Introduction to Creating Sheet Metal Parts
Creo 2.0

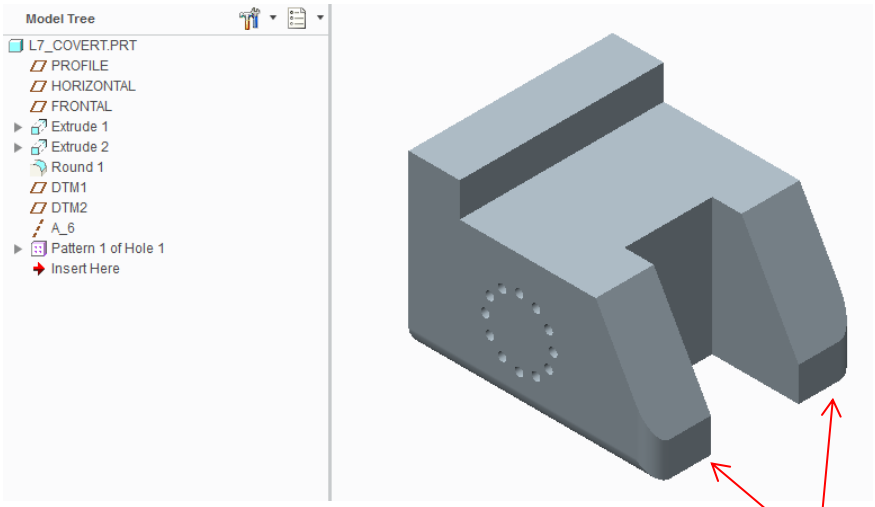
Goals:

1). To develop a general overview of creating sheet metal parts by converting solid geometry to sheet metal.
2). To understand how flat pattern geometry is produced.
3). To produce appropriate project documentation.

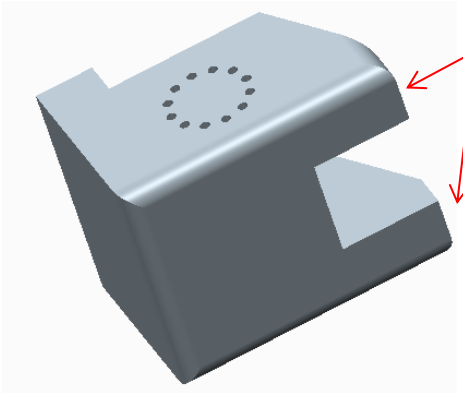
Rounded Corners are **highly** exaggerated for visualization purposes. Normal bend radius would be twice the material thickness. Notice the “rips” in the long vertical edges at the back, and the short horizontal edges in the front. Dimensional values are not important.

Level 7:
Method 1

Create base geometry with extruded protrusions, cuts, holes etc.

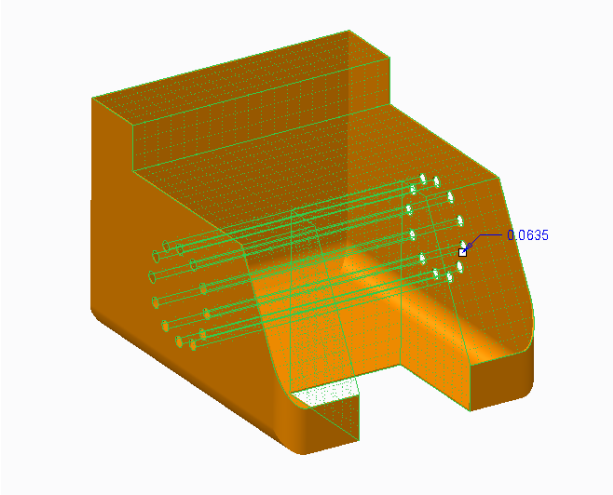


Add rounds at the edges where appropriate as shown below.



