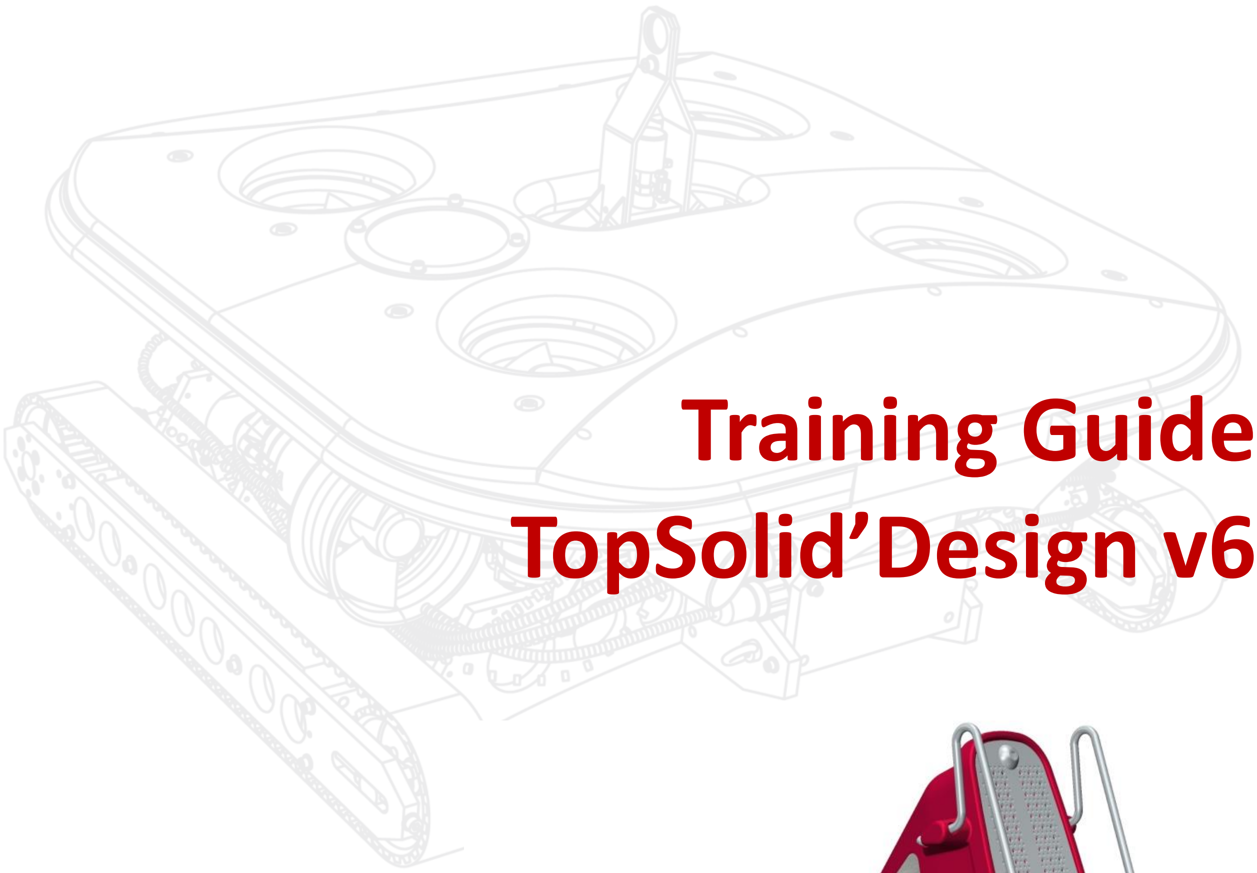

Training Guide
TopSolid'Design v6

MASTER YOUR
MANUFACTURING PROCESS

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Rev.02

Contents

Products organisation

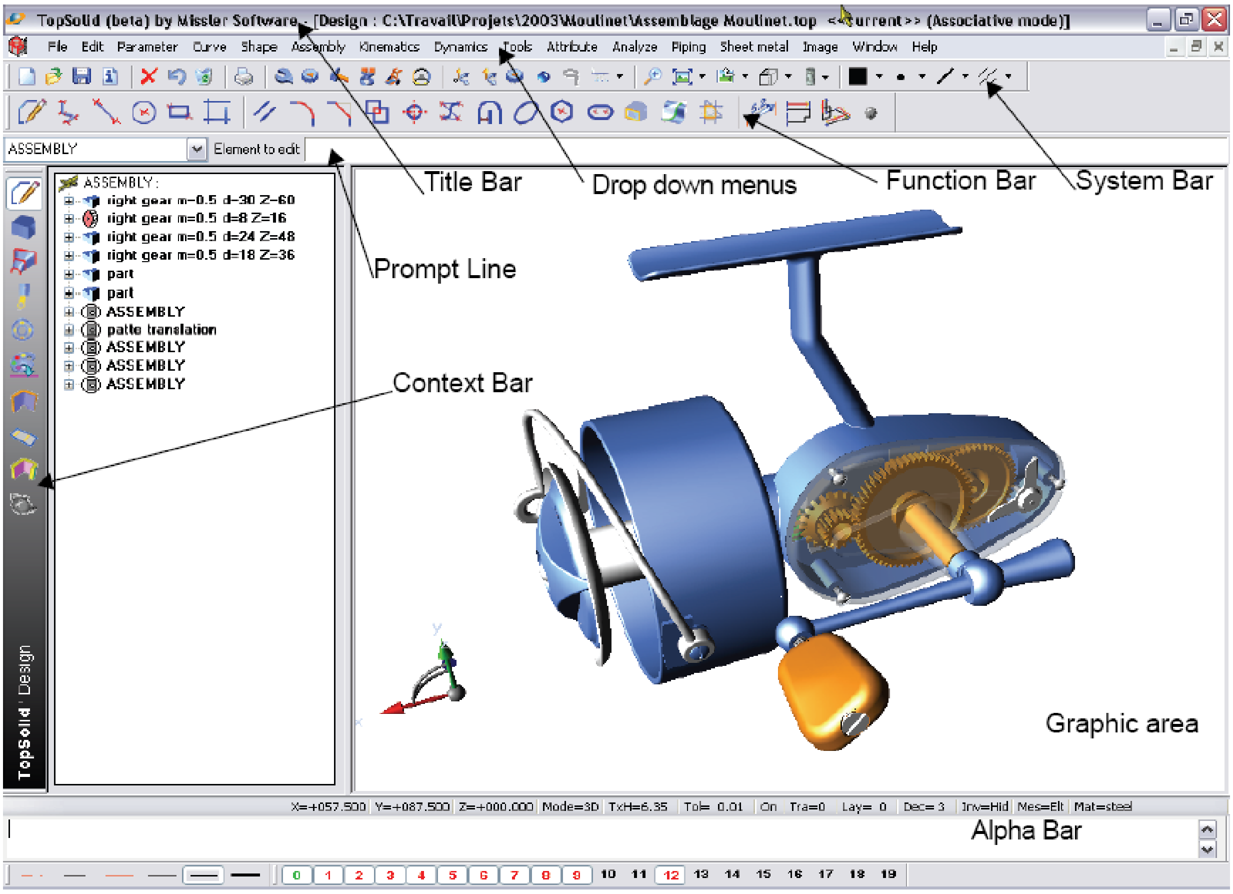
TopSolid is a powerful 3D CAD solid modelling package that runs in the Windows environment. TopSolid is the core product of a family of integrated software solutions developed by Missler Software that offer a global and integrated general mechanical solution for both design and manufacturing.

The training provided here is a pre-requisite for all other modules.

Introduction to TopSolid

I – TopSolid Interface

Below is the TopSolid environment in which you will work, and which is common to all TopSolid modules.



The status bar Provides feedback and allows the user to quickly set layers, colors etc. and set display tolerances and invisible parts. Click directly onto a value to change it.

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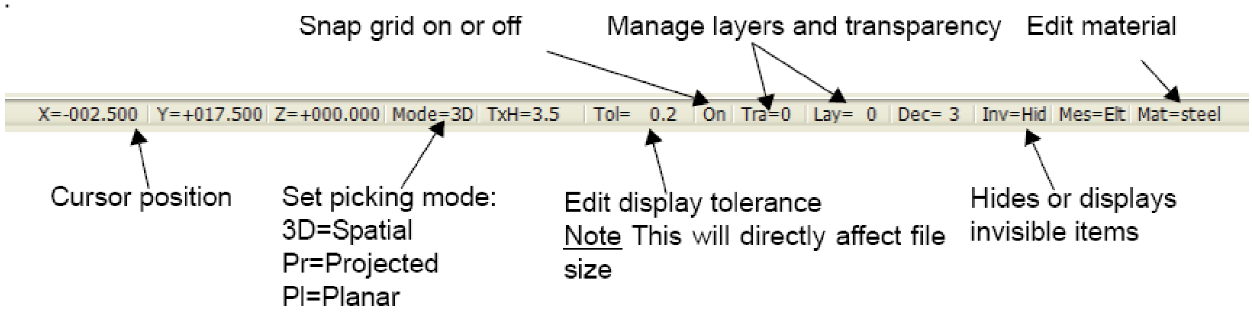
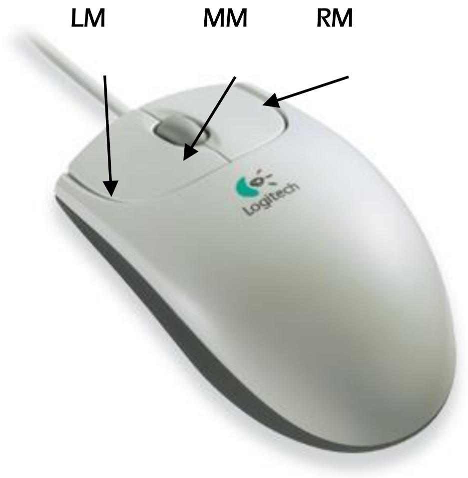


A – Mouse Functions

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

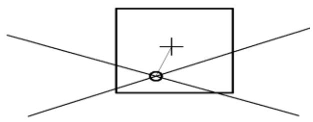


Left Mouse Button (LM):
- Selection of a function or an icon in the menu.
- Picking an element or creating a point.

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

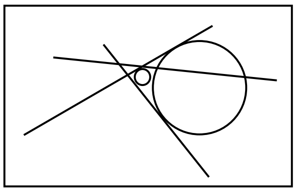
Right Mouse Button (RM):
•The first option of the current command is accepted when the right mouse button is clicked
•Or the context menu of the current command is displayed when held down.

Three further important uses
1. Intersection of 2 Items:
To obtain an intersection of two items left click and hold LM in the graphics area away from the intersection. Then move the mouse over the intersection and release the mouse key; - The size of the square can be changed using the + and – buttons on the keyboard.



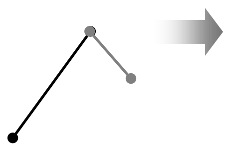
2. Rotative picking of items:
When the mouse is moved over an item, the nearest item is automatically highlighted. If this is not the required item, press and hold down the left mouse button and at the same time click the right mouse RM. Continue right clicking to alternate through the items at the current position. When the correct item is highlighted release the left mouse LM.

How would you select the small circle without zooming?
Use Rotative selection. Click **LM** as close as possible to the circle and while pressed continuously click the RM until the circle is highlighted.

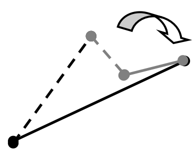


- The middle button has one more distinct property. When drawing lines for instance it will always create a NEW point even if you click onto an existing one.

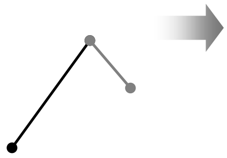
1 – Here we draw 2 separate lines that join at a point. All done with the left button.



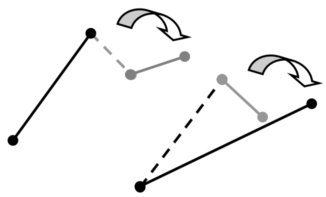
If we move the common point, both lines alter to remain joined.



2 – Here we draw 2 separate lines that join at a point however the second line was drawn with the right button.



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



B – Functions

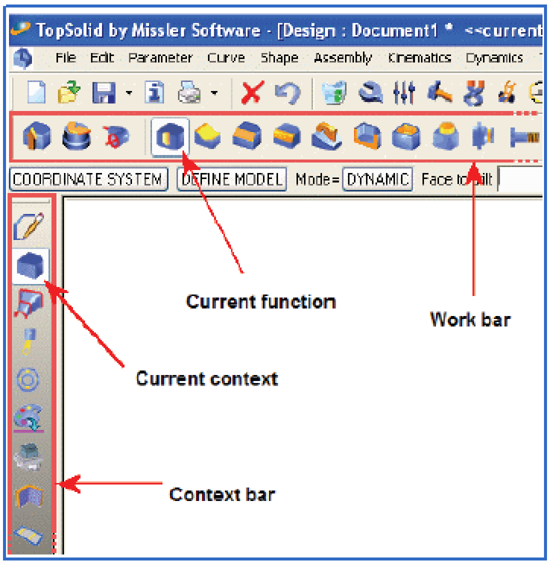
The Icons

There are two types of icons in Topsolid, simple icons and icons with options.


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


Icons with options carry out the selected task when you click the main part with **LM** and open a menu of options when you click the black arrow with **LM** or the main part with **RM**

- 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



Using the Context icon bar

Many of the functions are grouped together "in context" using the context bar (the vertical icons bar located on the left ofthe screen).

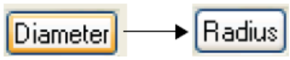
Selecting an icon will change the functions displayed in the work bar (horizontal icons bar located under the system icon bar), and in some cases the menus are also changed when you change the current context

The buttons

Buttons without choice:

– Some button allows you to switch from one option to another simply by clicking on it.

For example, when drawing a circle; by default the « RADIUS » option is activated. If you click the button it switches to « DIAMETER ».

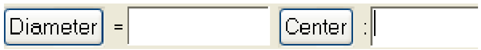


- Some buttons are used to validate an option.
For example in a duplication, clicking the  button validates this option.

Buttons with choice:

- For some options, TopSolid will wait for an input from the user, typed directly from the keyboard.

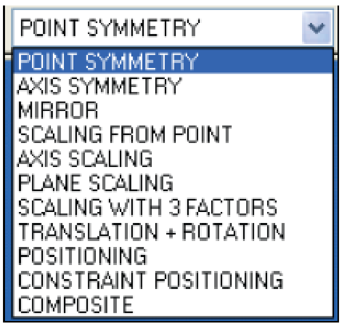
For example in this case the user can type in a diameter and centre location.



Note, this is not mandatory and can (in this case) be ignored by clicking on the screen and dragging the mouse for the diameter.

- Some buttons allow the selection of options from a drop down menu.

For example Transformation has a drop down box showing the other available options as shown here:



The keyboard actions

Using the ***"Shift"*** and ***"Ctrl"*** key along with the mouse can control dynamic movement of the screen.

- Holding down "***Shift***" and the **LM** mouse button causes a Panning of the screen.
- Holding down "***Ctrl***" and the **LM** mouse button causes dynamic rotation.
- Holding down both the "***Ctrl***" **LM** mouse will allow dynamic zoom.

Using the ***up*** and ***down*** arrows allows previous values used in some fields to be recalled

Here is the description of the function keys used in TopSolid:

Shortcut	Function
F1	On line help
F2	Information on points and elements
F3	Dynamic Zoom
F4	Dynamic Pan
F5	Dynamic Rotation about X
F6	Dynamic Rotation about Y
F7	Dynamic Rotation about Z
F8	Cancel Dynamic Rotation
F9	Dynamic Rotation
F10	
F11	Relocation of the floating icon bars
F12	Floating Windows On/Off

User defined shortcuts can be created using the Tools, Options menu..

Typing 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

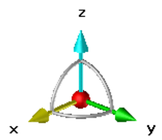
Polar Coordinates:
Defines polar coordinates with, length and angle XY plan, and a height in Z (length; angle, z). The Z height is optional.
Ex: 20; 45, 5

Spherical Coordinates:
Defines spherical coordinates with, length and angle in XY followed by angle in YZ plane of the current coordinated system (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

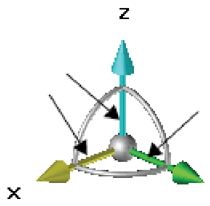
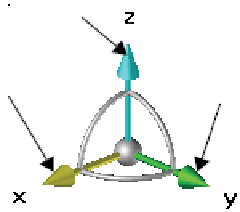
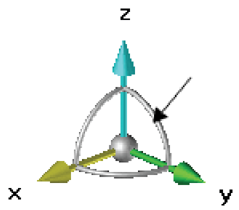
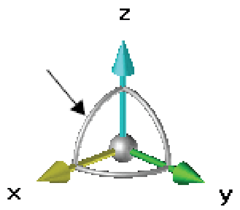
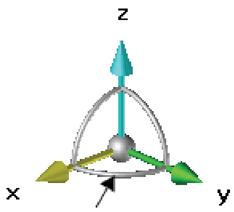
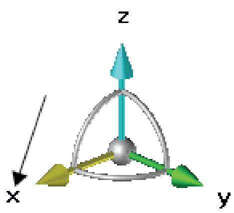
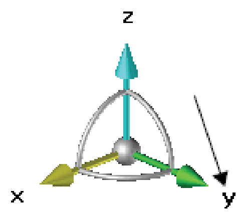
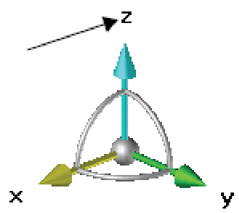
Note: To create a point with the coordinates 0,0,0, you can just type ENTER instead of typing the coordinates.

The Compass



The compass allows the user to modify different views of the screen depending on how it is selected. It allows for panning, rotation and translations.

To modify a view, click the **(LM)** of the mouse on a specific part of the compass, then move the mouse keeping the button pressed.

Transformation	Area(s) to click		
Displacement of the view (Panoramic)			
Spherical rotation			
Rotation along an axis			
	Rotation around X	Rotation around Y	Rotation around Z
Modification of a view orientation			
	View along X (Right)	View along Y (Back)	View along Z (Top)

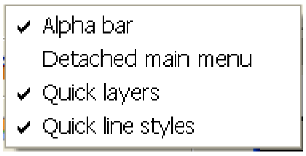
The compass may be positioned anywhere in the view or "hooked" to an element of the design by sliding/moving its centre point. Hooking the compass to an element allows the user:

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

A symbolic coordinate system, representing the current coordinate system, is maintained at the compass default position if the compass is moved (whether hooked at some point or left free somewhere in the view).

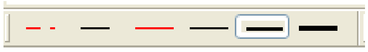
After moving the compass, it is possible to move it back to its place at the bottom left of the screen, and vice versa, by double-clicking. The compass may be temporarily hidden via its context-sensitive menu. Use the context-sensitive menu of the default coordinate system to make it reappear. When the compass does not appear in the view (i.e. if it has been hooked to an element that has passed outside the view), it may be retrieved by clicking on the default coordinate system.

Quick Menu Bar:

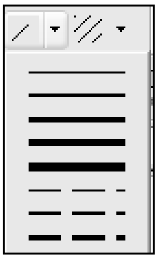


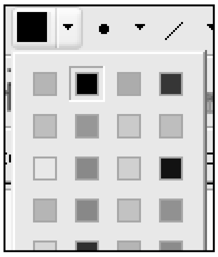
Right clicking **(RM) over** the main Title Bar brings up the "Quick Menu" Highlight "Alpha bar", "Quick Layers" and "Quick Line Styles".

When you highlight "Quick Line Styles" the following menu appears at the bottom left corner of the screen.

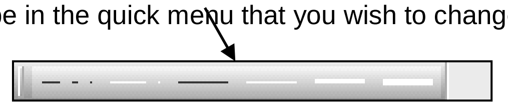


To select a line type for your design simply click on the required style on the line style menu with the left button of the mouse. If you wish to redefine a line style (colour, thickness, type) in the quick menu, first change the colour and line-type you require from the system bar. Then right click

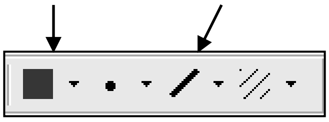




over the line-type in the quick menu that you wish to change

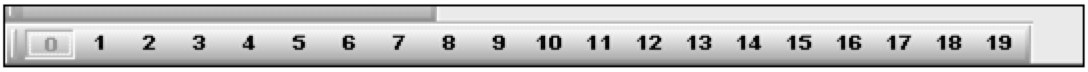


It is also possible to change the colour and the line-type without having to use the quick menus. Simply select the colour and line-type from the system bar directly



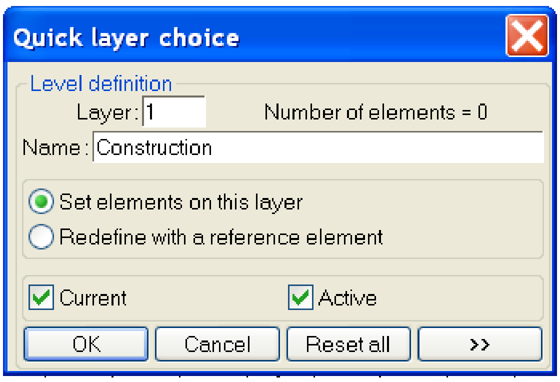
Quick Layers:

If you activate "Quick Layers" in the rapid menu, you will obtain a menu at the bottom of the screen as shown.

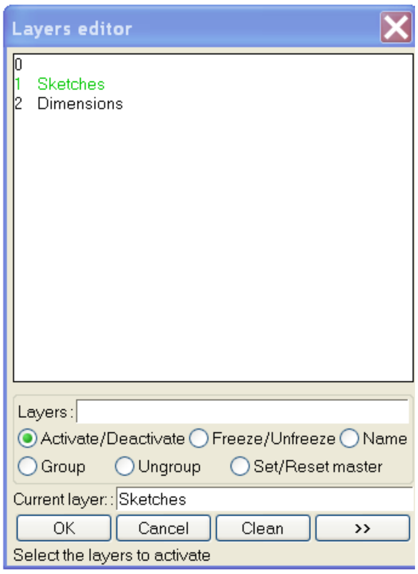


You can toggle a layer on and off by clicking on its number with the left button of the mouse. Red indicates that the layer is on. Clicking on it again turns it black which hides the layer. Green indicated the current layer. Right clicking on a layer number (e.g. 1) brings up the "Quick Layer choice "dialogue box

You can specify a name, e.g., "Construction", and set the layer as current and active



Managing the layers:



This dialog box appears by clicking "layers" on the status bar.

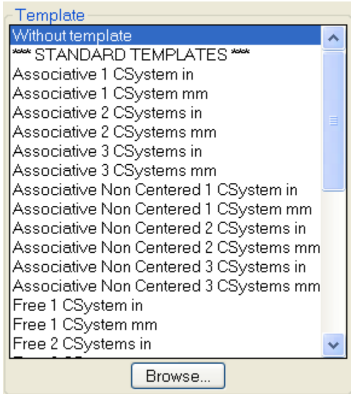
Configuring the layers need 3 parameters :

- The current layer is *green* in the dialog box and in the fast level bars.
- The active layer (visible) is *red*.
- The deactivated layers (invisibles) are *black*.

You can access several options in the bottom section of the dialogue box. You can freeze a level (make it inaccessible), rename it, group or separate it.
Groups appear in *blue* and have a name.

II - Main functions presentation

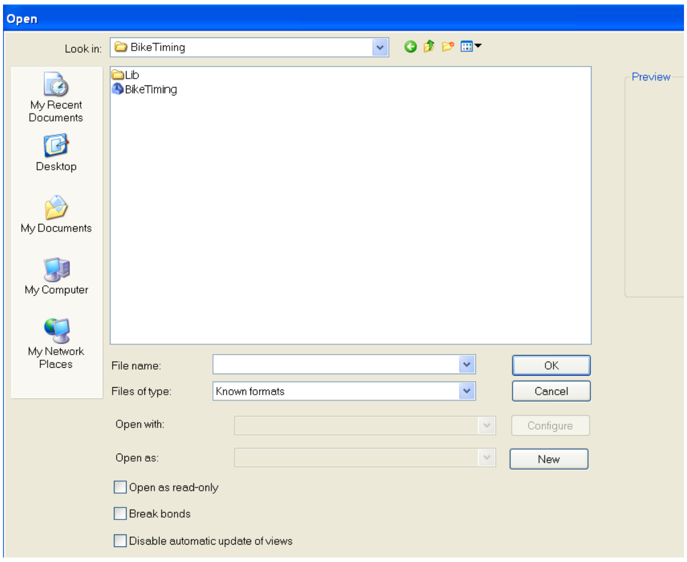
New document 



There are several types of files created in TopSolid;
.TOP for the models, .DFT for the drafts, the .CAM for machining..

For each file type there is a selection of Templates provided for creating new documents. User defined templates can be stored in the Config/Template directory of the software.

Open an existing document 



TopSolid shows a list of files in the current folder with the extension .top and .dft and also files supported by the direct interfaces STEP, IGES, DXF, DWG, Parasolid, and ACIS etc. Some direct interfaces are purchased separately.

Note:
New creates a new document.
The Configure button is active depending on the type of direct interface file used.

Save  or Save as  to save as a different name.

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. Files can also be saved in other formats such as STEP, IGES, DWG, DXF, etc.

Print

Prints the current document. Depending of the application used you have will have different printing options.

- Print the graphic zone of the screen.
- Print a selected area with the **(LM)** of the mouse
- Print in paper scale, with chosen dimensions.

Cancel

Cancels all the actions carried out within the current function but does not exit it, to quit the function press the Escape key

Undo

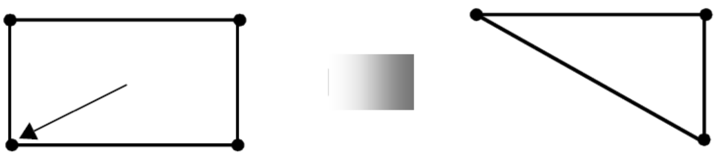
Undo the previous action within the current command.

Delete element

Deletes the selected elements. The option ALL THE ELEMENTS will allow, (after confirmation) the complete document to be erased.

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).

Insert element

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

Modify element

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

Move parents

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

Contour 

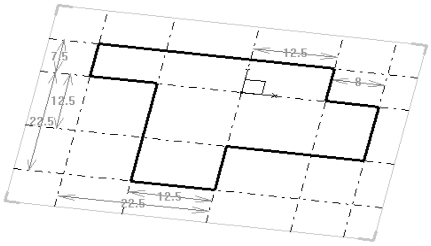
Creates contours over existing sketch lines, or on the grid of the active coordinate system.
Closed contours are automatically created when the start point is re-selected

Sketch lines 

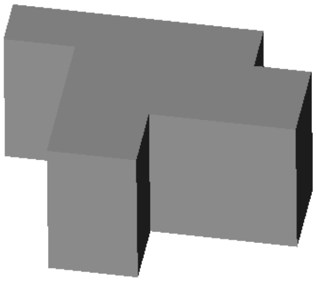
Sketch lines are created relative to points or elements, various option boxes allow for the change of angles etc.

Extruded shapes 

Creates an extruded surface or solid from a profile.
Generally if the profile is open a surface is created. If it is closed a solid is created.



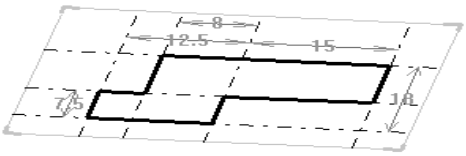
(Profile to Extrude)

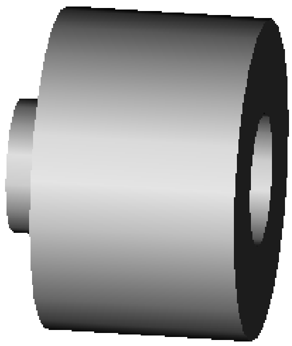


(Extruded Profile)

Revolved shapes 

Creates a revolved surface or solid from a profile around the selected axis.





II – Drawing basics

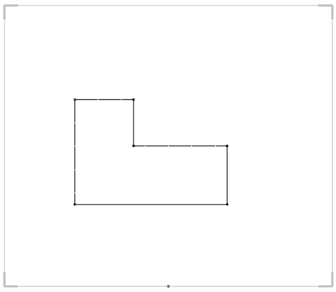
Creation of contours

There are two ways of drawing contours:

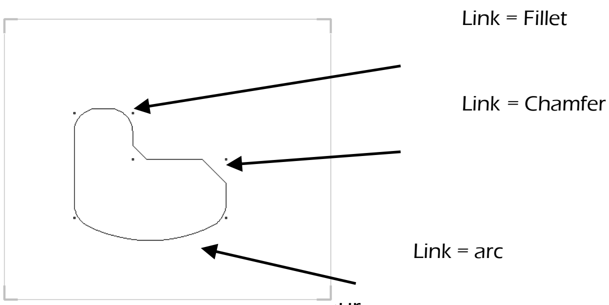
- Clicking point to point.
- Tracing over construction geometry.

Point to Point - Simple Contours:

To define a contour using points, lines are sketched defining the relevant points of the part. The actual dimensions / angles of the shape are defined later by dimensioning. Once the shape has been drawn you can use Modify to change the conditions at a vertex (chamfer, fillet or nothing) or between two points (change the link type line, arc, tangential) depending on whether you select near an end or at the middle of a side***.***



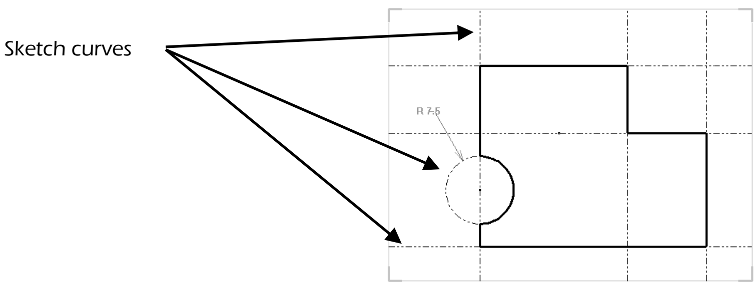
Contour with points



Modification of a contour

Creation from construction sketches.

To define the contour the user uses basic shapes (lines, circles…).
The dimensions of the contour depends upon the dimensions and positions of the sketch curves.



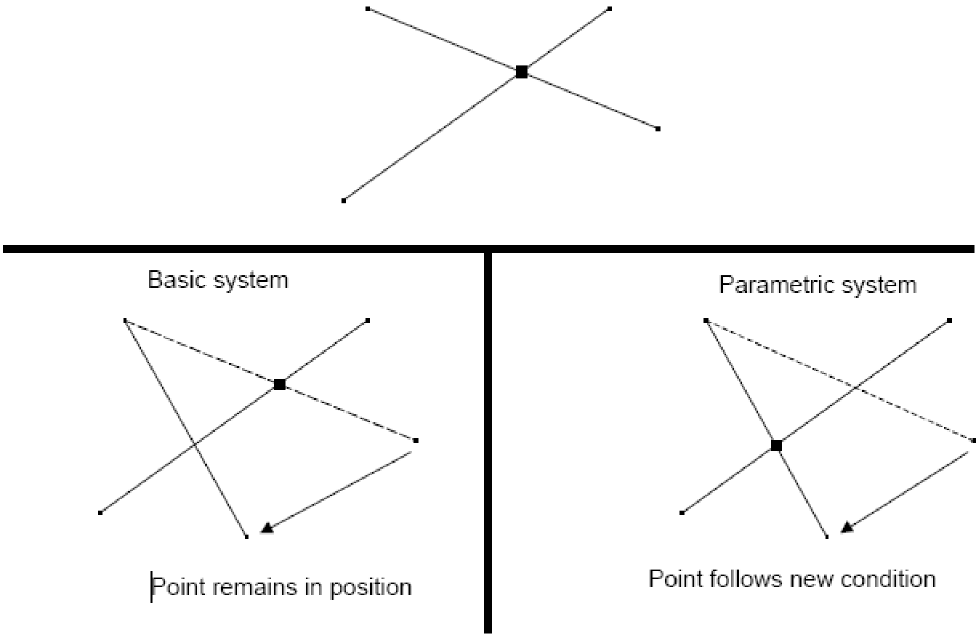
Contour by sketches

III – What does Parametric mean?

Parametric modelling allows the part to automatically link to the basic geometry from which it was created, so that changes can be automatically updated throughout the design process. A simple example follows.

In a traditional cad system when the operator creates a point at the intersection of two lines the point is created but if one of the lines moves later, the point does not automatically move with it.

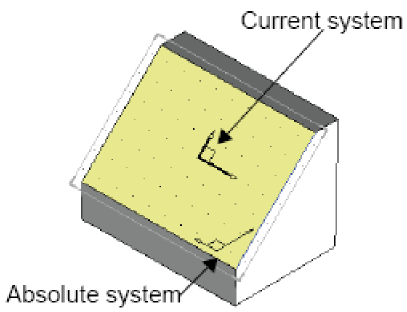
In a parametric system when the operator creates a point at the intersection of two lines it remembers this operation so that if one of the lines moves later the point is automatically updated to be at the new intersection point.



