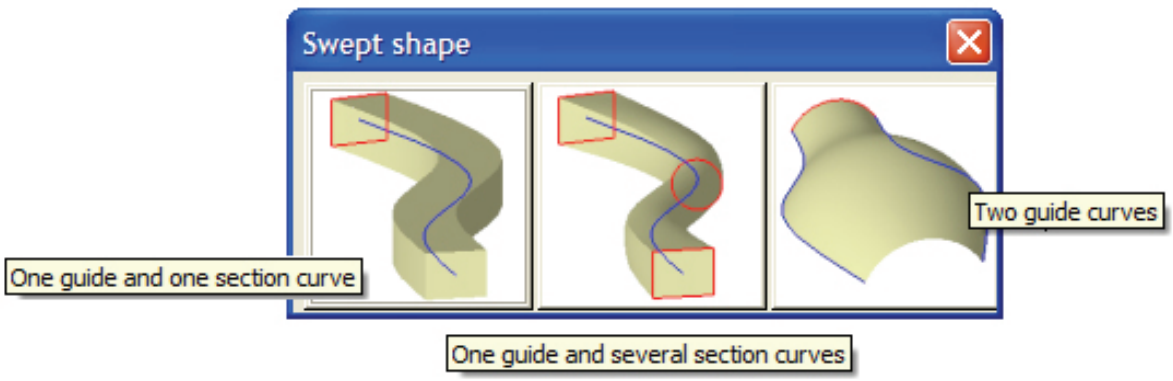
***Shape ruled on contour and point***

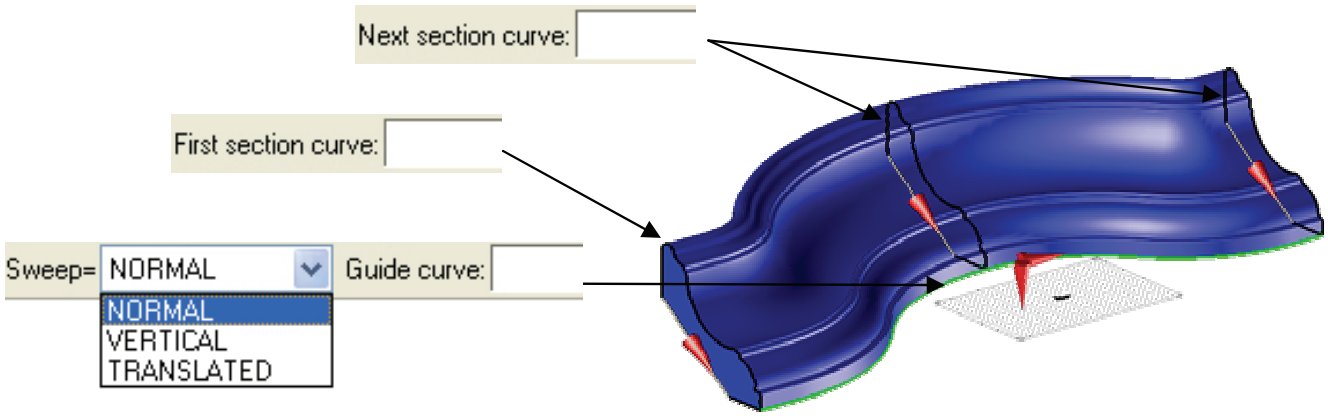
## Swept

Icon : 
Menu : *Shape* | *Other shapes* | *Swept*

Similar to the Pipe function, the Tube function offers more options, especially with regard to the choice of sections and guide curves (selection of several sections and guide curves, check sweep of the section in relation to the guide curve)

All types of complex surfaces can be created using these three modes.






## Multi-criteria selection.

By default, the selection icon is inactive. It becomes active in certain CAD functions and you can enter one or more entities:

- manually, by designating them in the 3D document
- according to physical criteria (color, layer, by box, etc.).

This function may be useful to change the color or the line type of several 2D entities and to repeat several parts.

Clicking on the selection icon opens a new tool bar that includes a range of options that can be used to select the elements you need.

The bar is divided into two zones.

The first is the 3-mode selection zone: Add, Filter, Remove.

The second zone is for the selection criteria.

***Examples of selection criteria :***

By type of part: (Shape, Contours, Parts, Points, References, etc.)

By layer (select all the elements in a given layer).

By physical criteria, superior or inferior.


A report in the alphanumerical bar indicates the number of selected elements.

# Control elements, display and layers

## Control elements

This function is used to hide or display the control elements.

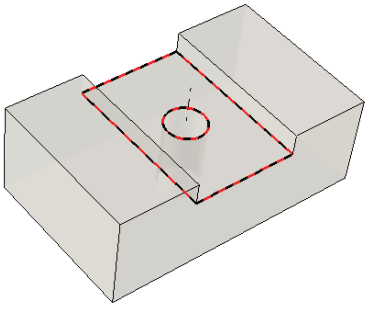
By default, certain functions hide the control elements during an operation.
This is to prevent the document from becoming too cluttered during construction.

***By way of example, the Extrude function automatically hides the contour section, while the Unite and Pocket functions hide the tools in use.***

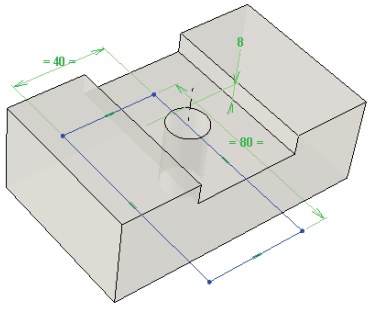
## Local mode

This option is used to select the operation whose elements are to be displayed or hidden.

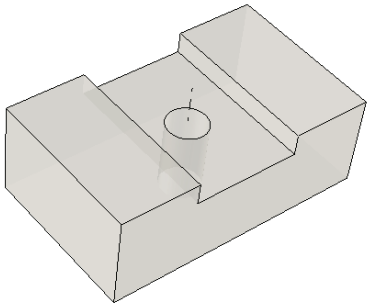
***Example: viewing the control elements of the pocket.***



***Select the operation***




***View the control elements***

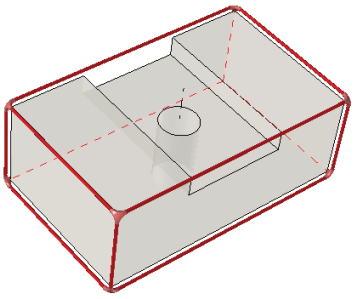


***Click on the operation again to hide the control elements.***

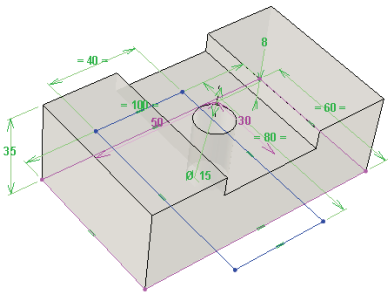
## Global mode

This function is used to hide or display all the controls of an element.
In this mode, use the **View** option to view or hide the control elements.

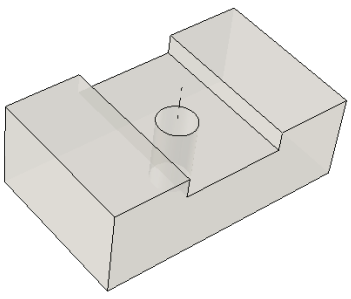
***Example: viewing the control elements of the part.***



***Select the part***




***View the control elements that make up the part***

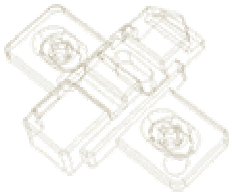


***Click on the part again to hide the control elements.***

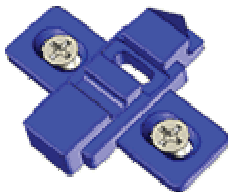
# Display

The display can be configured and adjusted for both the appearance of the 3D rendering (wire mode, shaded, realistic), the zoom and the working windows.

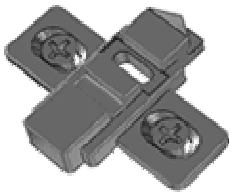
Click on the Rendering icon to open the list of views. The main display categories are shown below.



***Wire***



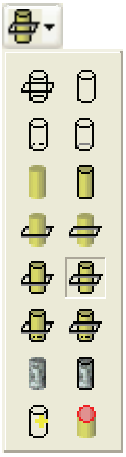
***Shaded***



***Realistic***

***Syntheti***

Click on the Rendering icon to open the list of types of rendering.
Along similar lines, there are also pallets for colors and types of lines and points.
Multiple selections can be used to edit several entities at the same time.



***Rendering***

***Colors***

***Line types***

***Point types***

There are two pallets for zooms and views.

Top view
Perspective
Configure view
Orientate view
Share view
Delete view
Vertical mosaic
Horizontal mosaic

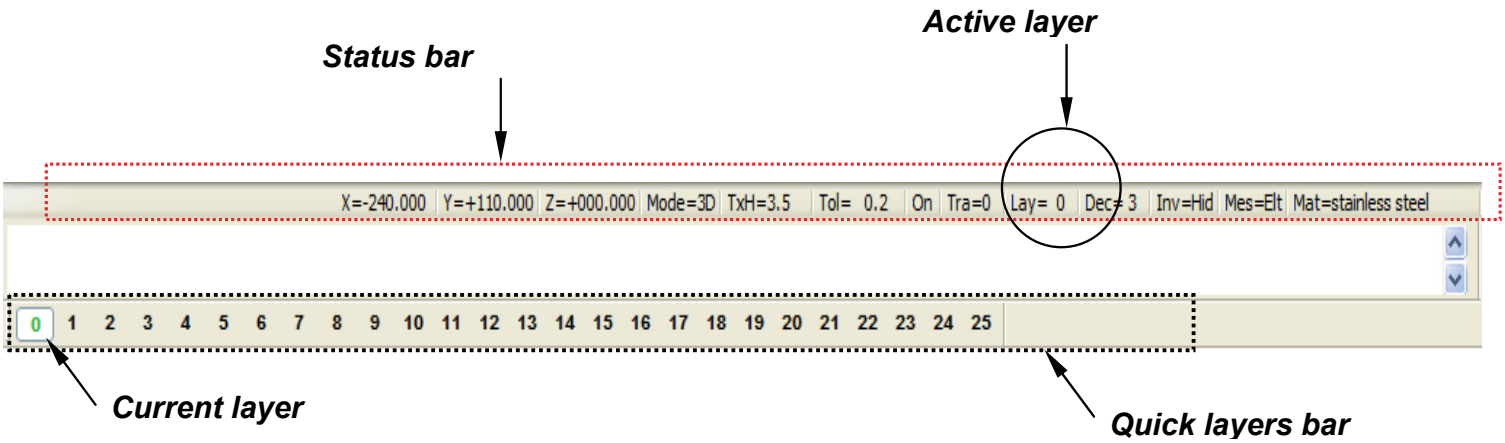
***Views***

Zoom in
Zoom out
Zoom on item
Zoom 100%
Zoom on current coordinate system
Undo zoom

***Zooms***

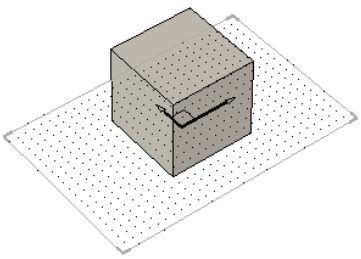
# Layers

Quick layers can be used to draw on the screen as is you were using several overlays. By default, there are 15 layers. The number of layers can be increased to 1,000, numbered from 0 to 999, using the ***Tools|Options|Icon bars*** command.
You can make one or more layers active or inactive at any time. This action makes the elements associated with the corresponding layer visible (active) or invisible (inactive). This action is similar to adding an overlay onto a drawing board.
When creating a new document, by default TopSolid'Wood assigns a current layer (0). This layer is green and is visible in the quick layers bar and the status bar.

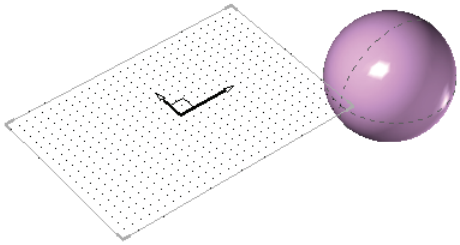



**« All created elements are stored in the active layer »**

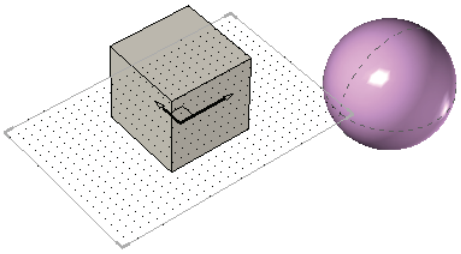
***Example :***



***Block inserted on the current layer (1)***



***Sphere insert on the current layer (2). Layer 1 is inactive***

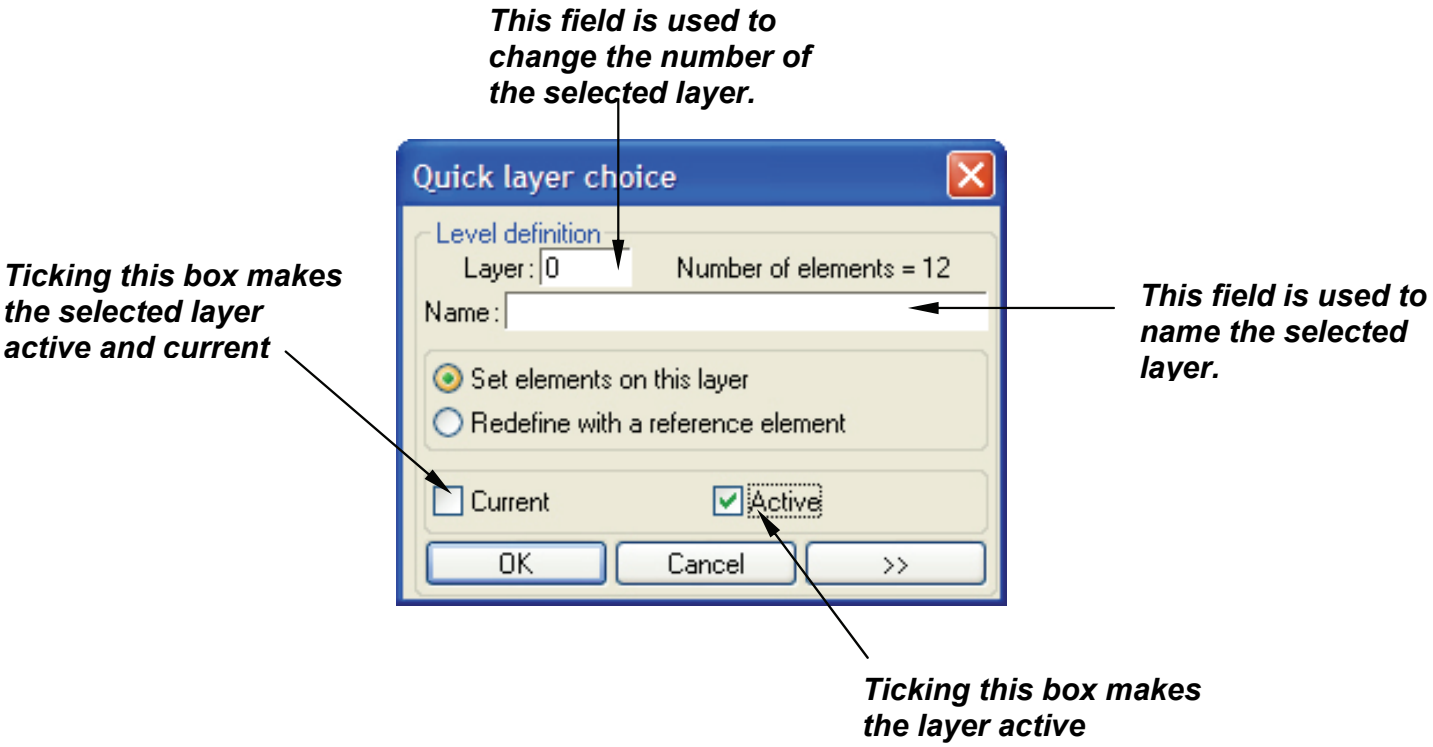


***When layer 1 is activated, the block reappears and layer 2 remains the current layer***

## Quick layers bar

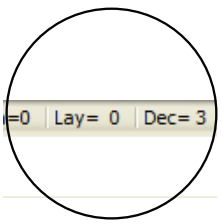
The quick layers bar is used for quick and easy layer management.


The left-hand mouse button **activates** the selected layer (red).
The middle mouse button makes the selected layer the **current** layer (green).
The right-hand mouse button opens the dialog box shown below.




## Layer editor

The layer editor is used to view all of the layers and to perform a number of operations (Group, Freeze, Activate, etc.).




Open the editor by clicking on the Layers tab in the status bar.

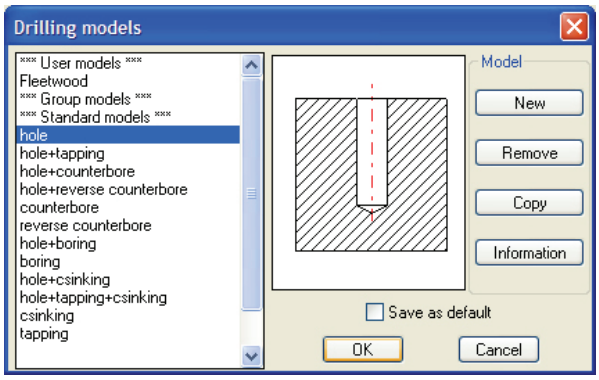
Refer to the on-line help for more details.

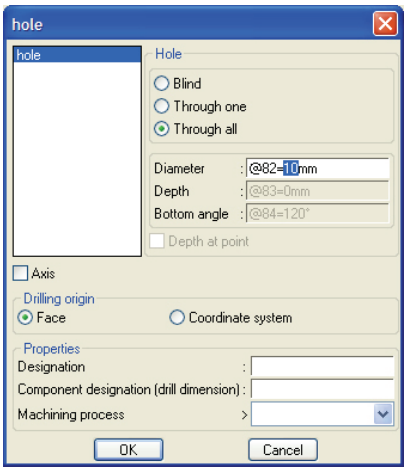
# Operations

# Drilling

Icon : 
Menu : *Shape* | *Drilling*

Is used to place drill holes on the shapes.




***Drilling templates are essential when using the multi-drilling function in the Wood menu.***

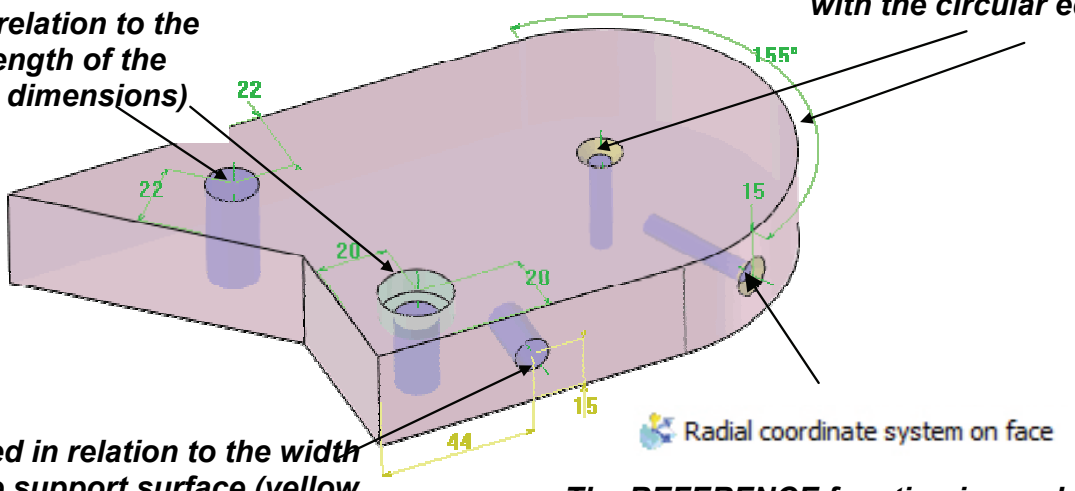
There are three ways of positioning a drill hole:

- **DYNAMIC**: the software automatically places the axis of the drill hole in relation to the edges
- **NON-DYNAMIC**: the user enters the position of the drill hole in relation to the edges.
  In certain cases, a dimension appears in yellow to indicate that the drill hole is centered on the support surface of the part
- **REFERENCE**: in some cases, when the surface to be drilled is not flat, a coordinate reference must be used to position the axis of the drill hole.

***Examples of possible positions of the drill hole on the surface :***

***Drill hole that can be positioned in relation to the width and length of the shape (green dimensions)***

***Drill hole concentric with the circular edge***



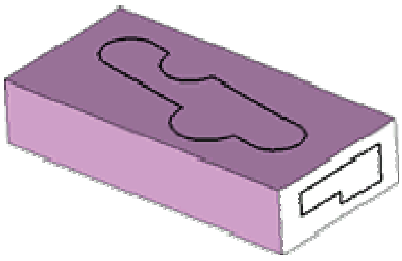

***Drill hole centered in relation to the width and length of the support surface (yellow dimension)***

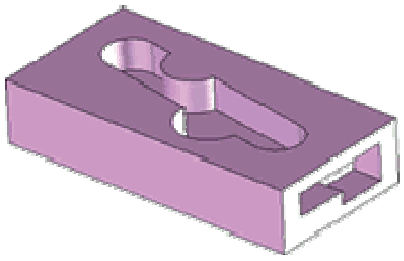
***The REFERENCE function is used to drill on uneven surfaces***

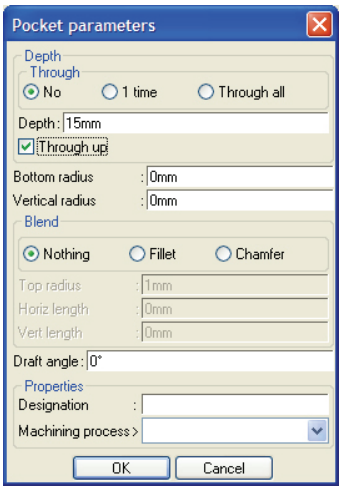
## Pocket

Icon : 
Menu : *Shape* | *Pocket*

Is used to remove material from solid shapes using a closed contour.



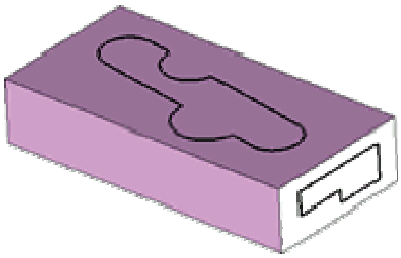


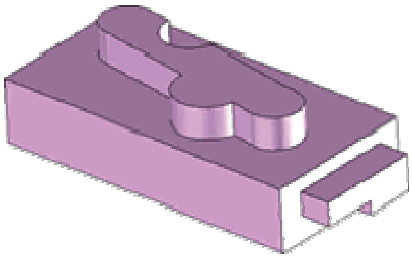



## Boss

Icon : 
Menu : Shape | Boss

Is used to add material to solid shapes using a closed contour.





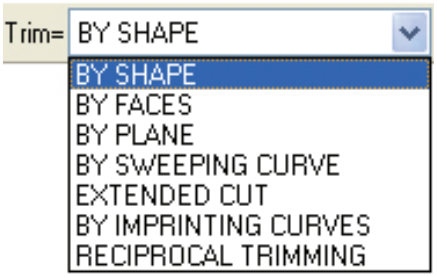

*For both of these functions, the contour must be in a plane that is parallel to the surface to be changed. Depth is measured from the surface.*

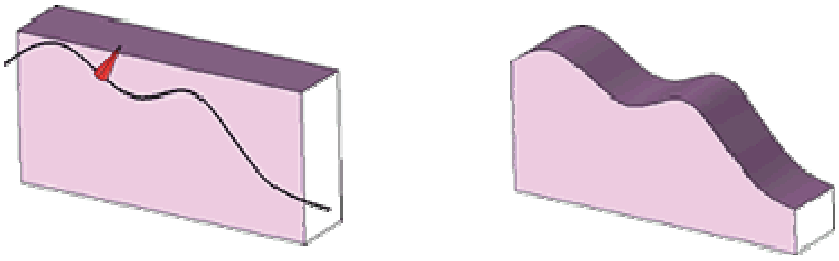
*The **CONTOUR IN PLACE** option means that the contour defines the height of the operation. This option is used to perform these operations on uneven surfaces.*

## Trim

Icon : 
Menu : *Shape* | *Trim*

Is used to remove material from solid shapes using different entities.

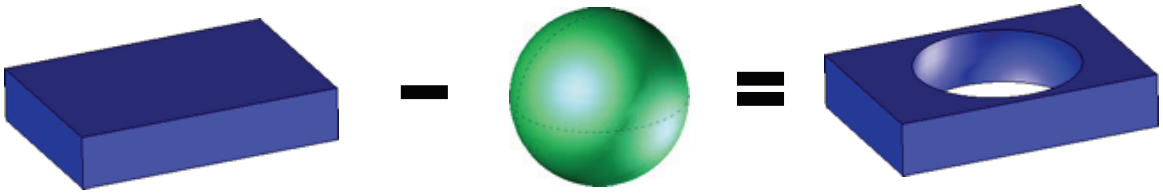


## Subtract

Icon : 
Menu : *Shape* | *Subtract*

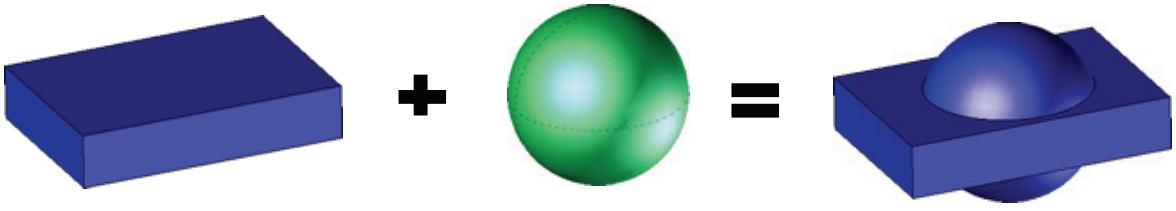
Is used remove one shape from another



## Unite

Icon : 
Menu : *Shape* | *Unite*

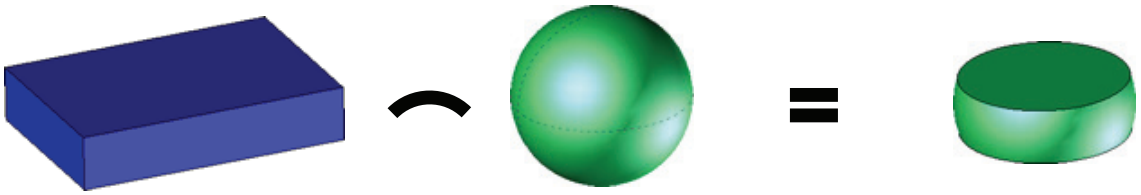
Is used add one shape to another



## Intersect

Icon : 
Menu : *Shape* | *Other Operations* | *Intersect*

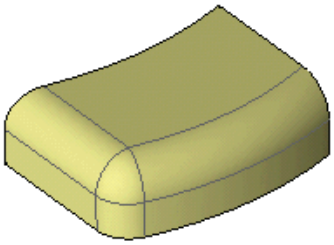
Is used to obtain the intersection of several shapes.



## Fillet

Icon : 
Menu : *Shape* | *Fillet*

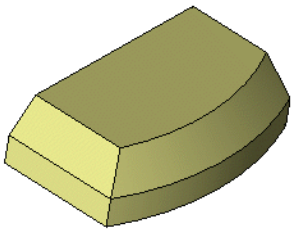
Creation of constant or variable fillets.
It is possible to create rounded corners.



## Chamfer

Icon : 
Menu : *Shape* | *Chamfer*

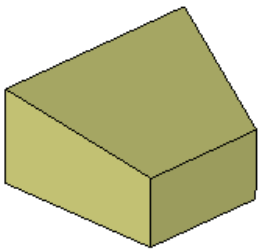
Is used to create chamfers on the edges of volume parts.
Chamfers can be created with one length and an angle or with two lengths.



## Draft

Icon : 
Menu : *Shape* | *Draft*

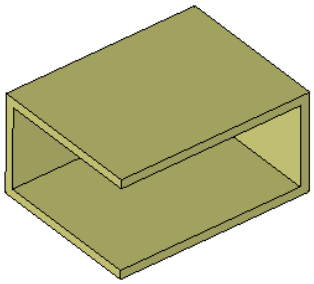
Is used to apply draft angles to one or more surfaces of a solid shape.
The surfaces must be flat, cylindrical or conical.
It is possible to create draft angles with flat planes.



## Shell

Icon : 
Menu : *Shape* | *Other Operations* | *Shell*

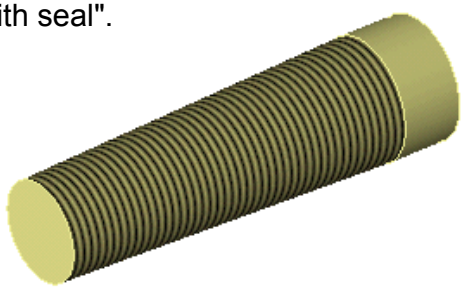
Is used to transform a solid into a shell by removing the surfaces.

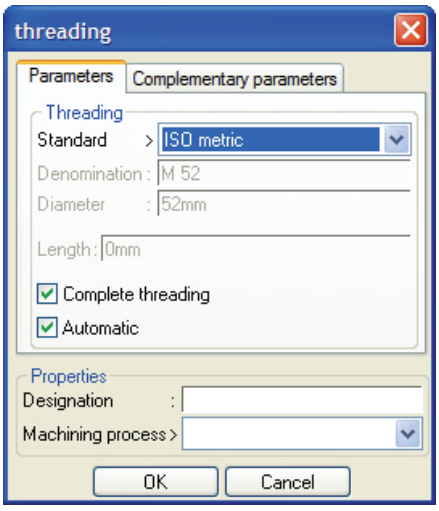


## Threading

Icon : 
Menu : *Shape* | *Other Operations* | *Threading*

Is used to thread cylindrical parts to different standards. A conical thread can be created on a conical or cylindrical shape by selecting the standard "Gas with seal".

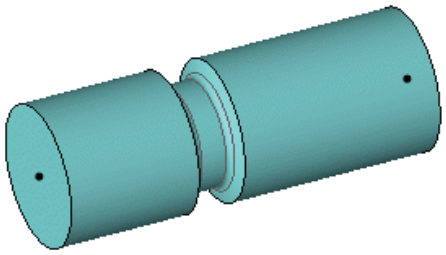


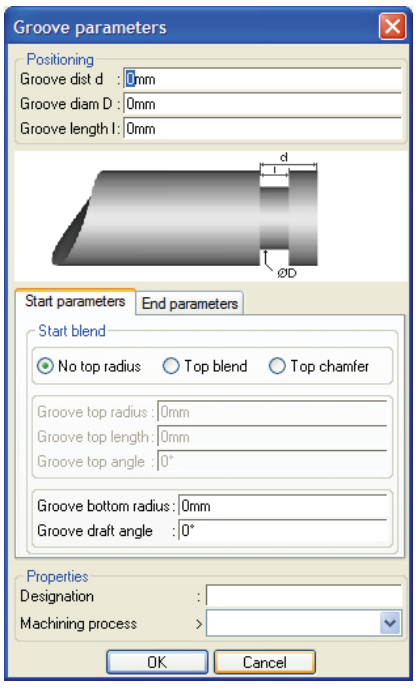



## Groove

Icon : 
Menu : *Shape* | *Other Operations* | *Groove*

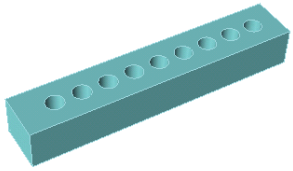
Is used to create grooves on cylindrical surfaces.



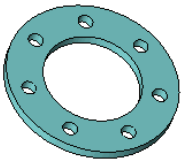


## Propagation of operations

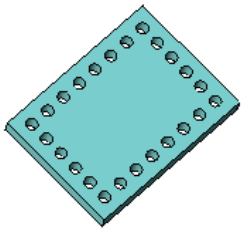
**Example with a drilled hole.**



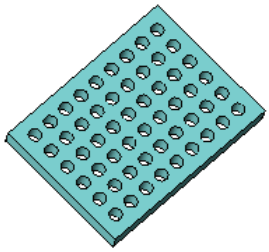
***Linear propagation***



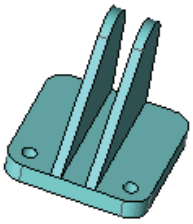
***Circular propagation***



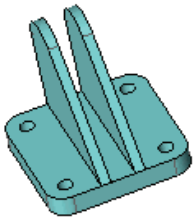
***Rectangular propagation***



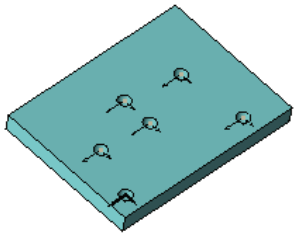
***Matrix propagation***



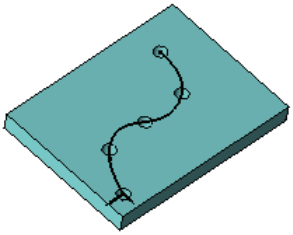
***Plane symmetry***



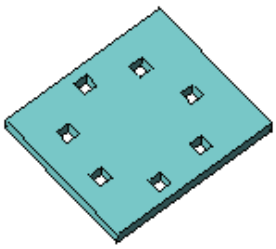
***Double plane symmetry***



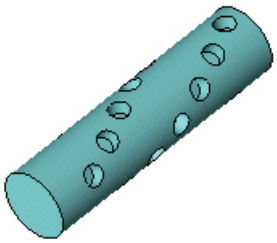
***By coordinate system***



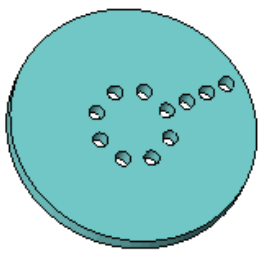
***On curve***



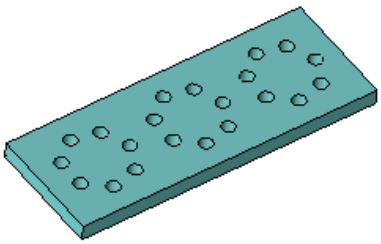
***Propagation by circular translation***



***Spiral propagation***



***Sum total of propagations***

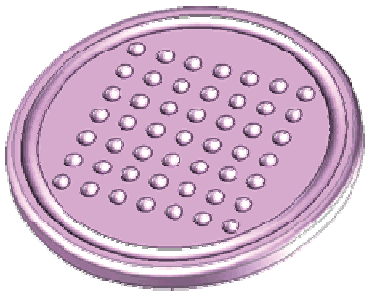


***Multiplication of propagations***

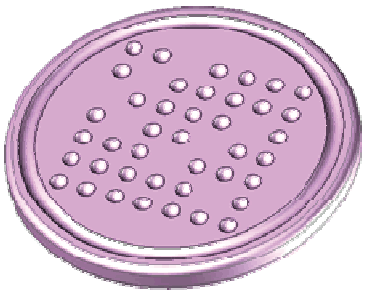
***Propagations can be applied to the following operations :***

- drilling 
- pockets 
- bosses 
- ribs 
- grooves 
- slots 
- subtractions 
- unions 
- trimming 
- contours 
- slots 
- folds 
- sawing 
- turning 
- tenon 
- mortise 
- fitting 

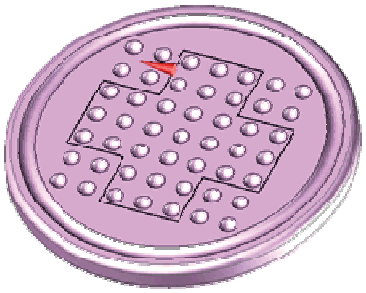
Propagations can be modified by removing certain occurrences :

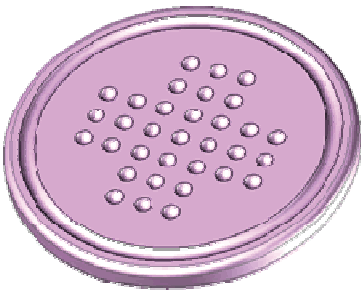


**EXCLUDE BY MANUAL SELECTION :**



**BY CLOSED CONTOUR**





# Coordinate Systems

The coordinate system bar works in a similar manner to the point bar.
A coordinate system can be compared with a work surface on which different elements of the 3D project are placed.
By default, each file has an absolute coordinate reference. The grid around this point indicates that this reference is the current reference.
All drawings are anchored in the XY plane of the current point of reference.
Several references can be created to simplify the design process.

To change the point of reference, click on  , then click on the new reference point.

The Tools | Coordinates menu includes a list of special coordinates that can be used to simplify your constructions

Certain coordinate systems are used frequently.

You must be able to create the following coordinate systems in particular :

***Coordinate system on point***

***Coordinate system 3 points***

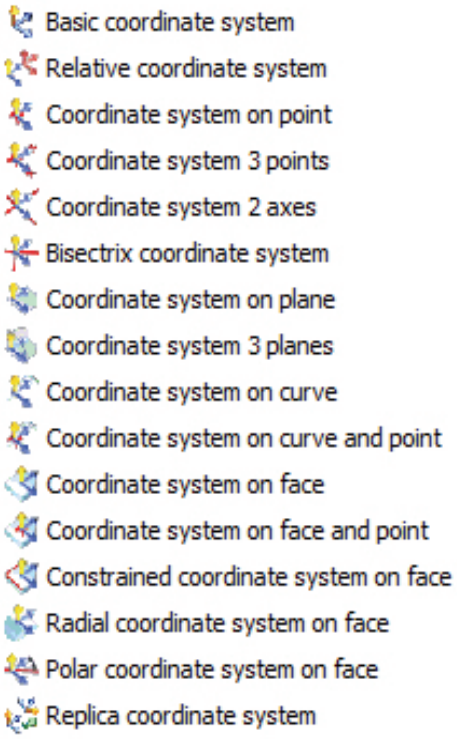
***Coordinate system 2 axes***

***Coordinate system on face***

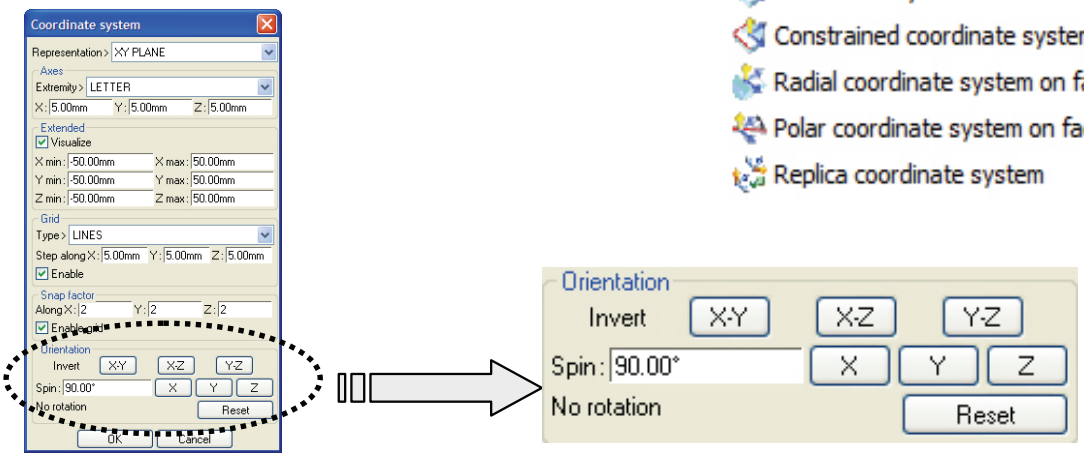
***Coordinate system on face and point***

***Constrained coordinate system on face***

***Replica coordinate system***

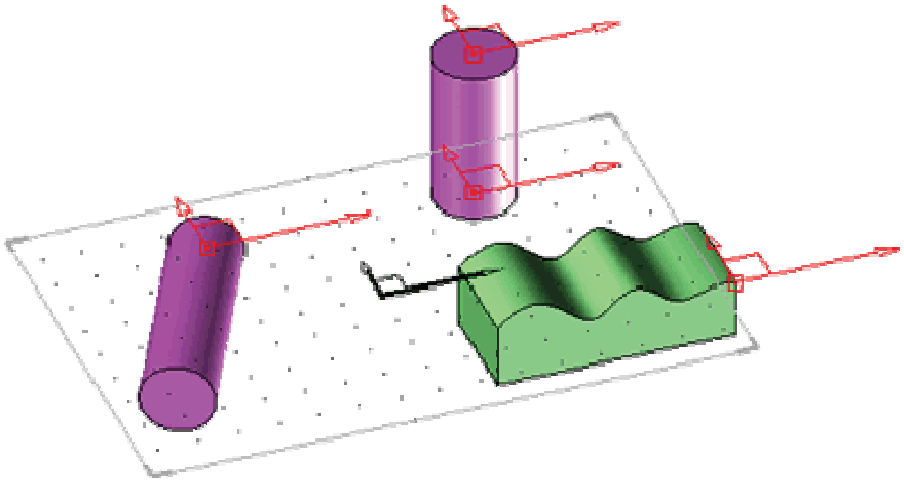



The orientation of the coordinate system can be modified using the  function. Each of the axes can be modified by a settable angular increment in the Orientation zone.

