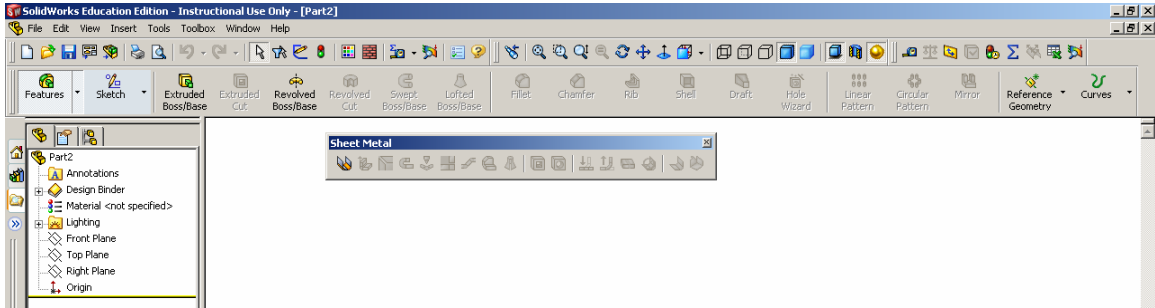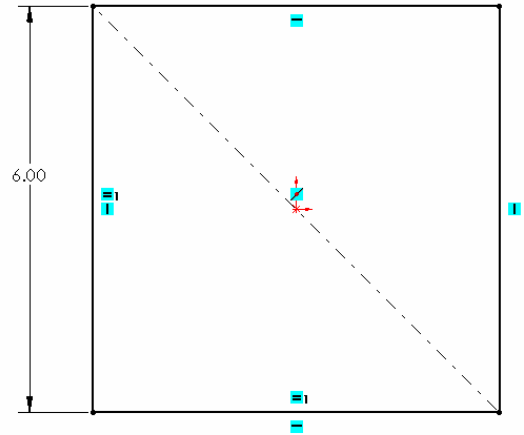# SolidWorks for Sheetmetal

1. Start a new inch part, right click on any toolbar and check the "Sheet Metal" tool for the Sheetmetal toolbar to be available.


2. Create a new Sketch on the front plane. Sketch a 6.00 inch square rectangle that is centered on the UCS origin. Exit Sketch


3. Orient your sketch to an Isometric view.
    The first tool on the sheet Metal toolbar is Base – Flange/Tab, we will use this to give our sketch thickness. Set the part thickness to .0747 inches (14 gage), K Factor .50 and leave Auto Relief to the default settings.
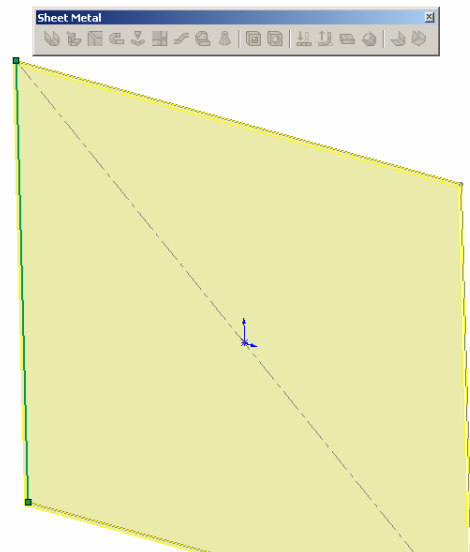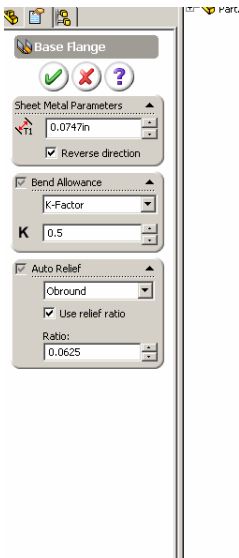
4. If you noticed there is no library of material for the K Factor and you were not prompted for the bend radius. The K Factor can be customized by Excel spreadsheets and the bend radius can be set after the part has thickness by editing the Sheet Metal feature in the browser bar. Set the bend radius to .125 inch.
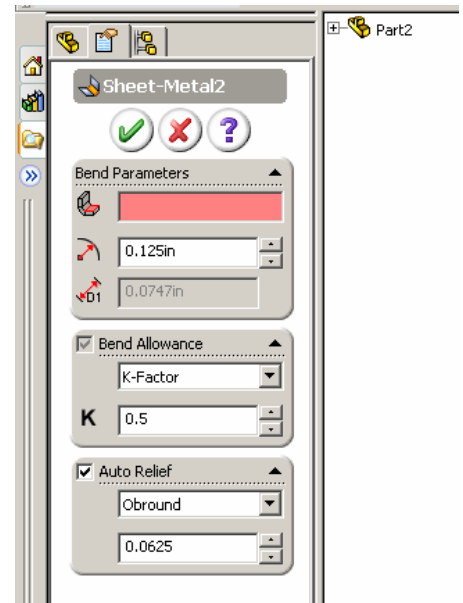
5. Next we are going to add an Edge Flange to one of the sides of the base part. We will add a 90 degree X 2.50 inch flange, create the settings so after the flange is complete the total measurement of the flange will be 2.50 inches and the original length of 6.00 inches will increase by the sheet metal thickness.
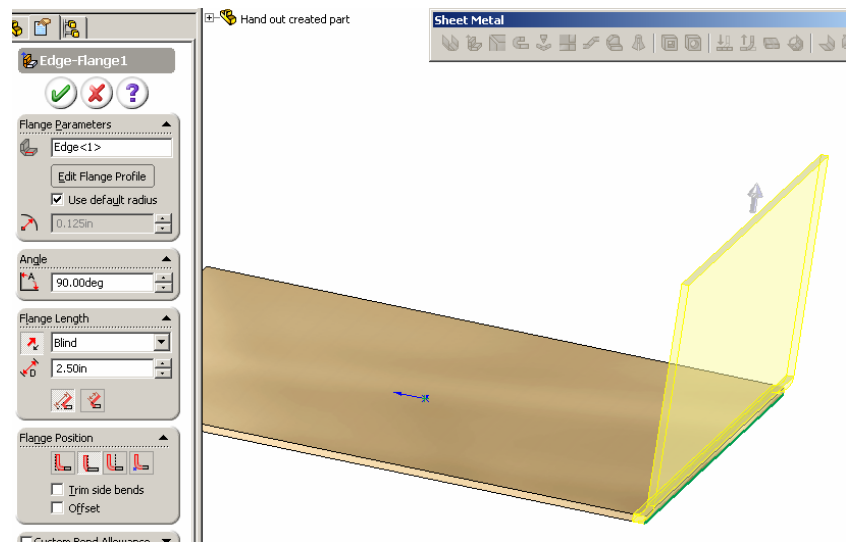
6. Create another Edge Flange on an edge perpendicular to the last flange. We want to add this flange as a 90 degree X 2.50 inch with the settings set so the final flange will also be 2.50 inches tall, however; on this flange we don't want the 6.00 inch base dimension to increase with the addition of material.

