

EXERCISE 2.1 – CREATING A FACE USING SHEET METAL

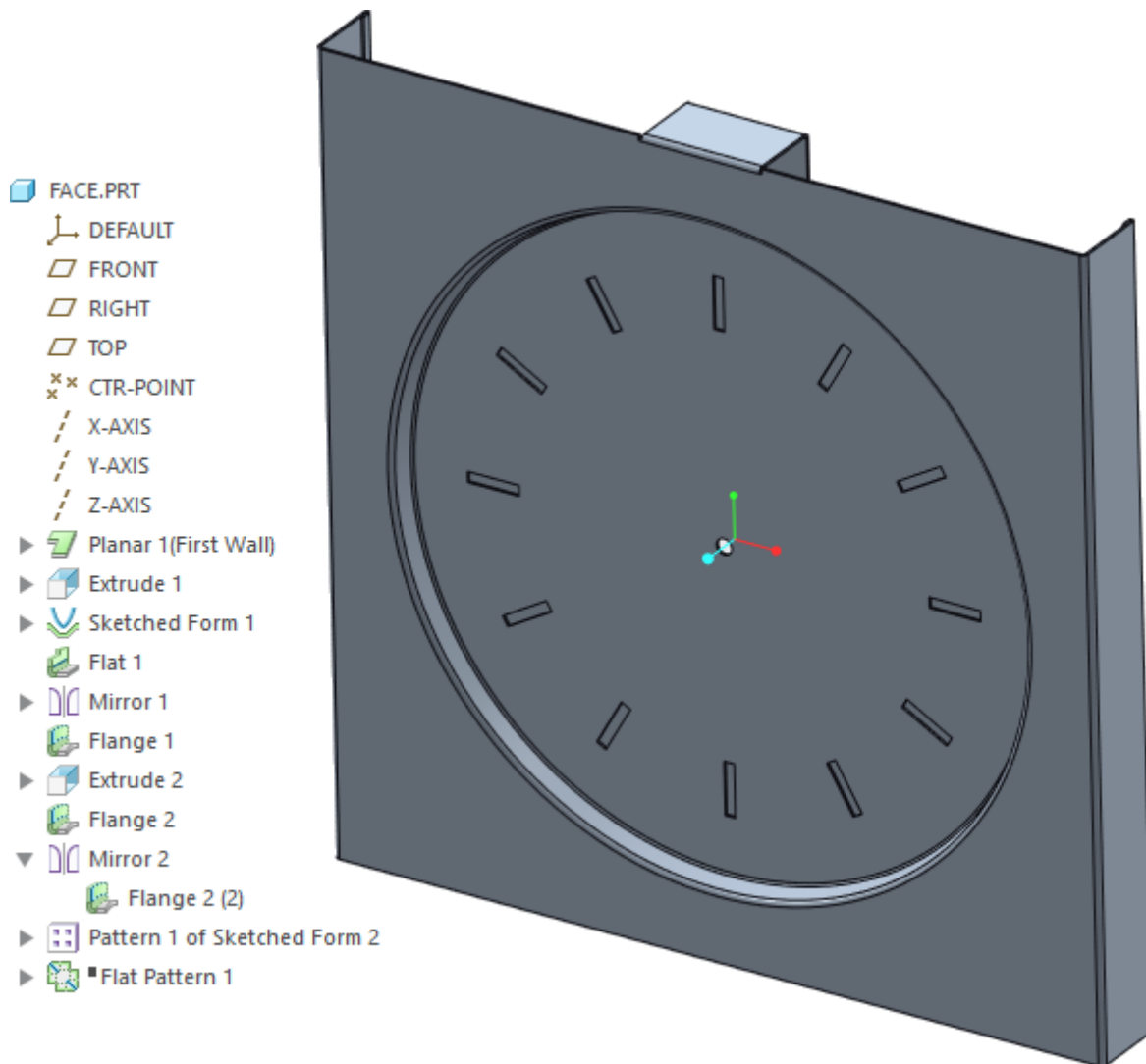


Figure 1: Clock face and its model tree.

Learning Targets

In this exercise you will learn

- ✓ To use sheet metal mode
- ✓ to use Flats and Flanges
- ✓ to use Form tool
- ✓ to use Flat Pattern
- ✓ to pattern features.

In this exercise, we create a clock face using sheet metal mode in Creo. Program version is Creo Parametric 6.0.2.0.

Starting

Start a new *Part* using *Sheetmetal* as a *Sub-Type* (Figure 2). Name it as *face*.

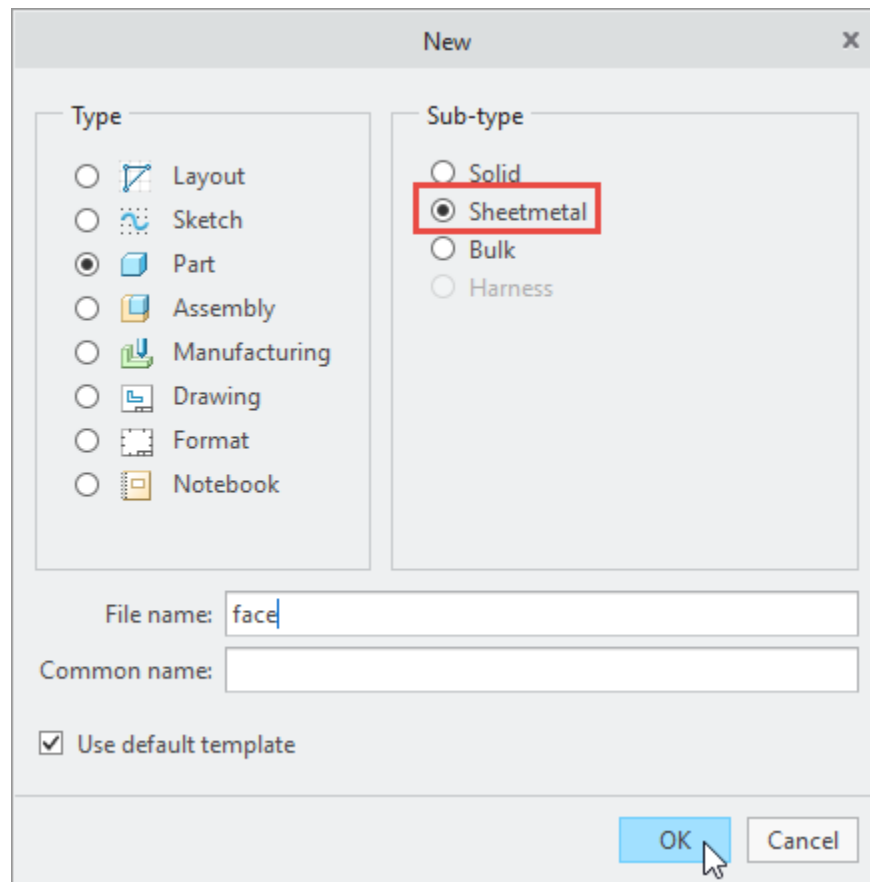


Figure 2: New dialog.

The First Wall

First, we need to define a first wall, a feature that defines the thickness of a sheet metal plate.

Select **Planar** (📐, *Shapes* group) and select FRONT plane. Now we need to sketch a profile. Create sketch as seen in Figure 3. When ready, accept the sketch.

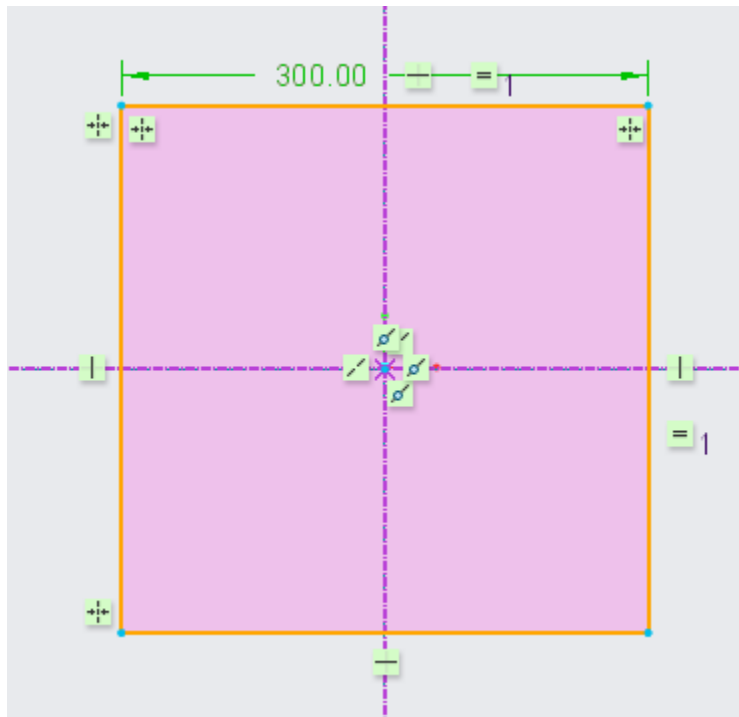


Figure 3: Sketch for Planar feature. Ensure, that you have only one dimension (300).

