Using Siemens NX 11 Software

The connecting rod

Based on a Catia tutorial written by Loïc Stefanski.

At the end of this manual, you should obtain the following part:
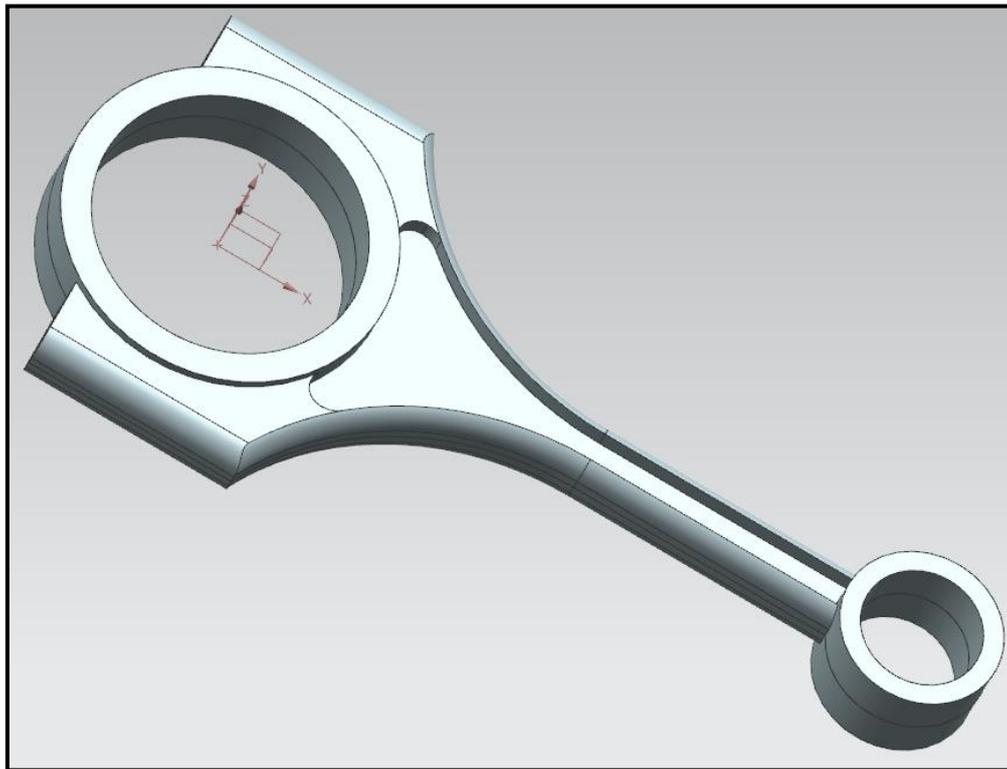
1 – Introduction.

Start NX 11 and open a new file of type *Part.*
Before any other operations, open a new file of type *Model*.
- In toolbar, select *New*.
- In the Filter list, select Model.
- Set the file name to *rod* and set its folder.
- Click *OK* to confirm.
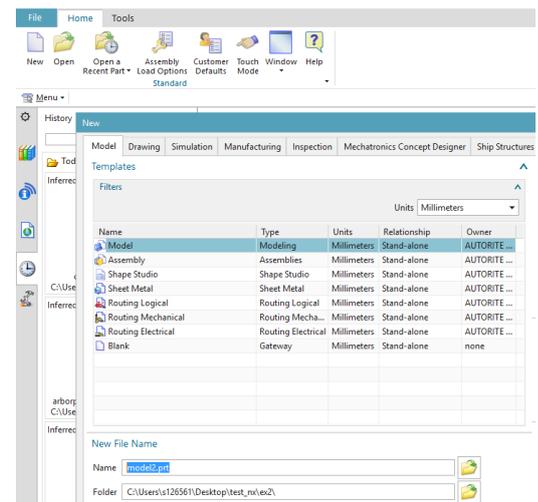
2 – Creation of a cylinder.

First it is needed to draw the cylinder profile in the sketcher.

- Click on the *Sketch* button. **Sketch**
- Create a new Sketch and select the plane XY in the *Create Sketch* dialog box.
- Draw a circle with an arbitrary radius (button ◯) centered at the origin of the axes.
- Double-click on the dimension and define a radius of **27 mm**.

- Click on the icon **Finish Sketch** to get back to the 3D mode.
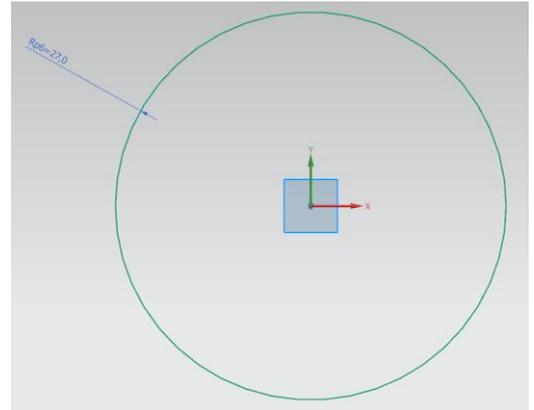
We will now extrude the last sketch.

