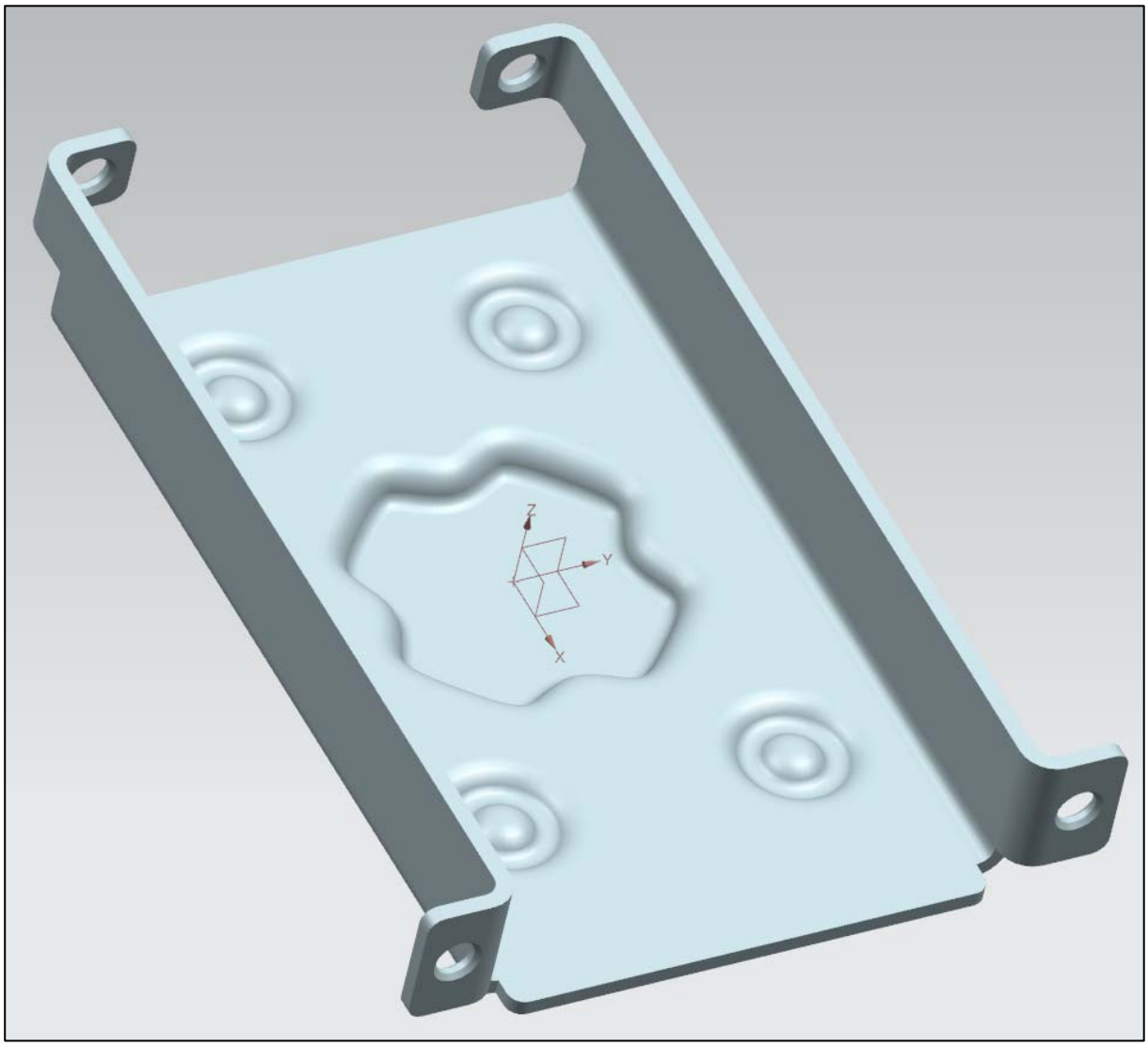
Using Siemens NX 11 Software

Sheet Metal Design - Casing

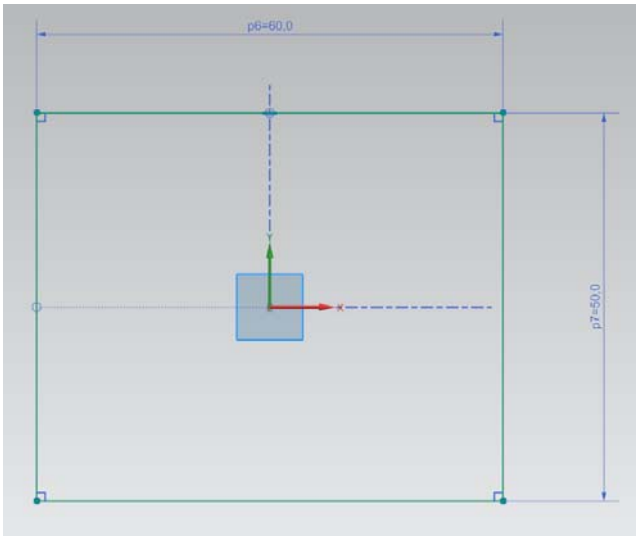
Based on a YouTube NX tutorial[1].



[1] *https://www.youtube.com/watch?v=-SIYi1Vz87k*

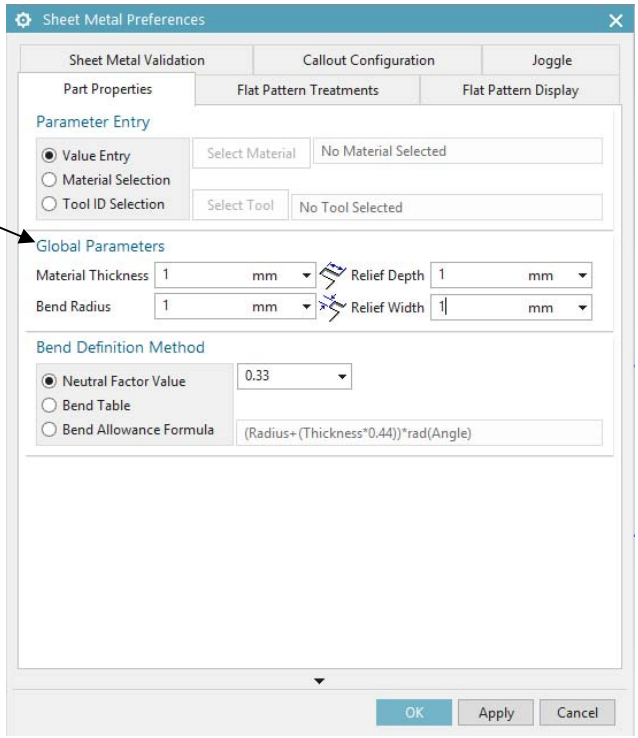
1 – Introduction.