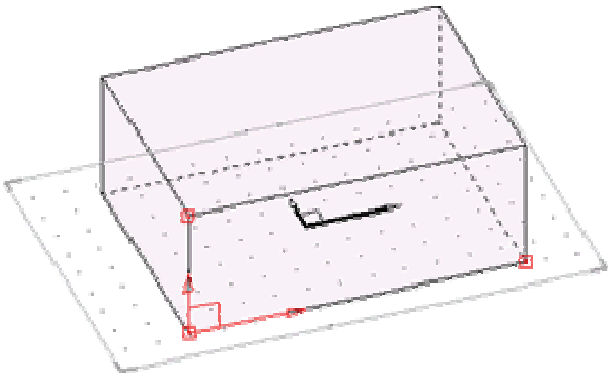


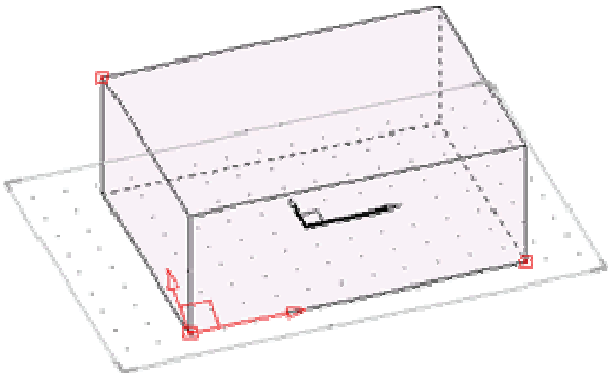
## Examples of the use of coordinate systems

***Coordinate system on point:*** Coordinate systems on points retain the orientation of the current coordinate system on which it depends when positioned on the designated point.

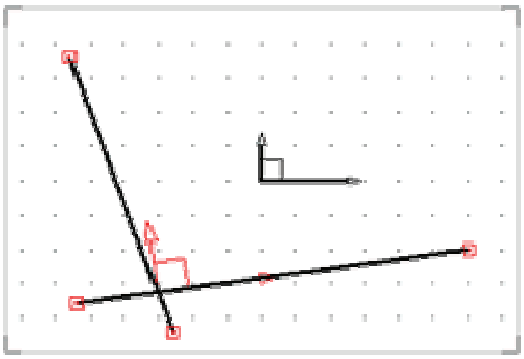


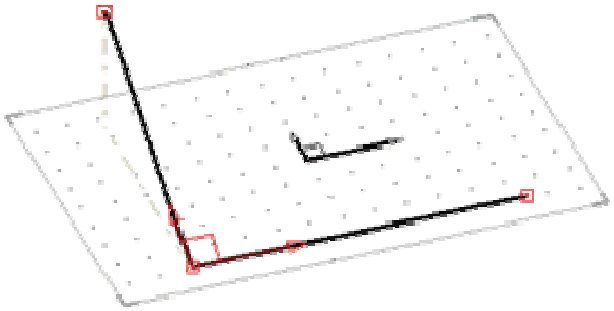
***Coordinate system 3 points:*** Coordinate systems 3 points are oriented using waypoints. The first point defines the origin 0 of the coordinate system, the second defines the position of the point for the X axis and the third defines the point for the Y axis.



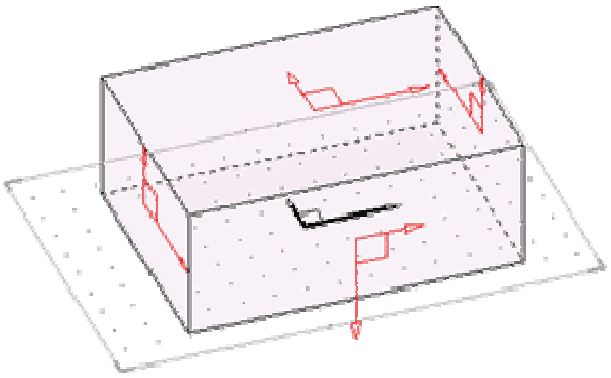


***Coordinate system 2 axes*** If there are more than two contours, the X+ and Y+ directions of the coordinate system are defined. The arrows indicate the positive direction of the axes.

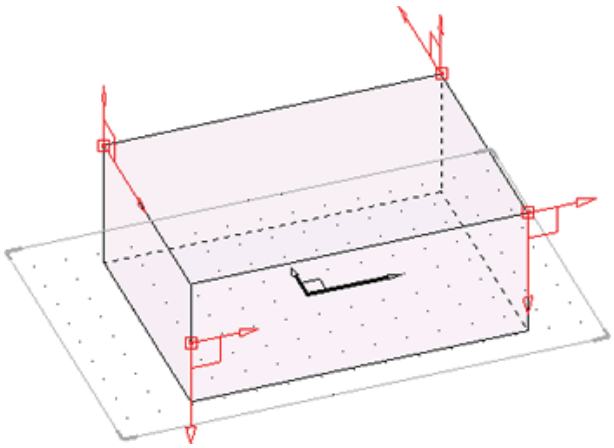




***Coordinate system on face:*** Creates a coordinate system positioned in the center of the selected surface.
Warning: the coordinate system is positioned in the center of the face at the time it is created, but it is not constrained. Changes to the geometry may cause the coordinate system to move.



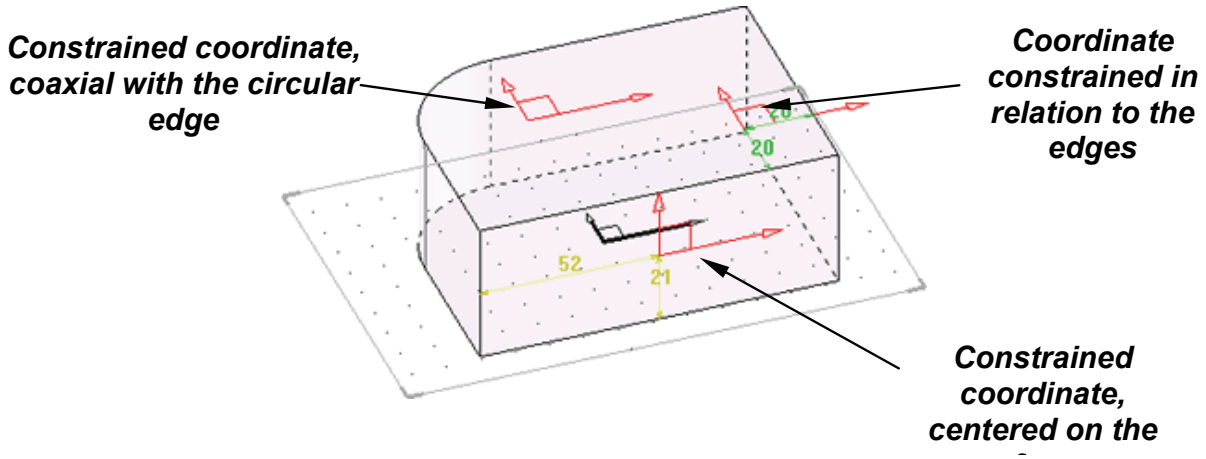
***Coordinate system on face and point:*** Creates a coordinate system whose orientation can be controlled using a reference surface and whose position can be controlled using a point.



***Constrained coordinate system on face:*** Is DYNAMICALLY placed on the selected surface. Behaves like a drill hole by positioning itself in relation to the nearest edges (green dimensions).
Depending on the position, it is automatically centered on the surface (yellow dimensions).

A coordinate system with two yellow dimensions is centered in relation to the length and width of the surface.

A single yellow dimension indicates that the system is centered in relation to one of the sides.




***Replica coordinate system:*** Creates a coordinate system by transformation in the same manner as the duplication and repetition functions.

***To change the orientation of a coordinate system manually, select the system, click on***  ***and change the orientation of the axes.***

Use the current coordinate system function to make a system the current system. 

# Memo : Duplicate – Repeat

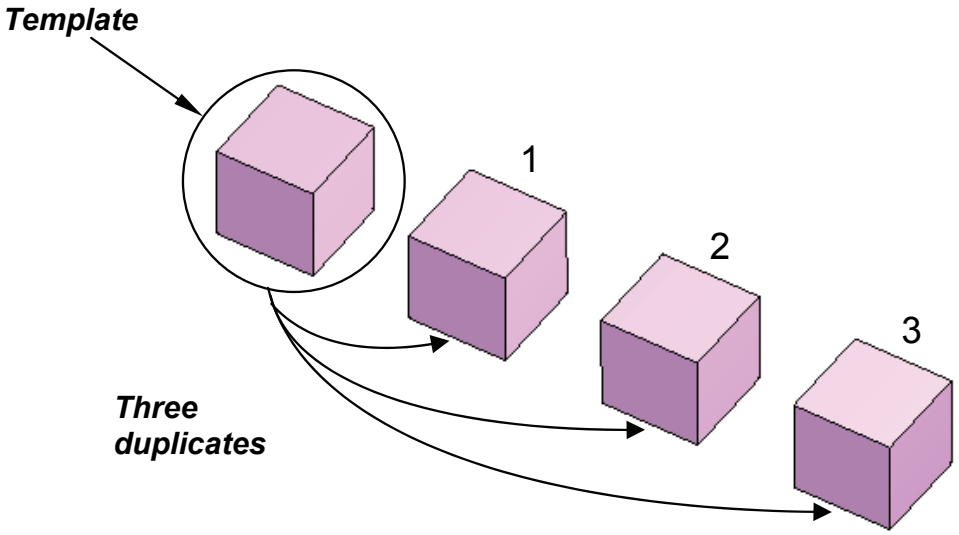
There are two ways to copy parts: **DUPLICATE** and **REPEAT**.

## Duplicate

This function is used to create one or more identical copies that are associated with the original.

***This function cannot be used to control the setting of the number of copies.***

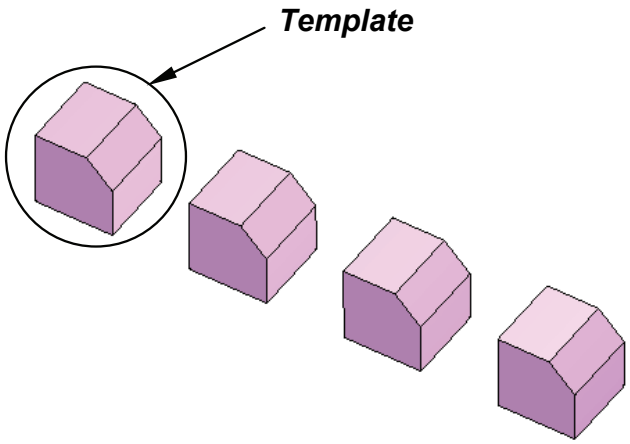
Each duplicate part is linked to the original template. The direction of the link is one-way: **from the template to the copy.**

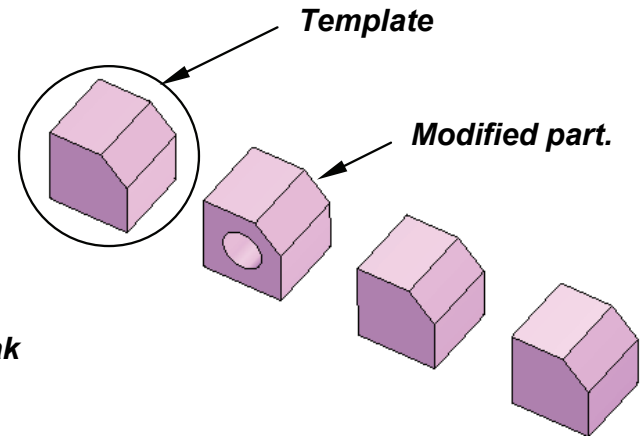



## Subsequent operations

When this option is used, any operations performed on the **Template** are automatically applied to the duplicates.

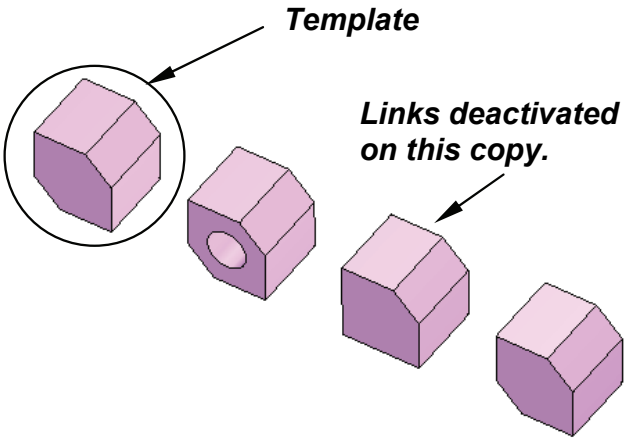
If a chamfer is created on the template, it is automatically created on the different copies by the link.






On the other hand, operations performed on the copies are not applied to the template.

***Making changes to a copy does not break the link with the template.***



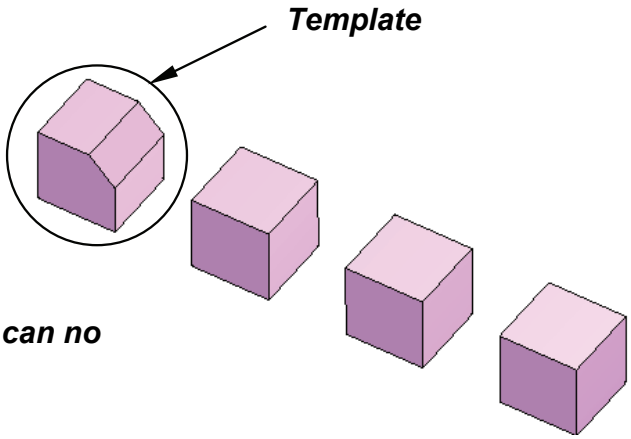

However, it is possible to break the link.

Select the  icon, then click on a copy to deactivate the link.
Use the **FOLLOW EXISTING OPERATIONS** option.

Changes to the template will be applied to the copies using the **active links.**

## Existing operations

This option is used to perform operations on the template without changing the copies.




If a chamfer is created on the template, the copies remain unchanged because the link is inactive

***Once the link has been broken and the template has been modified, the mode can no longer be changed.***

## Use in the BOM

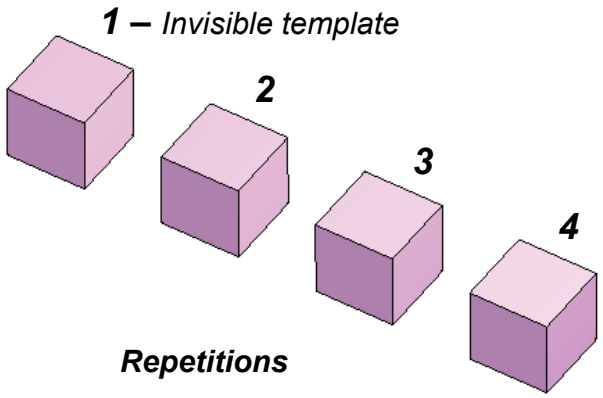
All duplicated parts are linked geometrically but can be defined independently. Each part has its own data (designation, material, dimensions, etc.).

| NB. | DESIGNATION | MATERIAL | PART TYPE | LENGTH | WIDTH | THICKNESS |
|---|---|---|---|---|---|---|
| 4 | Shaft | steel | HARDWARE | 160.0 | 10.0 | 10.0 |
| 2 | Rail | quercus | HARD WOOD | 1010.0 | 120.0 | 30.0 |
| 4 | Rail | quercus | HARD WOOD | 1020.0 | 65.0 | 35.0 |
| 4 | Leg | quercus | HARD WOOD | 225.0 | 60.0 | 60.0 |
| 1 | Glass | glass | GLASS | 910.0 | 910.0 | 8.0 |
| 1 | Base | quercus | HARD WOOD | 1020.0 | 1020.0 | 80.0 |

## Repeat

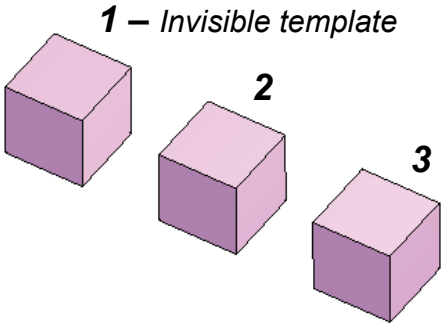
The same principle applies. The part to be repeated is selected, a transformation is chosen and then the distance and quantity are entered. The main difference is the link between the template and the instances. Any changes made to the template will be applied to the instances.

***When creating repetitions, instance number 1 is always superimposed on the template.***
***By default, the template becomes invisible. It can be made visible again.***
***Poor use of the feature can result in problems.***

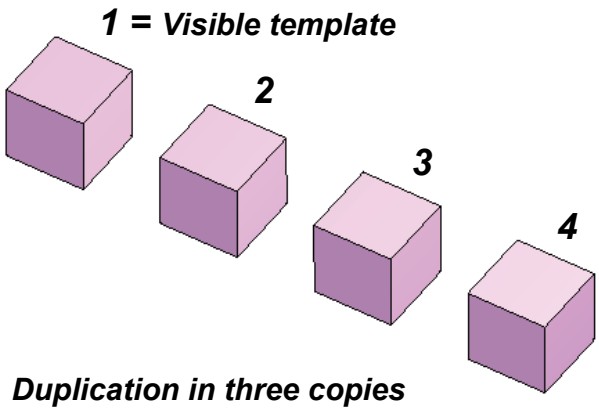



***Repetitions***

Unlike duplicates, the number of parts to be repeated can be configured.
The result varies depending on the quantity and type of instance used.



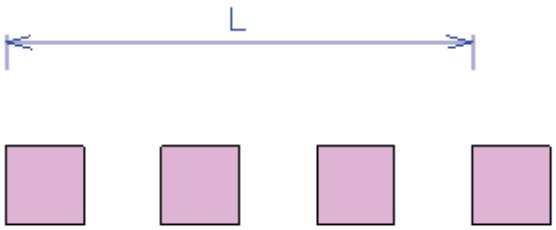

***Repetition in three instances***




***Duplication in three copies***

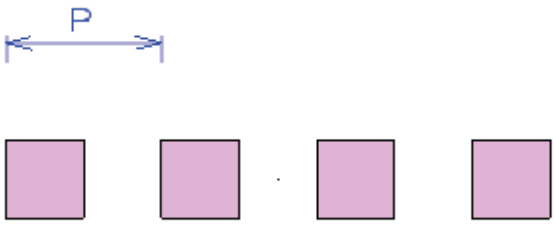
Two distribution modes are available for repetitions :

## Total distance :




The value is the distance "l" between the template and the furthermost instance. The number of instances is spread across a distance.

## Distance per instance




The value is the distance between two instances. A distance and a number are imposed.

There are two ways to edit the template: the construction tree or the wrench.

## Using the construction tree

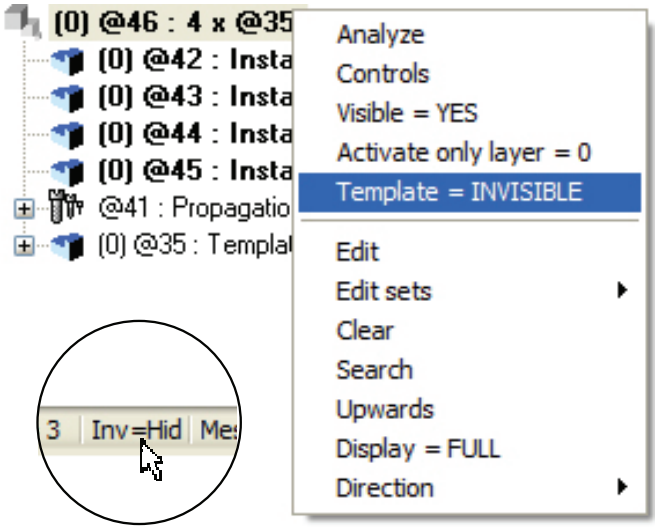
Open the construction tree and **Edit** the repetition.

This function is easy to use:

- if **the INSTANCES are in bold type face**, then the template is invisible,
- if **the TEMPLATE is in bold type face**, then the instances are invisible.


To make the template visible or invisible, right-click on the line that identifies the repetition.

In the drop-down list, select **Template = INVISIBLE** to hide the template and the reverse to view it.

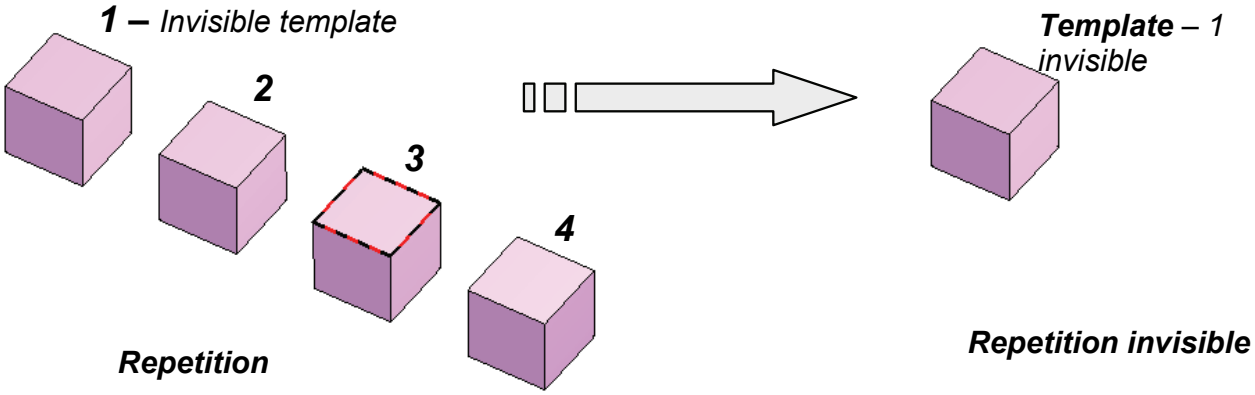



***You must be in visibility mode Invisible=Hidden.***
***In this way, you will not mix up the template and the repeated instances.***

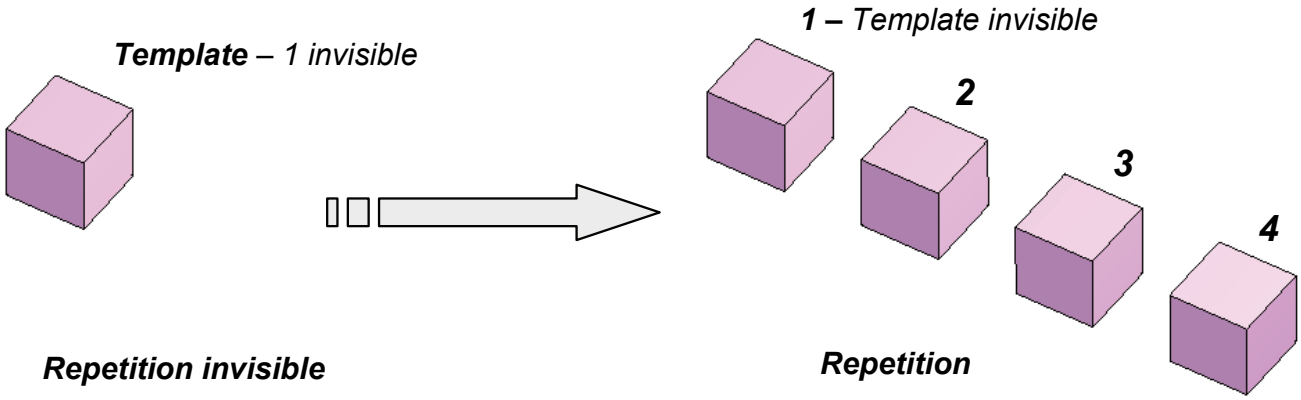
## Using the wrench

Start the Modify element function  .

Select one of the repeated instances, then select the **EDIT TEMPLATE** tab to view the template.

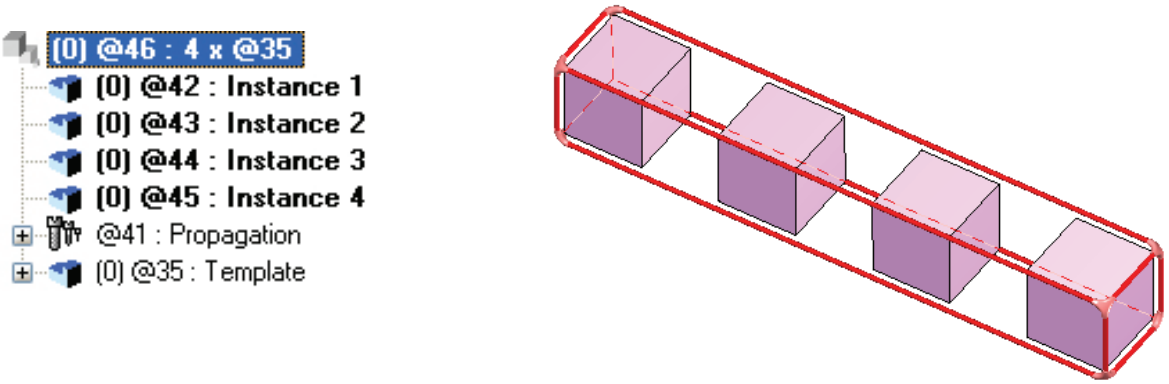


Use the ***Repeat*** function to return to the repetition.
Select the **SHOW REPETITION**, then show the template.
A selection filter can be used to designate just the templates.

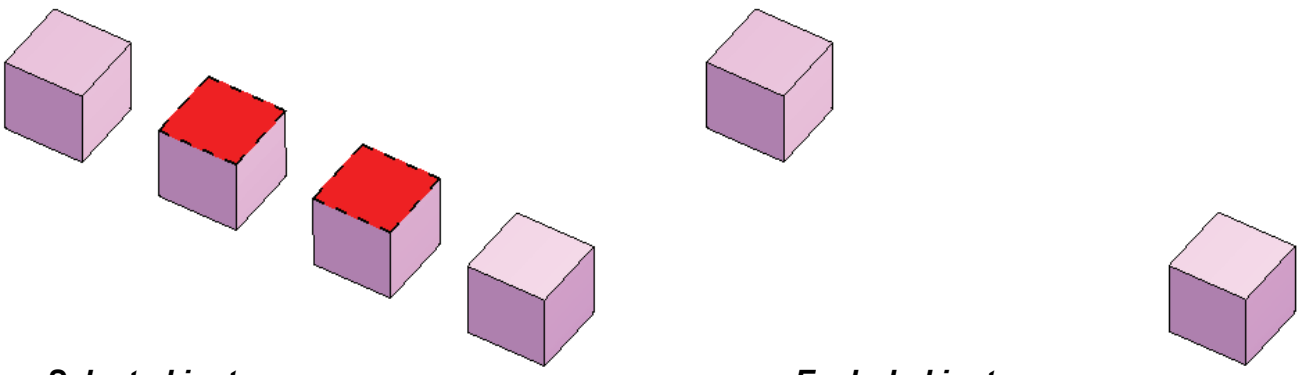



## Excluding instances of a repetition

The repetition function creates a set of repeated instances. Once or more of these instances can be excluded.




To exclude the unwanted instances, use the wrench  and show the repetition.
Select **PROPAGATION**, then **EXCLUDE INSTANCE.**
Show the instances to be excluded.



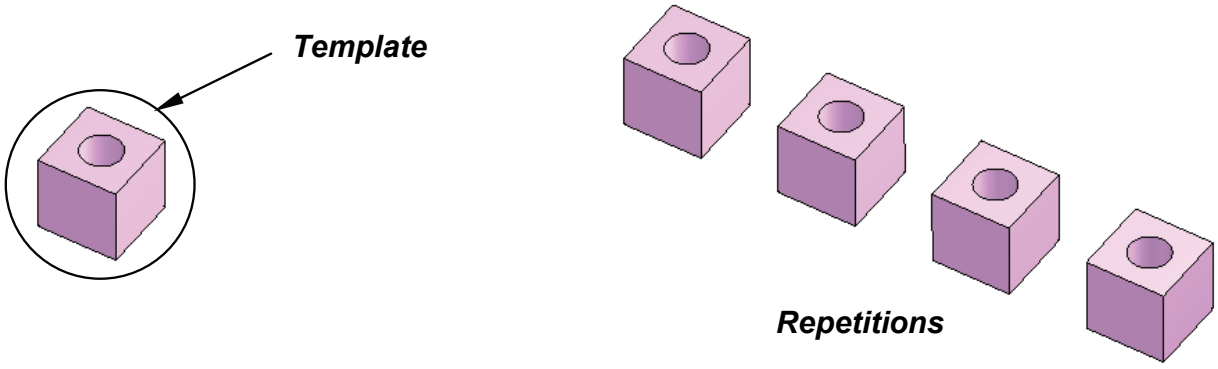
Click on **OK.**

***The excluded parts are not deleted.***

To return the parts, follow the same procedure as above and select **RETURN INSTANCES.**

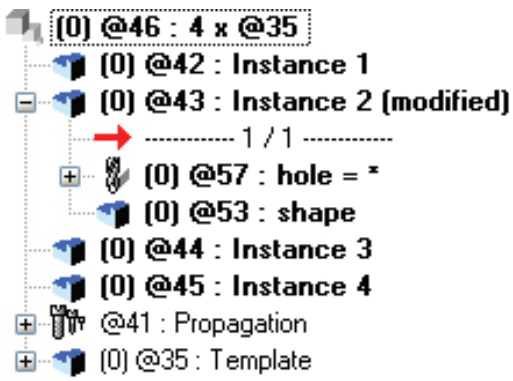
## Modifying the template

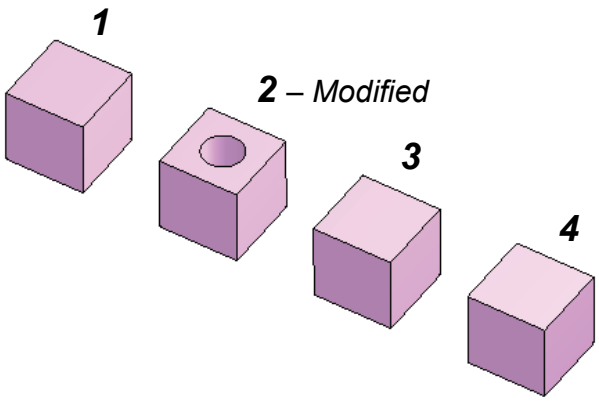
If the template is modified by an operation such as drilling, this operation appears on all the repeated instances.




## Operations on repetitions

On the other hand, if the operation is performed on one of the repeated instances, it is not applied to the others. The log file indicates which instance has been changed.



## Use in the BOM

All of the parts in a repetition are considered to be identical.
Any changes made to the data defined for the template (designation, material, dimensions, etc.) are automatically applied to the instances.

| 4 | Shaft | steel | HARDWARE | 160.0 | 10.0 | 10.0 |
|---|---|---|---|---|---|---|
| NB. | DESIGNATION | MATERIAL | PART TYPE | LENGTH | WIDTH | THICKNESS |

# Memo : Different assembly methods

# Introduction

TopSolid'Design is a software used to design 3D projects 3D.
Unlike in other software applications, it is possible to choose between different design methods when building the project.

The design method can be managed according to the project in order to simplify and optimize the work in hand.

The two main design methods are:

- Design in place.
- Design by reassembly.

These two methods offer a close fit and offer users increased flexibility and power.

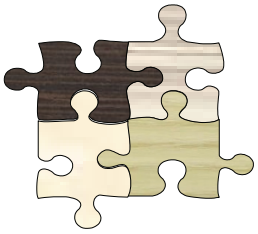
**Fundamentals.**

***Design in place.***

The principle of design in place consists in creating the project in a single document. All parts are designed in the same file.

Dimensions and constraints are used to position the parts in relation to one another. The result is an assembly of several parts.

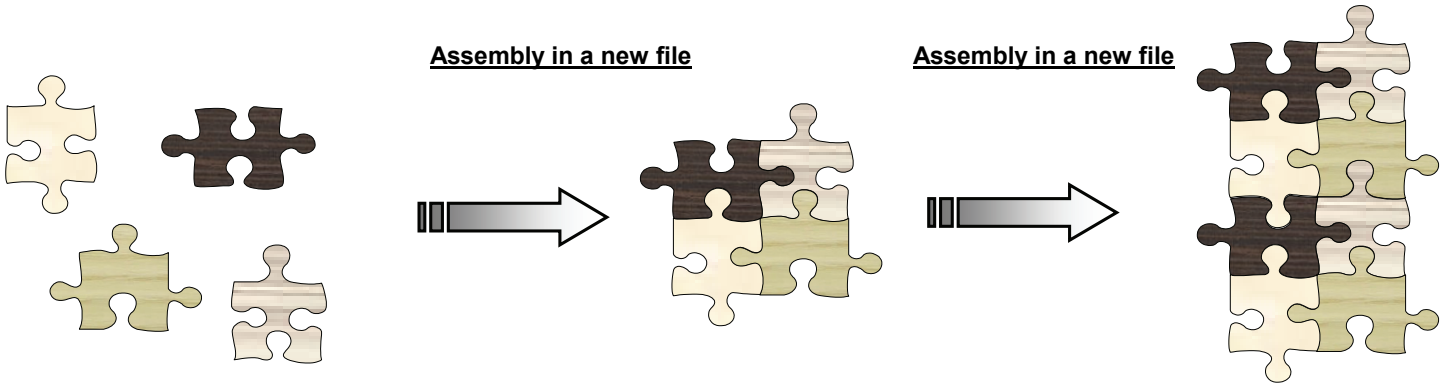
***Design by reassembly.***



Design by reassembly consists in creating a project by assembling several files. Users can use one of two methods to assemble the parts in the file:

- With positioning constraints.
- With key points.

The resulting file contains an assembly that refers to several other files.



# Designing a project

The choice between design in place and reassembly depends on the complexity and the usage of the elements to be drawn.

For the following project, we need to think about the different subassemblies of the unit.
In this example, the unit will be broken down into two main subassemblies :

- a box made up of:
    - four plated panels with 4/10 edges.
    - dowels.
    - cams.
    - casters.

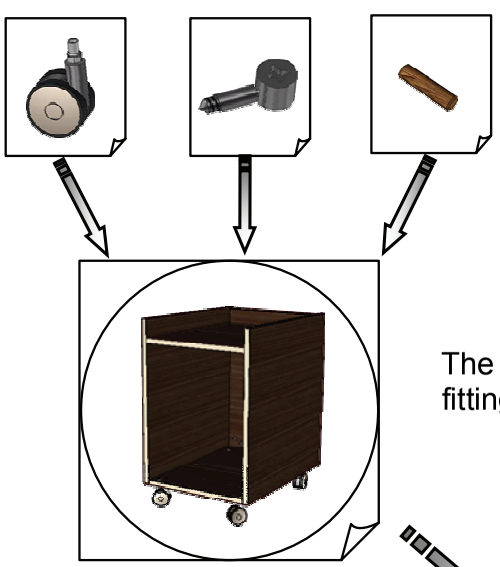
- Three drawers made up of:
    - a box with a grooved base.
    - one pair of runners with retaining screws.
    - one wooden front panel.
    - one handle.

This analysis gives us an idea of the design method to be used for each of the subassemblies.
The drawers will be drawn in a separate file and will then be inserted by reassembly, which means that they can be used in a future project.

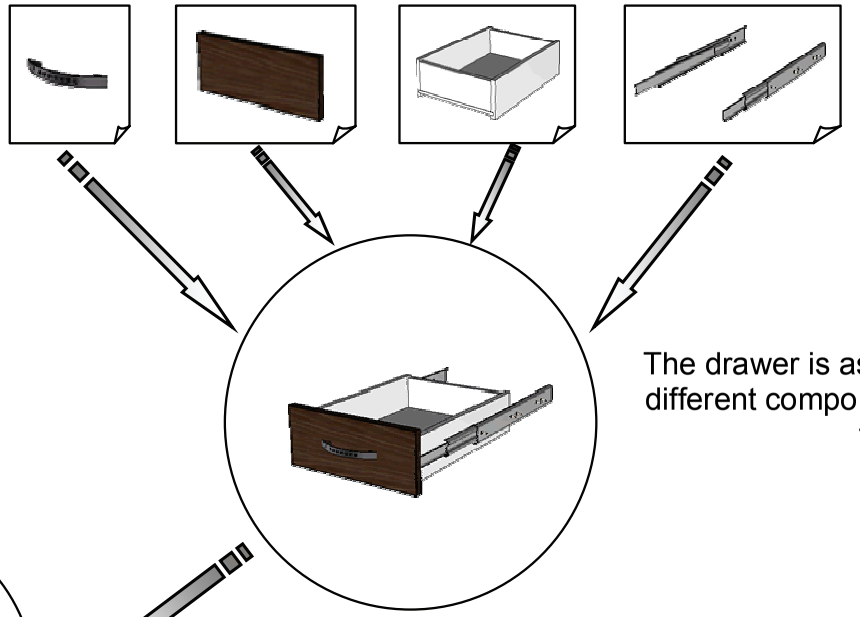
**Choosing the design method for the drawers**

The drawers can be drawn in a single file or built from the different parts (runners, front panel, handle, screws, box).
If the component parts of the drawers can be used in other projects, then it is advisable to draw them in separate files and then assemble them. This is the case of the handle and the sliders.
The drawer box will be drawn in a single file.

The design in place and reassembly methods can be used in the same project. There is no need to exclude one method in favor of the other. You simply need to understand the advantages of each method when making your choice.
Design in place is the simpler design method, since all of the parameters are contained in a single file, a fact that simplifies the relationships between the parts. However, colors, visibility and layers must be carefully managed in order to keep the number of elements in the file under control.
Design in place can be used to break down the project into subassemblies, which can be used in other projects, or to simplify the application of a project.
The following diagram proposes a design process that uses the design in place method and component reassembly



The box is designed in place. The fittings are inserted in the box file.



The drawer is assembled using the different components in the drawer file.

Design in place.

Design by reassembly

Unit created from the box and drawer insert three times.

3D files can be used to create large quantities of documents: drafts, programs, animations, etc.

Bill of materials (Excel or draft).

| | A | B | C | D | E | F | G |
|---|---|---|---|---|---|---|---|
| 1 | Nombre | Désignation | Matière | Type_de_pièce | Long | Largeur | Ep |
| 2 | 1 | Arrière | mélaminé | Panneau | 554 | 382 | 19 |
| 3 | 6 | Avant / Arrière | mélaminé | Panneau | 312 | 87 | 16 |
| 4 | 6 | Coté | mélaminé | Panneau | 450 | 110 | 16 |
| 5 | 1 | Coté droit | mélaminé | Panneau | 576 | 546 | 19 |
| 6 | 1 | Coté gauche | mélaminé | Panneau | 576 | 546 | 19 |
| 7 | 3 | Coulisse à bille | acier | | | | |
| 8 | 1 | Dessous | mélaminé | Panneau | 546 | 382 | 19 |
| 9 | 1 | Dessus | mélaminé | Panneau | 524 | 382 | 19 |
| 10 | 3 | Façade de tiroir | | Panneau | | | |
| 11 | 3 | Fond | mélaminé | Panneau | 328 | 450 | 8 |
| 12 | 3 | Poignée | acier | | | | |
| 13 | 4 | Roulette | | | | | |
| 14 | 30 | Vis Euro | acier | Quincaillerie | | | |
| 15 | 20 | boîtier minifix ep 19 | acier | | | | |

Draft documents.

