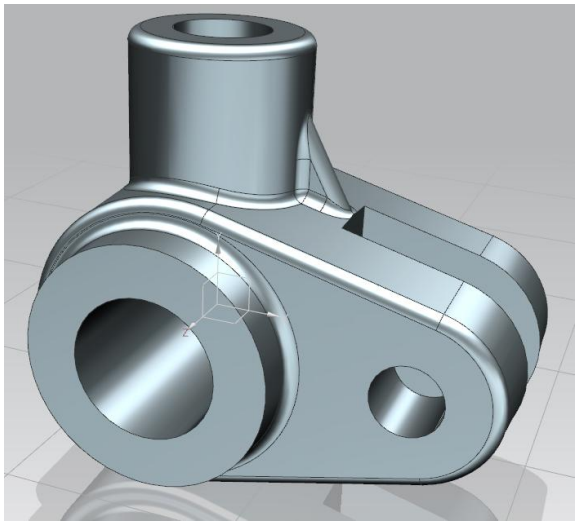
Siemens NX11 tutorials

The angled part

Adaptation to NX 11 from notes from a seminar Drive-to-trial organized by IBM and GDTech.

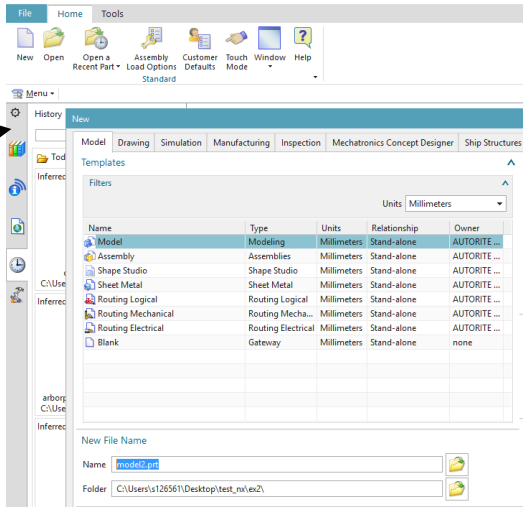
This tutorial will help you design the mechanical presented in the figure below, "from scratch".



1 – Introduction.

First, you will open a new file of type *Model*.

- In toolbar, select *New*.
- In the Filter list, select Model.
- Set the file name and its folder.
- Click *OK* to confirm.

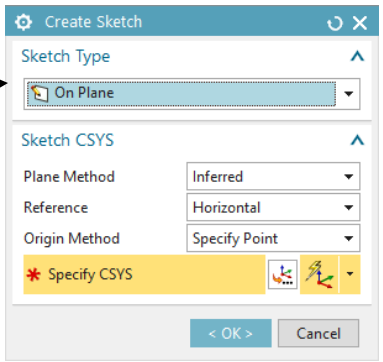


2 - Creating an extrusion.

Before creating a volume using extrusion, you must first select the plane on which to draw the profile.

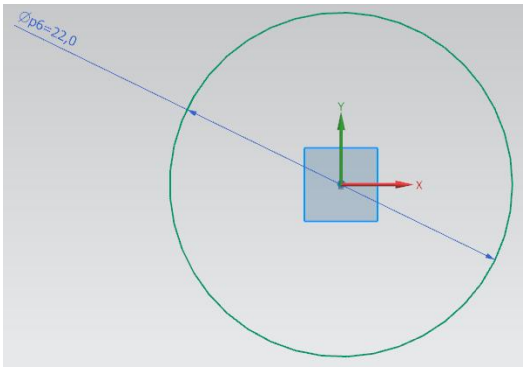
- Click on the *Sketch* button. 

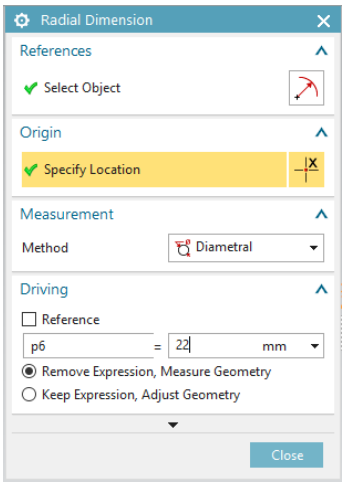
- Create a new sketch and select the plane XY in the *Create Sketch* dialog box.
- XY plan appears on the screen.



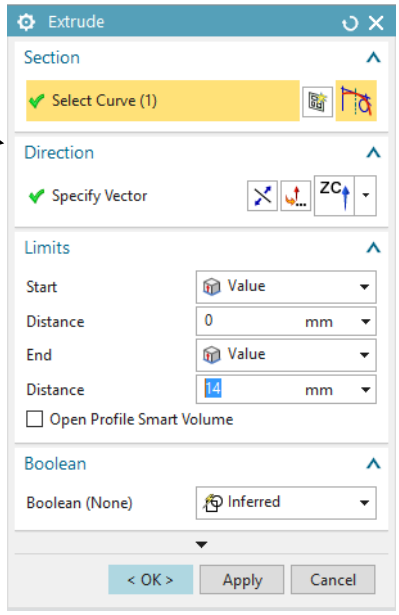
You must then draw the circle that will be the basis of the extrusion.

- Click the *Circle* button in the toolbar.
- Select the point of origin as the center of the circle.
- Click any point to define the circle.
- Double-click on the diameter constraint to open a dialog box.
- Set the diameter to **22 mm**.
- Click *Close* to accept the change.





- Get out of sketch mode using the button Finish Sketch.
- Select the *Extrude* button Extrude.
- The box dialog *Extrude* appears.
- Impose extrusion vector to be ZC.
- Impose the Start distance to **0 mm** and the End distance to **14 mm**.
- Click *OK* to confirm.

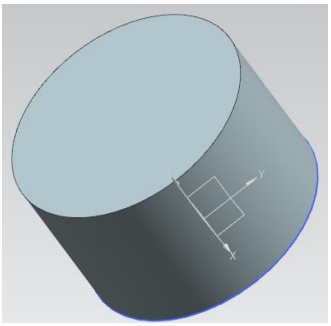


Manipulating objects.

1. To move hold down the Shift button as well as mouse middle button and drag the mouse (without releasing the buttons).

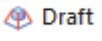
2. Rotation: hold down the middle mouse button and drag (without releasing the buttons).

3. Zoom: rotate the middle mouse button (wheel).

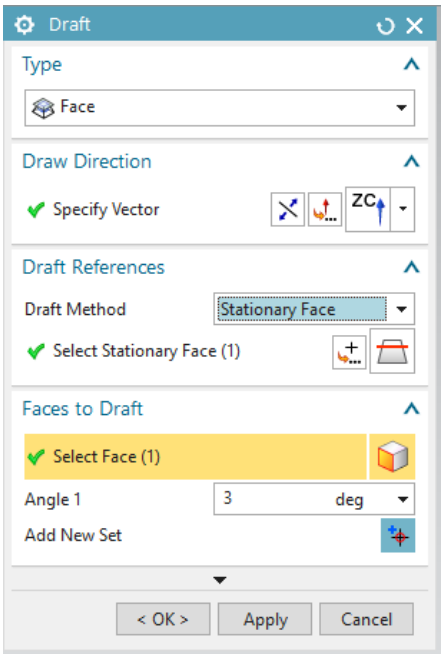


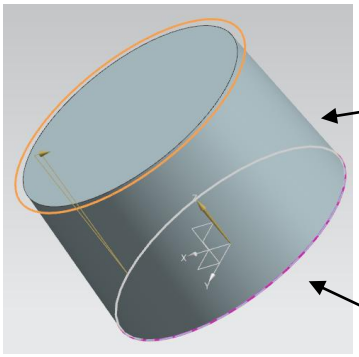
3 - Creating a draft angle.

Fallen serve unmold easily obtained by parts casting.