- Start NX 11 and create a new *Sheet Metal* model called casing.*prt*.
- Create a sketch in the XY plane and draw a rectangle centred at the origin with an x-length of **60 mm** and a y-width of **50 mm**.
- Exit the sketch mode.



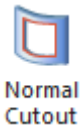
2 – Changing design parameters.

- Go to *Menu→Preferences→Sheet Metal…*
- In the *Sheet Metal Preferences* dialog box, set all the four *Global Parameters* to **1 mm**.

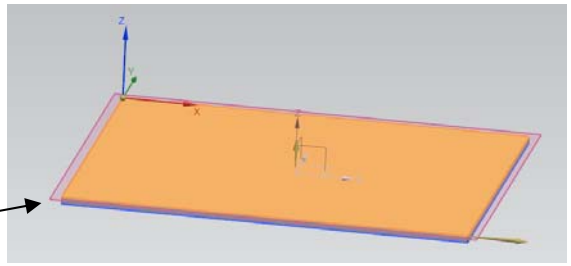


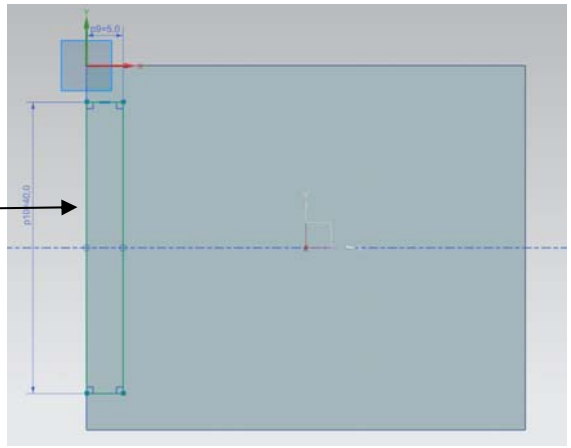
3 – Adding a solid sheet and cutout.

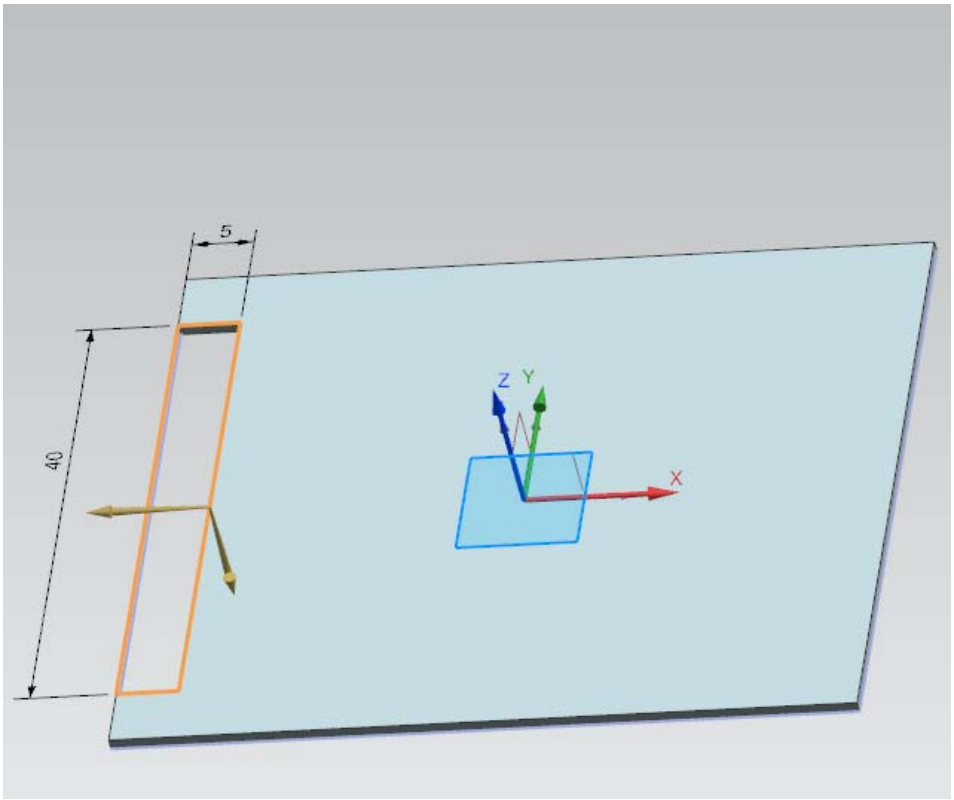
- Use the *Tab* button for creating a first rectangular metal sheet of **1 mm** in thickness.
- Create a new sketch in the plane **P** containing the upper face of the rectangular sheet and parallel to the XY-plane.
- In that sketch, draw a rectangle symmetric w.r.t the x-axis which is coincident with the left edge of the rectangular metal sheet. The rectangle is **5 mm** wide and **40 mm** long.
- Exit the sketch mode and use the *Cutout* button



for making a rectangular hole in the metal sheet.