Mark Eilers
Southeast Community College
Milford, NE
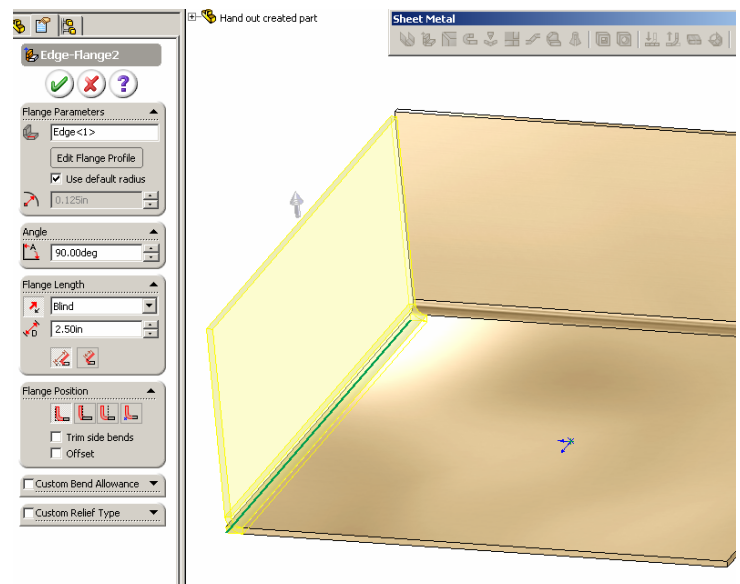
7. The next tool on the Sheet Metal toolbar is Miter Flange. We will create a Miter Flange on the combination of the first flange that we applied and the base feature. Create a new sketch on this face. Sketch a line that is .500 inch long from the outside edge. Exit sketch.
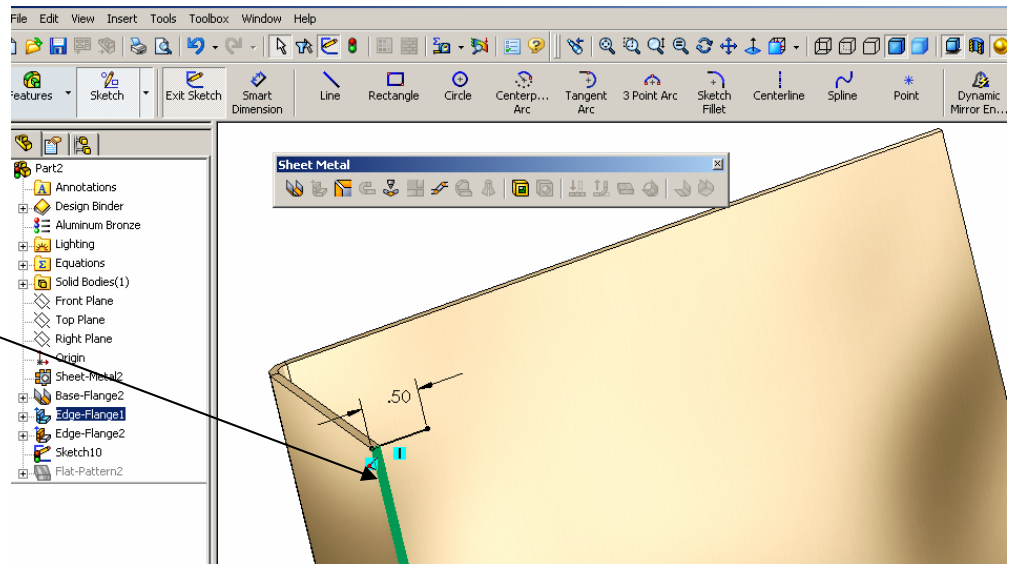
8. Select the Miter Flange button and the size of the flange is selected from the sketched .500 inch line. The first edge is also selected from the edge that the line was sketched from.

Select the outside edge of the bend and the far side edge of the original base feature.

We will keep the material inside of the part. Turn on trim side bends and leave the gap distance to .02 inch.

Set the start offset to 1.00 inch and the end offset to .500 inch.

Mark Eilers
Southeast Community College
Milford, NE
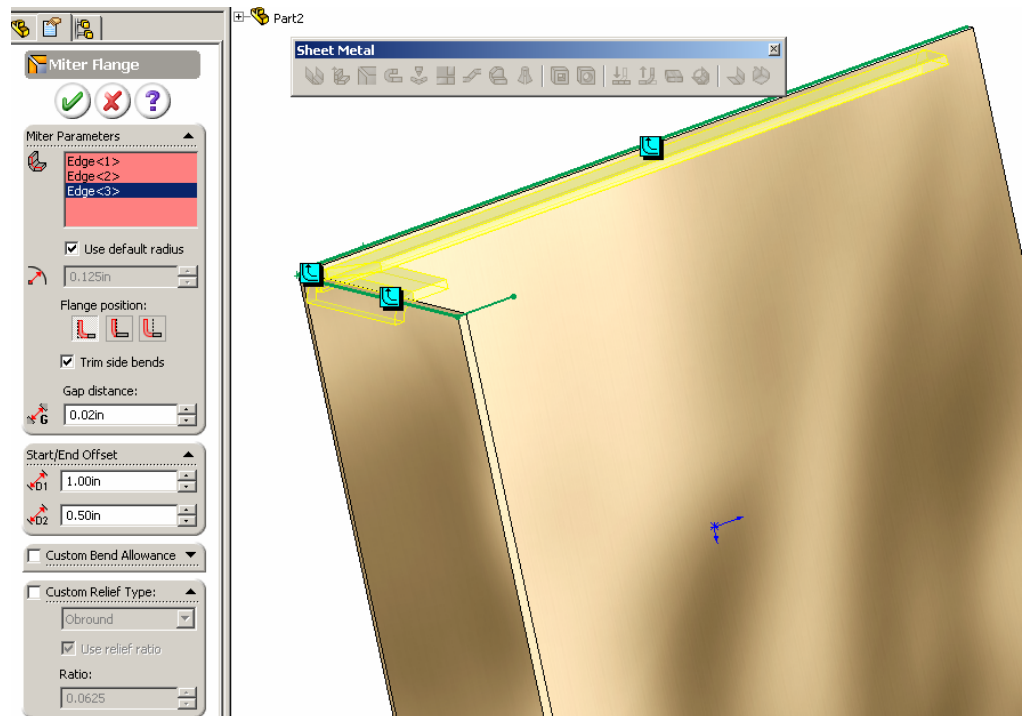
Finished part up to this point.
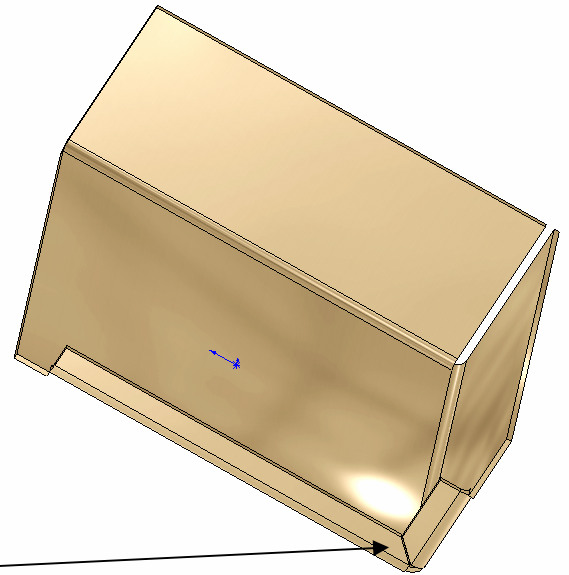
The Auto Relief creates the cuts at a transition point based upon a ratio of the sheetmetal thickness.