The coordinate systems

A coordinate system allows the creation of a work plane for the construction of elements. When we start a new document (associative 3 Systems mm) there are three coordinate systems: the absolute XY one, and a XZ plus a YZ coordinate system.

To change or create a coordinate system, select Current coordinate system from the system bar then pick a face or a coordinate system.
In the example shown, it is necessary to select the yellow face.



Numerous forms of coordinate system are available and these can be accessed from the coordinate system tool bar.

NOTE:

- This icon will allow you to set any coordinate system as the current or active one.
- This icon displays the coordinate system tool bar and allows the user to create a new coordinate system.
- If used alone the coordinate system created will NOT automatically become current, however if you use first of all, then the resultant coordinate system WILL automatically become current. The current (or active) coordinate system is drawn in a thicker outline.

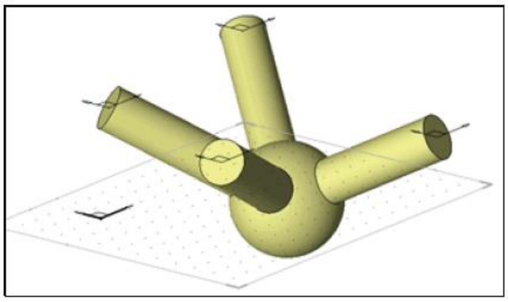
	Coordinate system on point
	Coordinate system through 3 points
	Coordinate system on profile
	Coordinate system on a profile and point.
	Coordinate system on face and a point
	Coordinate system constrained on a face
	Duplicated coordinate system

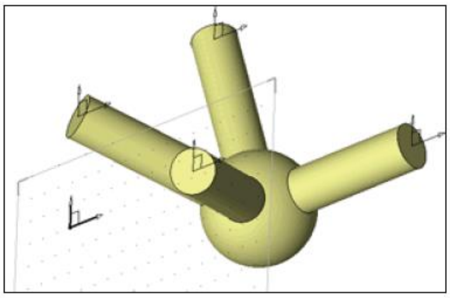
The most widely used coordinate systems are:

Examples of their use:

Coordinate system on a point:

Constructs a coordinate system positioned on a point, taking the orientation of the current coordinate system.



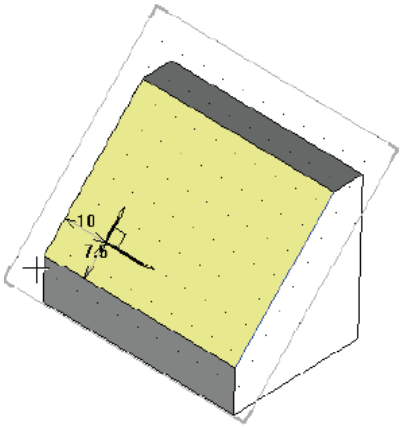


Coordinate system on curve and point:

Lets you to create a coordinate system that is based on a curve and a point.

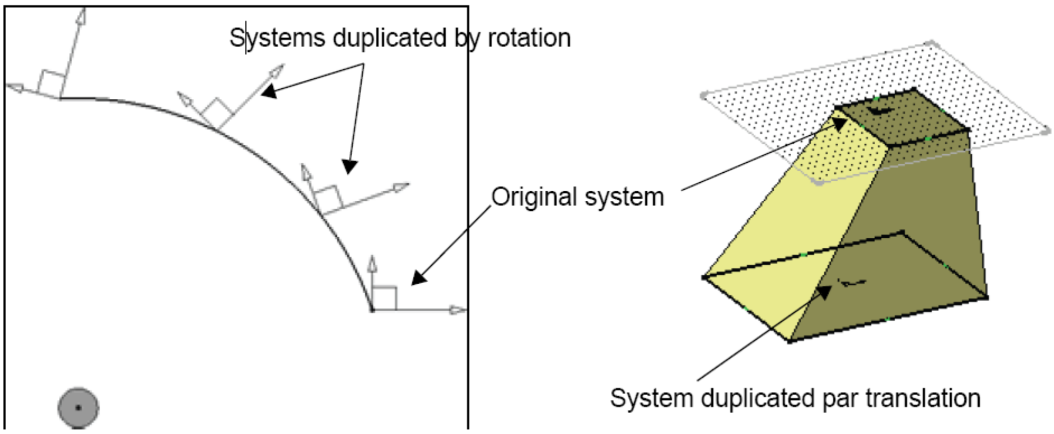
The curve defines the orientation of the coordinate system (XY normal in relation to the curve), the point defines its position.

 **Constrained coordinate system on a face:**
Creates a coordinate system placed on a face and positioned in respect of the edges or contiguous faces.



The DYNAMIC button enables searching for these edges or sides used.
Note: Very convenient as the two dimensions easily allow the coordinate system to be moved later.

 **Duplicated coordinate system:**
Creates a copy of a coordinate system by applying a transformation, translate, rotate etc.

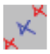
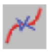


Points

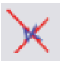
Points are elements that comprise a distinct position. They are maintained during associative mode design. Points are used to join dimensions, to impose dimensional and positional constraints. In contrast, in "free design mode" and "non associative curves mode" points are deleted immediately as soon as the function is changed, as they cannot be used to constrain an element.

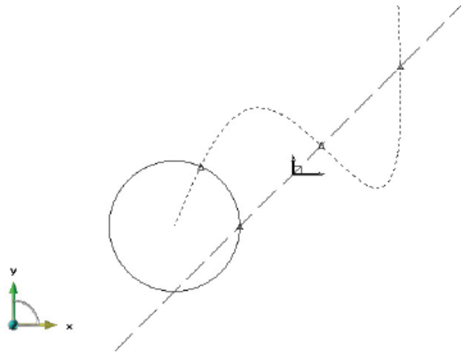
Creation of points in TopSolid is performed in several ways: First of all, during construction of b-spline curves, the user is actually tracing points without realizing it. These points include, for example, the centre of a circle that is placed on the grid, a line that is joined to the end of an existing curve, etc. The other method is to use a specific point creation function, by choosing the menu Tools, Points or the point's tool bar.

The most widely used points systems are:

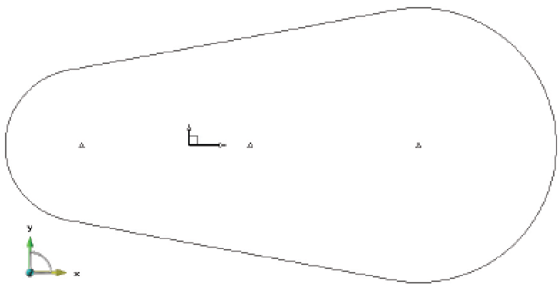
	Intersection point: point at the intersection of 2 curves.
	Middle point: a point created between 2 other points.
	Centre point: a point in the middle of an element. (lines or circles).
	Point on a curve: create a point attached to a curve.
	Barycentre point: a point at the centre of gravity.
	Duplicate point: a duplicated point which is translated from the original.
	Axis-curve/plane-face intersection point: The Intersection point between an axis/ curve axe and a plane/face allows you to create an intersection point between a curve or an axis an a face or a plane.

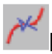
Using points

 **Intersection point:**
Creates an intersection point between curves.

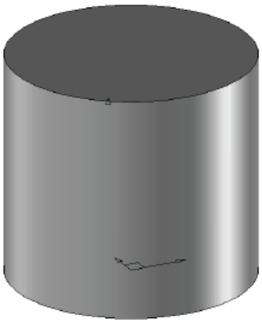


 **Middle point and Centre**  **point:**
Creates a centre point of the arcs and then the middle point between the two centre points.

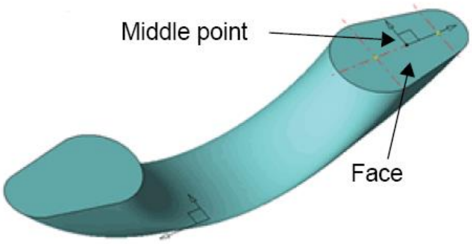


**Point on a curve:**

Creates a point on the upper edge of the cylinder.

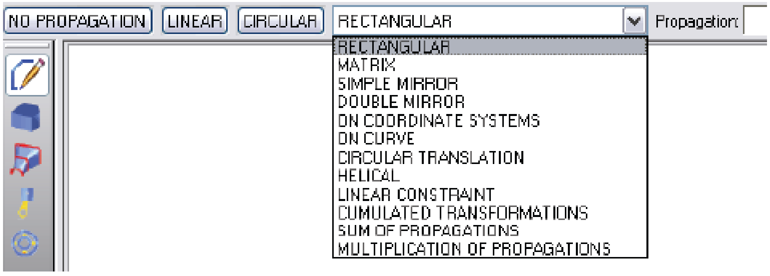


It is possible to combine the coordinate systems with points on coordinate systems with icons showing a red point. For example, in the following case, a coordinate system on face and point will be created, and the point will correspond to a middle point between 2 other points (which are actually the centre points of the 2 circles).



Transformation and associativity

A part or element can be transformed or moved from one place to another. Or it can be duplicated either singly or in multiples. The most common propagations are translation and rotation, but it may be symmetry, double symmetry, rectangular, etc. The important question with respect to the associativity mechanism is: what happens to the parents when such a propagation is applied?



There are numerous options for the user to choose from and the correct choice will depend on the required outcome.

Move and turn functions

These 2 functions apply the translation and rotation to the parents of the selected element. The entire element will be affected. For elements such as lines or circles and simplistic shapes this has very little impact. But if a part that you are applying a move and turn function to, has several other parts based from it, the result will also be applied to those parts. This may not give you the desired effect.

Repeat or duplicate?

When applying a propagation, it may be useful if the number of resulting elements were to be a parameter, so that it can be changed later. The Edit, Repeat function allows for this, however the Edit, Duplicate functions do not (but these are less complex in terms of associativity). The main effect of the Repeat function is to create a top level element other the copies, and the copies themselves are in fact considered as completely new elements. For example, you may have to detect an element inside the repetition to access it.

Repeat (or not repeat) any further modification to the original?

The duplicated or repeated elements may be allowed to follow any new operations (chamfer, holes,...) applied to the original. This has to be set by the user. By default, the subsequent operations to the original will be applied on the duplicated or repeated elements too.

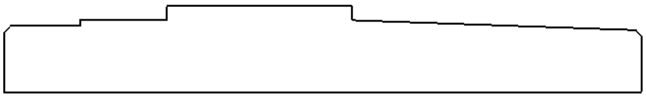
Linked or independent copies?

If you simply want to copy an element without any link back to the original, use the **Copy** with our reference option. This has the effect of breaking the associativity between the original and its copies.

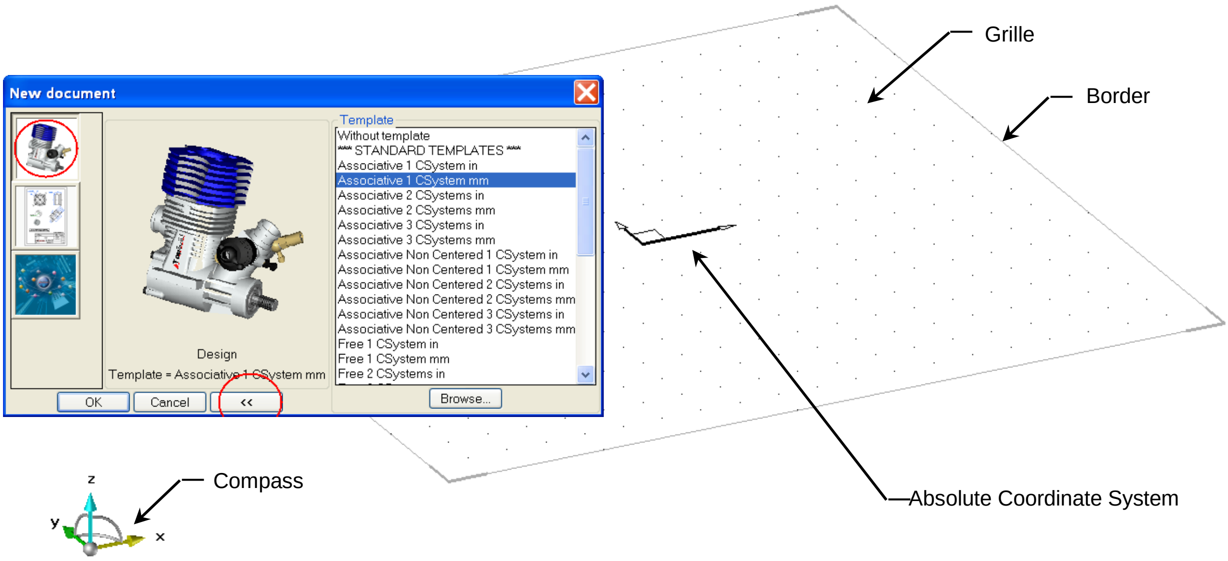
Exercise n°1 : Contour in Sketch mode

In this exercise you will:

- Create a contour using a sketch
- Use constraints
- Create dimensions
- Examine sketch limitations
- Create a chamfer
- Use control elements



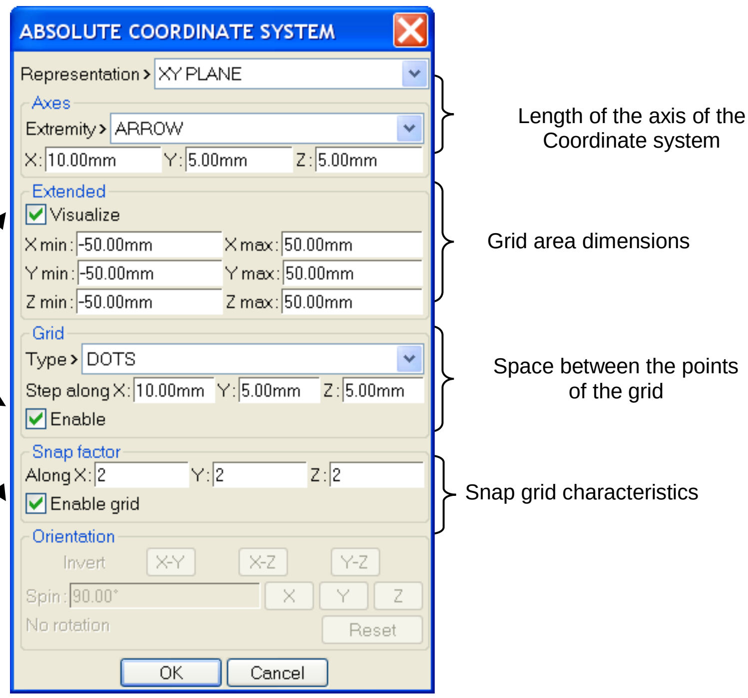
Create a new document in TopSolid
Choose the « Associative CSystem » template among the standard templates



Use the **Modify element** feature to modify the characteristics of the coordinated system axis.
Set the grid spacing to 10mm in x.
The snap factor is still 2, so the cursor snaps to half the grid spacing (5mm)

Enable/Disable frame, grid, snap grid

The first coordinate system in TopSolid is the absolute one. It can't be changed.



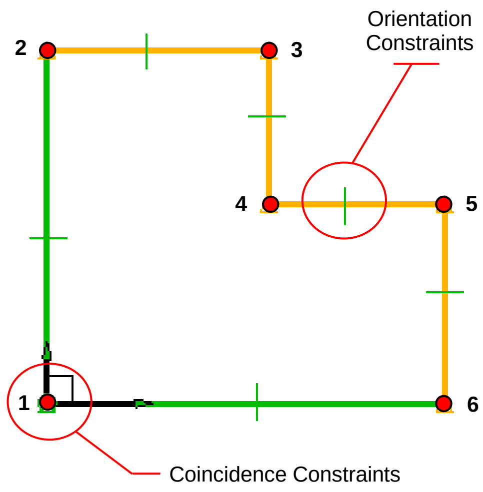
Switch to **Top View.** TopSolid'Design orientates the X and Y axis horizontally and vertically respectively
.

Activate the **Sketch** mode

Start a sketch. A green frame appears around the work space when the sketch mode is activated.
.

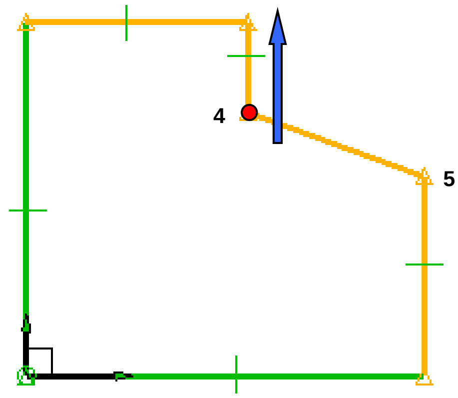
Create a ***Contour***. Use the points of the grid to draw the contour. Point (1) is located at the origin of the CSystem.

To close the contour, click on the starting point, here (1), or on the first segment [1;2]



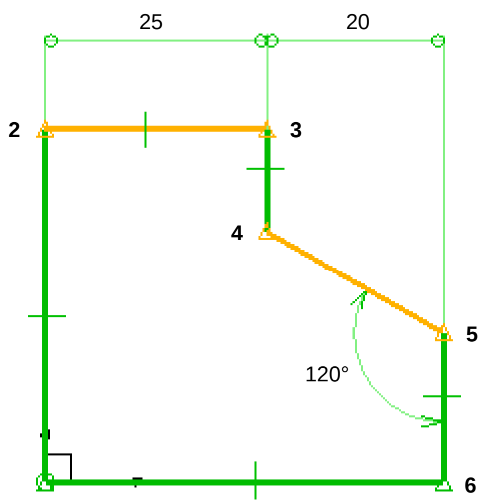
Delete the horizontal orientation constraint on X between points 4 and 5
.

Move point 4 upwards as shown



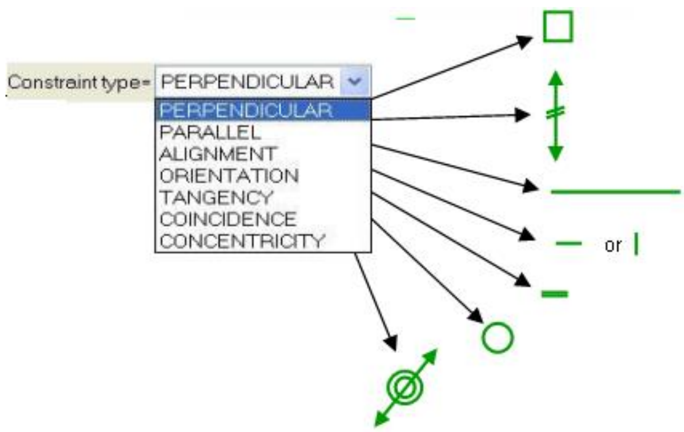
Constrain the length of the segment [2;3] and the segment [4;5]. In the dialog bar the orientation option is FREE by default, but you can force it to be HORIZONTAL, VERTICAL or PARALLEL.

Constrain the angle between the segment [4;5] and the segment [5;6].



Constraints on a contour

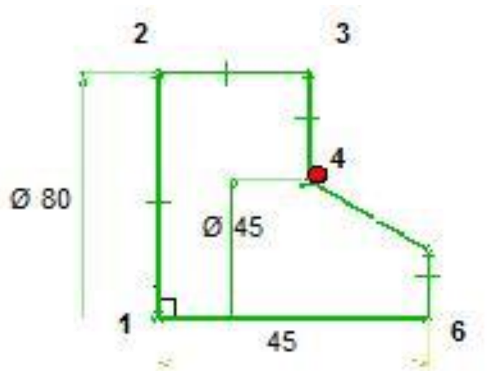
- In the tool menu the option Auto-Constraint is activated by default. As you draw the contour, TopSolid assign the geometrical constraints.
- The constraints are deleted with the **Delete element** feature.
- The constraints are added using the **Constraint** feature in the sketch menu



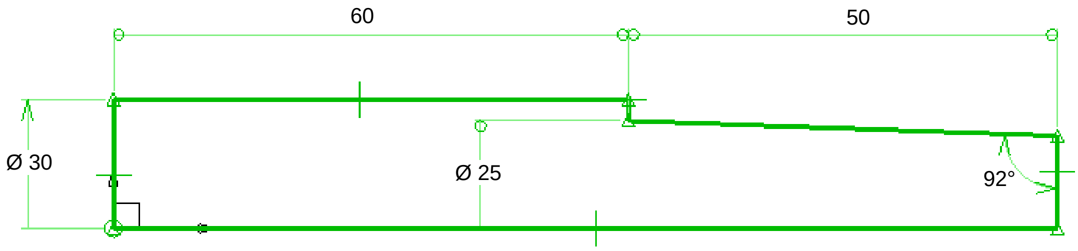
As we work with a revolved part, we can create a Half-part dimension between the revolution axis [1;6], the segment [2;3] and the point (4).

Assign a dimension to the length of the segment [1;6].
Before placing the dimension, set the
option constraint = NO.

This option stays enabled for the next dimensions



Use the ***Modify Parameter*** feature to adjust the dimensions to give the following results.



Enlarge the frame to cover the entire sketch using the **Modify Element** function. Click on the border, hold, and drag to the required position

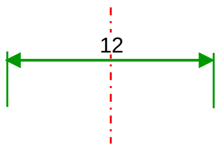
Use the **End Sketch** function to validate the sketch. All the construction elements become invisible
.

Dimensioning

- The role of a dimension is described by is color :
  Green indicates a driving (active) dimension which can be changed.
  Yellow indicates passive dimensions which are for visualization only.
  Red indicates an invalid dimension, as a result of over constraining.
  Orange dimensions are deactivated but can appear in draft views.

- To transform an active dimension to a deactive one, use the **Modify Element** feature on the dimension ,click the option CONSTRAINT then DISABLE

- to place a symmetry constraint on a dimension, use the Modify Element feature, accept the CONSTRAINT option and then click the symmetry axis of the dimension.
  It's also possible to put a symmetry constraint on the dimension after placing it. When you create it select the option SYMETRY CONSTRAINT .

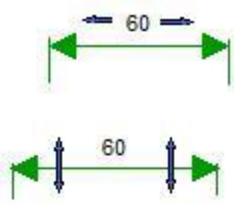


- To modify a dimension, click on it with the **Modify Parameter**.

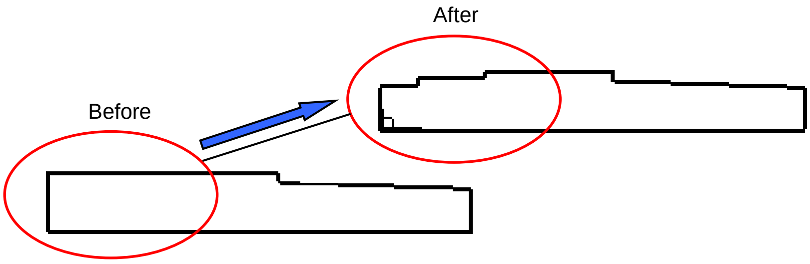
- To move a dimension, use the **Move Parent** feature

  Clicking on the number allows it to be moved along the line.

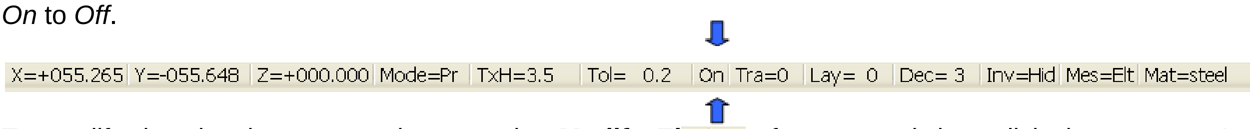
  Clicking on the line, allows the line to be moved up or down.
  .



Next we will modify the sketch as shown below;

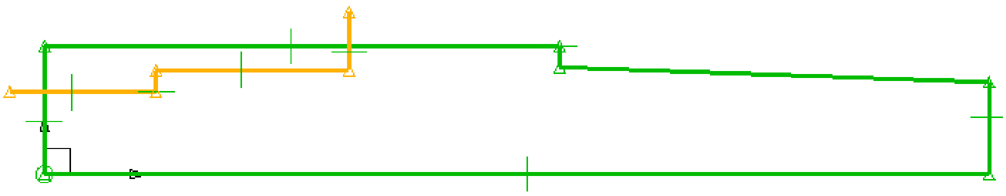


To be more accurate when you place your points, disable the snap grid in the current CSystem: click the *On* to *Off*.

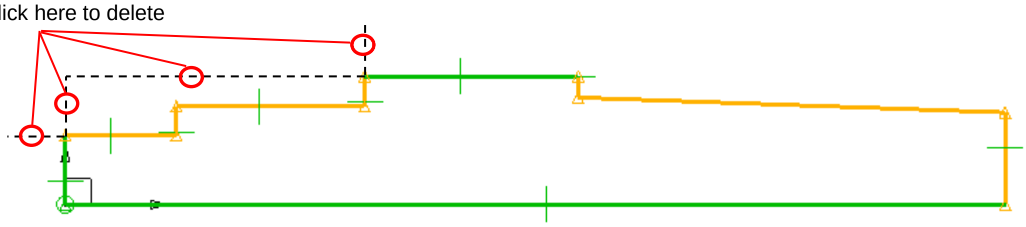


To modify the sketch or constraints use the ***Modify Element*** feature**,** and then click the contour. An automatic reactivation of the sketch occurs

Create a new open ***Contour*** on the same sketch with the following constraints.



Trim the first contour using the second and so on. In DELETE mode, click the lines that you want to delete. This operation can delete some constraints, so some lines will become orange.



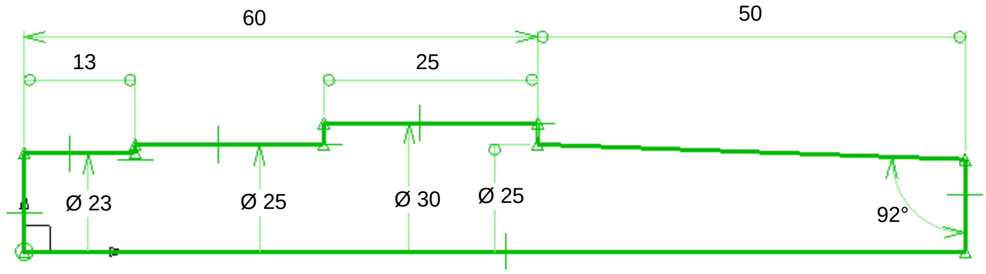
The contour in sketch mode

- A color code exists in the constraint mode.
  -Green lines are fully constrained and can't be moved or stretched.
  -Orange lines are not fully constrained and can be moved or modified.

A contour will be totally constrained when it is totally green.

- By default, the Trim function only acts on continuous lines, to activate dotted lines, switch 'consider construction lines' = ON

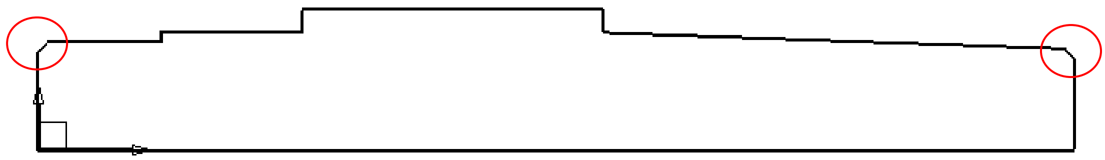
Constraint the contour to that it is totally constrained as below.



Use the End Sketch function. All the construction elements become invisible.

Activate the **Curve** context.

Make two chamfers of 1mm, and 45 degrees



Save the file in the training folder, under the name: « Shaft profile.TOP »

File extensions for TopSolid

3D Files :	**.TOP**	Quick view files :	**.PNG**
		Backups Files:	**.BAK**
		Locked Files :	**.LCK**
2D Files :	**.DFT**	Rescue Files:	**.RSC**

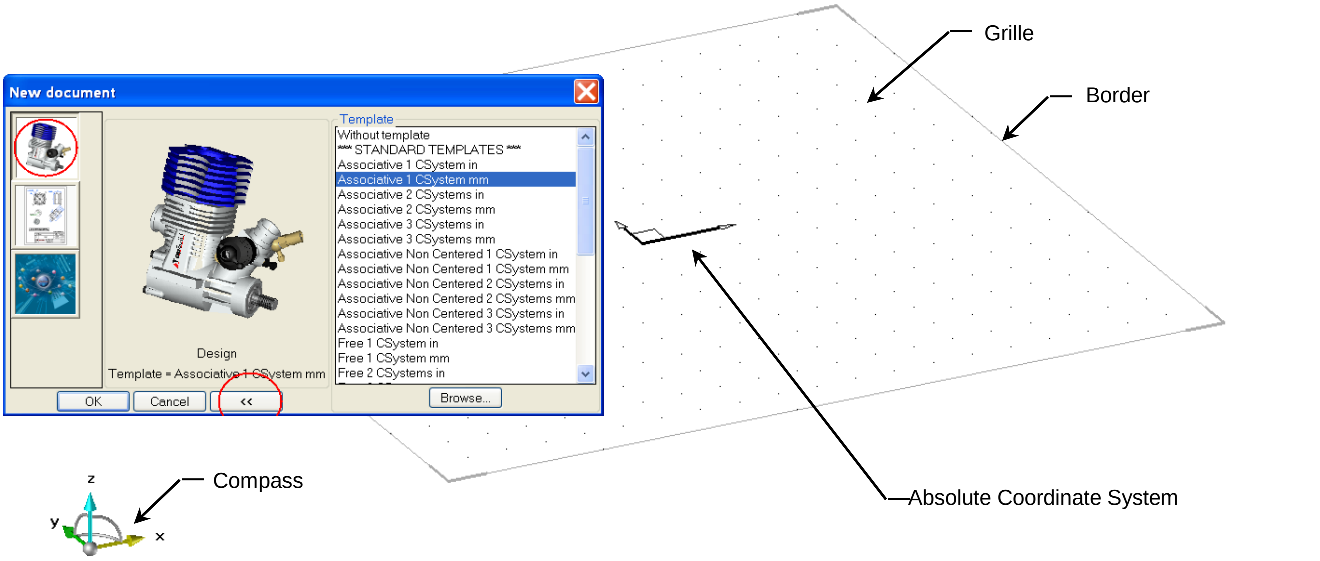
If a file is locked, delete the **.lck** file found in the directory using Windows files explorer.
.bak and **.rsc** should not be used directly. To use them they should be renamed with **.top** or **.dft** extensions.
The **.png** file is recreated every time you save a **.top** or **.dft**.

Exercise n°2 : Contours by Pass Over / Trace

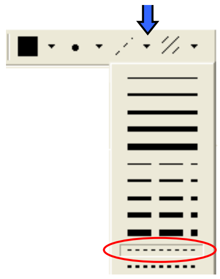
In this exercise you will create:

- Circles and construction lines
- Sketch lines
- Parallel lines
- Intersection points by passing/tracing over intersection lines

Create a new TopSolid'Design document.
Choose the « Associative CSystem » template from among the standard templates.



Switch to ***Top View.***



Change the **Line Type** to dotted lines which will be used to make construction lines.

Place horizontal and vertical Sketch Lines on the origin of the absolute CSystem
Use SWITCH TO VERTICAL and SWITCH TO HORIZONTAL to orient the lines

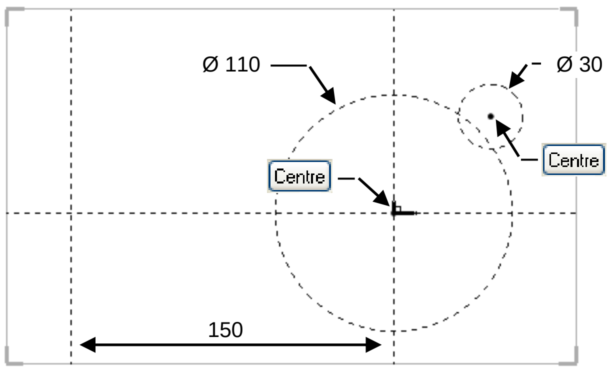
Make a parallel line at a distance of 150mm.

Enlarge the area of the absolute CSystem using Modify Element, sketch lines will stretch automatically..

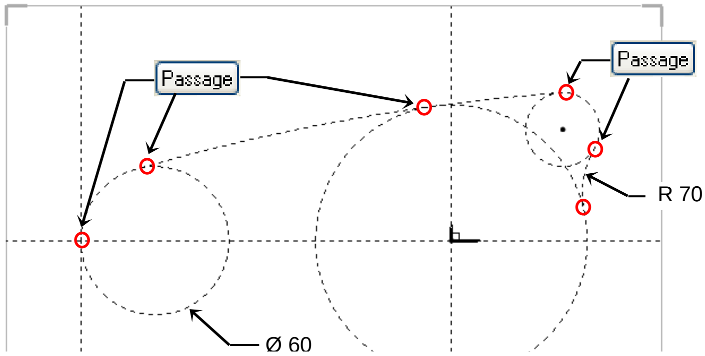
Draw the first **circle** on the origin of the CSystem, with a diameter of 110mm.

