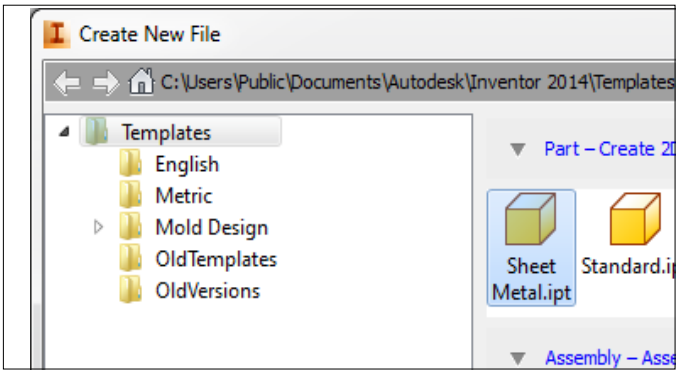
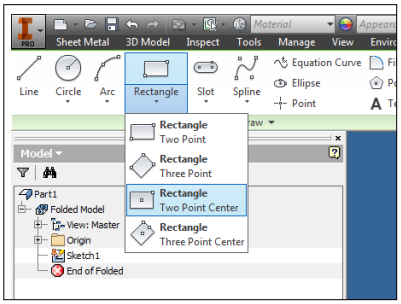
Sheet Metal Exercise. Conical Development

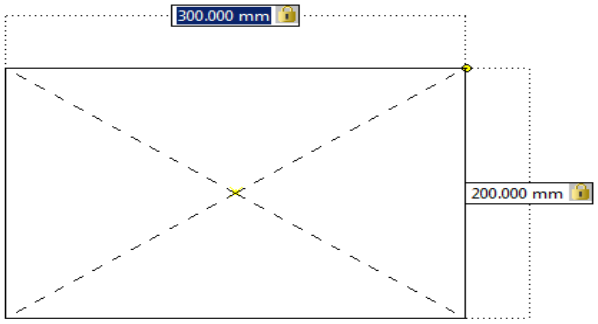
Circular to square sheet metal development



Step # 1. From the Create New File dialog box , select the *sheet metal.ipt* template.



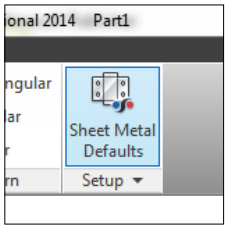
Step # 2. On the Sketch tab under the Draw panel select *Rectangle 2 point centre.*

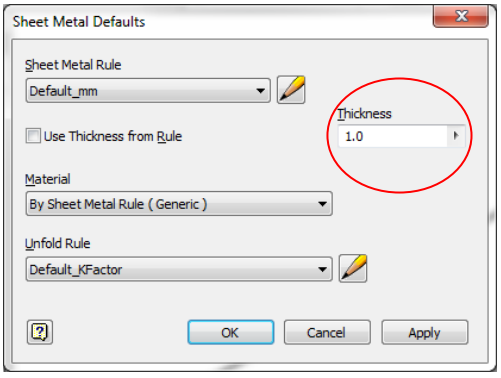


Step # 3. Place the cursor at the origin and then drag out the shape of the rectangle. Enter 300 mm for the length and 200 mm for the height. (Use the *TAB key* to toggle between the input boxes)

Finish the sketch.

Step # 4. On the Sketch Metal tab select the *Sheet Metal Defaults Tool*

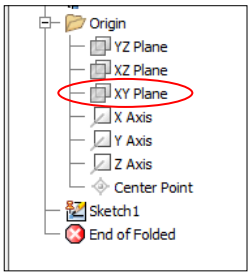


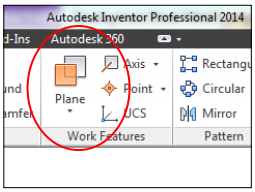


Step # 5. On the Sheet Metal defaults dialog box de-select the *Use Thickness from Rule checkbox* and set the thickness to 1.0

Click OK

Step # 6. Expand the Origin Increment in the browser.





Step # 7. On the Sheet Metal tab select the *Plane command.*

Left click on the XY Plane increment expanded in the previous step.