9. The next tool on the Sheetmetal toolbar is Hem. We are going to create a hem on the flanged side of the part that we started the Miter Flange on. Create a Rolled hem on the outside edge with the material inside. Roll the hem 270 degrees and give the hem an inside radius of .125 inches.
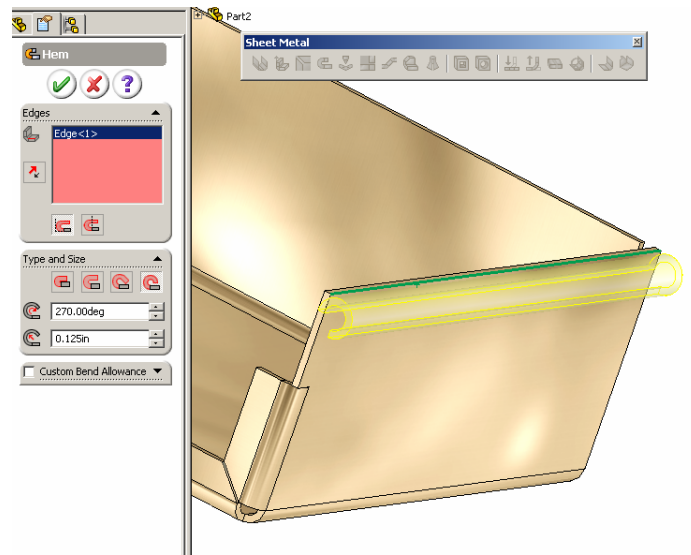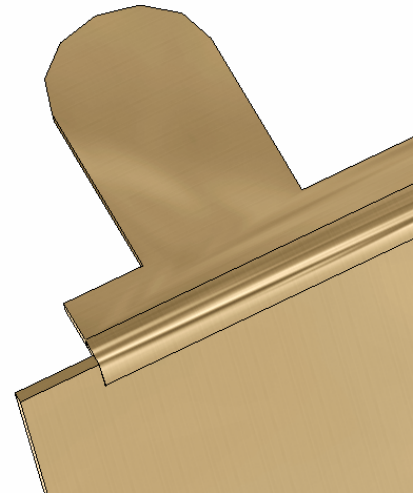

10. The next tool on the Sheetmetal toolbar is Sketched Bend. On the .500 face of the long side of the Mitered Flange we will create a sketch for the Sketched Bend.
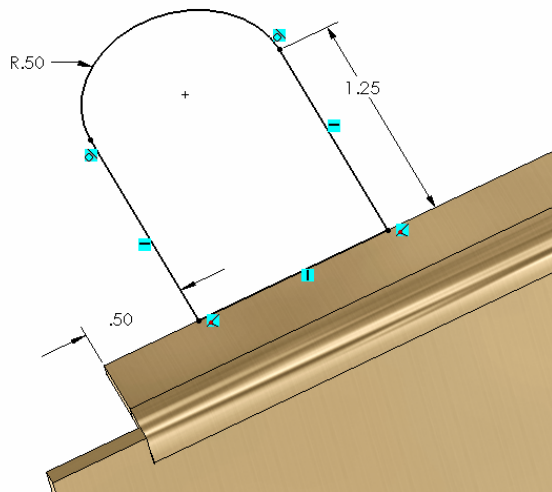
11. Once the sketched is complete select the Base – Flange/Tab tool on the Sheetmetal toolbar, this will give the last sketch the correct thickness based upon prior settings.

12. Next we need to create a sketch on this tab that will start and stop on the far edges of the tab. The sketched line can either be perpendicular to the edges or at a selected angle. I placed my line .375 inches from the miter flange. Exit sketch and select the Sketched bend tool. The first requirement is selected the Fixed Faces, then you can set the bend angle requirements.
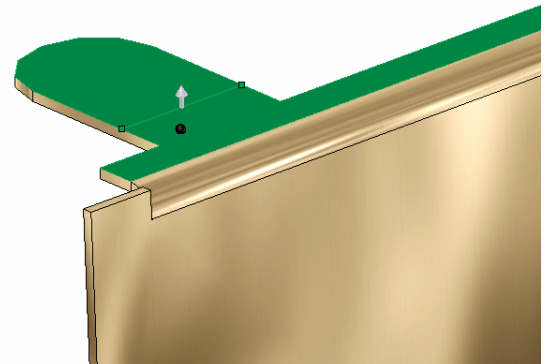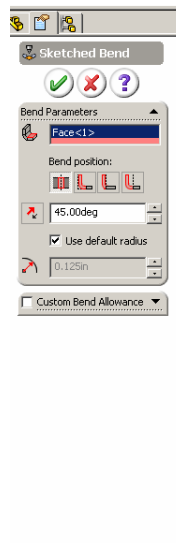
13. Next we will explore the Jogged tool. Create a sketch on the same face as the last step. Exit sketch, select the Base – Flange/Tab tool and give this new tab thickness.
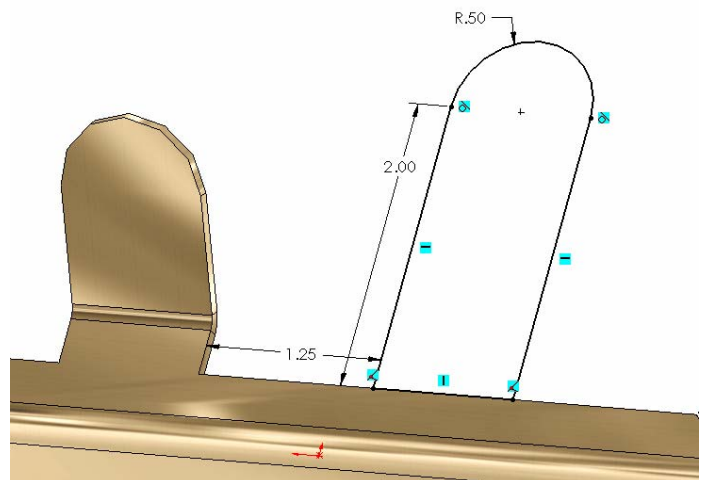
14. Create another new sketch on this tab and sketch a line that will be used for the Jog tool. Dimension as shown and exit sketch.
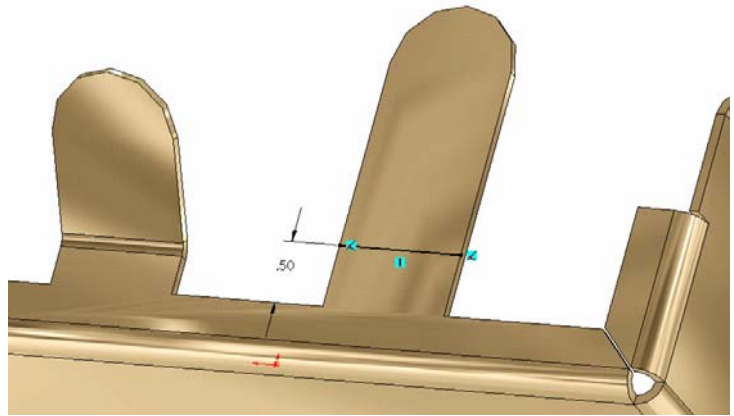
15. Select the Jogged tool. The first requirement is to select the fixed face. Then you can set the jog offset, the dimension position the jog position and the angle of the jog.
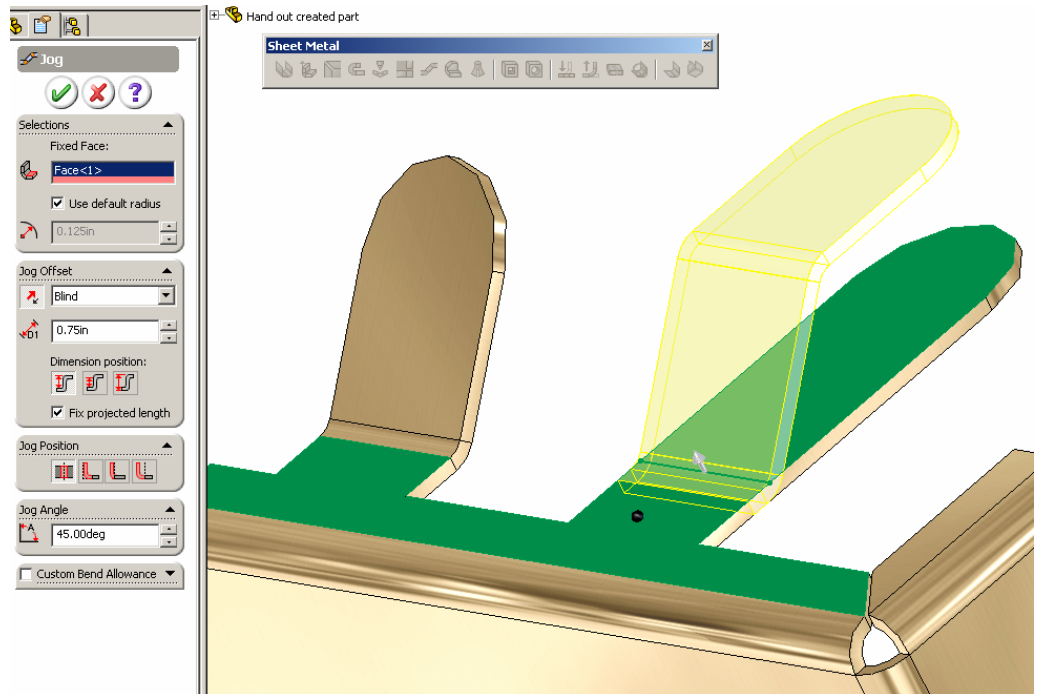
Example part up to this point