Draw a second **circle** of diameter 30mm, anywhere on the screen.

Move this circle to place it as shown on the diagram, using the ***Move parents*** feature.



Draw a circle of diameter 60mm, with its center on the horizontal line, and tangent to the vertical line.

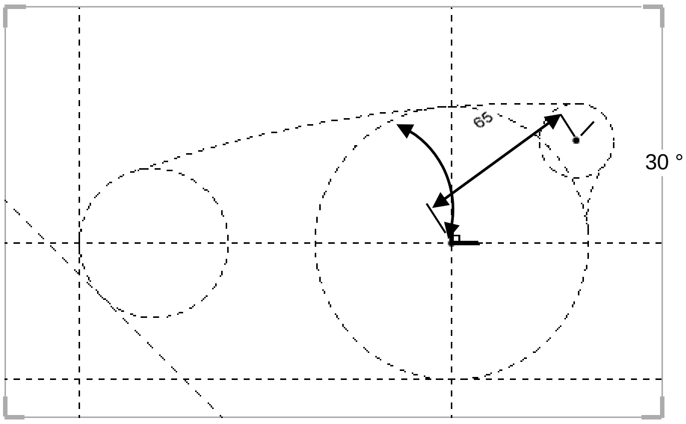


Draw a ***part circle*** radius 70mm tangent to the 110mm and the 30mm circles

Draw a ***part circle*** tangent to the 30mm, 110mm and 60m circles.

Draw a **sketch line** at 135° tangent to the 60mm circle.

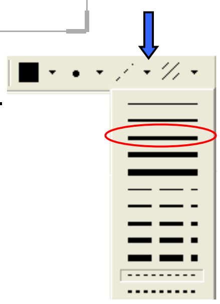
Draw a horizontal ***sketch line,*** tangent under the 110mm circle.



Constraint the position of the 30mm circle to the origin, using the ***dimension*** feature

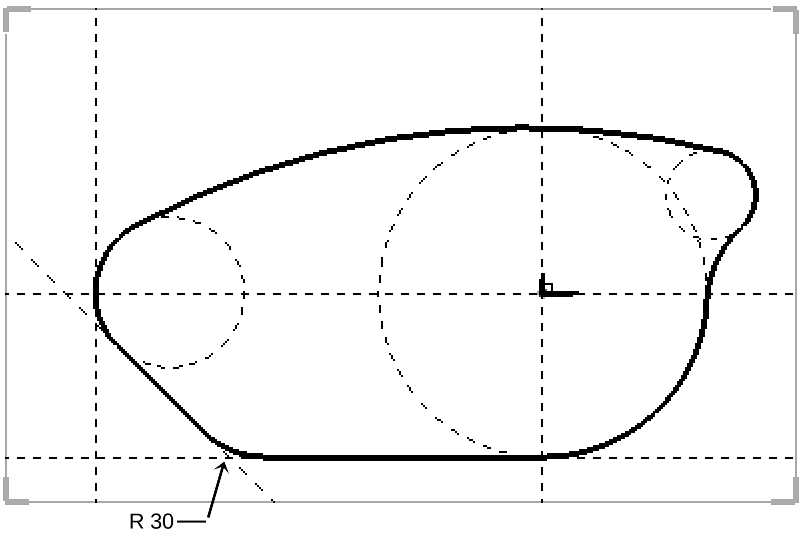
Add an angular dimension to totally constrain the circle. Using the Dimension function, click on the origin of the CSystem, the center of the circle and the horizontal line.
.

Change the **Line Type** to a thick continuous line. This will be used to trace the required contour.

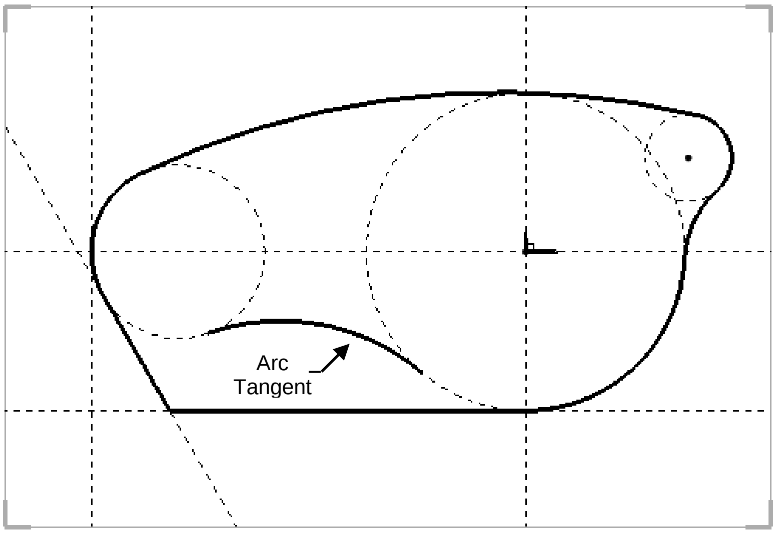


Draw the ***Contour*** by tracing over the construction lines as shown.
(don't try to pass by the intersection points, TopSolid will find it automatically)

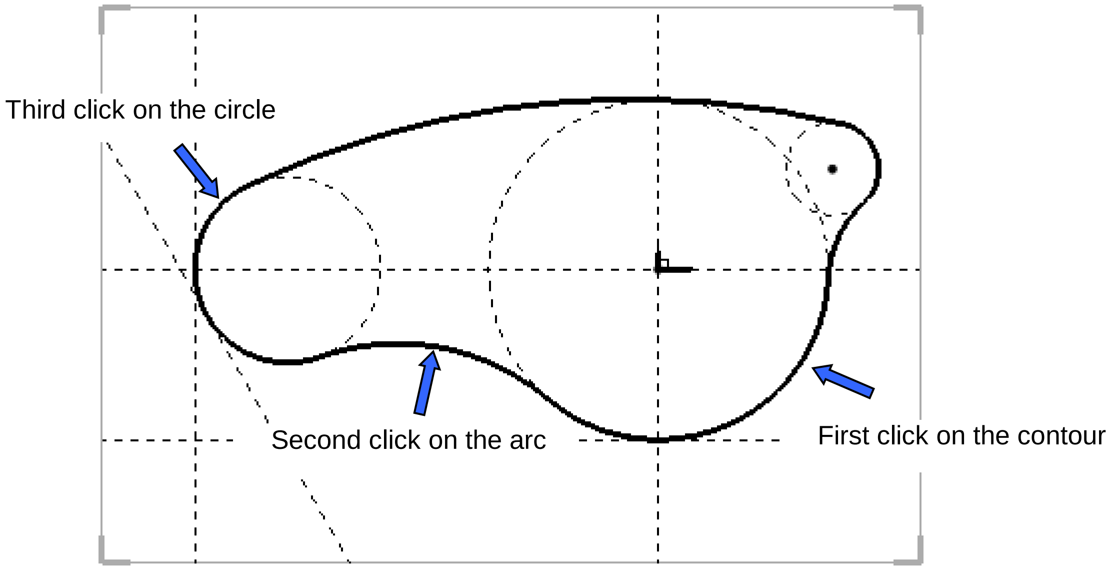
Create a 30mm ***Fillet.***



Create a tangent arc of diameter 150mm, to the 60mm circle and the 110mm circle.



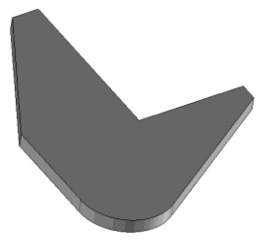
Insert this arc in the contour.
The construction direction affects the element section order in the contour.
If construction sense is the clockwise:
Click the contour on the 110mm circle, then the 150mm arc, followed by the 60mm circle.



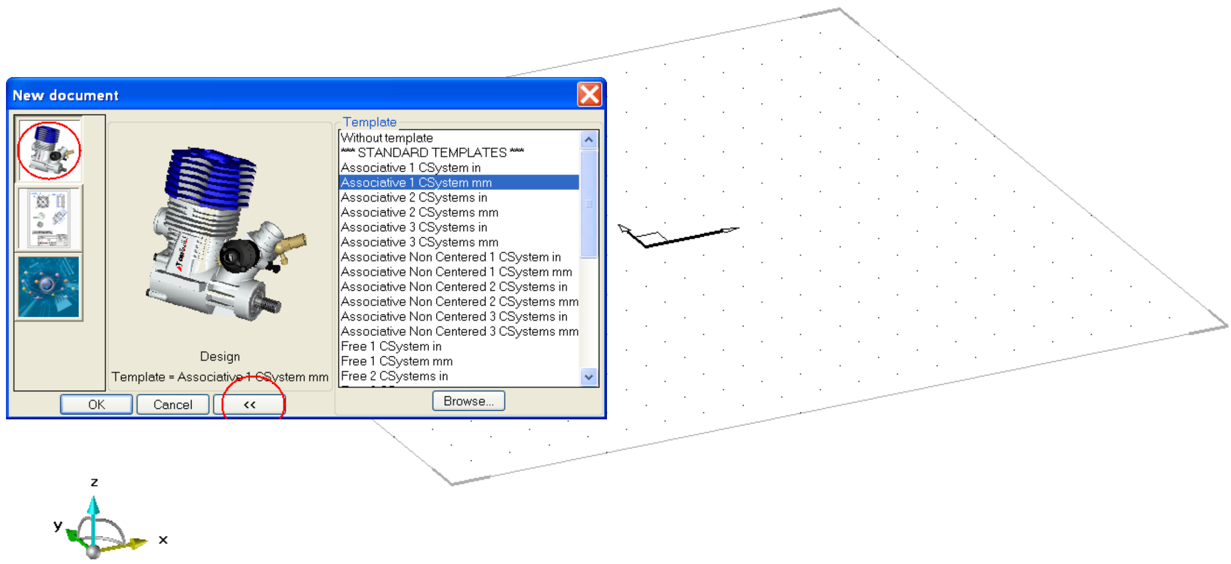
Exercice n°3 : Square Bracket

In this exercise we will examine:

- Creating an arc in a sketch
- Parametric dimensioning
- Extrusion command
- Modifying a sketch
- Concentric drilling in NON DYNAMIC mode
- Definition of a part
- Different views in a draft
- Filling the Title Block



Create a new TopSolid'Design document.
Choose the « Associative 1 CSystem » template from among the standard templates.

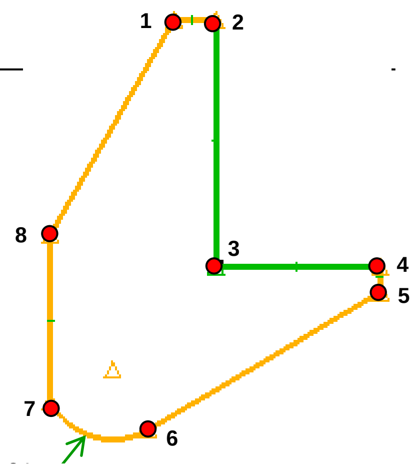


Switch to ***Top View***

Activate the ***Sketch*** mode.

Start a sketch. A green frame appears round the work zone when sketch mode is activated.

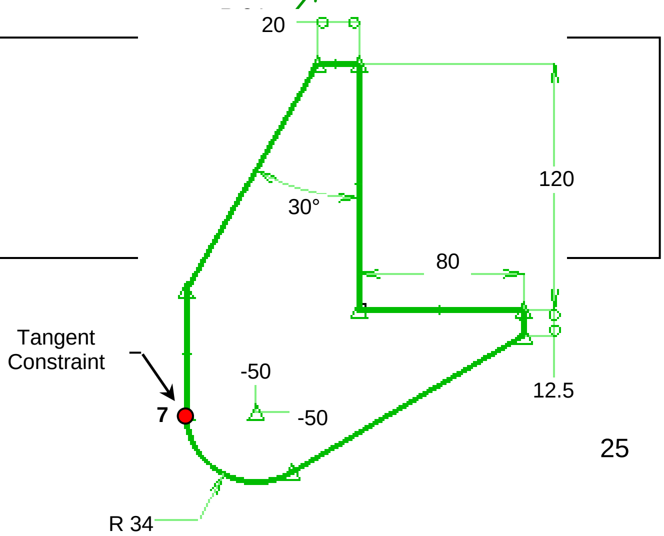
Create a ***Contour***.
Use the grid points to draw the contour. Point (3) is at the origin of the CSystem.
After creating point (6), type « a » on the keyboard to tell TopSolid'Design to draw a tangent arc. Then finish the contour.



Only the circle is dimensioned, its diameter is 34mm

Dimensioning

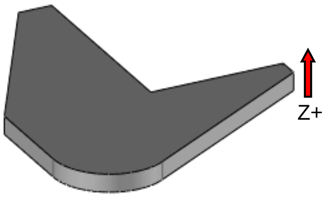
- When you modify a dimension of a sketch with only one dimension, the whole sketch transforms to maintain a homogenous profile.



Dimension the position of the center of the circle with Cartesian coordinates. Add the dimensions as shown.

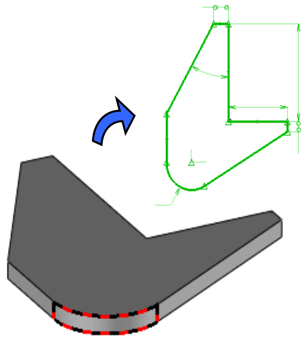
To make the sketch totally constrained, add the tangent ***Constraint*** on point (7) between the vertical part and the circle arc.
.

Extrude the contour in Z^+ to a height of 10mm.
The position of the mouse defines the direction of the extrusion.



Use ***Modify Element***, and click on the side of the part to make the construction elements appear.
Select the sketch in the list as the operation to modify, and then click OK

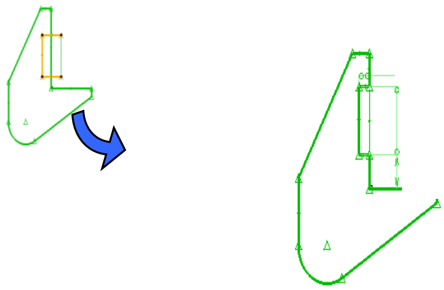
TopSolid'Design switches to top view and zooms in on the sketch



In the same sketch, create an open ***Contour*** with three points, to draw the profile of the groove.
The contour is orange because it is not yet constrained.

Trim and finish ***Dimensioning*** the contour to make it totally constrained.

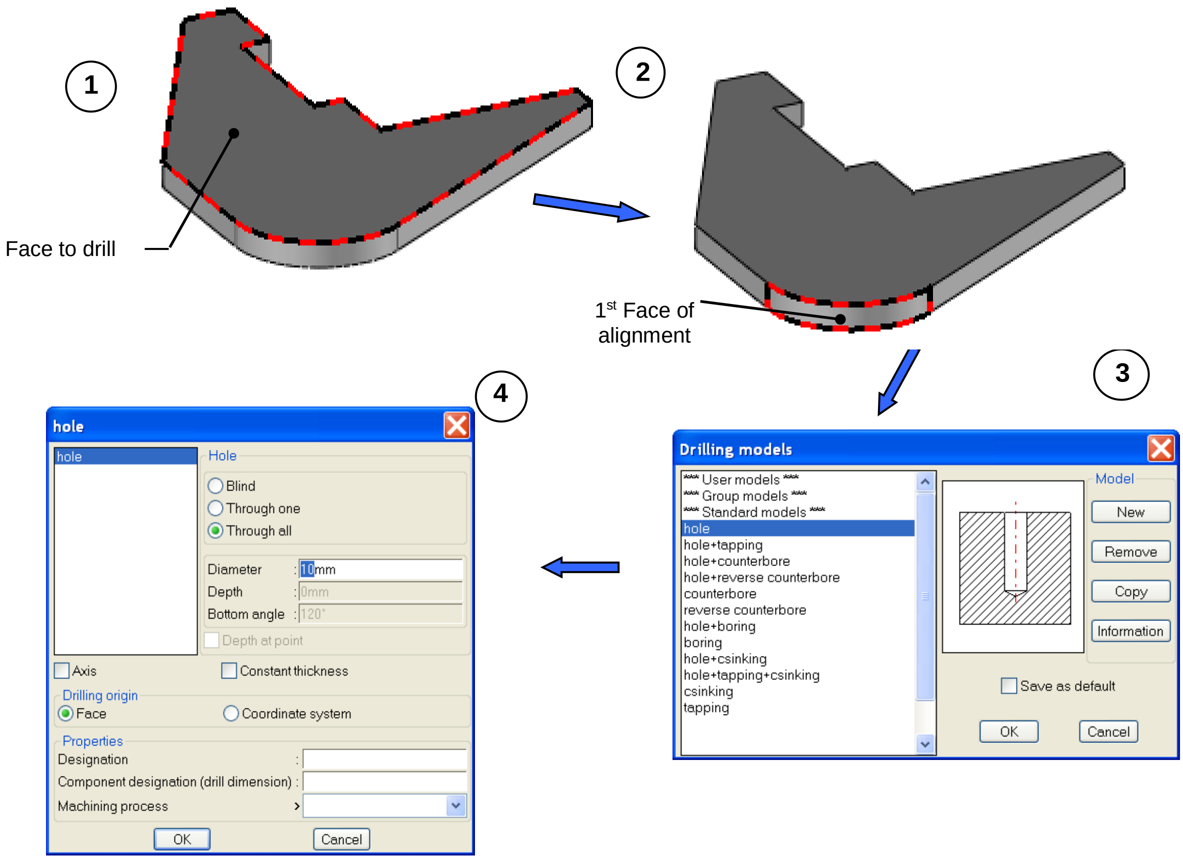
When you ***End Sketch,*** the modification is automatically done on the extruded shape and the contour elements become invisible.



<u>Extruded shapes</u>

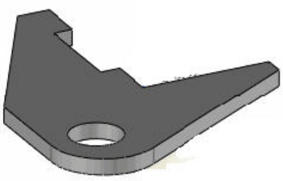
- The ***Modify Element*** function, can invert the direction of an extrusion by clicking on the arrow. The option Mode= NORMAL can switch to Mode= CENTER to extrude symmetrically about a plane.

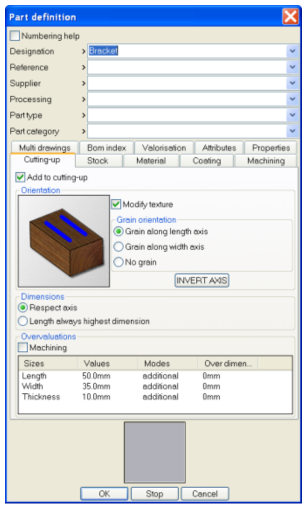
Drill a simple hole of Ø32mm, concentric to the cylindrical face in mode = NON DYNAMIC Ø32mm, using ***Drilling***



Define the designation of the part, using ***Assembly / Define Part***,
Then click on the part to insert in the assembly. Validate the STOP button.
In this case only fill the definition.

Save the file in the training folder with this name : « Bracket.TOP ».

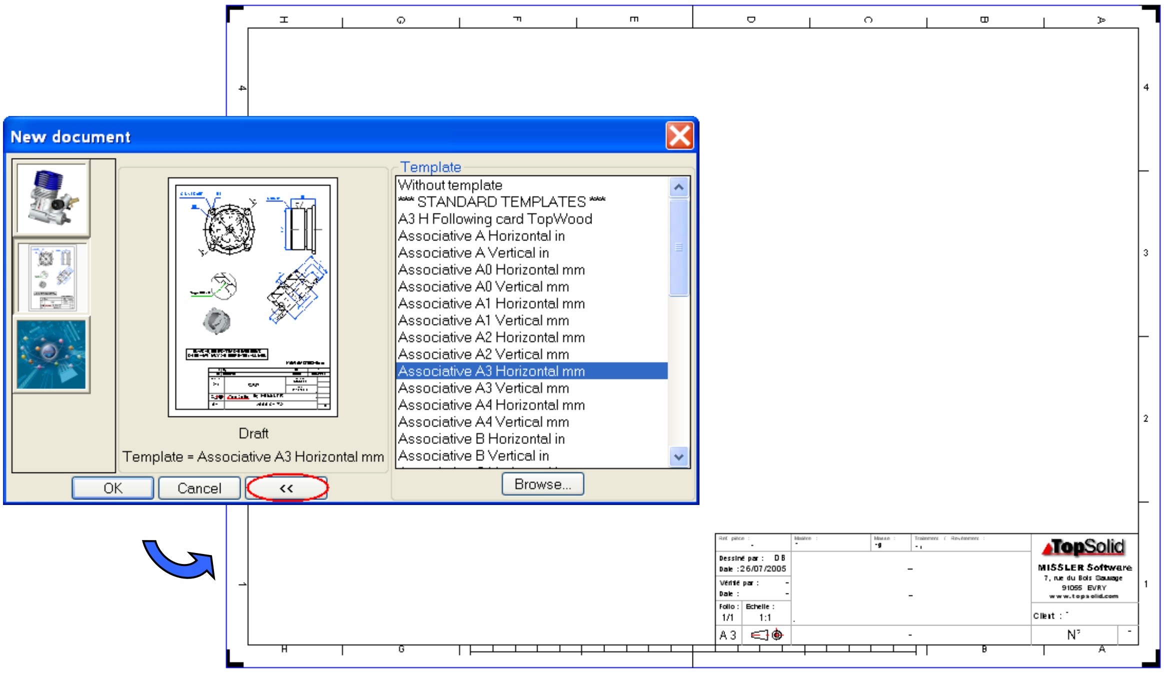




Define Part

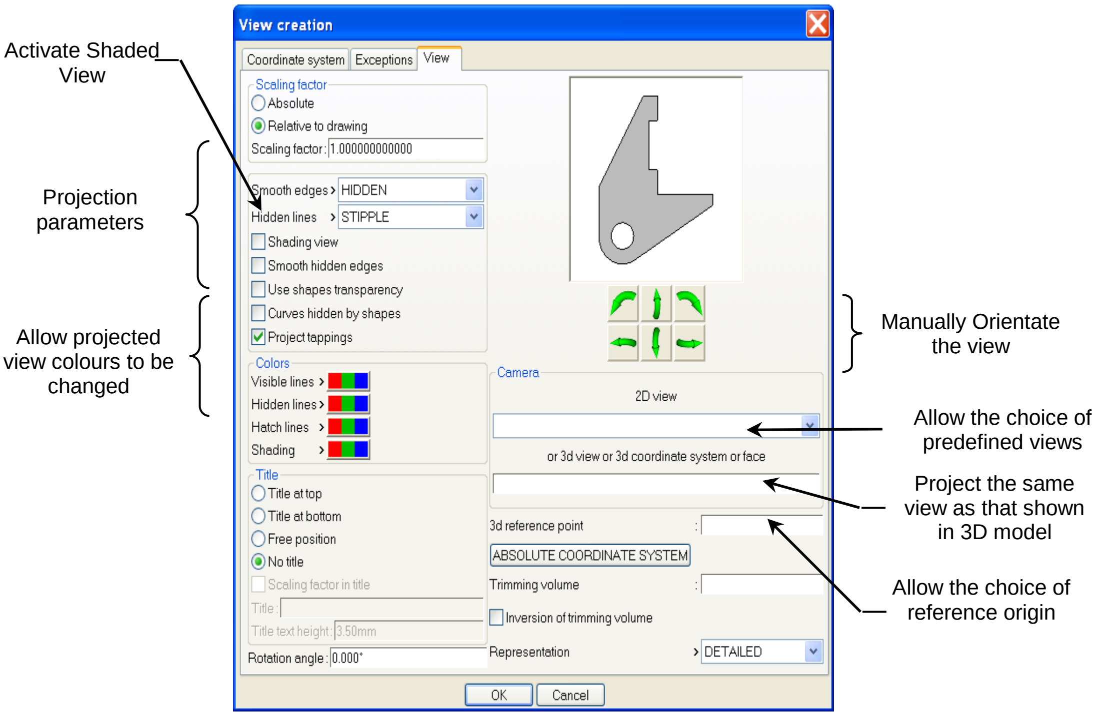
- Information in the Define Part dialogue box can be automatically retrieved later into the Bill of Materials draft file.

Create a new document – TopSolid 'Draft
From the template options choose « Associative A3 Horizontal mm ».



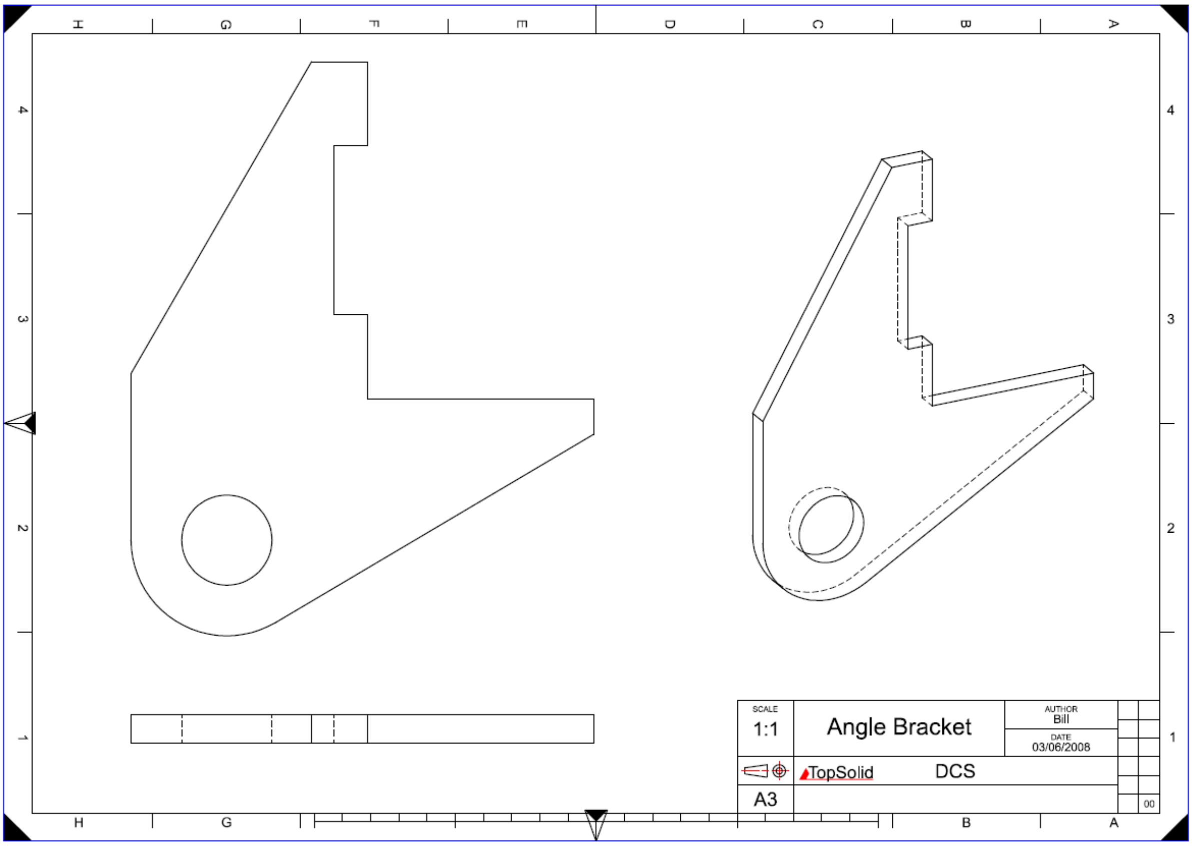
The work window automatically switches to vertical tile if only one 3D document is opened. If this is not the case, switch to vertical tile mode with the ***Window / Vertical tide***.

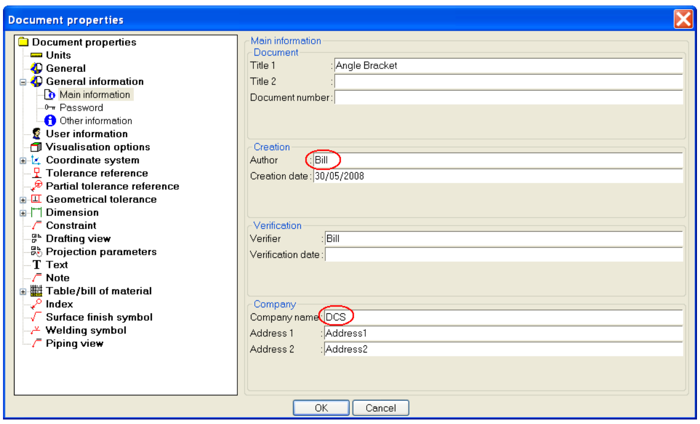
Click ***Main View ,*** then highlight the 3D part to bring up the View Creation dialogue box



After placing the main view, click AUXILIARY VIEW to set the other views and the perspective.

To move views use the ***move parents*** function.





Save the file in the training directory with the name : « Bracket.DFT ».

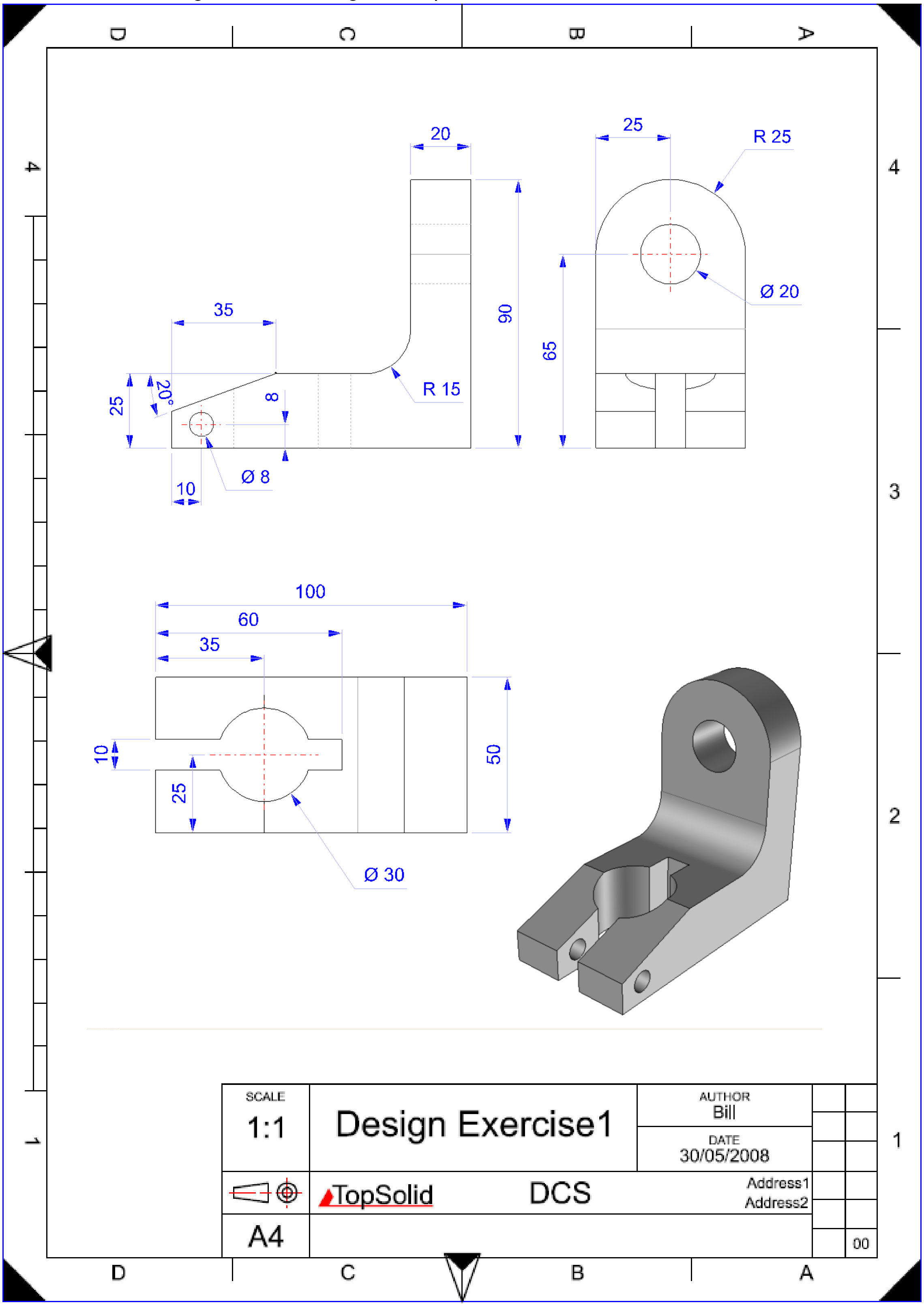
Title Block

- To fill the title block automatically, click ***File / Properties***.
  Choose the General Information /Main Information category and fill in appropriate boxes

Exercise n°4 : Angle Bracket

In this exercise you will look at:

- Creating a contour
- Drilling on a plane surface in DYNAMIC and NON DYNAMIC mode
- Creating extrusions
- Subtracting shapes
- Filleting and chamfering a solid part.