- Click on the *Draft* button 

.
- In the field *Draw Direction* select the ZC vector.
- In the field *Draft References* select as *Stationary Face* **the base of the cylinder included in the XY plane.**
- Select the **lateral face** of the cylinder as *Face(s) to Draft*.
- Click the yellow arrow to reverse it if necessary (it must be directed towards the interior of the cylinder)
- Impose a clearance angle of **3** degrees and have an overview.
- Click *OK* to create the body if the preview is correct (the part should become thinner at the top).





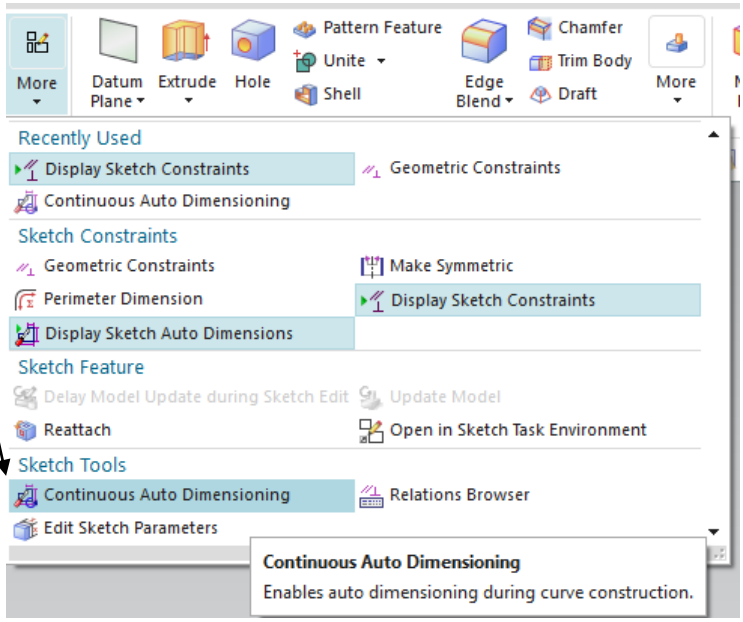
1. Face(s) to draft: lateral face

2. Draft reference: Base included in XY plane

3bis – Disable Auto Dimensioning.

From now, each time you create a new sketch, be sure to disable the *Continuous Auto Dimensioning*.

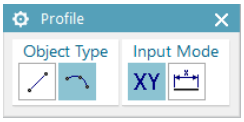
- In the toolbar menu, click on *More* and then disable the option *Continuous Auto Dimensioning*.

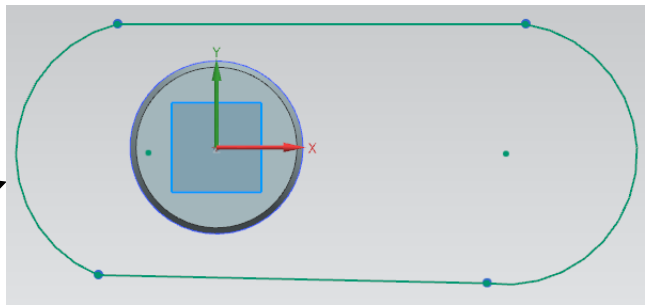


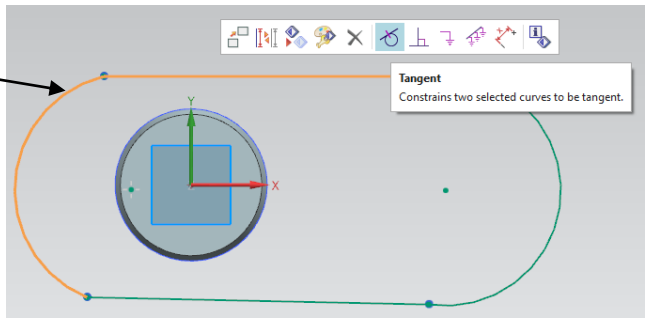
4 - Creating a second extrusion.

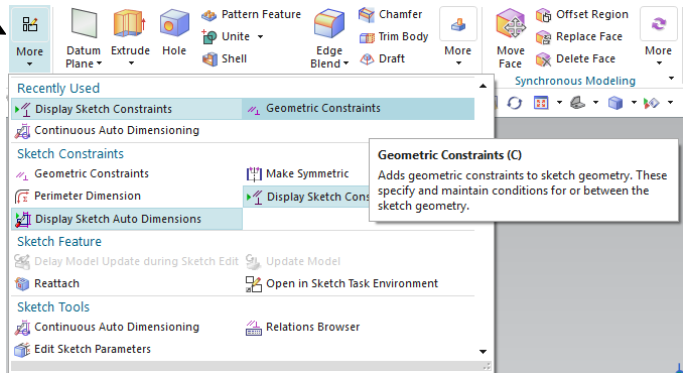
You must first create a profile that will be used in this extrusion. **Disable the *Continuous Auto Dimensionning*.**

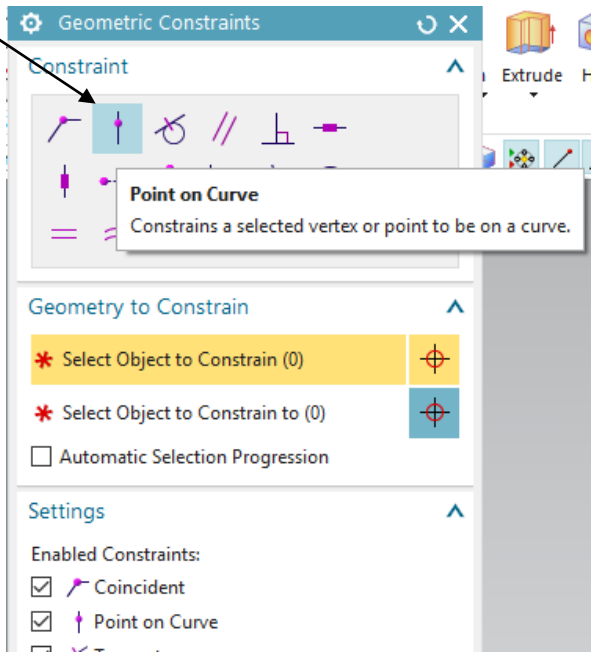
- Create a new sketch in the XY-plane.
- Using the *Profile* button and alternating between the *Line* and *Arc* type in the *Profile* dialog box, construct approximatively the hereafter oblong contour.
- Impose a tangent constraint at each connection between an arc and a line. To this aim, select an arc and a line (CTRL + left click) and click on *Tangent* in the appearing menu.
- Open the *Geometric Constraints* dialog box by clicking on the *More* button and then on *Geometric Constraints.*
- In the *Geometric Constraints* dialog box, select the constraint *Point on Curve*.



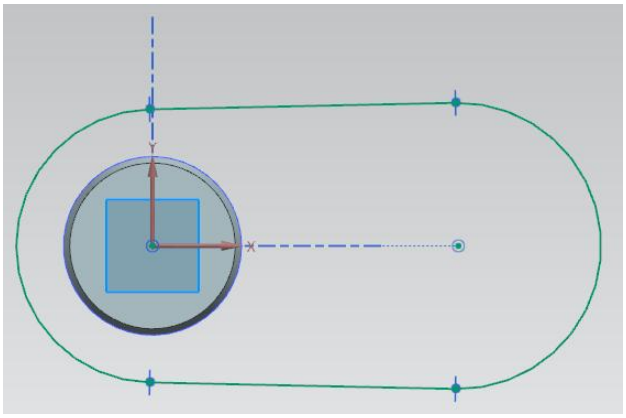








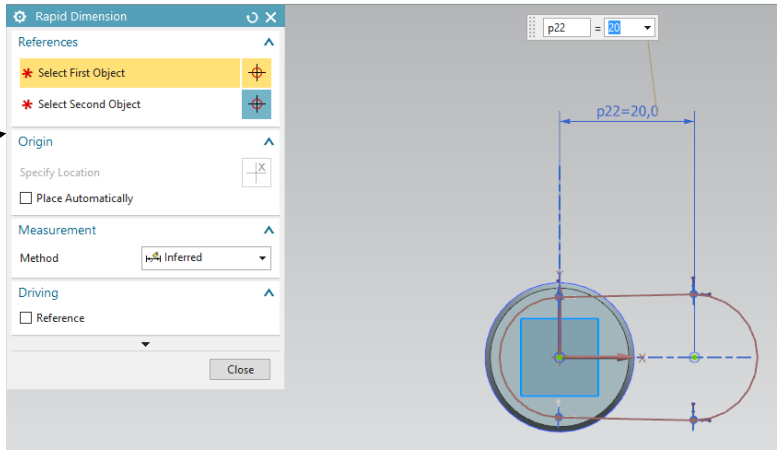
- Select the right arc center and the x axis and validate.
- Add two additional *Point on Curve* constraints by selecting the left arc center and the x axis and then the left arc center again and the y axis. You should get something similar to the figure on the right.