Mark Eilers
Southeast Community College
Milford, NE
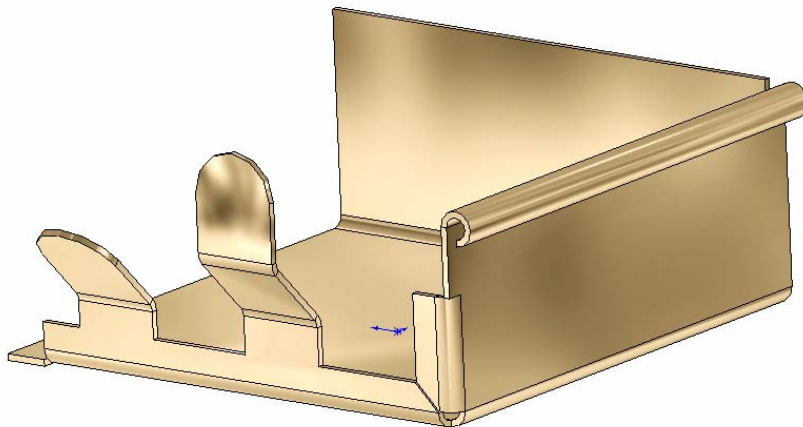
16. Add a 2.50 inch Edge Flange on the side of the base part that we haven't used yet.
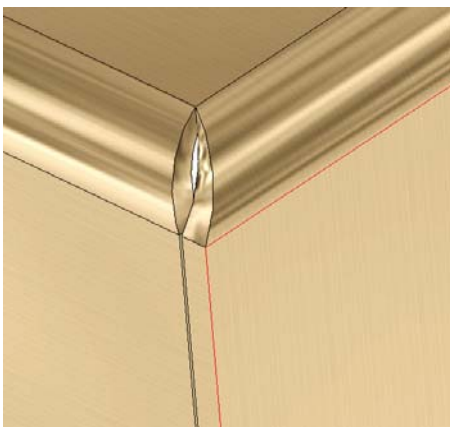
    Set the flange position to material inside and select the trim side bends option.

    Now we have a sheetmetal corner that we can use the Closed Corner tool on.

17. Select the Closed Corner tool and select the face that you would like to extend and the corner type that you prefer.
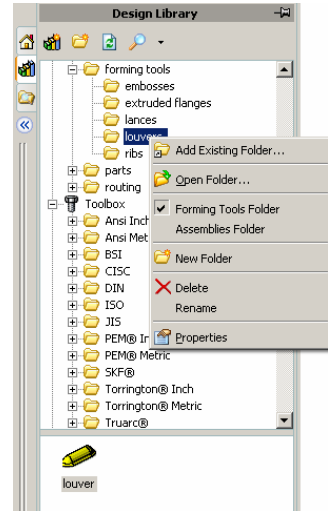
Finished corner

Mark Eilers
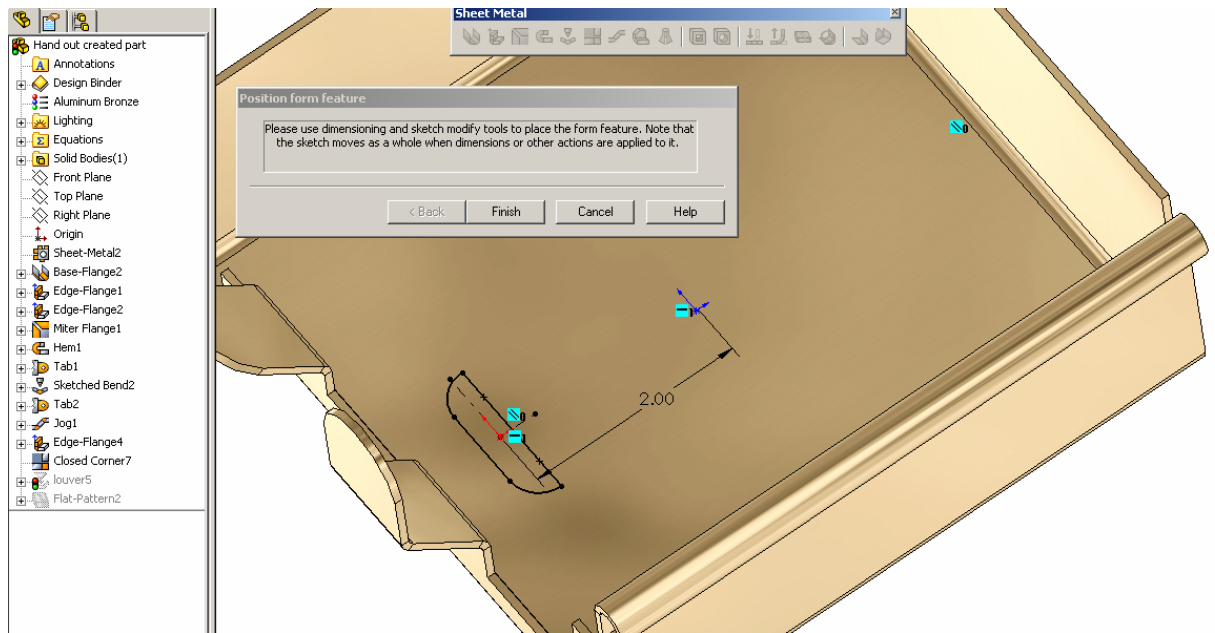Southeast Community College
Milford, NE
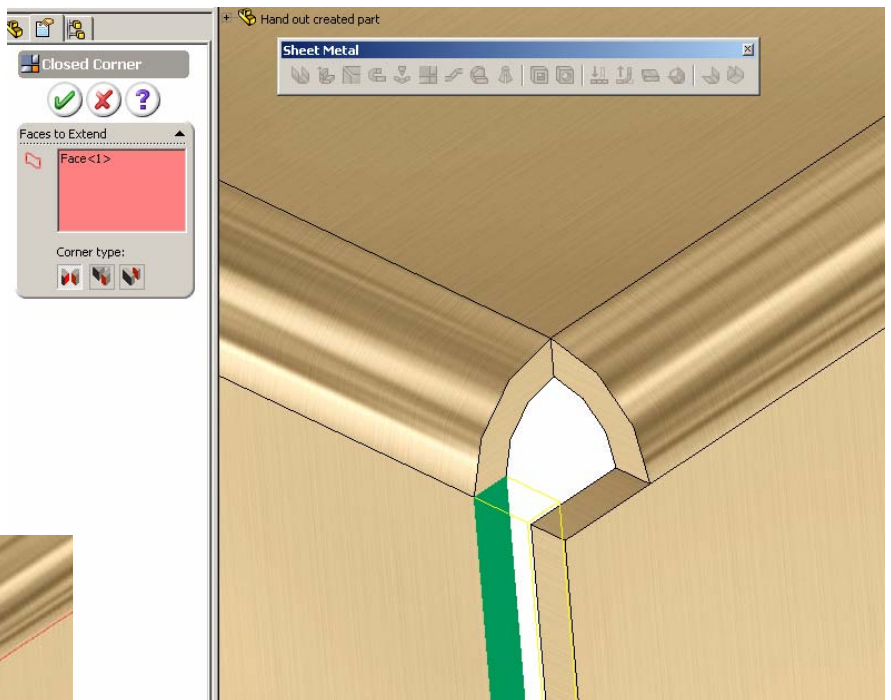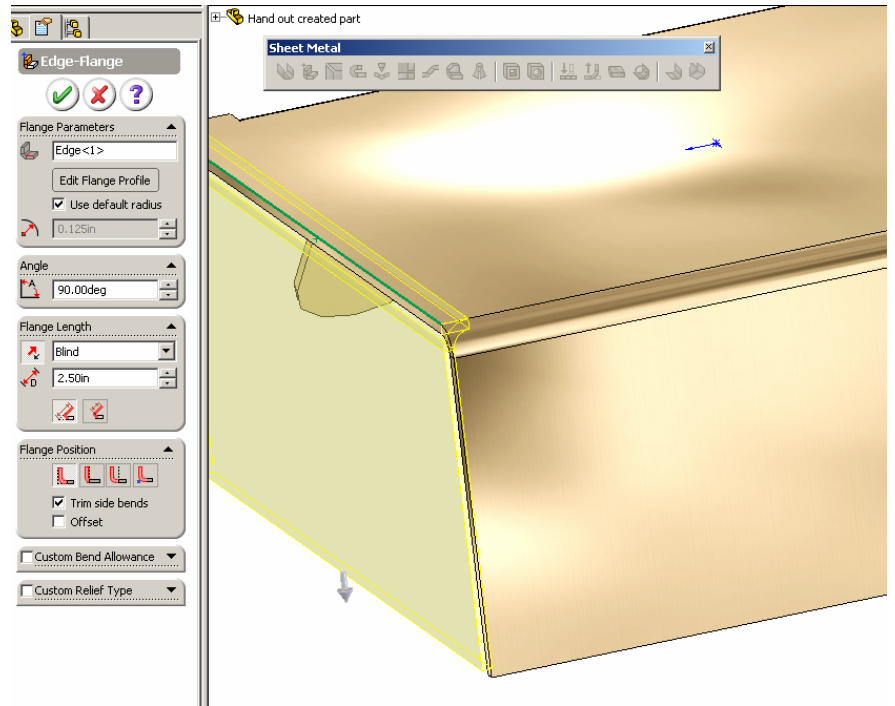
18. Next add louvers and pattern