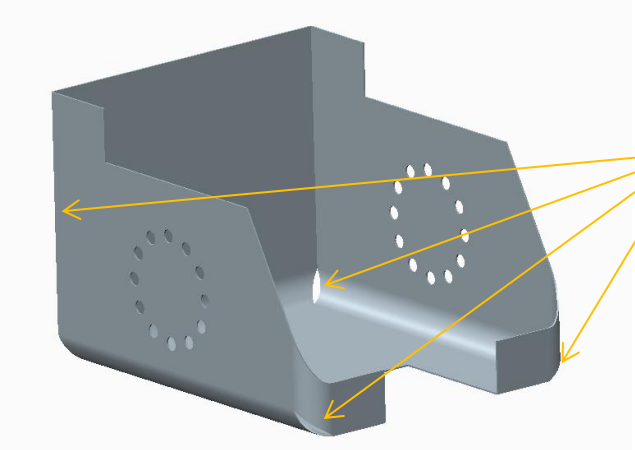
There are NO rounds on these edges. If there were you could not add an edge rip.

Convert to Sheet metal
In the Model Tab, select the down arrow in the Operations group.
Select Covert to Sheet metal.
Select Shell. Under the References tab, select the surfaces to remove. The thickness is related to the material (16g Galvanized).



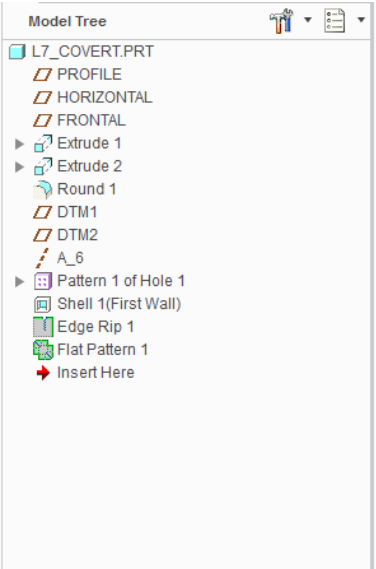
Use the Rip (Edge Rip) located in the Engineering region of the Model tab to create rips so we can unbend the part.

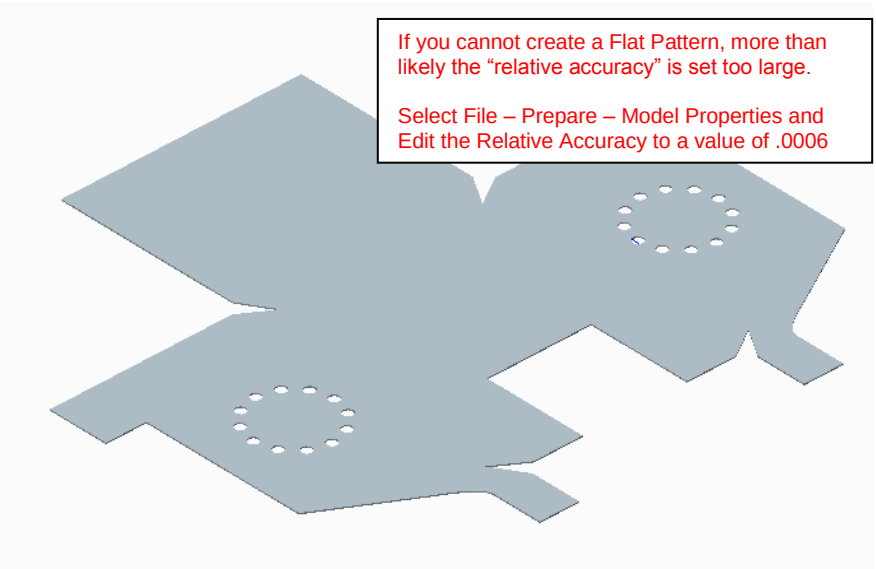
The rips are shown as you select the edges.



Rip these edges (Edge Rip), as well as the vertical edges in the back.

To flatten, use the *Flat Pattern* pick (Bends region of the Model tab), selecting the bottom surface as the Reference for fixed geometry.