Exploded view of the 3D project

# Workshop : Creation of a circular rail

On this workshop you will be able to learn to :

- Use 2D sketches,
- Use points and coordinate systems,
- Extrude with offset and draft angle,
- Create operations on shape (trim, subtract …),
- Propagate an operation

# Circular rail - Rail's design

**1** **Open a new design document**

Choose `Without Template` in the proposed list of **New Document** template models.

**2** **Create a rectangular contour**

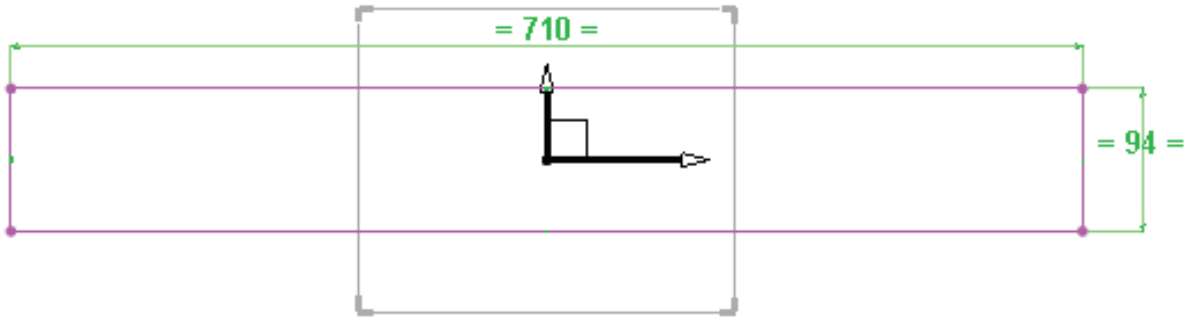
*Curve* \ *Contour* \ *Rectangular*

Draw a **RECTANGULAR Contour** using **AUTO DIMENSION.**

**3** **Add constraints**

Use the **Modify element** function to add a symmetry constraint on the **X** axis then on the **Y** axis.

Select the dimension to constrain, click on **CONSTRAIN** then click on the axis to use.




**4** Modify parameters

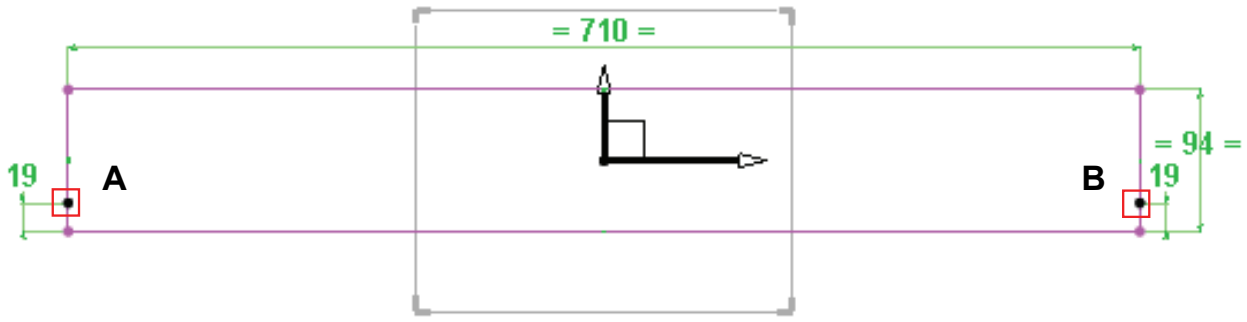
*Parameter* | *Modify parameter*

Modify the dimensions as above.

**5** Offset points

*Tools* | *Points* | *Offset point*

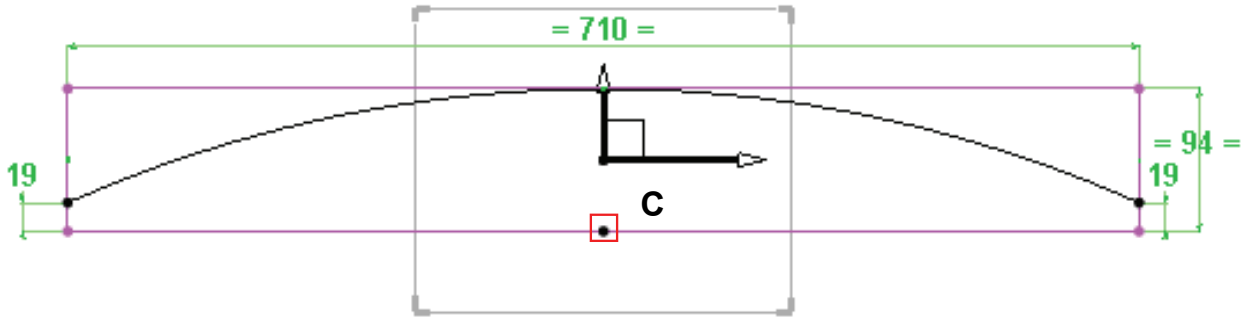
Create two points, **A** and **B**, with an offset of 19mm along **Y+** axis. (See below).



**6** **Center point**

*Tools |Points | Center key point*

Create a center point, C, on the bottom line of the rectangle.




**7** **Create an arc**

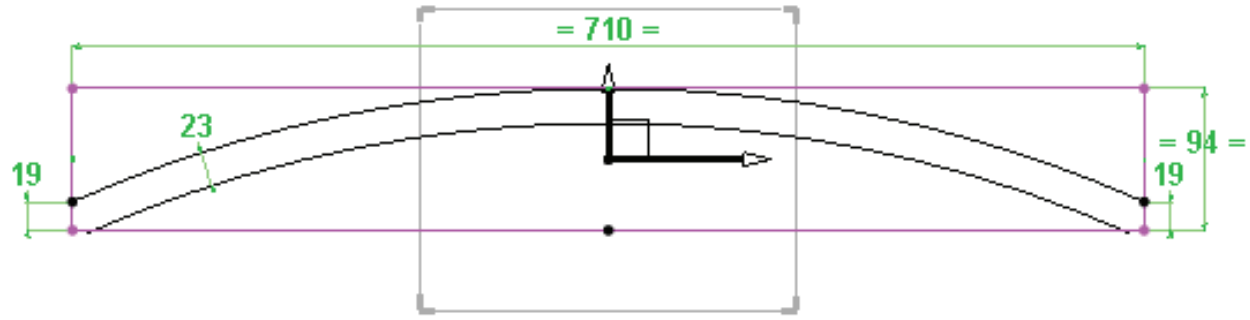
*Curve | Other curves | Circles | Circle 3 points*

Select the point A, then the top segment of the rectangle to be tangent, and to finish the point B.

**8** **Offset curve**

*Curve | Offset curve*

Create an offset curve of 23mm from the arc.



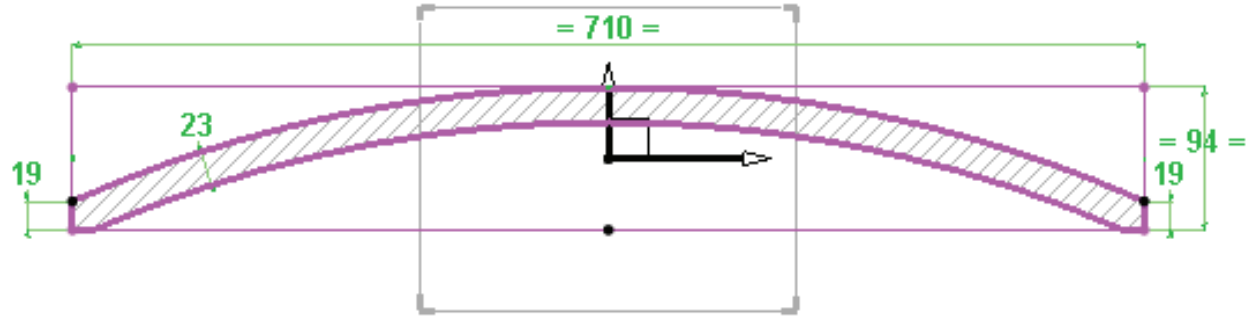

**9** **Activate layer 1**

Make layer 1 current

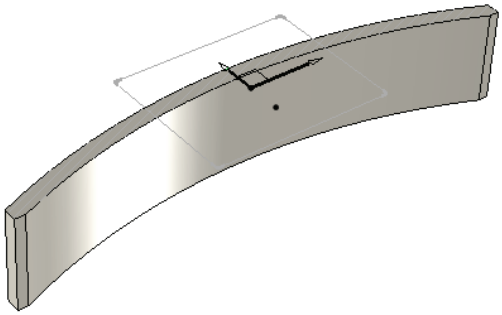
**10** **New contour**

*Menu \ Contour*

Create a closed contour like below (hatched area)



## Creation of the rails

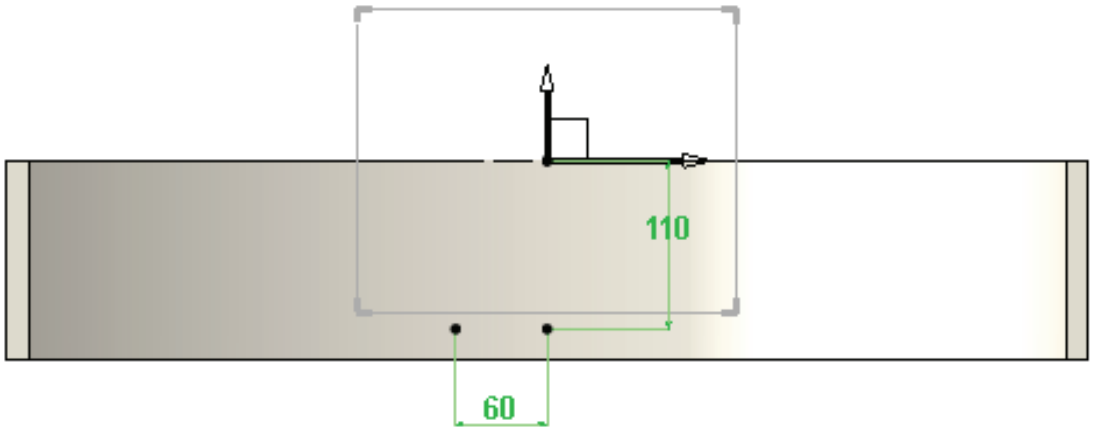


**11** **Extrude**
*Shape* | *Extruded*
Extrude the contour along **Z-** axis with a height of 130mm

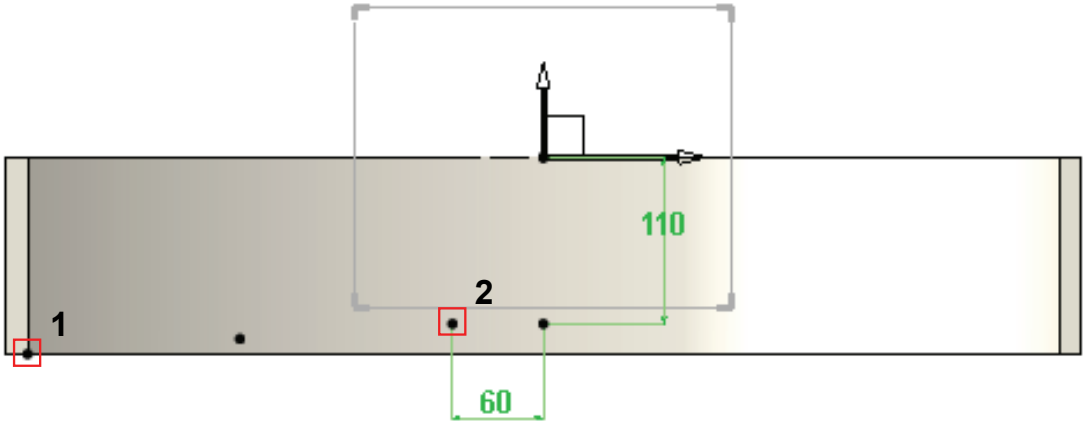
**12** **Activate layer 2**
Make layer 2 current

**13** **Coordinate system**
*Tools* | *Coordinate systems* | *Coordinate system 2 axes*
Click on **"Passing point"** and select the C point created previously. Select **X+** axis to define **X** axis, and **Z+** axis to define **Y** axis.
**Set current** this new coordinate system.

**14** **Offset points**
*Tools* | *Points* | *Offset point*
Create two offset points:
One along **Y-** axis with an offset of 110mm from the current coordinate system.
One along **X-** axis with an offset of 60mm from the offset point just created.



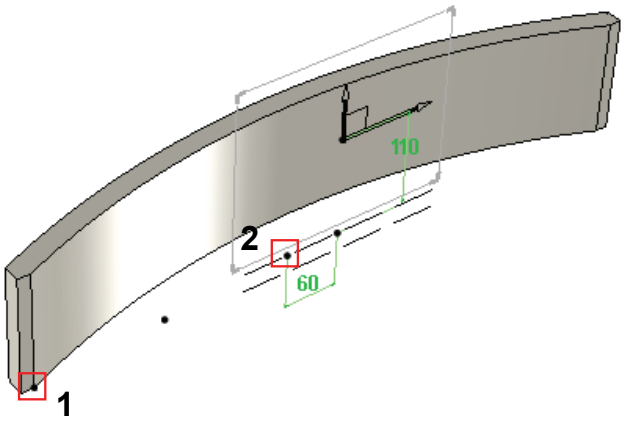

**15** **Middle point**
*Tools* | *Points* | *Middle point*
Create a middle point between the point **1** and the point **2**.



**16** **Sketch lines**

*Curve* | *Sketch line*

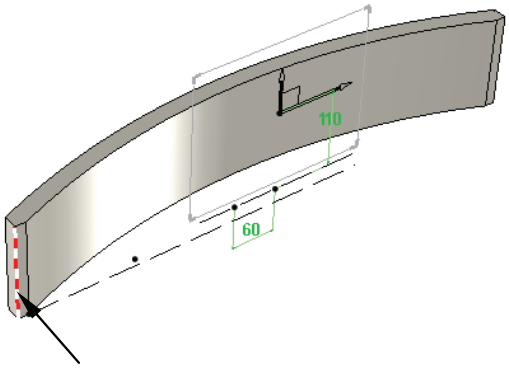
Create two sketch lines:
Passing by the point **1**
Passing by the point **2**.



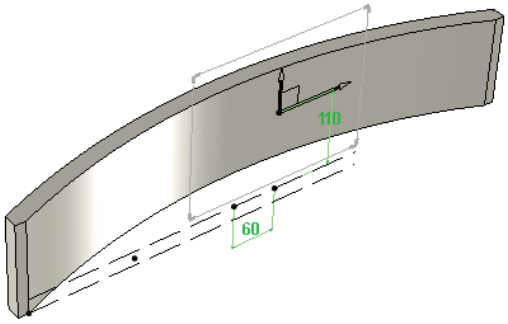

**17** **Trim a segment**

*Curve* | *Trim*

Trim both sketch lines by the left border edge of the rail.
Select a line, select the edge and then click on "**Make trim**".




**Edge to use**

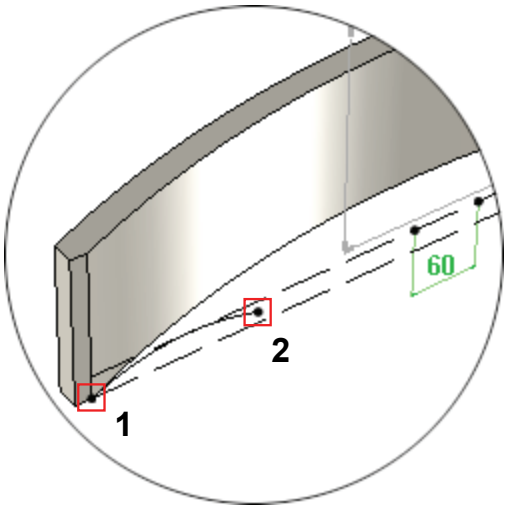



**Result**

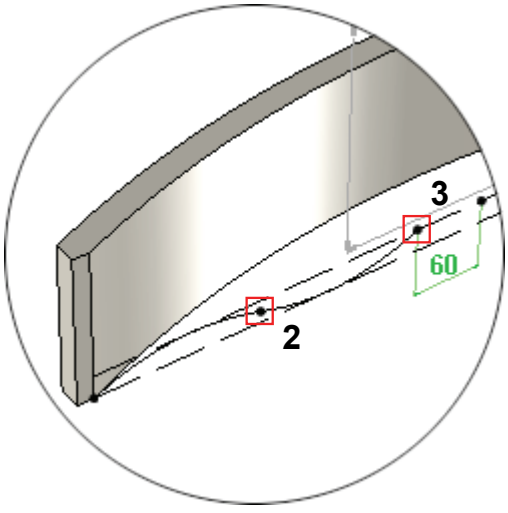
**18** **Create arcs**

*Curve* | *Other curves* | *Circles*| *Circle 3 points*

Select the point **1**, then the top sketch line to be tangent and finally the point **2**.



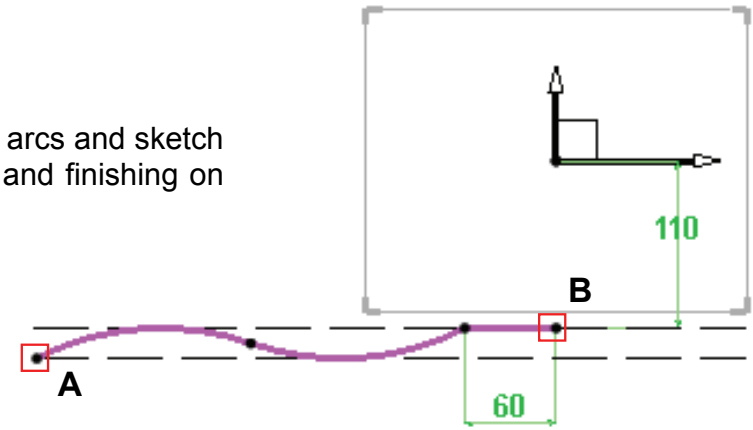

Select the point **2**, then the bottom sketch line to be tangent and to finish the point **3**.



**19** **Contour**

*Curve* | *Contour*

Create a **new contour** through arcs and sketch lines, starting from the point **A** and finishing on point **B**.

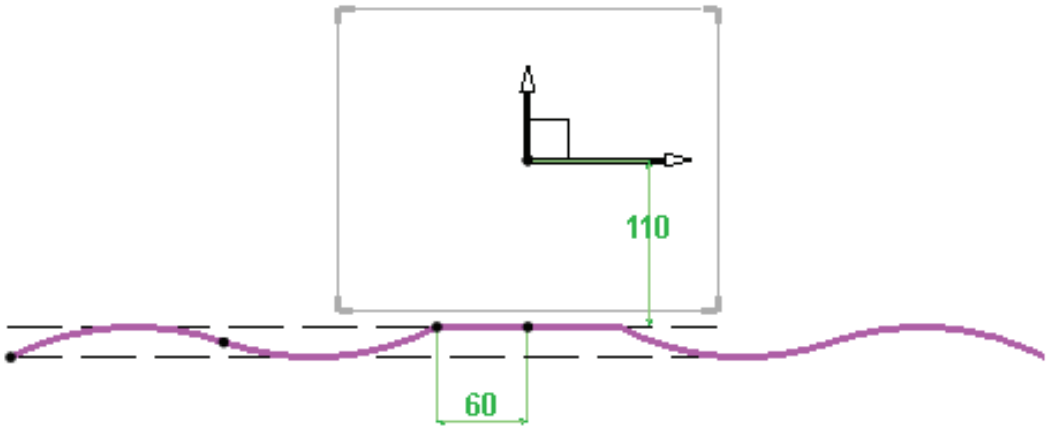



**20** **Pattern**

*Curve* | *Pattern*

A pattern allows to duplicate a curve and sew the result as a single curve.

Select the contour and choose a **simple mirror** using **YZ** plane.



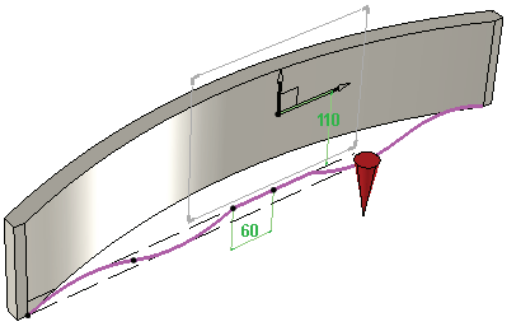

**21** **Trim**

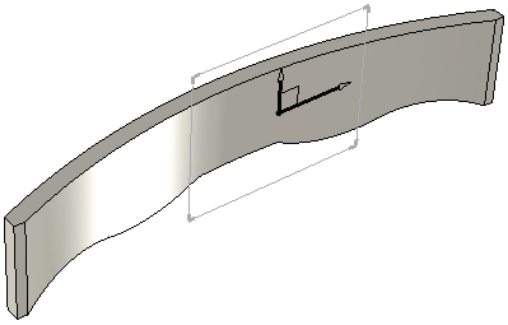
*Shape* | *Trim*

In the combo list **"Trim"**, select **"By sweeping curve"**. Select the rail and then the trimming contour.

The red arrow shows the side to remove.

Select **OK**.

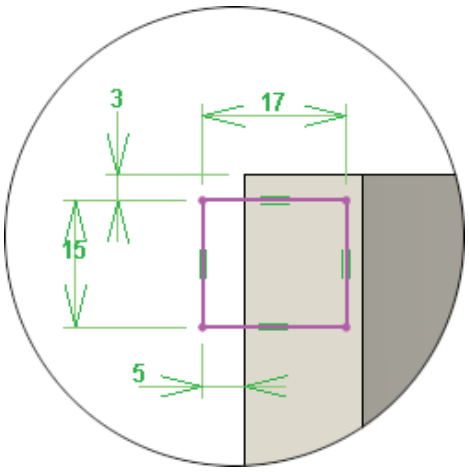


**22** **Activate layer 3**

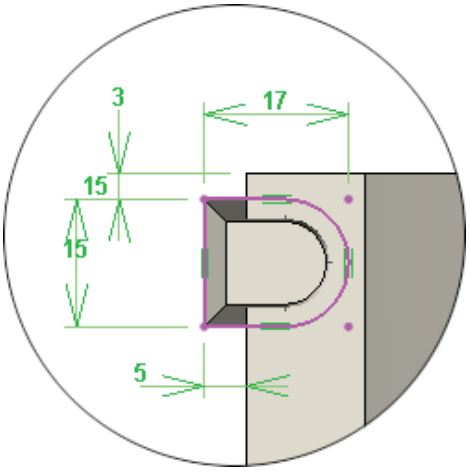
Make layer 3 current

## Blind dovetail




**23** **Rectangular contour**
*Curve* | *Contour*
Create a rectangular contour near the top left of the rail and use **"Auto dimension"**.

**24** **Dimensions**
*Tools* | *Dimension*
Adjust dimensions and position the contour like shown, 3mm from the top and 5mm from the left outside of the rail.




**25** **Fillet**
*Curve* | *Fillet*
Use the "**Local**" mode to create fillets of 7.5mm on the rectangle.

**26** **Extrude**
*Shape* | *Extruded*
Extrude the contour along **Z+** with an offset of -10mm, a draft angle of 10° and a height of 15mm.
Note: to manage the draft angle and the offset, use advanced options >>

**27** **Subtract**
*Shape* | *Subtract*
Select the rail and use the extruded shape as tool.

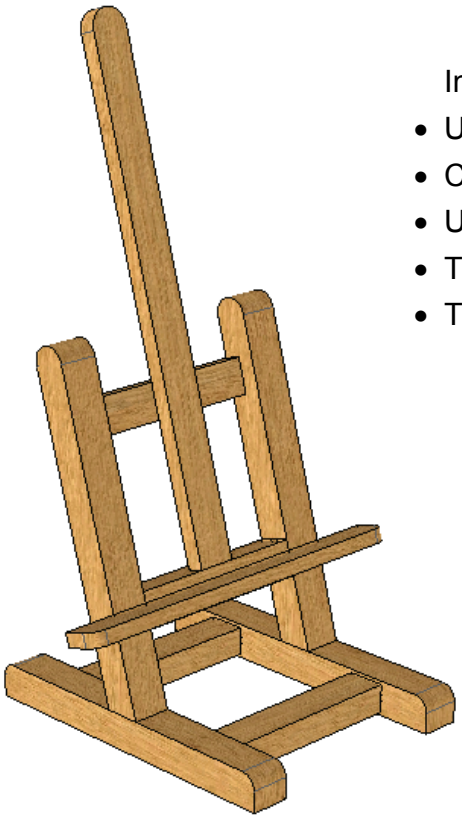
**28** **Propagate an operation**
*Shape* | *Subtract*
Select the subtraction operation just done.
Select in the list of propagation "**Multiplication of propagations**".

- Propagation n°1 :
  Use "**Linear**" along **Y-** axis. Click on "**Total distance**" to switch to "**Distance per instance**" of 26.5mm and a total number of 4.

- Propagation n°2 :
  Use "**Simple mirror**" and select **YZ** plane.

Select **STOP**.

# Workshop : Creation of an easel



In this workshop you learn to :

- Use various geometry tools,
- Create a thickened profile,
- Use the various points and coordinate systems,
- To create a tenon / mortice assembly,
- To repeat parts

## Create the geometry of the easel

**1** **Open a new design document**

Choose `Without Template` in the proposed list of **New Document** template models.

**2** **Create a rectangular contour**

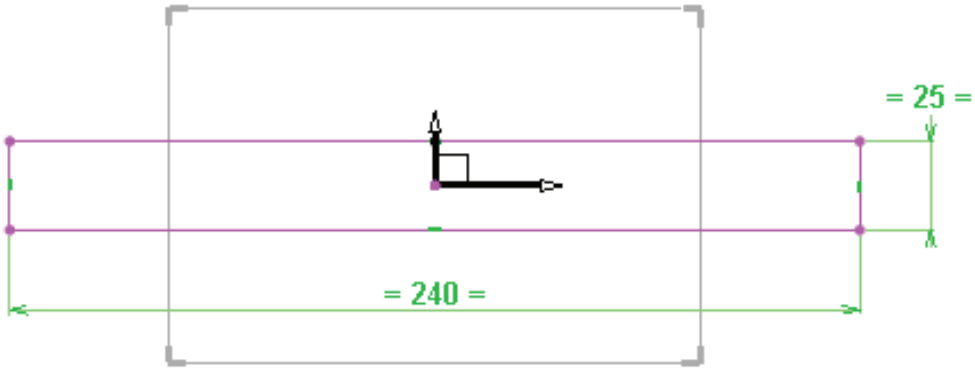
*Curve* \ *Contour* \ *Rectangular*

Draw a **RECTANGULAR Contour** and then click on **AUTO DIMENSION.**

**3** **Add constraints**

Use the **Modify element** function to add a symmetry constraint on the **X** axis, and then on the **Y** axis.

Select the dimension to constrain, click on **CONSTRAINT** then click on the axis to be used.




**4** **Modify parameter**

*Parameter* \ *Modify*

Replace the dimension values with the parameters created (Shown above).

**5** **Create a centre point**

*Tools* \ *Point* \ *Centre key point*

Create a centre point on the upper segment of the rectangle.

**6** **Create a line**

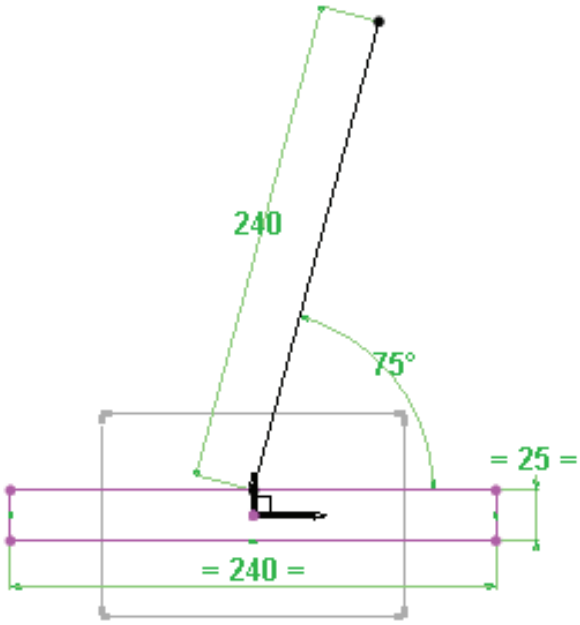
*Curve*\ *Line*

Draw a line using the previously created centre point as the starting point of the line.

**7** **Dimension a contour**

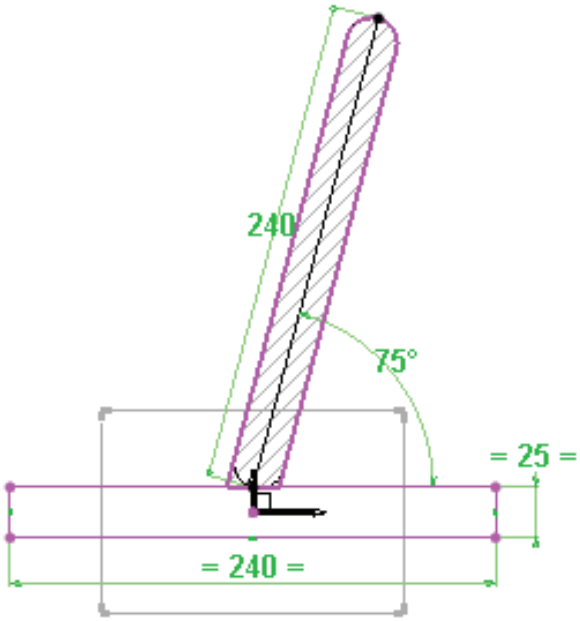
*Tools* \ *Dimension*

Dimension the line as shown opposite. Its length will be `240 mm` and its slope `75°`.



**8** **Thicken a line**

*Curve* \ *Thickened curve*
Thicken the previously created line using the **Symmetric = YES** mode and a value of `12.5 mm`.
From the list of End Types, select the **CIRCLES INSIDE** mode.
Select **OK**.



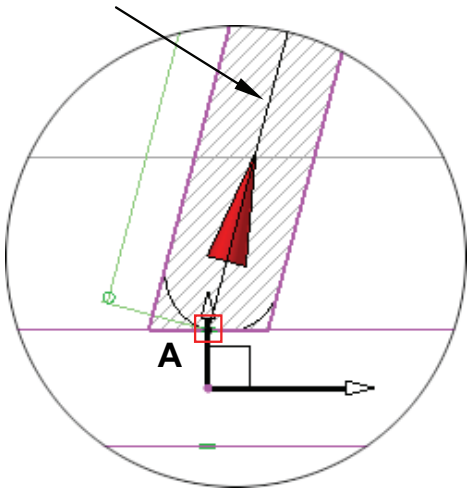

**9** **Create a contour**

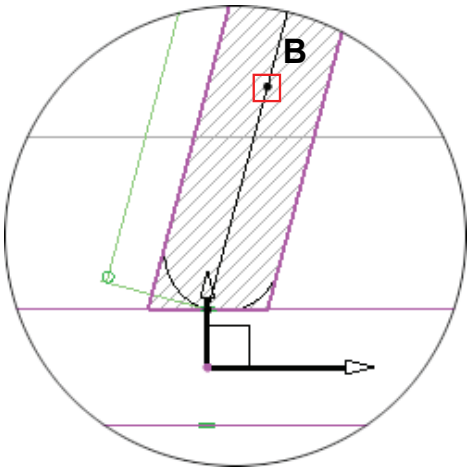
*Curve* \ *Contour*
Draw a **closed contour** as shown opposite. (Contour surrounding the hatched zone)

**10** **Activate layer 1**
Make layer 1 current.

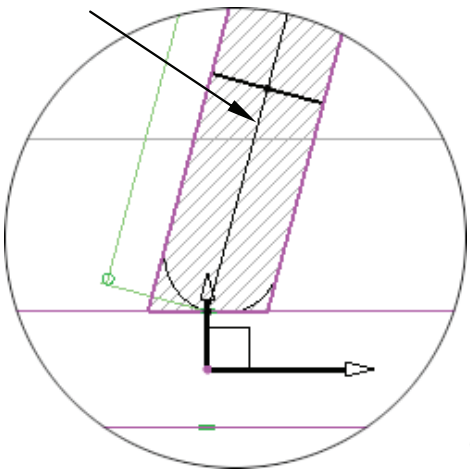
**11** **Create a point on a curve**
*Tools* \ *Point* \ *Point on curve*
Using the **CONSTRAINED POINT** mode, create a point on curve **B** at a distance of `50 mm` starting from point **A**.

***Reference curve***

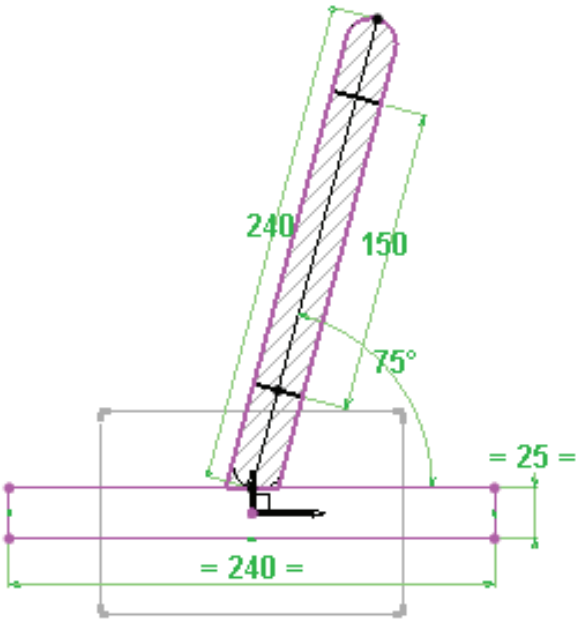



***Reference curve***



**12** **Create a line on curve**
*Curve* \ *Other curves* \ *Line on curve*
On the line shown opposite, create a line on the curve with a **CENTERED** alignment, a length of `25 mm`, in the **PERPENDICULAR** mode and using the previously created point on a curve (**B**).




**13** **Create an offset curve**

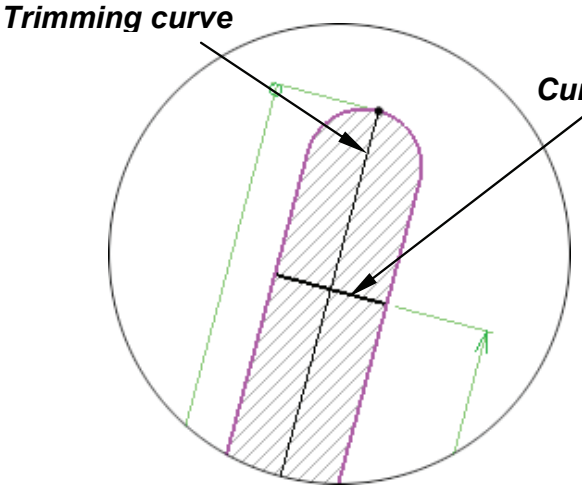
*Curve* \ *Offset curve*

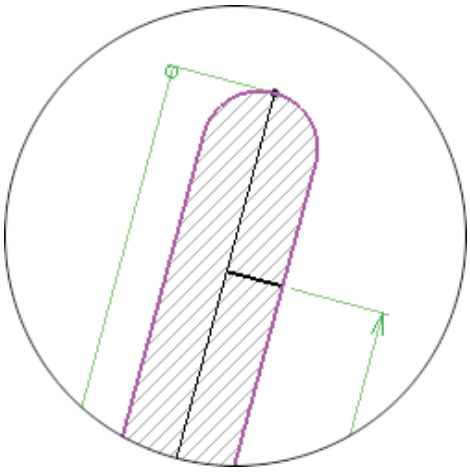
Create an offset curve, offset by `150 mm` from the previously created line on a curve.

**14** **Trim a curve**

*Curve* \ *Trim*

Trim the curve as shown below.



***Result***

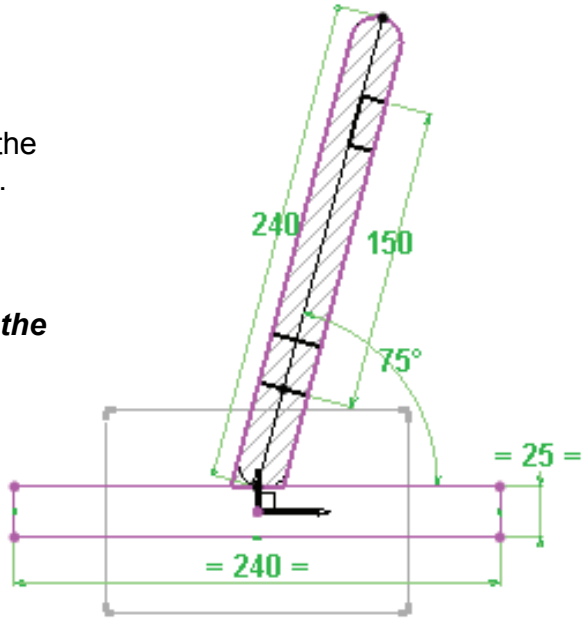
**15** **Thicken a line**

*Curve* \ *Thickened curve*

Thicken the two lines previously created using the **Symmetric = YES** mode and a value of `25 mm`. From the list of End Types, select the **LINES** mode.

***To modify the thickening direction, click on the red arrow on the screen.***

Select **OK**.



**16** **Create a rectangular contour**

*Curve* \ *Contour* \ *Rectangular*

Draw a **RECTANGULAR Contour** using **AUTO DIMENSION** with length `27 mm` and height of `20 mm`.

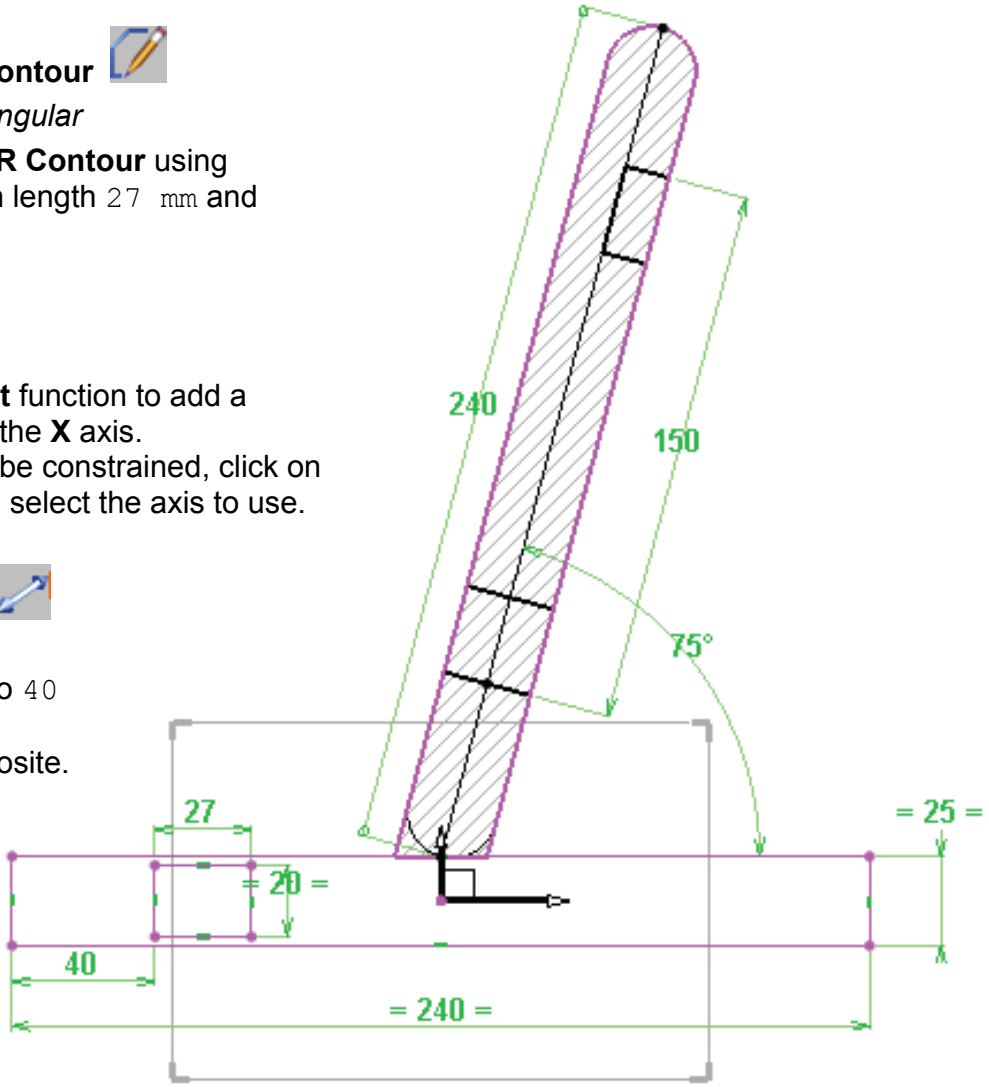
**17** **Add constraints**

Use the **Modify element** function to add a symmetry constraint on the **X** axis.
Select the dimension to be constrained, click on **CONSTRAINT** and then select the axis to use.