Left Click and hold on the highlighted sketch 1 as shown in Fig 1.0

While holding the left mouse button, drag the Workplane upwards.

Enter a value of 300 mm.

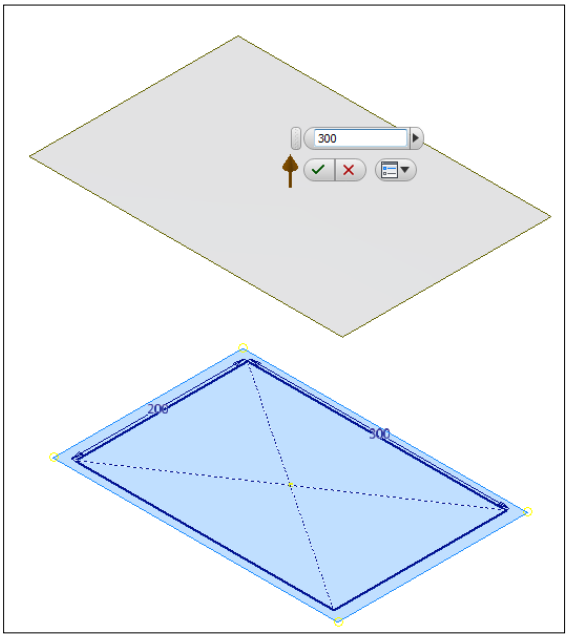


FIG 1.0

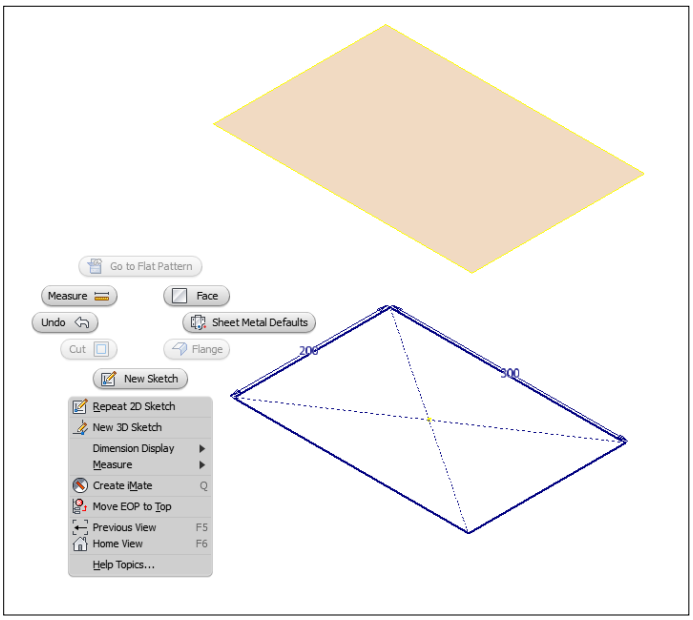


FIG 2.0

Step # 8. Right Click and select *New Sketch*.

Left Click on the border of the new Work plane just created.

Step # 9. On the new sketch create a circle with the Circle command, as shown in Fig 3.0 Give it a diameter value of 150 mm

Finish the sketch.

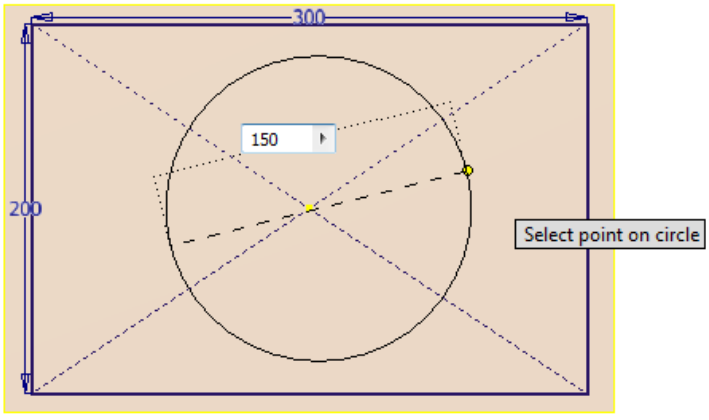


FIG 3.0

Step # 10. From the create tab select the *Lofted Flange command* as shown in Fig 4.0