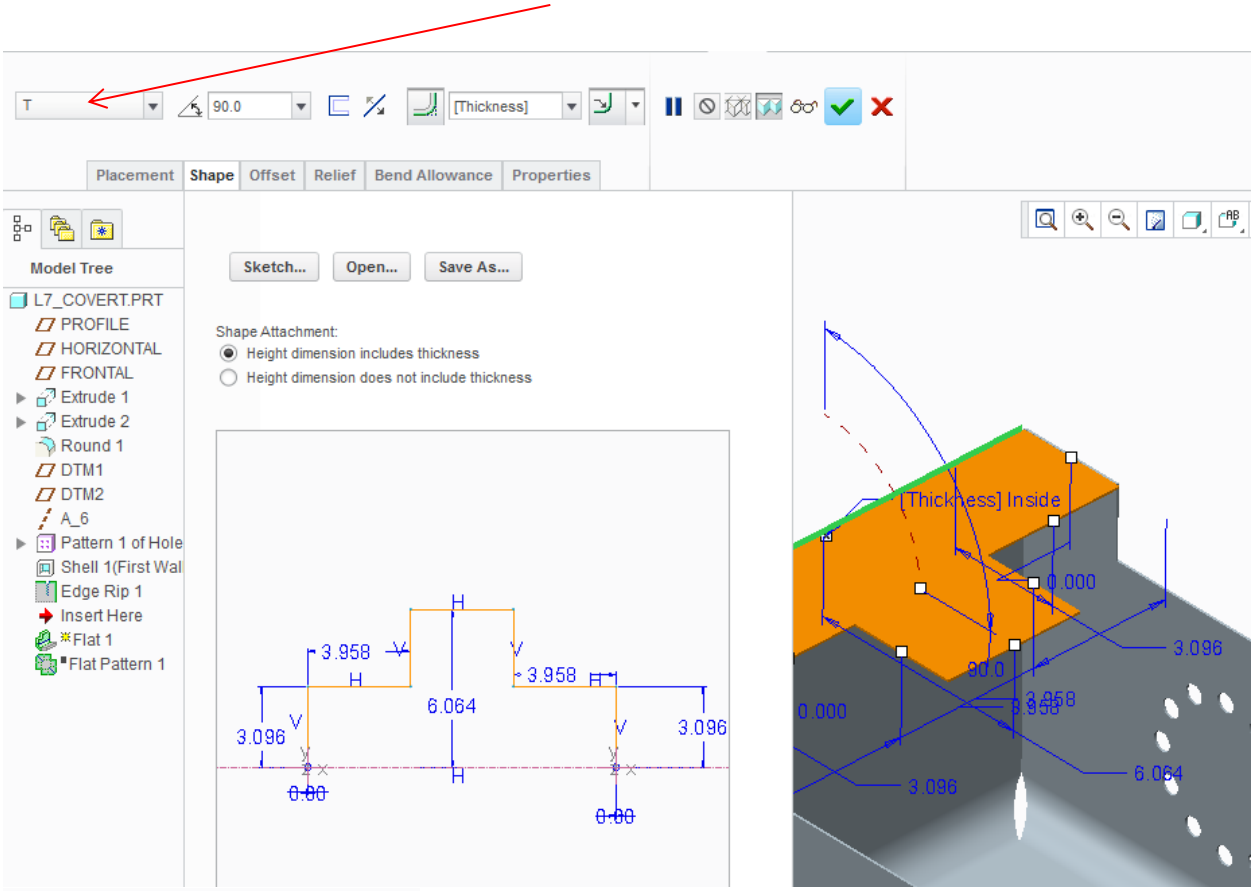
If you cannot create a Flat Pattern, more than likely the “relative accuracy” is set too large.

Select File – Prepare – Model Properties and Edit the Relative Accuracy to a value of .0006