**18** **Dimension a contour**

*Tools* \ *Dimension*

Dimension the contour to `40 mm` from the end of the mounting as shown opposite.




**19** **Activate layer 2**

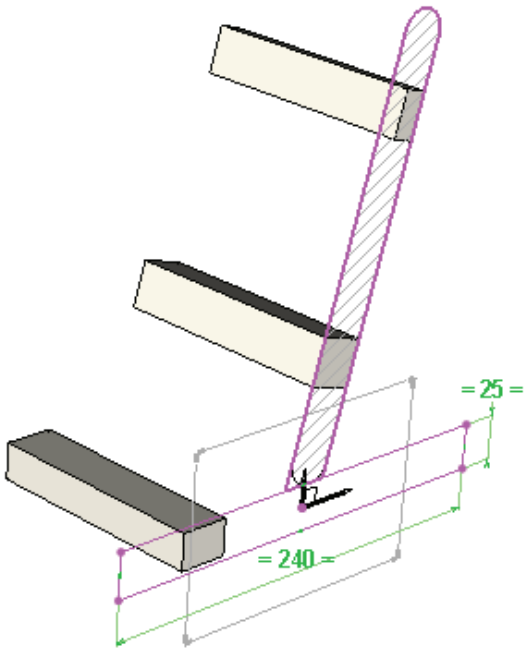
Make layer 2 current

## Model the easel

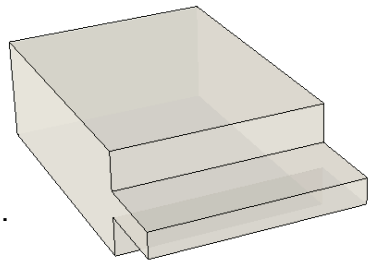
**20** **Extrude**

*Shape* \ *Extruded*

Extrude the 3 contours corresponding to the cross-pieces, with a value of `128 mm` in the **Z-** direction.



## Tenon the cross-pieces



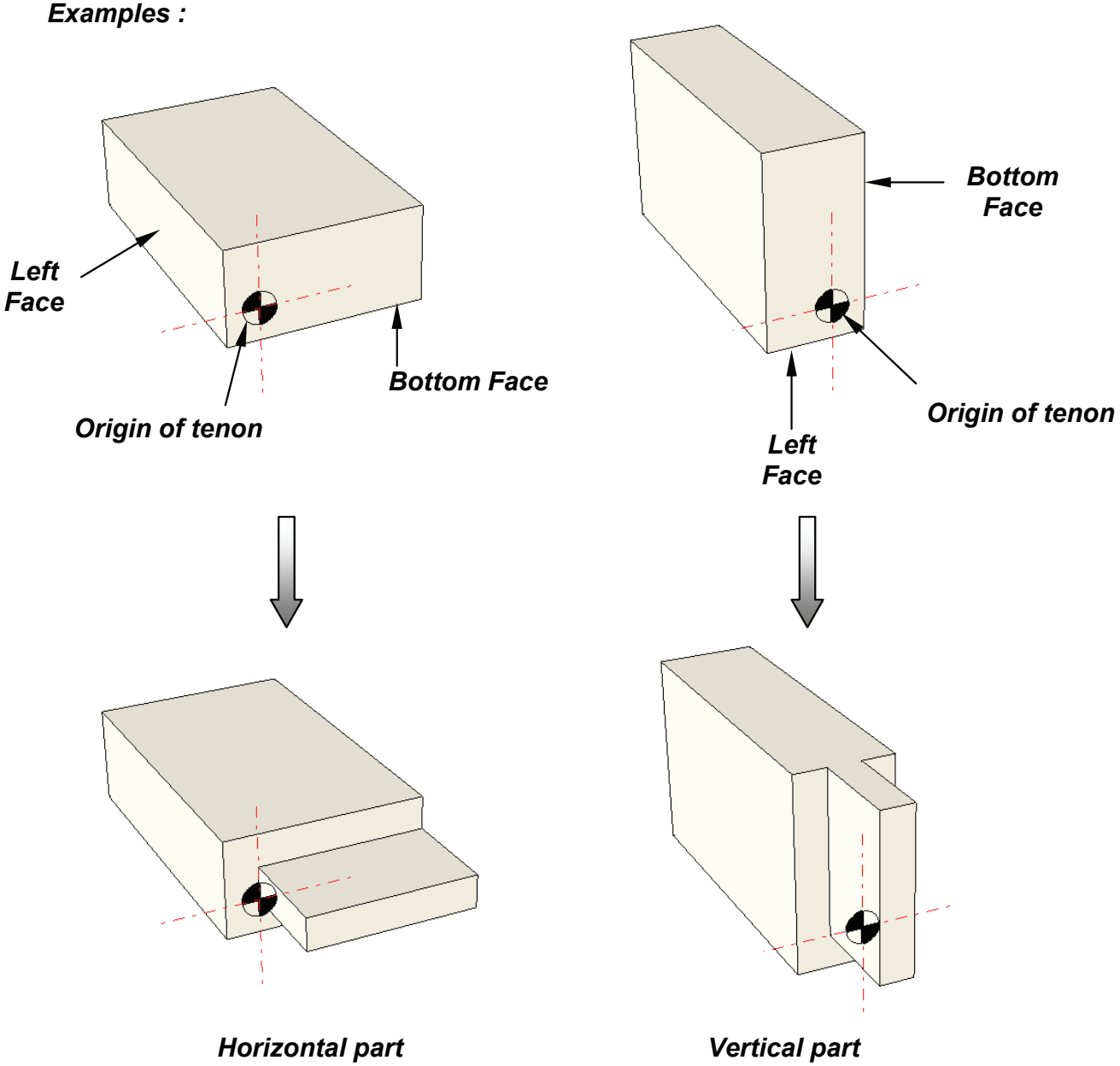
**21** **Tenon** 

*Wood \ Tenon*

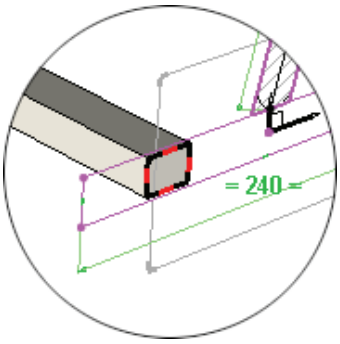
Create a horizontal tenon on one of the three cross-pieces.

A tenon must be created on a planar face.
The origin of the created tenon is the bottom left corner of the selected face.
The function dialogue window allows the user, to position the tenon, according to the orientation of the part.
To simplify the adjustment of the tenon operation, orientate the part horizontally during the selection of the faces.

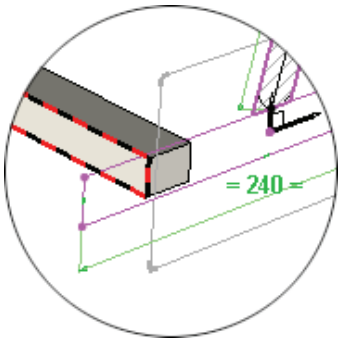
***Examples :***



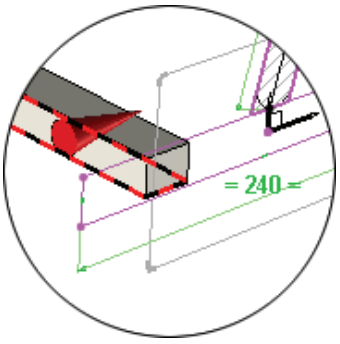
Select the **PERPENDICULAR TO SHOULDER** mode for the direction of the tenon, and then select the support face of the tenon.
Select the left face, then the bottom face.




***Support Face***




***Left Face***




***Bottom Face***

Select the **`Simple`** tenon from the list of **USER MODELS**.

Define the characteristics of the tenon by entering the following information :

- Width setting : `Total width`
- Thickness setting : `e centered`
- Value of L : `15 mm`
- Value of e : `5 mm`

Select **OK.**

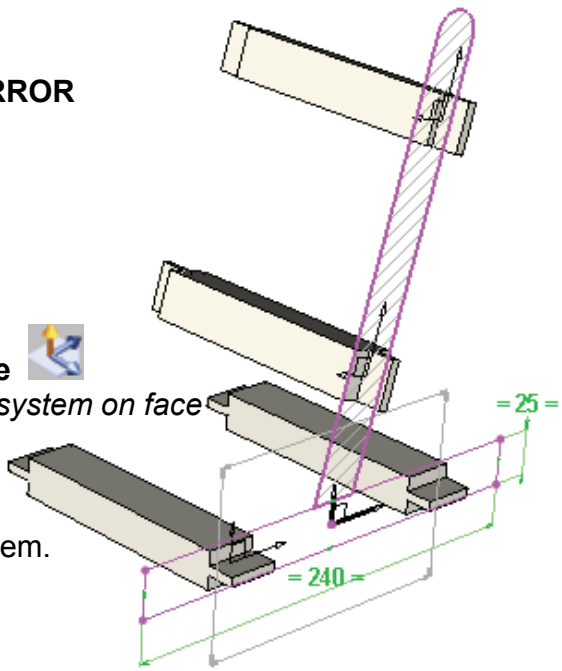
Use the option **PROPAGATION ON OPPOSITE FACE** to obtain the same operation on the other face of the part.

Repeat the procedure for the other two cross-pieces.

**21** **Repeat a cross-piece**
*Edit* \ *Repeat*
Select the bottom cross-piece and then **SIMPLE MIRROR** as the type of repetition. Then select the **YZ** plane

**22** **Activate layer 3**
Make layer 3 current

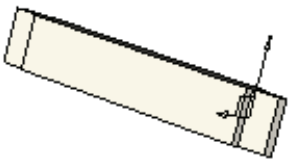
**23** **Create a constrained coordinate system on a face**
*Tools* \ *Coordinate system* \ *Constrained coordinate system on face*
Create a constrained coordinate system on a face centered on the length and the height of the central cross-piece.
Make the created coordinate system the current system.



**24** **Create a rectangular contour**

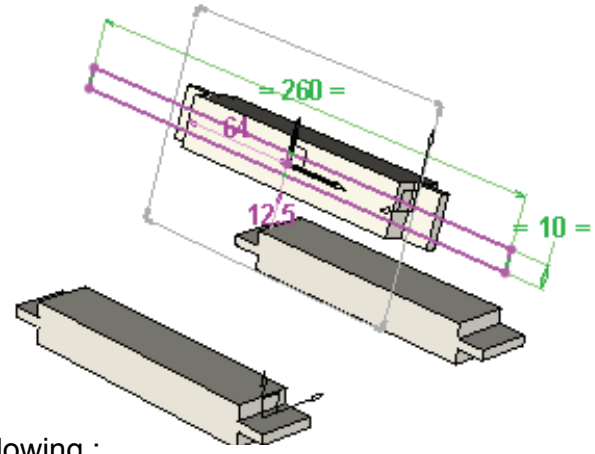
*Curve* \ *Contour* \ *Rectangular*

Draw a **RECTANGULAR Contour** using **AUTO DIMENSION.**



**25** **Add constraints**

Use the **Modify element** function to add a symmetry constraint on the **X** axis then on the **Y** axis.
Select the dimension to be constrained, click on **CONSTRAINT** and then select the axis to use.




**26** **Modify parameter**

*Parameter* \ *Modify*

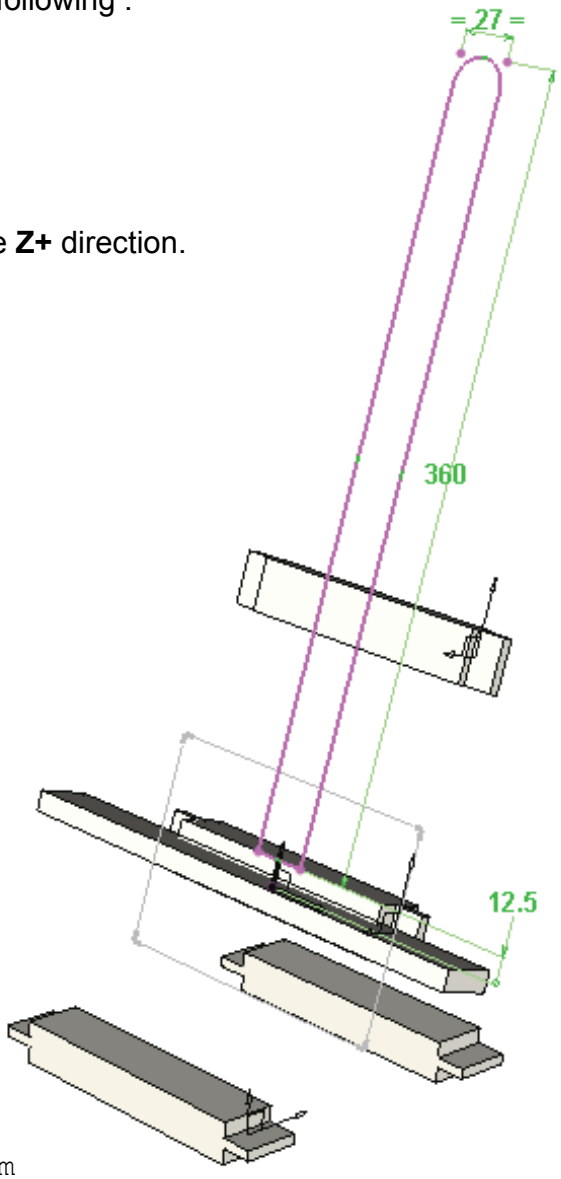
Replace the dimensions of the contour with the following :

- Length : `260 mm`
- Width : `10 mm`

**27** **Extrude**

*Shape* \ *Extruded*

Extrude the contour with a value of `27 mm` in the **Z+** direction.




**28** **Create a rectangular contour**

*Curve* \ *Contour* \ *Rectangular*

Draw a **RECTANGULAR Contour** using **AUTO DIMENSION.**

**29** **Add constraints**

Use the **Modify element** function to add a symmetry constraint on the **Y** axis.
Select the dimension to be constrained, click on **CONSTRAINT** and then select the axis to use.

**30** **Modify parameter**

*Parameter* \ *Modify*

Replace the dimensions of the contour with the following :

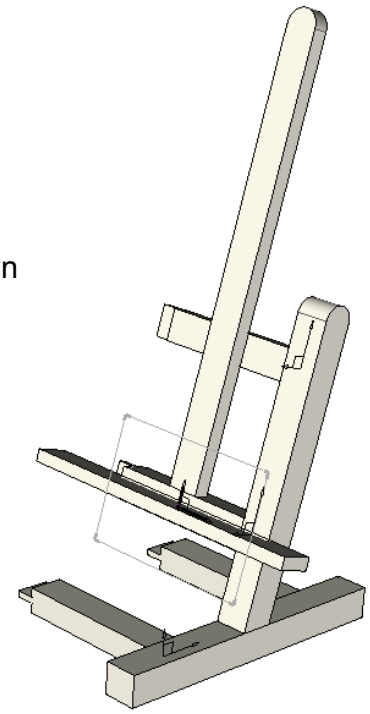
- Height : `360 mm`
- Width : `27 mm`

**31** **Dimension a contour**

*Tools* \ *Dimension*

Dimension the bottom of the contour to `12.5 mm` from the origin of the current coordinate system.

**32** **Fillet**
*Curve* \ *Fillet*
Create a fillet of 13.5 mm on the top of the contour.



**33** **Extrude**
*Shape* \ *Extrude*
Extrude the contour with a value of 12.5 mm in the **Z-** direction

**34** **Activate layer 4**
Make layer 4 current

**35** **Extrude**
*Shape* \ *Extrude*
Extrude the contours present on layer 0 with a value of 27 mm in the **X+** direction.

## Assembly by tenon-mortice

**36** **Tenon-mortice assembly**
*Wood* \ *Tenon-mortice assembly*
Create the assembly by selecting the option **FROM TENON OR MORTICE.**
Select one of the tenons in contact with an element, then the part to mortice.

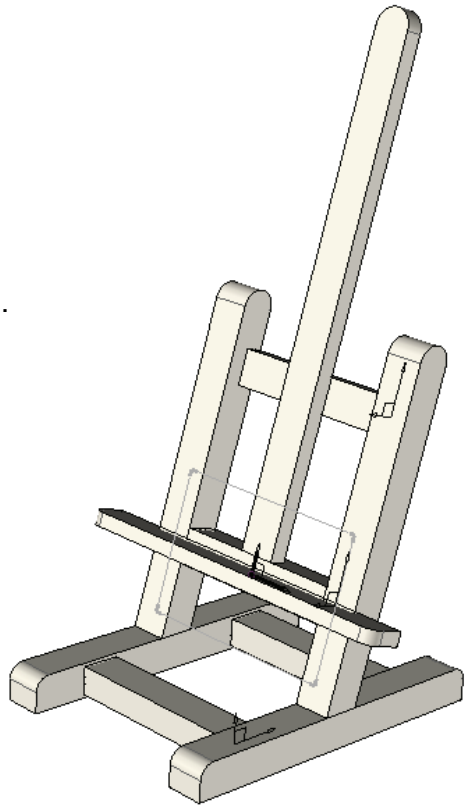
***By default, the function implements a selection filter that makes it possible to only select an edge belonging to a tenon.***

Define the mortices with the following values :

- Play on sides : 0.25 mm
- Play on bottom : 0 mm
- Allowance on cheeks : 0.25 mm

Select **OK.**

Repeat the operation for the other tenons of the easel.



**37** **Fillet**
*Shape* \ *Fillet*
Create a fillet of 10 mm as shown opposite.

**38** **Repeat**
*Edit* \ *Repeat*
Using the selection lasso , select the last two parts extruded.
Choose **SIMPLE MIRROR** as the repetition type, and then use the **YZ** plane.

# Workshop : Creation of a trestle

In this workshop, you will learn to :

- Use various geometry tools,
- Create a thickened profile,
- Use the various points and coordinate systems
- To create a tenon/mortice assembly,
- To repeat elements.

## Create the geometry of the trestle

**1** **Open a new design document**

Choose `Without Template` in the proposed list of **New Document** template models.

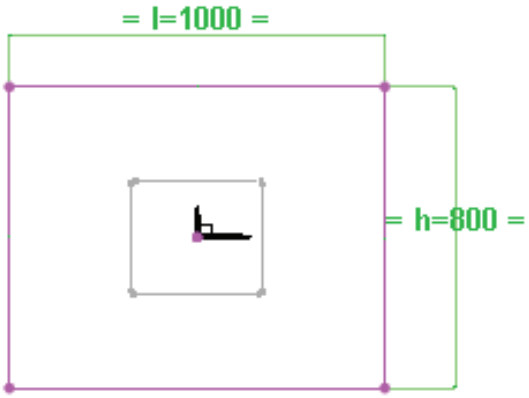
**2** **Create a rectangular contour**

*Curve* \ *Contour* \ *Rectangular*

Draw a **RECTANGULAR Contour** using **AUTO DIMENSION.**

**3** **Add constraints**

Use the **Modify element** function to add a symmetry constraint on the **X** axis then on the **Y** axis.
Select the dimension to be constrained, click on **CONSTRAINT** and then select the axis to use.




**4** **Create the parameters**

*Parameter* \ *Create*

Create the height and length parameters of the trestle: `H=800`, `l=1000`

**5** **Modify parameter**

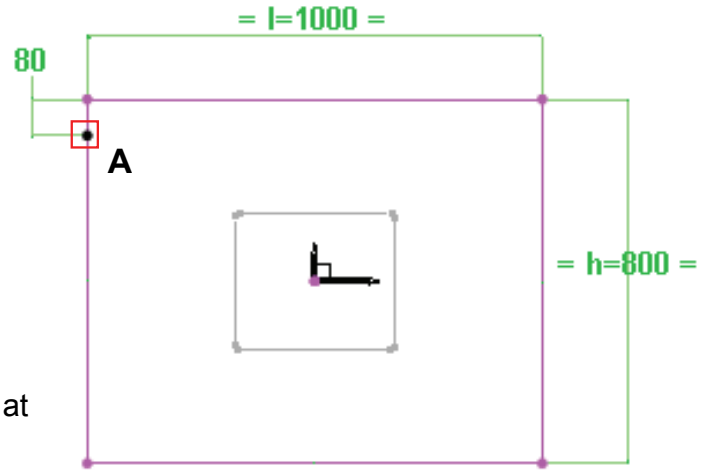
*Parameter* \ *Modify*

Replace the dimensions of the contour with the values shown (See above).

**6** **Create an offset point**

*Tools* \ *Point* \ *Offset point*

Create an offset point `80 mm` in the **Y-** direction, starting from the upper left point. The point created corresponds to point **A**.




**7** **Create a rectangle**

*Curve* \ *Rectangle*

Using the **DIAGONAL** mode, draw a rectangle starting from point **A** and finishing at the upper right corner of the large rectangle.

**8** **Create a rectangular contour**

*Curve* \ *Contour* \ *Rectangular*

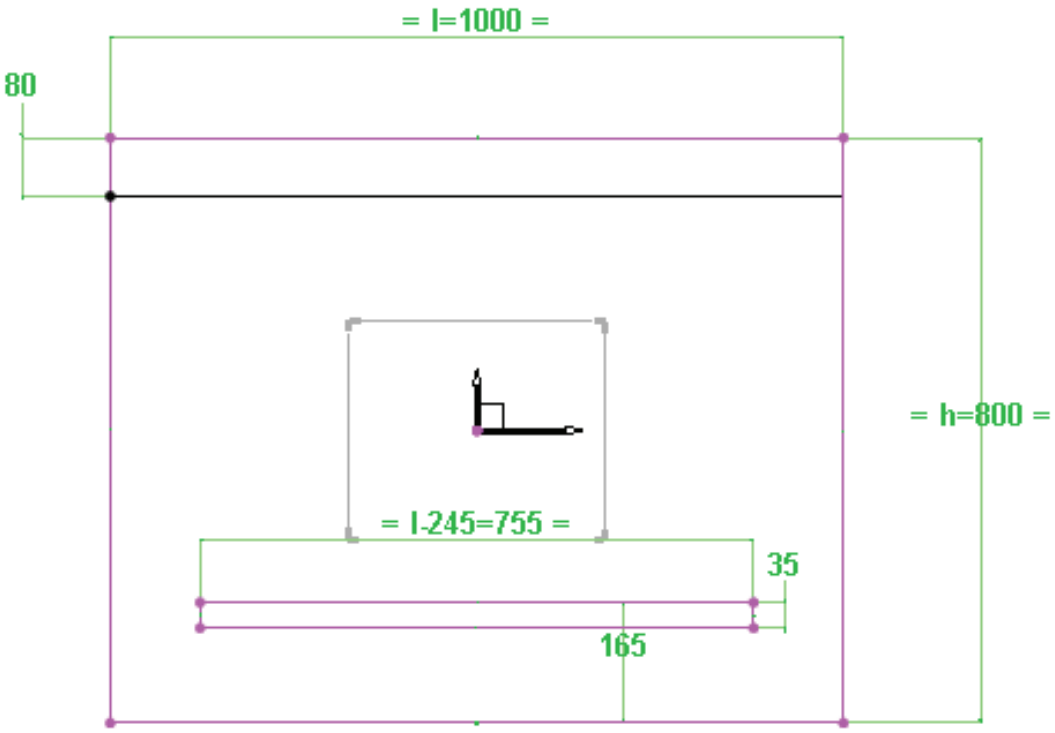
Draw a new **RECTANGULAR Contour** with **AUTO DIMENSION** below the coordinate system.

**9** **Add constraints**

Use the **Modify element** function to add a symmetry constraint on the **Y** axis.
Select the dimension to be constrained, click on **CONSTRAINT** and then select the axis to use.

**10** **Dimension a contour**

*Tools* \ *Dimension*
Dimension the top of the previously created rectangle to `165 mm` from the bottom.



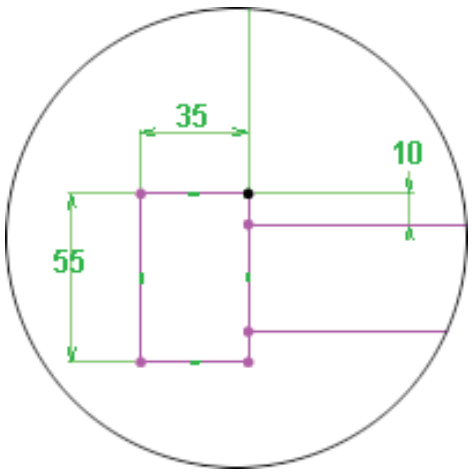

**11** **Modify parameter**

*Parameter* \ *Modify*
Replace the contour dimensions with the values opposite (See above).

- Length : `l-245 mm`
- Height : `35 mm`

**12** **Create an offset point**

*Tools* \ *Point* \ *Offset point*
Create an offset point `10 mm` in the **Y+** direction, starting from the upper left point of the previously created contour.




**13** **Create a rectangular contour**

*Curve* \ *Contour* \ *Rectangular*

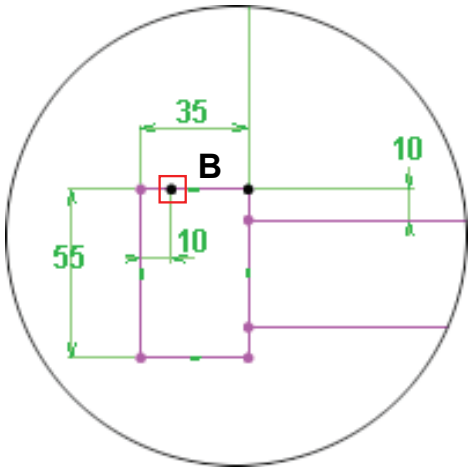
Draw a new **RECTANGULAR Contour** with **AUTO DIMENSION**.

**14** **Modify parameter**

*Parameter* \ *Modify*

Replace the dimensions of the rectangular contour with the following values :

- Width : 35 mm
- Height : 55 mm




**15** **Create an offset point**

*Tools* \ *Point* \ *Offset point*

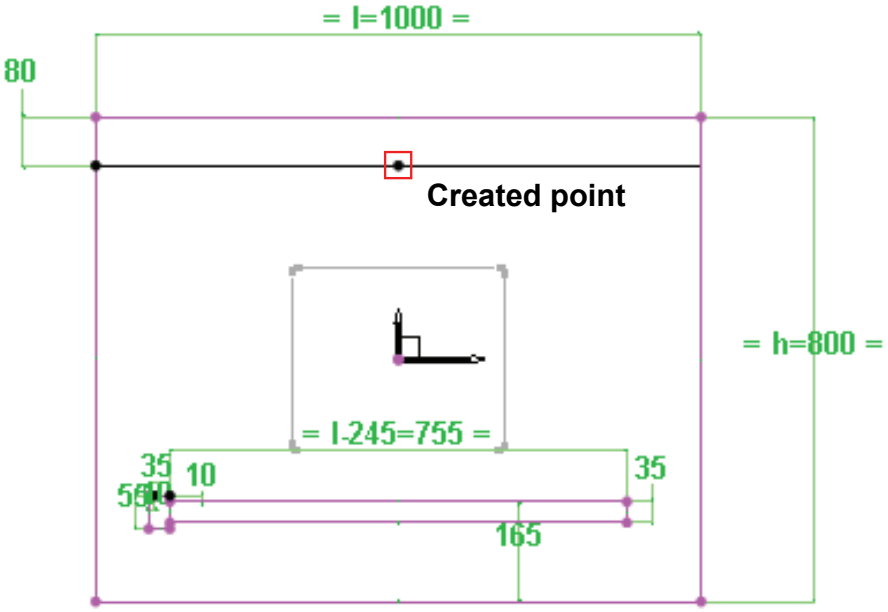
Create an offset point 10 mm in the **X+** direction, starting from the upper left point of the previously created contour.

The point created corresponds to the point **B**.

**16** **Create a centre point**

*Tools* \ *Point* \ *Centre key point*

Create a centre point on the lower segment of the upper rectangle as shown below.




**17** **Create an offset point**

*Tools* \ *Point* \ *Offset point*

Create an offset point 50 mm in the **X-** direction, starting from the centre point created previously.

The point created corresponds to point **C**.

**18** **Create a line**

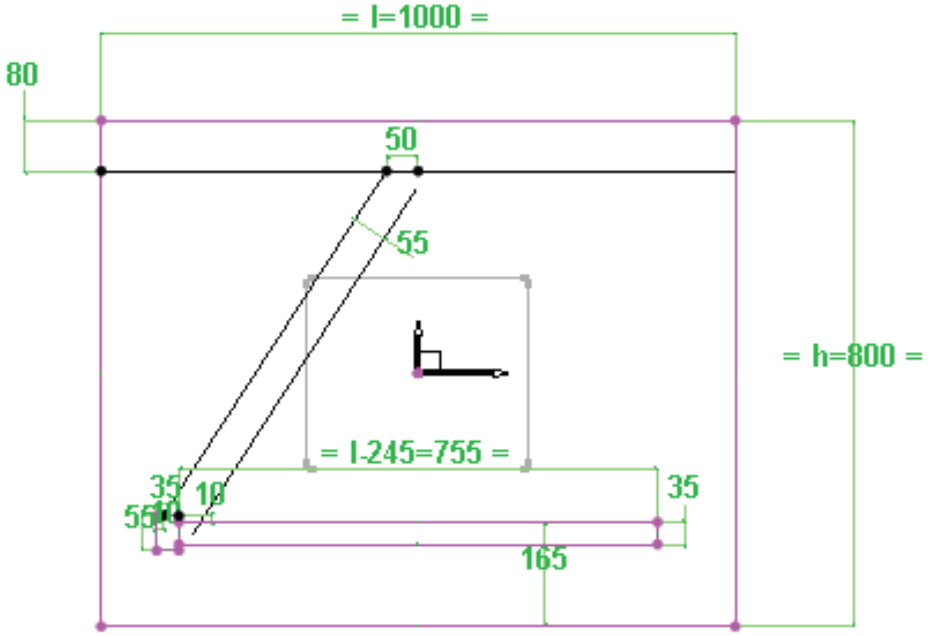
*Curve* \ *Line*

Draw a line starting on point **B** and ending on point **C**.

**19** **Create offset curves**

*Curve \ Offset curve*

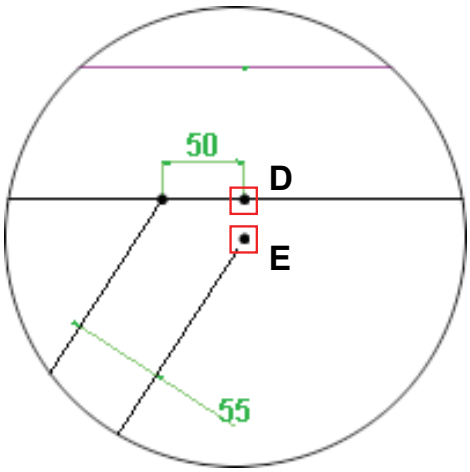
Create an offset curve, offset by `55 mm` from the previously created line. *(See the diagram below).*




**20** **Create a projected point**

*Tools \ Point \ Projected point*

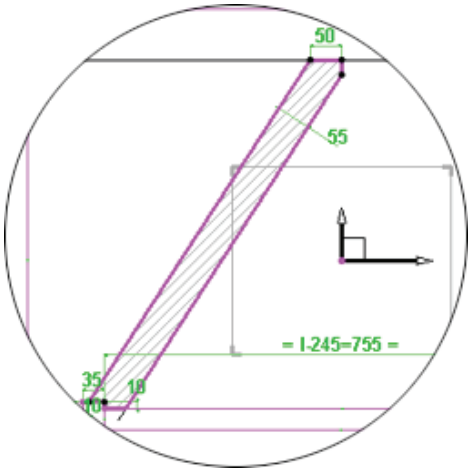
Create a projected point **E**, in the **Y-** direction to the offset curve previously created. Use point **D** as the reference point.




**21** **Create a contour**

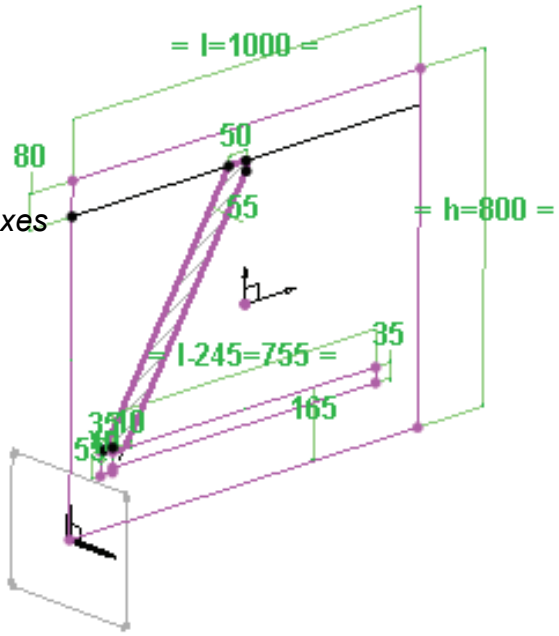
*Curve \ Contour*

Draw a **closed contour** as shown opposite while passing through the points **D** and **E** created previously. (Contour surrounding the hatched zone)

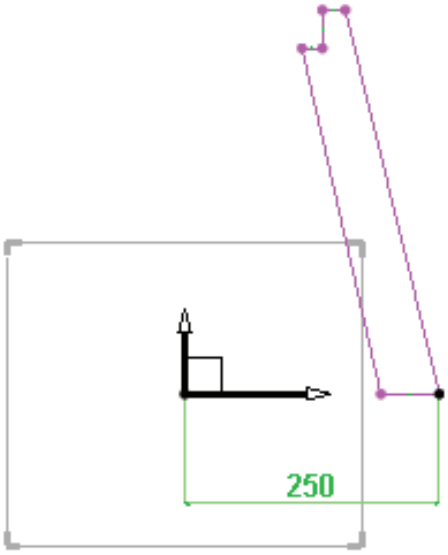


**22** **Activate layer 1**
Make layer 1 current.

**23** **Create a coordinate system**
*Tools* \ *Coordinate system* \ *Coordinate system 2 axes*
Select **THROUGH POINT**.
Click on the lower left corner of the enclosing rectangle, then click on **Z+.**
Use **Y+** as the **Y** axis.
Make the created coordinate system current.

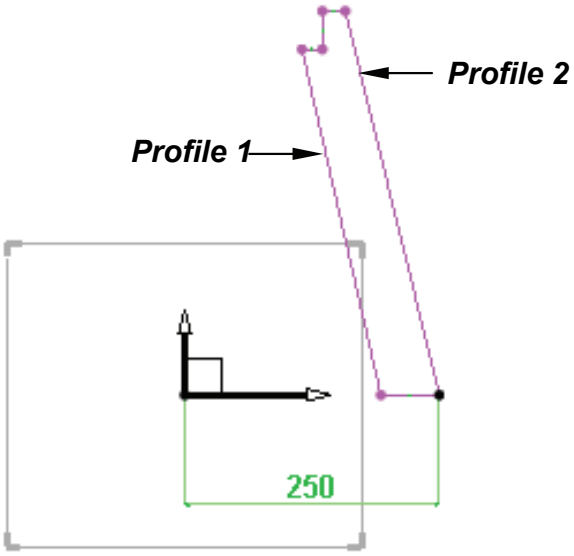



**24** **Create an offset point**
*Tools* \ *Point* \ *Offset point*
Create an offset point `250 mm` in the **X-** direction, starting from the origin of the current coordinate system.



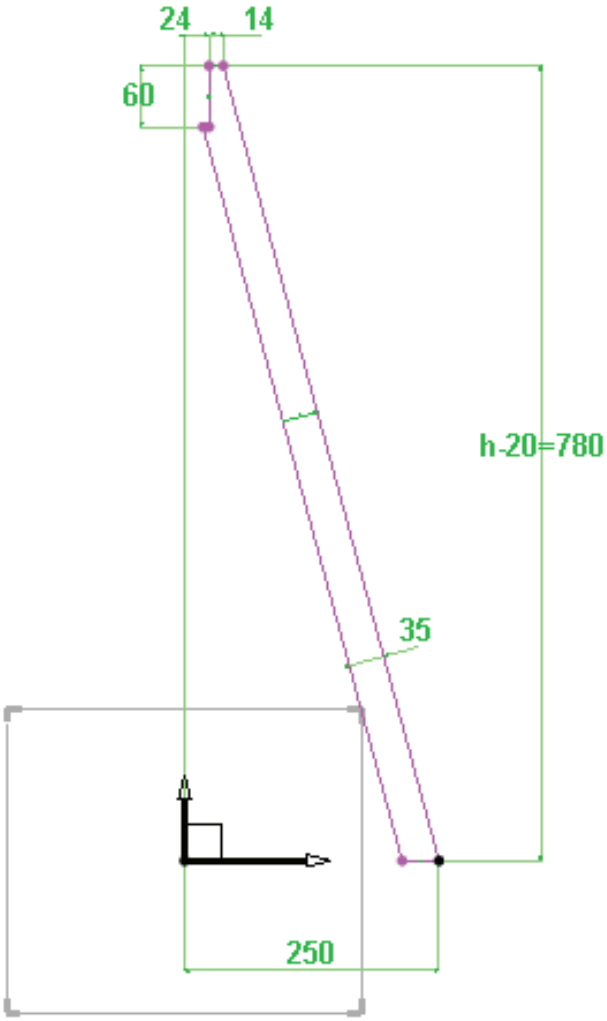

**25** **Create a contour**
*Curve* \ *Contour*
Draw a closed contour as shown opposite.
Use the previously created point as the starting point of the contour.

**26** **Add a constraint**
*Tools* \ *Constraint*
Add a parallel constraint between two elements.
Choose **PARALLEL**, from the drop-down list.
Select the profiles as shown opposite.
A parallel constraint appears.





***Representation of a parallel constraint***




**27** **Dimension a contour**

*Tools* \ *Dimension*

Dimension the contour as shown opposite.

**28** **Activate layer 2**

Make layer 2 current.

## Model the trestle

**29** **Extrude**

*Shape* \ *Extruded*

Extrude the contour a value of `70 mm` in the **Z-** direction with an offset distance of `70 mm`.

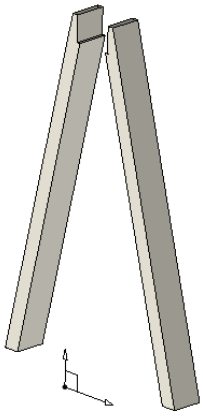
**30** **Repeat**

*Edit* \ *Repeat*

Make the **ABSOLUTE COORDINATE SYSTEM** current.

Select the foot, then choose **SIMPLE MIRROR** as the repetition type from the list.

Use the **XY** plane.



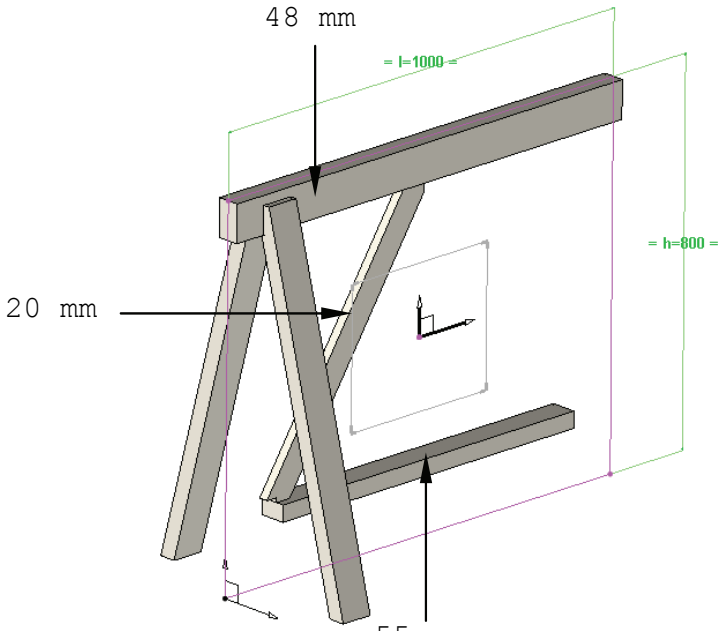
**31** **Activate layer 3**

Make layer 3 current.