Go to the Design Library and make the Forming Tools
Folder the current folder by using the context menu.

19. Drag and Drop the louver to the surface that you
    would want the forming punch to come from. (The first
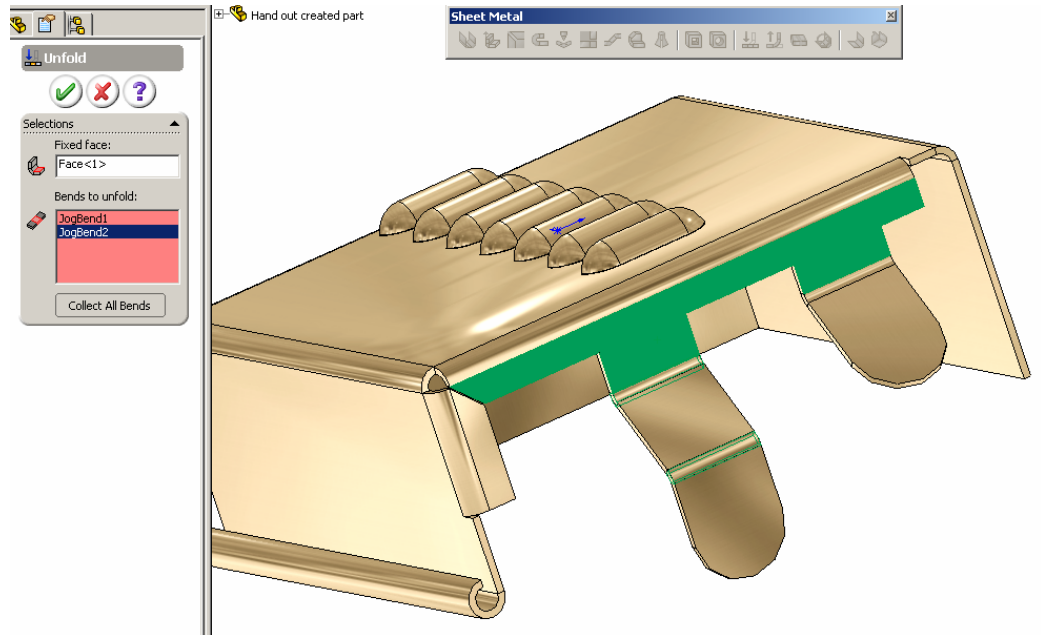   surface that the forming punch would contact)

Apply the Geometric Constraints and Dimensions for your requirement. And finish

20. Pattern the louvers.

We want 7 louvers that have a spacing of .75 inches.

21. Now we will use the Unfold tool to unfold the jogged tab. Select the fixed face and then select the two folds that need to be unfolded.

22. Next we will create an opening on the tab that we just flattened out. Create a sketch on the face of the tab, sketch the geometry, constrain and dimension. Exit sketch.
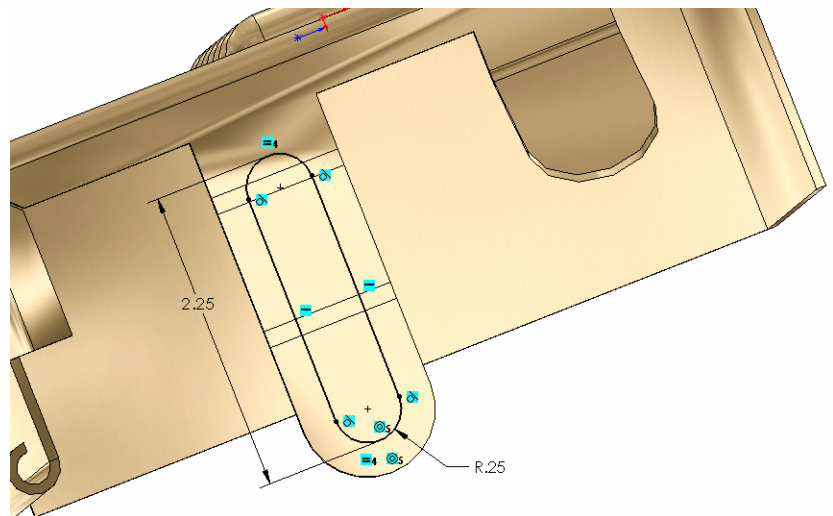
23. Using the Extrude Cut on the Sheetmetal toolbar we will cut out the last sketch, linking the sheet metal's thickness to the depth of the cut.
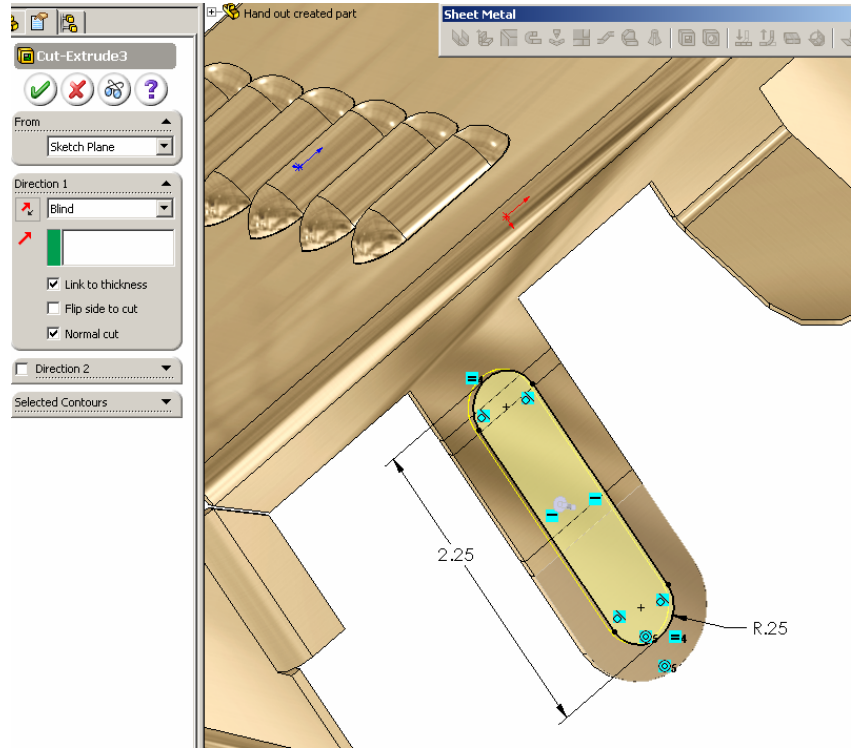