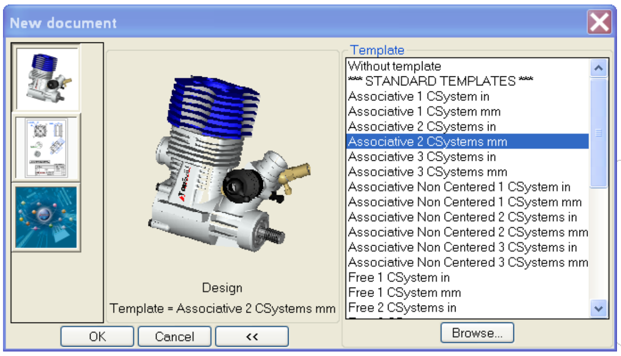


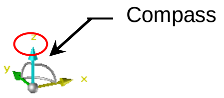
Exercise n°5 : The Knob

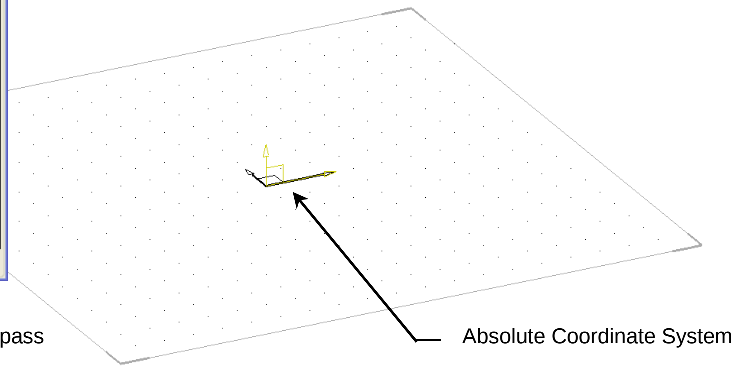
In this exercise you will use/Examine :

- Sketch lines
- Construction Circles
- Construction lines
- Limitation with the "keep" mode
- Revolved shapes
- Extruded shapes
- Limitation of shape by surface
- Embossing on a plane surface
- Concentric drilling in DYNAMIC mode
- Part view on a draft

Create a new TopSolid'Design document.
Choose « Associative 2 CSystem » from among the Standard Template.







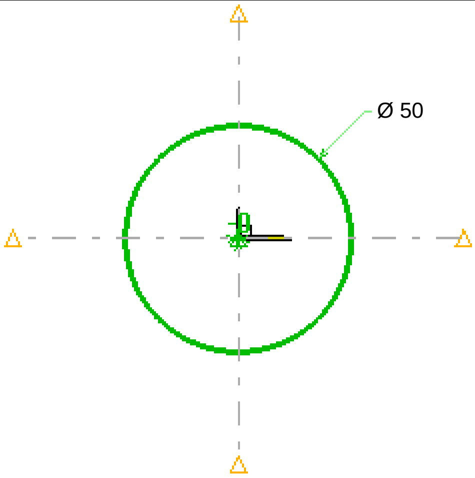
To switch to top view, left-click the Z of the compass.

Activate ***sketch*** mode***.***

Start a sketch.

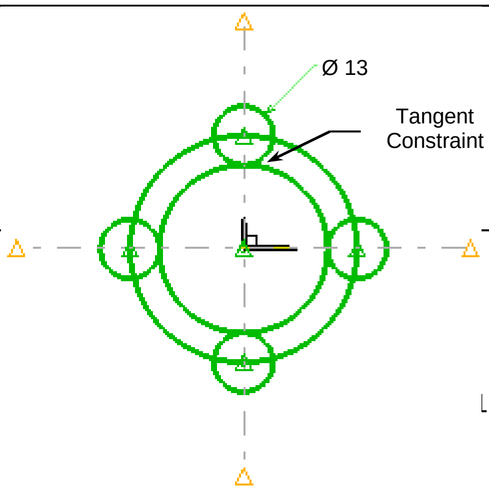
Create a horizontal sketch passing by the origin of the absolute CSystem, then validate SWITCH TO VERTICAL to create a vertical sketch line which also pass trough the origin. Sketch lines are thick, and are considered as construction lines.

Draw a Ø50mm circle centered at the origin of the CSystem.



Construction point

- When designing a geometrical shape, it is possible to set the origin of the current CSystem by hitting ENTER on the keyboard. A coincidence constraint is automatically created on the origin point. This only works in sketch mode.



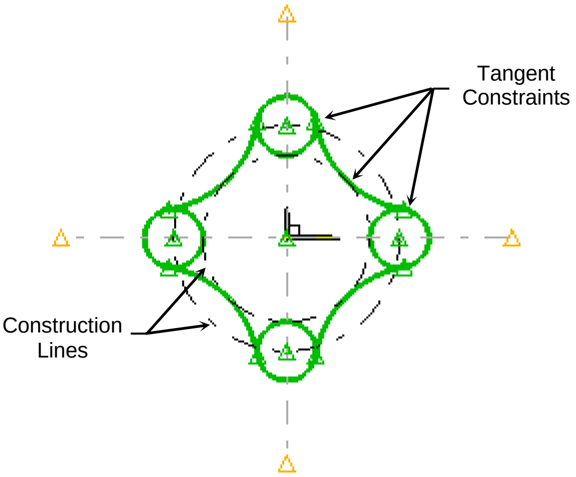
Draw a Ø13mm ***Circle*** centered at the intersection of the Ø50mm circle and the vertical sketch line. To detect the intersection while drawing the circle, click the left button in an empty zone and drag the cursor over the intersection point. When the intersection is found, the two intersecting entities become red.

Repeat this operation to draw a Ø13mm circle at each intersection.

Draw a ***Circle*** centered at the origin of the absolute CSystem and tangent to one of the four Ø13mm circles. To calculate the tangent point, click one of the Ø 13mm circles.

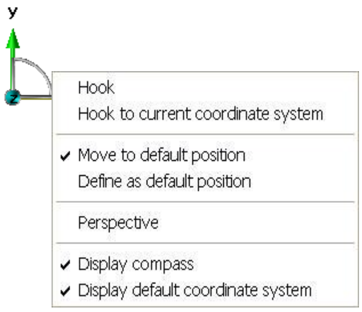
Draw four tangent arcs.
With the ***Circle*** function, switch the option from CENTER to PASSAGE. Then click the three entities the arc is tangent to.

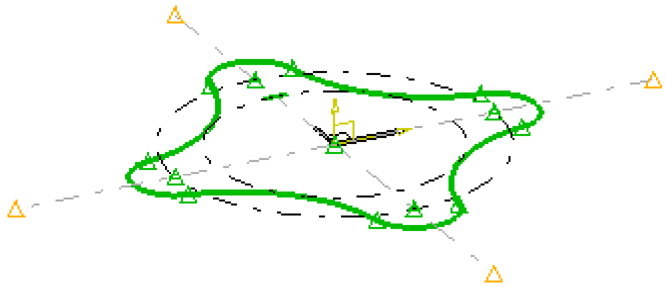
Define the two circle centered on the origin as construction entities. Construction entities are shown in fine mixed line.



To finish the contour, ***Trim*** the Ø 13mm circles, switch the mode to KEEP then click the circles.

Switch to the perspective view by clicking the compass with the third button. Choose the Perspective option in the menu.

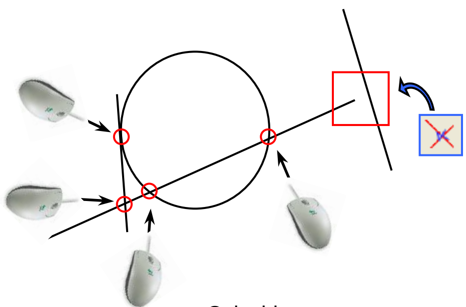




To finish, ***Validate the sketch.***

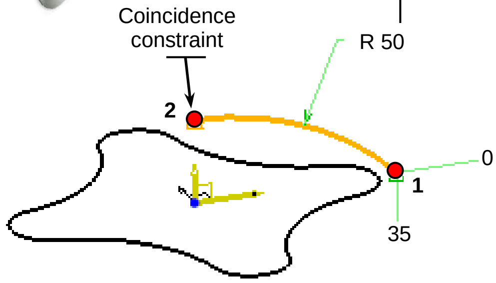
Construction point

- Automatic creation of intersection points can only be found if there is a true visual intersection.



Current CSystem - Click on the yellow coordinate system and make it current

Start a new sketch.



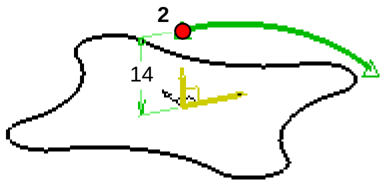
Draw a 50mm radius ***Arc*** passing from point (1) to (2), switch the option from CENTER to PASSAGE.

Dimension the point (1).

Place a coincidence constraint between the Y axis of the yellow CSystem and the point (2).

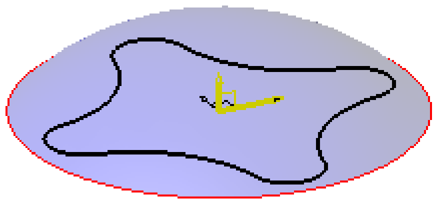
The contour remains orange, and it is possible to move the point (2) along the Y axis of the yellow CSystem.

Place a dimension between the X axis of the yellow CSystem and point (2). The contour turns green.



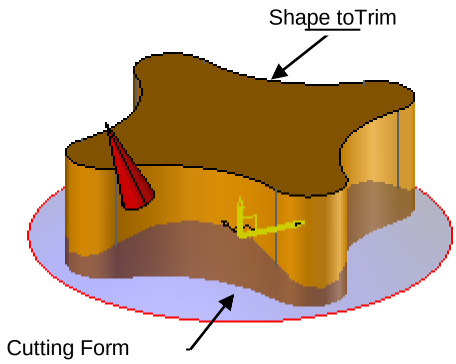
Create a ***Revolved shape*** by sweeping the arc 360° around the Y+ axis.

.

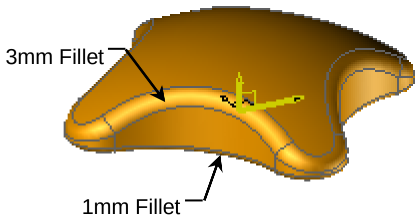


Extruded shape - Extrude the profile of the knob 20mm upwards

Trim the extruded shape with the revolved one. After selecting the cutting form, select the side to delete with the red arrow.



Create 3mm and 1mm fillets on the top and lower edges of the part as shown.



<u>Forme Extruder</u>

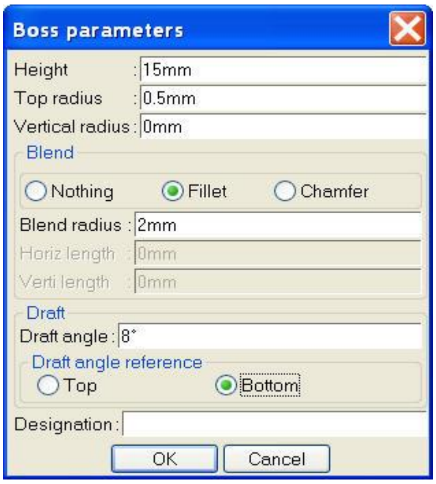
- When creating an extrusion, you can extrude a contour to another face by switching the mode options from Mode = HEIGHT to Mode = TO.
- In the advanced options >> of the ***Extrude*** function, you can place a draft angle on all vertical sides and also specify an offset from the starting curves.

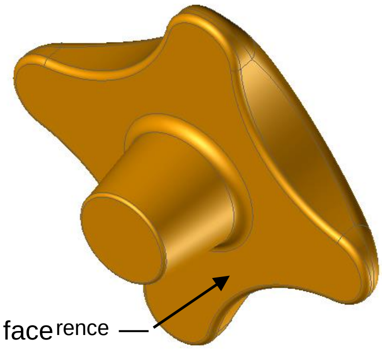
Current Coordinate System – select the option "Named Coordinate system" = ABSOLUTE CSYSTEM

Start a new sketch.

Draw a Ø15mm ***Circle*** centered on the origin of the absolute CSystem.

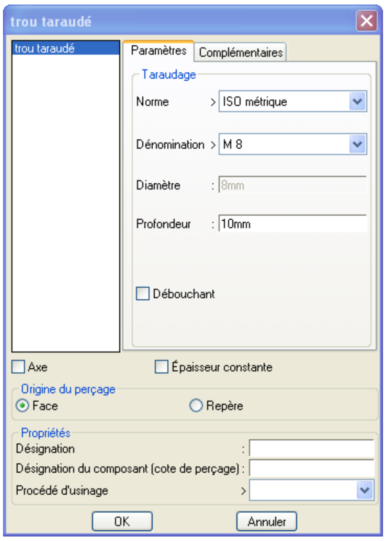
Boss 15mm from the lower face, top radius of 0.5mm, blend radius of 2mm and a draft angle of 8°





Reference face rence

Create a ***Threaded Hole*** M8 concentric to the boss, stay in DYNAMIC mode. The thread depth is 10mm.



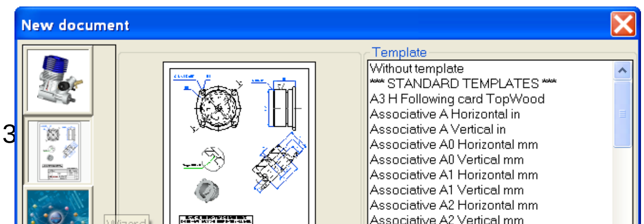
Assembly / Define Part – set designation = « Knob », also set the material of the knob to bakelite.

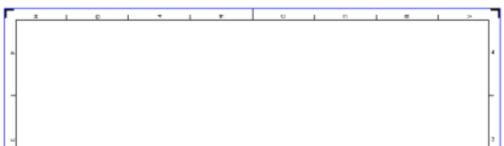
Save the file in the training folder with this name : « Knob.TOP ».

Analysis

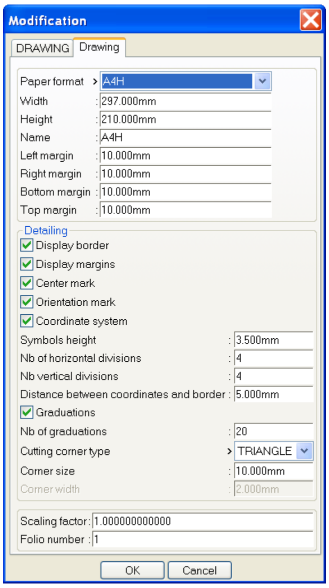
- Now that the material is known, it is possible to carry out a ***physical analysis of the part*** such as part weight, volume and its surface area
- Note the ***Analyze / Weight, Surface, Volume*** functions display the results in the alphanumeric bar. The number of decimal places is 3 and can be adjusted in the state bar DEC=3. This option doesn't affect dimensions

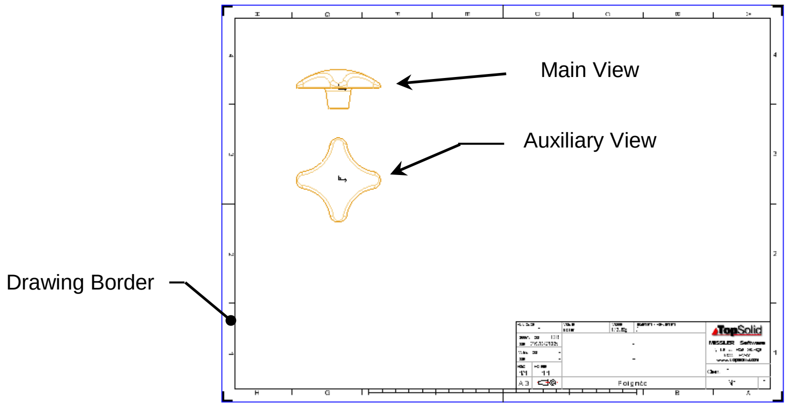
Create a ***New*** TopSolid'Draft document
Select « Associative A3 Horizontal mm» from the template list.

3 


are

Place the part on the draft using ***Main View***
Then create an AUXILIARY VIEW to give the following.





Use ***Modify Element*** and click on the border to change the Paper format to A4H

Use ***Move Parents*** reposition the views.

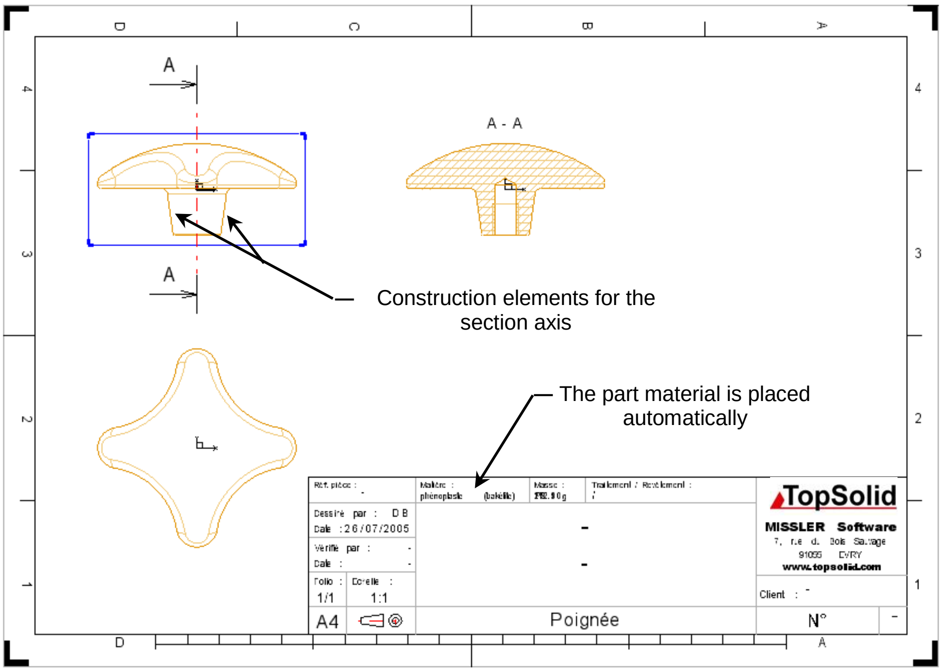
DFT Format

- The dialogue box where you change the paper format, also allows you to change the scale factor. It is this scale factor which appears in the title block.

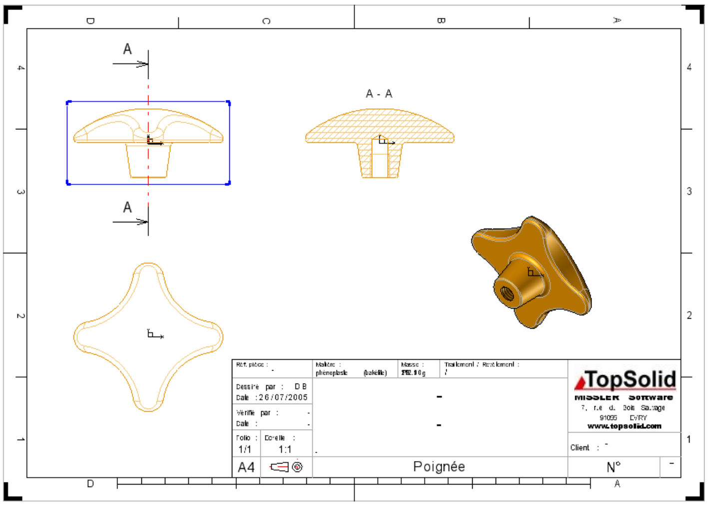
- You can customise the layout of a draft document, such as border, scale, title block and then save this document as a template file. If you save the file in the directory ...c\V69\Local\englishUS\template\ the file will appear in the list of templates when you start a new drawing.

Create an axis on the cylinder of the boss using the ***Axes*** function. Choose the option AXIS BETWEEN TWO ELEMENTS and then pick the 2 drafted edges of the boss. We will use this axis to create the partial section.

Make a ***Partial section*** using the axis A-A



Project an other auxiliary view from the main view as shown below.



Save the file with the name: « Knob.DFT ».

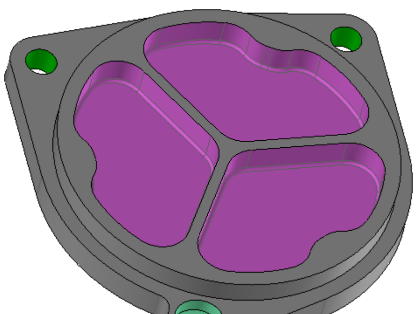
<u>Mise en plan</u>

- When you move the main view, the auxiliary views move with it to maintain allignement.
- If the 3D model file is closed, you can use the function ***View / Edit*** Model to automatically open the it.

Exercise n°6 : Cover

In this exercise you will Use:

- Construction lines and circles
- Dimensioning
- Sketch Extrusions
- Link types in the contour function



- Extruded shape
- Boss function
- Pocket function
- Propagation operation
- Coloring operations

Create a new TopSolid'Design document.
Choose among the templates « No template ».

Activate the ***Sketch*** menu**.**

Start a sketch.

Create a 60mm radius Circle on the origin of the CSystem.

Draw a ***horizontal*** and a ***vertical*** sketch line passing through the origin of the CSystem

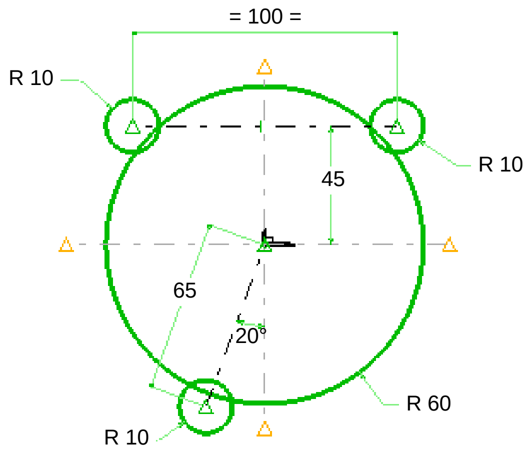
Create horizontal ***Line*** , and ***dimension*** its length 100mm, and its position 45mm from the X axis.

Use the ***Modify Element function*** to place a symmetrical constraint on the 100mm dimension about the Y axis.

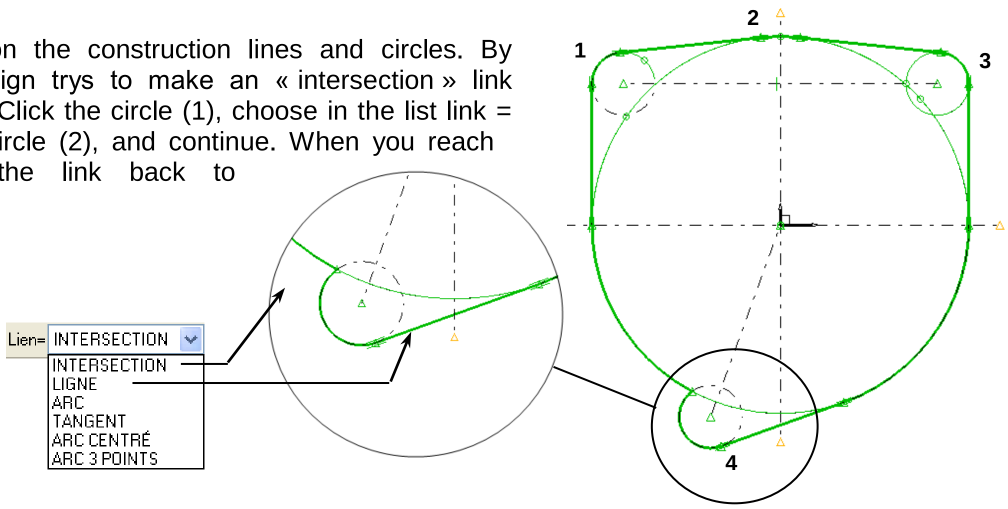
Create an oblique line length 65mm, 20° to the Y- axis.

Create 3 ***Circles*** of 10mm radius at the end of each lines.

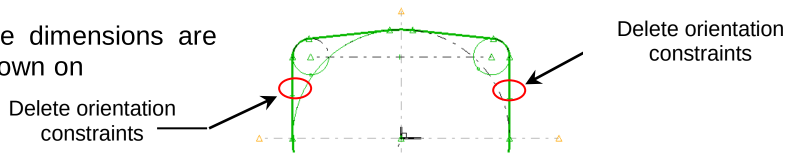
Switch all the lines and circles to construction elements.



Create a ***Contour*** on the construction lines and circles. By default, TopSolid'Design trys to make an « intersection » link between the entities. Click the circle (1), choose in the list link = LINE then click the circle (2), and continue. When you reach circle (4), switch the link back to INTERSECTION.



The contour is over-constrained, and some dimensions are red. ***Delete*** the orientation constraints as shown on the diagram,.

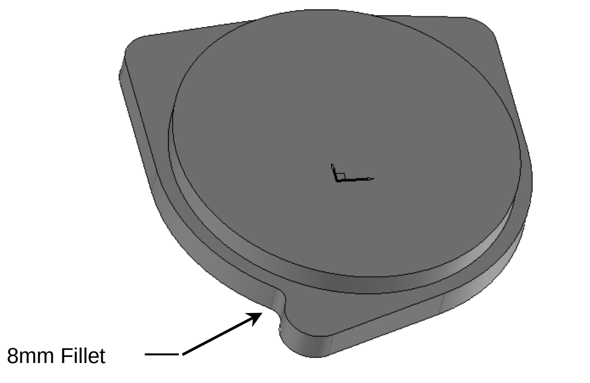


Extrude the profile 10mm in Z+ direction.

Create an 8mm fillet.

Start a new sketch

Draw a 55mm radius ***Circle***.

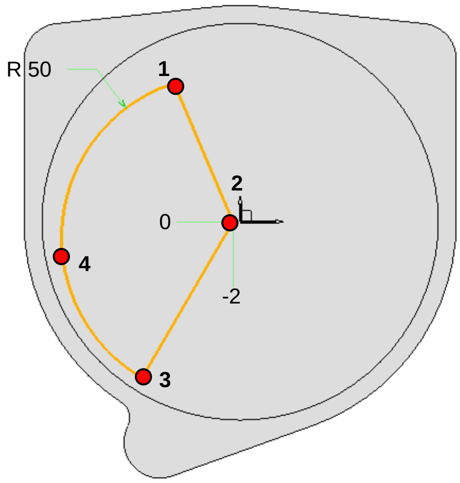


Use this circle to make a ***Boss*** 8mm high.

Start a new sketch

Create a new ***Contour*** passing through the points (1), (2) and (3). To close the contour with an arc choose a 3 POINTS ARC link, click an intermediary point (4) and then point (1).

Adjust the radius dimension to 50mm

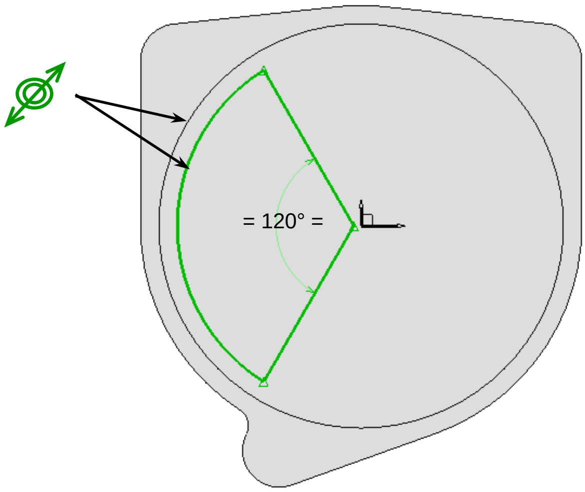


Dimension the angle between the two lines of 120°

Place a symmetrical constraint on this angle about the X axis.

Place a concentric ***Constraint*** between the arc and the boss edge.

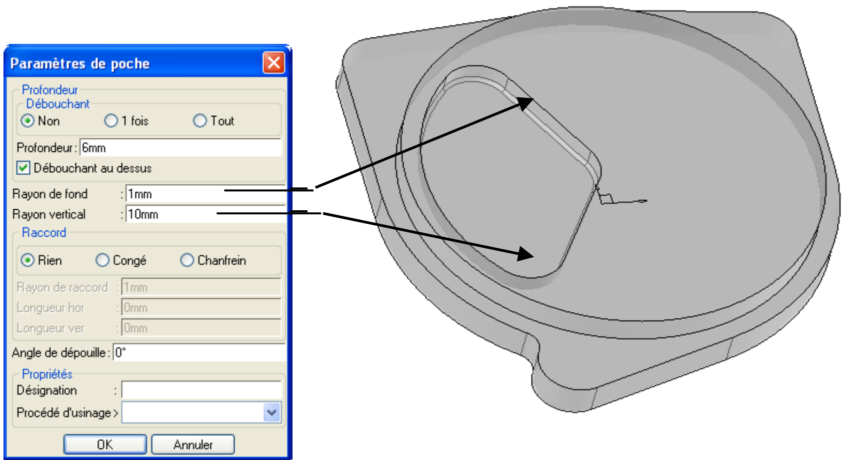
To finish, ***End sketch***



<u>**Operations on Functions**</u>(drilling, pocket, boss, fillet …)

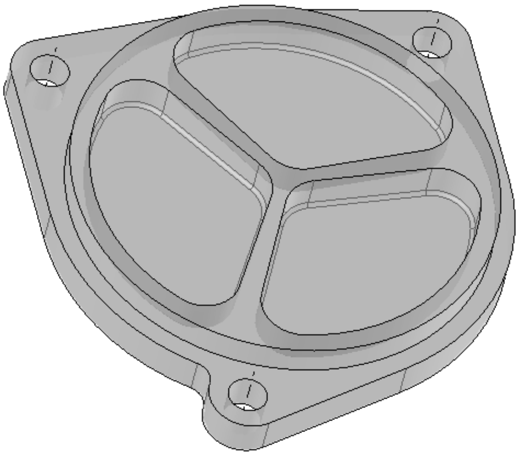
- To modify the characteristics of an operation, use ***Modify Elements.***
- To delete an operation, use ***Extract Element***.
- To copy an operation, use the ***Propagate Operation*** function.

Make a 6mm deep pocket from the top face of the part



Create a CIRCULAR ***Propagation*** 360° about Z+ axis of the pocket.

Create three boring ***Holes*** of Ø 10mm.

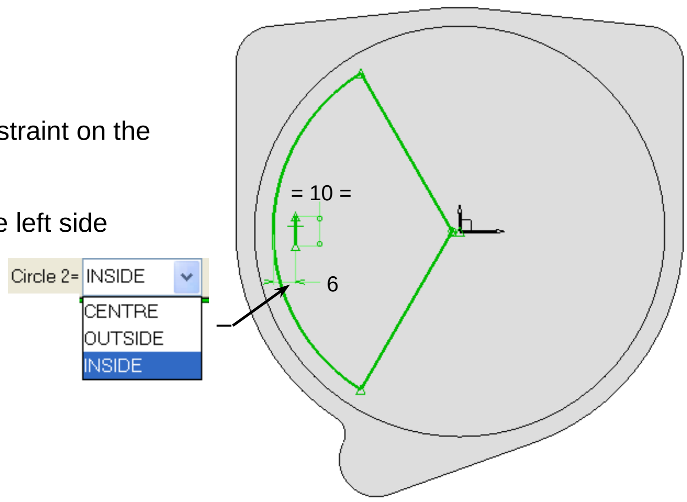


Modify the sketch used to create the pocket.

Draw a vertical ***line***.

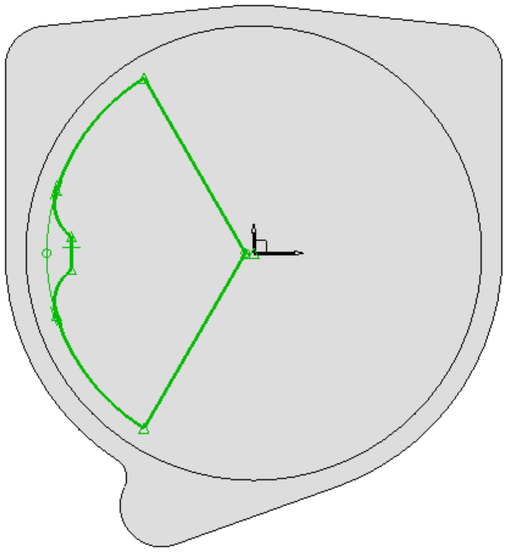
Dimension the line and place a symmetrical constraint on the dimension about the X axis.

Place a 6mm dimension between this line and the left side of the circle. By default TopSolid'Design dimensions from the center of the circle, choose INSIDE from the list before placing the dimension.



Pockets and Bosses

When creating a pocket or a boss, the z position of the profile used is not important. The depth or height is determined by the reference face chosen. If the reference face is at an angle, it is necessary to position the profile in the z direction to be either the bottom of the pocket or height of the boss. In this case use the option PROFILE IN PLACE.