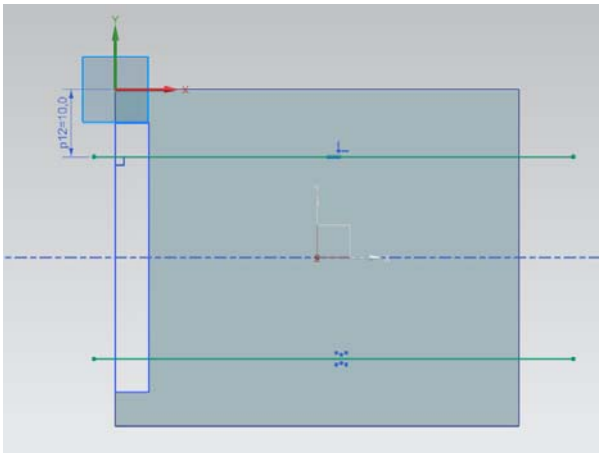


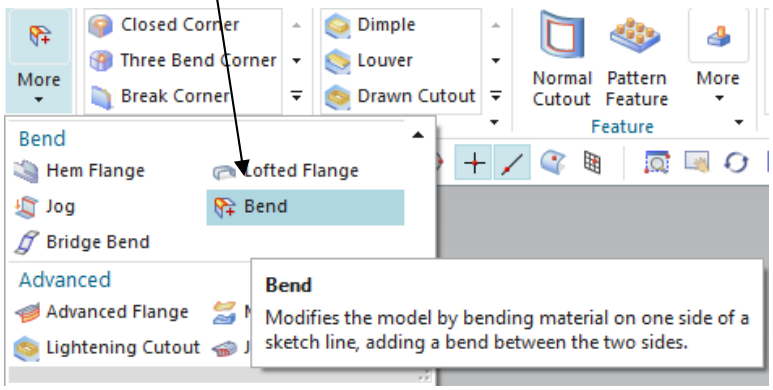




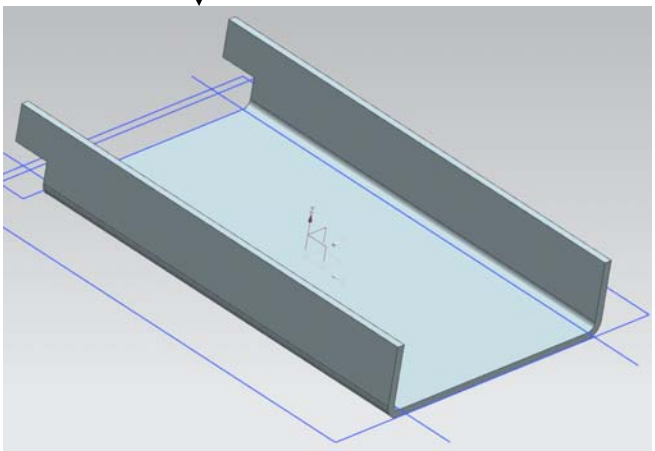
4 – Bending.

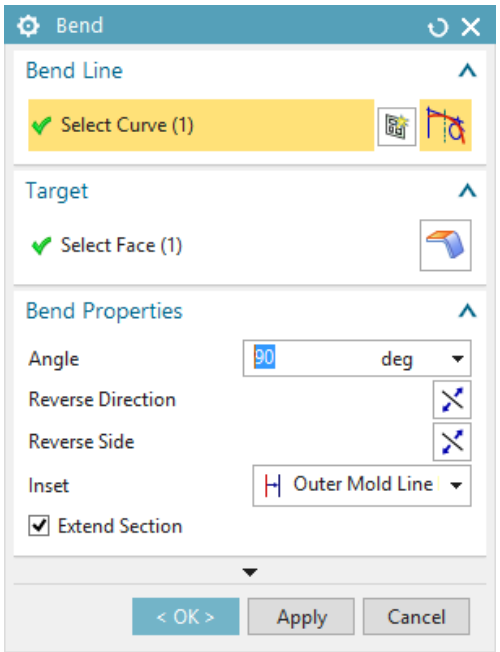
- Again, draw a sketch in the plane **P** consisting in two symmetric horizontal lines w.r.t the x-axis. The upper horizontal line is located at **10 mm** from the upper left corner of the metal sheet.
- Exit the sketch mode and click the *Bend* button, under the *More* button of the *Bend* field.





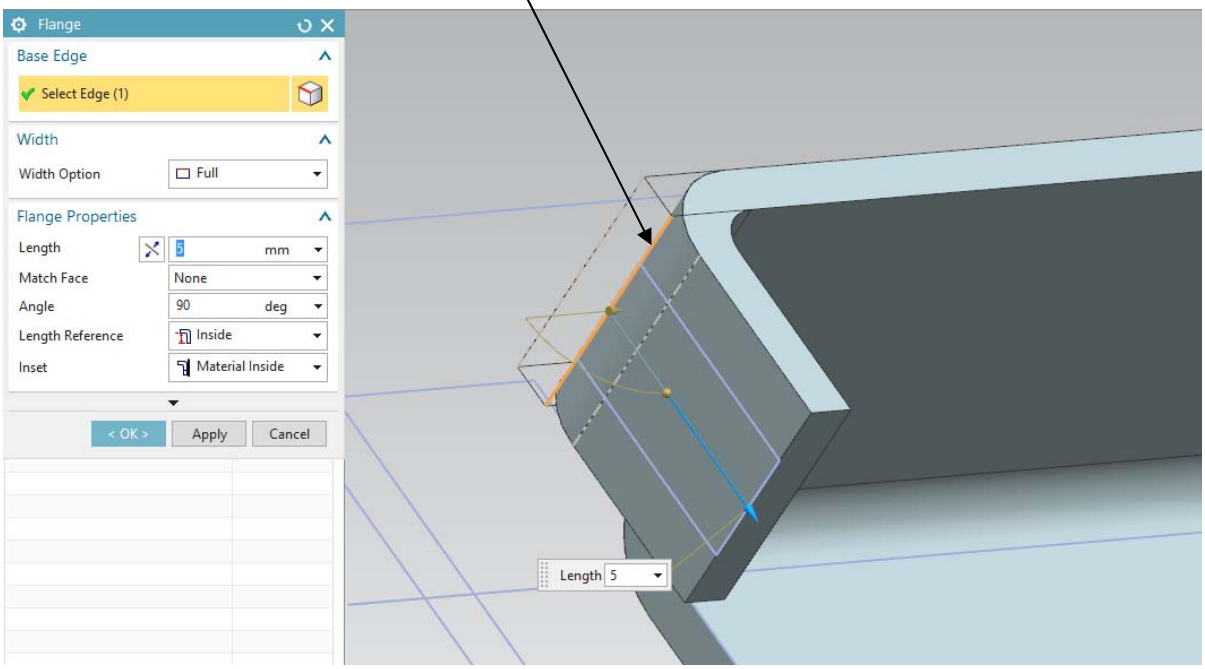
- In the *Bend* dialog box, select one line as B*end Line* and use an angle of **90 degrees**.
- Redo the same procedure for the other, symmetric, line.
- **Note**: if needed, use the *Reverse Side* option for obtaining the same result as below.