Next, impose the distance between the two arc centers.

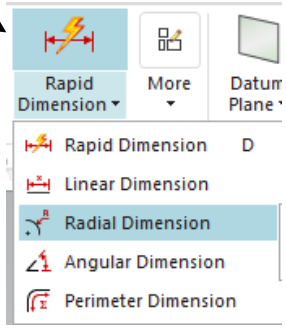
- Click on the *Rapid Dimension* 

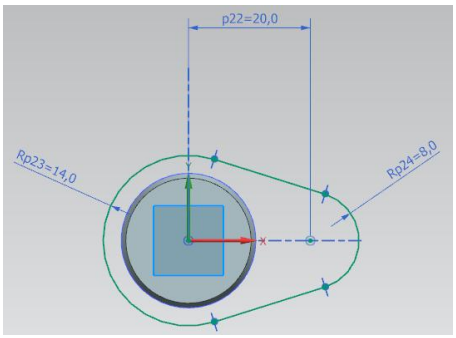
button and in the *Rapid Dimension* dialog box select the two arc centers and impose a length of **20 mm**.



Finally, impose the arc radii.

- Under the button *Rapid Dimension*, select *Radial Dimension*.



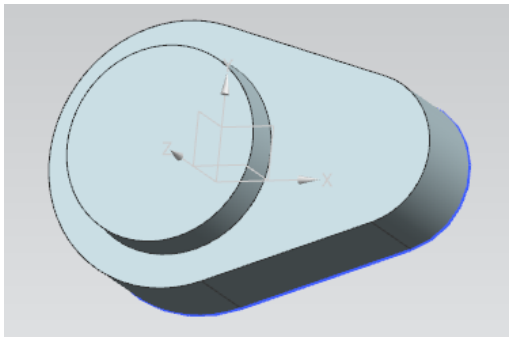


- Impose for the left arc a radius of **14 mm** and for the right arc a radius of **8 mm**.

- Get out of sketch mode (button 

)
- Create an extrusion (button 

) of **10 mm** based on the profile you just draw.

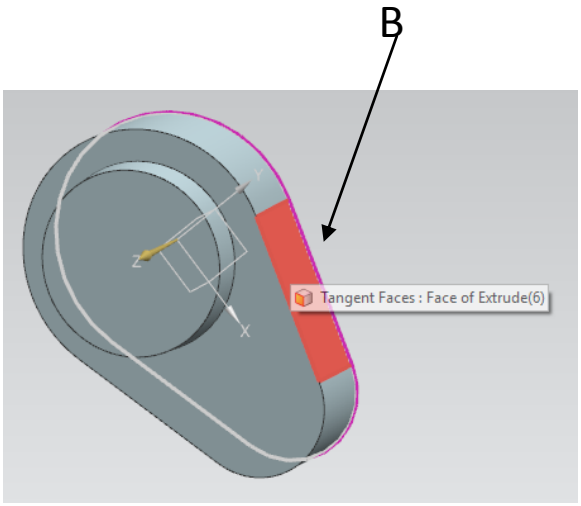


5 - Creating a second draft angle.

A draft identical to that performed on the cylinder will be created on the second extrusion

- Select the face B as shown in the figure.
- Select the inner face as draft reference (the back face with respect to the figure).
- Apply using *OK*.

A new draft is then applied to the second extrusion.



6 - Creating a hole.

- Click on the *Hole* button 

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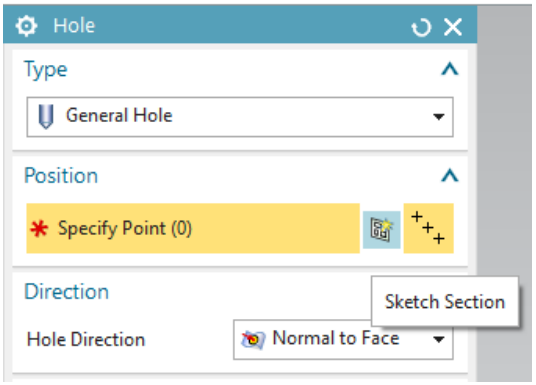
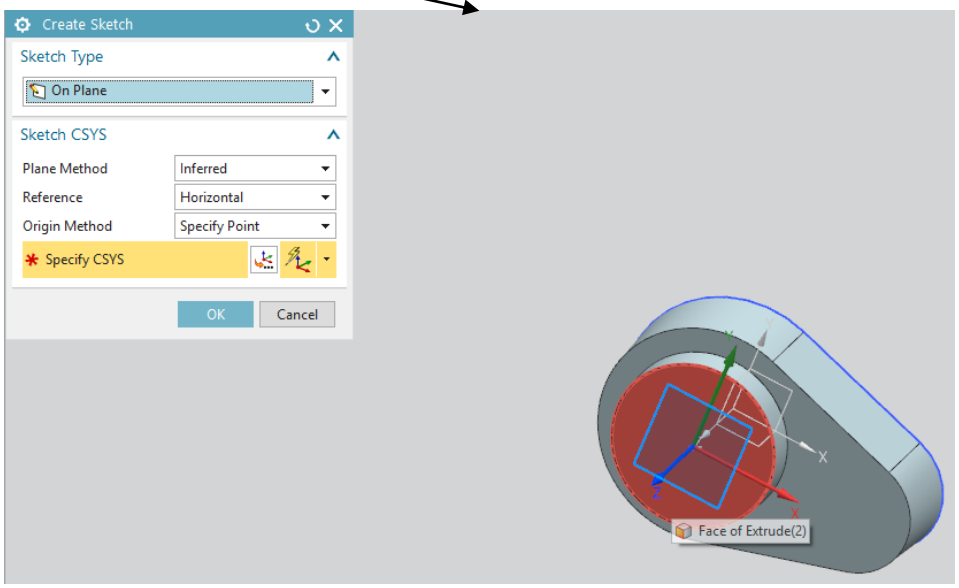
In the *Hole* dialog box you have to specify the center point of the hole, which will be the origin.

- Under the field *Position*, click on the button *Sketch Section*.

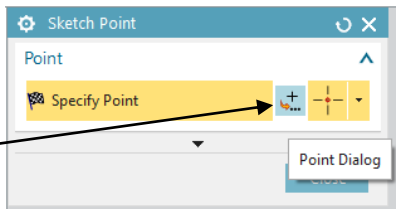


A new dialog box appears named *Create Sketch* which will help us
for creating a sketch for the hole.

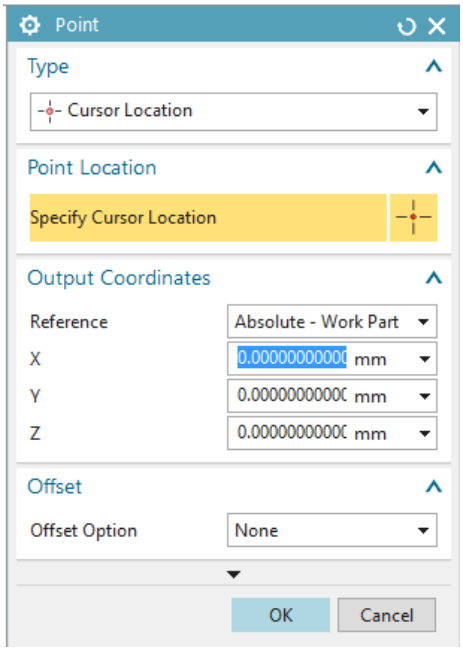
- Select the top of the cylinder as reference plane and click on *OK*.