**32** **Extrude**

*Shape \ Extruded*

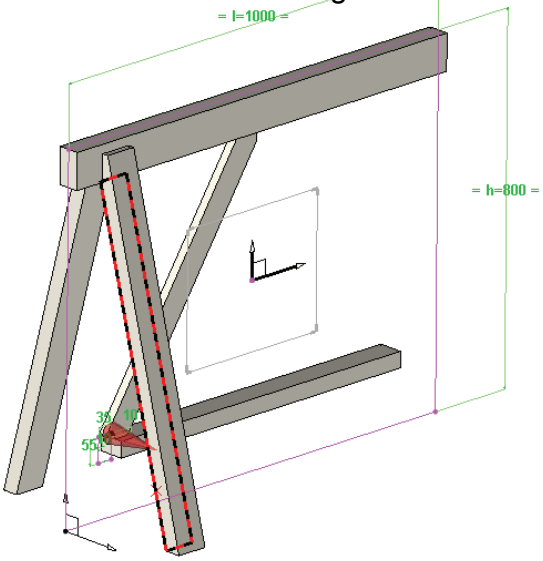
Extrude the following contours using the **CENTERED** mode with the extrusion values shown :



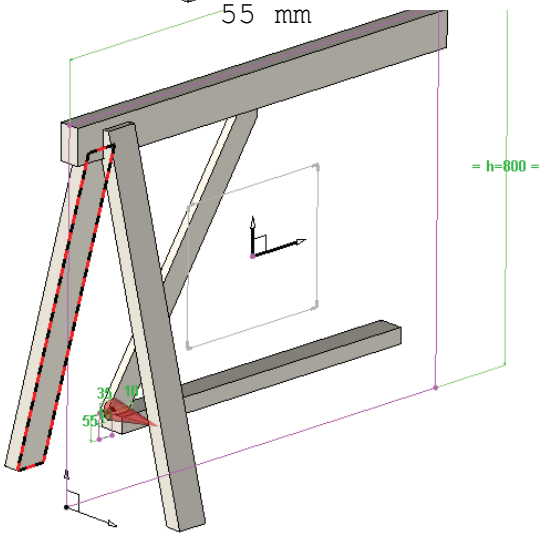

**33** **Extrude**

*Shape \ Extrude*

Extrude the last contour using the **CENTERED** mode. Use the **TO** mode and use the following faces :

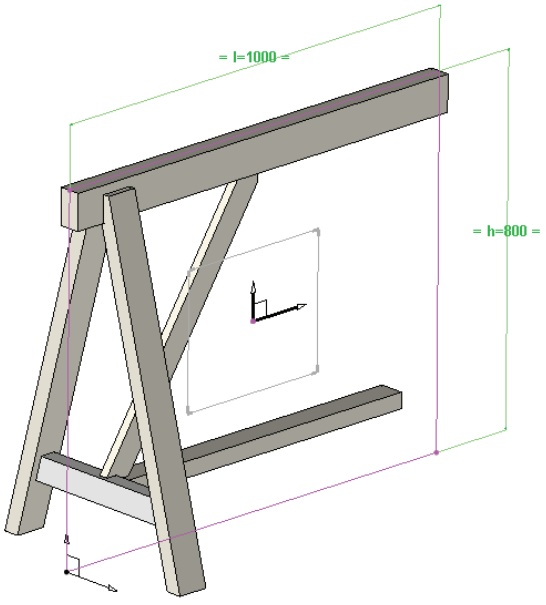



***Interior face of the front foot***




***Interior face of the back foot***

***Result***

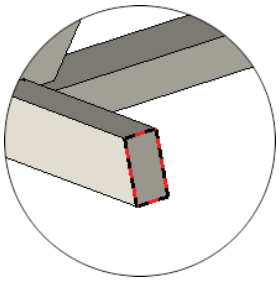


**34** **Turn off layers**
Turn off layers 0, 1 and 2

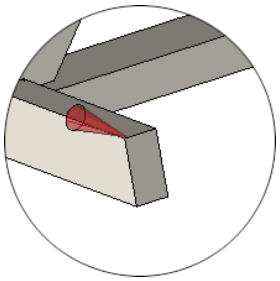
## Tenon the lower cross-piece

**35** **Tenon** 
*Wood* \ *Tenon*
Create a horizontal tenon on the lower cross-piece.

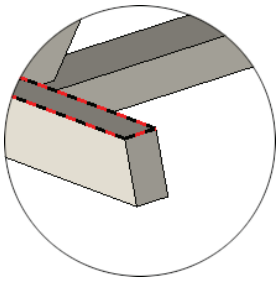
Selection the **MANUAL** mode for the direction of the tenon, and select the support face of the tenon.
Click on an edge to define the direction of the tenon.
Click on the red arrow if the direction is to be changed.
Select the left face, then the bottom face of the tenon.



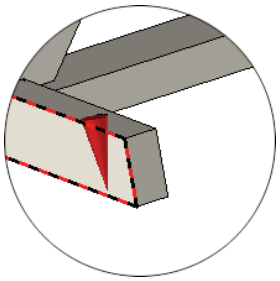
***Support face***



***Direction of tenon***



***Left face***



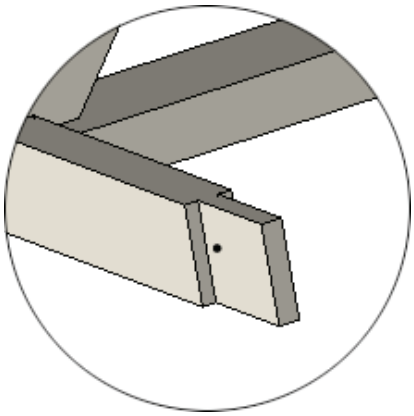
***Bottom face***

Select the **simple** tenon from the **USER MODELS** list.

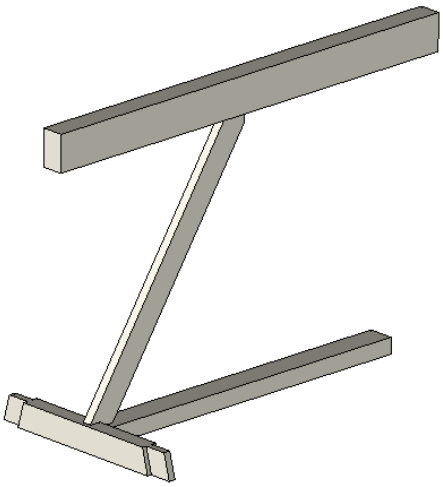
Define the characteristics of the tenon by entering the following information :

- Width setting : `Total width`
- Thickness setting : `e centered`
- Value of L : `50 mm`
- Value of e : `15 mm`

Select **OK.**



***Result***



Use the option **PROPAGATION ON OPPOSITE FACE** to obtain the same operation on the other face of the part.

**36** **Activate layer 2**
Make layer 2 current

## Assembly by tenon-mortice

**37** **Tenon-mortice assembly** 
*Wood* \ *Tenon-mortice assembly*
Create the assembly by selecting the option **FROM TENON OR MORTICE.**
Select one of the tenons in contact with an element, then the part to mortice.

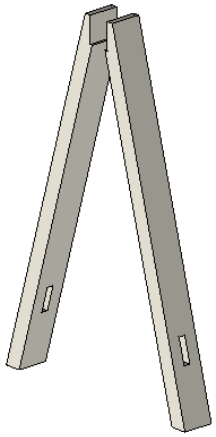
***By default, the function implements a selection filter that makes it possible to only select an edge belonging to a tenon.***

Define the mortices with the following values :

- Play on sides : 0.25 mm
- Play on bottom : 0 mm
- Allowance on cheeks : 0.25 mm

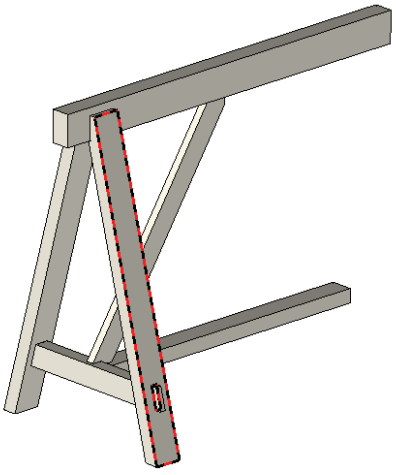
Select **OK.**

Repeat the procedure to create another tenon/mortice assembly using the other tenon.

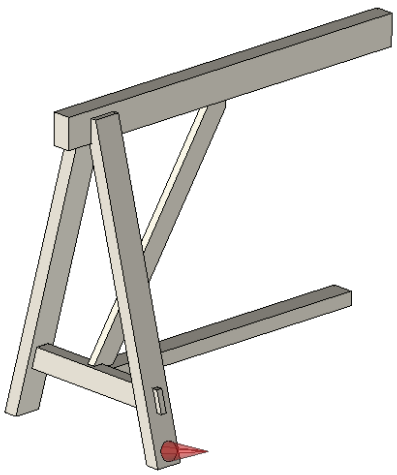


***To view the result, turn off layer 3.***

**38** **Trim a part by a plane** 
*Shape* \ *Trim*
Choose from the list the **BY PLANE** mode, then click on the lower cross-piece.
Use the front face of the foot as the Trimming Plane.
The direction of the red arrow corresponds to the side which the material will be removed from. This must point towards the outside of the part.



***Trimming Plane***

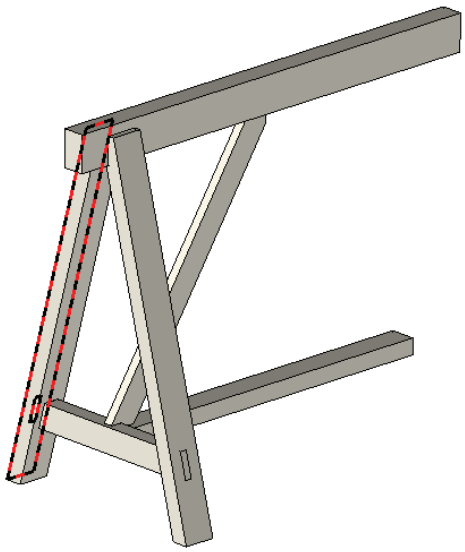


***Direction***

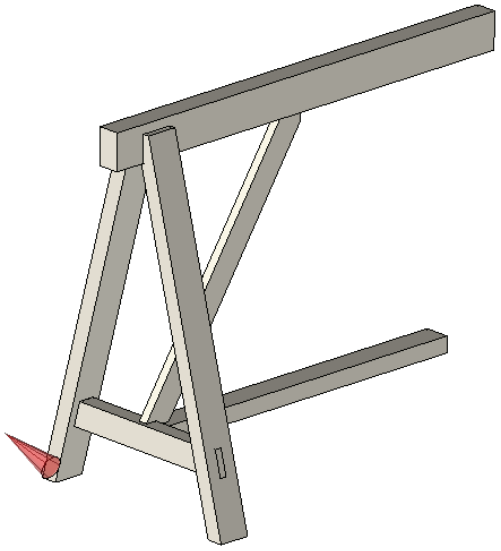
Select **OK** to confirm.

Ensure **Hide tools = NO** is selected and then select **OK.**

Follow the same procedure to trim the length of the second tenon.
Use the same options as above.



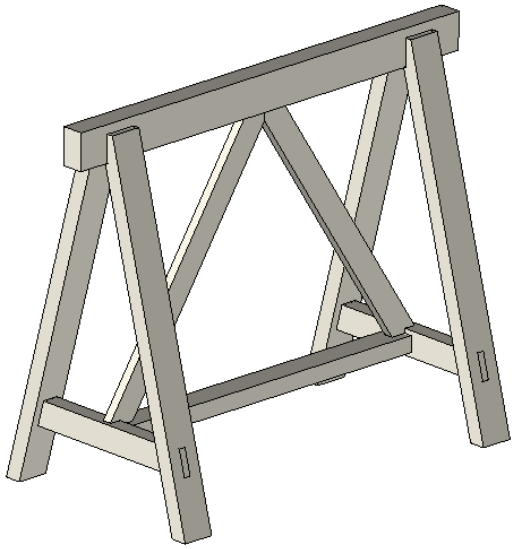
***Trimming plane***



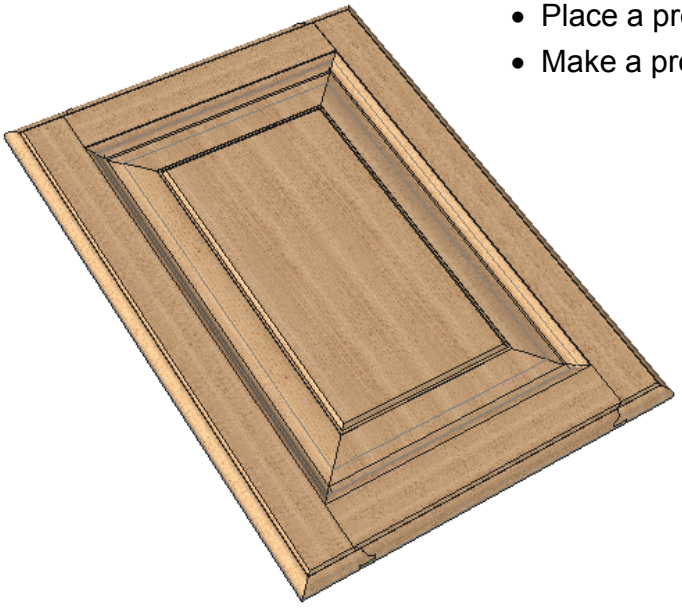
***Direction***

**39** **Repeat** 

*Edit* \ *Repeat*
Select the feet.
Choose **SIMPLE MIRROR** as the repetition type, and then use the **YZ** plane.
Now select the cross-piece for the supports, then the bottom cross-piece in order to obtain the result shown below.



# Workshop : Creation of a door



On this workshop you will learn to :

- Model the parametric elements,
- Place a profile,
- Make a profile / counter-profile assembly.

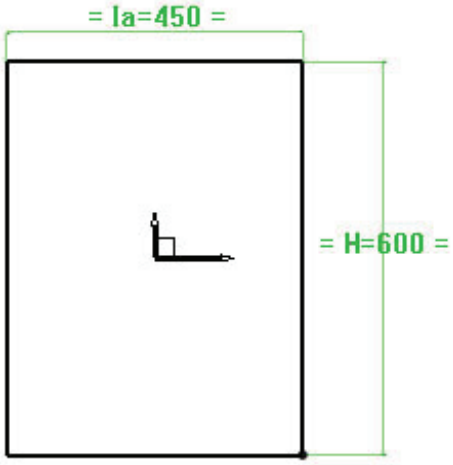
## Create the geometry for the door

**1** **Open a new document**

Choose `Without Template` in the proposed list of **New Document** template models.

**2** **Create a rectangular contour**
*Curve \ Contour \ Rectangular*

Design a **Contour RECTANGULAR** in **AUTO DIMENSION.**




**3** **Add the constraints**

Use the function **Modify element** to add a symmetry constraint on the **X** axis then on the **Y** axis.

**4** **Create the parameters**
*Parameter \ Create*

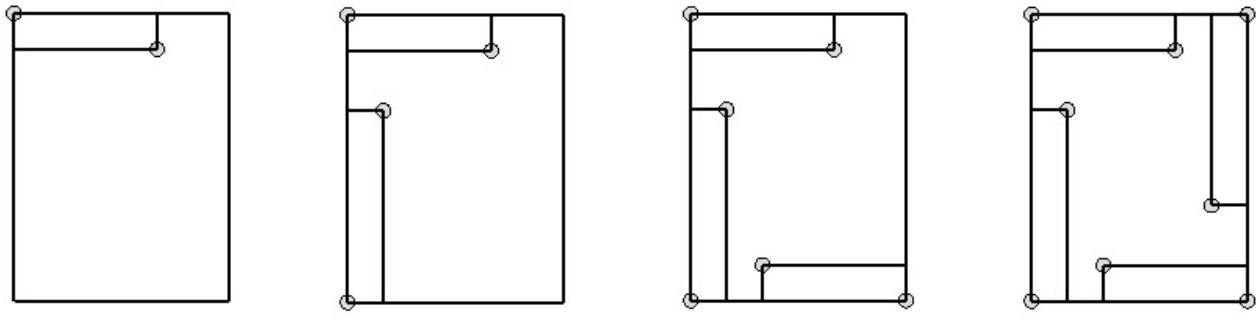
Create the length and width parameters for the door: `H=600`, `la=450`

**5** **Modify parameter**
*Parameter \ Modify*

Change the contour's dimension for those of their parameters (See below).

**6** **Create 4 rectangular contours**

Using the function **RECTANGULR CONTOUR**, trace 4 rectangular contours anchored to the 4 corners of the previously constructed rectangle, then make the **auto dimension** of each entity.

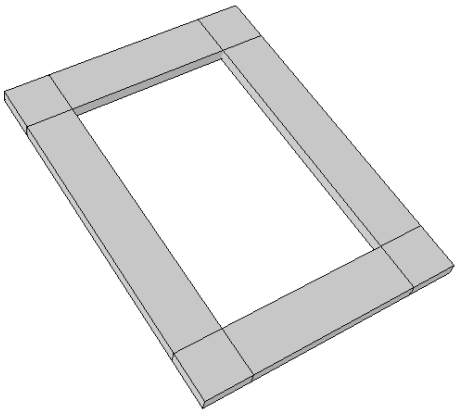


**7** **Dimension the 4 rectangular contours**

Change the dimensions for the corresponding `H` and `la` parameters.
Create an `s` parameter with a value of 80mm and apply it on the width of the 4 rectangles.

**8** **Activate layer 1**
Make current layer 1

**9** **Extrude the jambs and transoms**
*Shape \ Extruded*
Create an extruded shape pm the **Z+** axis with a value of 22 mm from each rectangular contour.



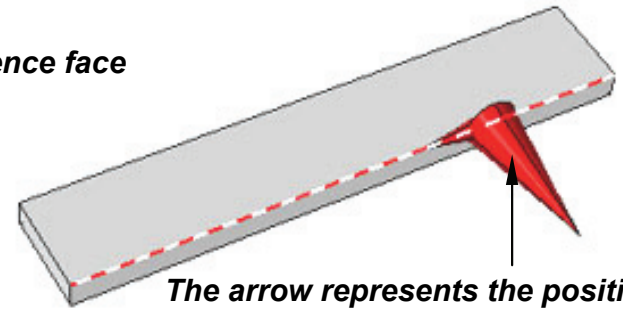
**10** **Verify and save**
Modify the values of the `H` and `la` parameters to validate the construction.

## Profiling the jambs and transoms

**11** **Internal molding**
*Wood \ Molding*
Make the molding on the upper transom by selecting

- the reference face (upper face of the shape)
- the path of the tool (the longest edge of the shape)

***Reference face***



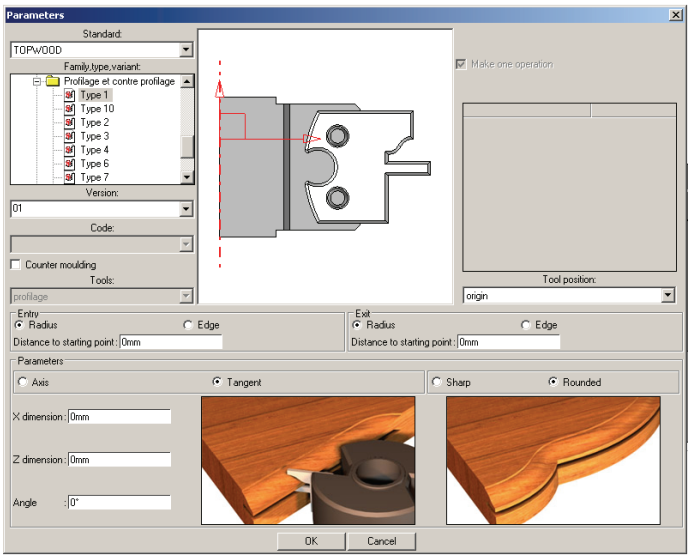
***Tool's path***

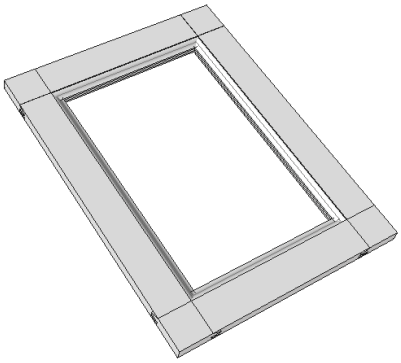
***The arrow represents the position of the tool's axis, so it must be oriented towards the outside of the part***

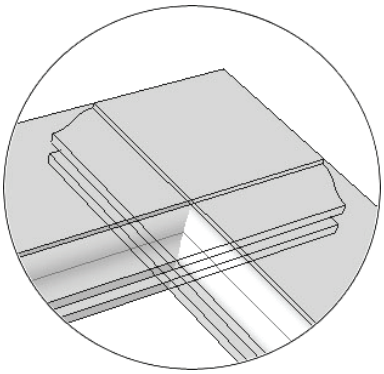
Select the `Profilage et contre profilage/Type 9` tool.

Select **OK**.

Use the option **COPY MOULDING** to make the same machining on the other 3 parts. Select the face and the tool path for each part.







**COPY MOULDING** will merge the type of molding applied on the jambs and transoms.

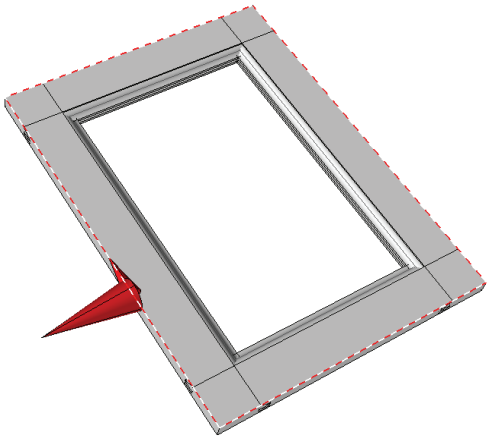
**12** **External molding** 
*Wood \ Molding*

Make the external molding of the door with the option

Apply it to one of the parts of the frame by selecting: Join edges=

- the reference face (upper face of the shape)
- the path of the tool *

* *In this case the path tool is made by the successive selection of the 4 edges corresponding to the lengths of the 4 parts (the tool path will appear as a red and white rectangle on the screen).*

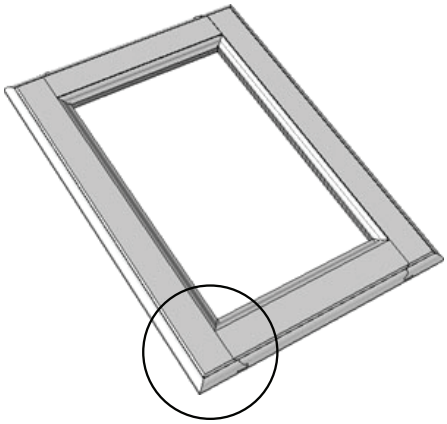


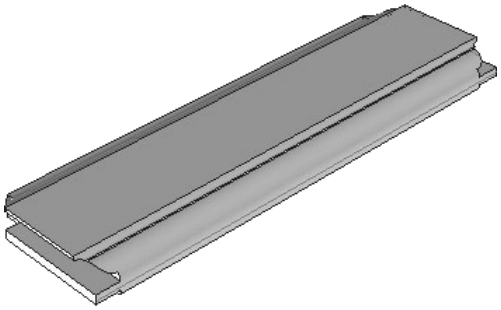
Select a `Profilage/Type 1`.tool
Click **OK**.

## Counter molding of the transoms

**13** **Counter molding** 
*Wood / Counter molding*
Make the counter molding on the upper transom of the frame by selecting:

- the shape to modify >> the transom to counter molding
- the reference molding >> an edge on the internal molding of the jamb



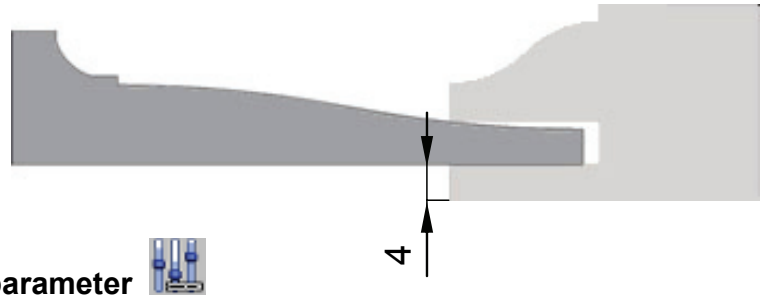


## Model the panel

**14** **Activate layer 3**
Make current layer 3

**15** **Create a rectangular contour**
Design a **Contour RECTANGULAR** in **AUTO DIMENSION** centered on the absolute coordinate system.

**16** **Extrude the panel**
*Shape \ Extruded*
Extrude the contour with a value of 15mm following **Z+** with an offset value of 4mm with the help of the option >> in order to match the panel to the slot.

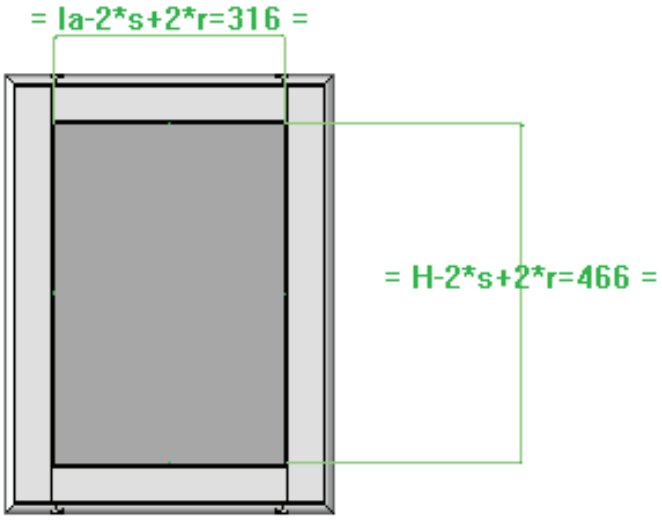



**17** **Create a parameter**
*Parameter \ Create*
Create a parameter `r` with a value of 17mm

**18** **Modify parameter**
*Parameter \ Modify*
Change the parameters of the panel to the following values:

- the -2*s+2*r for the width,
- H -2*s+2*r for the height.

*In order to make the following operation easier, deactivate the layers 1 and 2 to hide the jambs and transoms.*



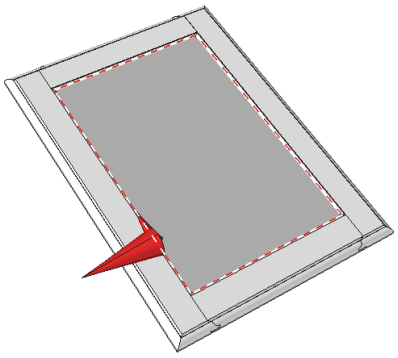

## Profiling the panel

**19** **Molding of the flat band**
*Wood \ Molding*
Execute the molding on the central panel by selecting:

- the reference face (upper face of the shape)
- the tool path (rectangular contour or the edges of the upper face).

Select the `Flat band/Type 1 tool`

Click **OK**



# Workshop : Creation of a kitchen table

In this workshop you will learn to :

- Model the parametric elements,
- Define a key point,
- Make a tenon/mortise assembly,
- Repeat and duplicate parts,
- Make mitre cuts

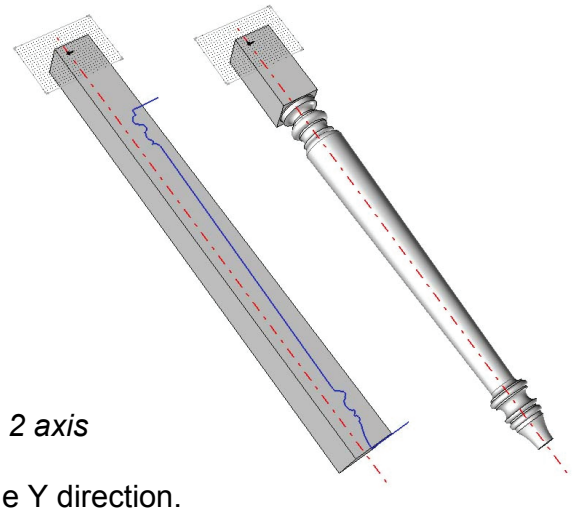
## Create the table's legs

**1** **Open a design document**
*File \ Open*
Open the **Pied** file available at this address :
ftp.topsolid.com/public/TopSolidWood/Support TopSolid'Wood 2008.

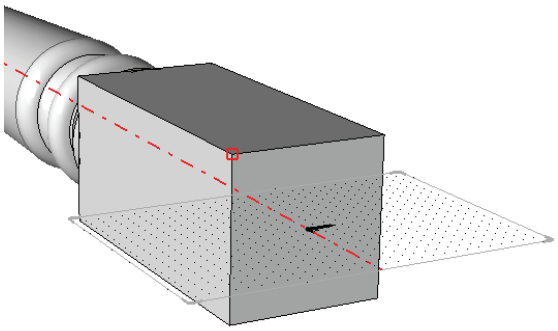
**2** **Extrude the leg**
*Shape \ Extruded*
Create an extruded shape in **CENTERED** mode with a value of 65 mm from the rectangular profile.

**3** **Turning**
*Wood \ Turning*
Select the part to machine, then the profile.
Use the length's axis as the turning axis.
Confirm the direction.

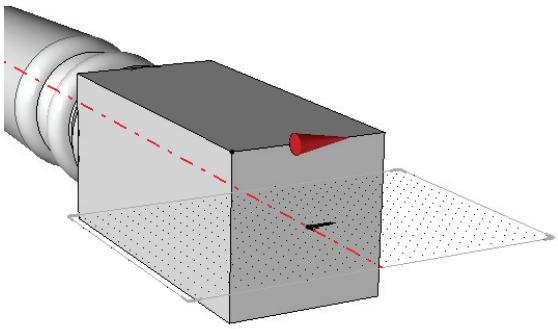
Select **OK.**



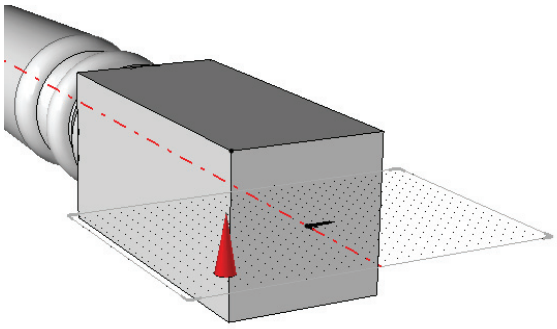
**4** **Create a coordinate system**
*Tools \ Coordinate system \ Coordinate system 2 axis*
Create a coordinate system following 2 axis.
Select the passing point, the X direction then the Y direction.



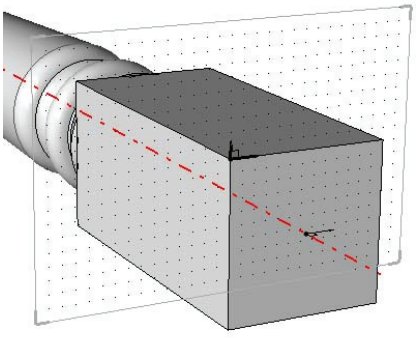
Passing point



**X+** direction



**Y+** direction



Make current

**5** **Define a key point**
*Assembly* \ *Define component* \ *Define key points*
Click on the previous coordinate system, name it as `fr1` and define it as `leg coordinate system.`

**6** **Leg's definition**
*Wood* \ *Define part*
Define the part giving:

- A designation: `Leg`
- A material: `Oak`

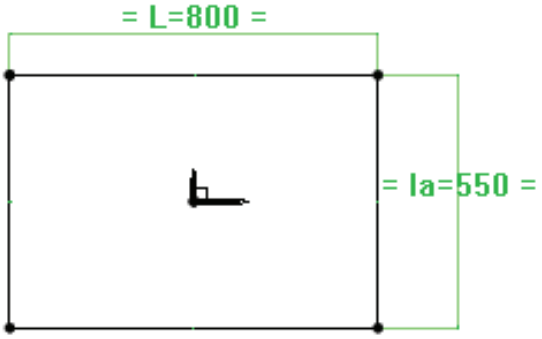
**7** **Save**
*File* \ *Save*
Save the filer

## Create the transoms

**8** **Open a new document**
Select **New** design document from the proposed list `Without a template.`

**9** **Create a rectangular contour**
*Curve* \ *Contour* \ *Rectangular*
Draw a **RECTANGULAR CONTOUR** in **AUTO DIMENSION.**

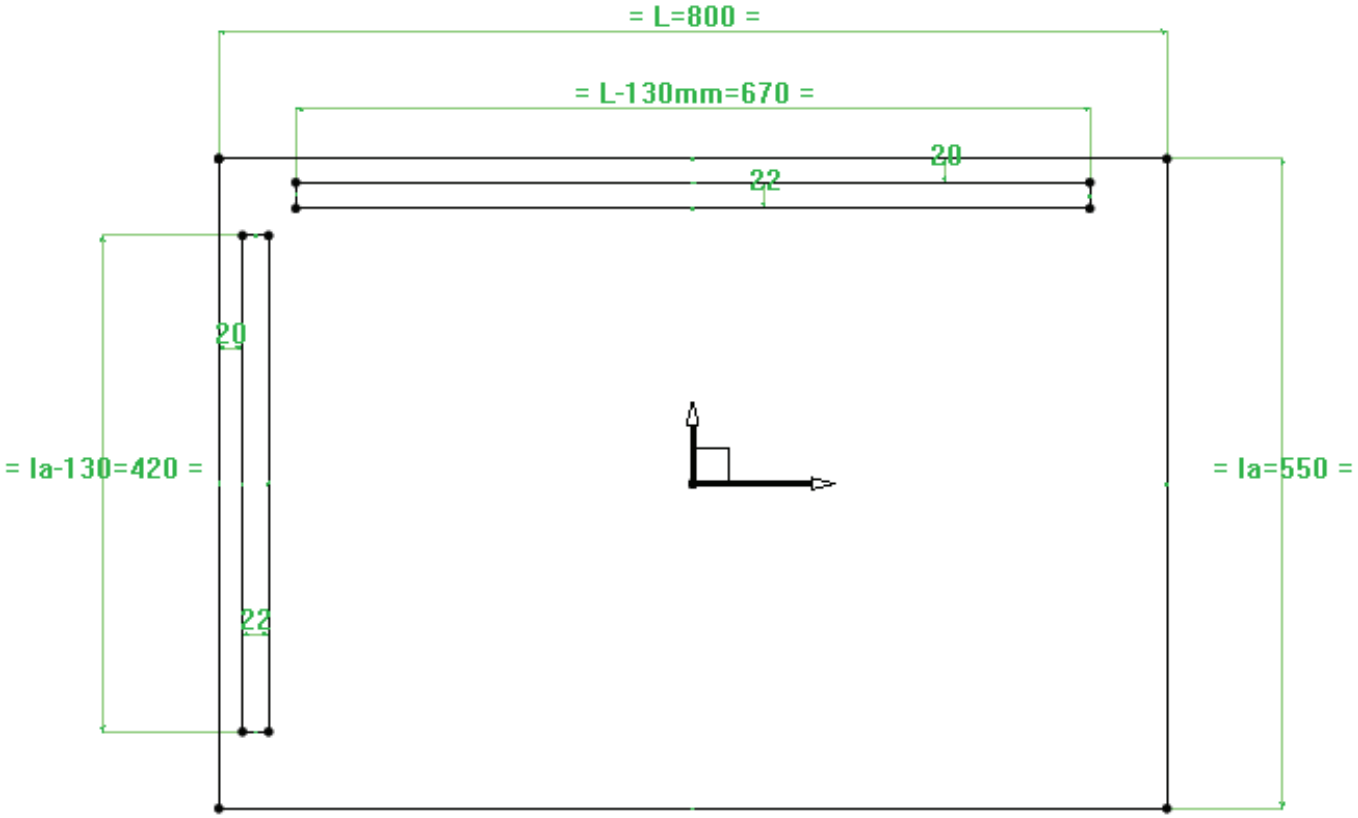
**10** **Add the constraints**
Use the function **Modify** element to add a symmetry constraint on the **X** axis en then on the **Y** axis.




**11** **Create the parameters**
*Parameter* \ *Crate*
Create the table's length and width parameters: `L=800`, `la=550`

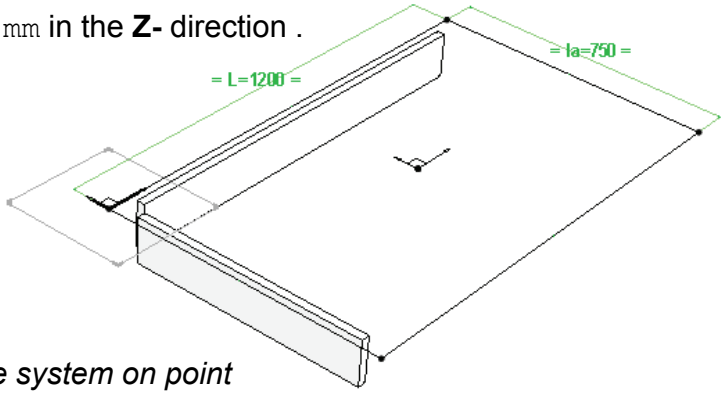
**12** **Modify parameter**
*Parameter* \ *Modify*
Substitute the dimensions of the contour by their parameters (See above).

**13** **Create 2 rectangular contours**
*Curve* \ *Contour* \ *Rectangular*
Design 2 rectangular contours as the table's transoms. Each one of these must be constrained on the length and have an offset of 20 mm.




Values for the transoms: `L-130 mm` for the length
`la-130 mm` for the width
`22 mm` for the thickness

**14** **Extrude the transoms**
*Shape* \ *Extruded*
Extrude each part with a value of `120 mm` in the **Z-** direction .




## Import the leg

**15** **Activate layer 1**
Make current layer 1

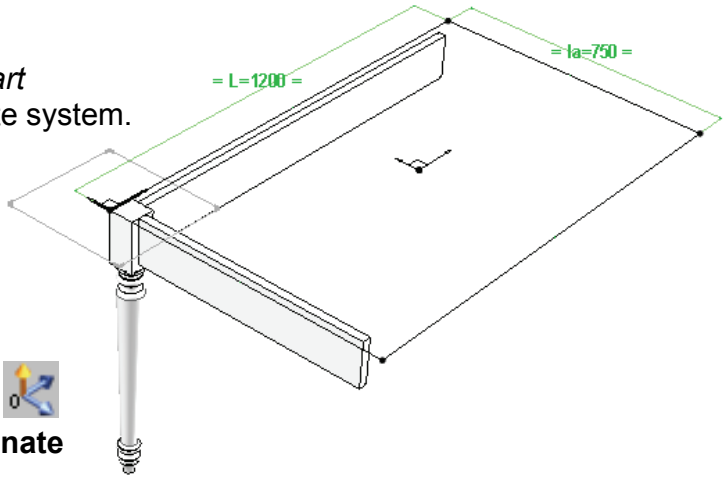
**16** **Create a coordinate system**
*Tools* \ *Coordinate system* \ *Coordinate system on point*
Create a coordinate system on the upper left corner of the rectangle.
Make it current

**17** **Inset the leg**

*Assembly \ Include sub assembly - part*

Insert the leg on the current coordinate system.