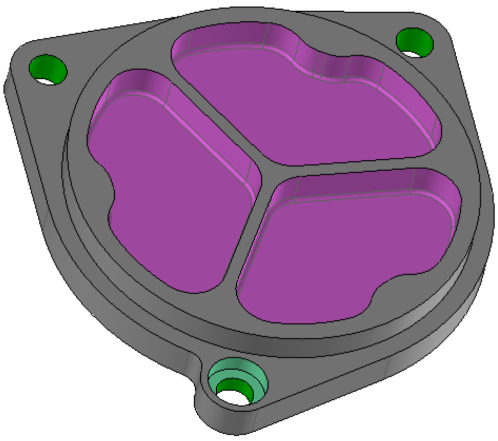


Draw two 10mm ***Arcs*** passing through the extremities of the line and tangent to the circle.

Trim the contour to give the following result.

End Sketch, the changes are automatically applied.

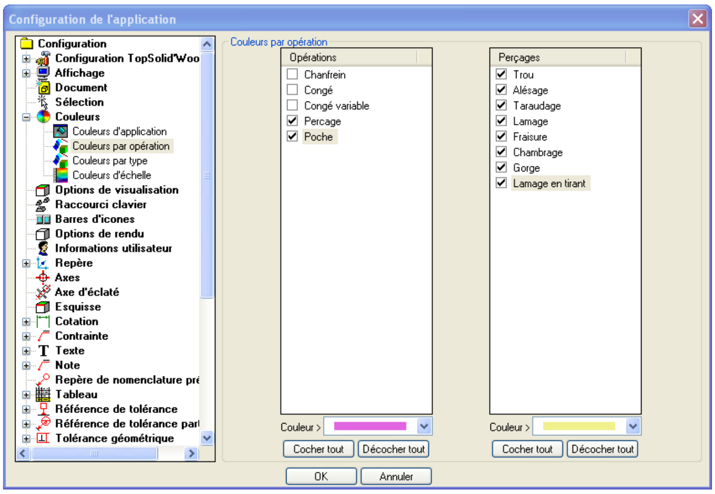
Replace the simple hole shown below by a counter bore hole, using the ***Modify Element*** function on the drilling. In the dialog box, click on the REPLACE MODEL button, and select « hole+counterbore ». The counter bore is Ø 16mm and 5mm deep and the hole is Ø 10mm.



To distinguish the different operations, ***Color*** the faces of the pockets and the holes.

The ***Color*** function in the Shape menu allows you to manually change the colors.

Use the color setting in the ***Tool / Option / Colors / Color by operation*** to set automatic detection. Tick the operations you wish to color.



Use ***File / Regenerate*** to update the model. Validate THIS DOCUMENT

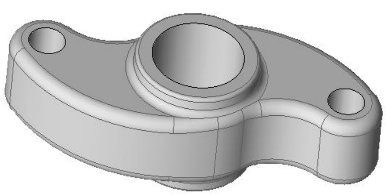
Type « Cover » as the part definition; use the ***Assembly / Define part*** function.

Save the file in the training folder with the name: « Cover.TOP ».

Exercice n°7: The Roller

In this exercise you will :

- Create a contour by tracing over sketches
- Use Cartesian dimensions



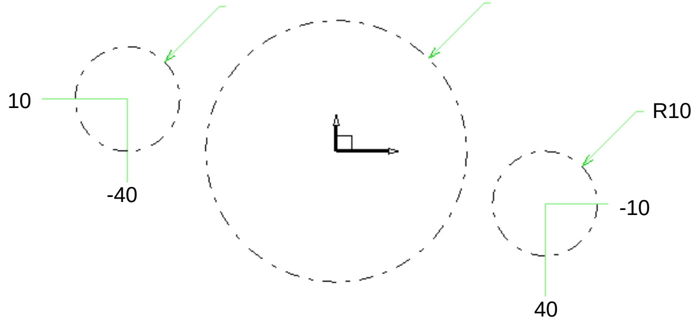
- Extrude a shape
- Use the boss, pocket and drilling operations
- Use the Propagate command

Crate a ***New*** TopSolid'Design document.
From the standard models select « Without template" model ».

Activate the ***sketch*** mode***.***

Begin a sketch.

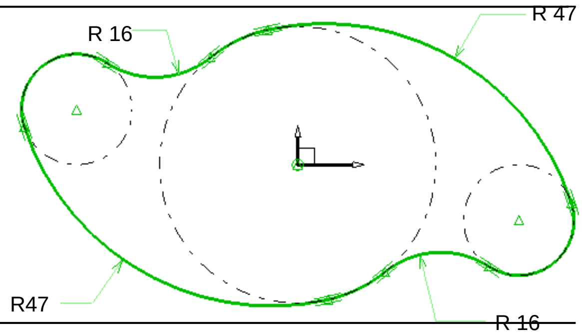
Trace three ***Circles*** as shown.



Dimension, using Cartesian coordinates, the center of the 10 mm radius circle from the axis of the CSystem.

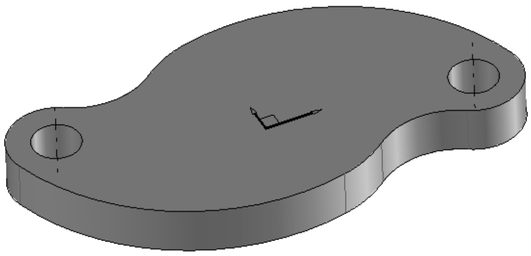
Transform the circles into construction elements.

Construct the ***Contour*** by clicking the circles.
The link between the circles is ARC



Extrude the contour 10mm using alignment=CENTERED. This will create a symmetric extrusion on both sides of the contour.

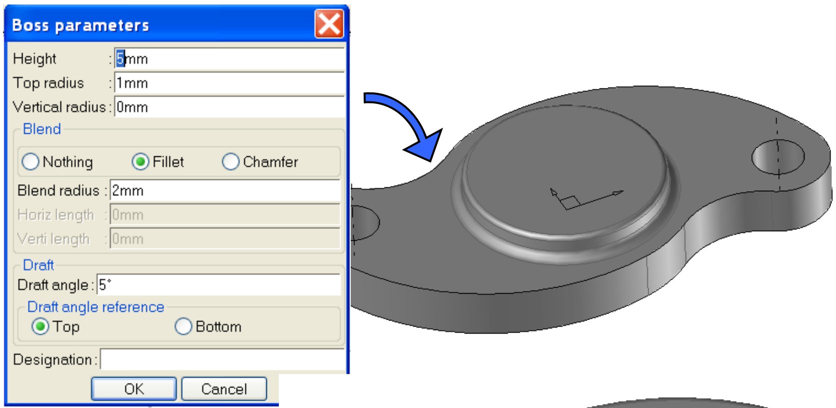
Create two boring holes of Ø10 mm. Use the mode = NON DYNAMIC.



Start a sketch.

Draw a ***Circle*** of radius 20mm centered on the origin of the CSystem.

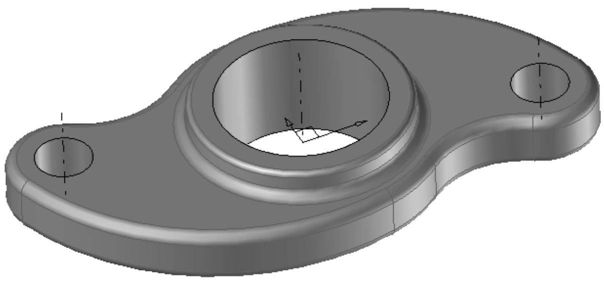
Create a **Boss** from thee circle, using the values shown.



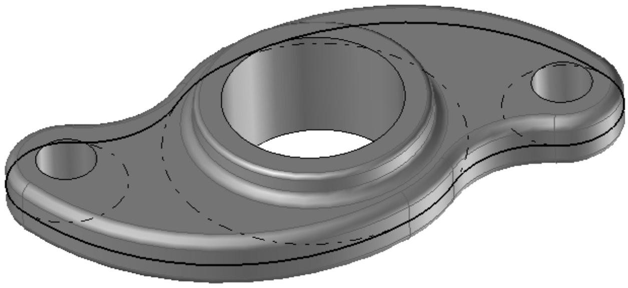
Propagate the boss using Simple Mirror about the XY plane

Bore a Ø30mm hole using ***Drilling*** DYNAMIC at the center of the boss.

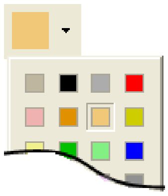
Add 2mm fillets on both surfaces.



To show or hide control elements, use the ***Control Element*** feature**.** Click, on an element to treat, in this case the vertical surface of the 10mm extrusion. TopSolid'Design will make the contour appear along with its dimensions. To hide the dimension click again but this time on the contour.

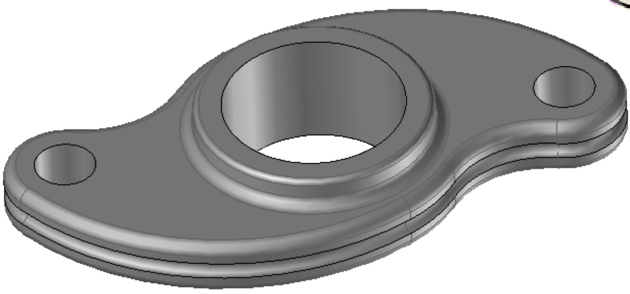


Change the current color in the menu ; choose orange from the list.



Create a wire shape Ø 3mm along this profile. Use the ***Pipe*** function, from the list choose the option Pipe = TUBE SHAPE then click the contour.

Substract the orange pipe shape from the part.

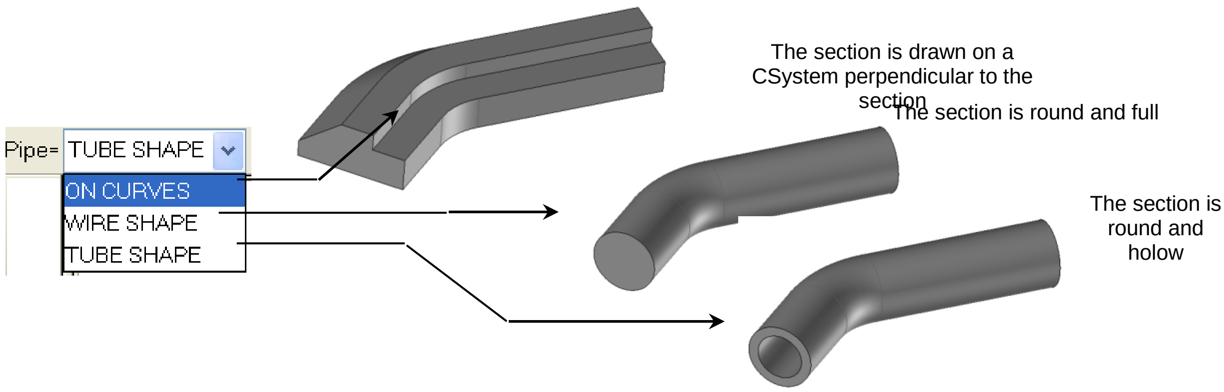


Type « Roller » as part definition, use the ***Assembly / Define Part*** function : « Roller.TOP ».

Save the file in the training folder with the name : « Roller.TOP ».

Pipe Shapes

You have three options when you create a pipe shape :



Create a TopSolid'Draft ***New Document***..
From among the standard templates choose « Associative A3 Horizontal mm ».

Change the plane format, switch to an A4 format with a scale of 1 :1.

Project the views as shown below..

Place an ***Axis*** on the main view, using the option PROJECTED AXIS.

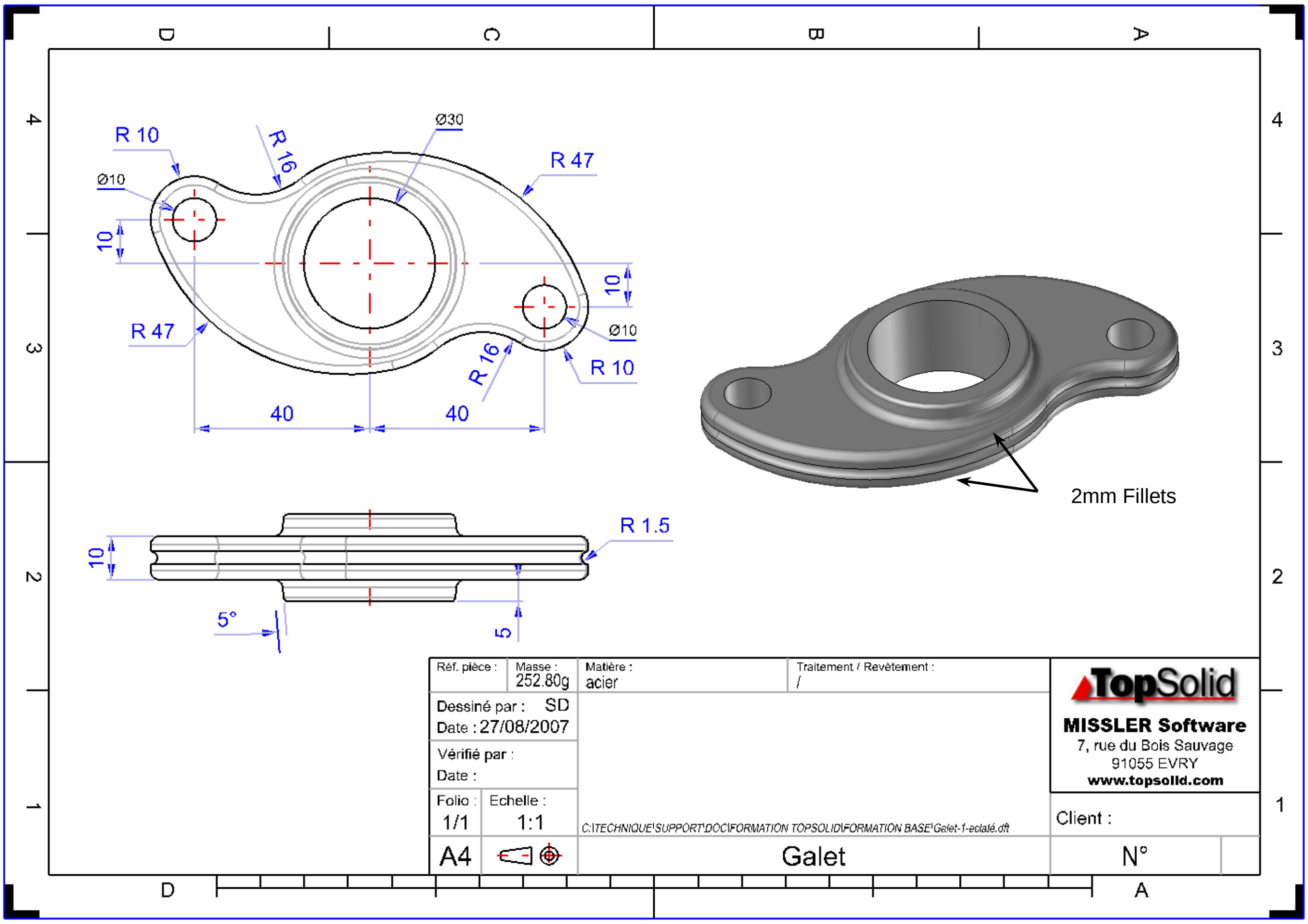
Dimension the different views.

Fast dimension for all standard dimensions (length, Radius, angle)

Drilling dimension for dimensioning holes

Draft dimension for the draft angle of the boss.

Add a ***Note*** for the fillets on the perspective



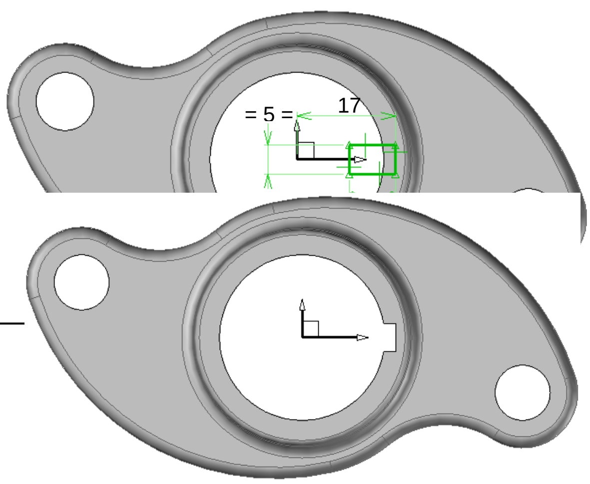
Save the file in the training folder with the name : « Roller.DFT ».

Use the ***View / Edit Model function*** to re-open the 3D model..

Start a sketch

Create a RECTANGULAR contour

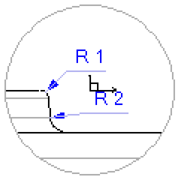
Constraint its position to the CSystem.



Create a ***Pocket*** for the key groove.

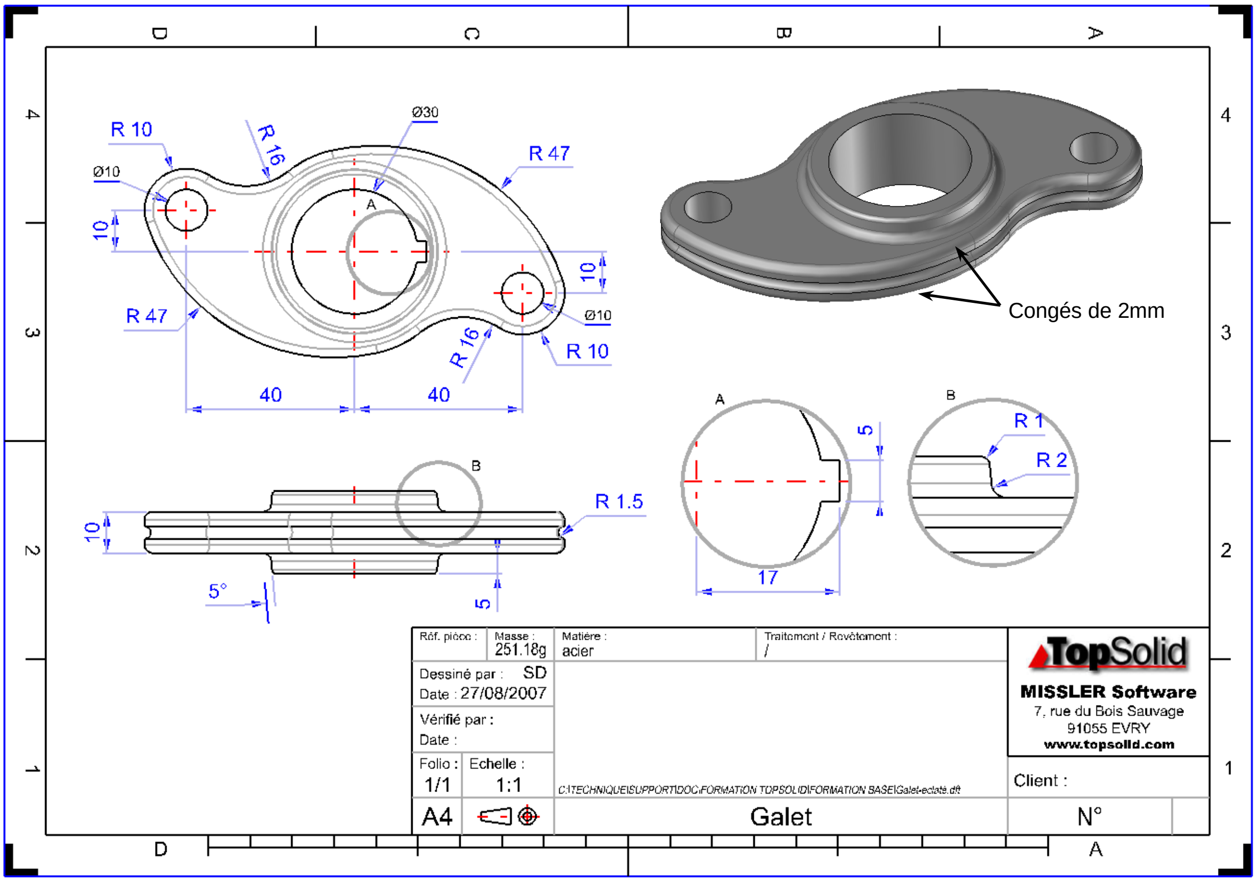
Save the file.

Open the draft file and note that it has been updated.



Create a ***Detail view*** at a scale of 2 :1 on the key groove and the boss

Add the missing ***Dimensions***.



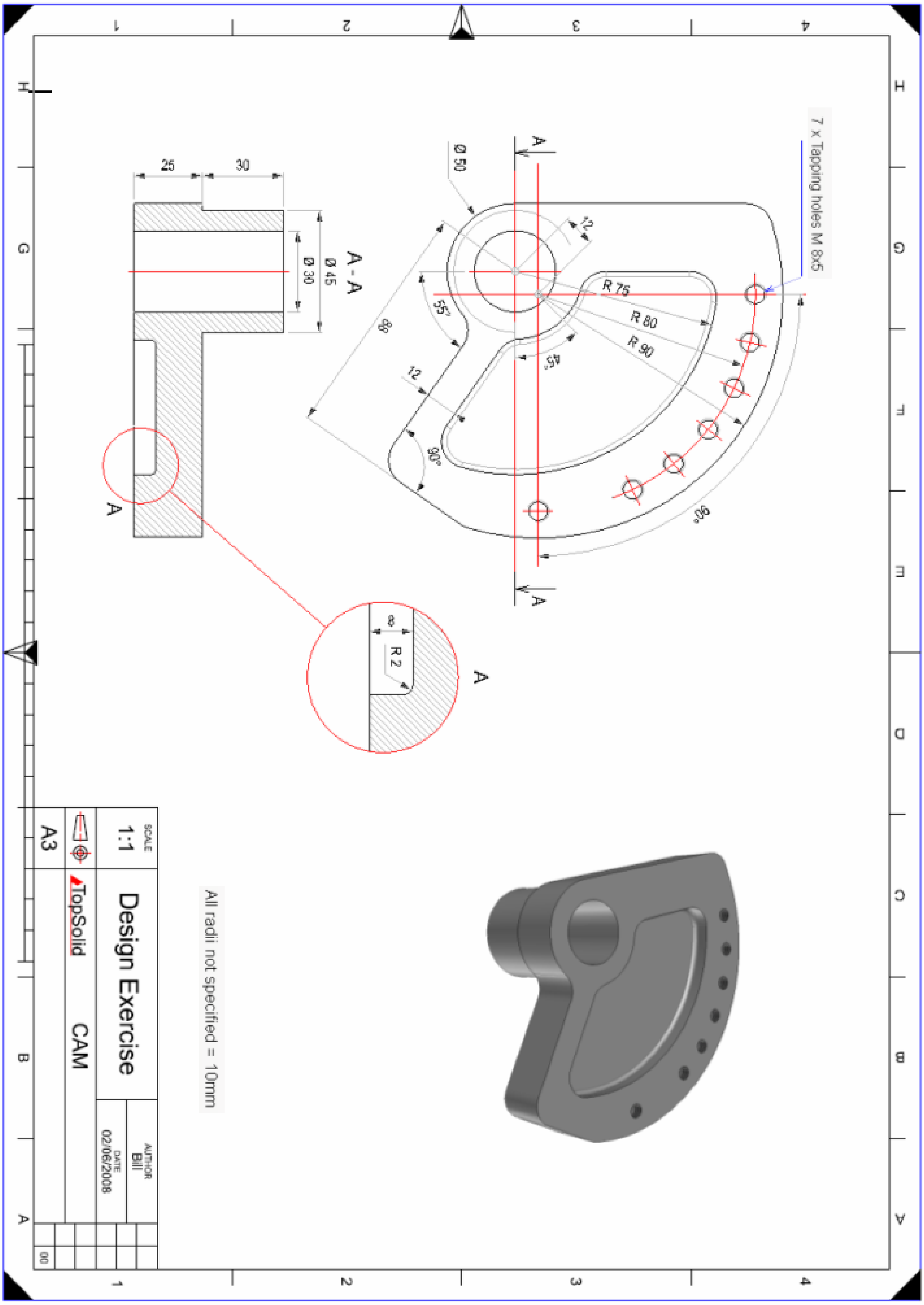
.

Save the file.

Exercise n°8 : Cam

Key Functions:

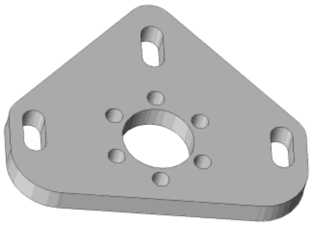
- Contour in sketch mode.
- Drilling on face and point in mode NON DYNAMIC
- Profile Extrusion
- Pocket operation
- Fillets and chamfers



Exercise n°9 : Bracket Plate

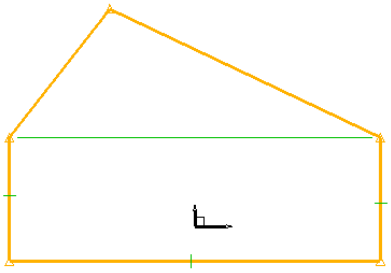
In this exercise you will :

- Construct a sketch
- Use Pilot dimensions
- Adding constraints to a contour
- Duplicate (Copy) a contour
- Create a pocket
- Drill holes on a face and at points
- Propagate (Copy) operations



Create a TopSolid'Design ***New Document***.
Choose « Associative model ». from among the standard templates

Draw a ***Contour*** using points with automatic constraints of alignment and orientation

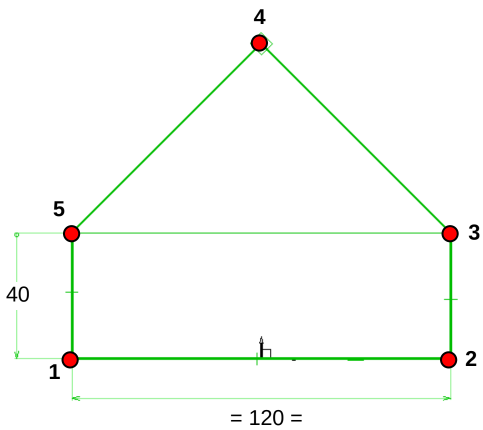


Dimension and ***Adjust*** the dimensions.

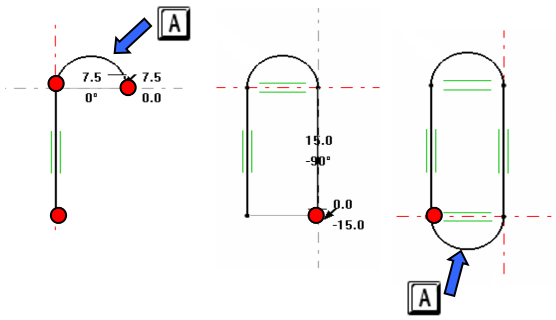
Add the following ***Constraints***

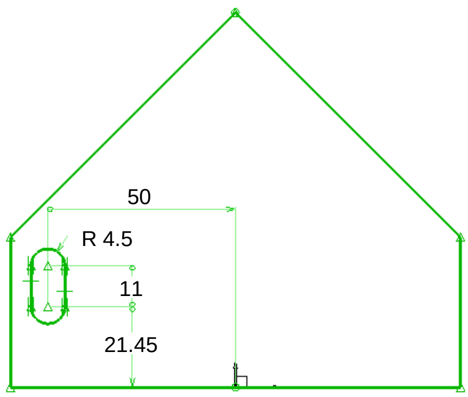
Align segment [1,2] with the X axis.
Make point (4) coincident with the Y axis.
Make the segments [3,4] and [4,5] perpendicular.

Add a symmetry constraint on the 120mm dimension about the Y axis.



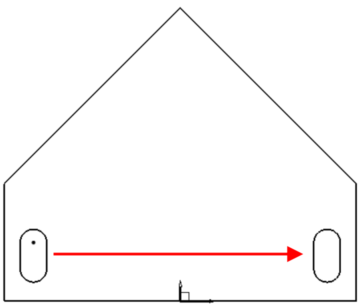
On the same sketch, create an oblong profile.
Start the 'oblong with a vertical line, then press « A » on the keyboard to make a tangent arc. Continue the contour as shown below



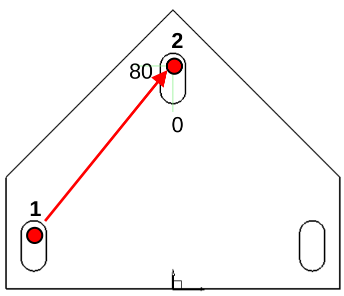


Dimenson the oblong profile.

Copy the oblong contour using PLANE SYMETRY, and click the YZ button.



Symetric about the YZ plane



Translation between 2 poins

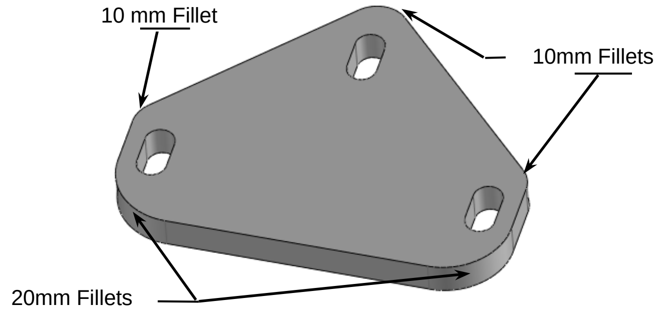
Copy the oblong contour by TRANSLATION between two points. The starting point is point (1), and the end point is point (2) with coordinates (0,80).

Use a ***Dimension*** to constrain the position of point(2).

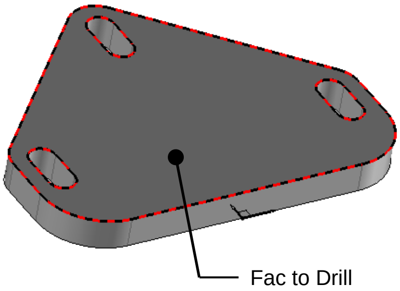
Validate the sketch.

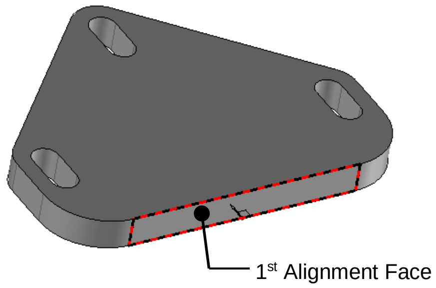
Extrude the outside contour by 10mm in Z^+.
The Mode Sketch = GLOBAL allows all sketches to be extruded in one go.

Add 10mm fillets on the part, as shown.

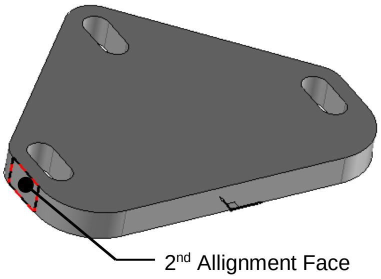


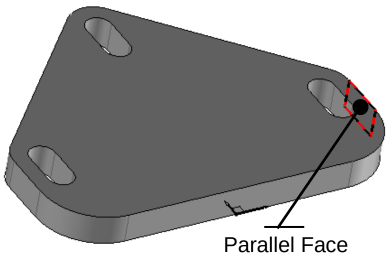
Create a simple Ø30 mm hole. Switch to NON DYNAMIC mode and select reference surfaces for the position of this **drilling**.



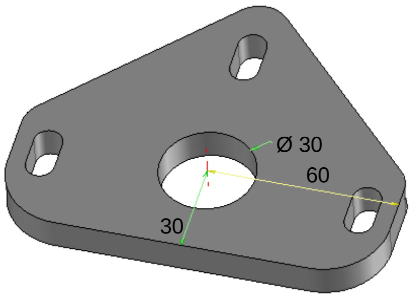


After clicking the first alignment surface, specify a distance of 30mm. The direction of the red arrow indicates the direction of the hole in relation to the alignment surface.

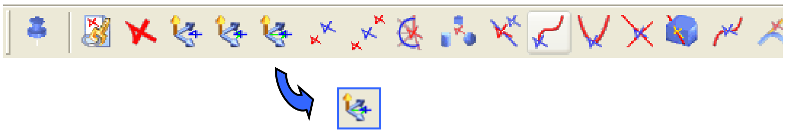




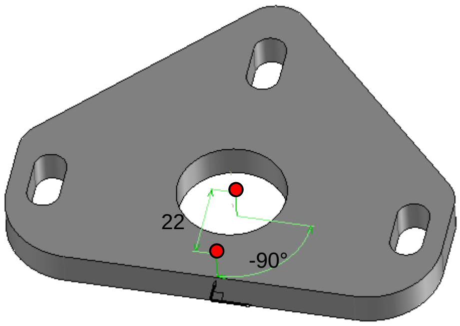
After clicking the 2nd alignment face, do not specify a distance; instead click the opposite parallel face. In this case the hole will be constrained at the middle of the two faces.
.