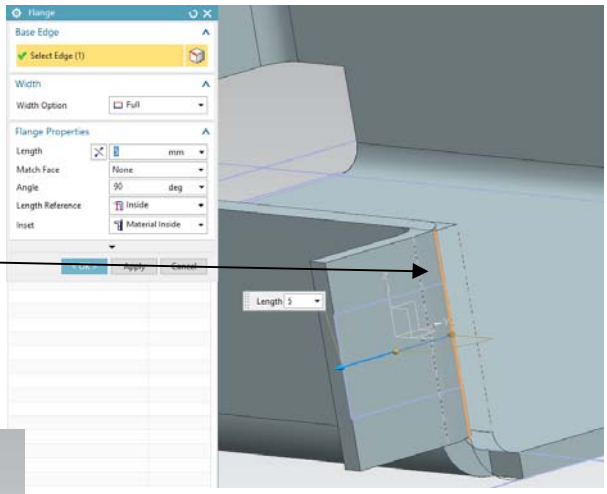


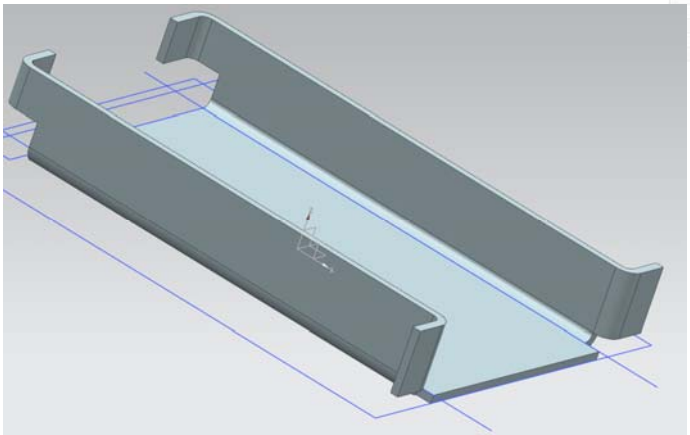
5 – Adding flanges.

- Click on the *Flange* button . Use a *Length* of **5 mm** and apply the flange on the shown edge of the metal sheet.



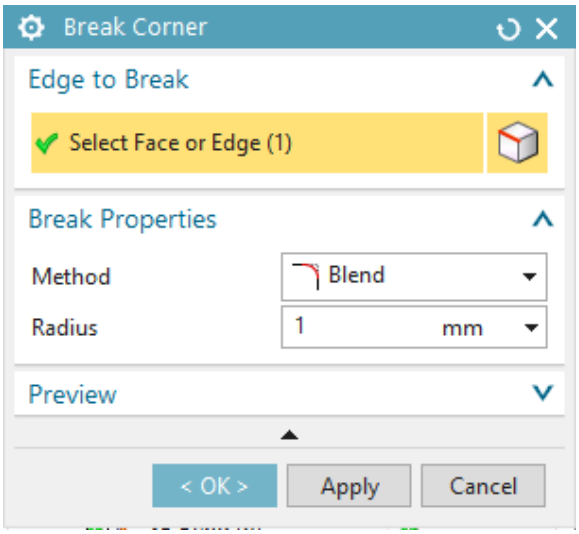
- Apply the same flange to the corresponding symmetric edge of the metal sheet.
- Also, apply the same flange on the below shown edge of the metal sheet, and its symmetric counterpart.
- The final result to obtain is shown here below.

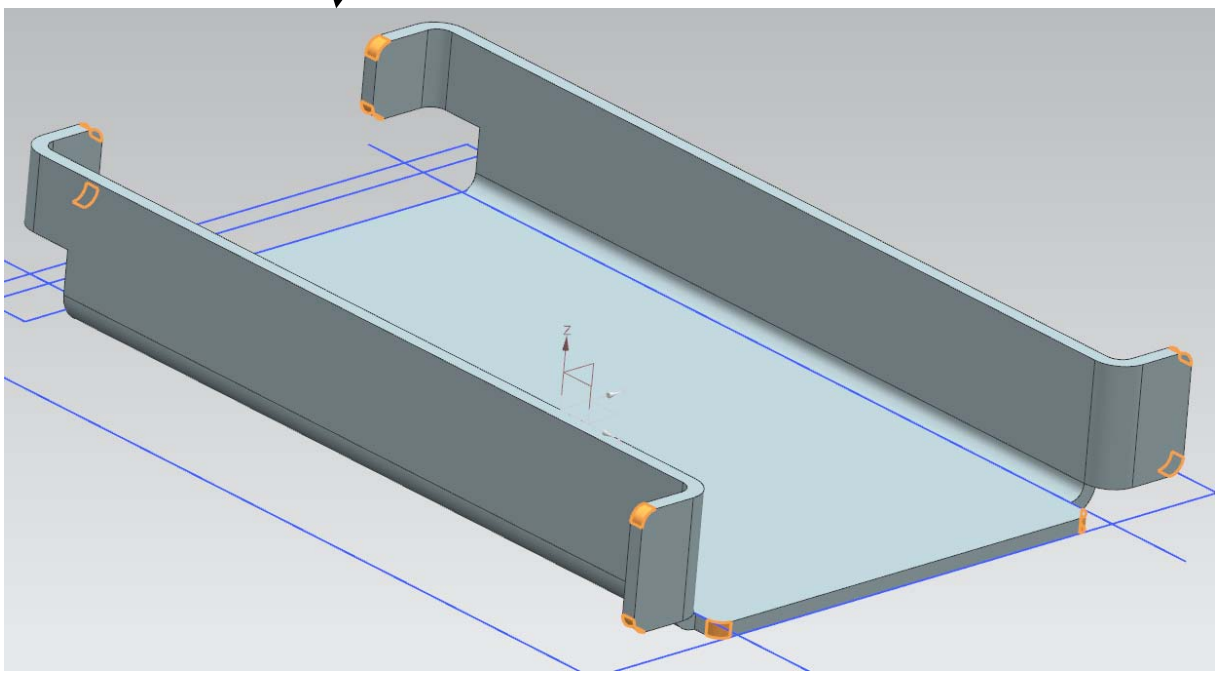




6 – Breaking corners.