24. Now we can refold the tab by using the Fold tool on the Sheetmetal tool bar, returning the tab back to its original shape.

Mark Eilers
Southeast Community College
Milford, NE
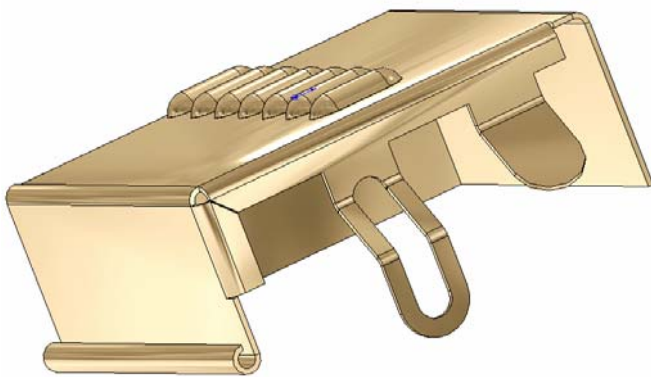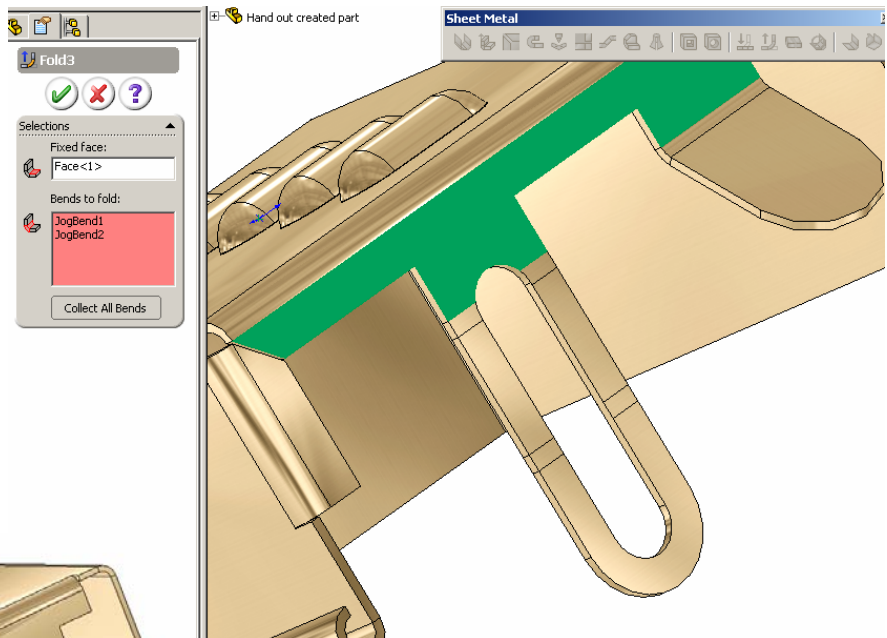
25. Now we can create a flat sheet metal layout of the part by selecting the Flatten button on the Sheet Metal toolbar.
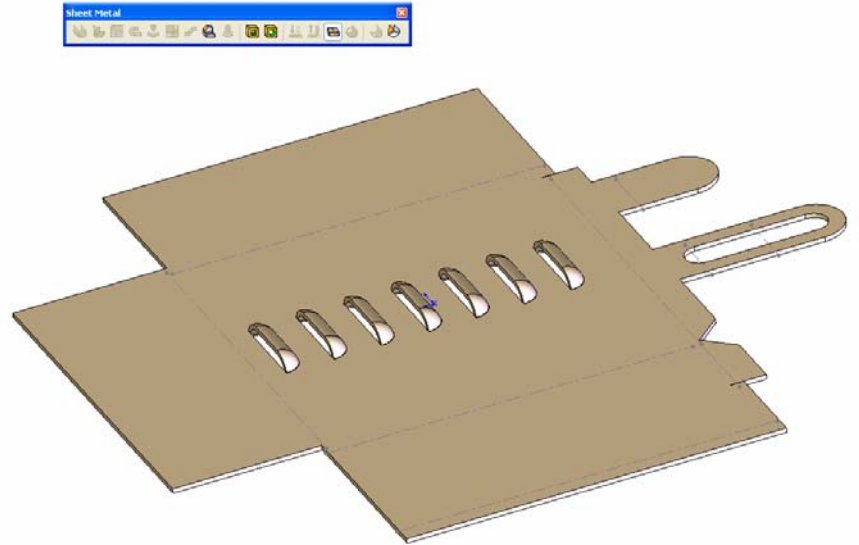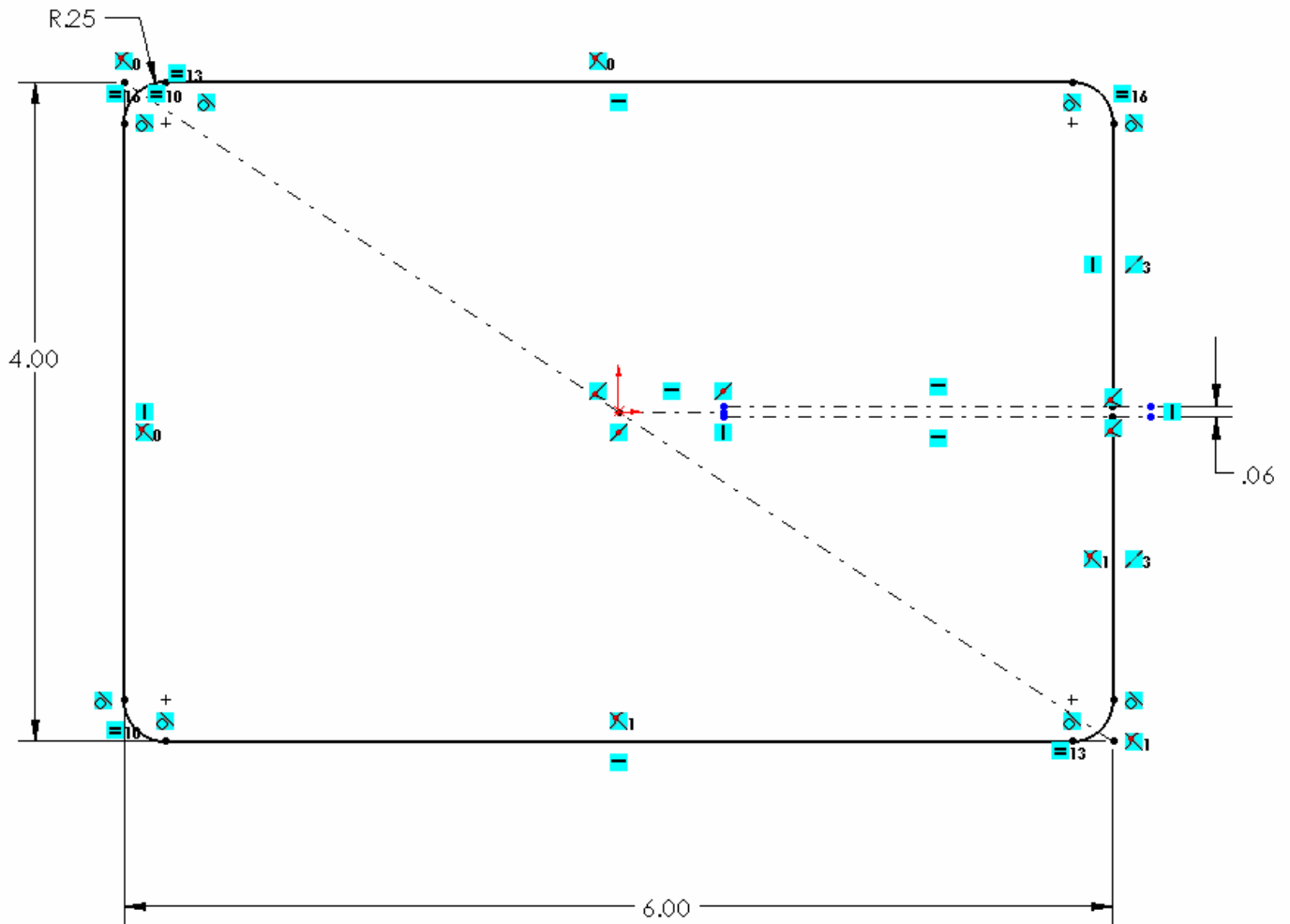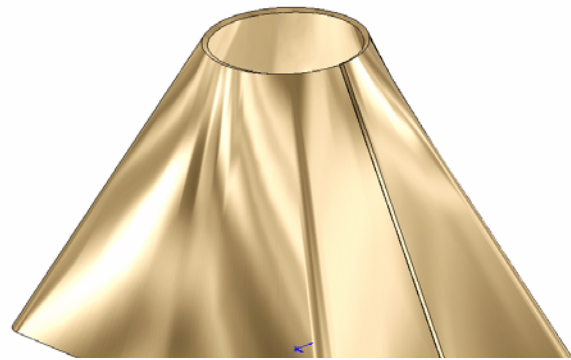