Give 1 as thickness (top left corner in the dashboard) and accept feature (**MMB**).

The Middle Cut

Next, we cut a hole in the middle. Select **Extrude** (📐, *Shapes* group) and select FRONT as a sketching plane. Make a circular sketch (diameter of 6) and cut it through (default in the sheet metal mode). Be sure that the cut directory is right (otherwise the cut will fail; if happens, just flip the direction arrow).

Using Sketched Form

We punch a form using a sketched shape. Select **Sketched Form** (📐) from *Engineering* group (Figure 4).

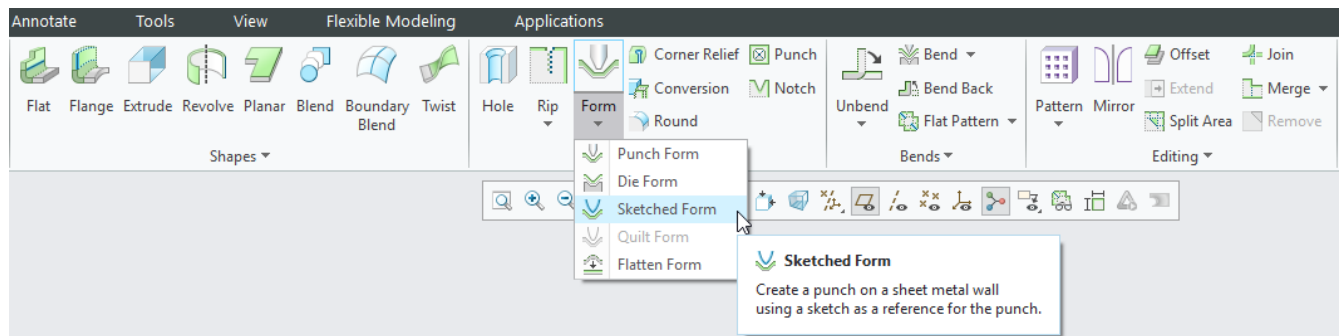


Figure 4: Selecting *Sketched Form*.

Select the front surface of the part as the sketching plane and create a sketch as seen in Figure 5. Accept sketch when ready.

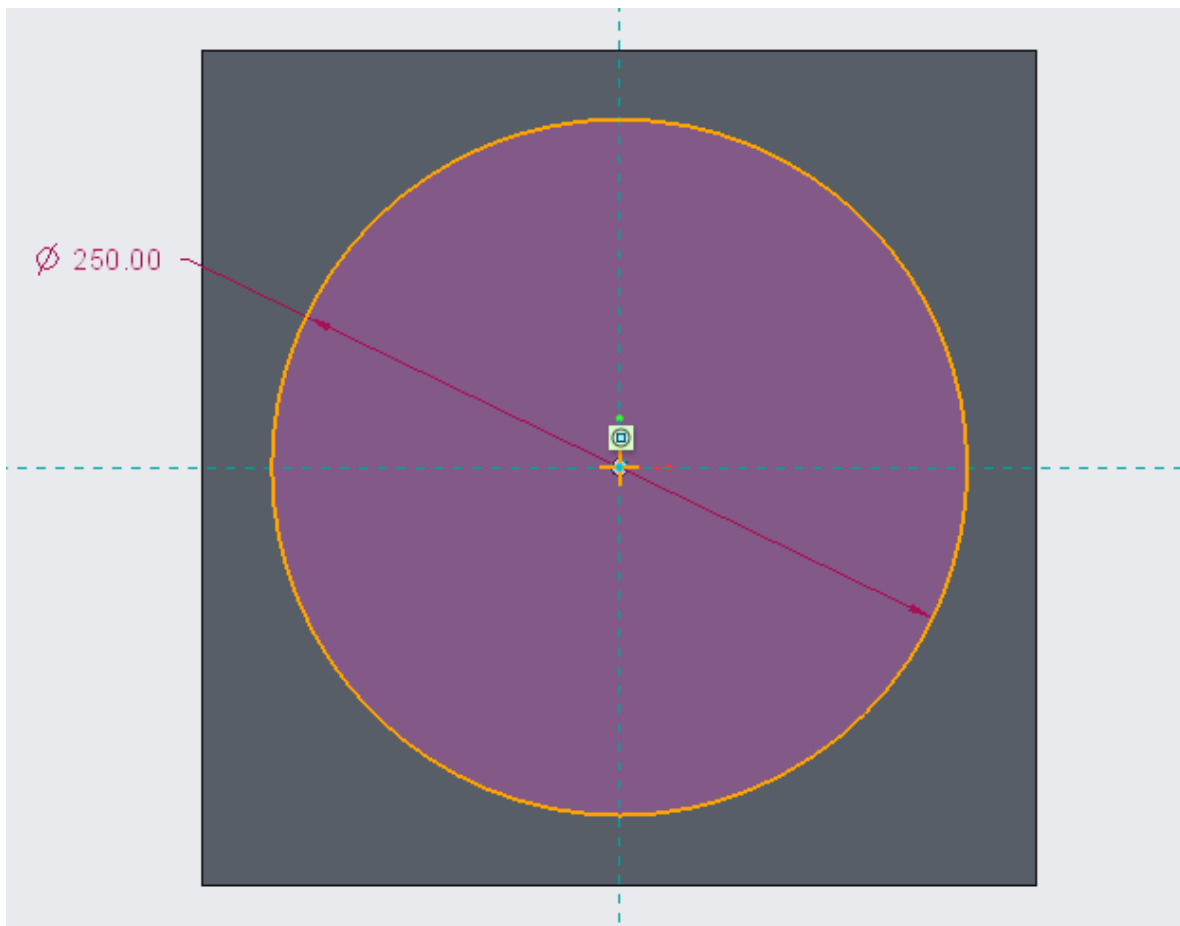


Figure 5: Sketched shape for *Sketched Form*.

Change punching direction (away from the screen) by clicking the red arrow (note the one next to sketch). Give **10** as deep. Open **Options** tab and check **Add taper** and give **15°** as angle. Select also both rounds using default values (Figure 6). When ready, accept the feature.