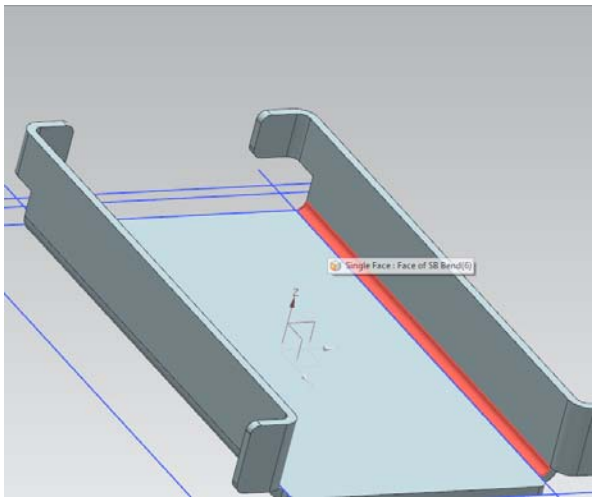
- Click on the *Break Corners* button Break Corner . In the *Break Corner* dialog box, use the *Blend* method with a *Radius* of **1 mm**.
- Apply the *Break Corner* to the edges shown in figure below.



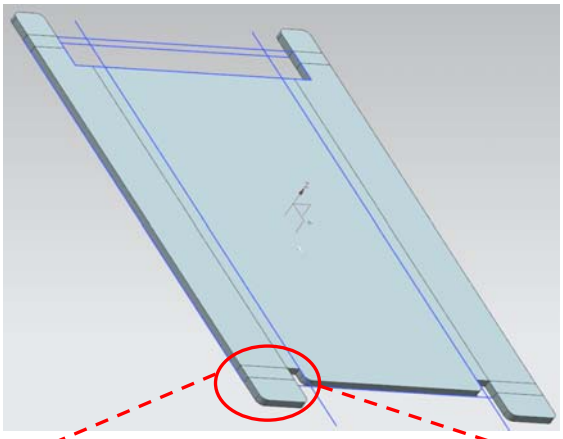


7 – Unbending the metal sheet.

- Click on the *Unbend* button Unbend .
- In the *Unbend* dialog box, select as *Stationary Face* the larger horizontal (inside the plane **P**) face of the metal sheet. Select as *Bend* face one of its adjacent face.



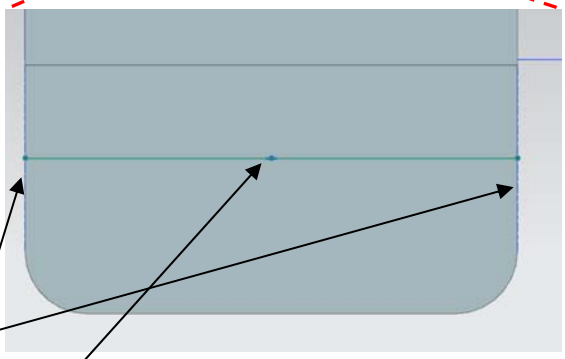
- Redo as many similar operations as necessary in order to obtain a flat metal sheet.
- **Warning**: keep in mind (or take some notes) the order in which you unbend the faces. You will need to re-bend these faces in the **reverse order** at the end of this tutorial.



8.a – Adding holes (sketch).

We will first sketch four points figuring the centers of the four wanted holes.

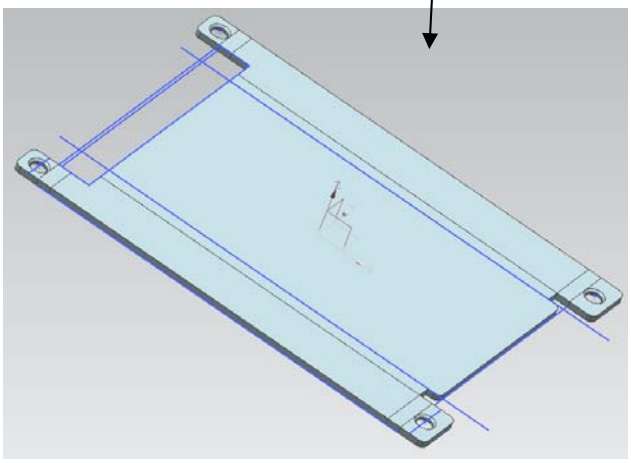
- Create a sketch in the **P** plane.
- In the lower left part of the metal sheet draw a line  which extremities are located on the middle of the shown segments.
- Draw a point  on the centre of this line.
- Finally, from that point create a symmetric point w.r.t the x-axis of the *Datum Coordinate System*.
- Similarly, create two other symmetric points (w.r.t. the x-axis of the *Datum Coordinate System*) in the upper part of the metal sheet.
- The first point will be located on the middle of a line. The extremities of this line will themselves be located on the middle of segments of the metal sheet.

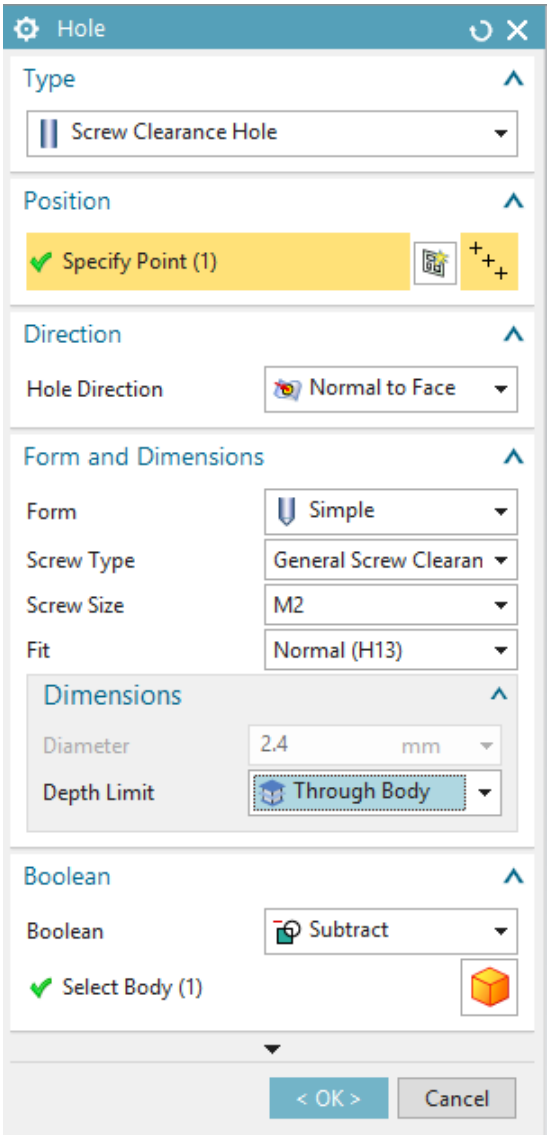


8.b – Adding holes.

- Click on the *Hole* button Hole under the *More* button of the *Feature* field.