Click **STOP.**




## Repeat the leg and the transoms

**18** **Make a coordinate system current**

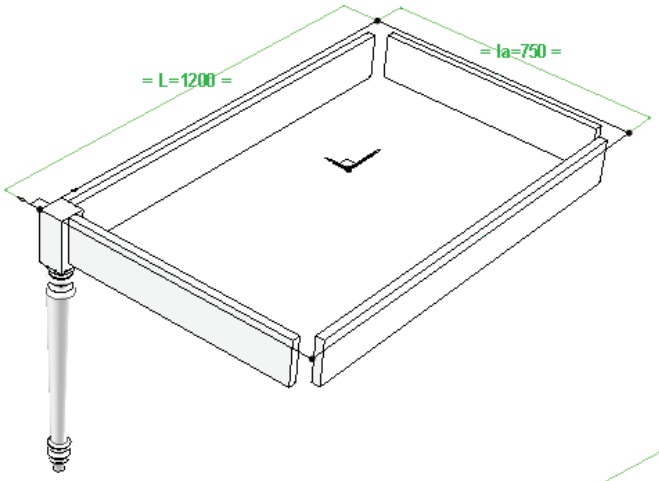
Make current the **ABSOLUTE coordinate system**

**19** **Repeat the transoms**

*Edit \ Repeat*

Select the part to repeat and choose the repetition type **SIMPLE MIRROR**:

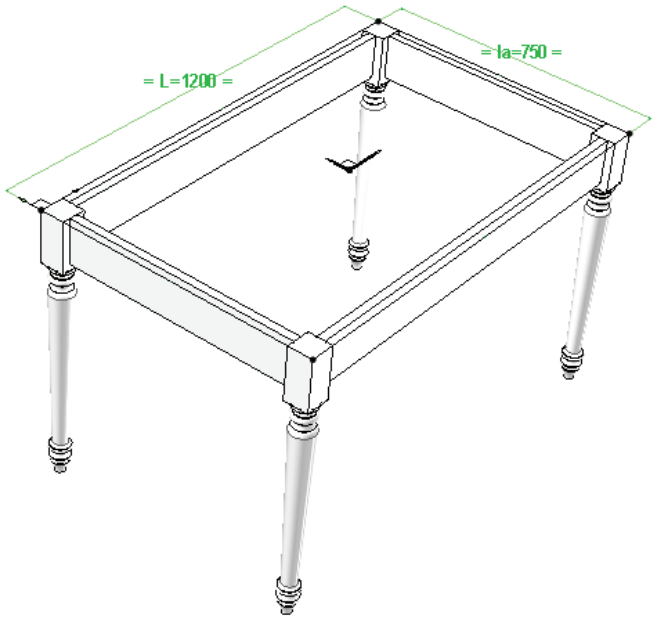
- for the length's transom use the plane **XZ**.
  for the width's transom use the plane **YZ**.



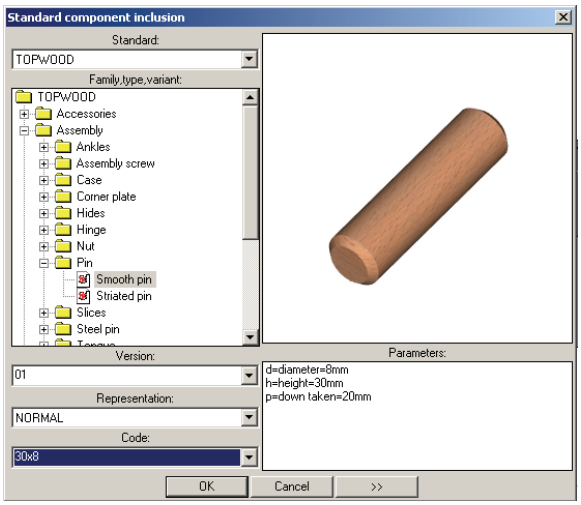

**20** **Repeat the leg**

*Edit \ Repeat*

Select the leg and choose the repetition type **DOUBLE MIRROR** with the planes **XZ** and **YZ.**



# Pin assembly



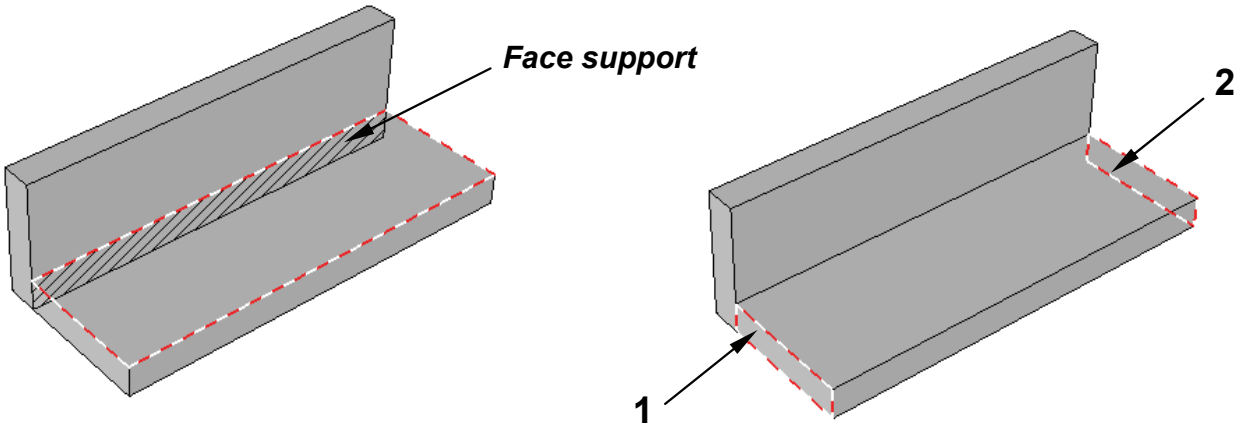

**21** **Pin assembly** 

*Wood* \ *Other processes* \ *Pin assembly*

Select the component `30 x 8 Smooth pin` available in the standard `TOPWOOD/Assemblies/Pins`.

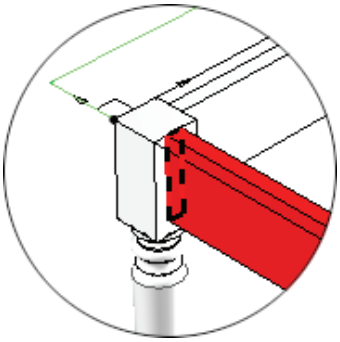
Define the position of the assembly by selecting:

- The support face for the hardware* - Hatched Zone -
- The start face or edge - 1 -
- Select **CENTERED AUTOMATICALLY**
- The term face or edge - 2 -

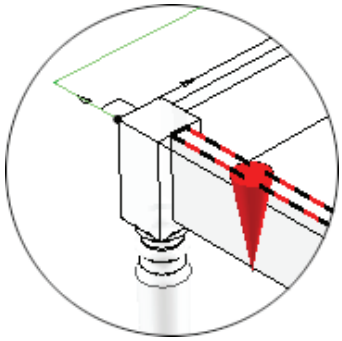



** a filter allows the user to only select faces in contact with other faces.*

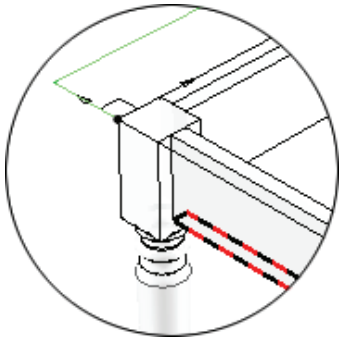
Select the support face, which is the edge of the transom in contact with the foot.
Select the upper face of the transom as the start face.
Click on **CENTERED AUTOMATICALLY**, then select the bottom face of the transom.



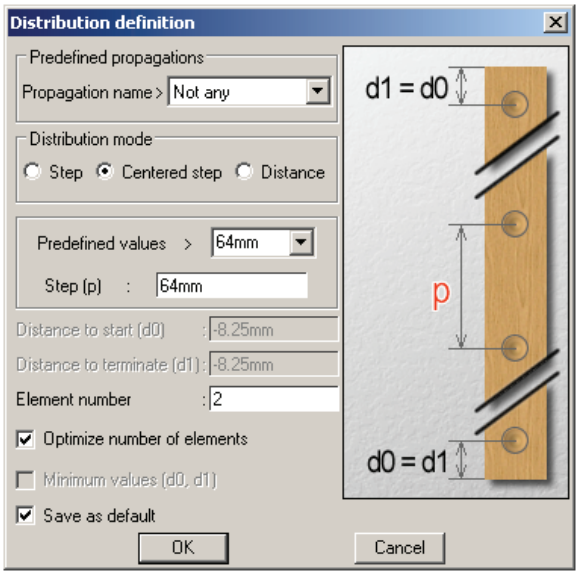
***Support face***



***Start face***



***Term face***

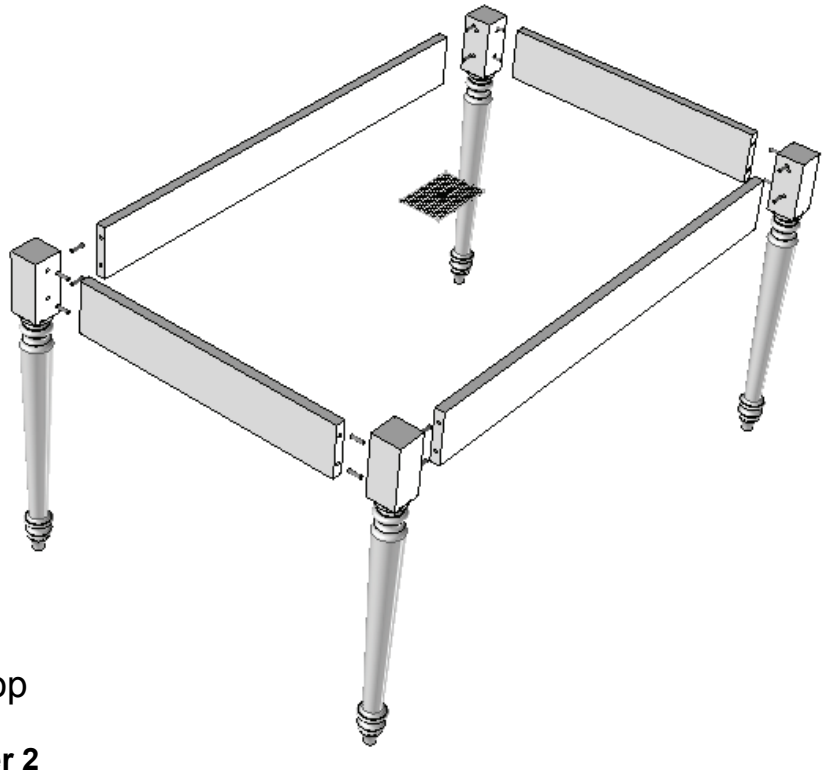



Fill in the **distribution mode** with the following information :

Distribution mode: `Step centered`
Step : `64 mm`
Check Optimize the number of elements.

Select **OK.**

Use the **COPY PROPAGATION** to obtain the same distribution on the other assemblies.



## Create the table top

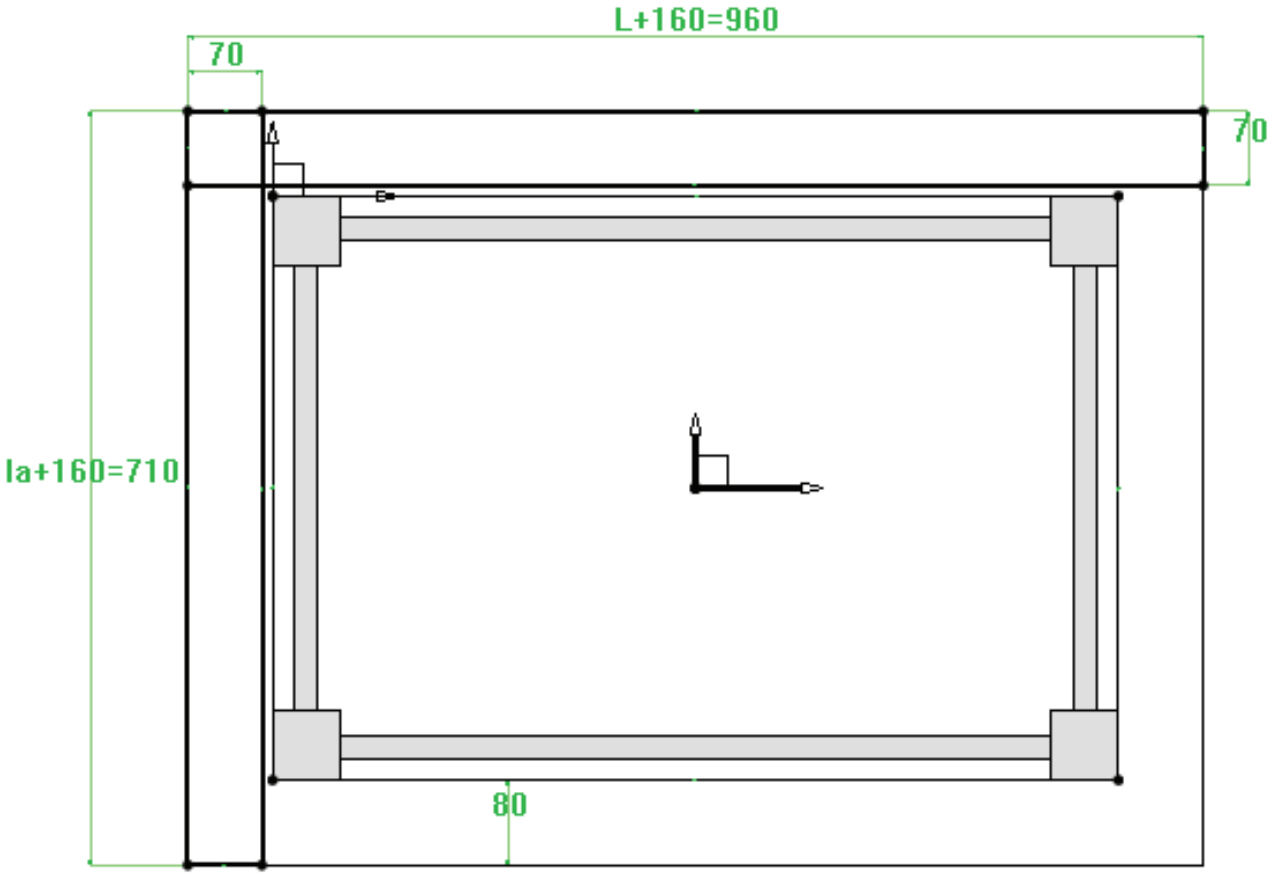
**22** **Activate layer 2**
Make current layer 2

**23** **Create a parallel profile**
*Curve* \ *Parallel*
Make a parallel to the initial rectangle of `80 mm`.

**24** **Create 2 rectangular contours**
*Curve* \ *Contour* \ *Rectangular*
Design 2 rectangular contours corresponding to the sides of the table top.

**25** **Modify parameter**
*Parameter \ Modify*
Change the dimensions of the contours by its corresponding parameters (See below).

Value for the table top sides: `L+160 mm` on the table length
`La+160 mm` on the table width
`70 mm` for the side's width

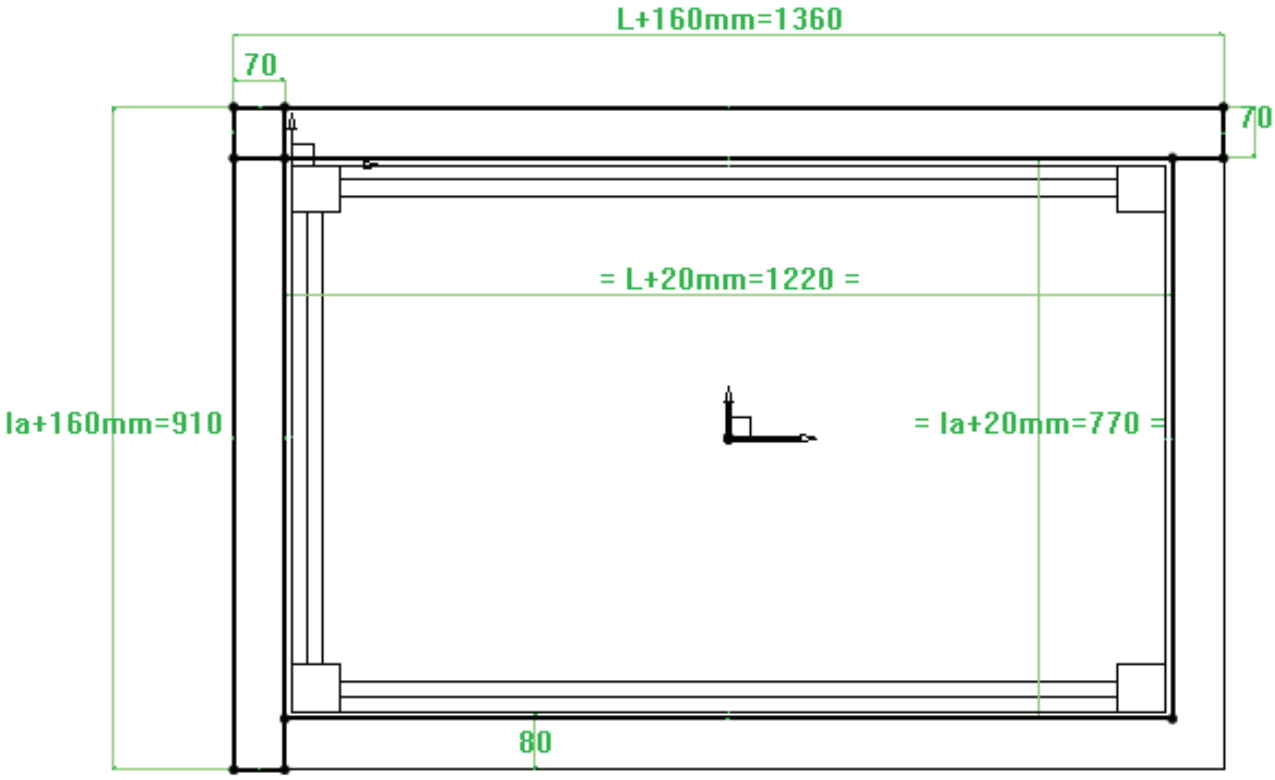



**26** **Create a rectangular contour**
*Curve \ Contour \ Rectangular*
Draw a **RECTANGULAR CONTOUR** in **AUTO DIMENSION.**

**27** **Add the constraints**
Use the **Modify** element function to add symmetry constrain on the **X** axis then on the **Y** axis.

**28** **Modify parameter**
*Parameter \ Modify*
Select the dimension to modify, then take the option **REPLACE** and enter the corresponding parameter:

- `L+20 mm` for the length
- `la+20 mm` for the width
-

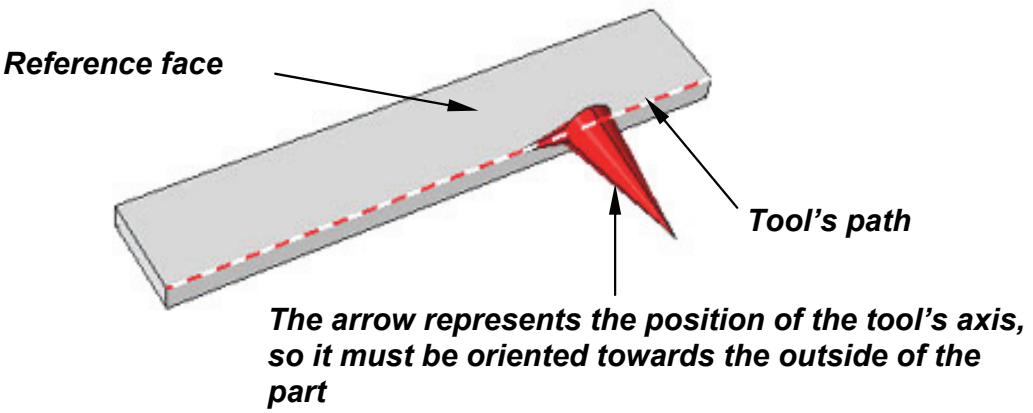



**29** **Extrude the sides of the table top**
*Shape \ Extruded*
Create a extruded shape on **Z+** with a value of 22 mm from the rectangular contour.

## Moulding the top sides

**30** **External molding**
*Wood \ Molding*
Make the molding on one of the top sides by giving

- the reference face (upper face of the shape)
- the tool path (longest edge of the shape)




Select the tool as `Profilage/Type 1`.
Click **OK**.

Use the option **COPY MOULDING** to make the same molding on the second table top side.

## Duplicate the table top sides

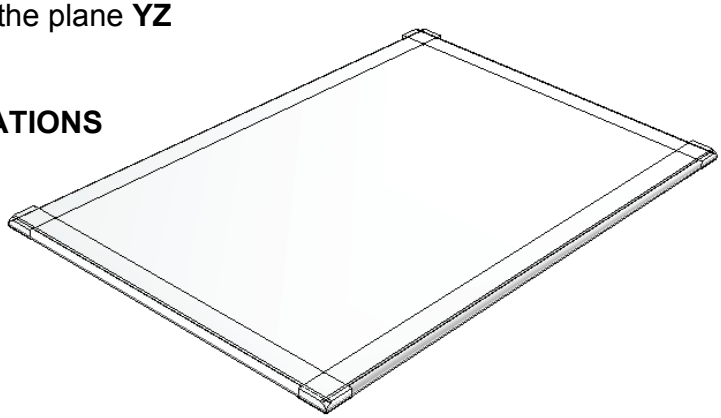
**31** **Duplicate the table top sides** 

*Edit \ Duplicate*

Select the **MIRROR** type of repetition:

- for the length's sides use the plane **XZ**.
- for the width's sides use the plane **YZ**

Choose the option **EXISTING OPERATIONS** and select the parts to duplicate.



## Cutting the sides' corners

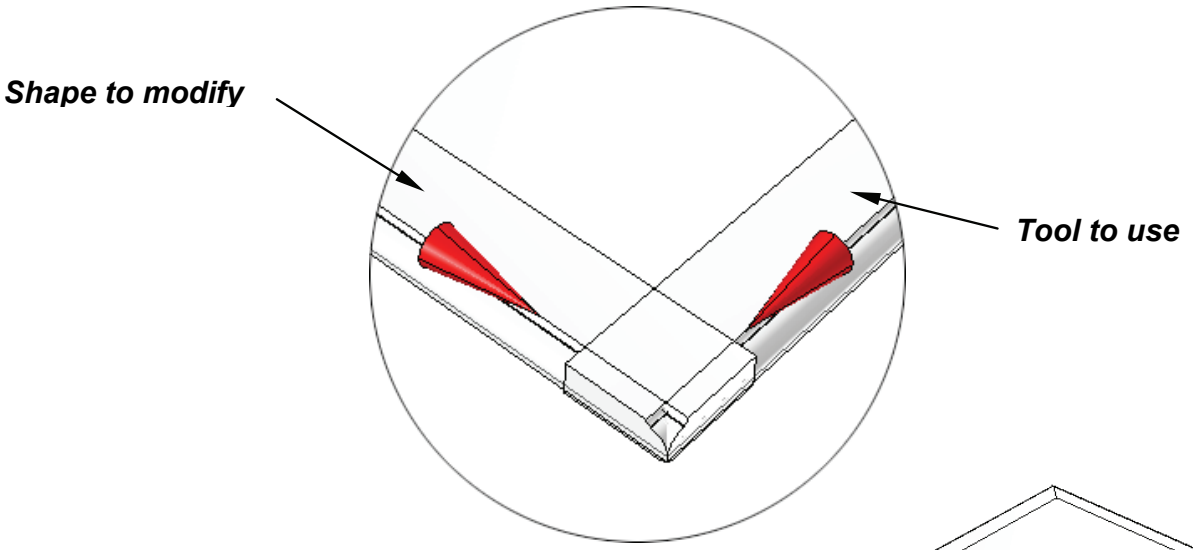
**32** **Miter cut** 

*Wood \ Cutext miter*

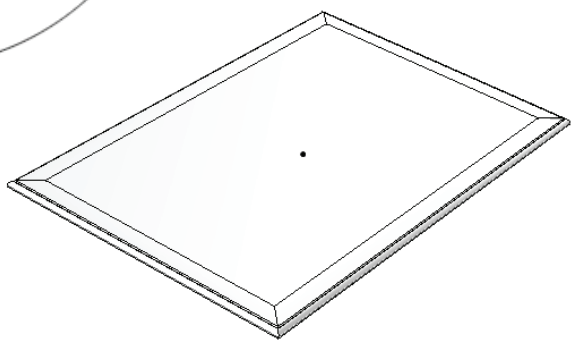
Make a miter union on the sides. Specify:

- the form to modify and the direction*
- the tool and the tool's path

** the requested directions (here with the red arrows) are to determine the bisector where the parts are going to be sawn*




Repeat the operation con the rest of the corners.

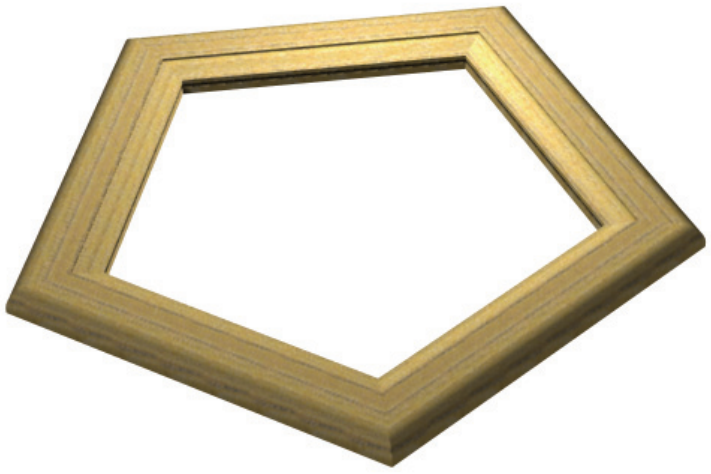


The kitchen table is finished.

# Workshop : Creation of a molding

In this workshop you will learn to :

- Build a parametric contour,
- Define the key points,
- Save a standard,
- Create a catalogue,
- Insert a  molding component,
- Use the molding process.



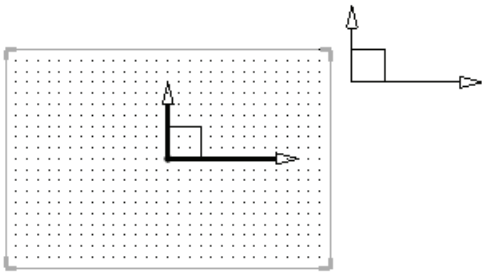
The creation of a molding implies to follow certain conditions described below. If one or more of the conditions are not respected the created tool will not work at all.

# Creation of a molding

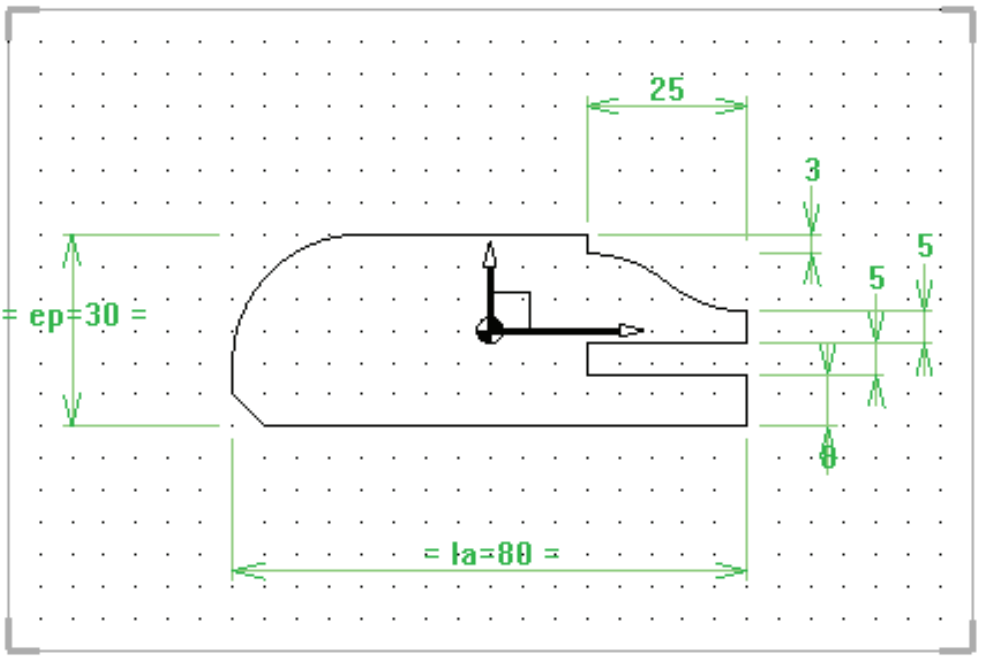
## Create the molding geometry

**1** **Open a new document**
Select from the list a **New** document `Without template`.

**2** **Create a coordinate system**
*Tools* \ *Coordinate system*
Create a XY coordinate system and make it current.

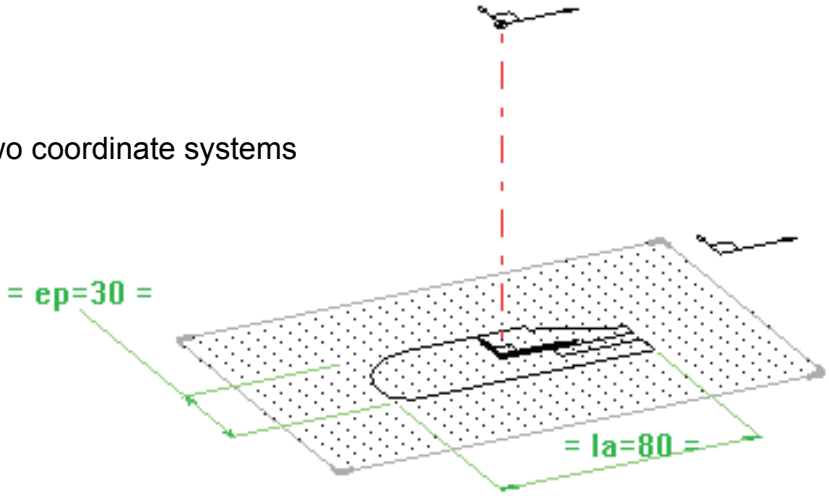


**3** **Create a contour**
*Curve* \ *Contour*
Create the contour corresponding to the section of the molding with its dimensional parameters.

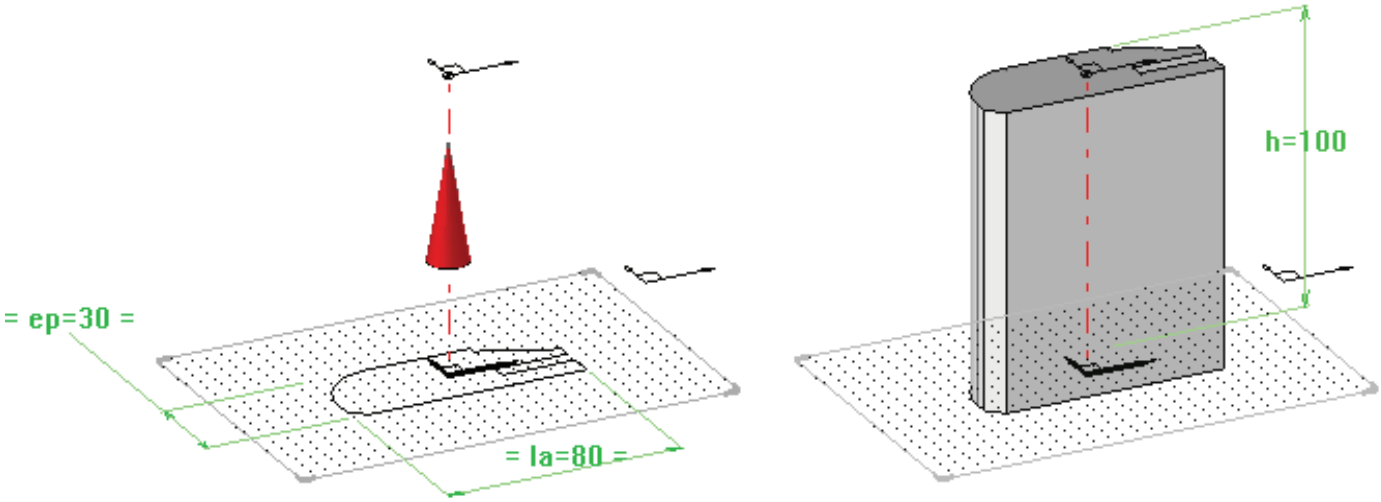



**4** **Create a duplicate coordinate system**
*Tools* \ *Coordinate system* \ *Duplicated coordinate system*
Create a coordinate system from the current one and duplicate it by a translation following **Z+** with a value `h=100mm`.

**5** **Create a line**
*Curve* \ *Line*
Create a line between the two coordinate systems

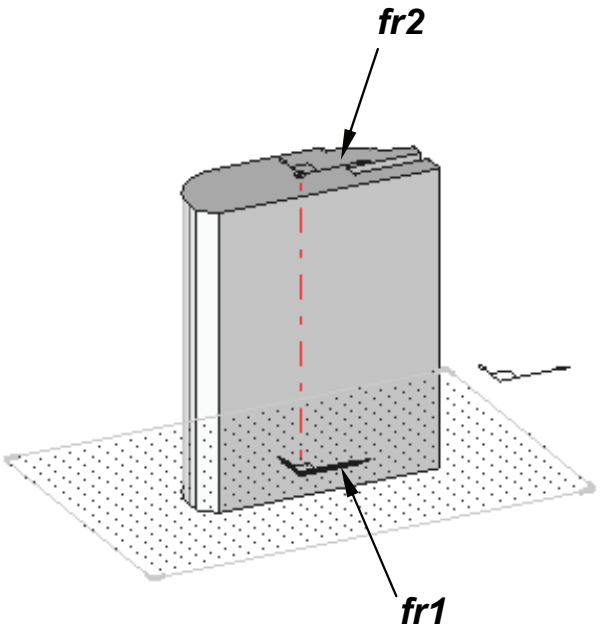



**6** **Extrude the profile**
*Shape* \ *Extruded*
Extrude the section of the profile with an option of direction. Select the line (arrow pointing up) then click on the duplicate coordinate system in order to define the height of the part.




## Definition of the part

**7** **Definition of the part**
*Wood* \ *Define part*
Define the element.




## Declare the elements.

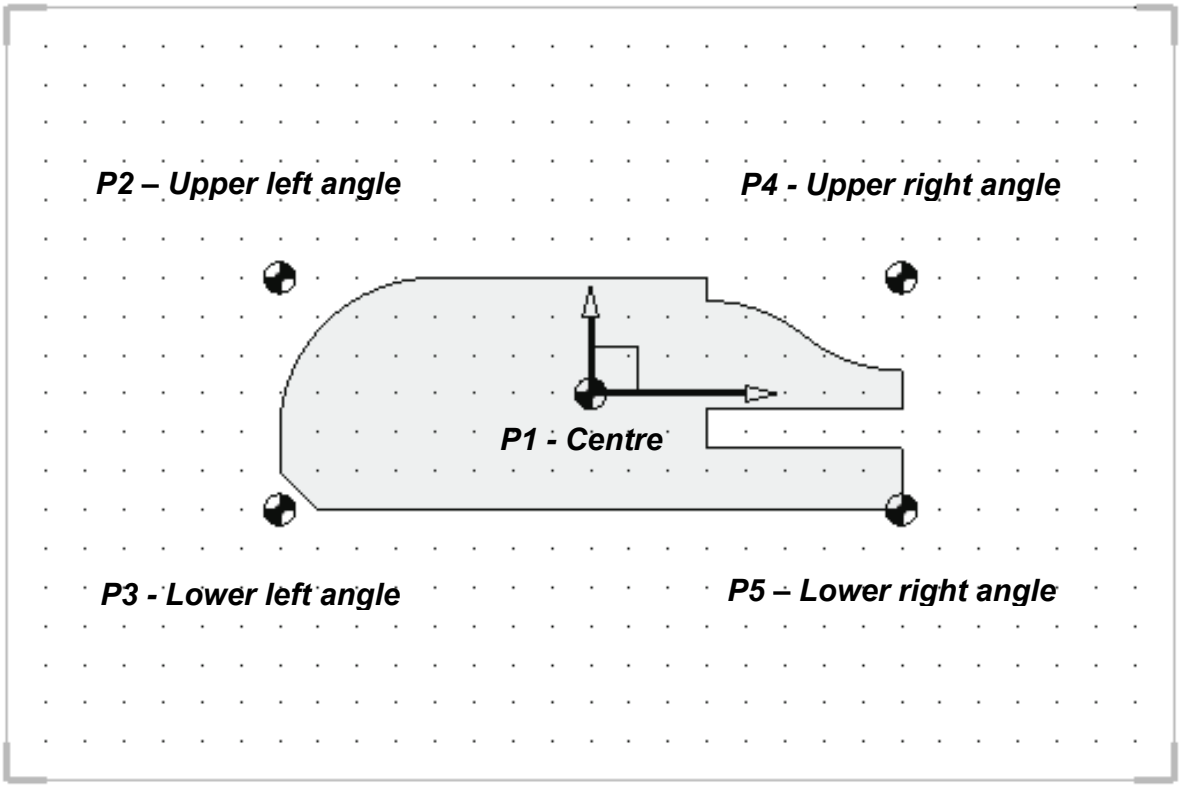
**8** **Name the elements**
*Edit* \ *Name*
The name for the origins is **imposed:**

- lower origin: fr1
- upper origin: fr2.

**9** **Define a driver**
*Assembly \ Define component \ Define drivers*
Define the parameter `h` as a driver.

**10** **Define a key point**
*Assembly \ Define component \ Define key points*
Create the points that will serve to insert and position the component, or recover the points that have served to create the section.
Name each key point as follows :

- name of key point: P1
- designation of key point: Centre



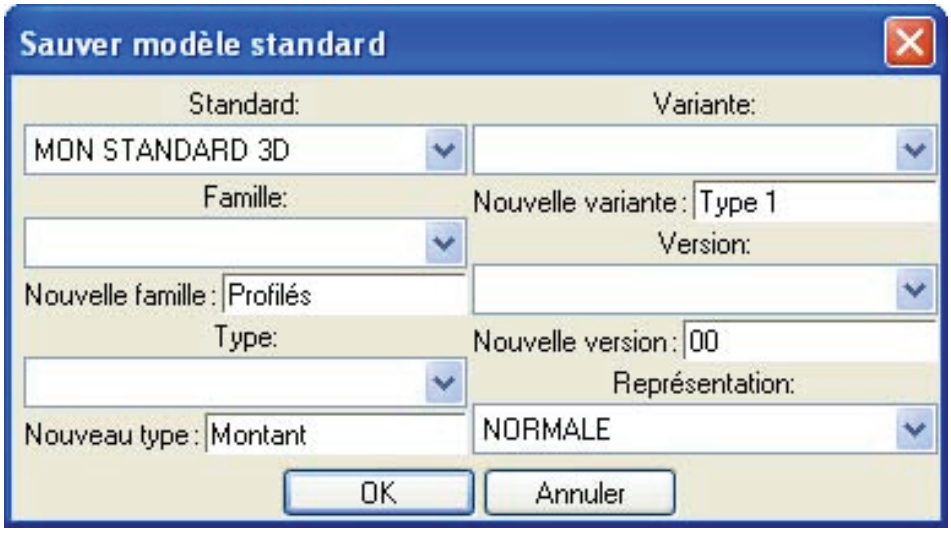

**The points must be attached to the plane of the grille of the coordinate system fr1.**

## Save the template

**11** **Save the template**

*Assembly* \ *Define component* \ *Edit* \ *Save template*
Select **SAVE STANDARD TEMPLATE**

Save it in MY STANDARD 3D \ `Profilés \ Molding \ Type 1.`




If the profile has several sizes or settings it will be possible to define a catalogue.

**12** **Define the extruded component**
*Assembly* \ *Defined component* \ *Define extruded component*
It will be possible to automatically make miter and plane cuts.
Run the function and select the extruded part. The program is going to create the necessary files for the processes y the document contains all the necessary elements for the creation of an extruded component.