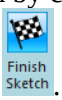


- The plane where the hole will begin has been defined. In the new *Sketch Point* dialog box, click on the *Point Dialog* button.



- The *Point* dialog box opens. Make sure that the X, Y and Z *Output Coordinates* are all set to 0 and click *OK*.



- Close the *Sketch Point* dialog box and exit the sketch by clicking the *Finish Sketch* button 

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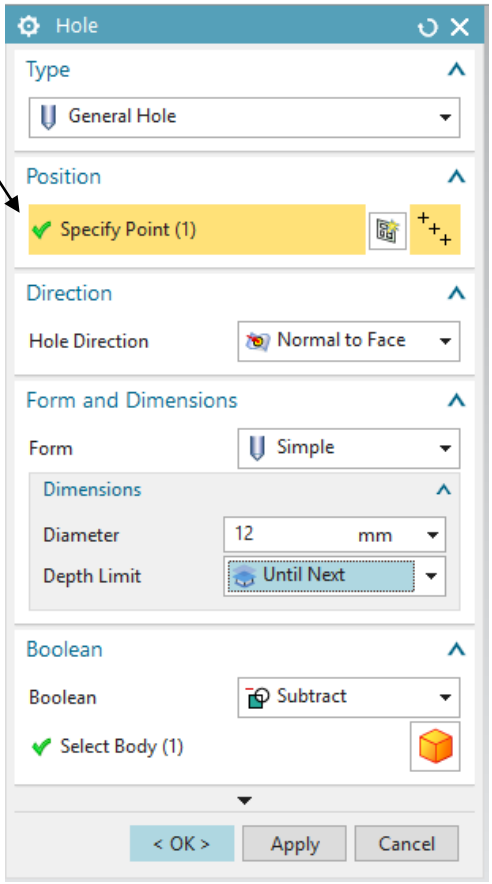
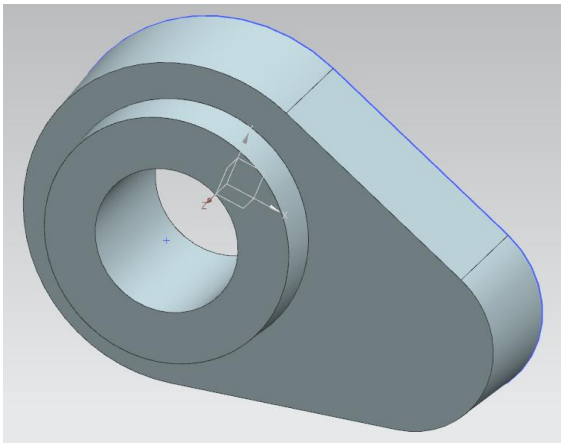
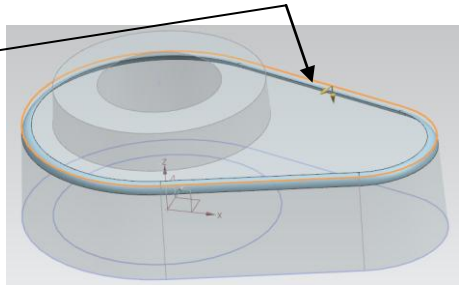
- Now, the *Position* field of the *Hole* dialog box should contain one specified point.
- In the field *Form and Dimension* select as form a simple hole, a diameter of **12 mm** and the *Until Next* depth limit.

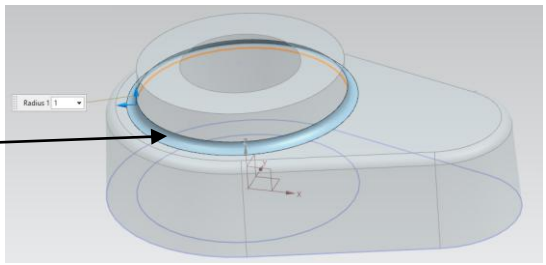


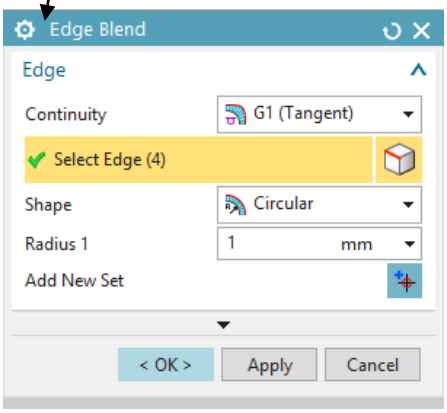


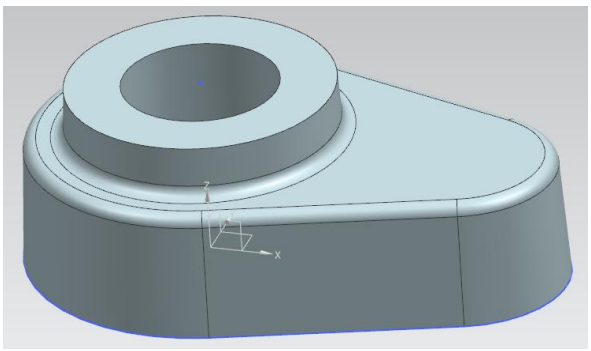
7 - Adding fillet.

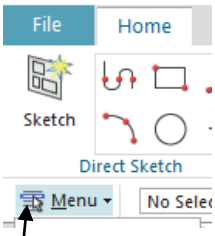
- Click the *Edge Blend* button .
- Select the four upper edges of the oblong contour.
- Set a radius of **1 mm** and click *OK* to confirm.
- Redo the above operations on the upper edges between the cylinder and the oblong contour.







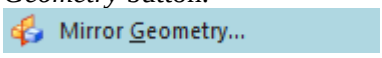




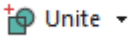
8 – Symmetric copy of the part.

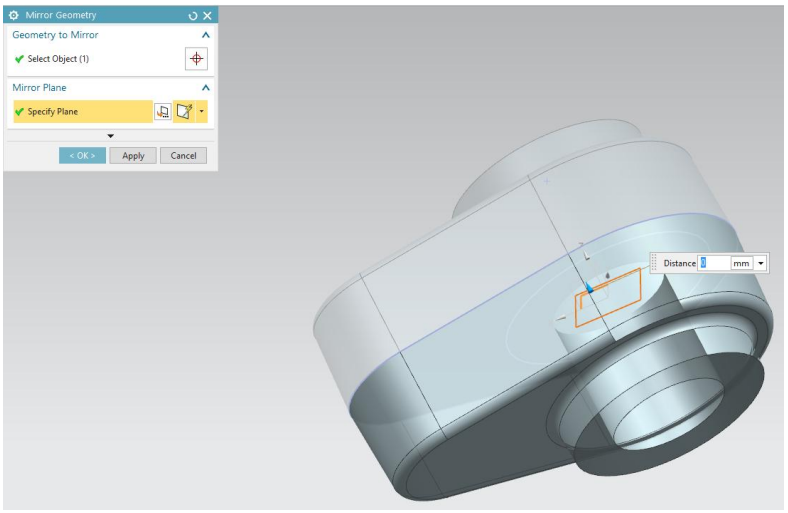
The part being symmetrical, it was much easier to not only draw one half and then duplicate the volume obtained.

- Click on the *Menu* button, then *Insert*→*Associate Copy*→*Mirror Geometry* button.



- Select the whole object you have so far and select as *Mirror Plane* the XZ plane.
- Click *OK* in the dialog box that appears.
- Finally, unite the object with its symmetric copy using the *Unite* button.