- Select the *Extrude* button **Extrude**.
- In the *Extrude* dialog box, set the direction of extrusion to axis ZC.
- Under the tab *Limits* set the start distance to **9 mm** and the end distance to **0 mm**.
- Under the tab *Offset* set a two-sided offset, a start value of **0 mm** and an end value of **6 mm**.
- Click *OK* to confirm.
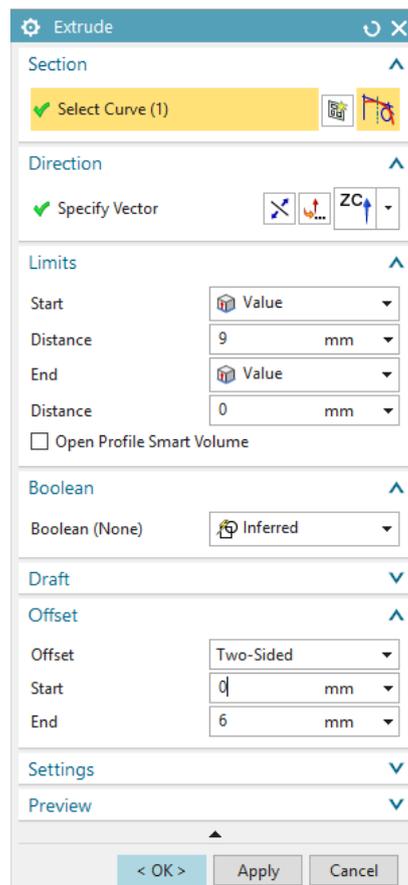
- Before to continue, disable the option *Continuous Auto-Dimensioning* under the *More* button located in the toolbar.
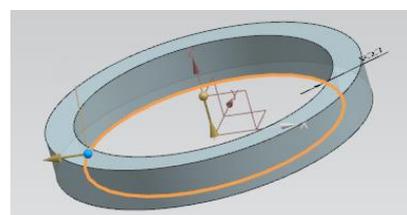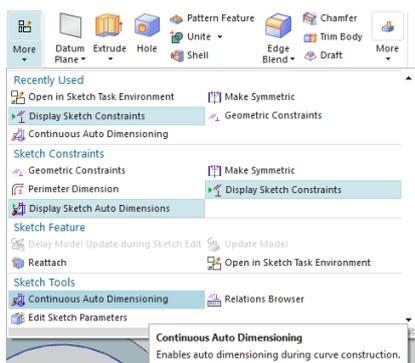
3 – Creation of the small cylinder.
The procedure is the same as before.

- Go to into the Sketcher module and select the XY plane.
- Draw a circle near the first one, with its centre located on the horizontal axis. Enforce this by imposing a *Point on Curve* geometric constraint (located under the *More* button of the toolbar).

- Click on the *Rapid Dimension* button and in the *Rapid Dimension* dialog box select the two circle centers and impose a length of 150 mm.
- Under the button *Rapid Dimension*, select *Radial Dimension*.
- Impose for the circle a radius of 12 mm.