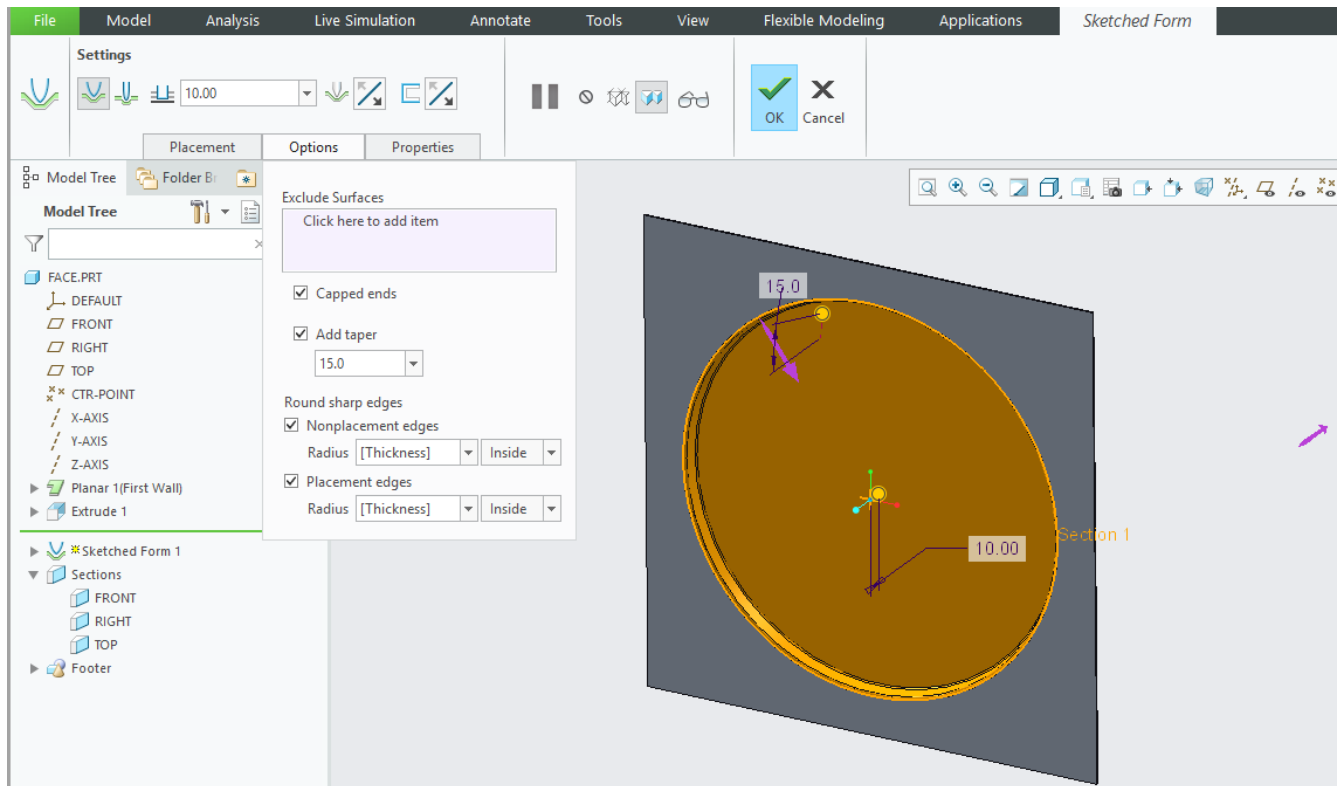


Figure 6: Ready-to-accept *Sketched Form*. Notice the arrow pointing away in the right (punching direction) and arrow pointing inside sketched circle (punched area).

Using Flat

Next, we add bends to our model. Select **Flat** (📐, *Shapes* group) and select right top edge of the Planar feature (see Figure 7). You can press **CTRL+D** to see the default view (all planes positive).

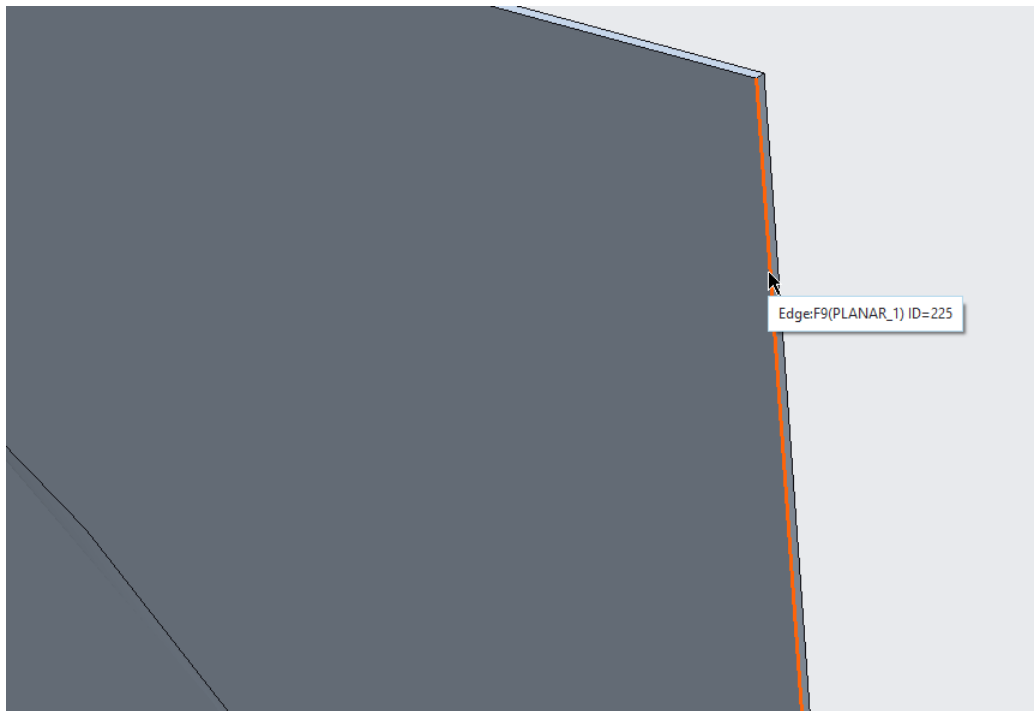


Figure 7: Selecting the Planar feature's top right edge.

Change the length of the added flat to **30** and check that your model looks like in Figure 8.
Accept the feature.

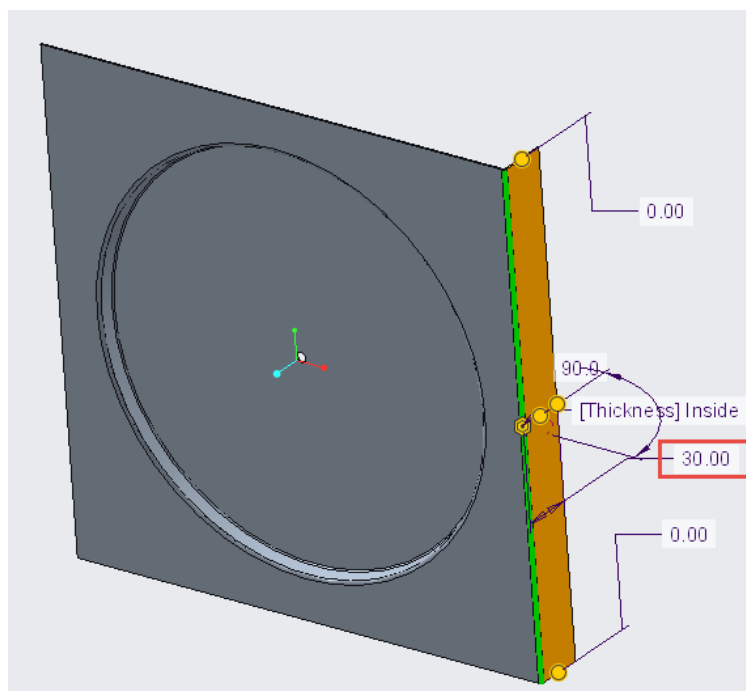


Figure 8: Ready-to-accept Flat. Changed dimension highlighted.

Mirror

Select previously created Flat and select **Mirror** (🪞) from *Editing* group (Figure 9). Select RIGHT as mirroring plane and accept the feature.

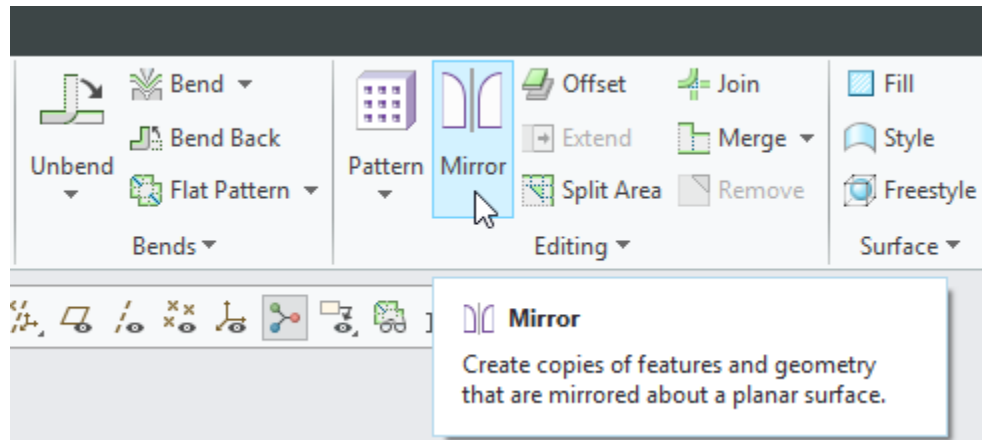


Figure 9: Selecting Mirror tool.