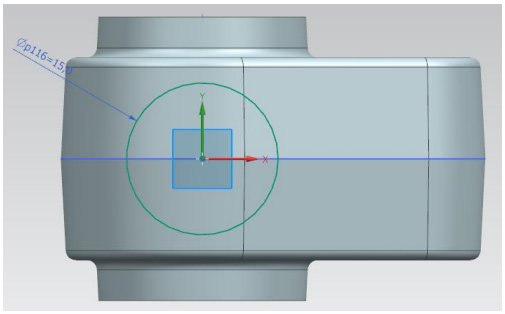


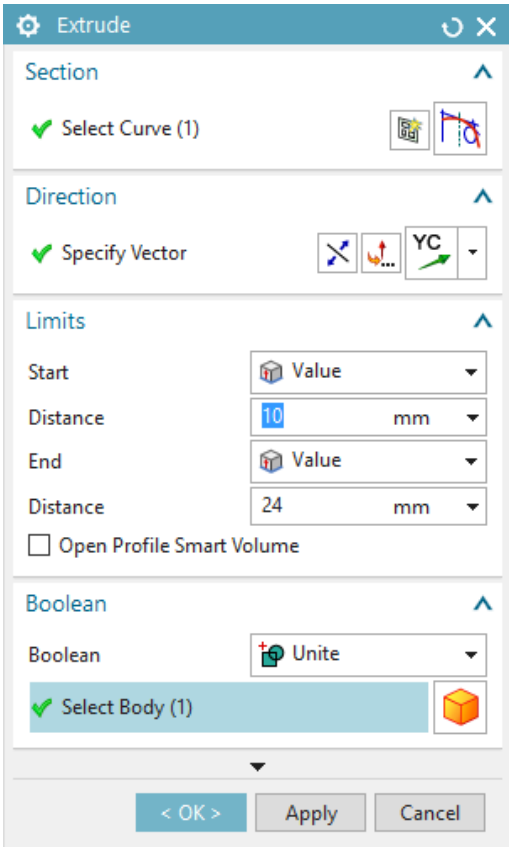


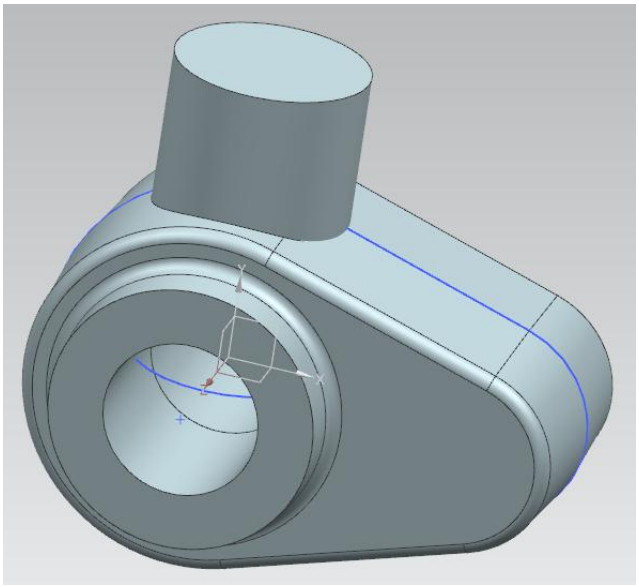
9 - Creating a third extrusion.

- Select the XZ plane and enter sketch mode.
- In this sketch, create a circle centered at the origin.
- Using a constraint, impose to the circle a diameter of **15 mm**.
- Get out of sketch mode and click the *Extrude* button 

.
- In the box dialog just opened select *Unite* in the *Boolean* field and select the object you construct so far.
- As direction vector, specify the YC axis.
- Set **10 mm** as the start distance.
- Set **24 mm** as the end distance.
- Click *OK* to confirm.



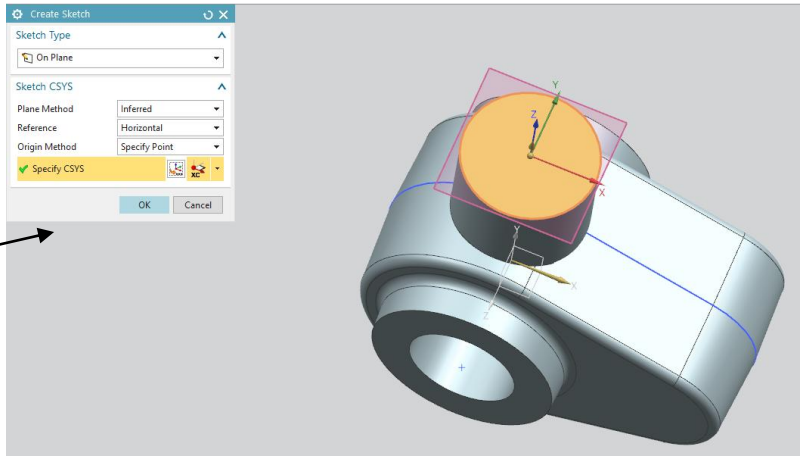


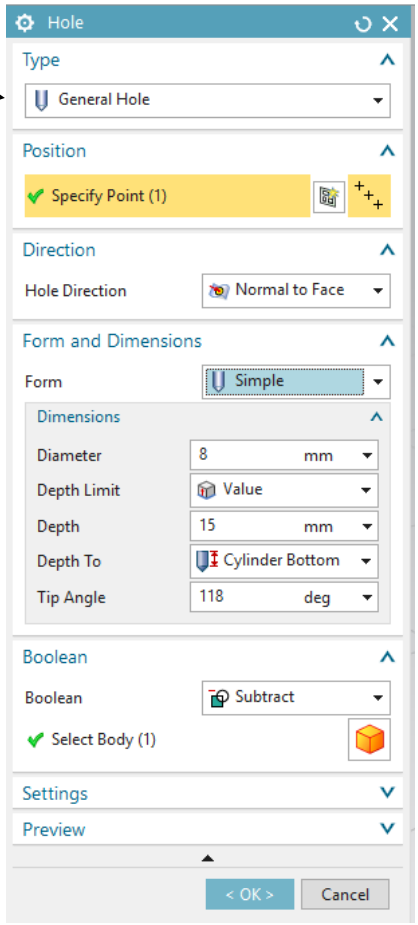


10 - Blind hole creation.

- Click the *Hole* button 

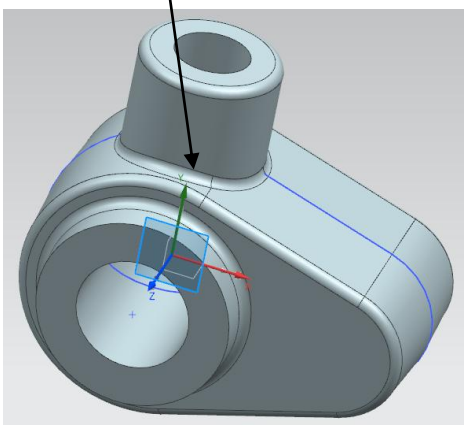
.
- As already done earlier, create a sketch defining the hole position. This time the sketch plane contains the upper face of the cylinder you just extrude.
- In the *Point* dialog box, make sure that the X, Y and Z *Output Coordinates* are all set to 0.
- Set a simple hole of **8 mm** diameter and **15 mm** depth.
- Also set the *Depth To* field to *cylinder Bottom*.





- Click the *Edge Blend* button 

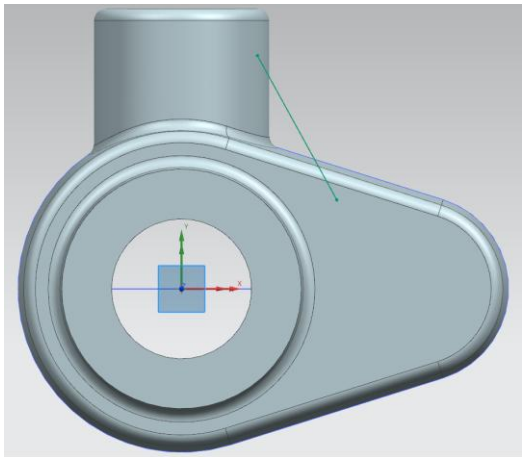
and select the side of the upper cylinder.
- Set the fillet radius to **1 mm**.
- Confirm.
- Redo the same operation for the edge at the junction of the cylinder with the body.