- Exit of the *Sketch* mode (icon).
- Finally, create an extrusion of 10 mm then a shell of 4 mm external thickness for the small circle.
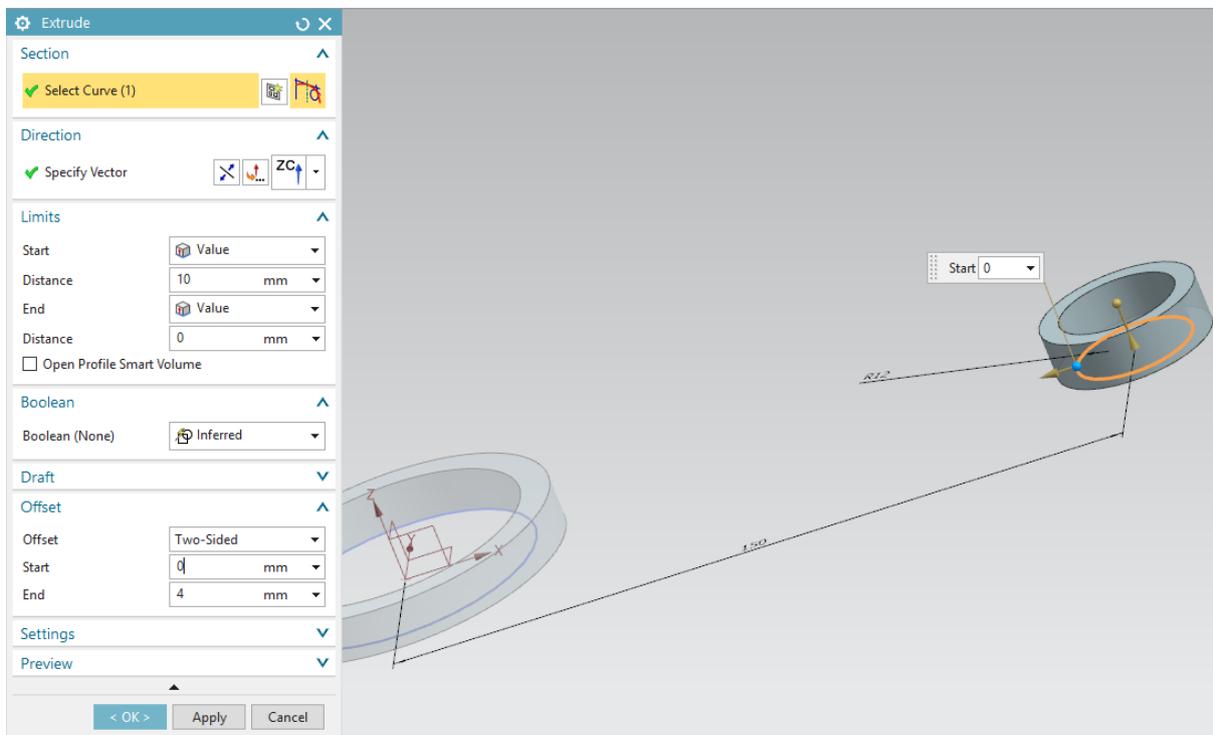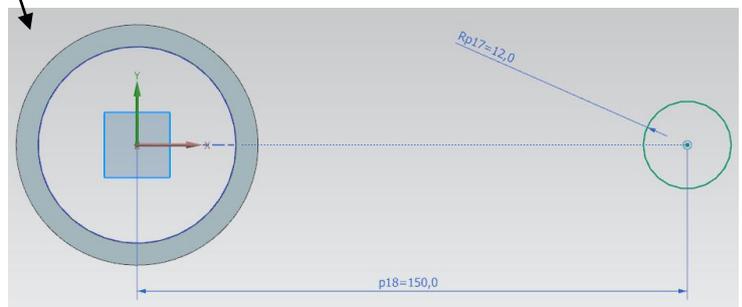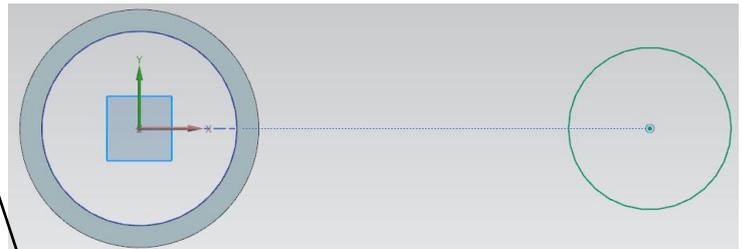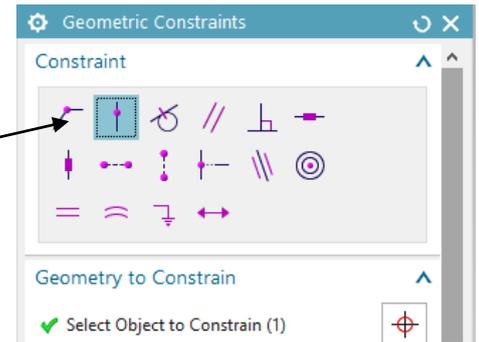
4 – Creation of a datum grid.

A datum grid will guide us in drawing the next sketch.