**13** **Save**
*File / Save*
Save the file.

## Define a catalogue

**14** **Create a catalogue**
*Assembly* \ *Define component* \ *Edit catalogue header*
When using the function and selecting **All parameters and texts except drives**, a **Excel** table will open.
Enter the values for the parameters following the different codes.
Save the Excel file

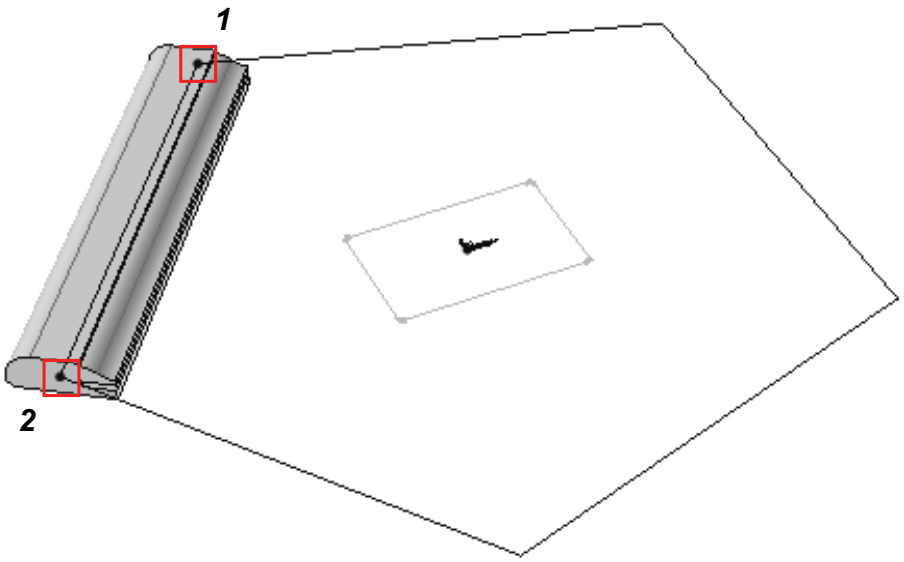
| | A | B | C |
|---|---|---|---|
| 1 | $code | ep | la |
| 2 | 60 x 22 | 22 | 60 |
| 3 | 80 x 30 | 30 | 80 |
| 4 | 100 x 35 | 35 | 100 |

## Insertion of the molding

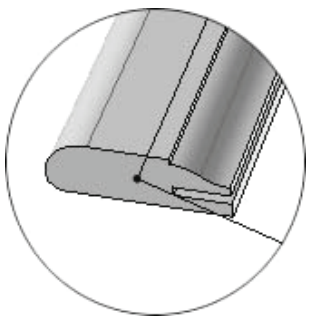
**15** **Open a new document**
Select from the list a **New** document `Without template`

**16** **Create a regular polygon profile**
*Curve* \ *Other curves* \ *Regular polygon*
Create a regular polygon of 5 sides with a diameter of 500 mm.

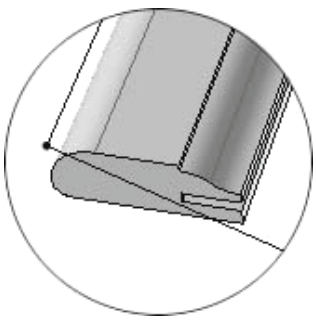
**17** **Insert a standard component**
*Assembly* \ *Include standard*
Select the component `profilé` available on the standard `MY STANDARD 3D /Molding / Type 1.`
Select the 1st passing point then the second one.



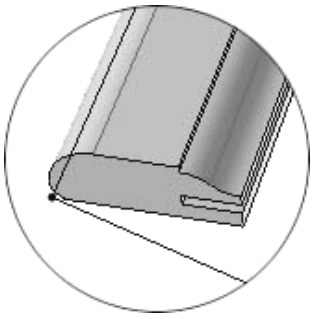

In order to position the profile it will be possible to choose one of the key points previously created.



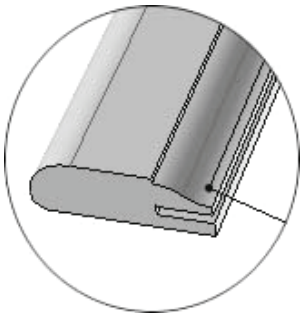
***Key point P1 - Center***



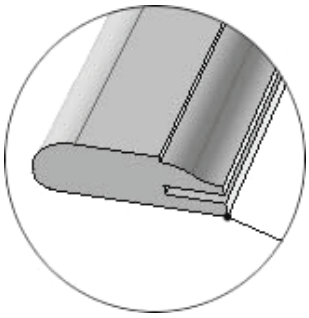
***Key point P2 – Upper left angle***



***Key point P3 – Lower left angle***

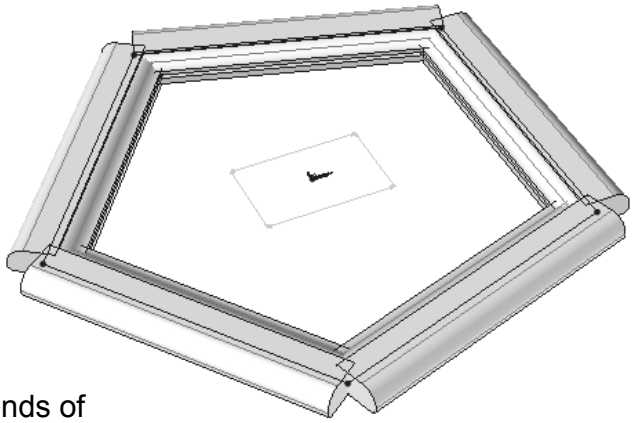


***Key point P4 –***
***Upper right angle***



***Key point P5 –***
***Lower right angle***

Do the same on all the points



The direction of the extruded profile depends of the order of selection of the passing points.

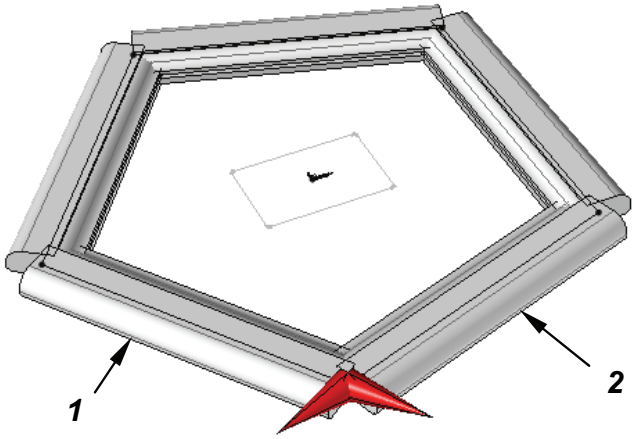
**17** **Use the process**

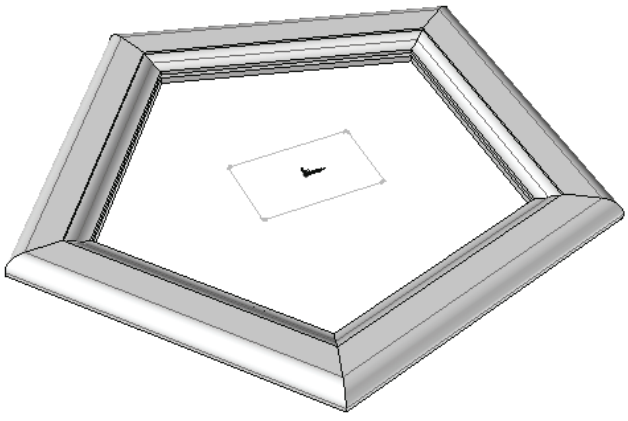
*Assembly \ Use process*

Select one part to activate the processes of the molding. (1).
Select **MITER CUT**, then click the second part that will have the cut (2).

Click **OK**.

Repeat the same procedure to make the cuts on the other parts.



# Workshop - Creation of a wooden bicycle

In this workshop, you will learn to :

- Model simple parts
- Create an assembly using constrained positioning

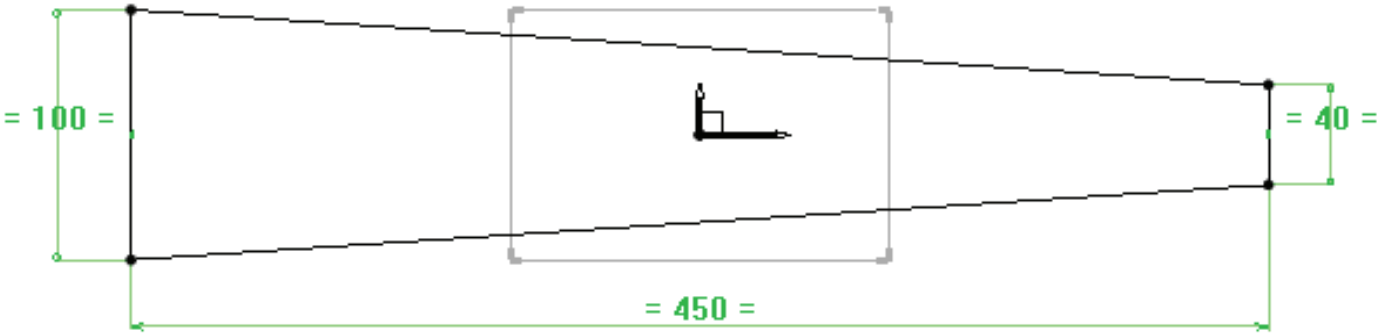
## Create the frame

**1** **Open a new design document**
Choose `Without Template` in the proposed list of **New Document** template models.

**2** **Create a contour**
*Curve* \ *Contour*
Draw the **Contour** shown below.

**3** **Dimension the contour**
*Tools* \ *Dimension*
Dimension the contour as shown below.

**4** **Add constraints**
Use the **Modify element** function to add a symmetry constraint on the **X** axis then on the **Y** axis.

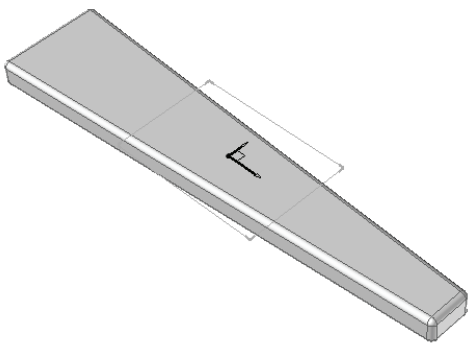



**5** **Modify parameter**
*Parameter* \ *Modify*
Replace the dimensions of the contour with the values shown (See above).

**6** **Fillet**
*Curve* \ *Fillet*
Use the **GLOBAL** mode and create a `5 mm` fillet on the contour.

**7** **Extrude**
*Shape* \ *Extruded*
Extrude the contour a value of `70 mm` in the **Z-** direction

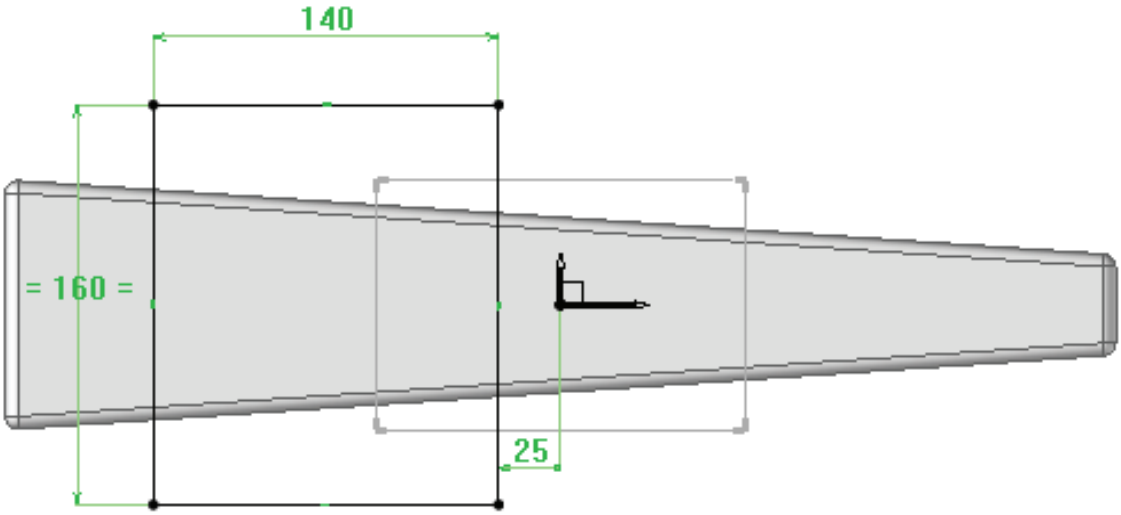
**8** **Fillet**
*Shape* \ *Fillet*
Create `5 mm` fillets on the edges of the extruded shape.



**9** **Activate layer 1**
Make layer 1 current.

**10** **Create a rectangular contour**
*Curve* \ *Contour* \ *Rectangular*
Draw a **RECTANGULAR Contour** using **AUTO DIMENSION.**

**11** **Add constraints**
Use the **Modify element** function to add a symmetry constraint on the **X** axis.

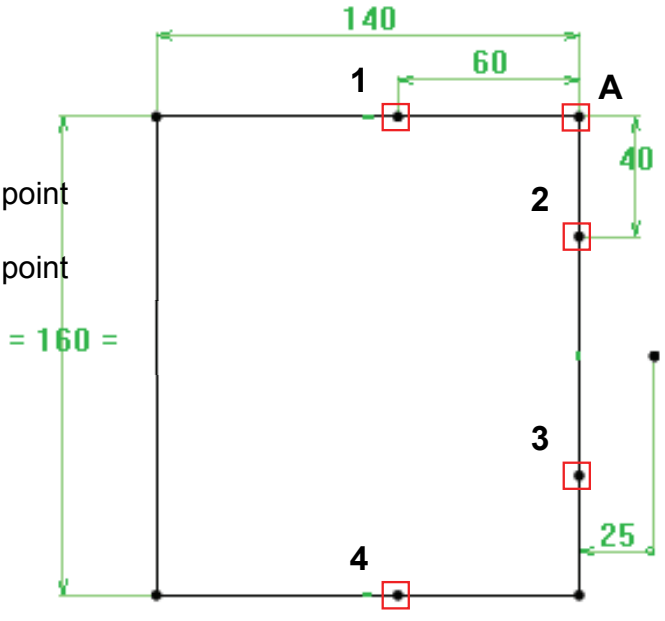



**12** **Position the contour**
*Tools* \ *Dimension*
Position the rectangular contour `25 mm` from the origin of the coordinate system in the **X-** direction.

**13** **Modify parameter**
*Parameter* \ *Modify*
Replace the dimensions of the contour with the values shown (See above).

**14** **Create offset points**
*Tools* \ *Point* \ *Offset point*
Create the following offset points :

- Point **1** is positioned `60 mm` from point **A** in the **X-** direction.
- Point **2** is positioned `40 mm` from point **A** in the **Y-** direction

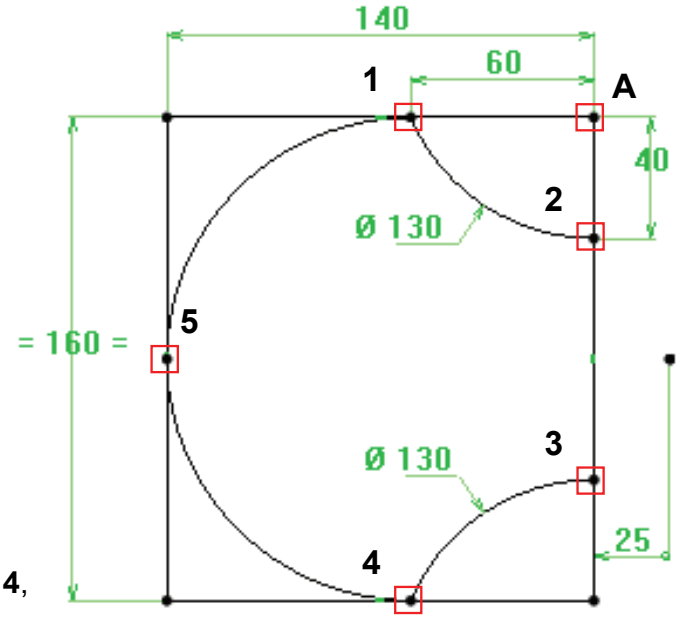



**15** **Create duplicate points**
*Tools* \ *Point* \ *Duplicate point*
Create the duplicate points **3** and **4** using **AXIS SYMMETRY,** the **X** axis and the points **1** and **2**.

**16** **Create a centre point**
*Tools \ Point \ Center key point*
Create the centre point, **5**, on the rectangular contour.

**17** **Create the circular arcs**
*Curve \ Circle*
Create a circular arc of `Ø130 mm` using the **Passing point** mode and the points **1** and **2**, then a second arc using the points **3** and **4**.

Create a third arc using the points **1,5** and **4**, again using the **Passing point** mode.

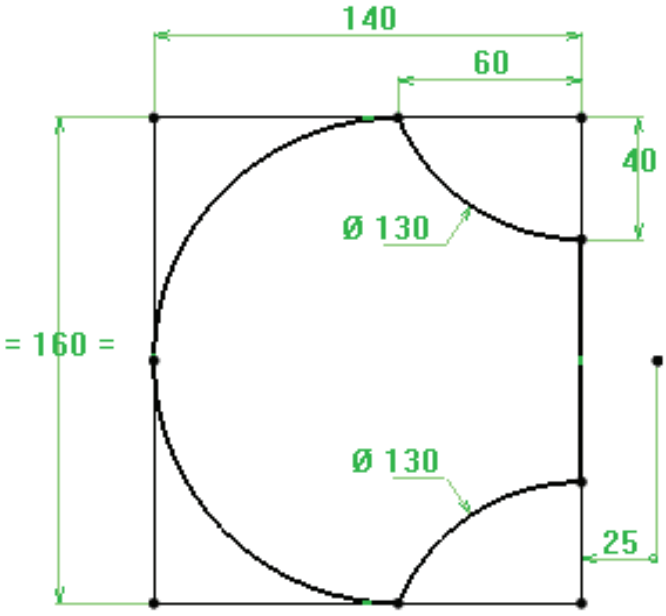


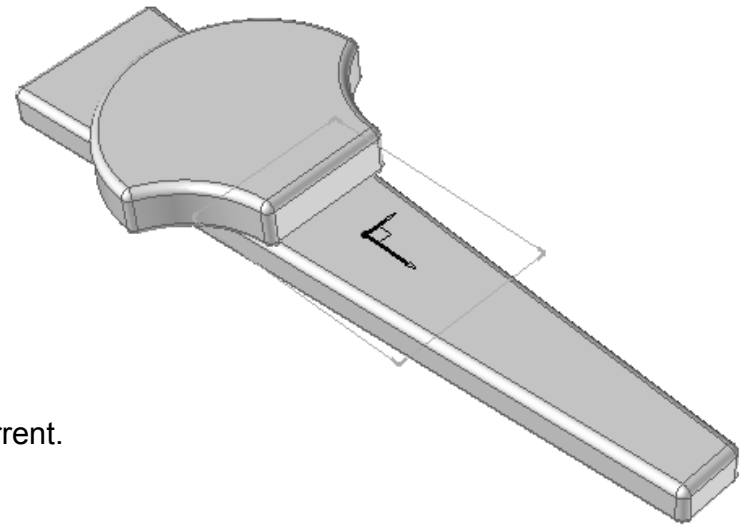

**18** **Create a traced contour**
*Curve \ Contour*
Trace the **Contour** shown opposite.

**19** **Fillet**
*Curve \ Fillet*
Use the **GLOBAL** mode and create a fillet of `5 mm` on the contour.

**20** **Extrude**
*Shape \ Extruded*
Extrude the contour with a value of `30 mm` in the **Z+** direction.

**21** **Fillet**
*Shape \ Fillet*
Create `5 mm` fillets on the edges of the extruded shape.

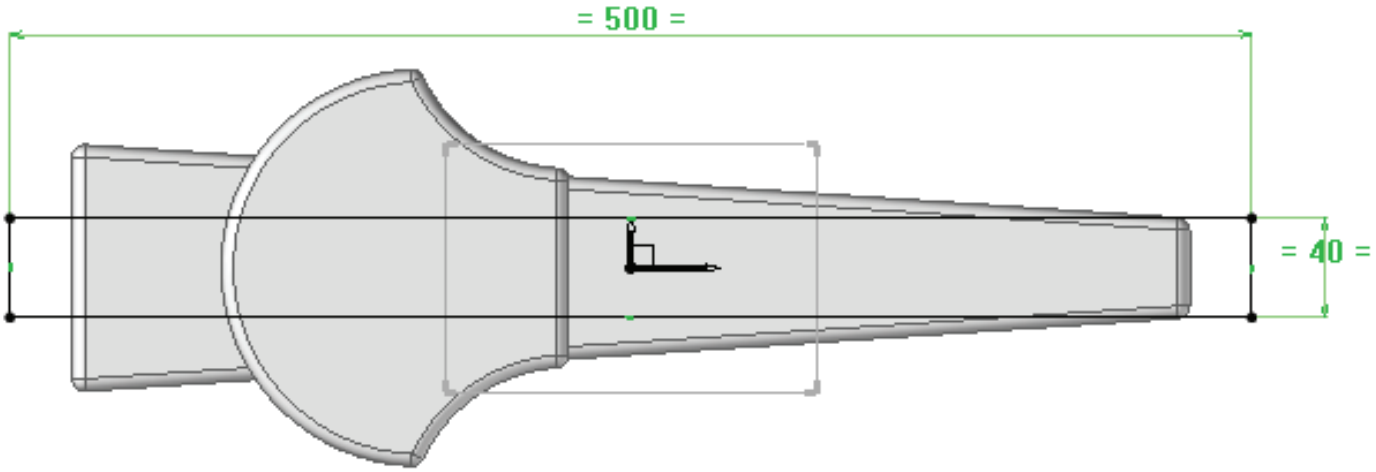


**22** **Activate layer 2**
Make layer 2 current.

**23** **Create a rectangular contour**
*Curve \ Contour \ Rectangular*
Draw a **RECTANGULAR Contour** using **AUTO DIMENSION.**

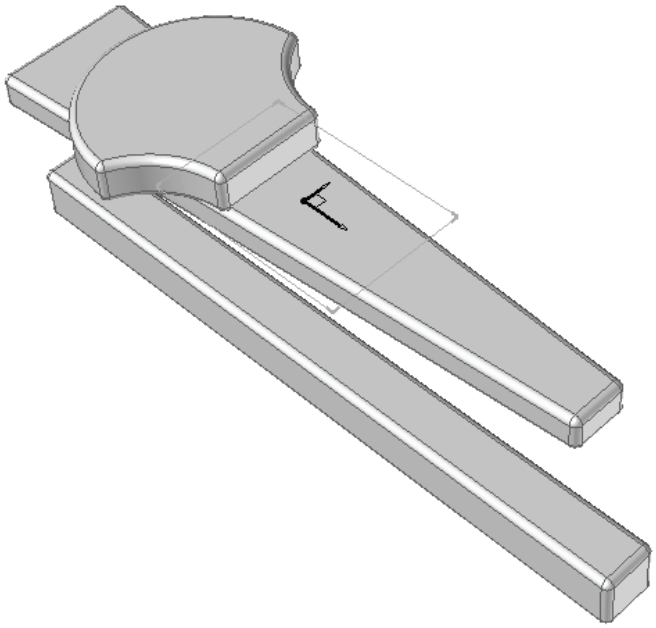
**24** **Add constraints**
Use the **Modify element** function to add a symmetry constraint on the **X** axis then on the **Y** axis.




**25** **Modify parameter**
*Parameter \ Modify*
Replace the dimensions of the contour with the values shown (See above).

**26** **Fillet**
*Curve \ Fillet*
Use the **GLOBAL** mode and create a fillet of 5 mm on the rectangular contour.

**27** **Extrude**
*Shape \ Extruded*
Extrude the contour a value of 40 mm in the **Z-** direction with an offset distance of 120 mm.

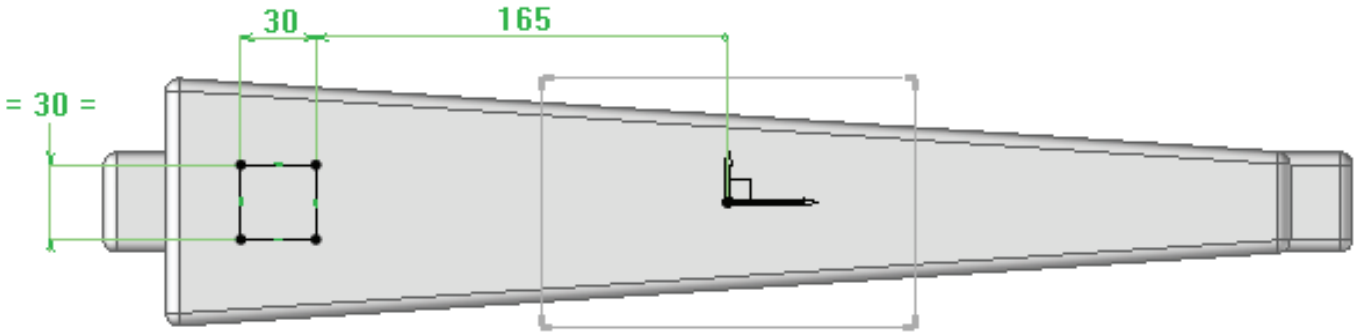


**28** **Fillet**
*Shape \ Fillet*
Create 5 mm fillets on the edges of the extruded shape.

**29** **Activate layer 3**
Make layer 3 current.

**30** **Create a rectangular contour**
*Curve* \ *Contour* \ *Rectangular*
Draw a **RECTANGULAR Contour** using **AUTO DIMENSION.**

**31** **Add constraints**
Use the **Modify element** function to add a symmetry constraint on the **X** axis.

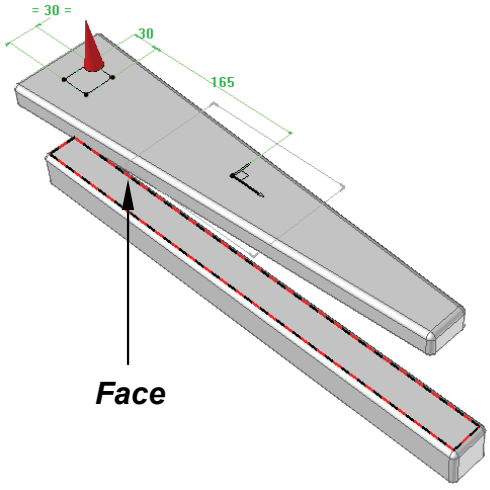



**32** **Position the contour**
*Tools* \ *Dimension*
Position the rectangular contour `165 mm` from the origin of the coordinate system in the **X-** direction.

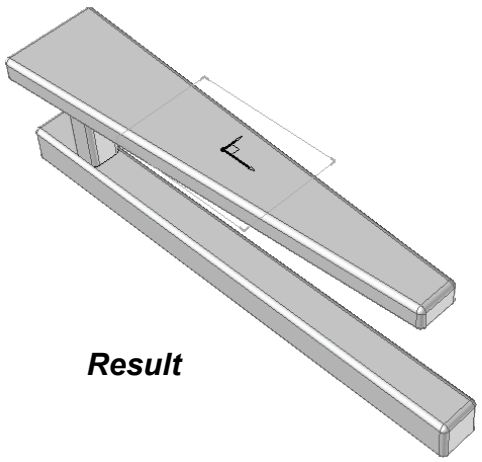
**33** **Modify parameter**
*Parameter* \ *Modify*
Replace the dimensions of the contour with the values shown (See above).

**34** **Fillet**
*Curve* \ *Fillet*
Use the **GLOBAL** mode and create a fillet of `5 mm` on the contour.

**35** **Extrude the separation panel**
*Shape* \ *Extruded*
Extrude the contour in the **Z-** direction with an offset distance of `25 mm`.
Use the mode **TO**, and then click on the top face of the lower part as shown below.




***Face***



***Result***

**36** **Repeat**

*Edit \ Repeat*

Select the part created in the previous step and then using the **SUM OF PROPAGATIONS** repetition type, enter the following characteristics :

- Propagation number 1 : **LINEAR** in the **X+** direction,
  Total distance of `60 mm`,
  Total number of `2`.
- Propagation number 2 : **LINEAR** in the **X+** direction,
  Total distance of `285 mm`,
  Total number of `2`.

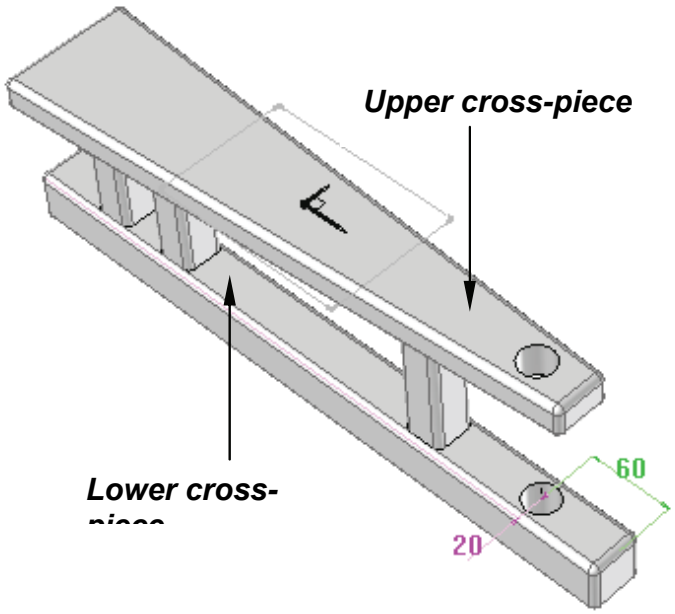
Click on **STOP**.

**37** **Drill**

*Shape \ Drilling*

Create a boring hole of `Ø25 mm` on the lower cross-piece of the frame**.** Centre the hole on the width of the part, and `60 mm` from the front face.

Create an identical drilling on the upper cross-piece. Use the **COORDINATE SYSTEM** mode, and select the coordinate system from the previously created hole.




**38** **Activate layer 4**

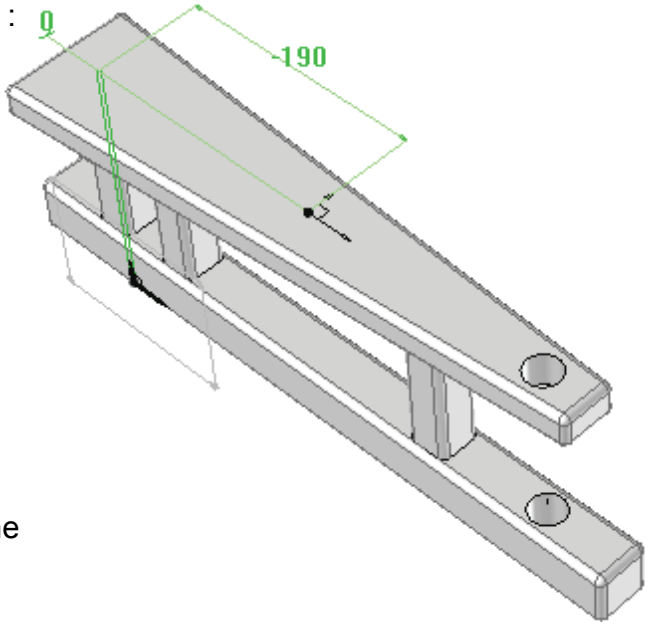
Make layer 4 current.

**39** **Create a Cartesian point**

*Tools \ Point \ Cartesian point*

Create a Cartesian point starting from the **ABSOLUTE COORDINATE SYSTEM**. The coordinates for the point are as follows :

- X coordinate : `-190`
- Y coordinate : `0`
- Z coordinate : `-180`




**40** **Create a coordinate system**

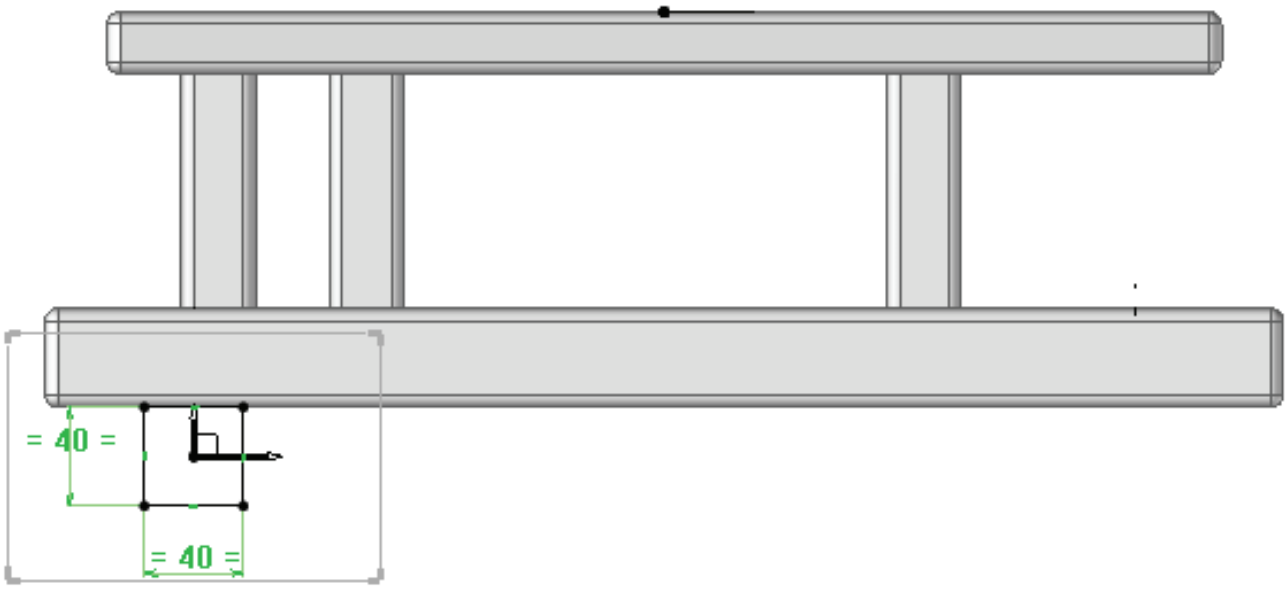
*Tools \ Coordinate system*

Create a coordinate system on 2 axes.
The **THROUGH POINT** is the point created in the previous step.
Select the **X+** for the X direction.
Select **Z+** for the Y direction.

Make the created coordinate system the current coordinate system.

**41** **Create a rectangular contour**
*Curve \ Contour \ Rectangular*
Draw a **RECTANGULAR Contour** using **AUTO DIMENSION.**

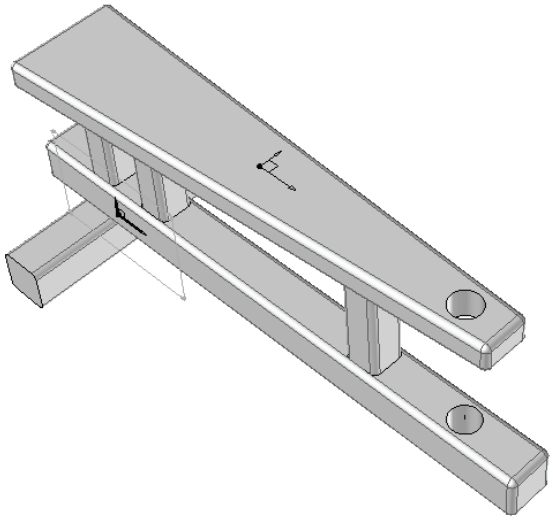
**42** **Add constraints**
Use the **Modify element** function to add a symmetry constraint on the **X** axis then on the **Y** axis.




**43** **Modify parameter**
*Parameter \ Modify*
Replace the dimensions of the contour with the values shown (See above).

**44** **Fillet**
*Curve \ Fillet*
Use the **GLOBAL** mode and create a fillet of 5 mm on the rectangular contour.

**45** **Extrude**
*Shape \ Extruded*
Extrude the contour a value of 200 mm using the **CENTERED** alignment mode.

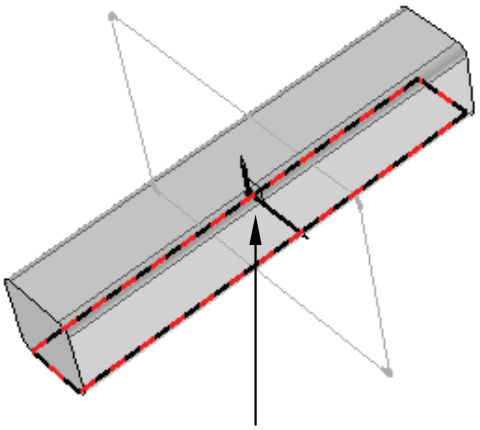


## 46 Slot

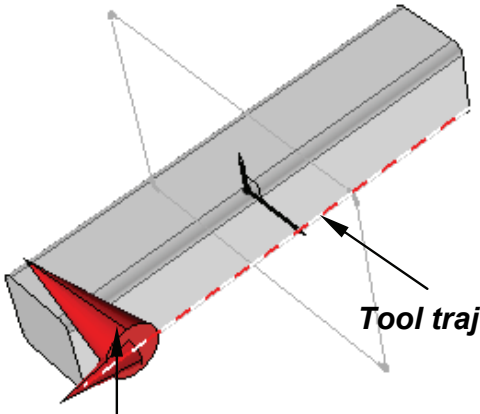
*Wood \ Slot*

Create a slot on the previously created part by defining the following:

- The reference face (lower face of the part)
- The tool trajectory (the longest edge of the part)



***Reference face***



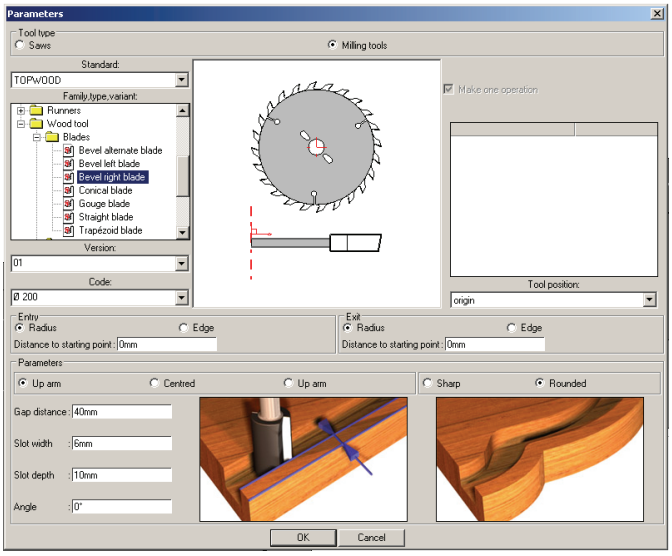
***Tool trajectory***

***The red arrow represents the position of the tool axis, it must point towards the interior of the part.***

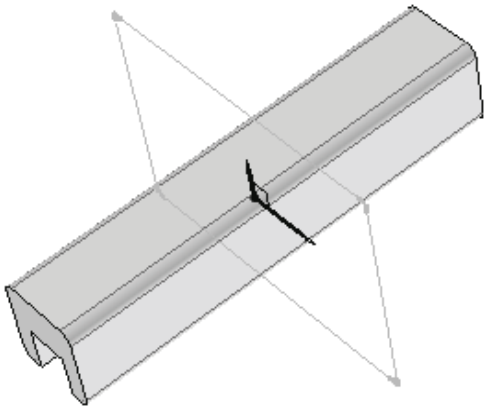
Select the `Bevel right blade`.

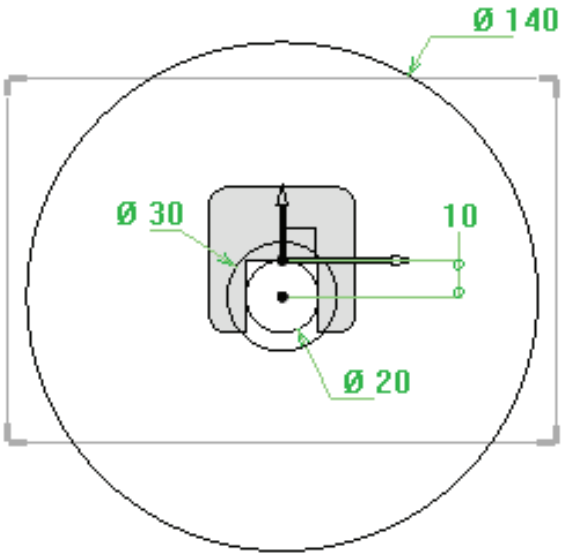
Define the machining values of the slot by creating a parameter for each of them.

- Gap distance = 10 mm
- Slot width = 20 mm
- Slot depth = 20 mm




Select **OK.**



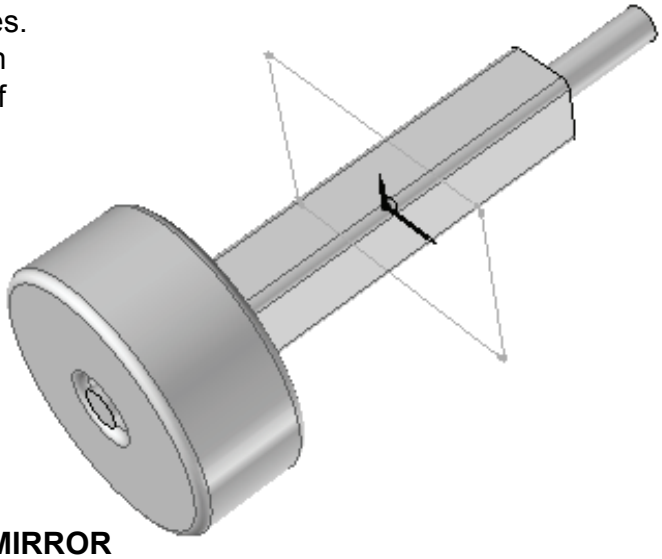



**47** **Activate layer 5**
Make layer 5 current.

**48** **Create circles**
*Curve* \ *Circle*
Draw 3 circles of Ø20 mm, Ø30 mm, Ø140mm. The centers of these should be 10 mm from the coordinate system in the **Y-** direction.