Hanger Using Flange

Next, we create a hanger. The *Flange* tool creates new material, but unlike *Flat*, you define added material by sketching a curve (in *Flat* you sketch an area).

Select **Flange** (🔧) from *Shapes* group. Select topmost edge of the part as attachment edge (see Figure 10).

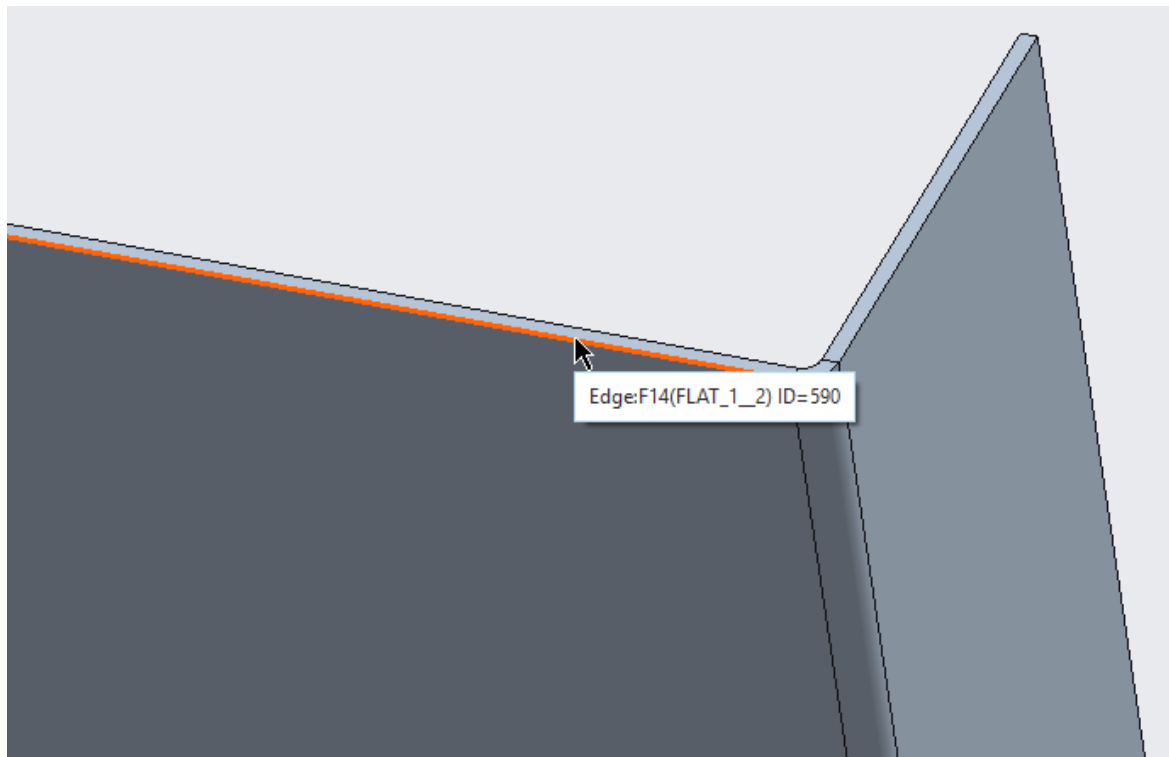


Figure 10: Selecting edge for *Flange*.

Select **User Defined** from dashboard as a wall profile type (Figure 11). Then select **Shape** tab and select **Sketch...** *Sketch* window opens, select **Sketch**.

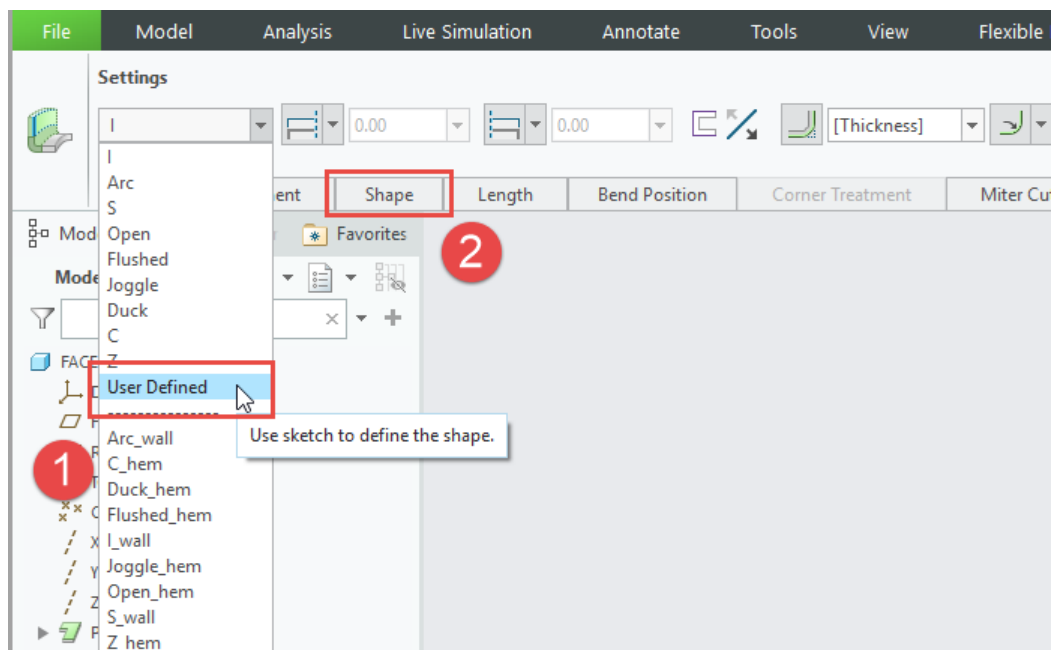


Figure 11: Selecting *User Defined* as a wall profile.

Open **References** (hold **RMB**, select **References...** from the list) and select entity from the *Flat* feature as a reference (see Figure 12).

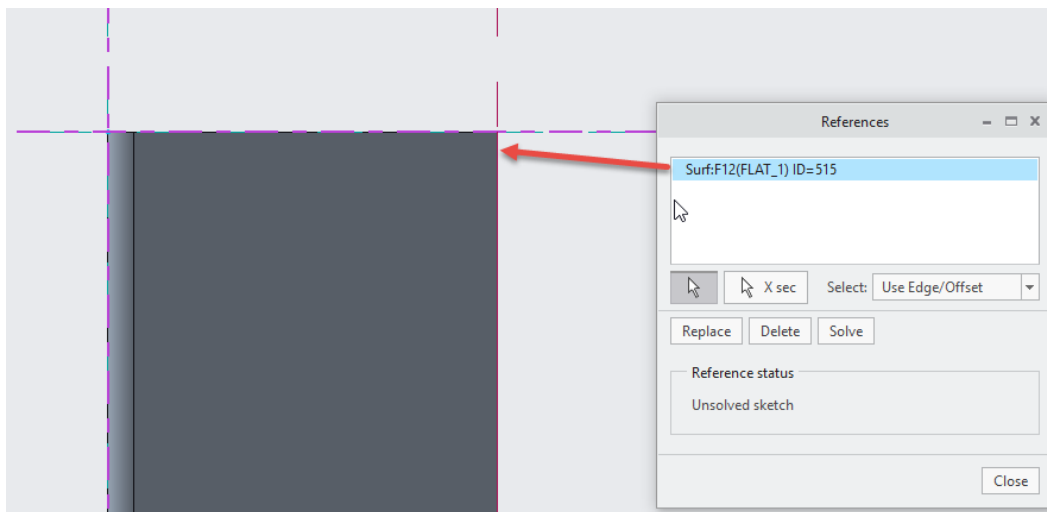


Figure 12: Reference selected.

Then using previously created reference, create an open sketch containing two lines as seen in Figure 13. When ready, accept the sketch.