- First, create a datum plane by clicking *Menu->Insert->Datum/Point->Datum Plane*. `Datum Plane...`
- Select as reference the XY plane.
- Move the lowest left corner of the *Datum Plane* on the origin.
- Resize the plane to a *Length* of **170 mm** and a *Height* of **60 mm**.
- Then, create a datum plane grid by clicking *Menu->Insert->Datum/Point->Datum Plane Grid*. `Datum Plane Grid...`
- In the *Datum Plane Grid* dialog box, set the *Major Line Spacing* to **10 mm**.
- In the *Select Datum Planes and Grids* field, select the datum plane we just defined.
- NOTE: you will need to select the datum plane in the *Part Navigator*. `☑ Datum Plane (25)`
- Validate.
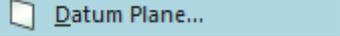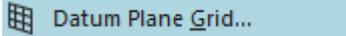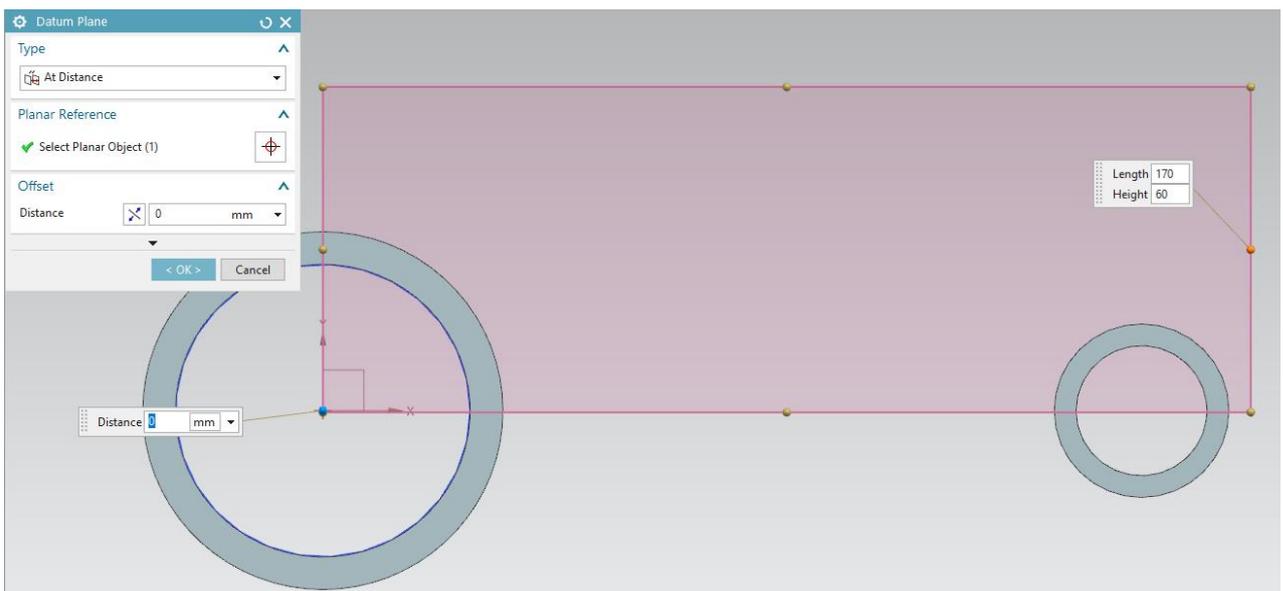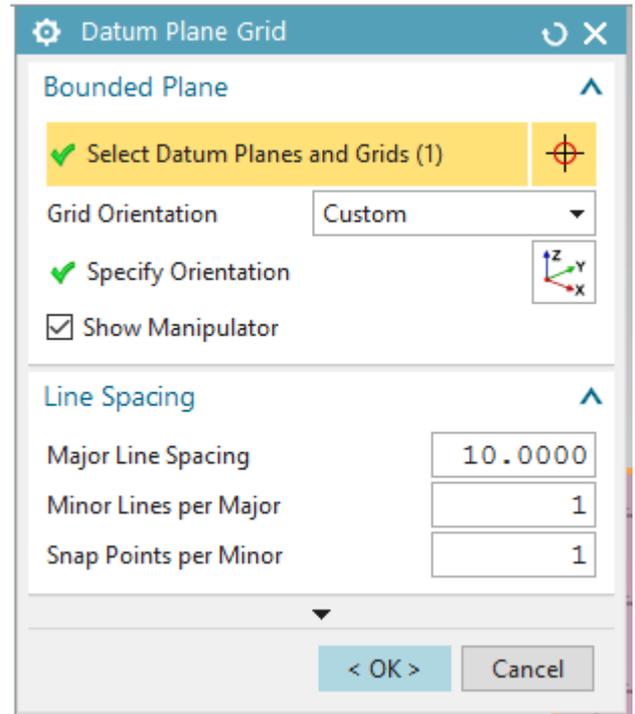
5 – Creation of the connecting rod.

By symmetry, we can draw only one part of the body.

- Enter in the *Sketch* mode in the plane XY.
- Click on the *Profile* button and create a profile similar to the profile illustrated on the right. This profile is made of several line segments (Start from point A to end at point B).
- The circle arc is then created to close the curve using the tool *Arc* of *Profile*.
- Impose a tangent constraint at point B.
- The curve should now be closed.
- Exit to 3D mode and extrude this drawing by 7 mm.
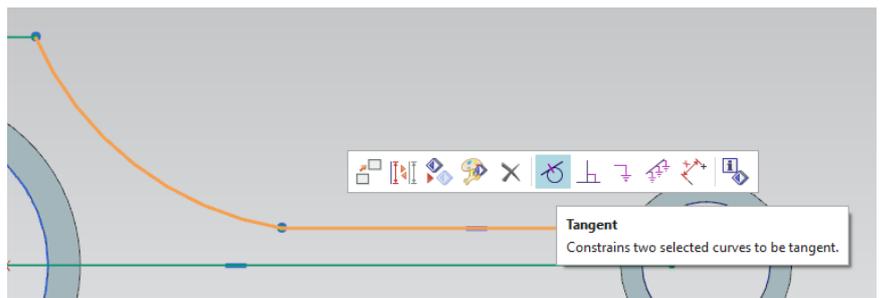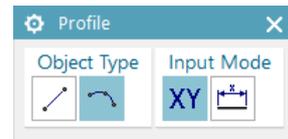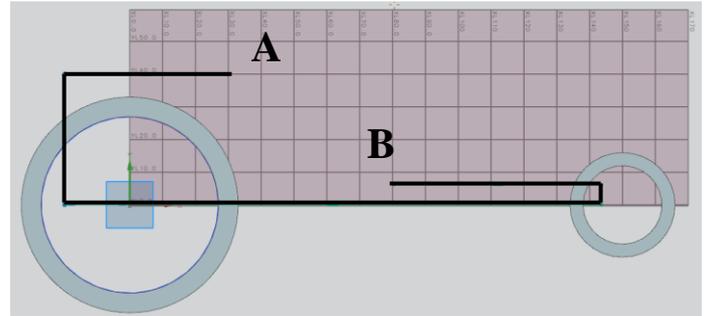
We now have to create a *pocket* in this last part.
- Go in the *Sketch* mode and select the upper face. You should be drawing on this face.