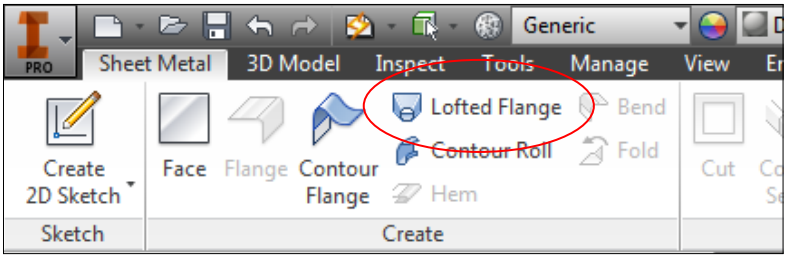


FIG 4.0

Select the circle sketch for *Profile 1* and the rectangle sketch for *Profile 2*. A lofted flange will be created like that in Fig 5.0

OK the dialog box.

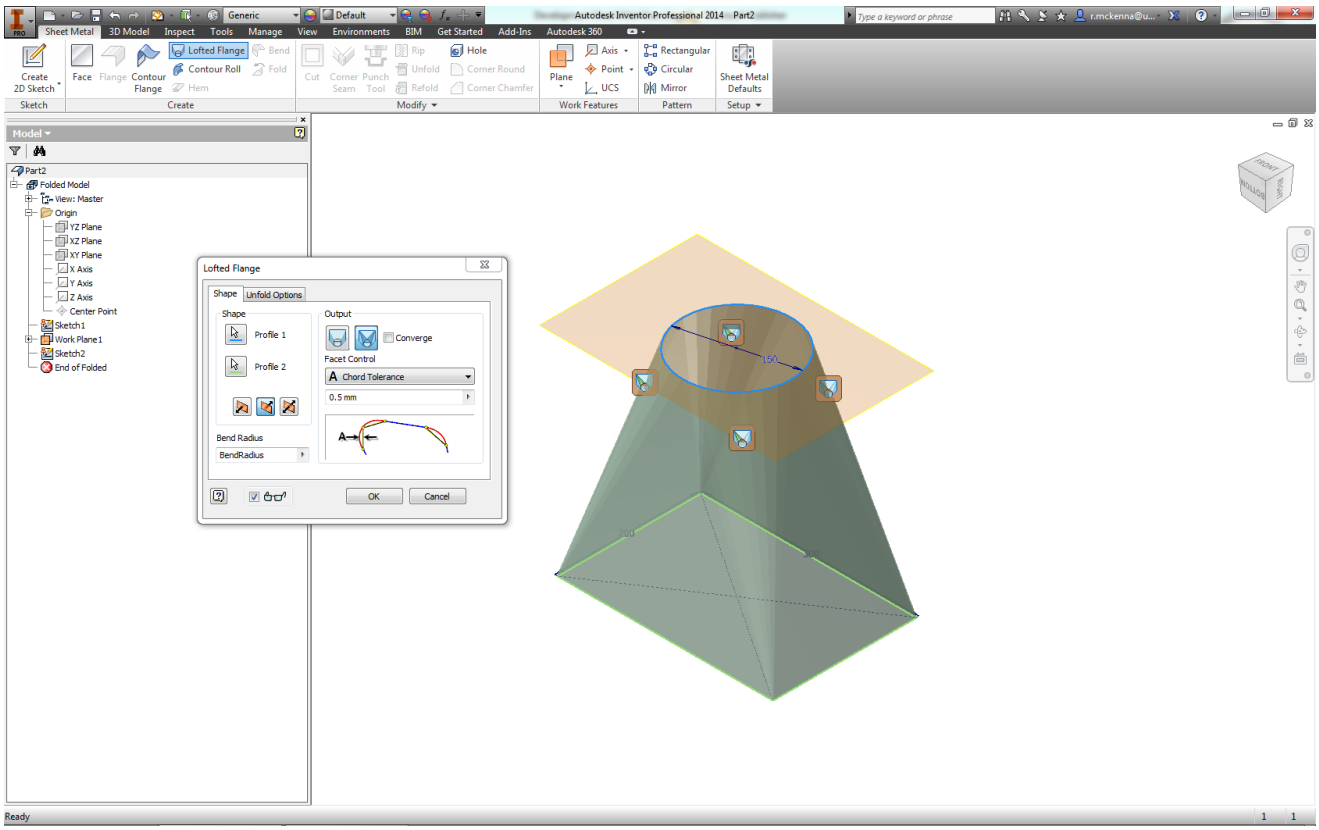
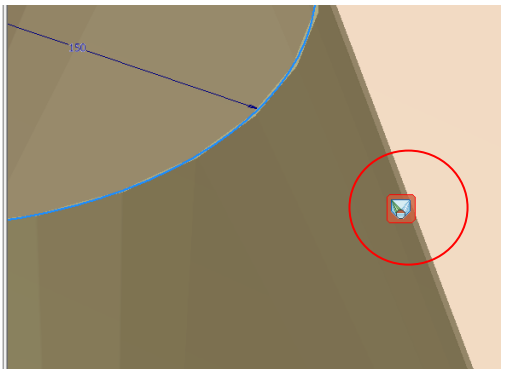