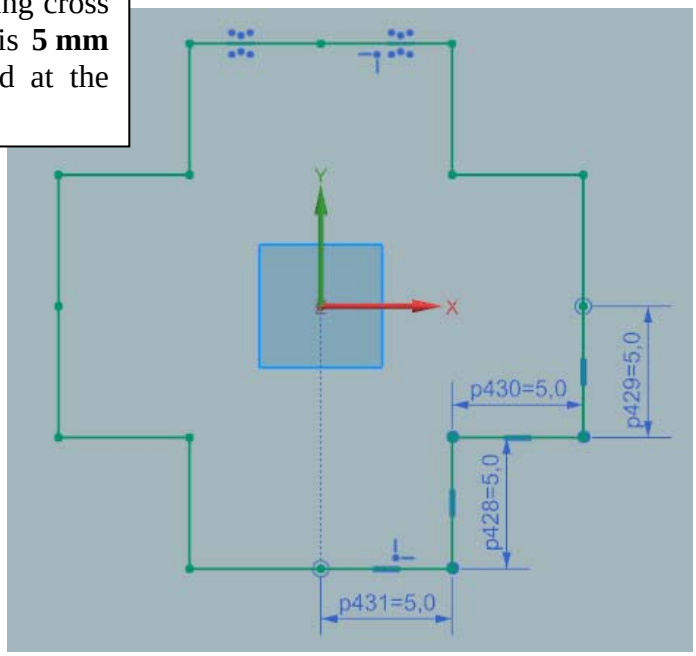
- In the H*ole* dialog box, use *Screw Clearance Hole* as *Type*, with a *M2 Screw Size* and a *H13 Fit*. Set the *Depth Limit* option to *Through Body*.
- Create four holes centred on the four previously defined points.



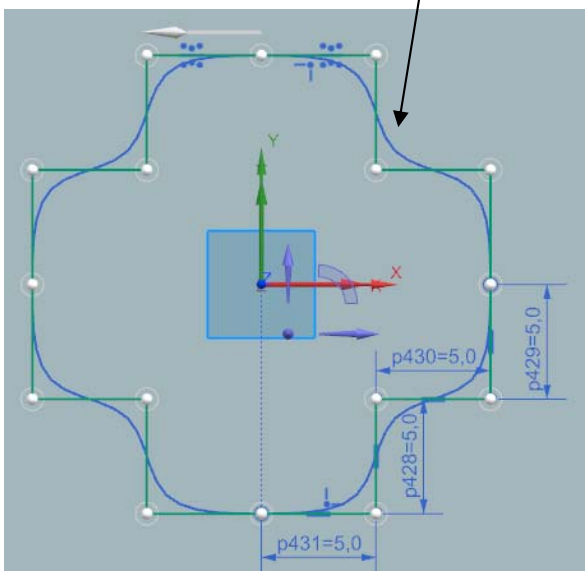


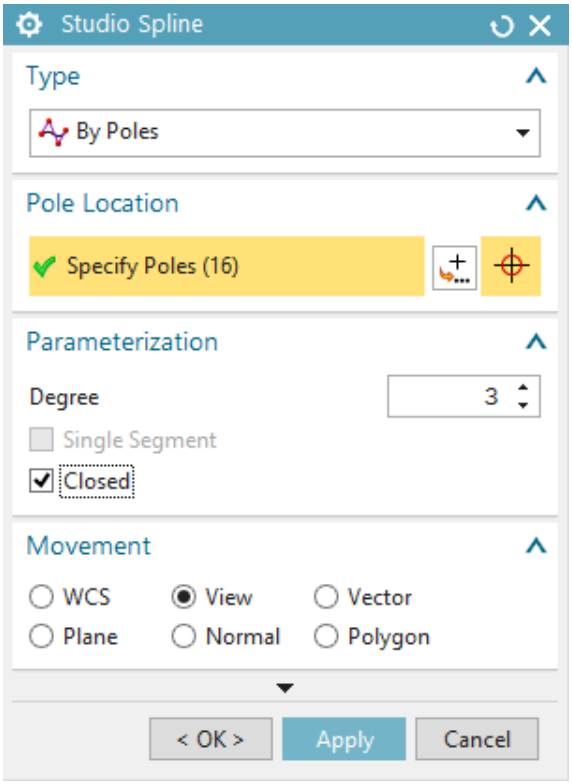
9.a – Adding a dimple (sketch).

- In the **P**-plane, create the following cross sketch, where each (half-) side is **5 mm** in length. The sketch is centred at the origin (0,0,0).