We will draw offset curves of the last contour one by one and then connect these offset curves together.
- Extend the *Direct Sketch* toolbox by clicking on the little triangle button.
- In the *Curve Rule* drop-down menu (located below the *Extrude* button), select *Single Curve*. This will allow us to select the curves one by one.
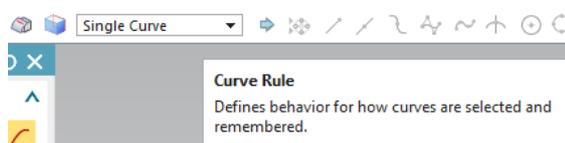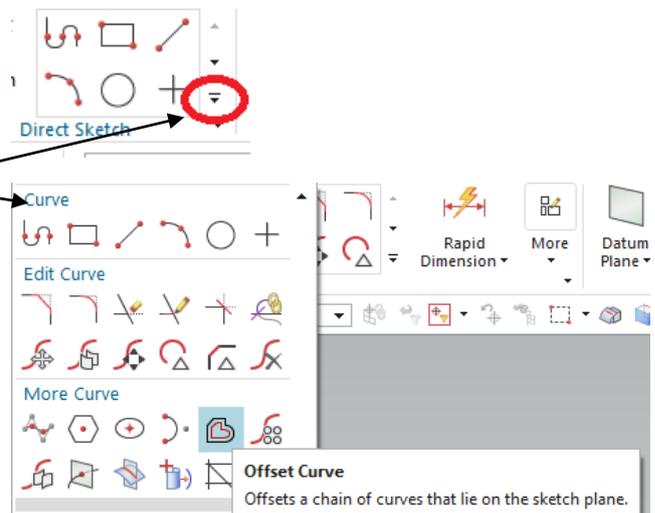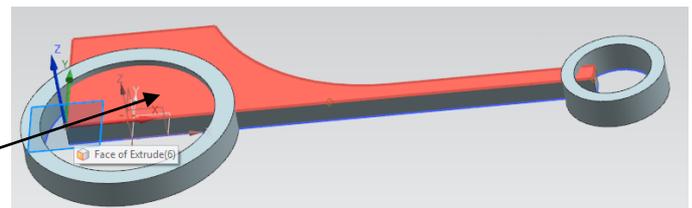
Drawing the offset curves.

- Select the top line of the last sketch and impose a distance offset of **8 mm** in the *Offset Curve* dialog box.
- Select the left line of the last sketch and impose a distance of **15 mm**.
- Select the bottom line of the last sketch and impose a distance of **0 mm**.
- Select the arc and its connected line at point B of the last sketch and impose a distance of **3 mm**.
- Select the right line of the last sketch and impose a distance of **2 mm**.
- WARNING: be sure to draw the offset curves in the right direction. If the direction is wrong, click on the blue arrow in order to reverse the direction.
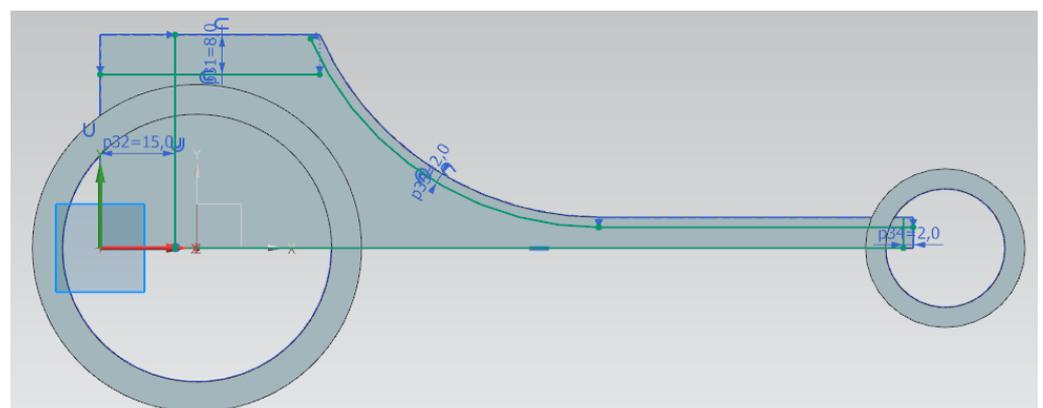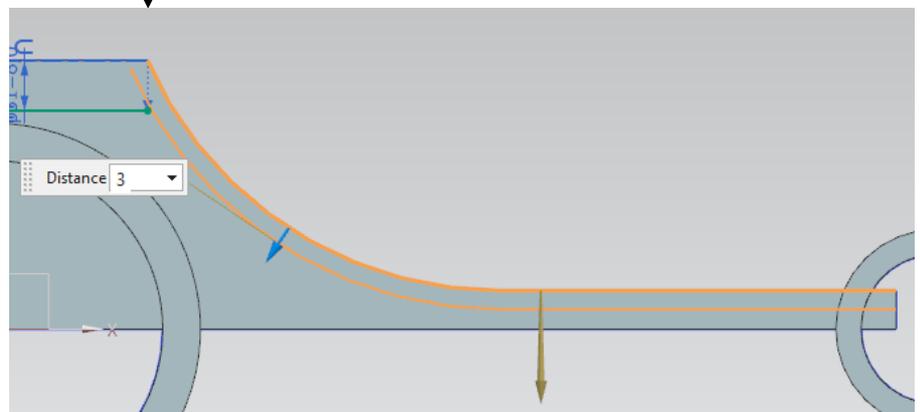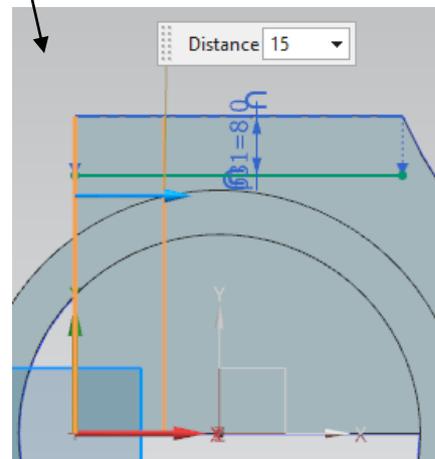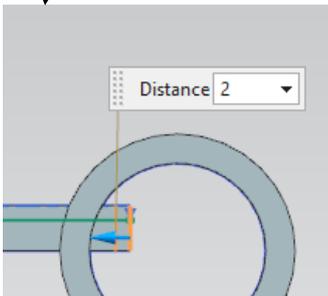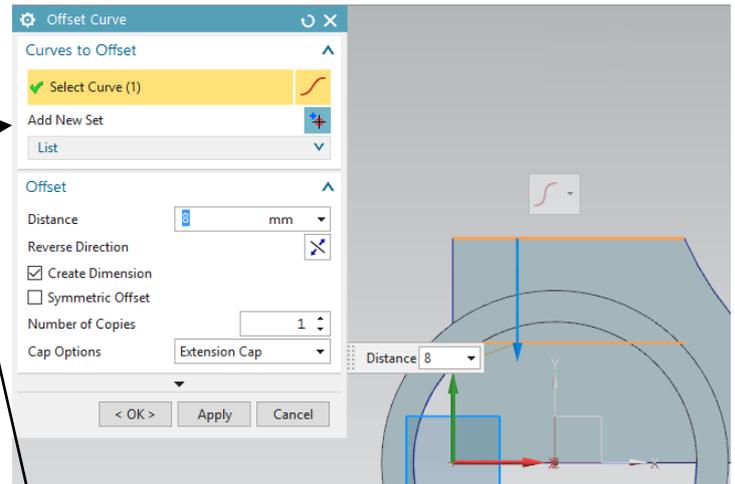
Connecting the offset curves.