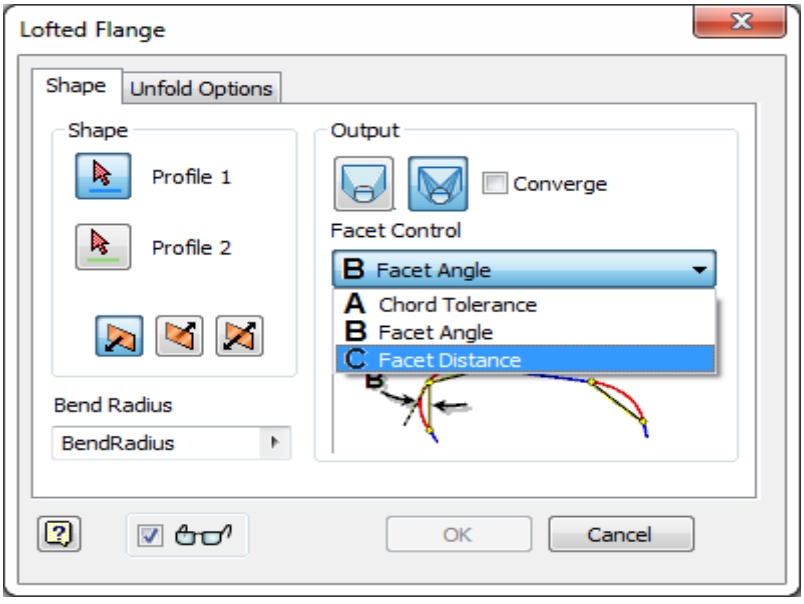
FIG 5.0

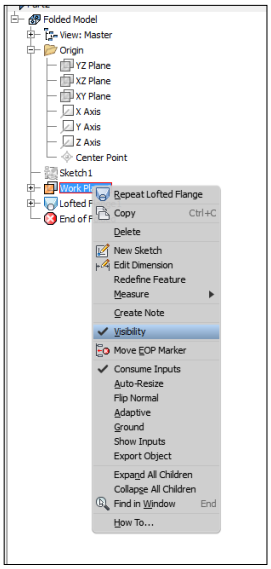
Note. *Experiment with values in the Lofted Flange Dialog box, alter the way Inventor calculates the bend facets Globally.*

(Facet Angle, Chord Distance or Facet Distance)

Alternatively select one of the Bend "glyphs" in Fig 5.0 to edit the bend method locally.