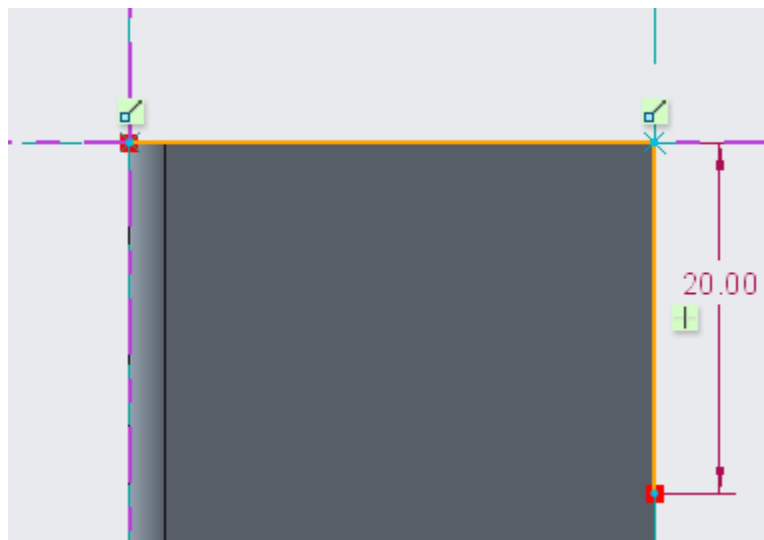


Figure 13: Two lines sketched.

Select **Length** tab to control the length of the bend. From drop down menus, select **Blind** and give value of **-125** to both fields (Figure 14).

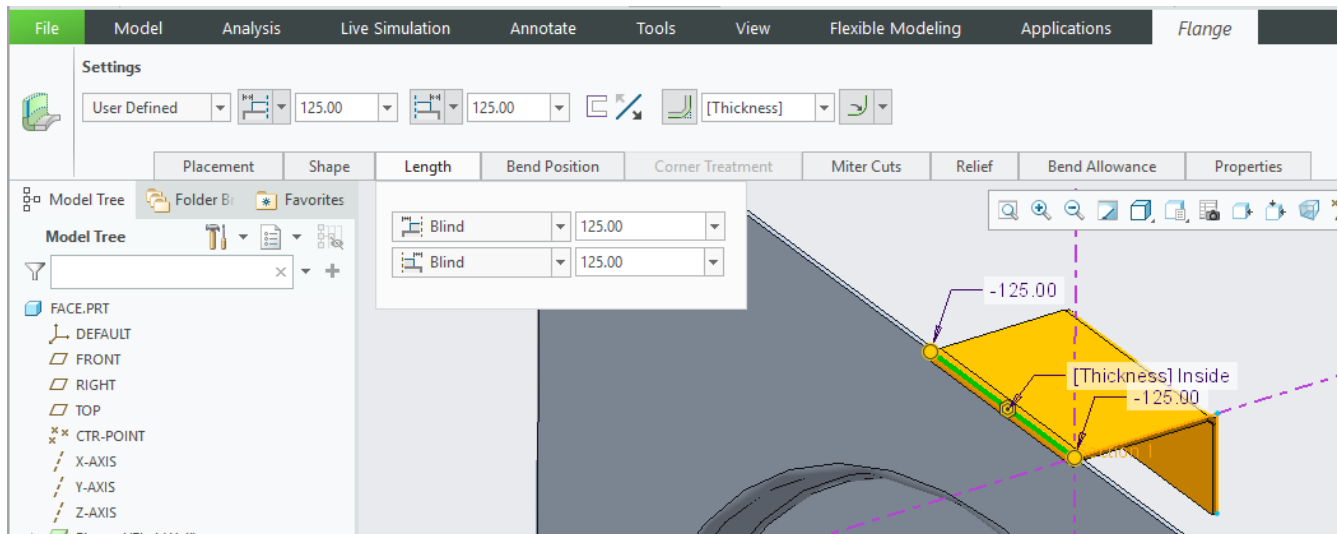


Figure 14: Changing values in Length tab.

Next, we add some cuts to help bending. Select **Relief** tab and change relief *Type* to **Obround**. Default values (*Up to Bend* and *Thickness*) should be fine (see Figure 15). When ready, accept the feature.

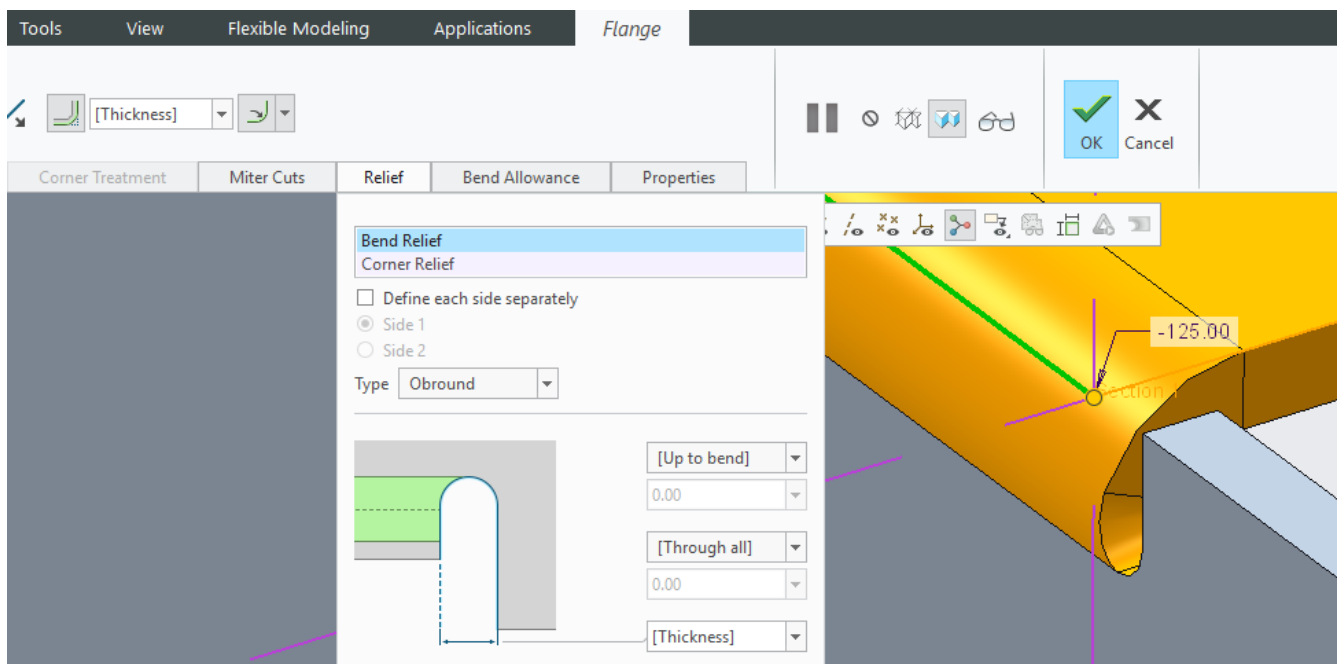



Figure 15: Added Reliefs.

Hanger Cut

Our hanger needs some cut to help placing the clock face on the wall. Select **Extrude** (, *Shapes* group) and start a sketch on the back surface of the Flange (see Figure 16).

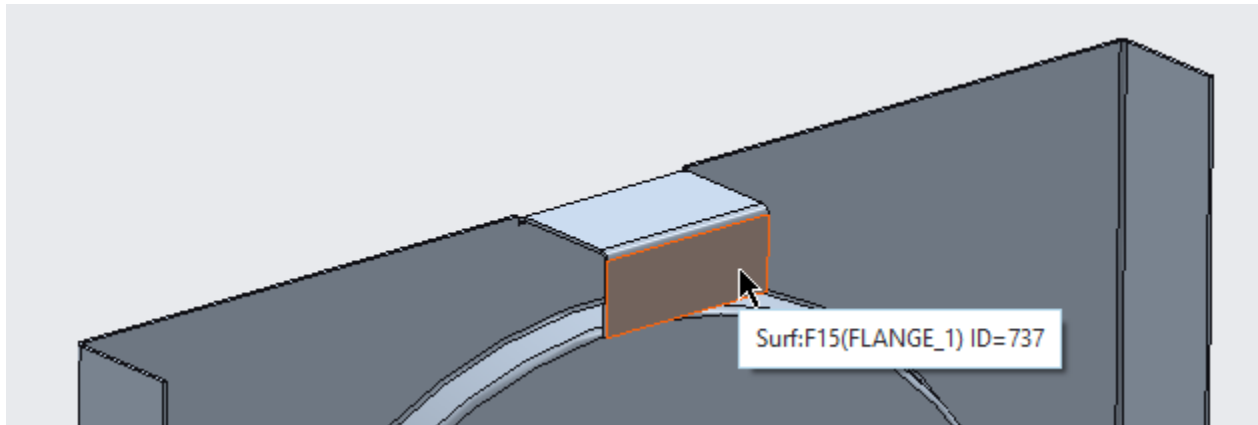


Figure 16: Location for the sketch.