As you can see on the last figure of the previous page, the offset curves are not connected. We will connect them using two tools located in the *Direct Sketch* toolbox:

Quick Extend and *Quick Trim* .

- Using the above two tools, extend and trim the offset curves in order to get the following closed sketch (blue curve).
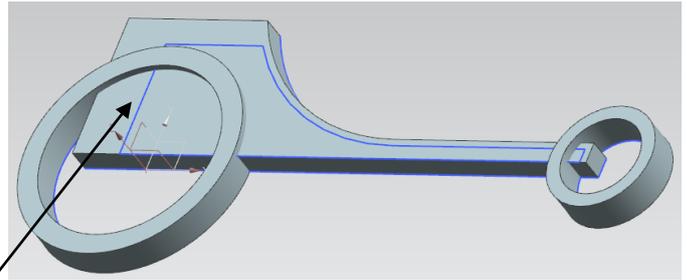
Creating the pocket.

We will use a Boolean operation associated to an extrusion in order to get a pocket.

- Exit to 3D mode

- Click on the *Extrude* button and select the last sketch.
- In the *Extrude* dialog box, select ZC as the axis of extrusion. Set the start distance to **0 mm** and the end distance to **-4 mm**.
- In the *Boolean* tab, set the *Boolean* field to *Subtract*.
- Click *OK* to validate.
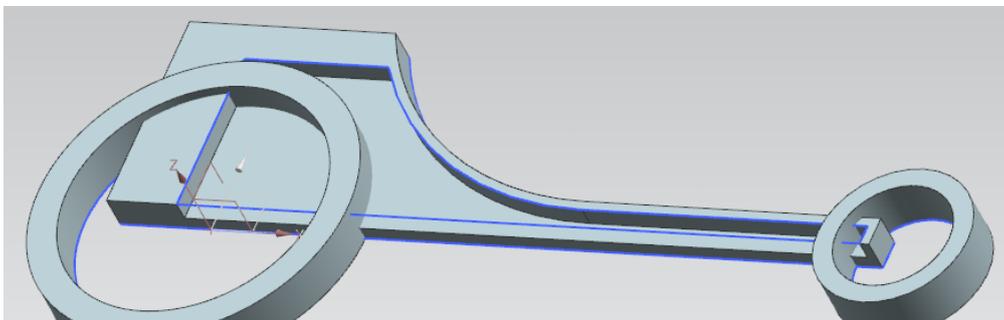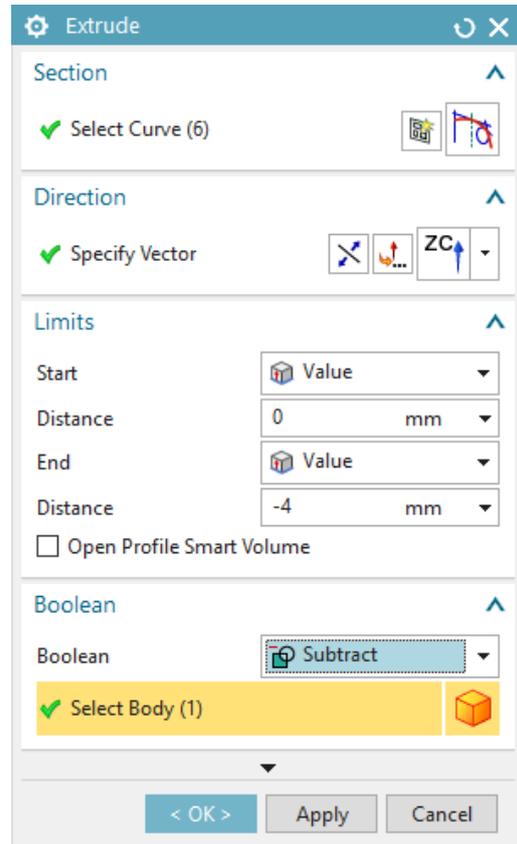
The remaining of the part is now created using symmetry.