Create a polar **Point** with the following characteristics

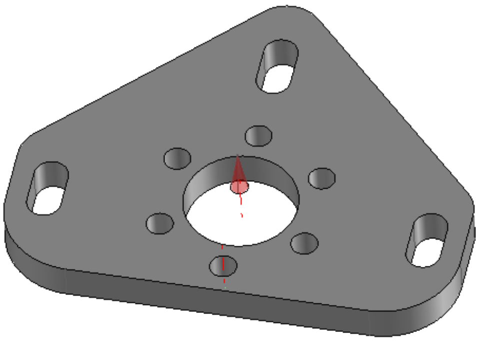


- Its origin is set at the center of the drilling
- Its X direction is X+,
- Its polar radius is 22mm,
- Its polar angle is -90 °.



Create a boring hole Ø7mm in NON DYNAMIC mode on the polar point.

Carry out a CIRCULAR **Propagation** of the Ø7mm hole about the axis of the Ø30mm hole. (Do not directly use the Z axis of the current Csystem, as in this case it is not correctly positioned). Total angle is 360° and the Total number is 6.

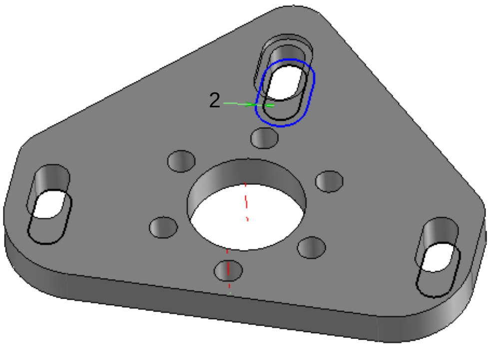


Make the **Control elements** of the pockets visible

Create a **Parallel offset** 2mm from the exterior on the upper profile.

Make a 2mm deep **pocket** on this parallel profile, with the upper surface as the reference surface.

Make the **Control elements** of the pockets invisible



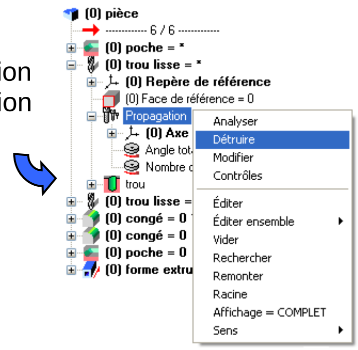
Define the part with « Bracket Plate » as part definition, using the ***Assembly / Define Part*** function.

Save the file in the training directory with this name : « Fixation plate.TOP».

Propagation d'opération

- To modify the NUMBER OF INSTANCES, the TOTAL DISTANCE, or EXCLUDE INSTANCE of a propagation operation use **Modify Element**.
- To delete a propagation, you must open the construction tree and delete the propagation. The **Extract** function removes both the propagation and the operation.

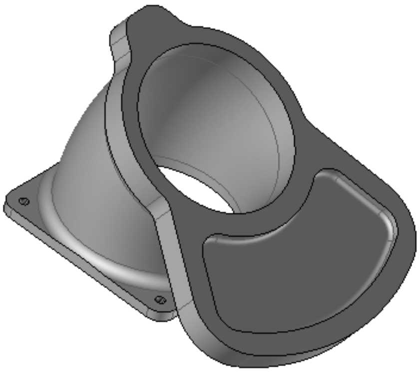
.



Exercise n°10 : The Bend

In this exercise you will:

- Construct a sketch
- Create Pilot dimensions
- Add constraints to a contour
- Copy edges
- Create pockets with vertical fillets
- Extrude using the Global mode
- Create a Csystem
- Propagate operations
- Join features with union
- Profile a shape
- Thicken



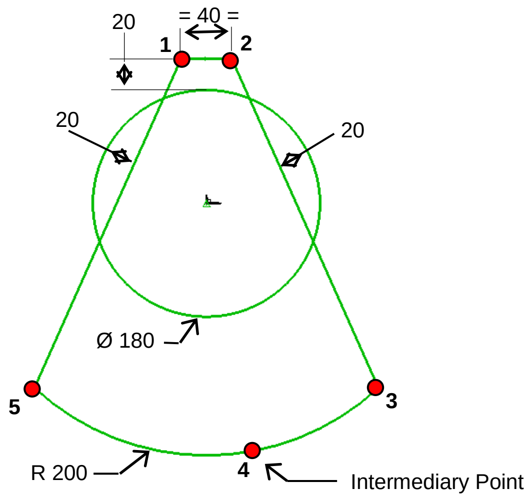
Create a TopSolid'Design ***New Document***.
Select « no template » from among the standard templates

Activate ***sketch mode.***

Start a sketch.

Draw a Ø 180mm ***Circle*** centered on the origin of the CSystem.

Build a ***Contour*** using points ; between the points (3) and (5) choose from the list the ARC 3 POINTS link

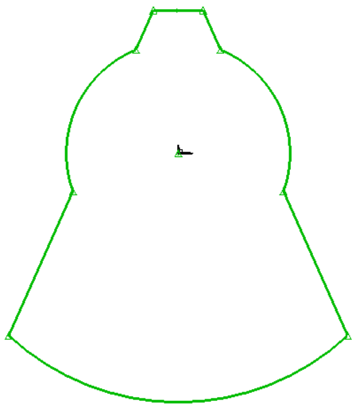


Dimension and ***Adjust*** the dimensions.
For the three 20mm dimensions, TopSolid'Design by default suggests CENTER, instead choose INSIDE from the list.

Place a concentric constraint between the two circles.

Place a symmetric constraint on the 40mm dimension about the Y axis.

Trim the contours to obtain the following result.



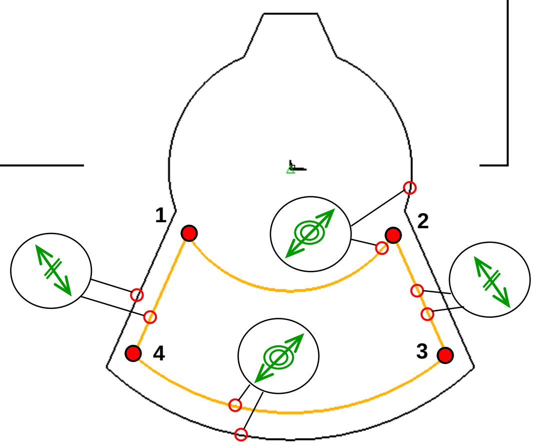
Merge the 20mm dimensions together. Use the Merge function under the Parameter menu.

Use ***Modify Paramter*** to changee the 20mm dimesnioons to 25mm

End Sketch.

Chaning a merged dimension

- To change a merged dimension click ***Modify Parameter,*** then pick the dimension, select REPLACEMENT and type in the new value.



Start a new sketch.

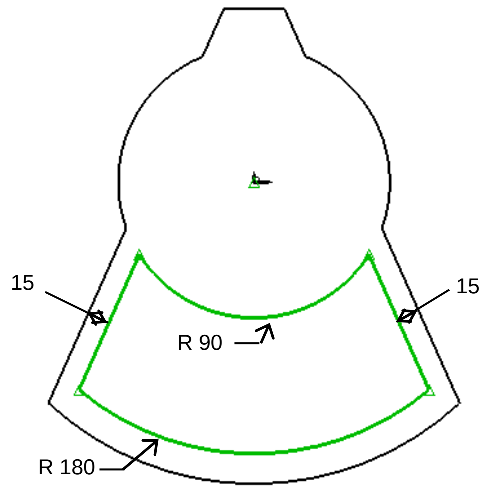
Create the following ***Contour*** using points. Between the points (1),(2) and the points (3),(4) choose ARC 3 POINTS from the list of options.

Add concentricity and paralellism ***Constraints*** as shown

Dimension and ***Adjust*** the dimensions. The sketch turns green, when it is totally constrained

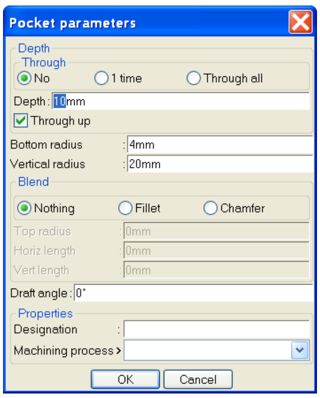
Extrude the exterior sketch 25mm by the Z- axis

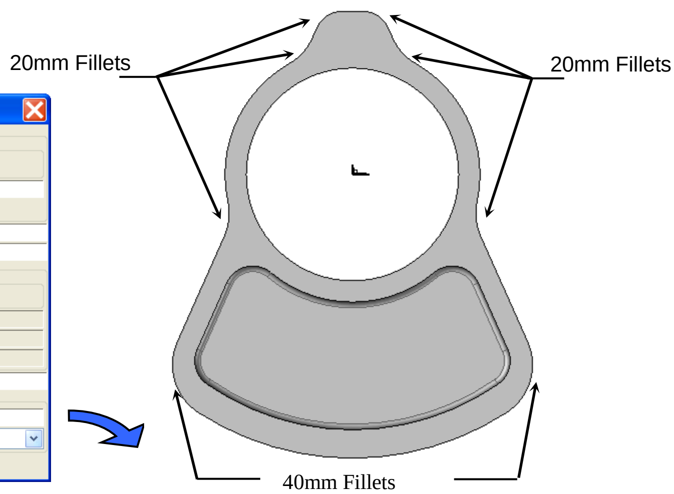
Drill a hole Ø 150mm at the center.



Fillet the vertical edges of the part as shown.

Create a 10mm deep ***Pocket*** on the lower surface of the part.
The vertical radius is 20mm and the bottom radius is 4mm.
.

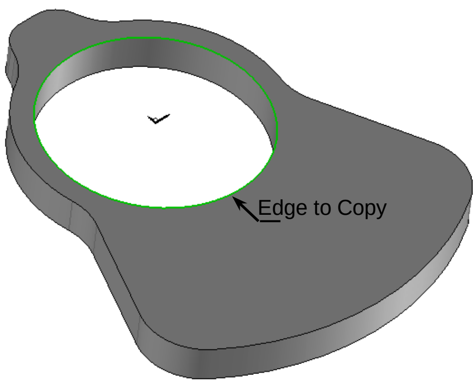




Start a new sketch.

Copy the edge of the drilled hole on the upper surface of the part**.**

Validate the sketch.



Create a CSystem perpendicular to the current one. Select ***Tools*** – ***Coordinate system*** - YZ SET AS CURRENT

Start a new sketch.

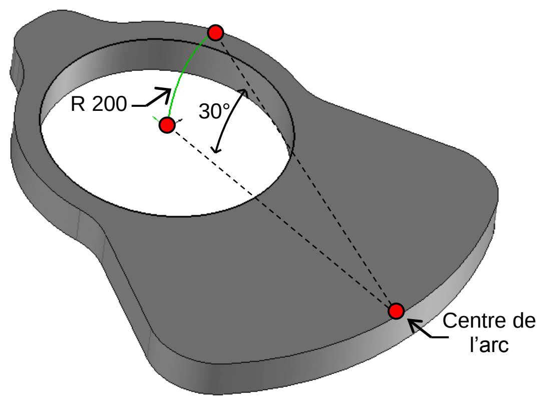
Draw a circle ***Arc by two points ,*** radius of 200mm passing by the origin of the CSystem.

Create the center point of the arc with the ***Point tool*** function.

Add a coincidence constraint between the center point and the X axis.

Add an angular dimension on the arc.

Validate the sketch.

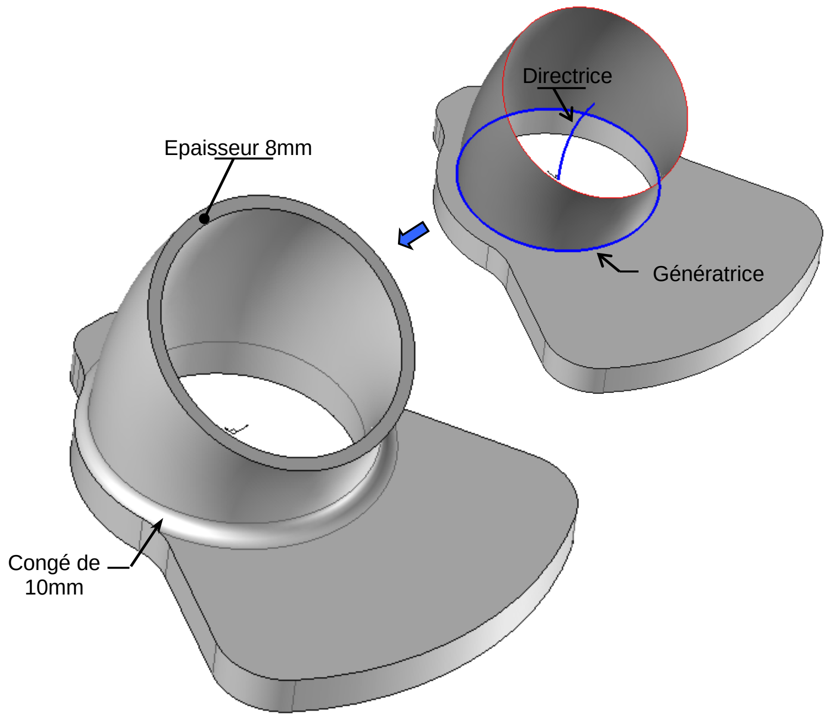


Create a ***Pipe*** with the arc as the guide curve and the circle as section curve. Choose Type= SURFACE TYPE.

Thicken the pipe surface to 8mm from the exterior.

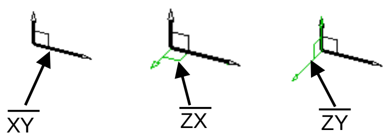
Unite the two Solids

Make a 10mm connection with the ***fillet*** function.

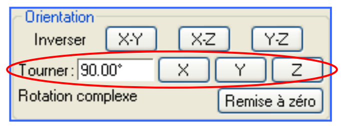


The Coordinated Systems

- Using the ***Coordinate system*** function in the tool menu , it is possible to quickly create two coordinate system perpendicular to the current one. The current Csystem is always named XY in TopSolid'Design.

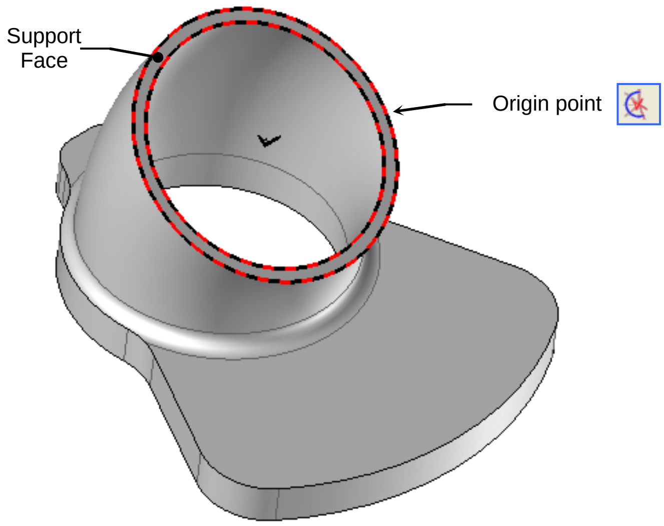


- To turn a CSystem, use the ***Modify Element*** function. In the dialog box choose a rotation angle, then click the X, Y or Z button. Note it is the axis of the selected Csystem, that is affected, not the current coordinate system.



Create a CSystem using a surface and point.

Select the upper surface of the bend and place its center on the outer circular edge.



Make current this CSystem.

Start a new sketch.

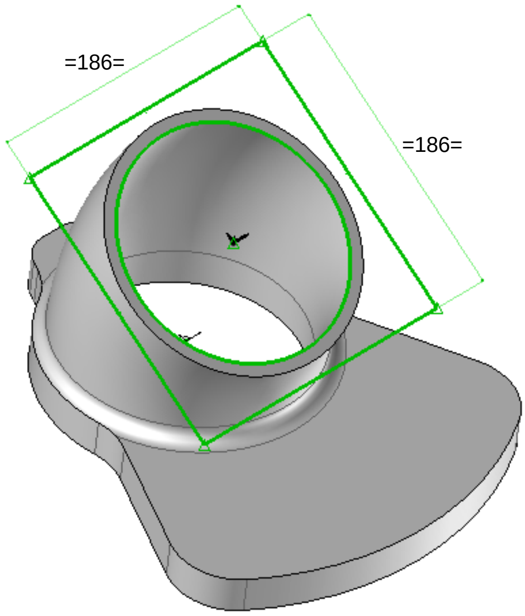
Copy the edge inside of the bend.

Draw a RECTANGULAR ***Contour***.

Dimension and ***Adjuste*** the dimensions.

Add ***Symetry Constraints*** on the 186mm dimensions of the square about the X and Y axis

Validate the sketch.



Extrude this sketch 10mm in the Z+ direction. When mode = GLOBAL is selected both the circle and the rectangle are extruded at the same time.

Create 4 vertical 20mm fillets

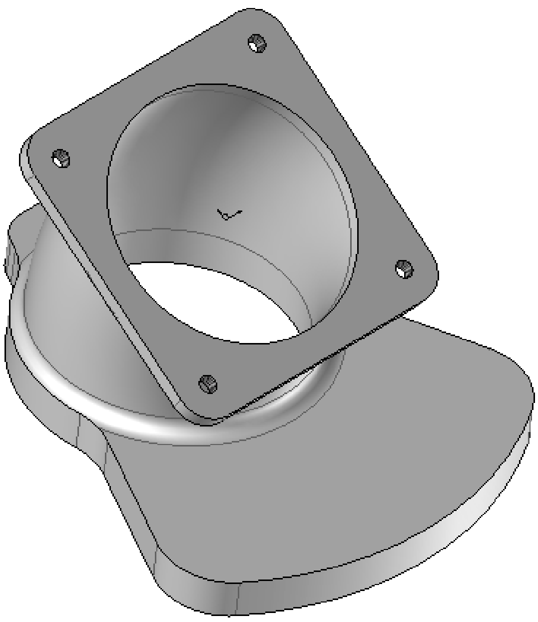
Make an M10 threaded hole concentric to one of the vertical fillets

Propagate the treaded hole using a CIRCULAR array 360° about Z axis.

Unite the two solids.

Assembly Define - Part
Define the part - using « Bend » as the definition

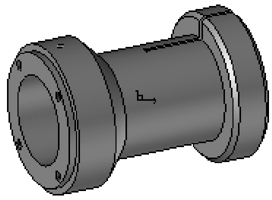
Save the file in the training directory with the name : « Bend.TOP».



Exercise n°11 : Coupling

In this exercise you will;

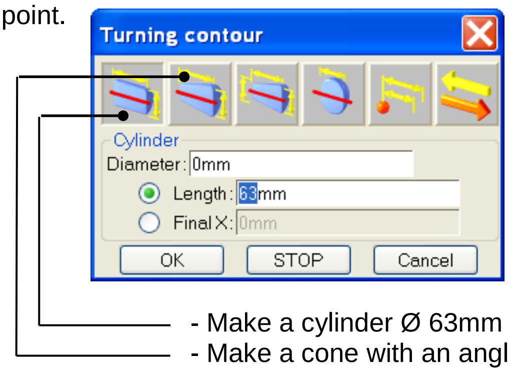
- Construct a contour using points
- Use parametric dimensioning
- Create a Revolved Protrusion
- Place drilled holes on a face usind a point and CSystem
- Create a Groove

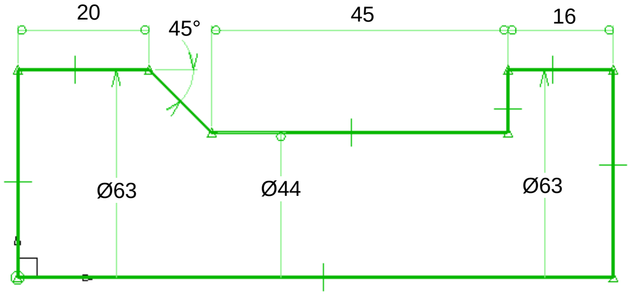


Create a New TopSolid'Design.
Choose « Associative no template »from the standard templates

Begin a new sketch

Creare a contour using points specifiying the TURNING option. Use the origin of theCSytem as the starting point.





- Make a cylinder Ø 63mm with a length of 20mm OK
- Make a cone with an angle -45° on a Ø 44mm OK
- Make a cylinder Ø 44mm with a length of 45mm OK
- Make a cylinder Ø 63mm with a length of 16mm OK

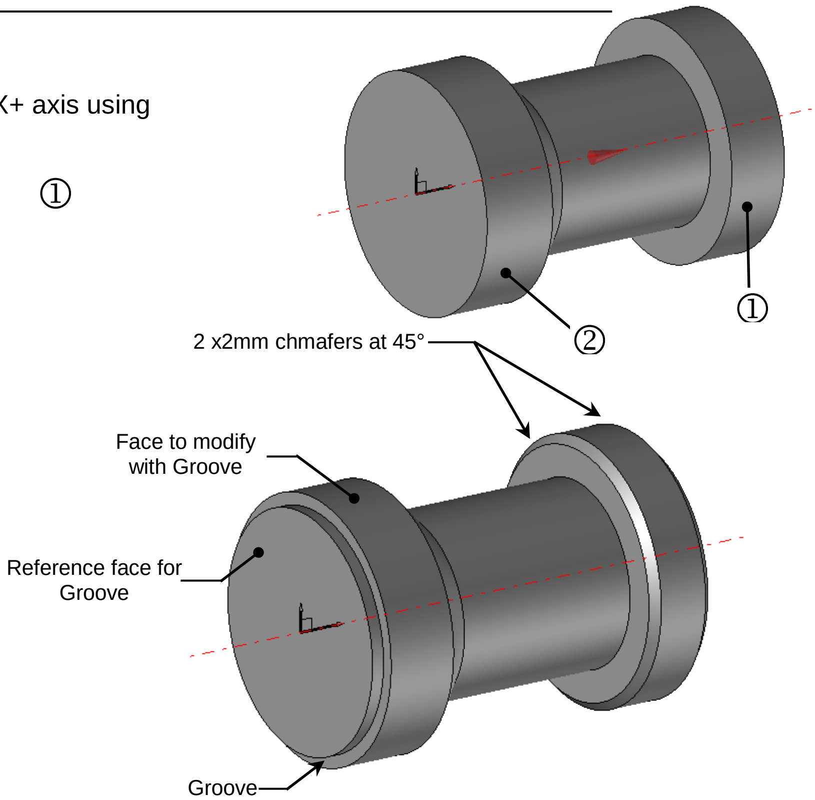
Click STOP and designate the profile to close the contour.
The contour is automatically dimensioned.

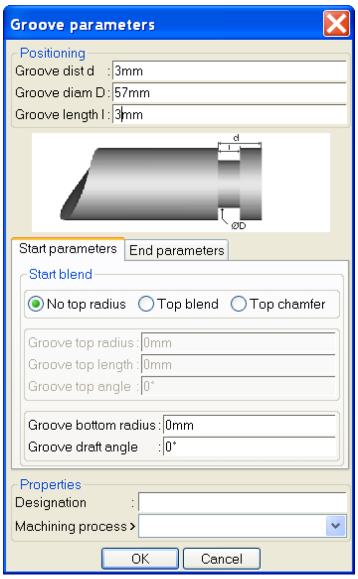
End Sketch.

Create a revolved protrusion about the X+ axis using the function ***Create Revolved Shape***.

Place 2 x 2mm x 45°chamfers on face ①

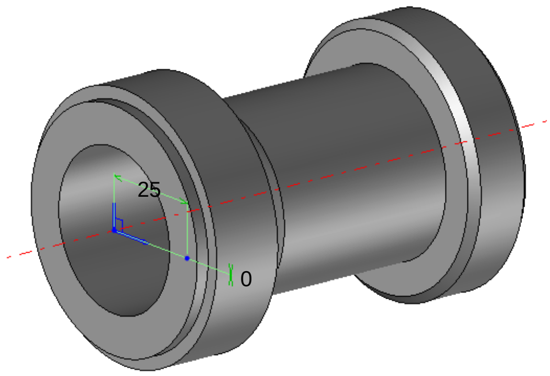
Make a Groove on face ②
Using the ***Shapes/ Groove menu***





Create a Boring Ø38mm H7 through the center of the part

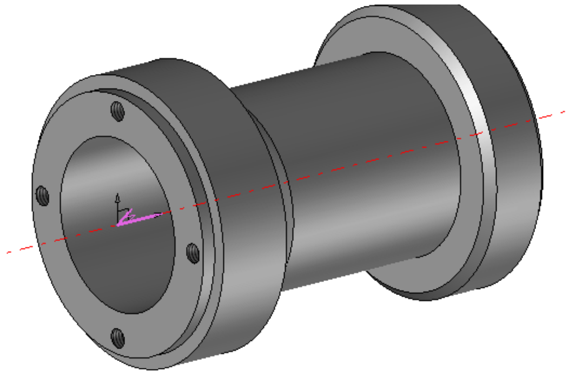
Place a CSystem on the front face.
Use the function ***Tool/Coordinate System,*** and select ***Coordinate System on Face and point***



To orientate the coordinate system as shown, Click on the red arrow which appears when creating the CSystem. If the arrow is not visible, use the function Modify Element to turn the CSystem.

Make this CSystem ***Current***

Construct a polar point. Use the function ***Tool / Point*** and ***select Polar Point*** to place a point centered on the origin of the CSystem, at a radius 25mm and angle 0° about the X+ axis. SET CURRENT



Use this point to drill an M5 hole to a depth of 12mm.
It is necessary to change the default mode DYNAMIC to NON DYNAMIC, because in NON DYNAMIC mode the hole will center itsel about the cylindrical part,

Propagate the threaded holes using CIRCULAR with angle 360° and number set to 4.

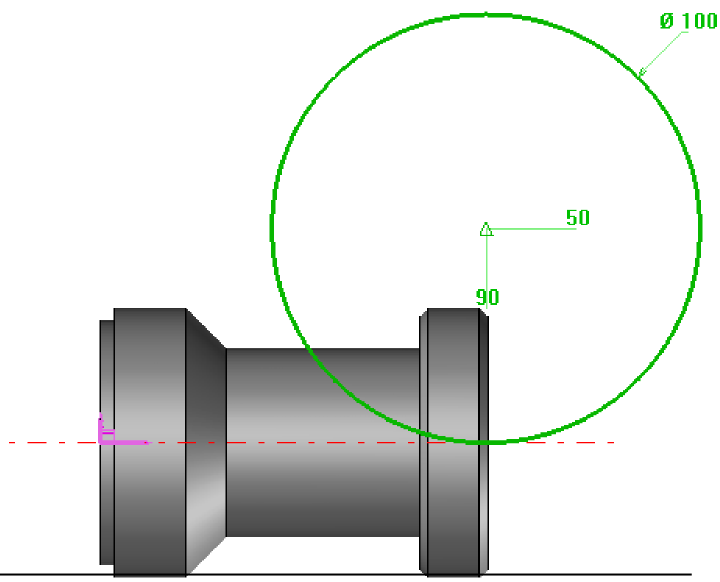
To facilitate a circular cut at an inclined angle, ***Edit/Duplicate*** the absolute coordinate system 45° about the rotation axis of the part.

Make this duplicated CSystem ***current***

Switch to ***Top View.***

Begin a new sketch.

Construct a ***Circle*** Ø100mm passing through the point (90,50).

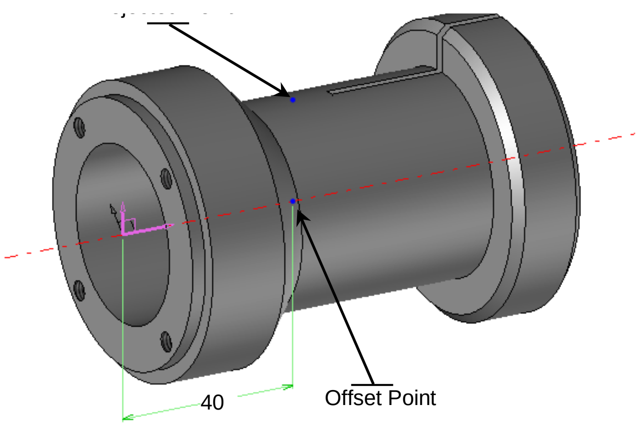


End Sketch.

Trim the part using the circle. Use the function ***Shape/Trim*** and select trim = BY SWEEPING CURVE click the option >> and set the 1st and 2nd extrusion lengths to 1mm.

Place an ***Offset Point*** 40mm along the X+ axis.

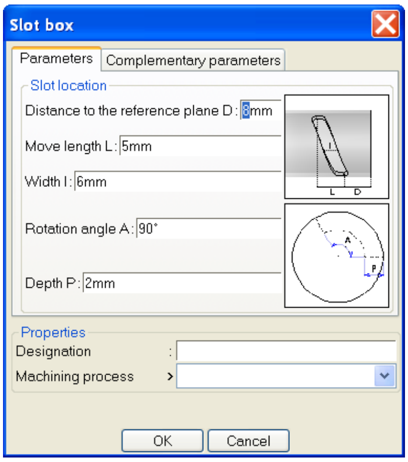
Place a ***Projected Point*** along the Y+ axis on the cylindrical surface.

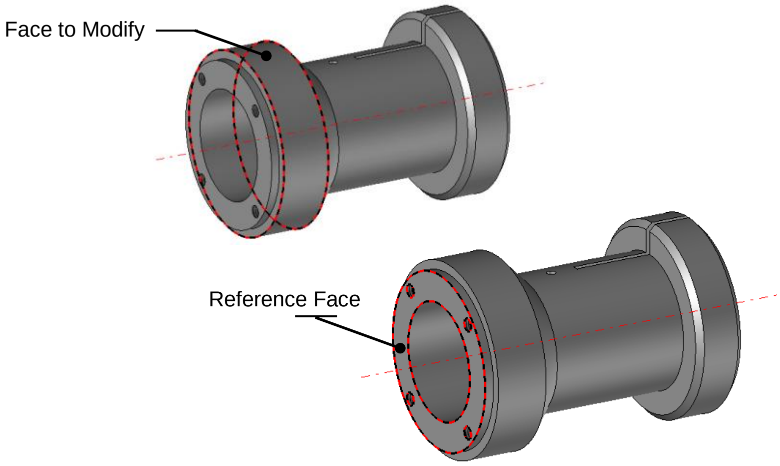


Create a CSystem on the projected point using ***Surface and Point.***
Pick ***CSystem on point*** first, then ***Projected Point.***
Use Modify Element or orientate the new CSystem

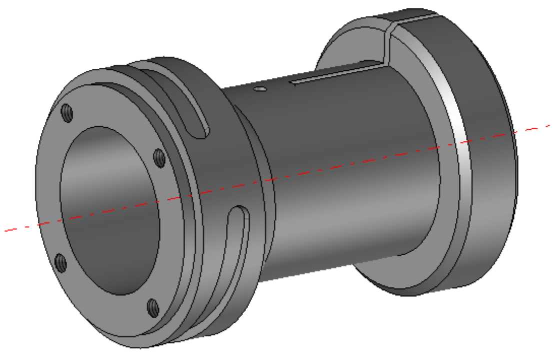
Make a ***Simple hole*** Ø3mm with "Through One" option on this reference plane. Use the option FRAME OR SKETCH select the CSystem followed by the cylinder as the reference face.

Use ***Shape/Mechanical Other Operations/Helical*** with the following parameter to create as helical slot as shown





Propagate the helical slot using the CIRCULAR option 360°about the center axis of the part.
There are 3 helical slots.



Make a CSystem on the center of the back face of the part. Use the function ***Tool/Coordinate System***, and choose ***Coordinate system on face and point***

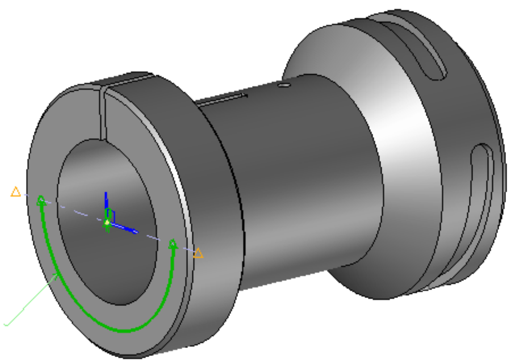
To orientate the CSystem, click on the red arrow that appears when you create the coordinate system. If the arrow is not visible, use the modify element function to rotate the CSystem

Begin a new sketch

Construct a Ø50mm circle centered on the CSystem.

Create a ***Sketch Line*** passing through the origin of the CSystem.

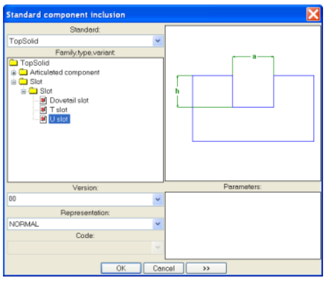
Trim the circle with the sketch line. Note, as this is a construction line you must use set "Consider construction line" = YES
.