Using suitable references and entities, create a sketch as seen in Figure 17. When ready, accept the sketch and make a cut (and accept the feature).

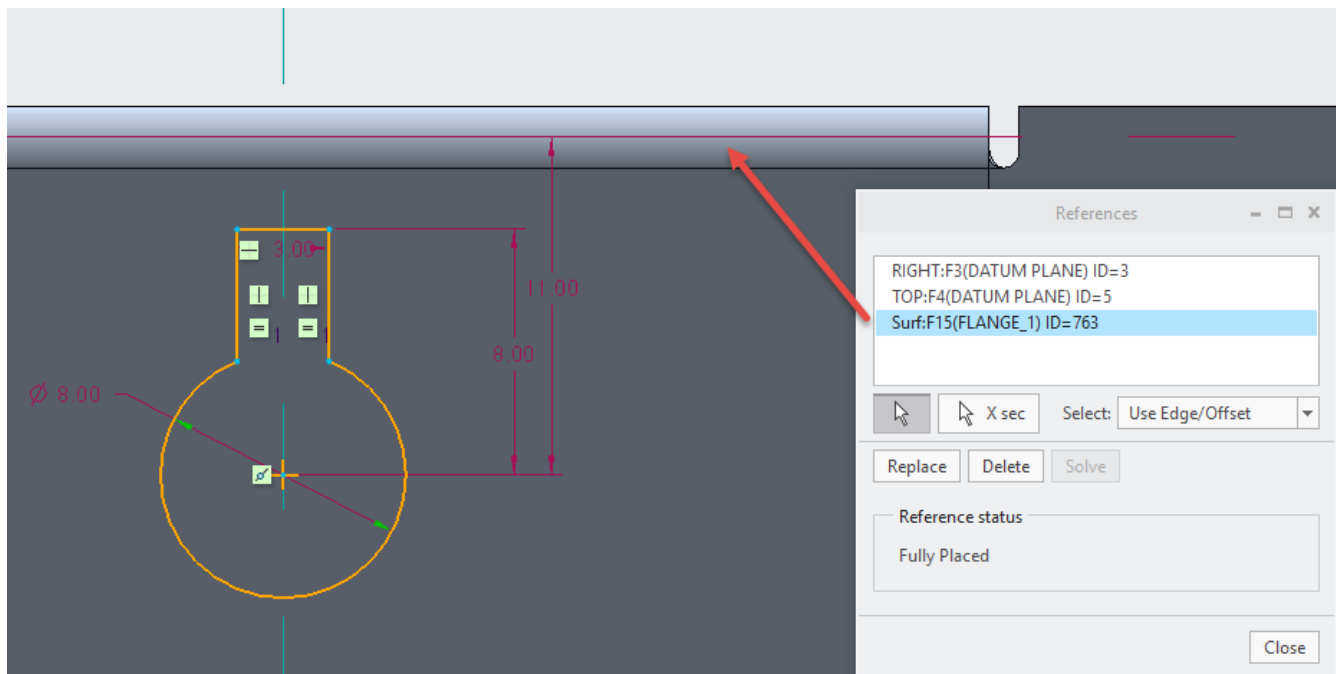


Figure 17: Sketch for hanger cut.

Flanges

Select **Flange** (🏠, *Shapes* group) and select an end edge of the first *Flat* feature as a bending edge (see Figure 18).

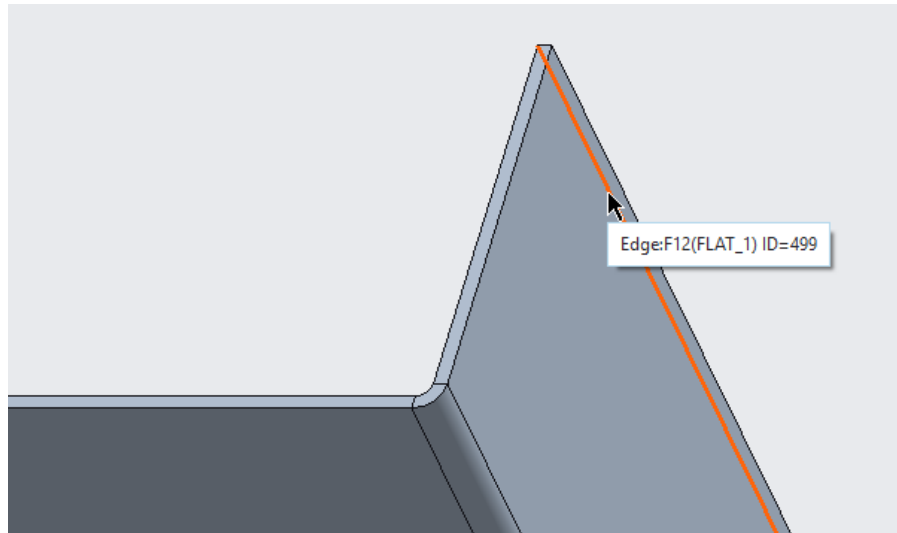


Figure 18: Selecting edge. View orientation is *Default* (CTRL+D).

Change shape to **Flushed** (leftmost drop-down menu in the dashboard) (Figure 19). When ready, accept *Flange* feature.

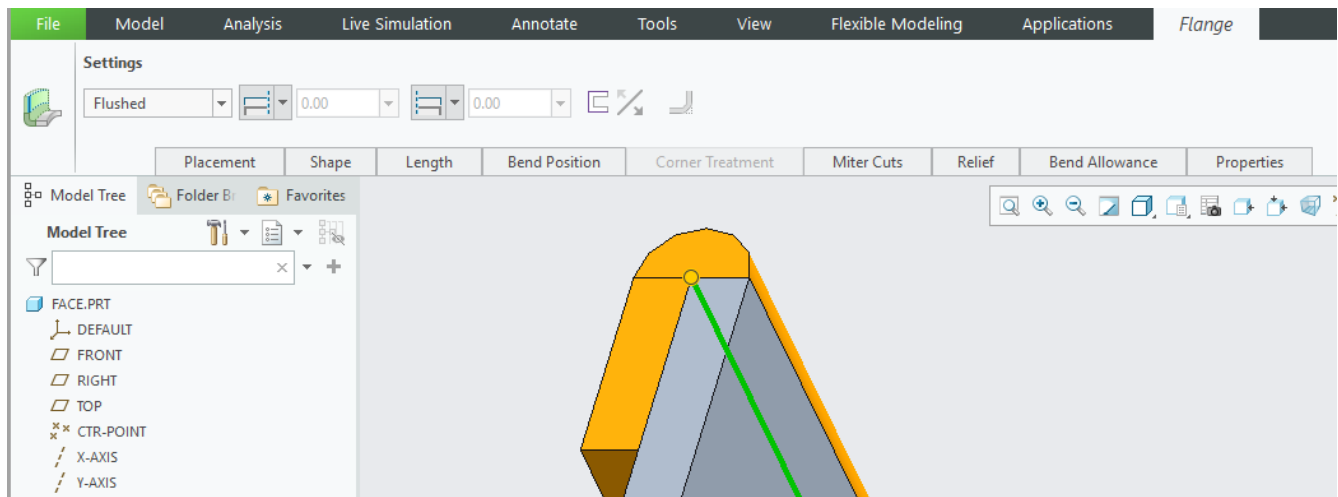


Figure 19: Offset type changed.

Mirror

Using **Mirror** (🔍, *Editing* tab), mirror previously created *Flange* using RIGHT as a mirroring plane.

Punched Hour Symbol

Clock face usually has some indicators for time related items. Next, we punch one hour symbol and then we pattern it.

Select **Sketched Form** (📐, *Engineering* group) and select previously punched area as a sketching plane (Figure 20).