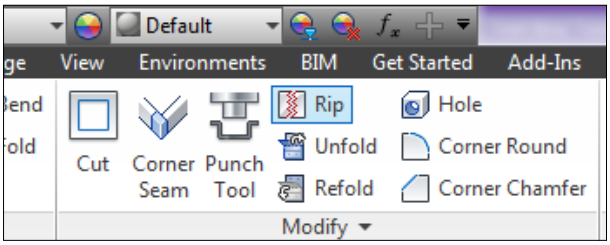


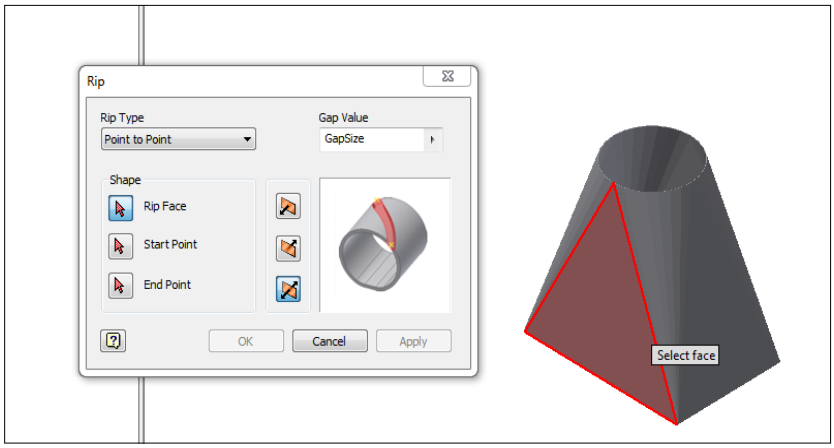


Control the visibility of the work plane by right clicking on it in the browser and unchecking *"Visibility"*. Alternatively right click on the work plane itself, and select "*Visibility"*

Step # 11. In order to create a flat pattern, the model must have a gap in the model or "*RIP"*

From the *Modify Tab* select the *RIP Command.*



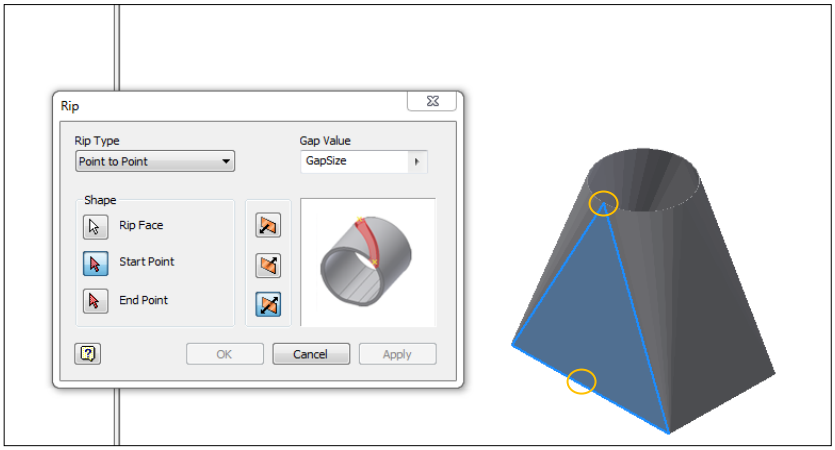


In this example, change the *Rip Type to Point to Point.*

Select the *"Rip Face"* button , then left click on the face where the Rip will be situated.

As seen in Fig 6.0

FIG 6.0



Once the face is selected, used object snaps to find the *Start* and *End* Vertices of the face.

The vertices locations are highlighted in Fig 7.0

After selecting the vertices, Hit OK

Note. *Experiment with the value in "Gap Value"*

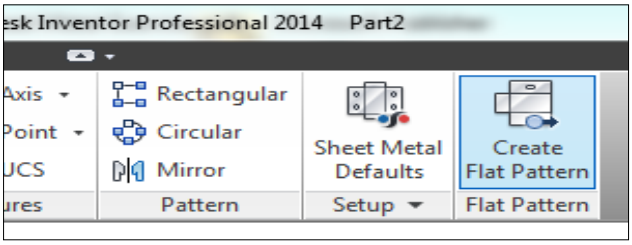
GAPSIZE by default is related to the sheet metal thickness value.

FIG 7.0

Step # 12. Create a flat pattern.

From the Flat Pattern Tab, select the *Flat Pattern* command.