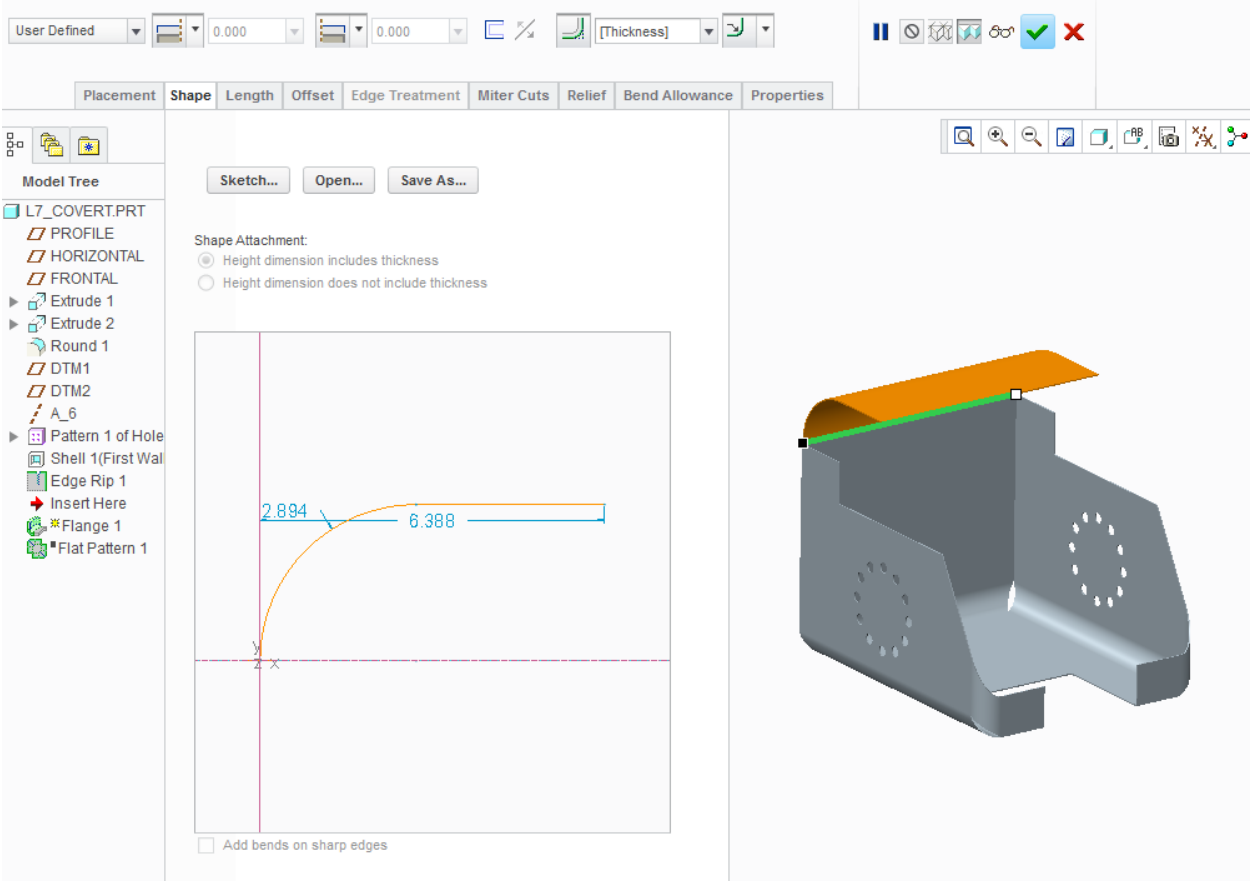
Flat wall creation

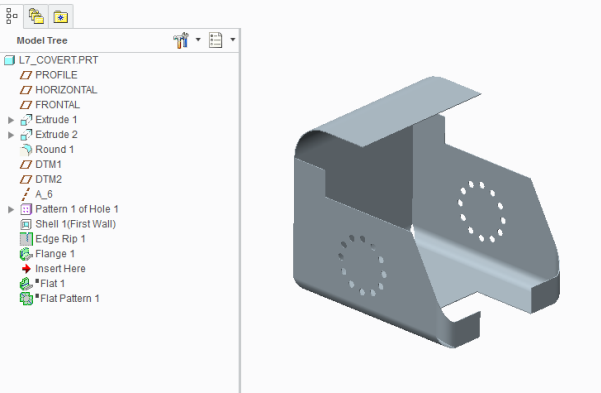
Select Flat from the Shapes region in the Model tab.

Generate a flat wall similar to the following (note the T *Shape*):