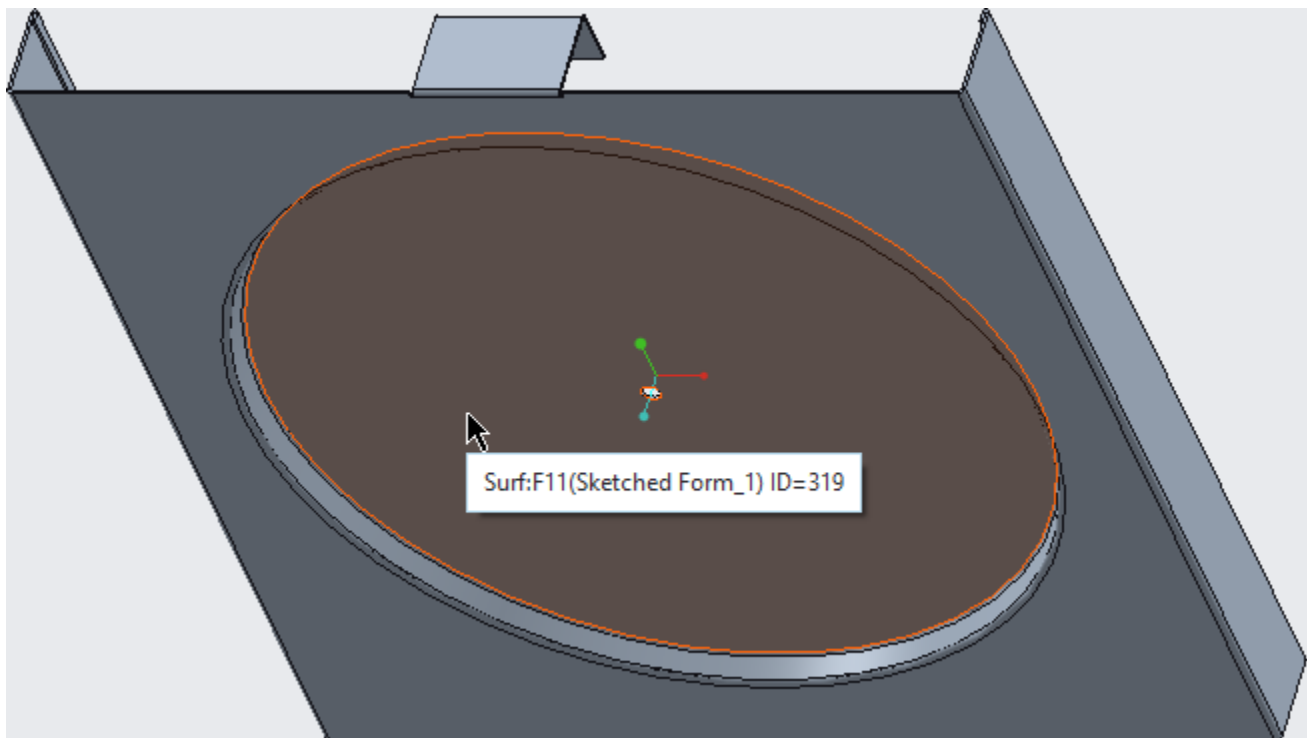


Figure 20: Selecting sketching plane.

Create a Circle (🕒, *Sketching* group), select that circle (highlighted in green), hold **RMB** and select **Construction** (Figure 21). This created a construction entity, i.e. entity that is not punched.

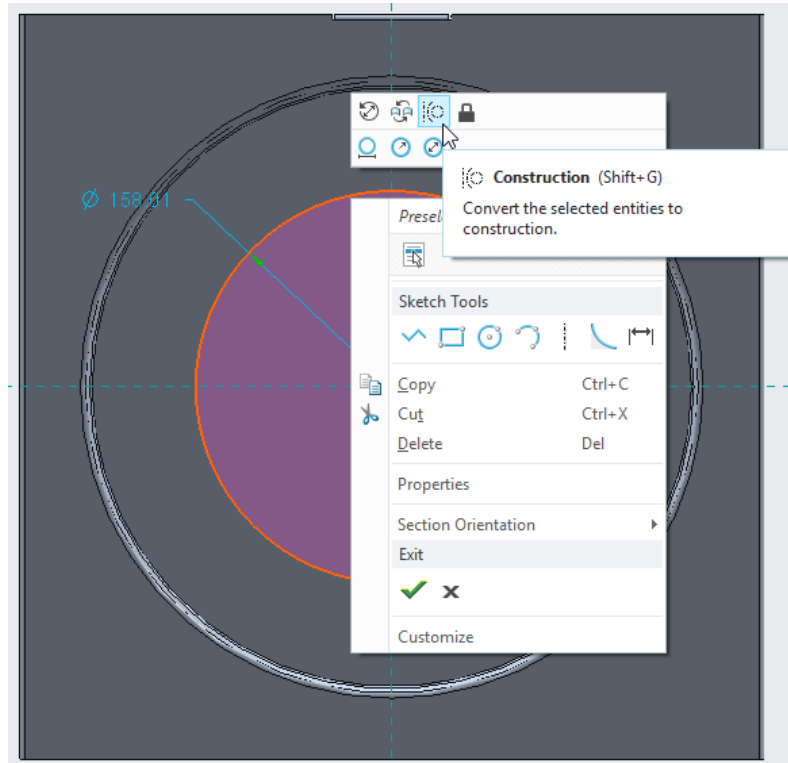


Figure 21: Circle selected and selecting Construction from RMB menu. Ignore dimension value.

Using **Center Rectangle** (, *Sketching* group), finish the sketch as seen in Figure 22.



Figure 22: Created sketch. Notice the construction circle.

When ready, accept the sketch. Change punching direction to pointing away and give **1** as depth (Figure 23). Accept the feature.

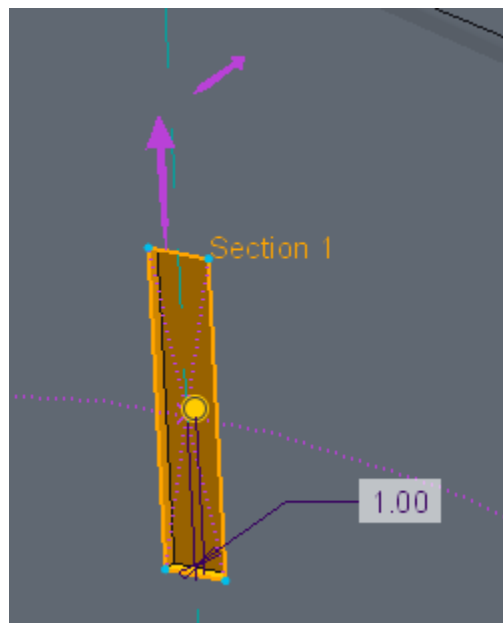



Figure 23: Punching direction changed and depth defined.

Pattern

Select previously created Sketched Form, hold **RMB** and select **Pattern** (Figure 24). Change pattern type to **Axis** and select one of the two axes in the model (remember to turn **Axis Display** on). Click  to change dimension schema and finish dashboard as seen in Figure 25. Accept the pattern.

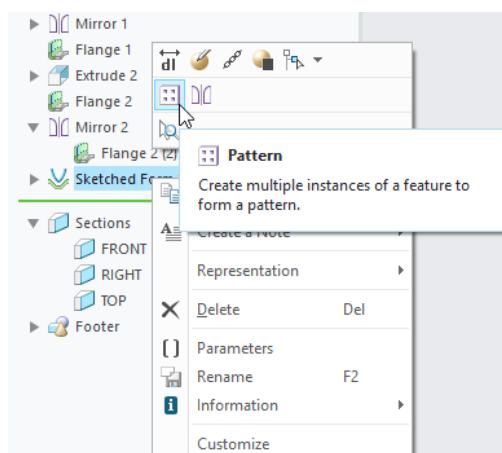


Figure 24: Selecting Pattern from RMB menu.