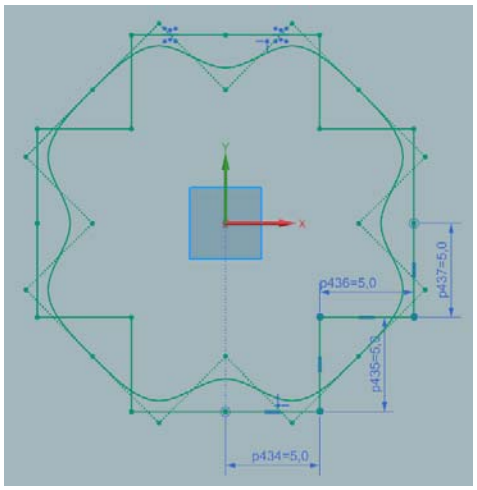


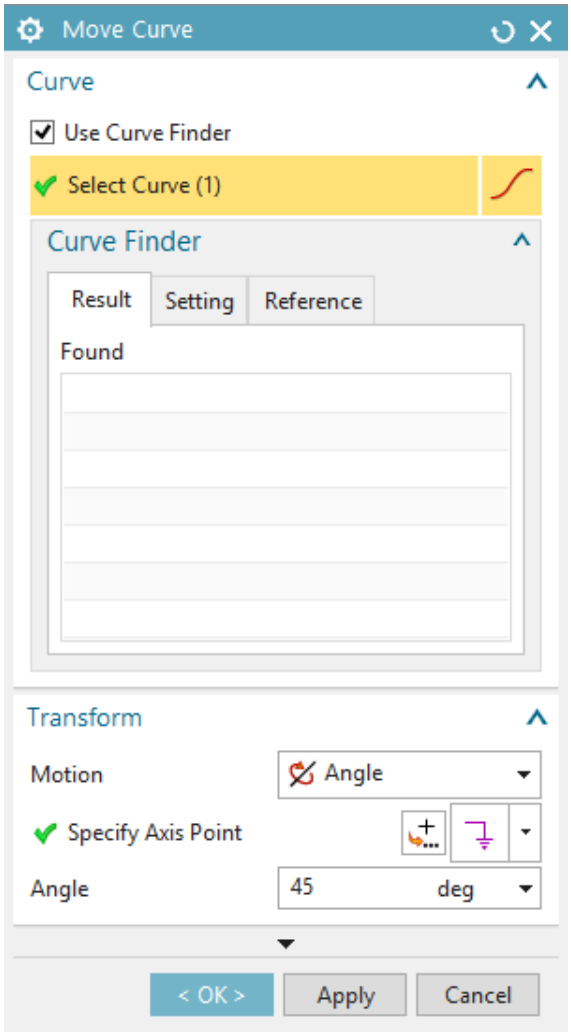
- Click on the *Studio Spline* button  .
- In the *Studio Spline* dialog box, set the *Type* field to *By Poles*. Check the *Closed* option.
- Select one by one all the 16 points of the sketch in a **clockwise (or anti-clockwise) fashion**.
- Click *OK* to validate the creation of the new spline curve.



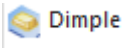


- Click on the *Move Curve* button  .
- In the *Move Curve* dialog box, select the spline curve you just drawn.
- Set the *Motion* option of the *Transform* field to *Angle*.
- Specify the origin (0, 0, 0) as axis point.
- Finally, enter an angle of **45 degrees** and click *OK* to validate.

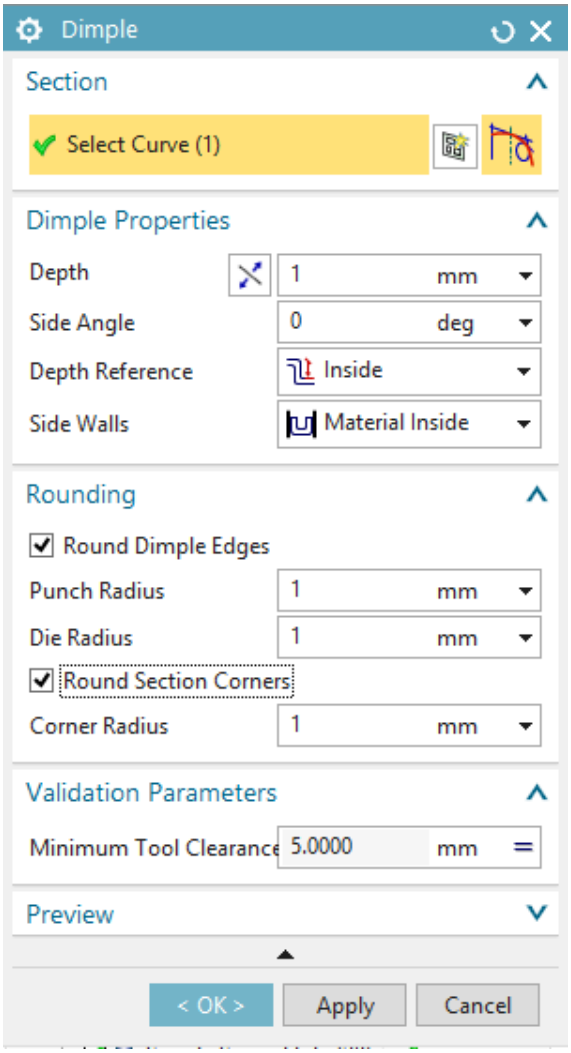




9.b – Adding a dimple.

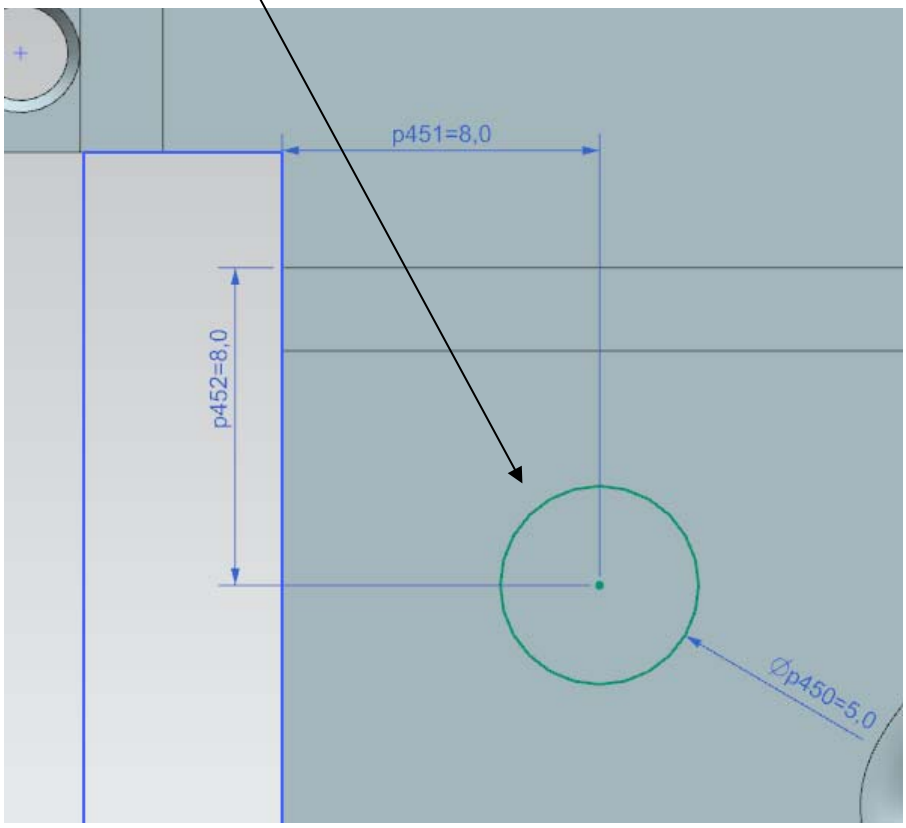
- Click on the *Dimple* button 

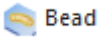
and select the spline curve.
- Set the *Depth* option to **1 mm** and *Side Walls* to *Material Inside*.
- Expand the dialog box (little black triangle) and expand the *Rounding* field by left-clicking on it.
- Set all the parameters of the *Rounding* field to **1 mm**.
- Click *OK* to validate your dimple.