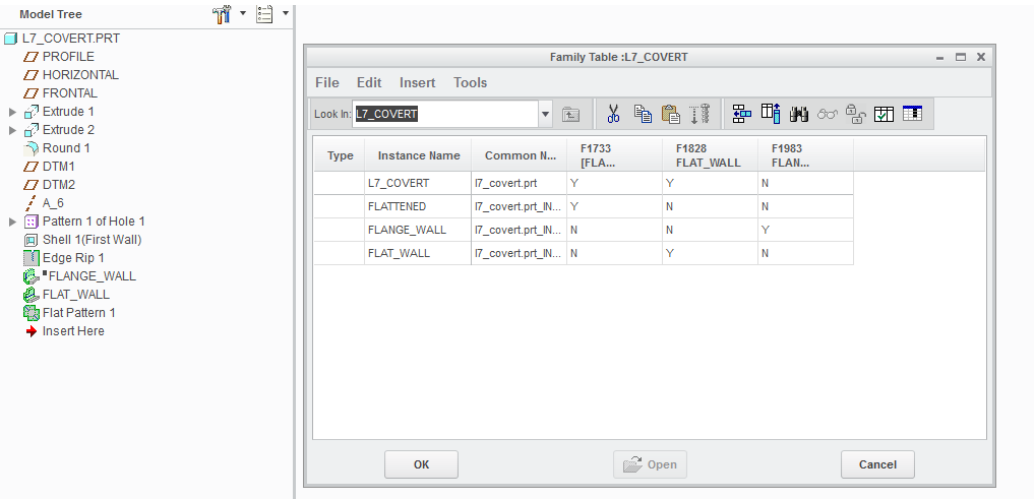
Suppress the Flat wall and insert a Flange wall similar to the one shown.
Use the inside edge of the vertical piece for the Placement edge and select Sketch on the Shape tab to produce the arc and straight line. You must have tangent conditions at both ends of the arc.



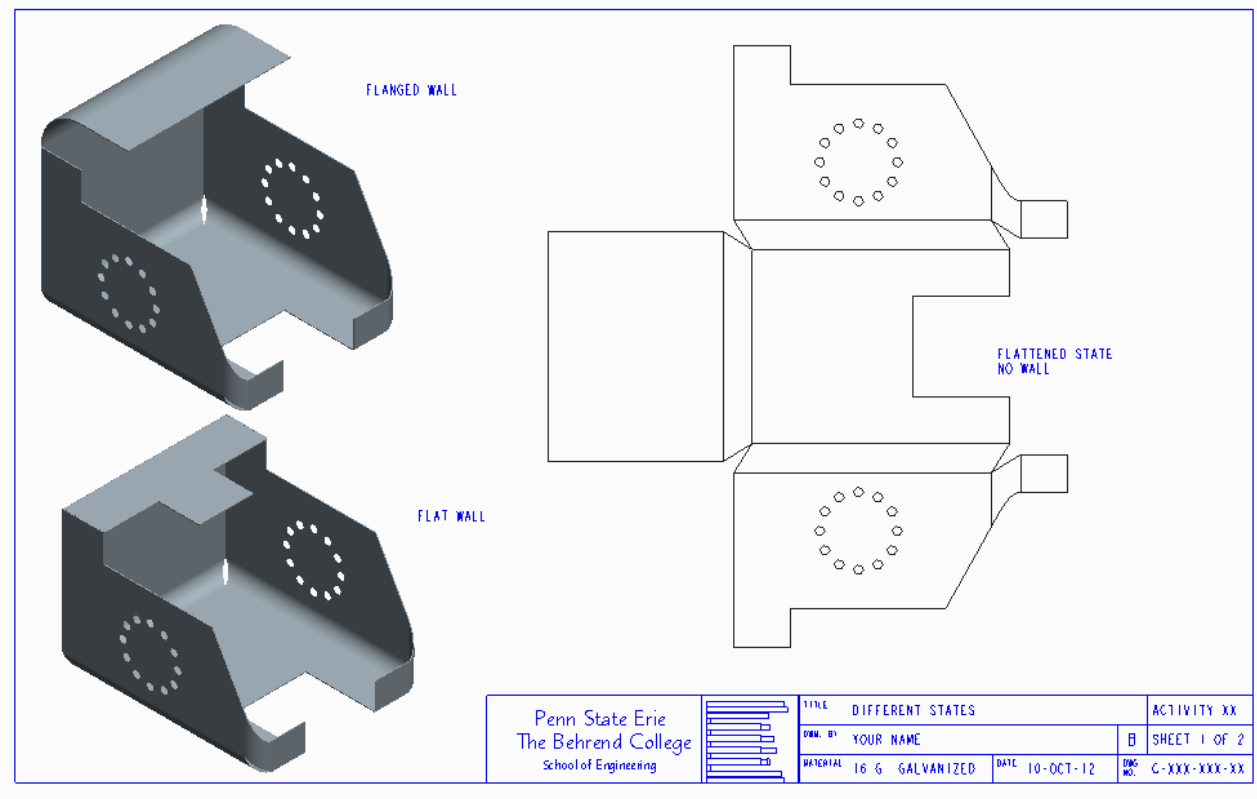


Create a family table that has instances that are flattened (with no flanged or flat wall) and fully formed (both flat and flanged walls). **Make sure you "verify" your instances.** Any changes to geometry will require you to re-verify the instances, otherwise you will get a regeneration warning when trying to save.