11 - Creating a stiffener.

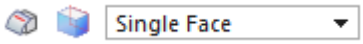
Stiffeners are used to stiffen a body subjected to mechanical stress.

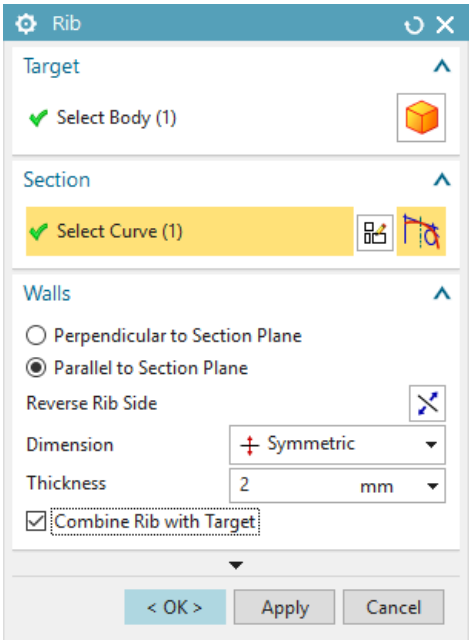
- Enter sketch mode and select the XY plane.
- Disable the *Continuous Auto Dimensioning* option.
- Click the *Line* button  and draw a line in arbitrary oblique position as indicated here below.

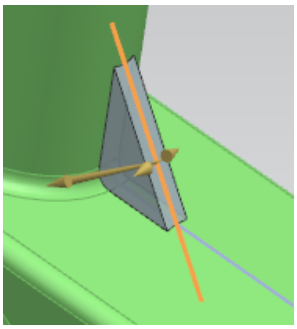


- Get out of sketch mode and click the *Rib* button 

located under *Menu →Insert →Design Feature*.
- In the *Rib* dialog box set as *Target* the body you have so far. Set as *Section* the curve you just sketched.
- In the *Walls* field select the option *Parallel to Section Plane*, with *Dimension* set to *Symmetric*.
- Set the *Thickness* to **2 mm** and make sure that the option *Combine Rib with Target* is checked.
- Finally, click *OK* to validate the creation of the stiffener.
- **Note**: if you do not manage to select the wanted faces, check if the selection rule is set to *Single Face*.





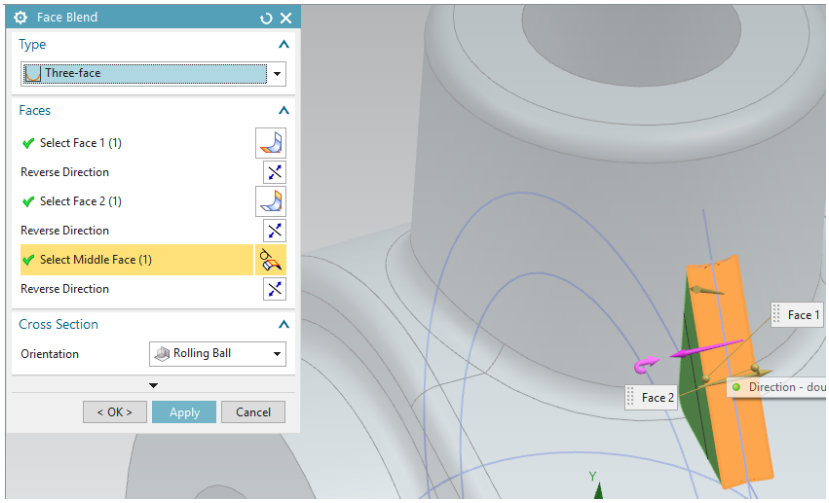


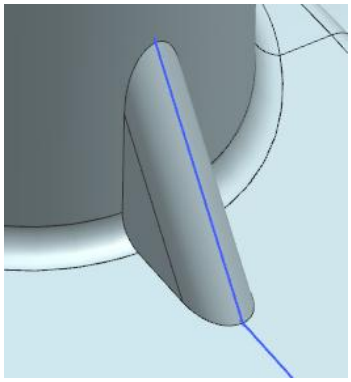
You will now add a three-face blend to the stiffener newly created.

- Click the *Face Blend* button 

located under the *Edge Blend* 

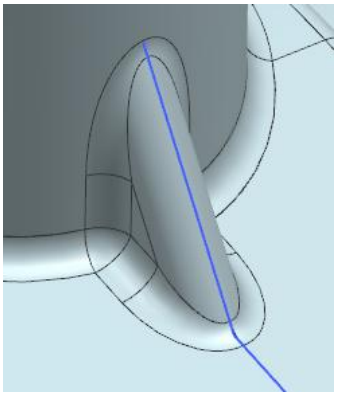
button already used.
- In the *Face Blend* dialog box, select *Three-face* as *Type*. As faces 1 and 2, select the two vertical visible faces of the stiffener.
- As middle face, select the remaining visible face of the stiffener.
- Click *OK* to validate.





The next step consists in adding a fillet around the edge of the stiffener

- Click the *Edge Blend* button (which is now under the *Face Blend* button).
- Select the edge of the stiffener (eg the junction of the stiffener and the upper cylinder)
- Set a radius of **1 mm**, and click *OK* to confirm.

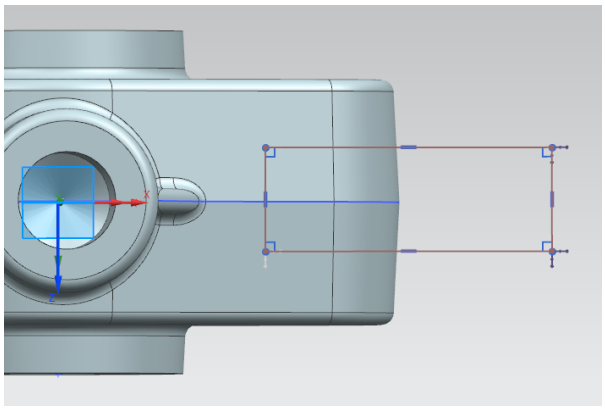


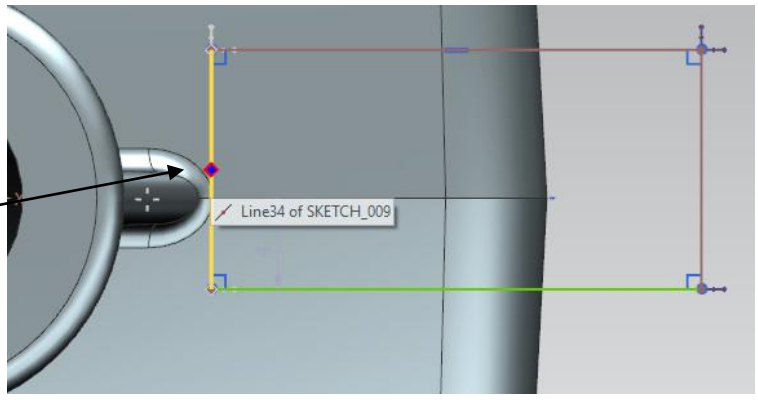
12 - Creating a cutout.

You are going to draw a rectangular shape to create a cutout in the part.

- Select the XZ plane and enter sketch mode.
- Draw a rectangle in the approximate position shown against (2 clicks for two opposite vertices of the rectangle) by using the rectangle button  .