26. Now we can use the Corner-Trim relief options to add a corner relief. Set the Relief Options to .1875, select the edge in the corner and set the Relief style to Circular.
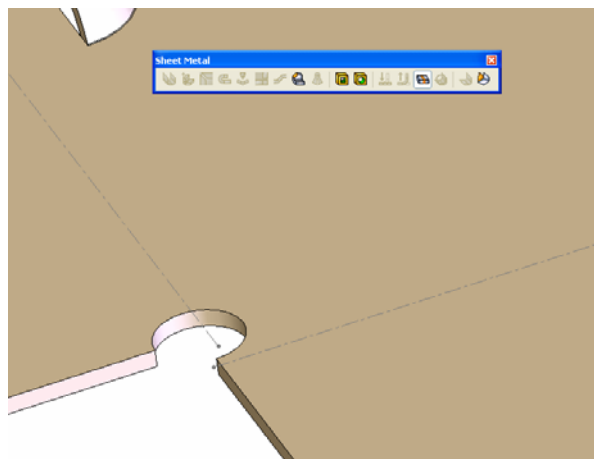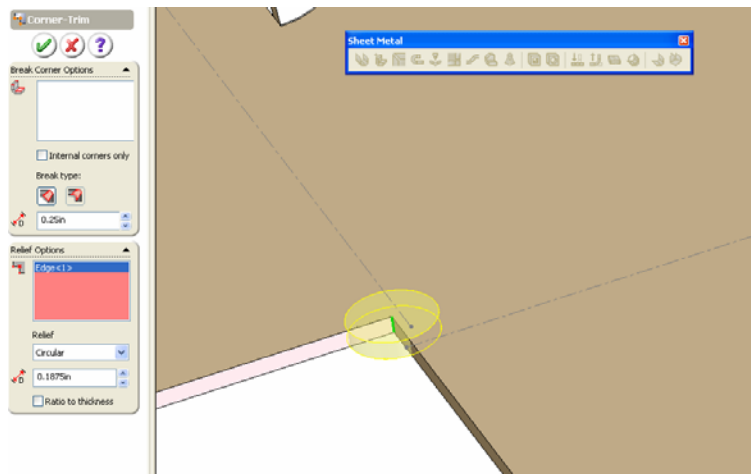
Now let's use SolidWorks to create a sheet metal transition.

27. Start a new inch part and create a new Sketch on the front plane. Constrain the geometry and dimension as shown.
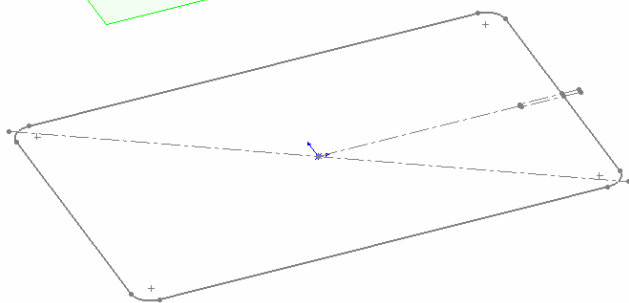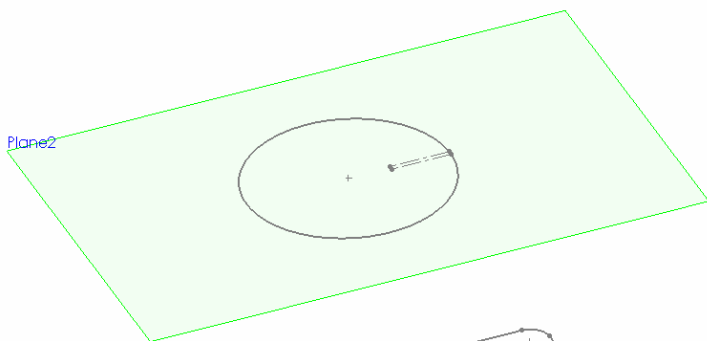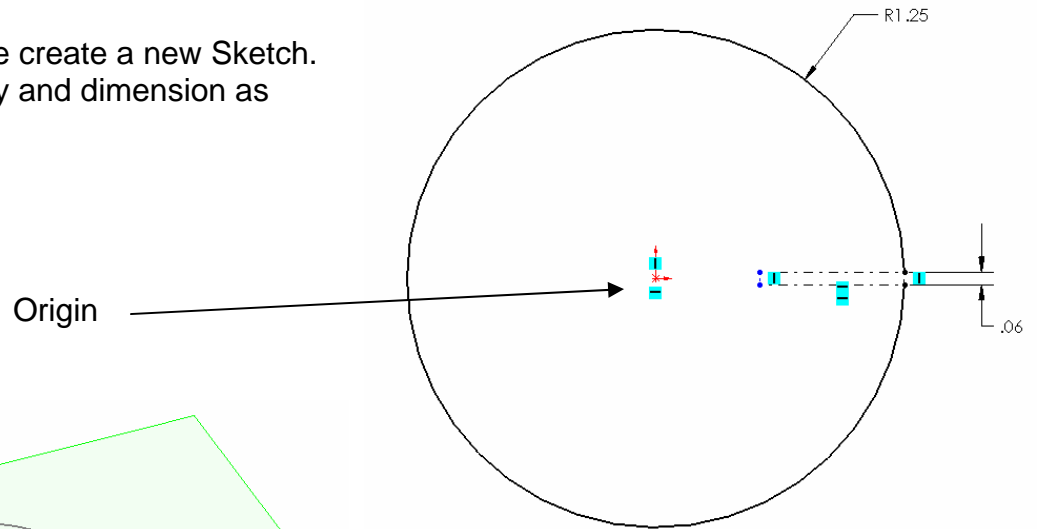

R.25

4.00

.06

6.00

The rectangular geometry on the right side of the sketch is used to create and center a gap in the sketch geometry. Finish sketch and Exit Sketch.

