Using Siemens NX 11 Software

Sheet Metal Design - Casing

*Based on a YouTube NX tutorial*

[1]https://www.youtube.com/watch?v=-SIYiIVz87k
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.

  ![Bend dialog box](image)

- In the *Bend* dialog box, select one line as *Bend 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. 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. 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 under the *More* button of the *Feature* field.
- In the *Hole* 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**.

![Image of Bead dialog box and 3D model with beads]({image_url})

• 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 Rebind button.
- 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.
- 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.
- 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.**