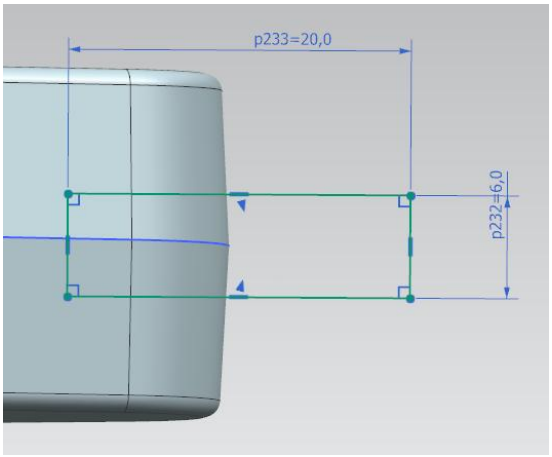
- Click on the *More* button and select *Geometric Constraints* in the *Sketch Constraints* field.
- In the dialog box, select *Point On Curve* as constraint.
- As *Object to Constraint* select the midpoint of a vertical edge of the rectangle.
- As *Object to Constraint to*, select the x-axis.

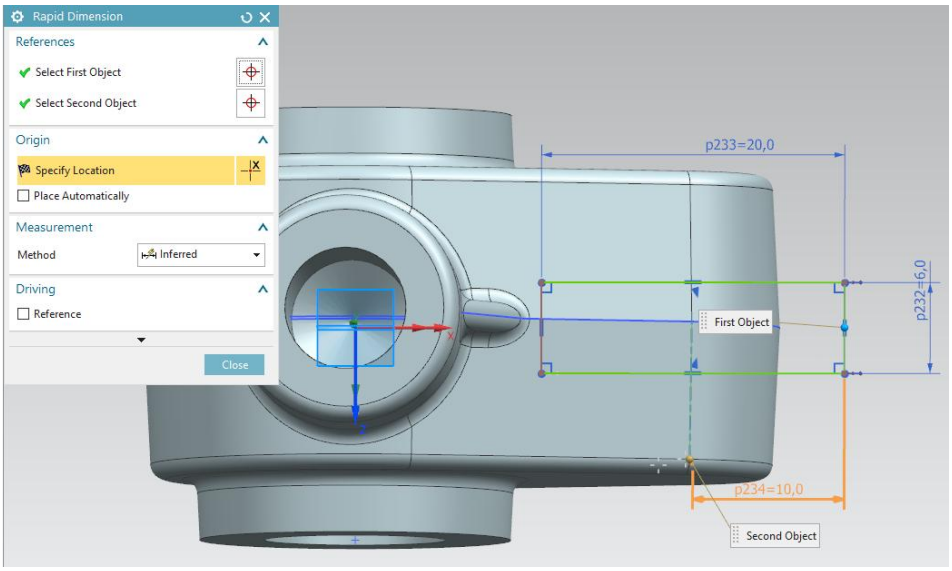




- Click on the *Rapid Dimension* 

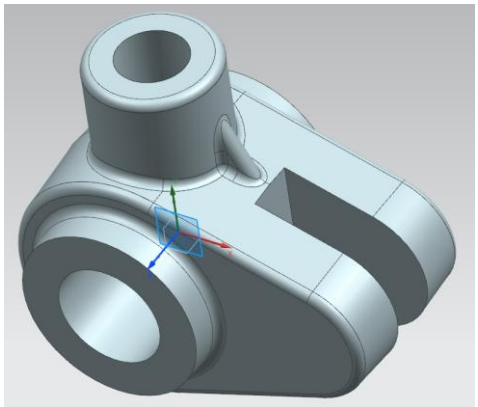
button and constraint the length and width of the rectangle to respectively **20 mm** and **6 mm**.
- In the *Rapid Dimension* dialog box, select a point of the rectangle and a point of the body as shown in the hereafter figure. Impose a distance of **10 mm** between those two points.

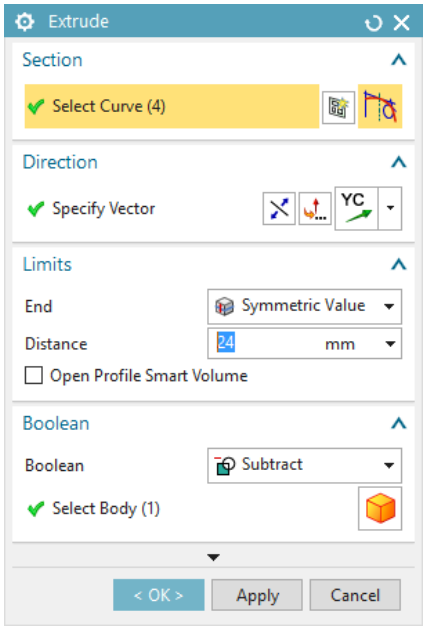




- Get out of sketch mode.
- Click the *Extrude* button 

.
- Make sure that the *Direction* field is the YC axis.
- In the *Limit* field, select as *End* value *Symmetric Value* and as *Distance* **24 mm**.
- Finally, set the *Boolean* field to *Substract*.
- Click *OK* to confirm.



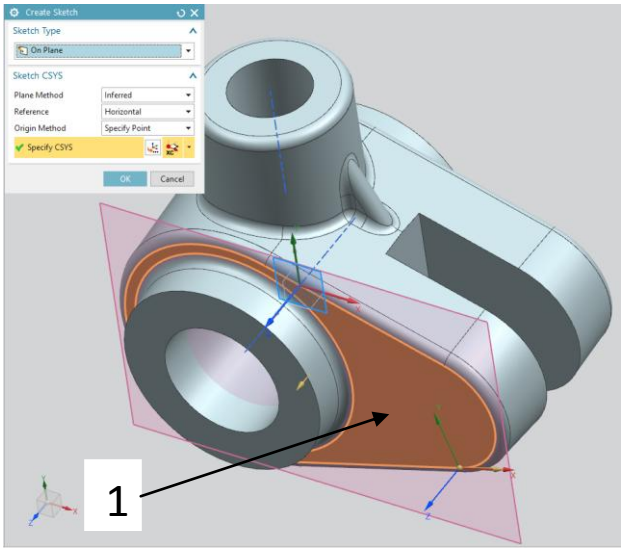


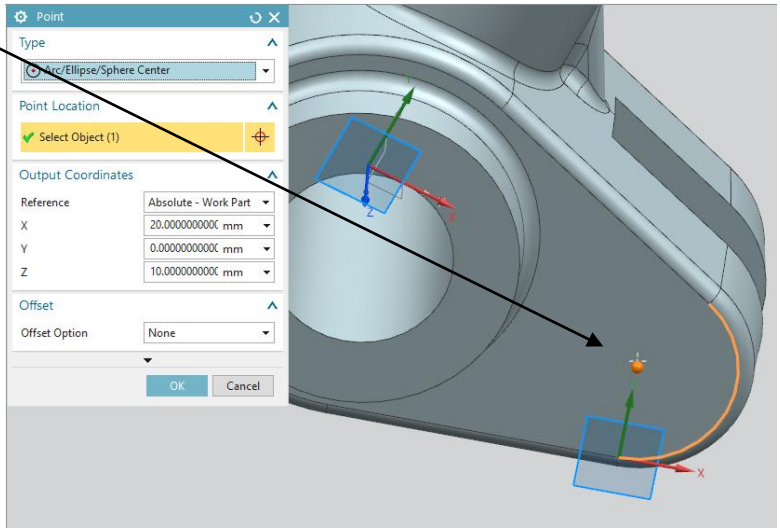
13 - Drill a hole.