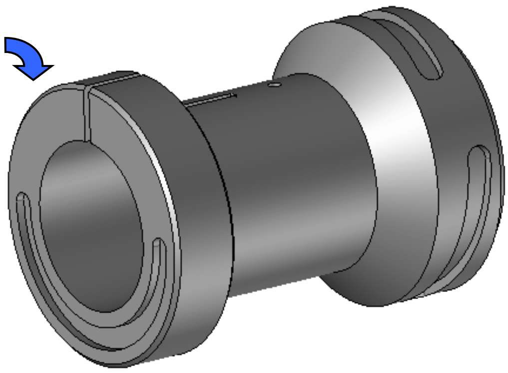


Ø 50

End Sketch.

Make a U slot on the back face using the arc a the sweeping sketch.
The Grove is 5mm wide and 2mm deep.

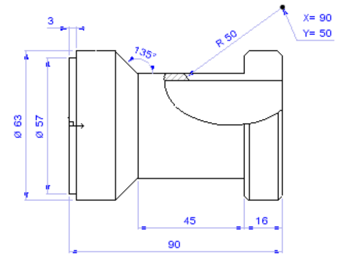




Define the part using « Shaft » as the designation of the piece.
Use the function Assembly / Define Part.

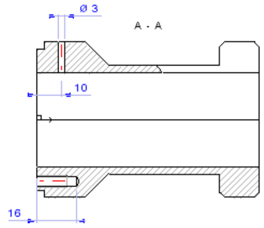
Save the file with the name « Shaft.TOP ».

Create a ***New*** Draft document. « Select Associative A3 Horizontal mm »



Project the views like those shown.

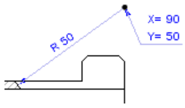
Add a ***Local Section*** on the principal view.

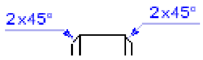


Create an ***Aligned Full Section*** on the view to the right.

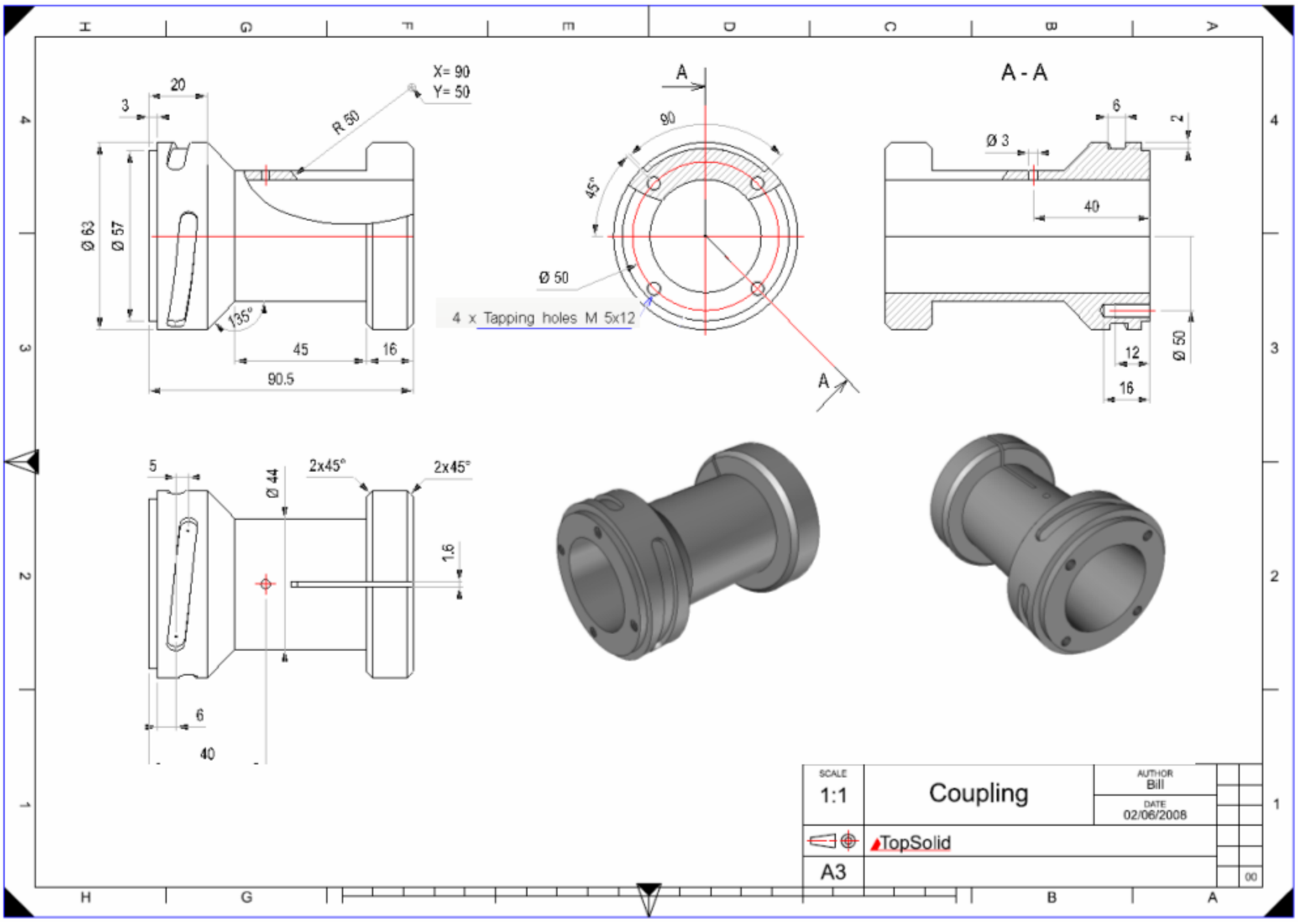
Completely Dimension the Draft :

- Drilling Dimension for the holes and threads
- Coordinate dimensions for the centre of the Saw radius
- Chamfer dimensions





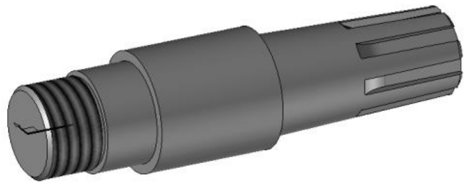
Save the file with the name: « Shaft.DFT ».



Exercise n°12 : Grooved Shaft (developed from Ex1)

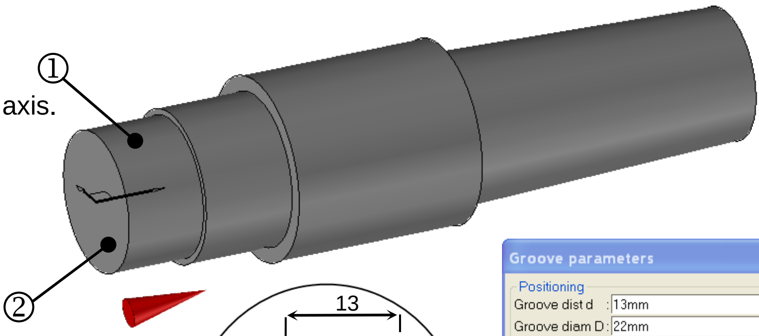
In this exercise you will use :

- Revolved shape
- Groove and threading operations
- Limitation of swept profile
- Shape subtraction
- Circular propagation

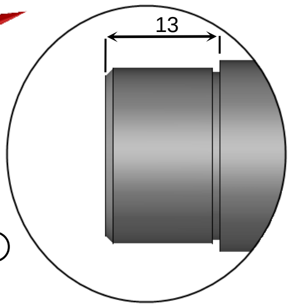


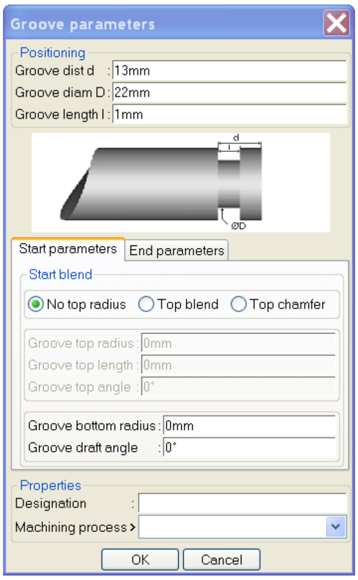
Open the « Shaft profile.TOP

Revolve the profile about the X⁺ axis.

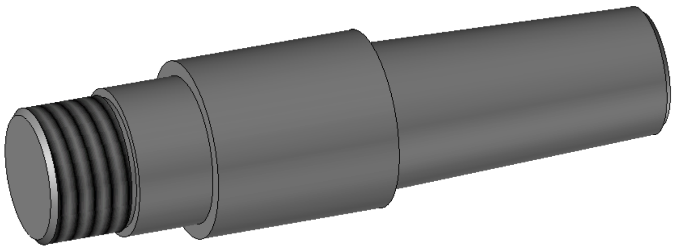


Create a **Groove** on surface ①
at a distance 13mm from face. ②





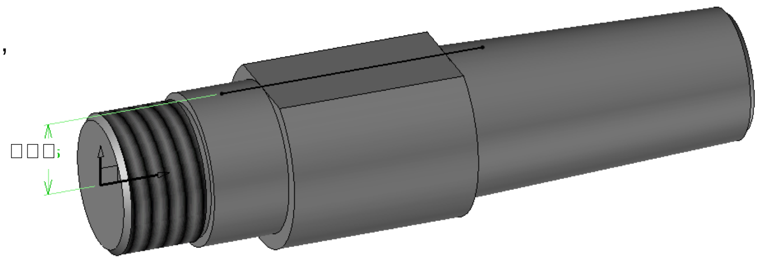
Make a complete ***thread*** on cylindrical surface ①
Pick surface 2 as the reference plane



Start a new sketch.

Create a horizontal ***line*** using two points, constrain it 12.5mm above the axis.

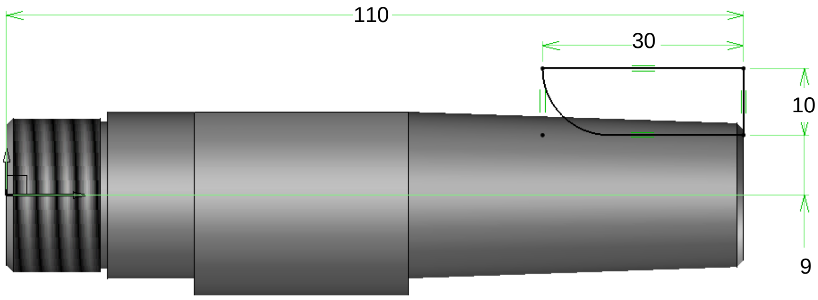
Validate the sketch.



Create a flat edge using ***Trim*** with the SWEEPING CURVE. option

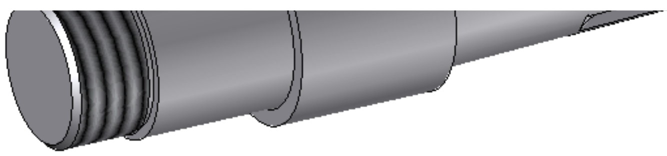
---

Create the Contour shown and constrain it as per the sketch opposite.



Use a 10mm fillet for the bottom left corner

Use ***Trim*** to create a slot from the profile.



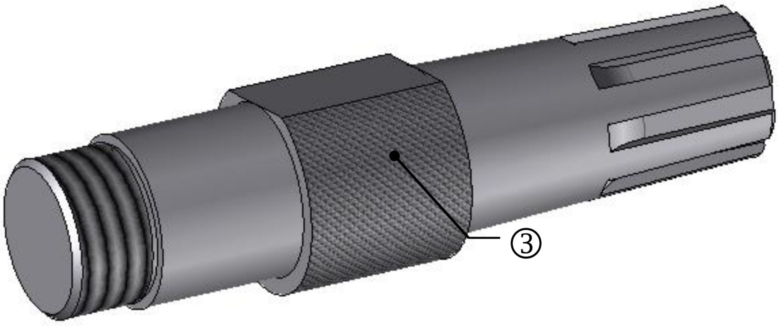
Use the "BY Shape" option, click on >>
And specify Extrusion Length and Second Extrusion length to be both 1mm.

Propagate the slot 360° about the X+
Using the Circular option. The number of slots is 8.

Create a ***Knurling*** 0f 1mm at 45° on the face shown ③

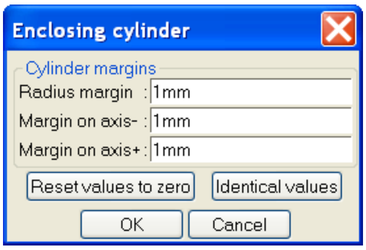
 → Knurling

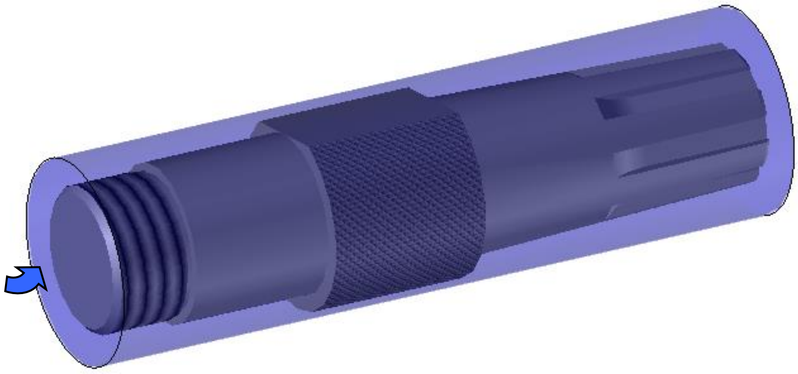
 → Slotting



Enclose the shaft in an ENCLOSING CYLINDER

Add a 1mm to the radius and to the + and – axis of the cylinder.





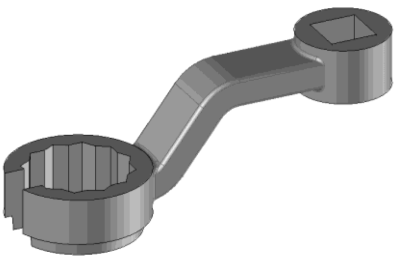
Define the part as « Grooved shaft » , using the ***Assembly / Define part*** function.

Save the file in the training directory with the name : « Grooved shaft.TOP »

Exercise n°13 : Connector

In this exercise you will use :

- Regular polygon
- Duplication
- Concatenated Profiles



- Layers
- Strip
- Profiled shape
- Extrusion of a face with an offset
- Unite and subtract operations

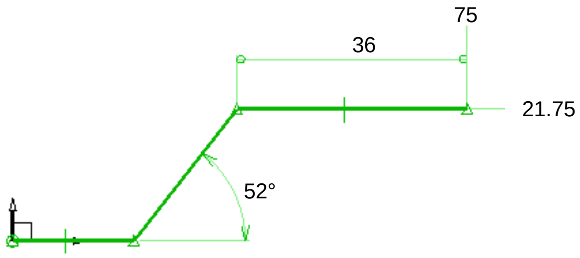
Create a TopSolid'Design ***New Document***.
Choose from among the standard templates « Associative no template ».

Start a new sketch.

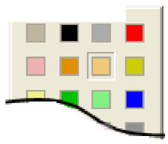
Make a ***Contour*** by points

Dimension and Adjust the dimensions

Validate the sketch



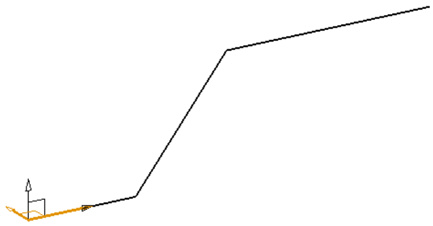
Change the current color in the options bar to orange



Create a CSystem perpendicular to the absolute one. Use the tool/CSystem operation and validate XZ.

Make current this CSystem

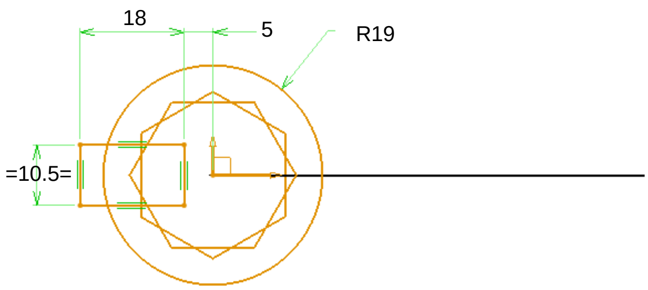
Use the ***Modify Element*** function to orientate the CSystem as shown



<u>The following circle, the hexagon and rectangles are created in sketch mode.</u>

Create a ***Circle*** radius 19mm centered on the origin of the CSystem.

Create a ***Regular Polygon*** with 6 faces and an internal diameter of 25.5mm.
Keep the same dimensions to make another hexagon with a 30° angular offset.
Use the option rotation angle 30°, and indicate the center of the hexagon.

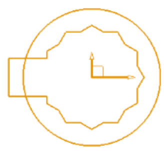


Create a RECTANGULAR ***Contour***, validate AUTO DIMENSION, then add a SYMETRY CONSTRAINT by the X axis and NO SYMETRY AXIS about Y

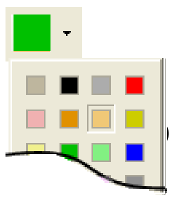
Adjust the dimensions to the rectangle as shown above.

Add a ***dimension*** to place the rectangular contour 5mm from the Y axis.

Merge the rectangular contour to the first polygon,
Then, ***Merge*** the result to the second polygon, in order to obtain the following result.

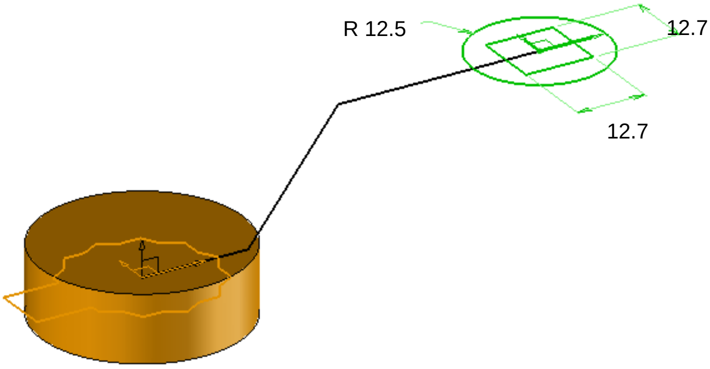


Extrude the circle of radius 19mm with an Alignment = CENTER and a height of12.5mm



Change the current color to green.

Create a new ***CSystem*** at the other end of the profile. Use the ***Tool / CSystem*** function. Click the point directly at the end of the profile, the orientation of the CSystem on a point depends on the orientation of the current CSystem.
To orientate the CSystem differently use ***Modify Element***.



Make this the ***Current coordinate system***.

Make a 12.5mm radius ***Circle*** centered on the origin of this CSystem.

Make a ***Rectangle*** with 12.7mm sides, also centered on the CSystem. Use RECTANGLE , AUTO DIMENSION, and SYMMETRIC CONSTRAINTS along X and Y.

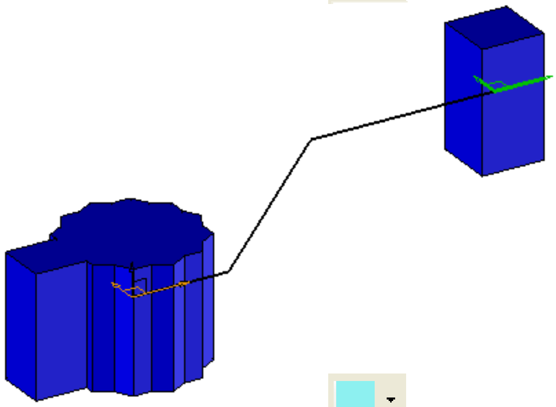
Extrude the 12.5mm radius circle with Alignment = CENTER to a height of 15mm.

Place the two cylinders created on layer 1. Use the ***Attribute / Layer f***unction.
Switch the option Modify the parts globally to YES in order to include sketch and construction elements from which the cyclinders were originally drawn. Turn layer 1 off.

Make Layer 2 Current

Change the current colour to blue.

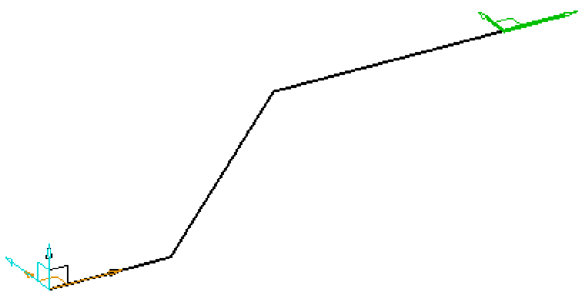
Extrude the square with an alignment centered on 20mm, then ***Extrude*** the profile concatenated with a centered alignment on 25mm.



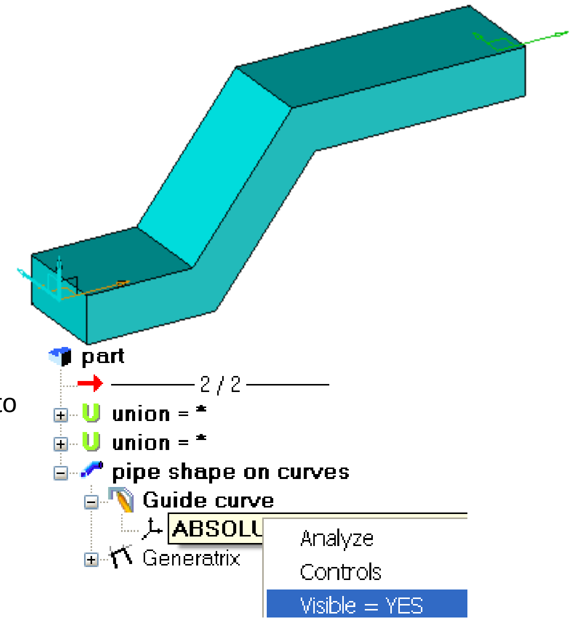
Make Layer 3 Current and turn off layer 2.

Change the current colour to cyan.

Create a CSystem perpendicular to the profile.
Use ***Tool / CSystem*** choose the CSystem on « profile and point ».
To orientate the CSystem correctly, use ***Modify Element*** and orientate with "spin"



Make current this CSystem.



Draw a 8mm, centered on the origin of the new CSystem.

Create a swept protrusion of the rectangle along the open profile.

Turn on Layer 1

Unite the two cylinders to the rectangle. Use Selection to select the two cylinders.

Open the feature tree, Edit the swept protrusion.
Right click the profile curve and switch the option Visible to YES with the left button.

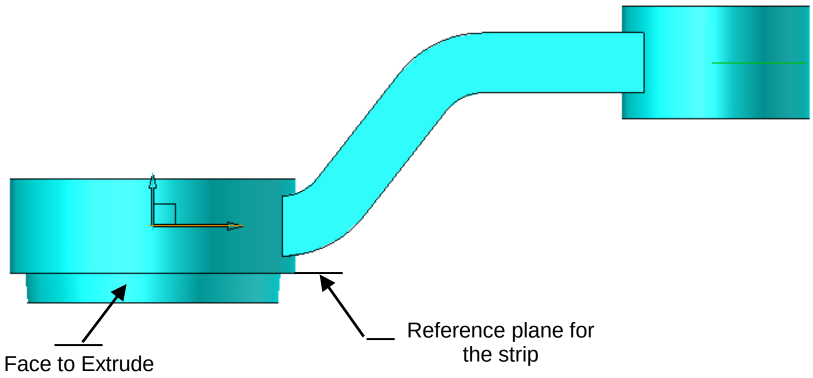
Use the ***Modify Element*** to edit the sketch.

Add 10mm fillets on the profile curve.

Validate the sketch

*Make the guide curve **invisible***

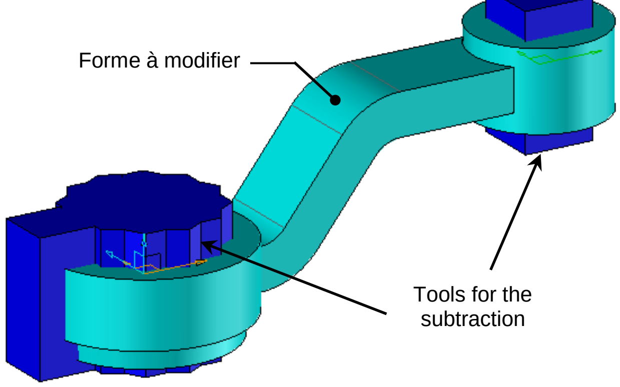
Shape/Boolean/surface operations / Extrude.
Use the plane face under the 19mm radius cylinder to make a 4mm high extrusion with an offset of 2mm.



Make a 5° on the cylindrical face of the cylinder.
The reference plane is the plain surface under the cylinder.

Turn layer 2 on and layer 1 off.

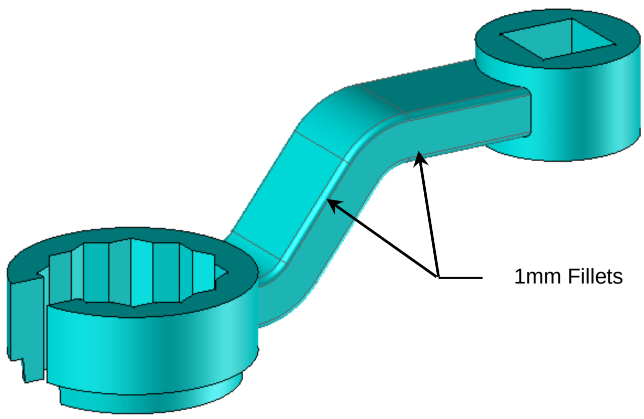
Subtract the two protrusion on layer 2 from the shape to modify. Just like when using the unite command you can select the two tools with the selection tool.



Make 1mm fillets on the main shape.

Assembly / Define Part
Define the part « Connecting Rod»

Save the file in the training directory with this name : «Connecting Rod.TOP »



Exercise n° 14 : Punch Assembly

There are two ways of creating an assembly within TopSolid'Design.

In-Place (or ***Top Down***)
Allows the designer to quickly design individual pats of an assembly within the one file.

Bottom Up
All parts of an assembly are first modelled separately. An new assembly file is then created and the individual parts are inserted into the assembly. These parts are then are constrained to one another using various different constraining techniques.

In this exercise we will examine the different assembly techniques. In addition, we will look at managing layers, placing the assembly within a draft document, and managing Bills of materials

I – In-Place (or Top Down) assembly

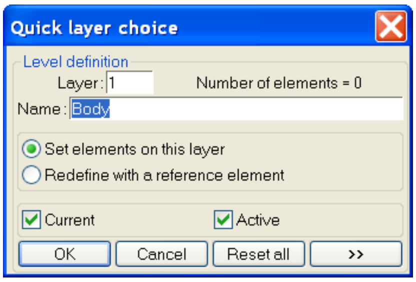
In-Place assembly design is useful for designing small assemblies, where individual parts are not required in other assemblies

A - Construction of the body

Create a TopSolid'Design ***New Document***.
Select « no template » from among the standard templates

Set layer 1 to the current layer. To do this right click over the "quick layer" menu at the bottom of the screen.

Name the layer « Body ».
Layer 1 turns green indicating this is the current layer



Create a CSystem on the XZ plane of the current CSystem and **make it current**.

Switch to top view in this CSystem.

Name the Csystem Rep/Body using the function ***Name*** from the ***Edit*** pull-down menu

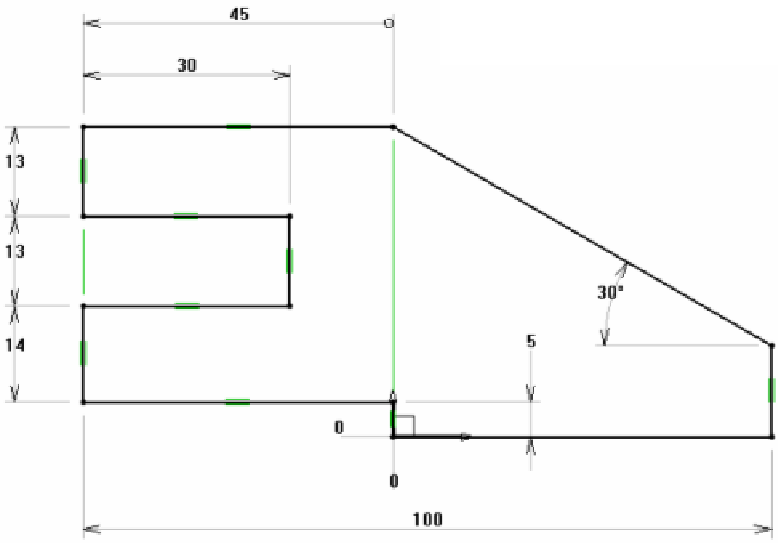
Create a new Sketch.

Create the contour shown using points. Place the contour about the origin as shown,

Dimension and ***Ajdust*** the dimensions.

Validate the sketch.

Extruded the contour to a height of 38mm



Create a pocket

Click Current ***coordinate system*** and highlight the inclined face .

Begin a new sketch

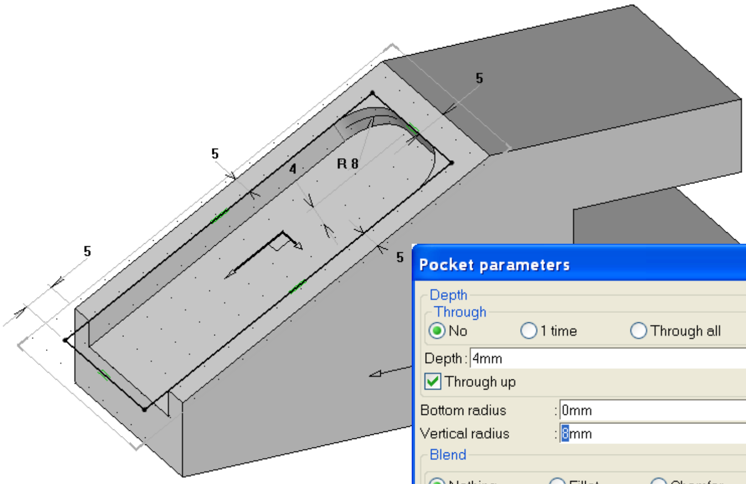
Construct a RECTANGULAR ***Contour***

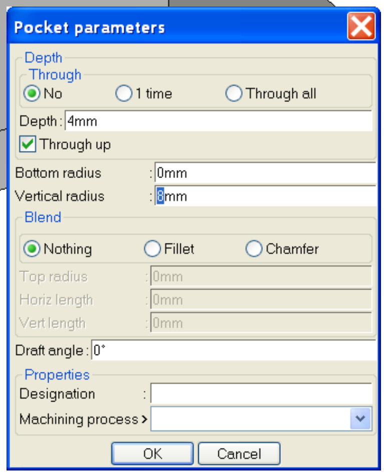
Dimension then ***Ajust*** the dimensions

Merge the dimensions together using the function ***Parameter / Merge from*** from the pull down menu.
Set the merged dimensions to 5mm.

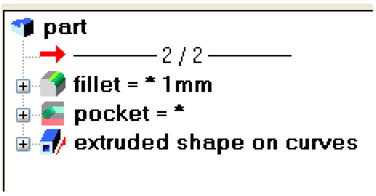
Validate the sketch.

Make a ***pocket***, depth 4mm, with 8mm fillets at the vertical corners.

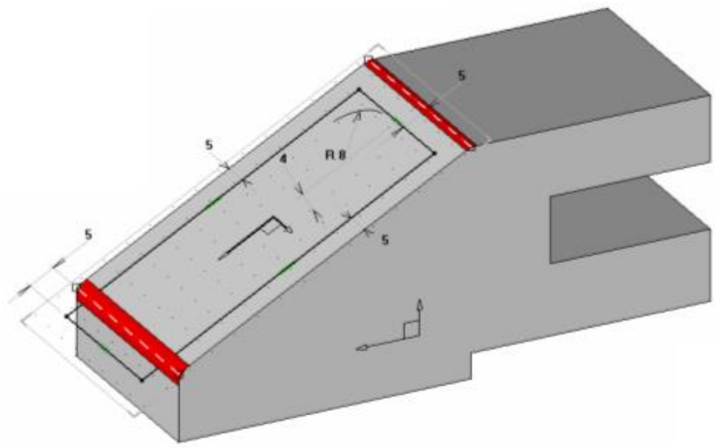




Fillet the bottom of the pocket with a value of 1 mm.
Open the Feature tree,
Move the cursor over the line extruded shape on curve.
Create 2 fillets R = 4mm on the top and bottom of the incline.