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

   Mirror Geometry...

- Select the connecting part 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. Unite ▾
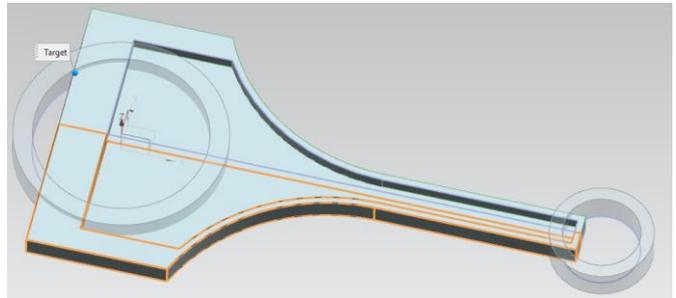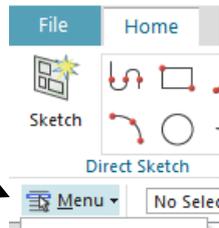
6 – Union of the two cylinders and the connecting part.

Here, we'll use the tool *Trim Body* in order to remove some part of the body while keeping some others.

- Click on the *Menu* button, then *Insert->Trim->Trim Body* button.

   Trim Body...

- First trim: select the connecting part as target and the external face of the big cylinder as *Tool Option*. Be careful to correctly trim the connecting part. Change the trim direction by clicking on the blue arrow if needed.
- **Note**: if needed, in the *Face Rule* drop-down menu, select *Single Face*. This will allow us to select faces one by one.

  Single Face ▾

- Validate with *OK* if the preview is satisfying.
- Second trim: redo the above operations by selecting this time the external face of the small cylinder as *Tool Option*.
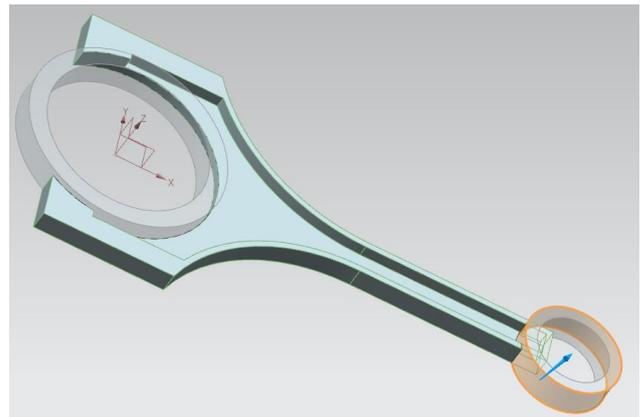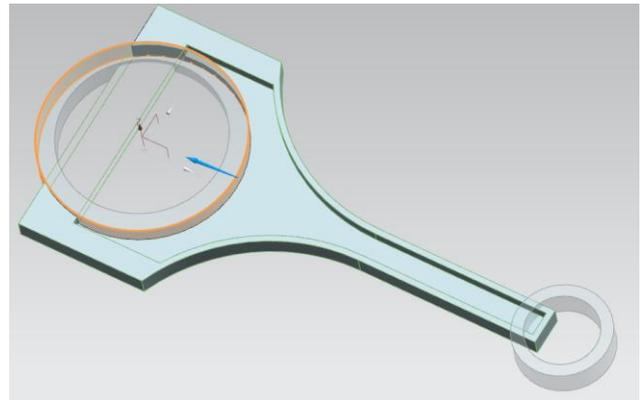
6 – Union of the two cylinders and the connecting part.

- Finally, unite the two cylinders with the connecting part using the last one as target and using the *Unite* button.
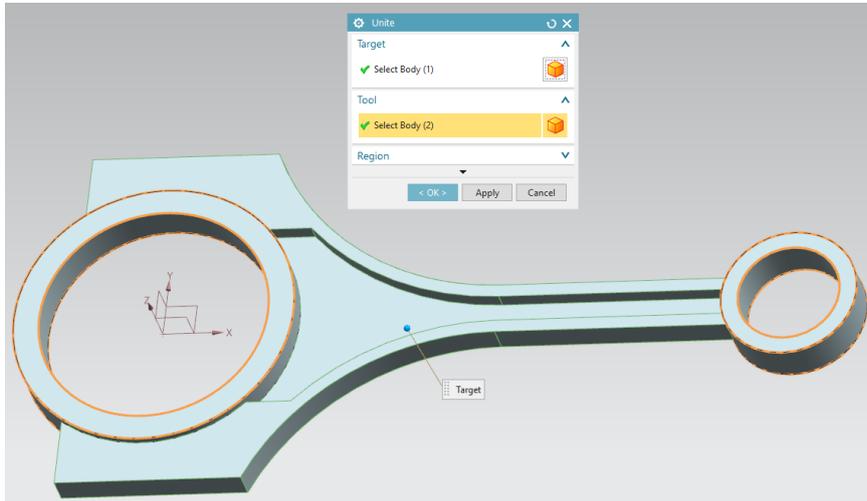
7 – Creation of the fillets.