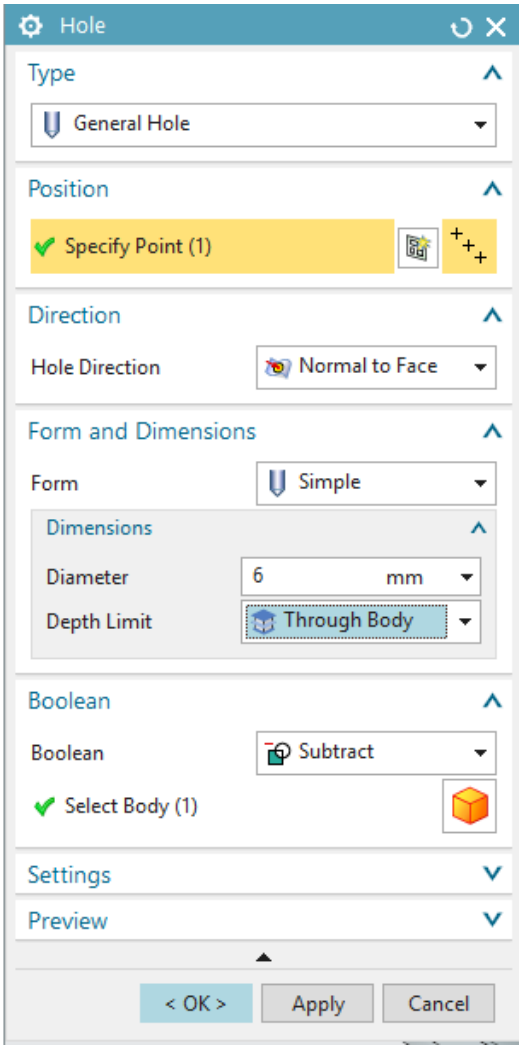
You will now drill a coaxial hole through the piece.

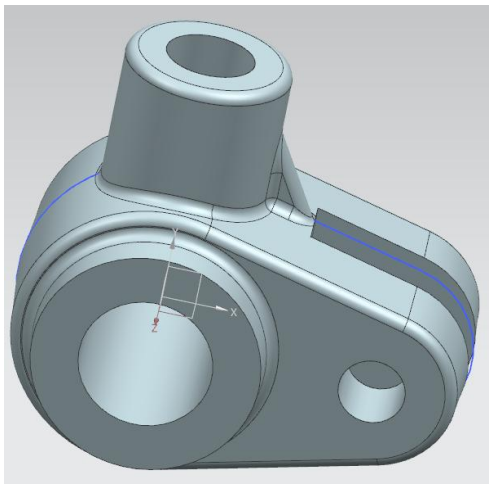
- Click on the *Hole* button 

. As already done, you will draw a sketch defining the hole position.
- In the *Create Sketch* dialog box, select face 1 as reference plane.
- In the *Point* dialog box, select the option *Arc Ellipse/Sphere Center* in the field *Type*.
- For specifying the *Point Location*, click on the right arc of the extruded oblong contour.
- In the *Hole* dialog box, set the diameter to **6 mm** and set the depth limit to *Through Body*.
- Confirm via *OK*.





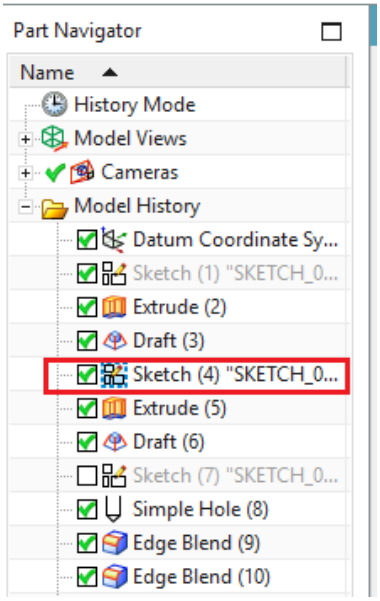




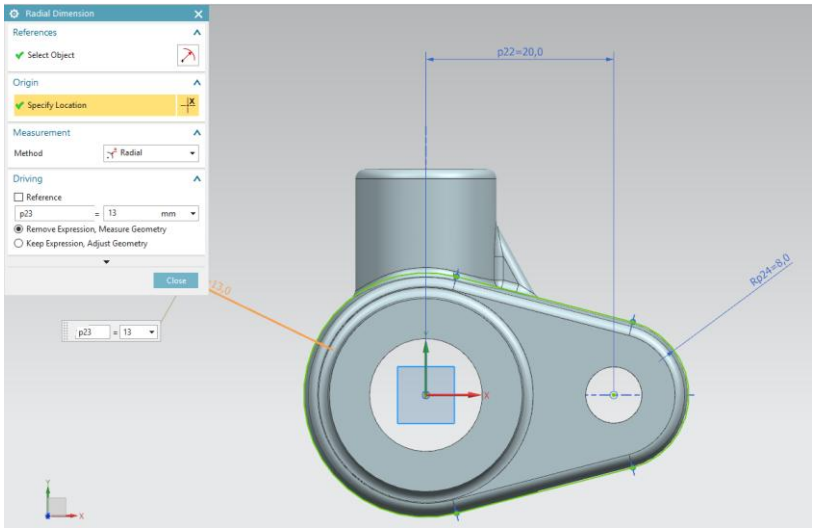
14 – Modification of the geometry.

While you had almost finished, you notice that you made a mistake with respect to plans that are provided, the first arc of the oblong profile should measure **13 mm** and not **14 mm**!

- Double click on the sketch corresponding to the oblong contour in the tree on the left of the screen (*Part Navigator*).

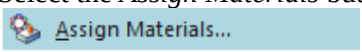


- Double click the dimension of the left arc of the oblong contour and replace the value of **14 mm** by **13 mm**.
- Confirm with *OK*.
- Get out of the sketch mode.
- The program automatically calculates the changes to make.
- The geometry is updated and reflects your changes.



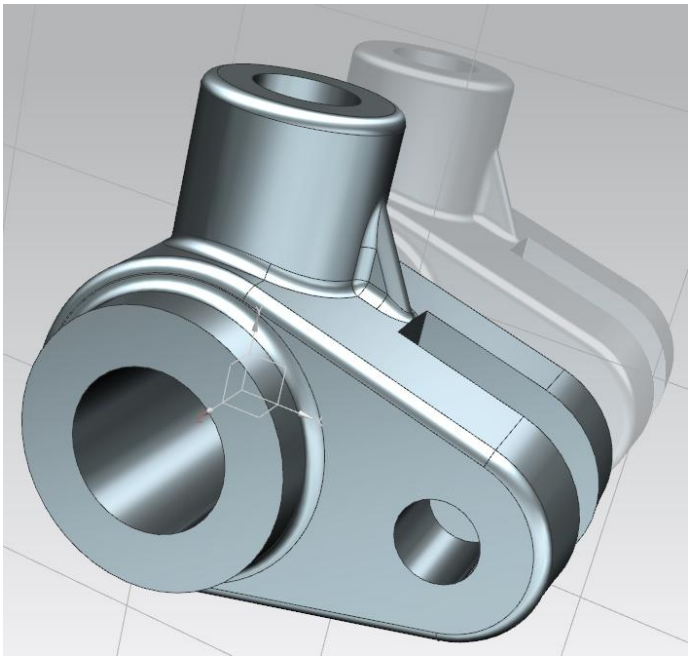
15 - Adding a material to the part.

Applying a material to a part not only provides a more realistic rendering, but also allows making calculations of stresses on the part according to the characteristics of the material used.

- Select the *Assign Materials* button 

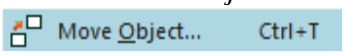
located in *Menu →Tools →Materials*.
- In the *Assign Materials* dialog box, select your body.
- Click on the material *Iron_40* in the material list.
- Click *OK* to confirm.
- In the header of the toolbar, click on *Render*.
- In the new toolbar, click on the *True Shading* button 

.



16 – Moving an object.

You just realize that you used the wrong planes during the whole object design. Indeed, the ground plane should lie under the object.
In order to correct that, you will rotate the object by **90°** along the x axis.

- Select the *Move Object* button 

under *Menu→Edit*.
- In the *Move Object* dialog box set the *Motion* field to *Angle*, the rotation vector to XC and the axis point to the origin.
- Finally, Set the angle value to **90°** and make sure that the *Move Original* radio button is checked.
- Click *OK* to validate.