If you are unfamiliar with the family table procedure, refer to the Precision LMS software, Sheetmetal design using Creo Parametric 2.0, Bending and Unbending Sheetmetal Models, Creating Flat States.

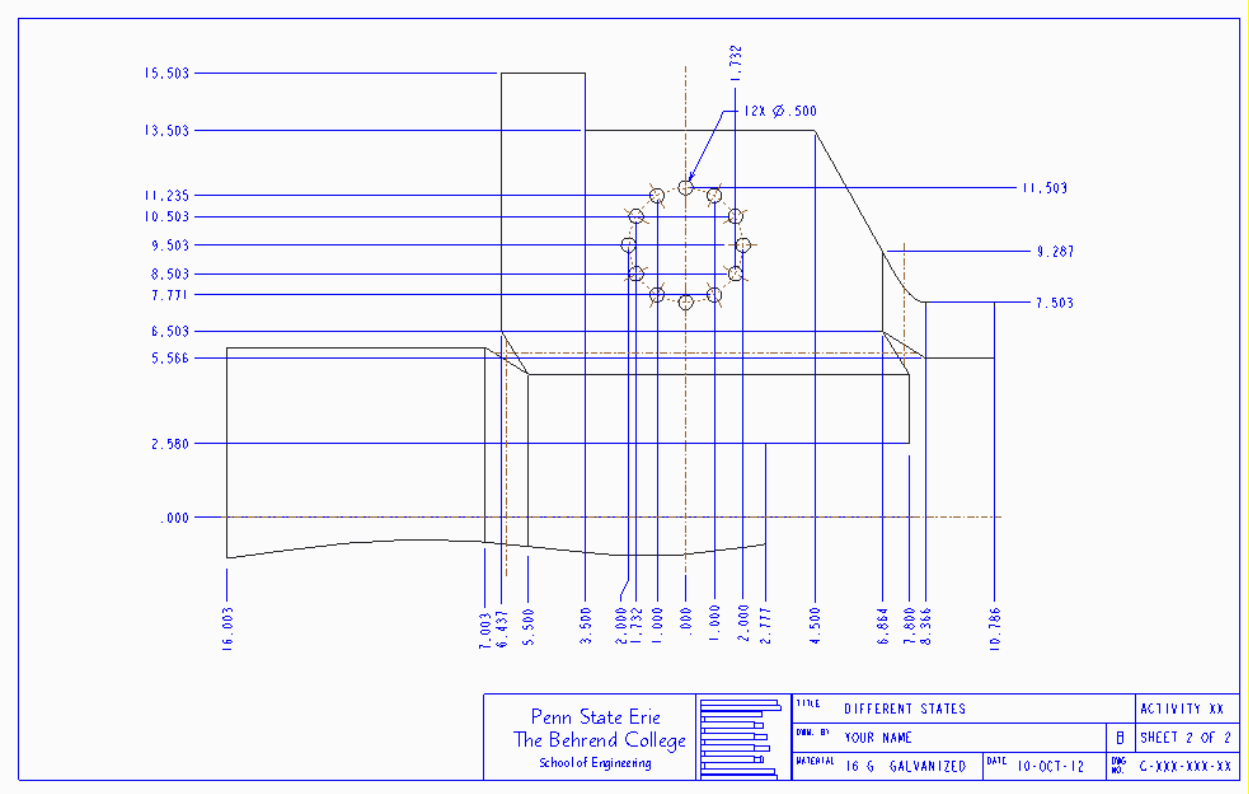


Create the B size drawing as shown below with the labeling. Print on A size paper.



On sheet 2 (see following figure), create a fully dimensioned detail drawing (B size) of the flattened geometry (partial view) using ordinate dimensions, nicely aligned. Tangent lines are displayed as phantom. You will need to create an axis that runs down the center of the part (at the part level) to show the centerline in the drawing if you don't already have one. To create a partial view, in the Drawing View dialog box, select the Visible Area Category and set the View visibility to partial view.

Use Auto Ordinate to dimension the view.



End Level 7