Similarly to a real structure, it is necessary to introduce fillets.

- Click on the icon *Edge Blend* .
- Select the shown edges two by two (see right pictures).
- Choose a radius of **5 mm**.
- Validate.
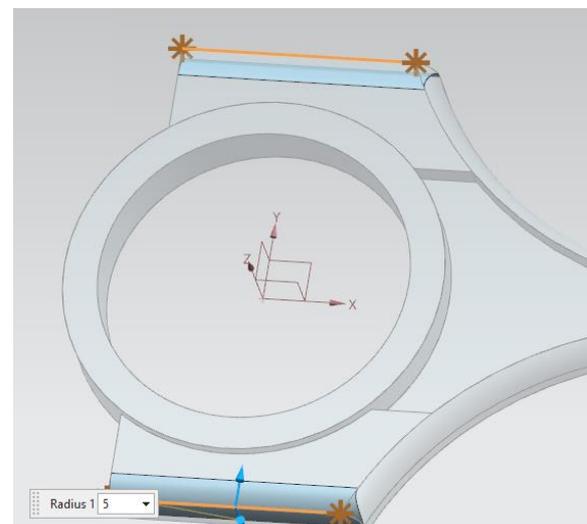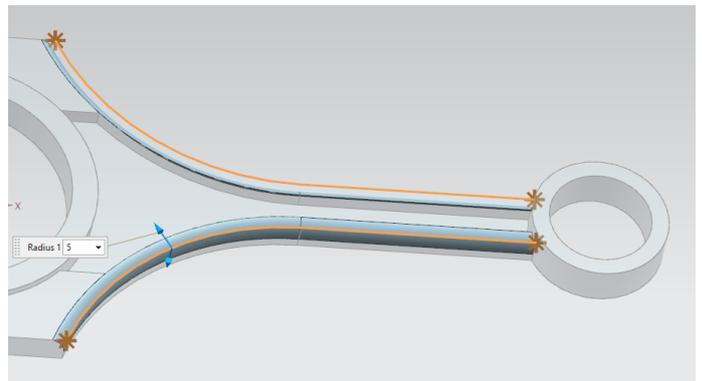
Insert also fillets between the cylinders and the connecting part.

- Use the button *Face Blend* Blend ▾ under

  Edge Blend Blend ▾ .
- In the tab *Type* select *Two-face*.
- Select the two showed faces in figure A and set a radius of **5 mm** and validate.
- Repeat the above operation for their symmetric counter-part in the rod.

- Redo a face blend with a radius of **4 mm** for the two faces showed in figure B and their symmetric counter-part.

A

B
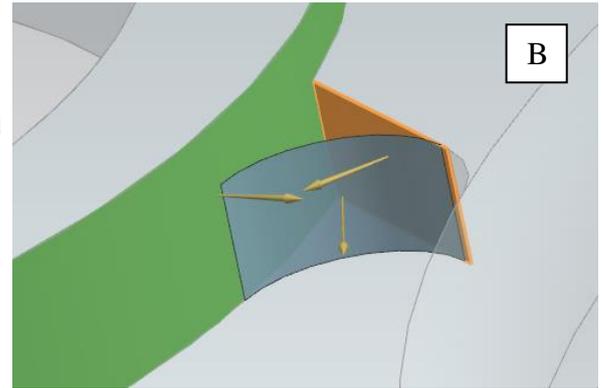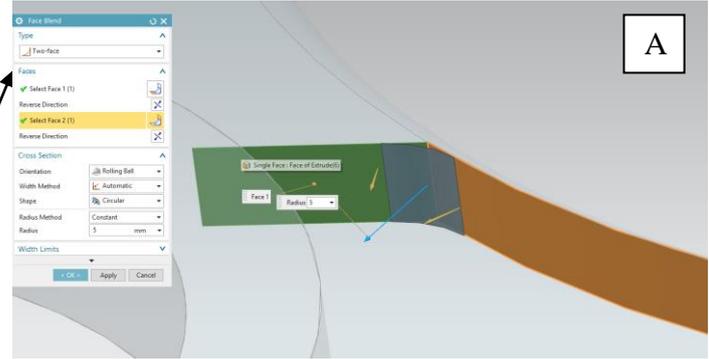
8 – Creation of the symmetric part.

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

  Mirror Geometry...
- Select the whole body and select as *Mirror Plane* the XY plane.
- Click *OK* in the dialog box that appears.
- Finally, unite the object with its symmetric copy using the *Unite* button.

  Unite ▾