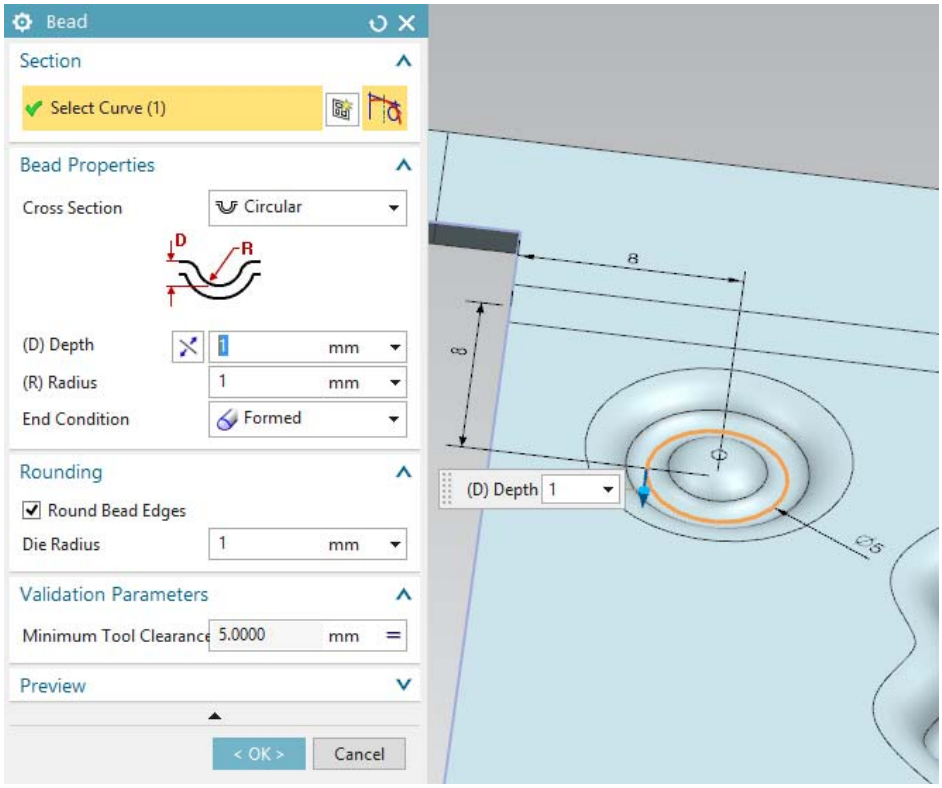
10 – Adding beads.

- In the **P**-plane, draw a circle of **5 mm** in diameter as shown.

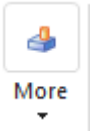


- Click on the *Bead* button 

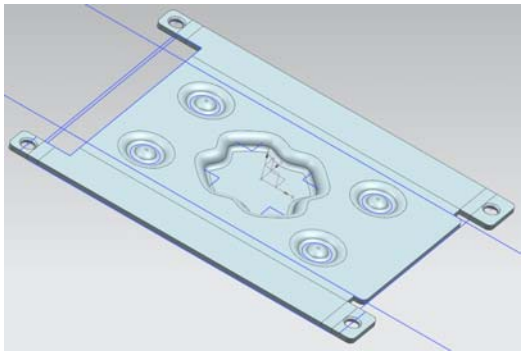
.
- In the *Bead* dialog box, select the circle you just drawn.
- Set the *Depth* and *Radius* options to **1 mm**, and the *Cross Section* to *Circular*.
- Expand the dialog box (small black triangle) and expand the *Rounding* field.
- In the *Rounding* field, set the *Die Radius* to **1 mm**.



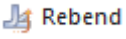
- Click on the *Mirror Feature* button

, under the *More* button 

of the *Feature* field.
- Select the bead and its corresponding sketch as *Feature to Mirror*.
- Select the YZ plane as the mirror plane and click *Apply* to validate.
- Then, select the two beads (and their corresponding sketches) and redo a feature mirror by choosing this time the XZ-plane.



11 – Re-bending.

- Click on the *Rebend* button 

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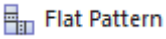
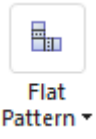
- Re-bend the faces by selecting them in the **reverse order in which they were bend**.
- Hide the sketches. You should obtain the same result as the one shown at the beginning of this tutorial.

12 – Drafting.

- Click on the *Flat Pattern* button 

, under *Flat Pattern* menu 

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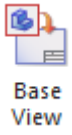
- Select the central flat face of the metal sheet and validate. This will create a flattened version of our metal sheet.
- If an information dialog box opens, just click *OK*.
- For the moment, the flattened version is not visible. To make it visible, go in the *Part Navigator* and expand the *Model View* tree.
- Double-click on the object named *FLAT-PATTERN#1*.

- Create a new drafting file of size A4, and add a new *Base View* 

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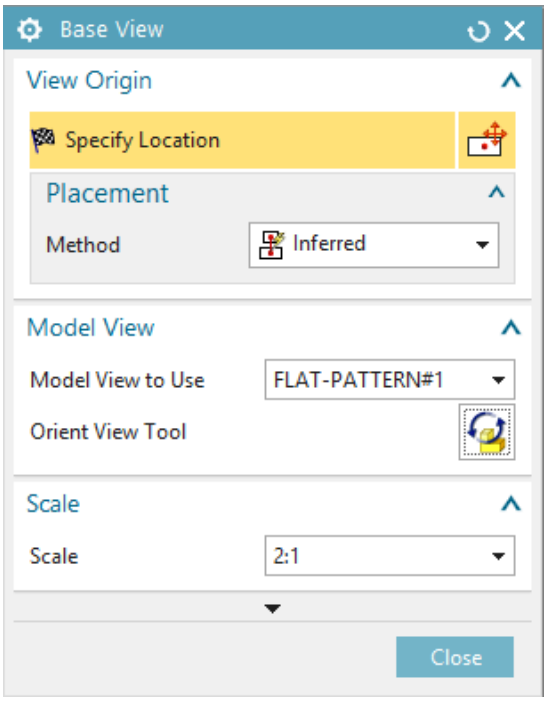
- In the *Base View* dialog box, select in the option *Model View To Use FLAT-PATTERN#1*.
- Set the *Scale* to *2:1*.



- The draft will probably be too big for fitting inside the available space of the form.
- Rotate it using the *Orient View Tool* button  in order to make it fit.
- Add the rotated draft to its form.
- Delete the text that is automatically added with the draft.
- **Do not forget to save your draft file.**