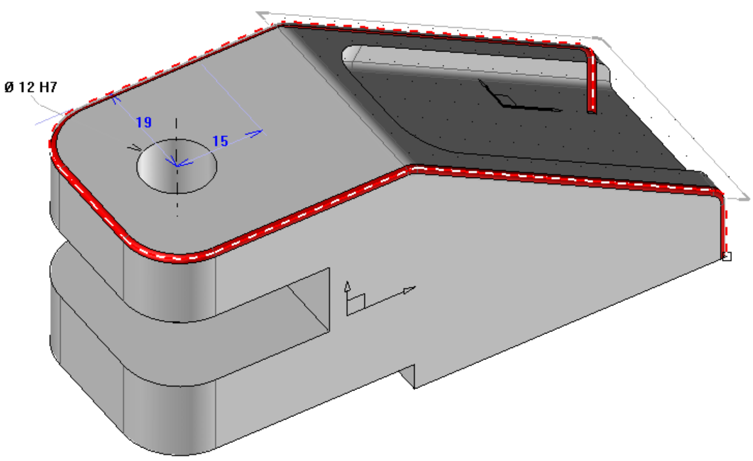


Move the cursor to the top of the feature tree.



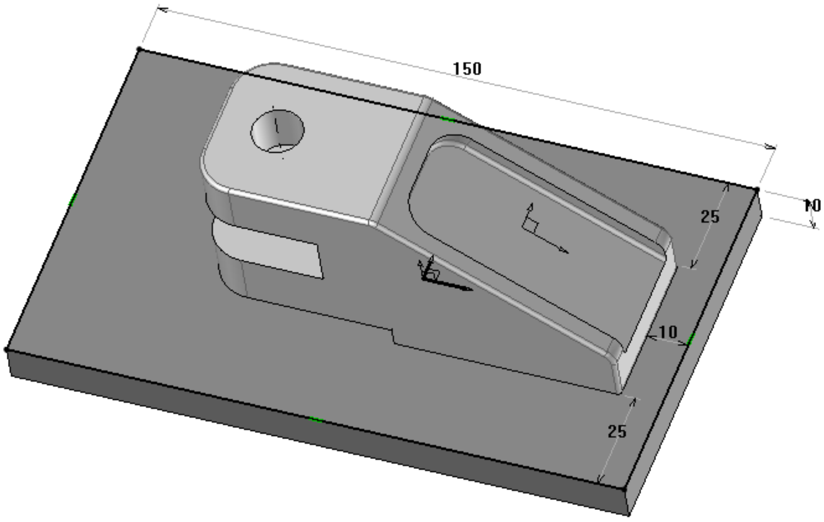
Make a Ø12 H7 boring hole on the top face contrained as follows. When creating the hole remember to switch to NON DYNAMIC mode and use the through all option

Add four R=10mm fillets to the front of the part and a 1mm one on the top.



B – Creation of the base

Make the ***absolute coordinate*** system current.



Make layer 2 current

It is also possible to do this by placing the mouse over « Lay=1 » and clicking. For current layer type in 2. you can also click on name to enter a name fo the layer.

Name it « Base ».

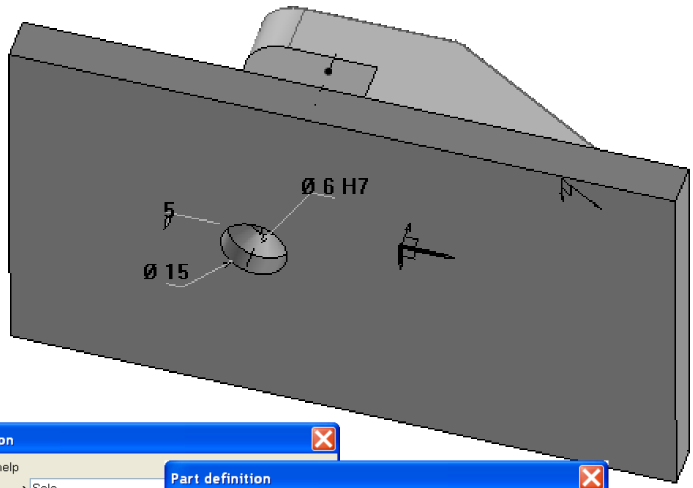
Create a rectangular contour and dimension it as shown opposite.

Extrude the contour 10mm

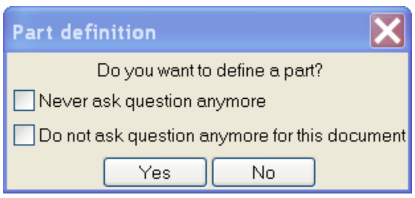
.

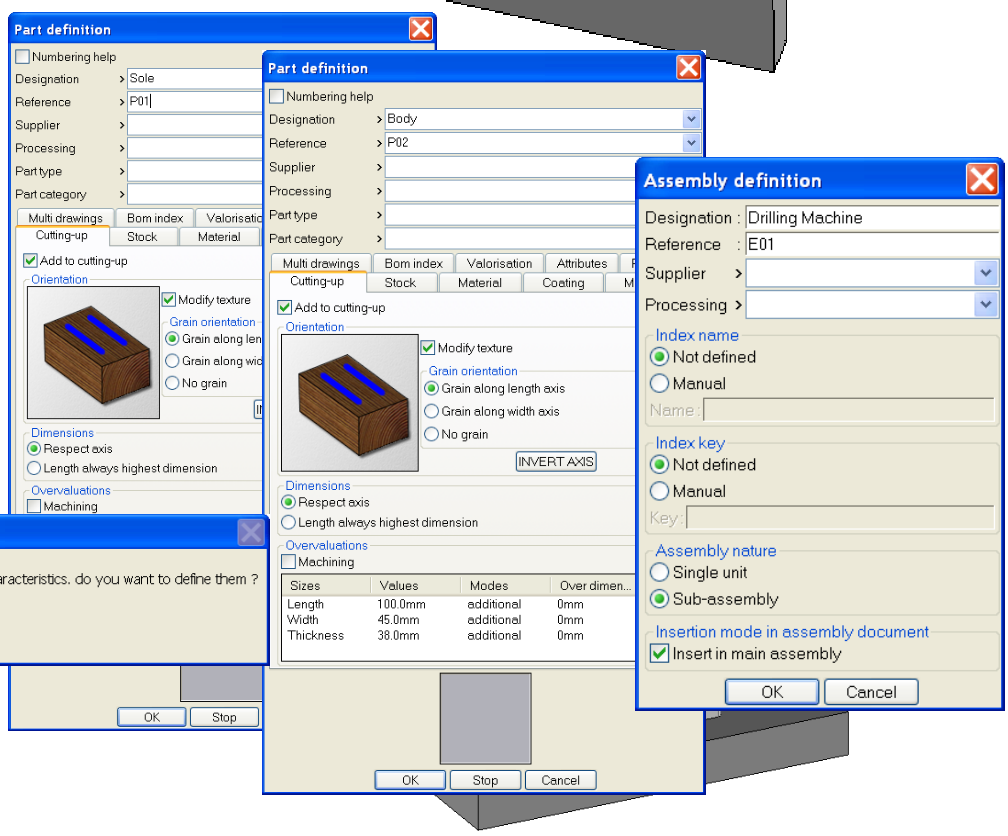
Create a Ø6 H7 hole on the lower face of the sole.
When doing this stay in NON DYNAMIC mode, hightlight the face to drill and select the center of the CSystem of the Ø12mm hole.

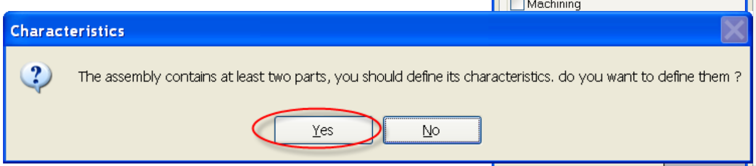
Then drill a Ø15mm hole, depth = 5mm at the same place using the center of the Ø6mm hole as a reference.



Save the files
Name it « Punch.top ».







C - Construction of the punch

Place a ***coordinate system on face and point***.
Select the top face of punch followed by the center point of the Ø12 hole.

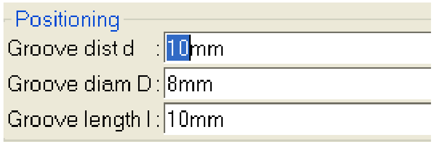
Make this current

Name the Csystem Rep/Punch using the function ***Name*** from the ***Edit*** pull-down menu

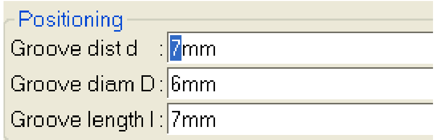
Make layer 3 active and name it « Punch ».
Hide layers 1 and 2 so that it is easier to work on the assembly.

Construct a cylinder, centered on the current CSystem, with a Ø12 and a height of 74mm in the Z+ direction and alignment set to CENTERED

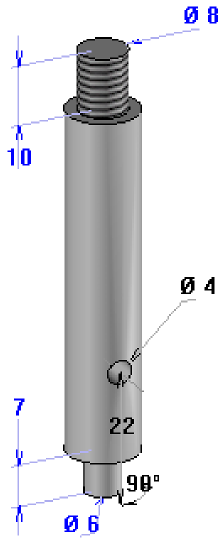
Place a groove on top of the cylinder with the following parameters :



Repeat for the lower half with the following parameters:

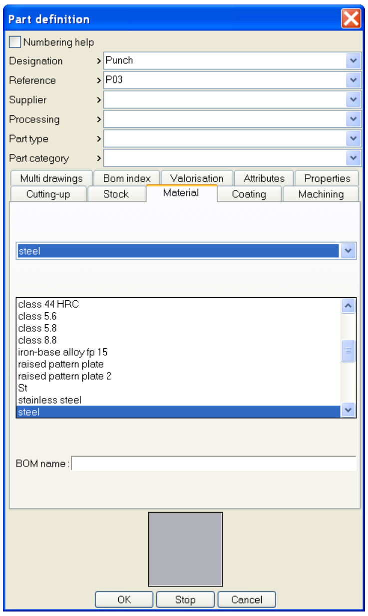


Make a Ø4 drilling using mode = NON DYNAMIC with Coordinate system = Radial
Place it at a distance=22mm and angle=90° form the base.



Don't forget to place a screw *Thread* on the top groove.

Use ***Define / Part*** to set the characteristics of the part.



Save the Part

II – Bottom Up Assembly

When placing a number of parts within as assembly using the Bottom-up approach, two methods of assembly are possible:
placement by constraints or placement using CSystem.

A – Positioning using constraints

In this method we use reference faces on different parts to align parts with each other.

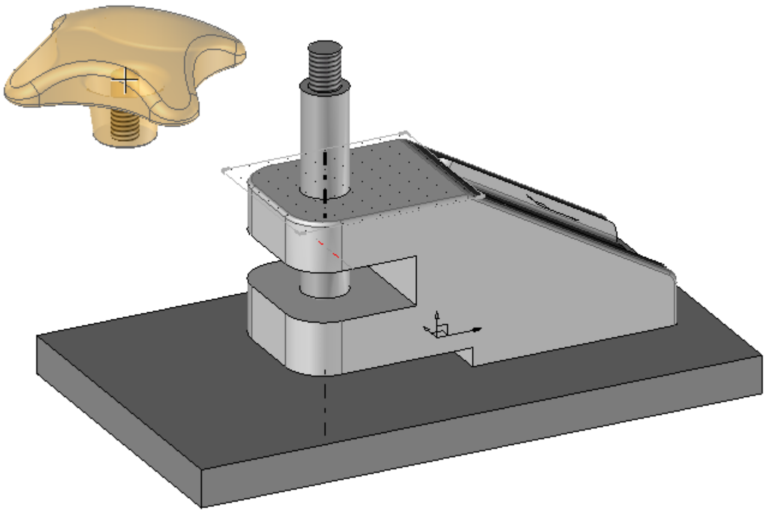
The Knob

Here we will place the knob part we designed earlier into the assembly.

First activate and make layer 4 current. Then name it «Knob ».

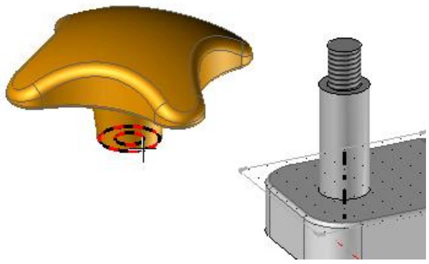
Next activate the assembly menu.
Select ***Include Sub assembly / Part,*** Explore and find the part Knob.top you created earlier***.***

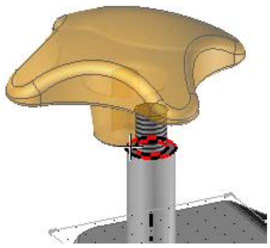
Click anywhere on the screen to place it into the assembly.



Next select the base face of the knob followed by the top face of the shaft as shown. This place a Mate relationship between the two faces.

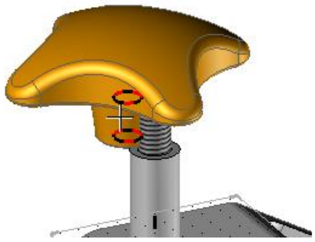
For the MATE distance specify 0.

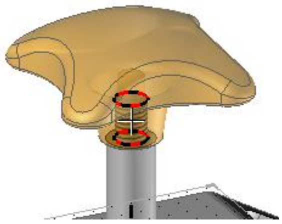




Now select the drilled hold on the knob and the axis of the shaft. TopSolid will automatically place an axial relationship between both.

To finish select STOP, No PROPAGATION





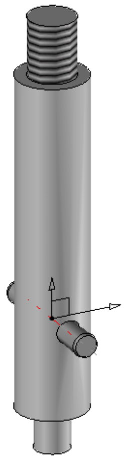
B – Positioning using Reference Planes

Positioning by CSystem consists in using a CSystem built in the file of origin to make it correspond with another CSystem located in the document of assembly.
This type of positioning is particularly well adapted to the standard components on which it is easy to envisage and declare reference frames.

This method involves using the CSystem created with the original part (to be inserted) and constraining it to another CSystem within the assembly.

This is particularly useful when inserting Standard component on which it is easy to see and define reference points.

The Pin



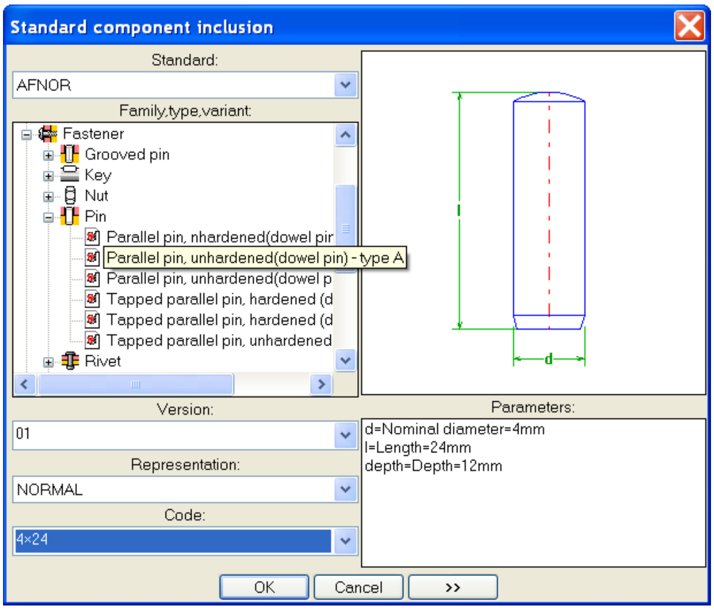
Before inserting the pin, make layer activate layer 5, make it current and name it « Pin ».
Turn off layers 1,2 and 4.

 Place an axis at the hole using the ***Curves / Axis*** function

 Create a coordinate system «On curve», select the option « middle » and select the axis.

 Insert the Pin using the ***Include Standard*** function

Select « Unhardened (dowel Pin) – type A » under is in the « AFNOR » Standard , family. Expand « Fastener », and « Pin » and
Choose the code « 4x24 » which corresponds to the dimensions of the pin.



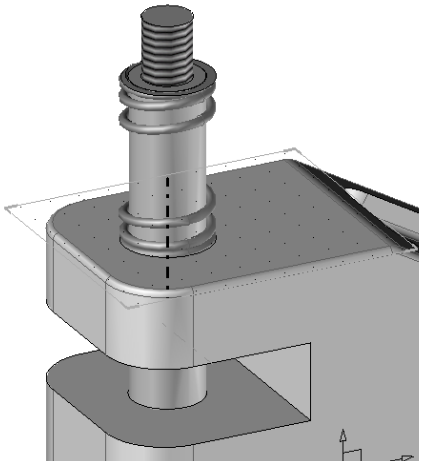
Select Key point = «CENTRE COORDINATE SYSTEM» and select the Csystem previously created in the middle of the hole

Accept the rotation angle by clicking STOP

In the same way, skip the (drilling) operation by also clicking STOP.

The Spring

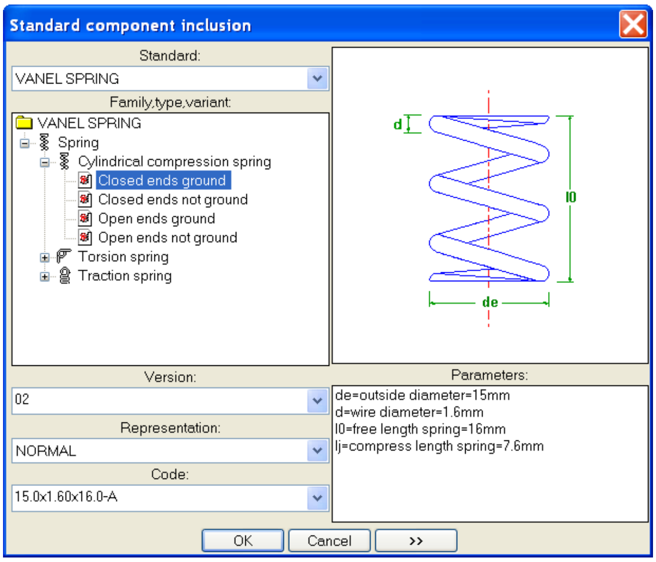
To pace the spring, we will again use the placement by CSystem option



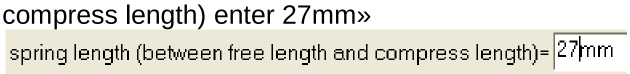
Show Layer 1, and make Layer 6 current and active. Name it « Spring ».

Use the ***Include/Standard*** function to insert the spring.
Select « VANEL SPRING » from the standard pul down menu.

Select « Closed ends ground» as follows:
Choose code « 15.0x1.60x32.0A ».



To the question:
«Spring length (between free length and compress length) enter 27mm»

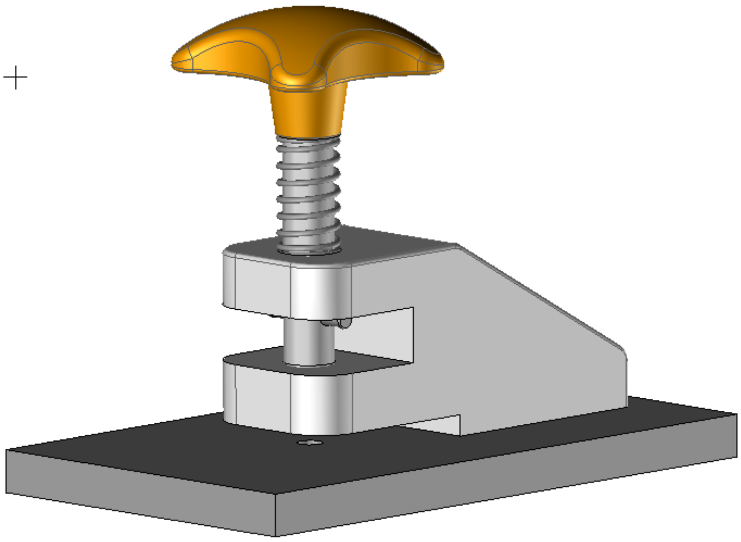


To calculate the length of the spring, use the function ***Distance*** from the ***Analyse*** menu. Specify the top of the body and base of the knob to measure.

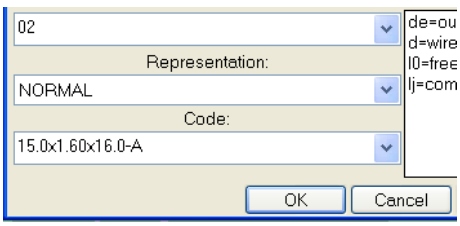
Set Key Point = « TOP FRAME » and click the current cooridinate system (which is on the top face of the body) to position the spring.

Click STOP when asked for the orientation of the spring.

Turn on all Layers and save the files.



To modify the spring use Modify Element followed by INTERCHANGE and select a different option
Caution: this will cause a regeneration and saving of the file which may take time with large assemblies.



III – Assembly Processing Functions

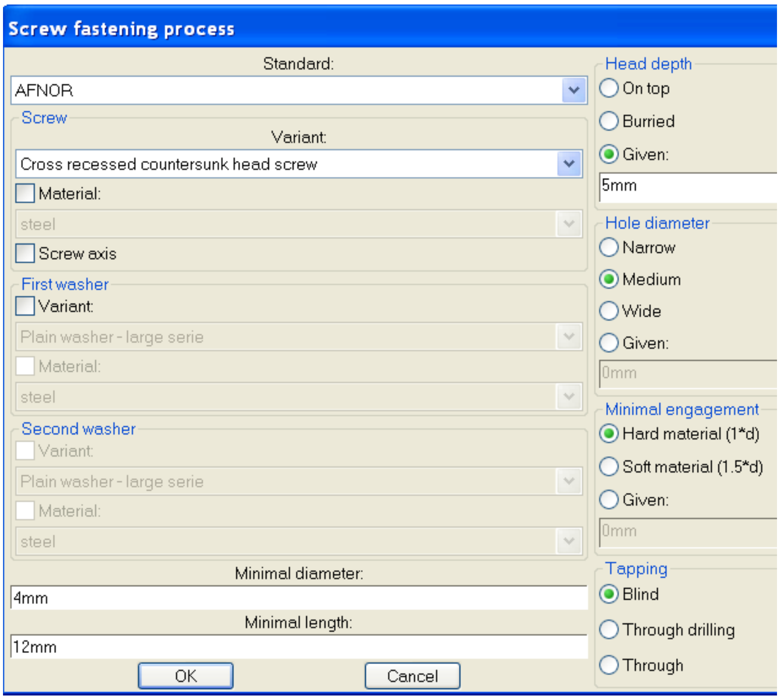
We will now look at directly bolting the base to the punch body. We will use an assembly processing functions that will allows us to select suitable bolts and at the same time mate the two parts together.

To start, reactivate the Absolute Coordinate system.

Make layer 7 current and active and name it « Screw ».

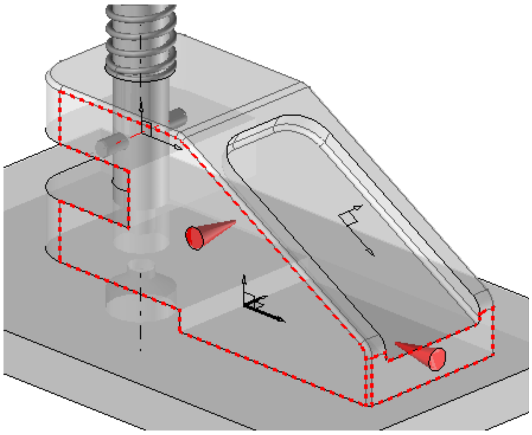
From the ***Attribute*** menu apply a ***transparency*** (3 or 5) to the Base and the Body.

Click on the ***Screw*** function in the assembly menu.

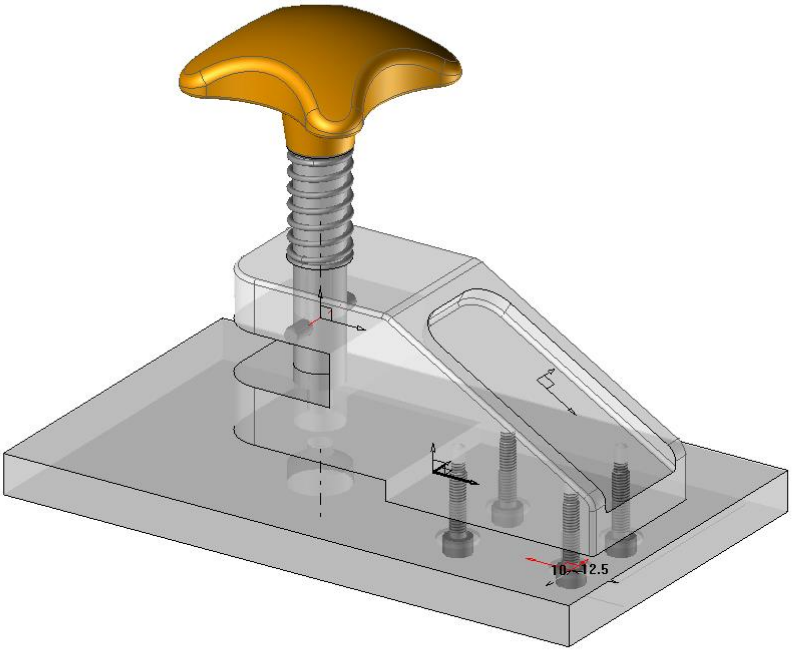


Here we can specify various parameters. In this case we specify a minimum diameter of 4mm and a minimum length of 12mm and a head depth of 5mm

To position the screw set the mode to NON DYNAMIC and pick the lower face of the Base as the reference face.



For the first alignment face specify a distance of 10mm.
For the second face specify a distance of 12.5mm.



For the parts to fasten, select the Base and Body and then STOP

A screw appears. Now select a « RECTANGULAR » PROPAGATION
In the direction X-, specify a distance of 30mm and 2 screws. In the direction Y+, type a distance of 20mm and also 2 screws.

Save the file.

IV – Draft mode and Bills of Materials

Here we will look at placing the allembly into a draft document, preparing the draft, and placing Bills of materials.

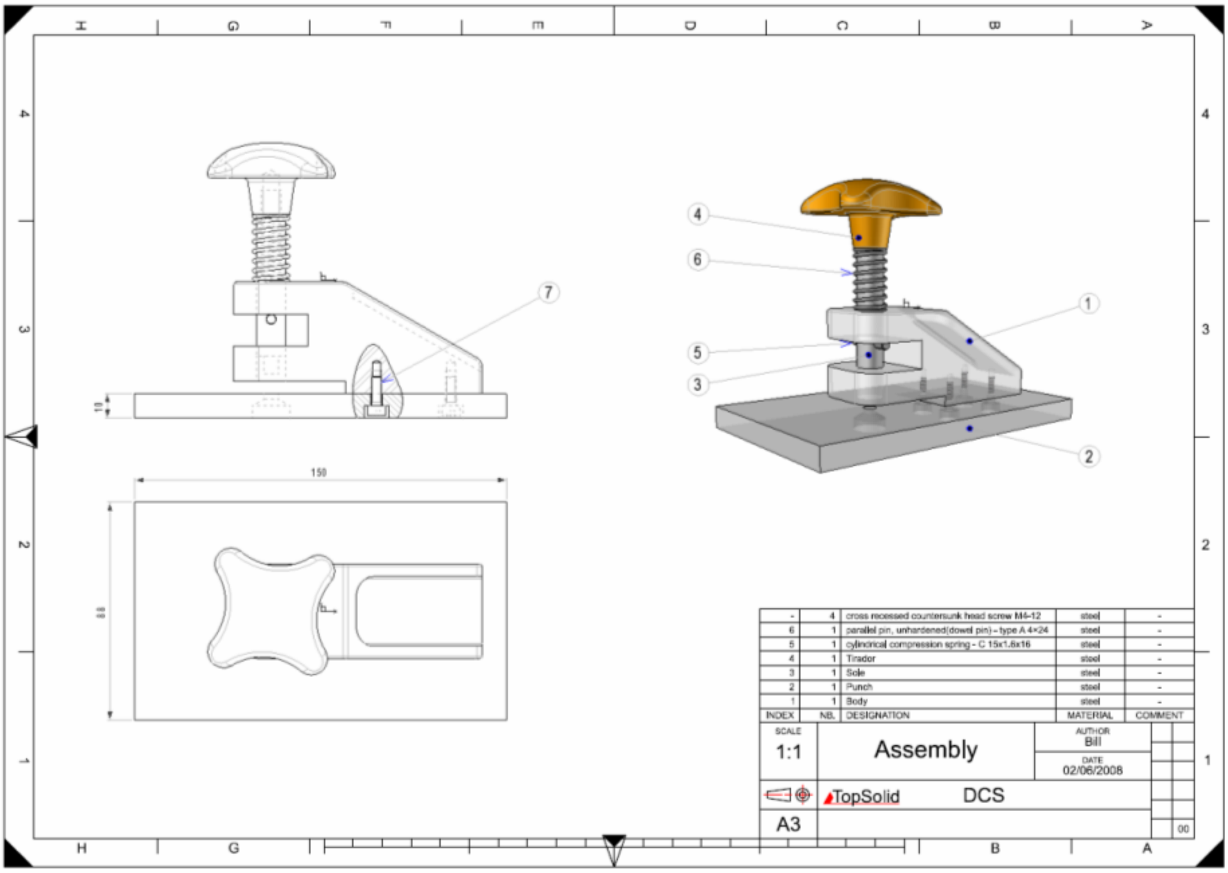
A – Complete Draft Assembly

Create a new Draft document. « Select Associative A3 Horiszontal mm »

Select ***Main view*** and click on ASSEMBLY to select all parts of the assembly.
We can pick all the parts the file by either selecting the assembly file from the pull down menu, using EXPLORE to search for the assembly file, or clicking on the 3d model if both the Draft and model are open on the screen.

Create the auxiliary views and place them on the draft.

Don't forget to change the scale factor by modifying the frame before placing dimensions.



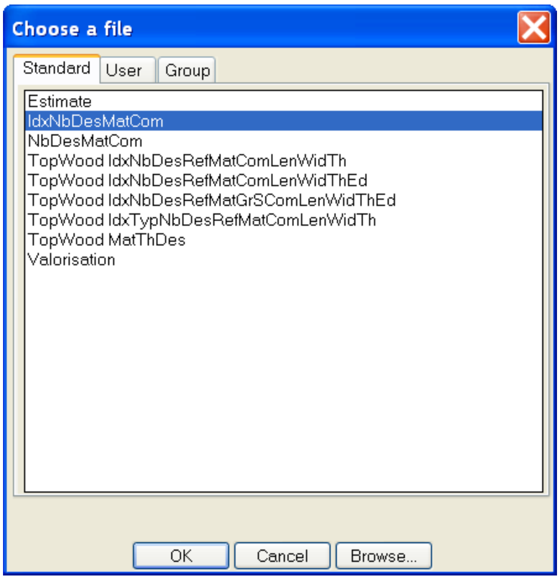
B – Bills of Materials (BOM)

Click on the ***Bills of Materials*** function.

Select a template file from the list. In this case we will pick the second option.

Click on the button ASSEMBLY to include all the parts of the assembly within the BOM.

From the drop down menu select an option on how the BOM will be build. In this case we will select Depth = « FLAT BOM »
Finally click on the Title block to position the BOM table above it.

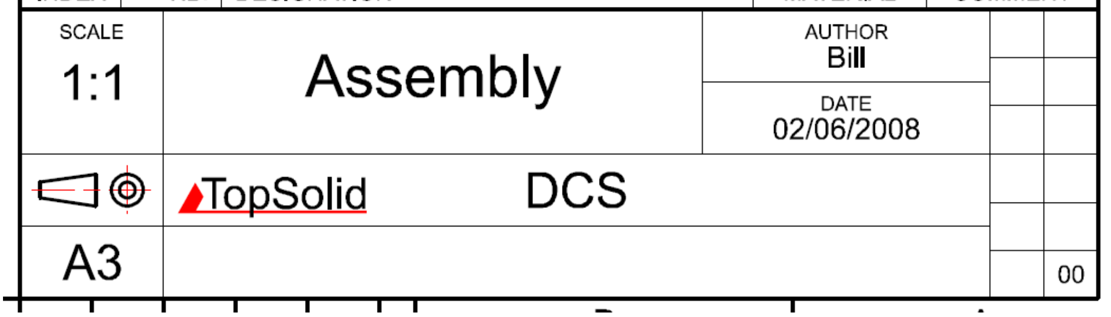


Use ***Automatic BOM Index to automatically*** place an index of parts

If parts are not automatically indexed, it is possible to manually index them using the ***Bills of Materials Index button***

To finish click on the regenerate button to update the BOM.

-	4	cross recessed countersunk head screw M4-12	steel	-
6	1	parallel pin, unhardened(dowel pin) - type A 4×24	steel	-
5	1	cylindrical compression spring - C 15x1.6x16	steel	-
4	1	Tirador	steel	-
3	1	Sole	steel	-
2	1	Punch	steel	-
1	1	Body	steel	-
INDEX	NB.	DESIGNATION	MATERIAL	COMMENT



Save the DRAFT file

The Draft file is automatically named with the name of the 3D model, but with a different file extension, namely, DFT.