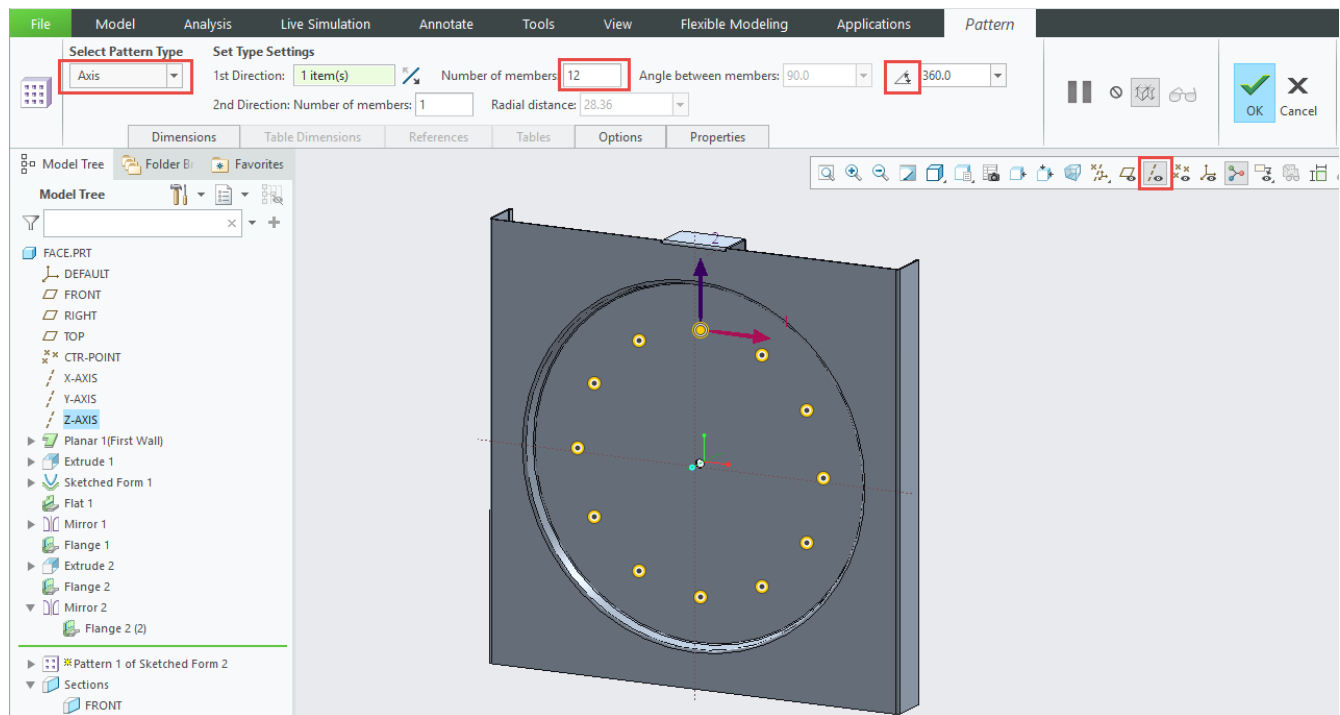


Figure 25: Ready-to-accept pattern feature. Center hole's axis selected as pattering axis.

Unbend

To see the needed sheet to manufacture this kind of clock face, select **Flat Pattern** (📐) from *Bends* group. Default values should be OK, accept the feature. Select *Flat Pattern* from model tree, hold **RMB** and select **Suppress**. (You can unsuppress it to see the flat state again)

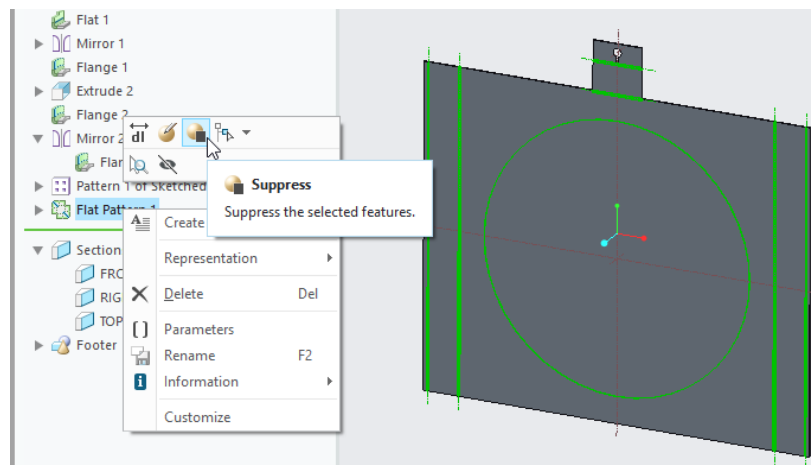


Figure 26: Selecting Suppress for Flat Pattern feature.

Your model should look like in Figure 1. Did you remember to save your model?

Parameters and Relations

Create three parameters that control the main shapes of the part. Parameters can be seen in Table 1. How those parameters affect the model can be seen in Figure 27.

Table 1: Parameters for the model.

Parameter Name	Type	Value
WIDTH	Real Number	300
DIAMETER	Real Number	250
HEIGHT	Real Number	30

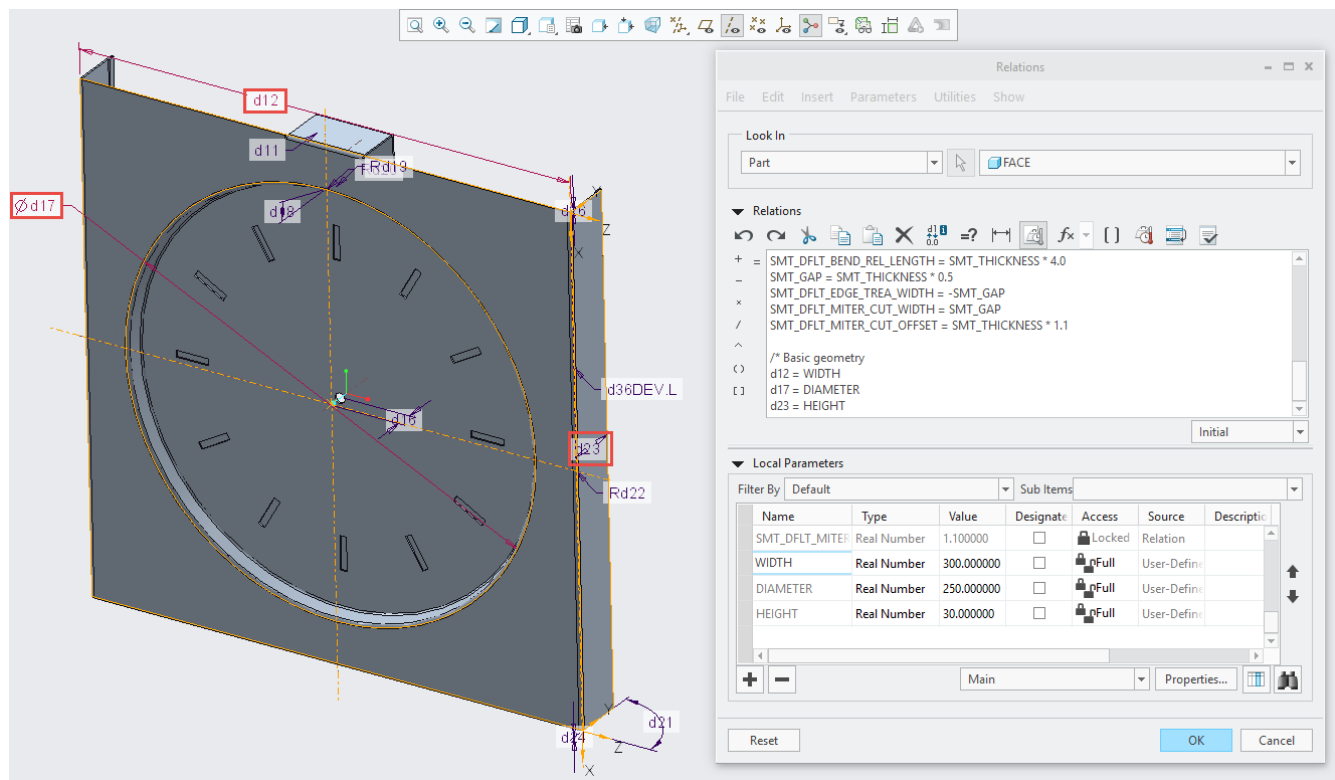


Figure 27: Relations and parameters. Four last lines in *Relations* created during this exercise.

This concludes this exercise.