**49** **Extrude**
*Shape* \ *Extruded*
Extrude the Ø20 mm circle a value of 200 mm using the **CENTERED** alignment mode.

Select the Ø30 mm and Ø140 mm circles. Extrude these circles a value of 60 mm in the **Z+** direction with an offset distance of 105 mm.



**50** **Fillet**
*Shape* \ *Fillet*
Create a fillet of 5 mm on the faces of the wheel.

**51** **Repeat the wheels**
*Edit* \ *Repeat*
Select the wheel, then choose **SIMPLE MIRROR** as the repetition type from the list.
Use the **XY** plane as the symmetry plane.

**52** **Definition of the part**
*Wood* \ *Define part*
Define the parts by entering :

- A designation :
- A material
- A type

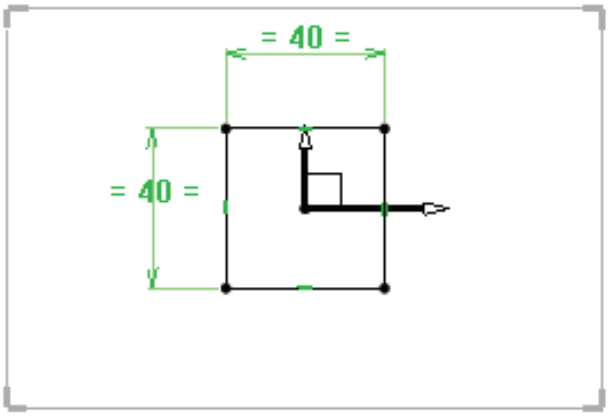
**53** **Save**
*File* \ *Save as*
Save the file as Frame**.top**

## Create the handlebars

**1** **Open a new design document**
Choose `Without Template` in the proposed list of **New Document** template models.

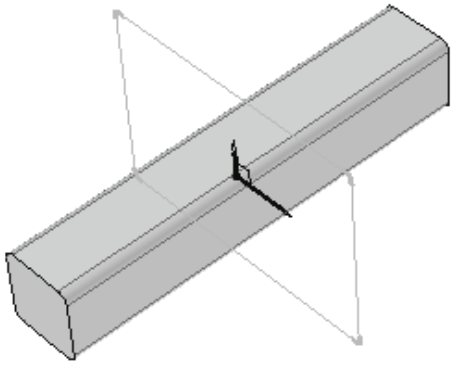
**2** **Create a rectangular contour**
*Curve* \ *Contour* \ *Rectangular*
Draw a **RECTANGULAR Contour** using **AUTO DIMENSION.**

**3** **Add constraints**
Use the **Modify element** function to add a symmetry constraint on the **X** axis then on the **Y** axis.




**4** **Modify parameter**
*Parameter* \ *Modify*
Replace the dimensions of the contour with the values shown (See opposite).

**5** **Fillet**
*Curve* \ *Fillet*
Use the **GLOBAL** mode and create a `5 mm` fillet on the rectangular contour.

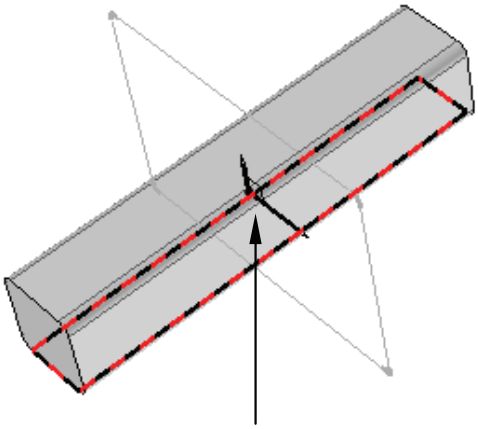


**6** **Extrude**
*Shape* \ *Extruded*
Extrude the contour a value of `200 mm` using the **CENTERED** alignment mode.

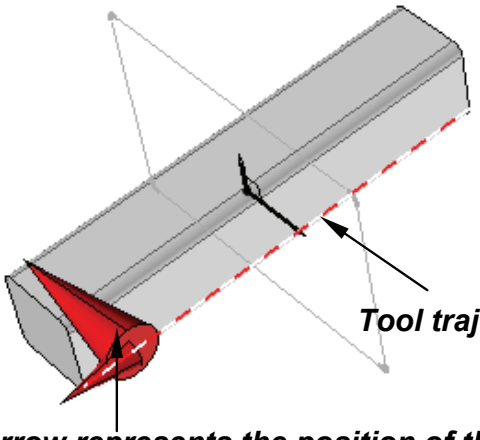
**7** **Slot**
*Wood* \ *Slot*

Create a slot on the previously created part by defining the following:

- The reference face (lower face of the part)
- The tool trajectory (the longest edge of the part)



***Reference face***

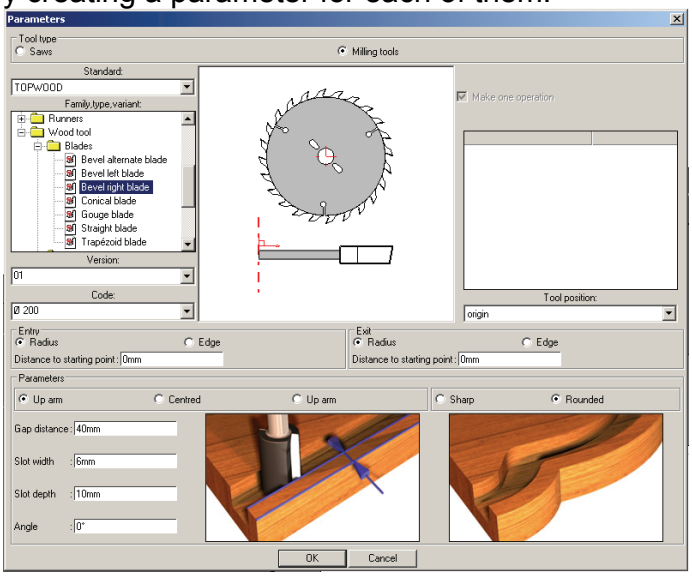


***Tool trajectory***

***The red arrow represents the position of the tool axis, it must point towards the interior of the part.***

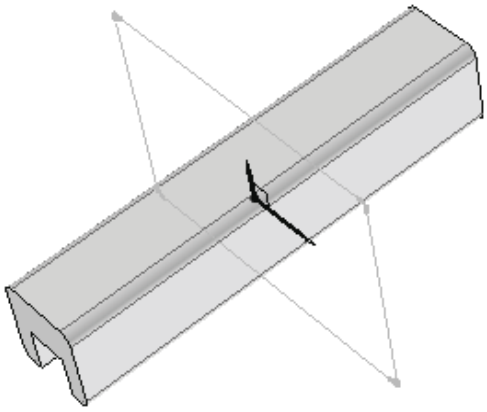
Select the `Bevel right blade`.

Define the machining values of the slot by creating a parameter for each of them.

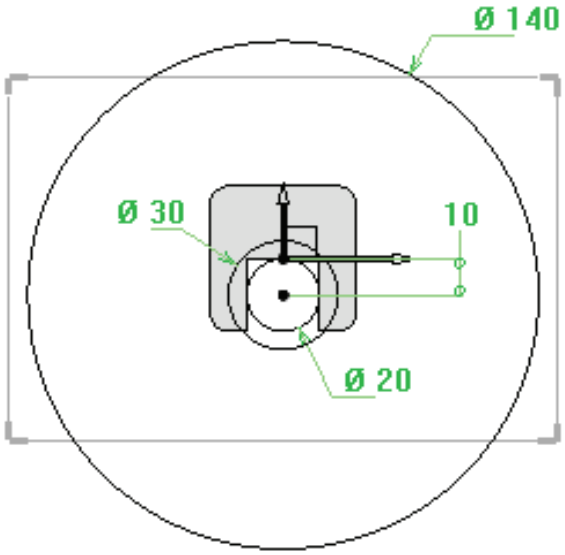



- Gap distance = 10 mm
- Slot width = 20 mm
- Slot depth = 20 mm

Select **OK.**



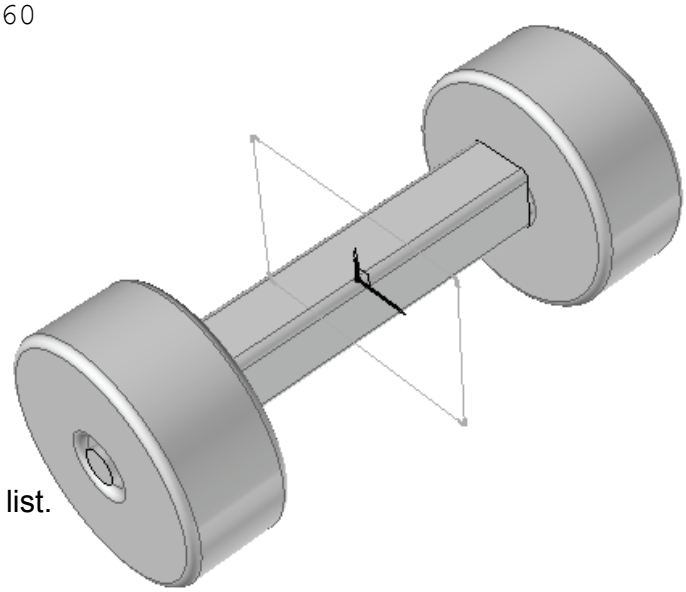
**8** **Activate layer 1**
Make layer 1 current.




**9** **Create circles**
*Curve* \ *Circle*
Draw 3 circles of Ø20 mm, Ø30 mm, Ø140mm. The centers of these should be 10 mm from the coordinate system in the **Y-** direction.

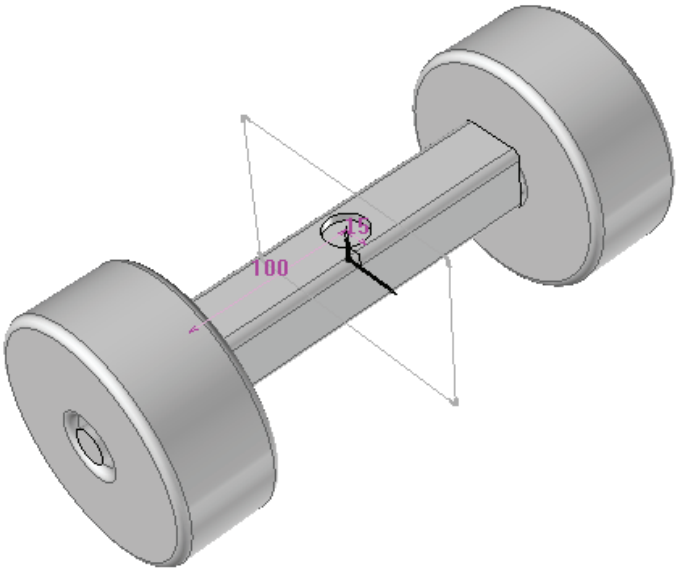
**10** **Extrude**
*Shape* \ *Extruded*
Extrude the Ø20 mm circle a value of 330 mm using the **CENTERED** alignment mode.

Select the Ø30 mm and Ø140 mm circles. Extrude these circles a value of 60 mm in the **Z+** direction with an offset distance of 105 mm.



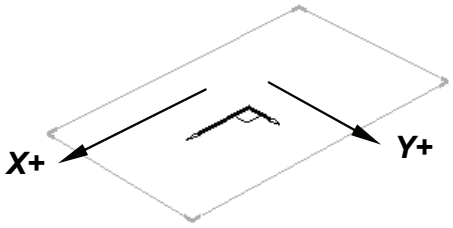
**11** **Fillet**
*Shape* \ *Fillet*
Create a fillet of 5 mm on the faces of the wheel.

**12** **Repeat the wheels**
*Edit* \ *Repeat*
Select the wheel, then choose **SIMPLE MIRROR** as the repetition type from the list. Use the **XY** plane as the symmetry plane.



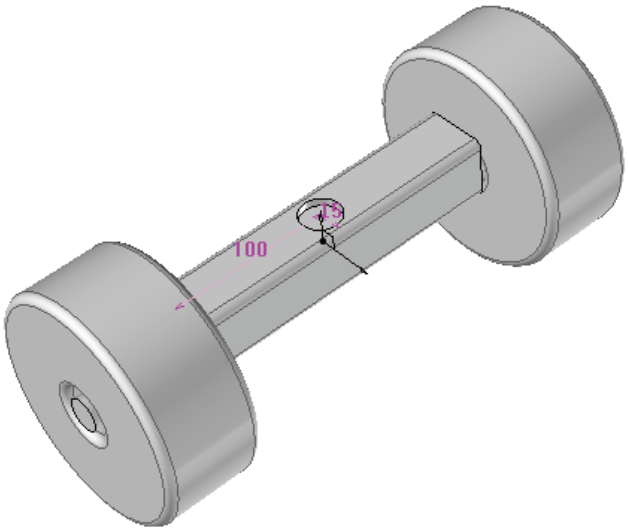

**13** **Drill**
*Shape* \ *Drilling*
Create a boring hole of Ø25 mm, 5 mm deep on the cross-piece**.** The hole should be centered on the width and length of the cross-piece.

**14** **Activate layer 2**
Make layer 2 current.




**15** **Create a duplicate coordinate system**
*Tools \ Coordinate system*
Create a duplicated coordinate system `310 mm` in the **Y+** direction from the **ABSOLUTE COORDINATE SYSTEM**.

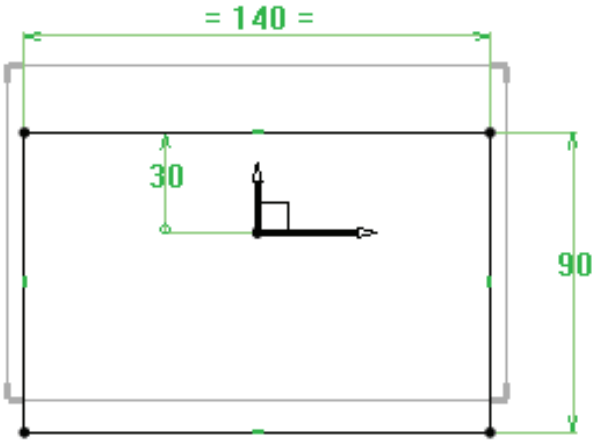
Make the created coordinate system current.




**16** **Add constraints**
Use the **Modify element** function to position the coordinate system as shown opposite.

**17** **Create a rectangular contour**
*Curve \ Contour \ Rectangular*
Draw a **RECTANGULAR Contour** using **AUTO DIMENSION.**

**18** **Add constraints**
Use the **Modify element** function to add a symmetry constraint on the **Y** axis.

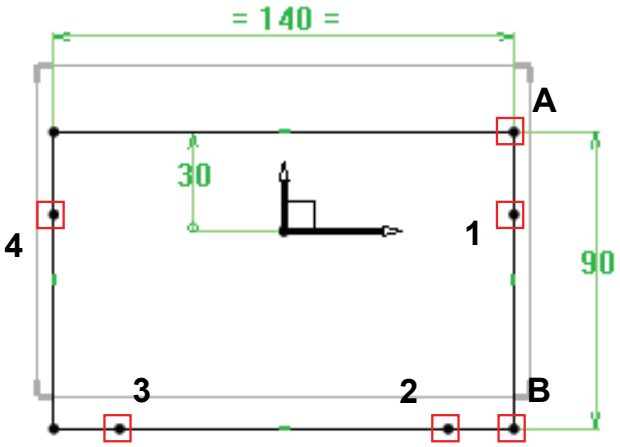


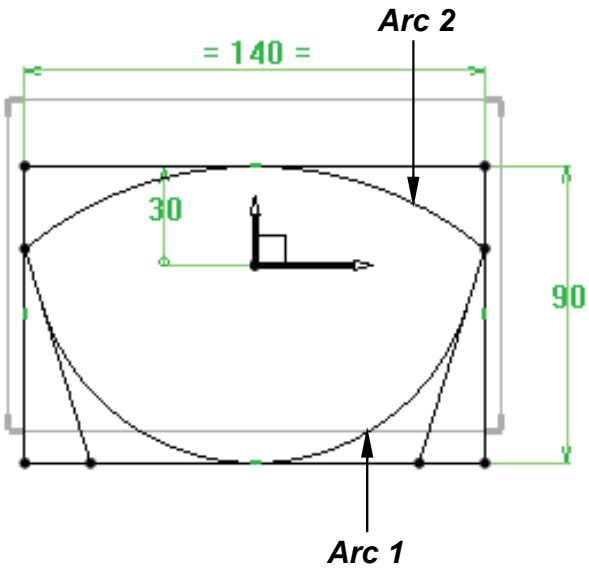

**19** **Position the contour**
*Tools \ Dimension*
Position the rectangular contour `30 mm` from the origin of the absolute coordinate system in the **Y+** direction.

**20** **Modify parameter**
*Parameter \ Modify*
Replace the dimensions of the contour with the values shown (See above).

**21** **Create duplicate points**
*Tools \ Point \ Duplicate point*
Create the following duplicate points :

- Point **1** is positioned `25 mm` from point **A** in the **Y-** direction
- Point **2** is positioned `20 mm` from point **B** in the **X-** direction
- Points **3** and **4** from points **1** and **2** using a simple mirror of the **Y** axis.

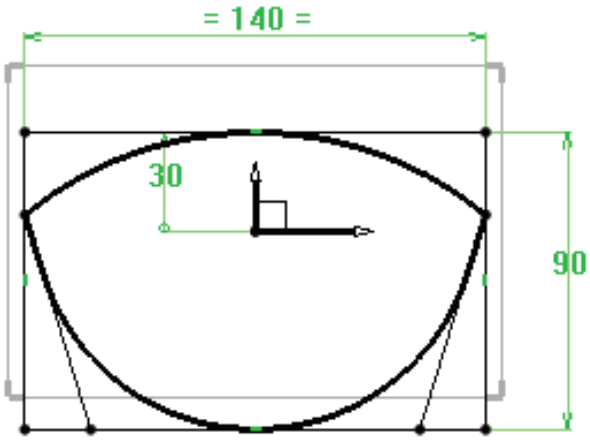





**22** **Create lines**
*Curve \ Line*
Create 2 lines. The first will be from point **1** to point **2** and the second from point **3** to point **4**.

**23** **Create a circle**
*Curve \ Circle*
Create a circular arc using the **Passing point** mode. Click on the two lines created previously and the bottom of the rectangle to create a tangent arc.

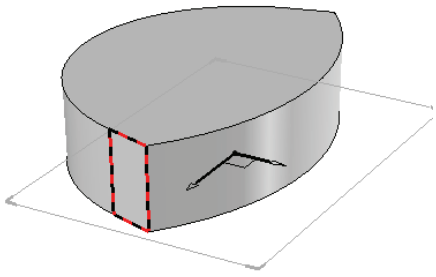
Create a second arc again using the **Passing point** mode, and select points 1 and 4 and the top of the rectangle as shown above.



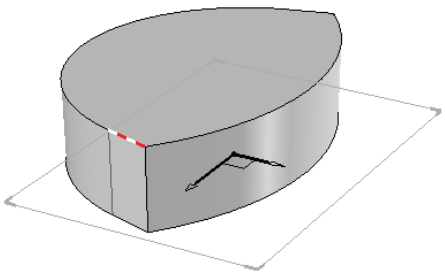

**24** **Create a contour**
*Curve \ Contour*
Trace the **Contour** shown opposite.

**25** **Extrude**
*Shape \ Extruded*
Extrude the contour with a value of `30 mm` in the **Z+** direction.

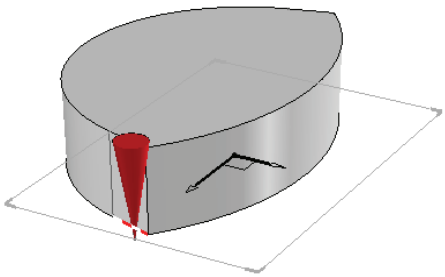
**26** **Drill**
*Shape \ Drilling*
Using the **NON DYNAMIC**, drill a `Ø20 mm` hole `25 mm` deep hole on the part**.** Follow the diagrams below to define the positioning :



***Face to drill***

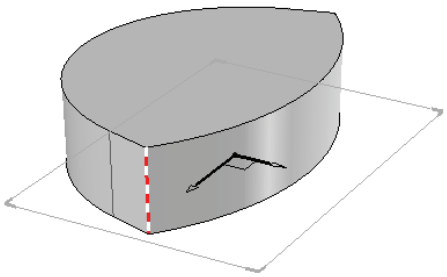


***First alignment face or edge***



***Parallel face or edge***

Enter a distance of 25 mm after selecting the second edge.

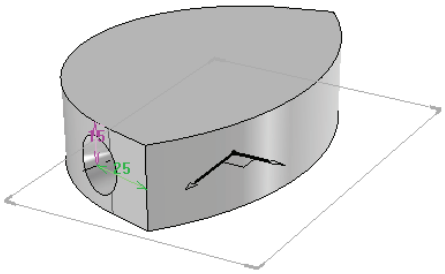


Select a normal drilled hole and enter the characteristics given on the previous page.

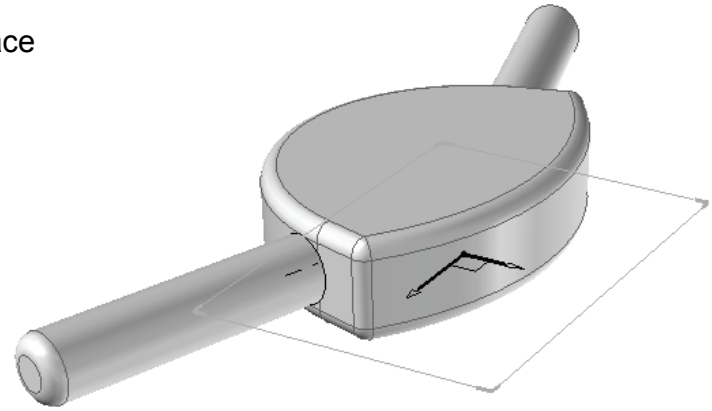
Select **Coordinate system** as the drilling origin.

Click on **PROPAGATE,** then select **SIMPLE MIRROR** and use the **YZ** plane to repeat the operation on the other face of the part.

***Second alignment face or edge***



**27** **Extrude**
*Shape* \ *Extruded*
Using the **FACE** extrusion shape option, create a shape by selecting the bottom face of the drilling.
The height of the handle is 120 mm.

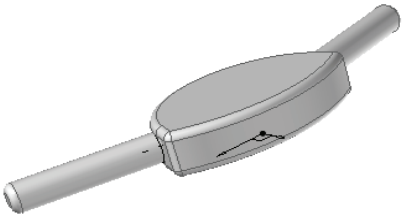


**28** **Fillet**
*Shape* \ *Fillet*
Create a 5 mm fillet on the end of the handle.

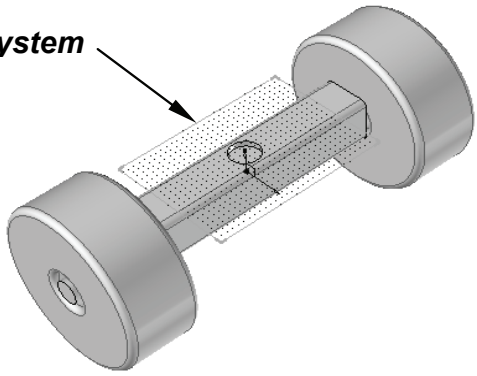
**29** **Repeat the handle**
*Edit* \ *Repeat*
Select the wheel, then choose **SIMPLE MIRROR** as the repetition type from the list.
Use the **YZ** plane as the symmetry plane.

**30** **Fillet**
*Shape* \ *Fillet*
Create a fillet of 5 mm on all the faces of the part.

**31** **Change the coordinate system orientation**
Use the **Modify element** function to modify the properties of the current coordinate system.
Click on **CURRENT COORDINATE SYSTEM**.
Modify the **Orientation** by spinning the **X** axis 15°.
Select **OK**.

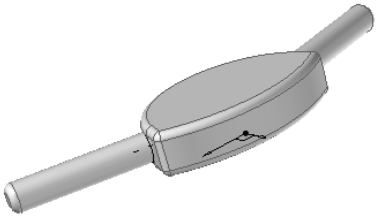


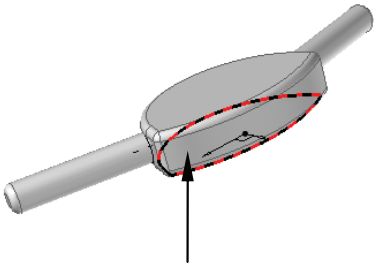
**32** **Change the current coordinate system**
Make the drilling axis of the Ø25 mm, 5 mm deep drilling on the cross-piece the current coordinate system as shown below.

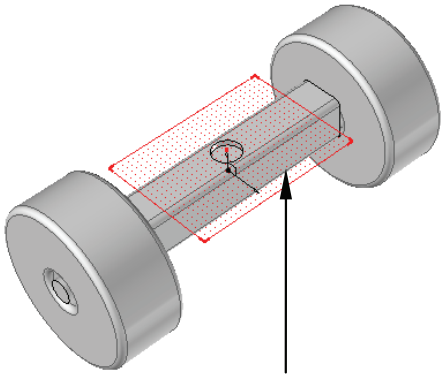


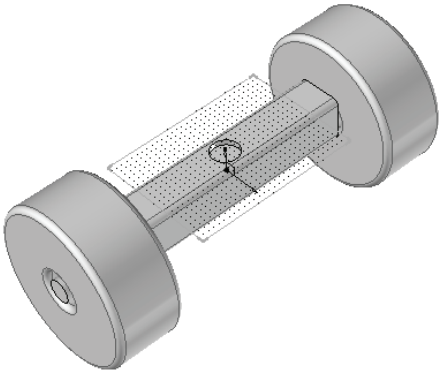

**33** **Drill**
*Shape \ Drilling*
Using the **COORDINATE SYSTEM** mode, drill a Ø25 mm, 5 mm deep hole on the lower face of the handlebar.

In the hole dialogue box ensure that the ***Drilling origin*** is set to the **Face** mode.





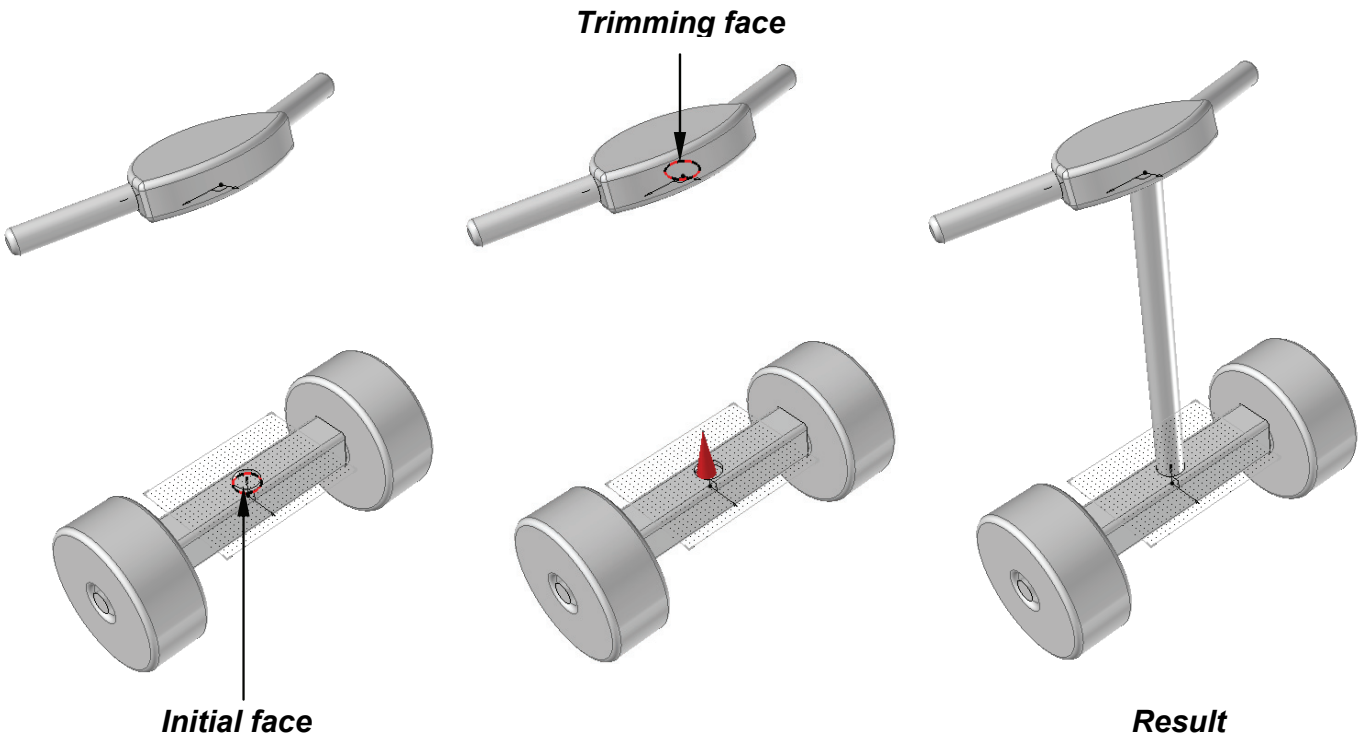
**34** **Activate layer 3**
Make layer 3 current.

**35** **Extrude**

*Shape \ Extruded*

Using the **FACE** extrusion shape option, create a shape by selecting the bottom face of the drilled hole on the lower cross-piece.

Use the mode, **TO**, and click on the bottom face of the drilled hole on the bottom of the handlebar.




**36** **Definition of the parts**

*Wood \ Define part*

Define the parts by entering :

- A designation
- A material
- A type

**37** **Save**

*File \ Save as*

Save the file as **Handlebar.top**

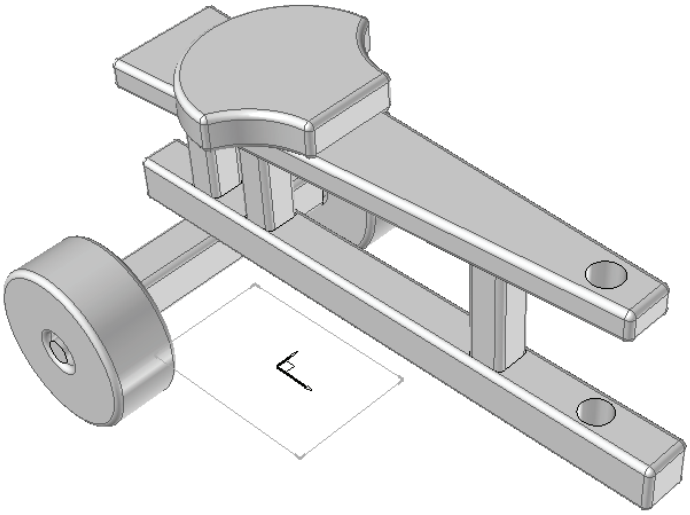
## Assembly of the bicycle

**1** **Open a new design document**
Choose `Without Template` in the proposed list of **New Document** template models.

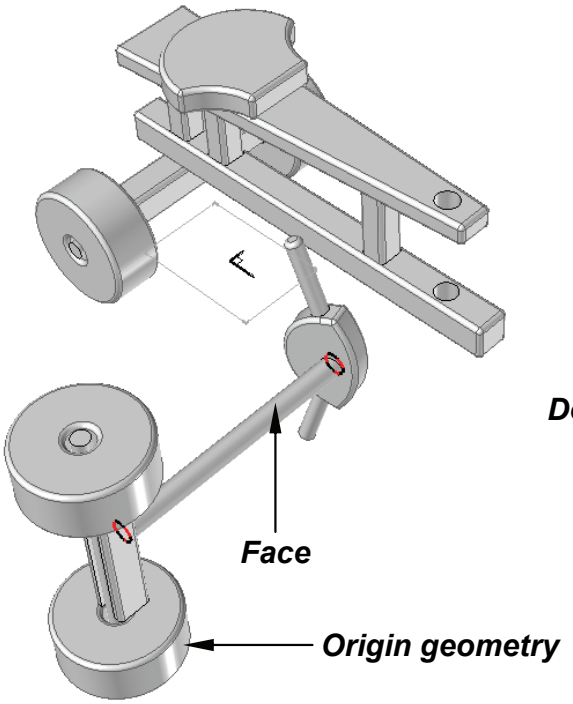
**2** **Insert the base frame**
*Assembly \ Include subassembly - part*
Insert the `Frame` in the document by clicking anywhere on the screen to define its insertion point.

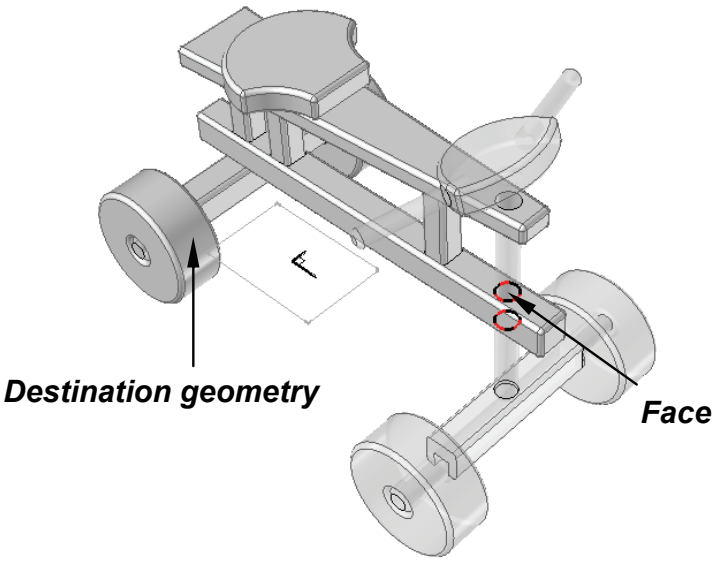
Select **STOP** and **NO PROPAGATION.**

Use the **OTHER COMPONANT** option to our insert the `Handlebar` and click anywhere in the document to define the insertion point.



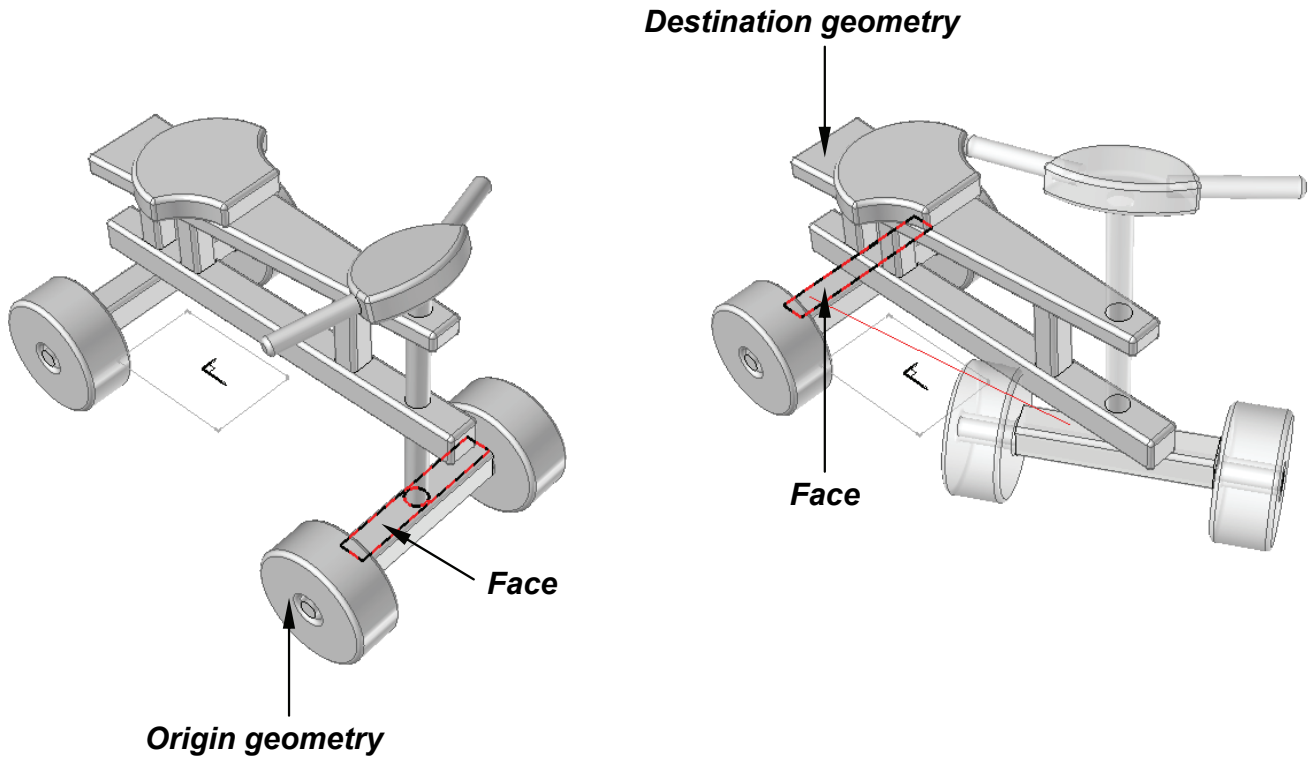
For the origin geometry, click on the cylindrical face of the fork in the handlebar as shown below and for the destination geometry, click on the interior face of the drilled hole on the lower cross-piece of the frame, also shown below.




Select **OK**

For the next origin geometry click on the upper face of the handlebar cross-piece as shown, and for the destination geometry click on the upper face of the lower cross-piece of the frame.



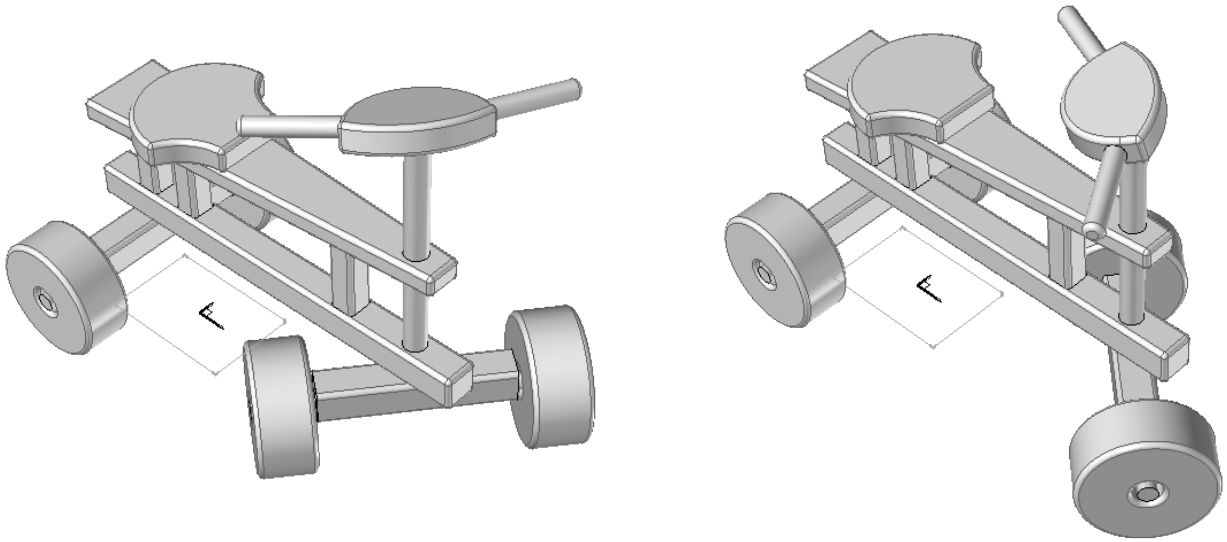

Select **OK**, **STOP** and **NO PROPAGATION.**

**7** **Move the assembly**

*Edit* \ *Move parents*

Select one of the two units that make up the assembly.

You can now move one of the parts by simply moving your mouse.



**8** **Save**

*File* \ *Save as*

Save the file as **Wooden bicycle.top**