28. Create a new offset Workplane 4.00 inches above the Front Plane.
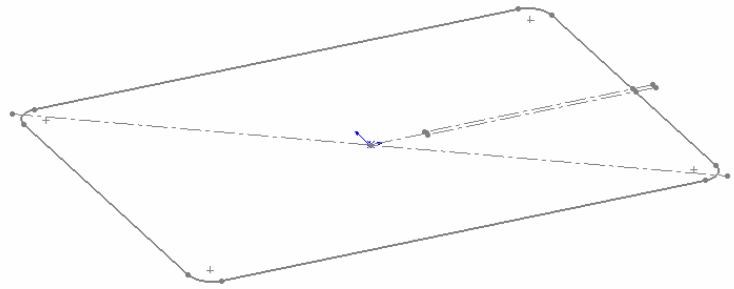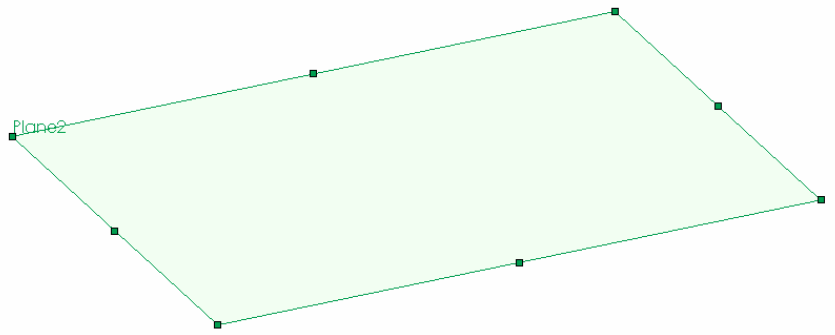
29. On this new work plane create a new Sketch. Constrain the geometry and dimension as shown.

Plane2

R1.25

Origin

.06

Plane2

Exit Sketch

30. On the Sheet Metal toolbar the Lofted Bend tool is now available. Select the Lofted Bend tool and then select the first sketch profile and then select the top sketch profile. Set the material thickness to .0747 inches (14 gage).

Reminder: It does matter where you select these profiles. Select them close to their profile gaps.

Mark Eilers
Southeast Community College
Milford, NE
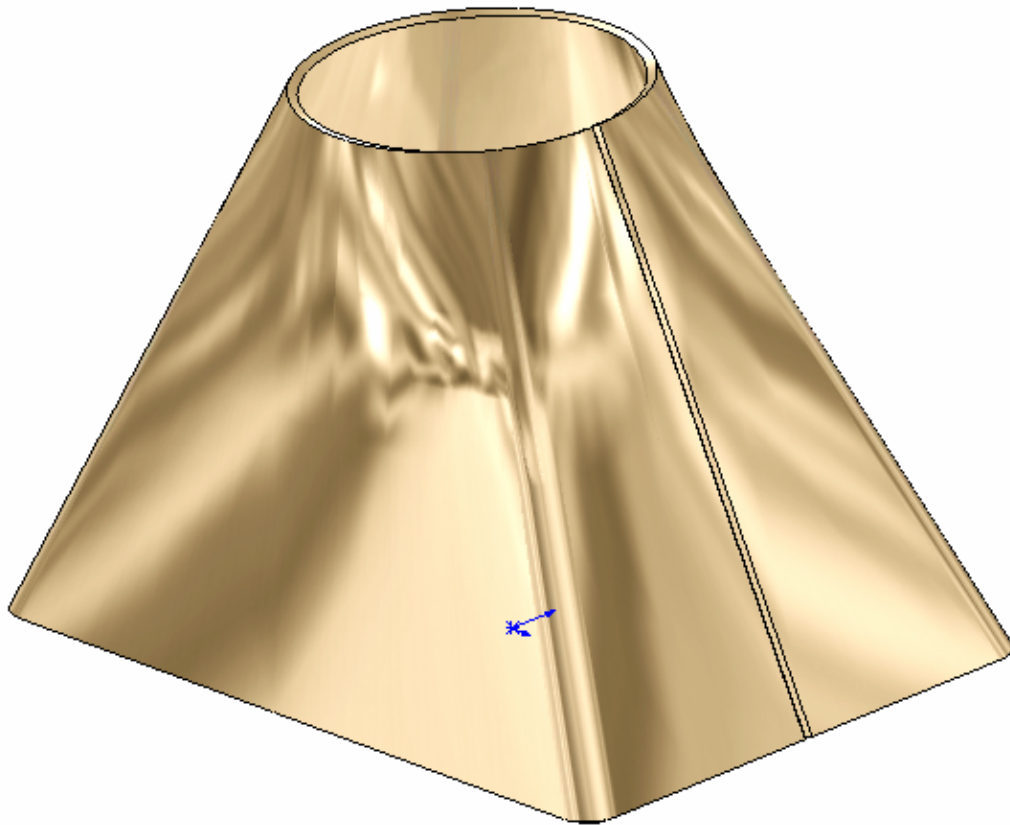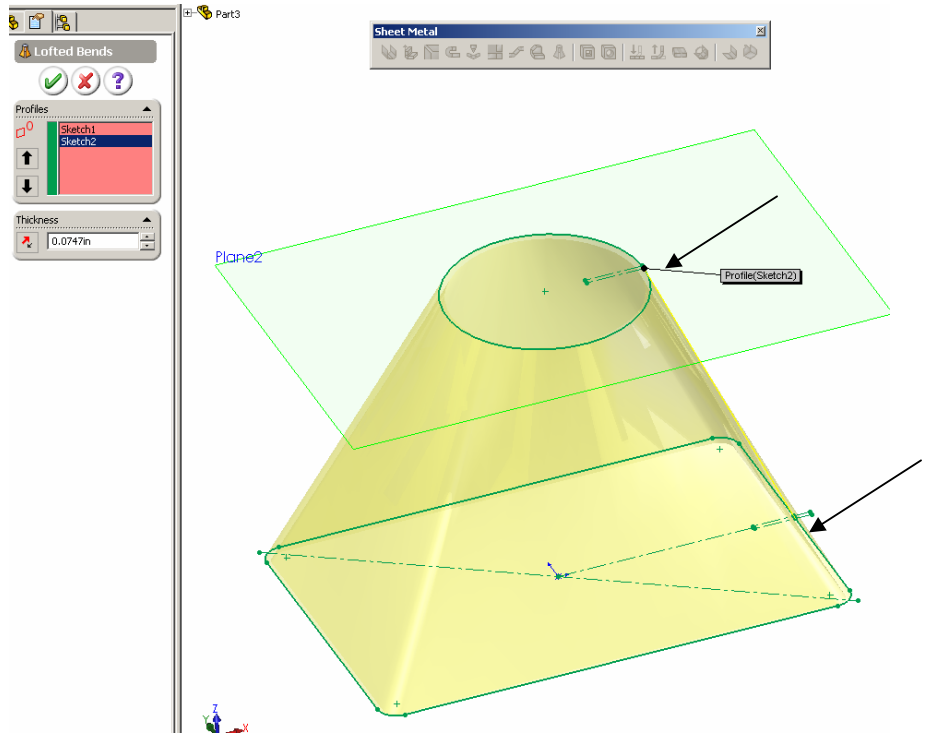
31. Select Flatten on the Sheet Metal toolbar and the sheetmetal transition that we just created will turn into a flat pattern layout.