A Flat Pattern of the Lofted Flange will be developed, as seen below in Fig 8.0



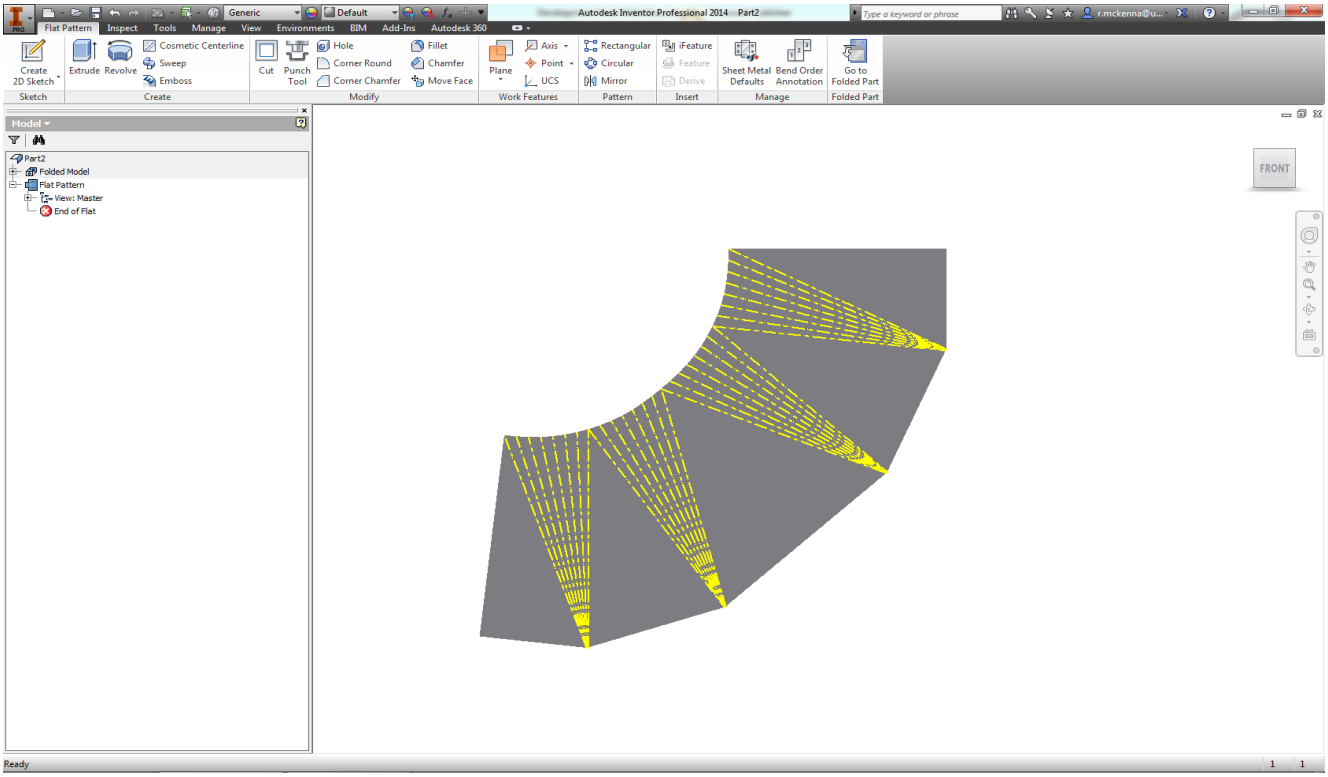


FIG 8.0

Note. To toggle between Flat Pattern and Folded Model, select the "Go to Folded Part" command on the Ribbon Menu.

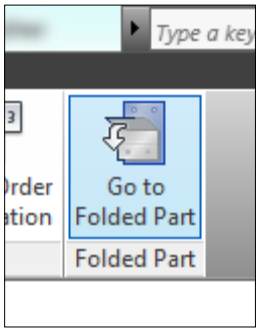


FIG 9.0

Alternatively double left click on the "Folded Model" increment in the browser.

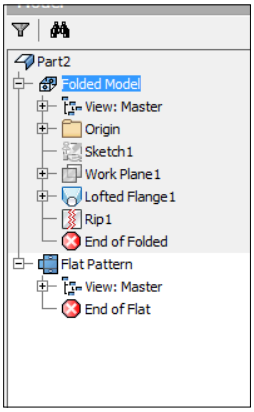


FIG 10.0