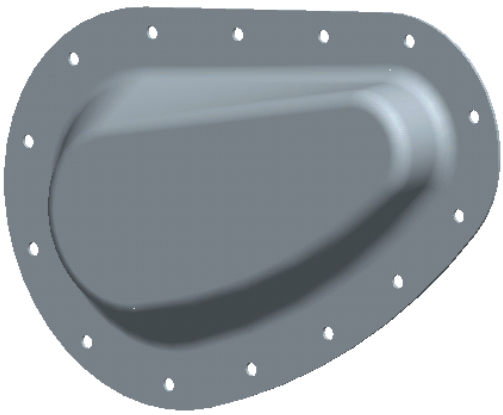
Oil Cover Design
Embedded Datums (Review)
Parallel Blends, Datum Planes (Review)
Composite Curves (New)
Release Creo 2.0

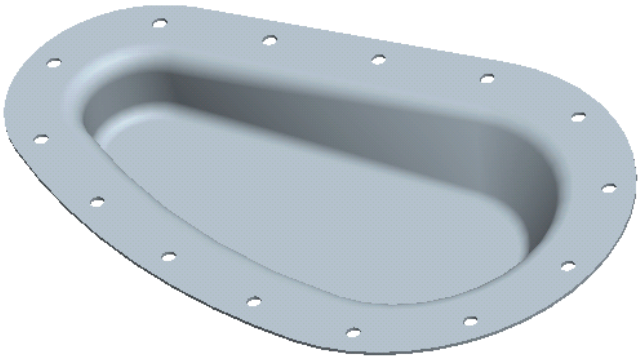
Level 7.0

Read the entire lab before beginning!

Model the Oil Pan and prepare a detail drawing showing a cross section of the part through the axis defining the upper and lower holes as well as fully detailing the holes.

This pan is used in a shallow bath lubrication (Type II) operation for chain drives. This is suitable for chain speeds of 650 to 1500 ft/min as the pan provides a sump for the oil into which the chain will dip continually. (Outside geometry on the left, inside geometry on the right)



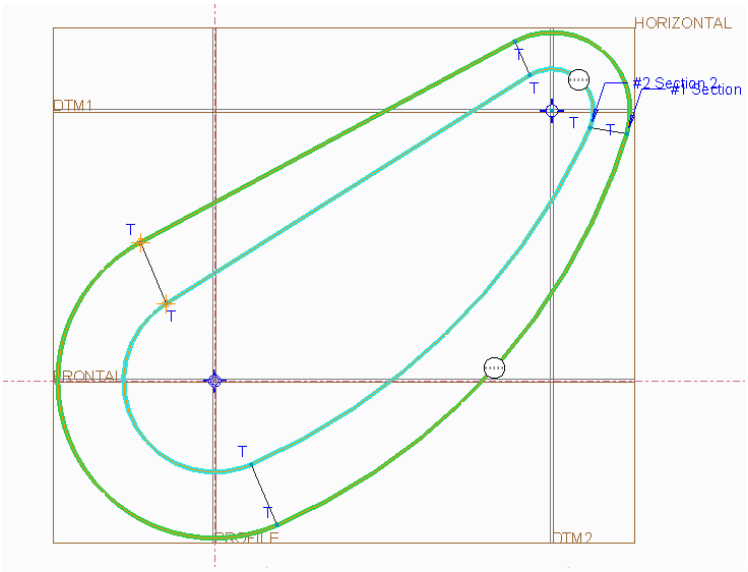


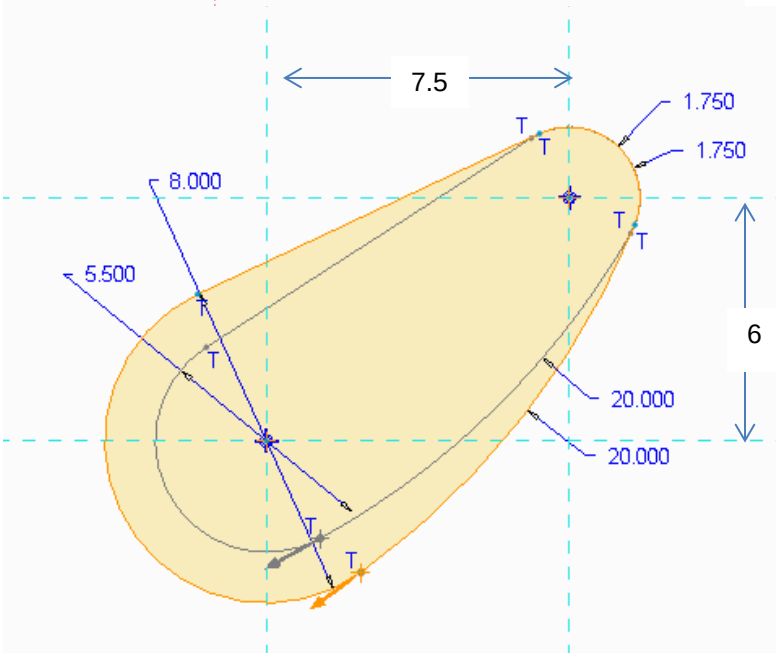
Rotatable PDF file: oil_pan.pdf

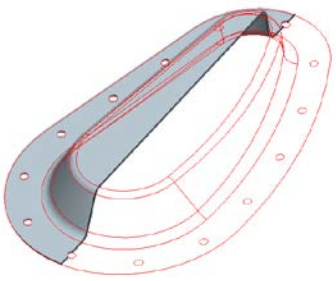
Modeling overview:
This part cannot be created with simple drafted geometry as the design call for the lower lobe to “taper” to a smaller size while maintaining the same size on the upper lobe. This will require the use of a parallel blend.

The part is to be a constant material thickness, requiring the use of the shell feature. A bolting flange must be provided where the location of the bolt holes is at a uniform distance from the edge of the flange. Each hole is to be located (patterned) the same distance along a datum curve that is offset from the outside edge. Additional Datum Planes, Curves, Points and Axis are required to accomplish this task.

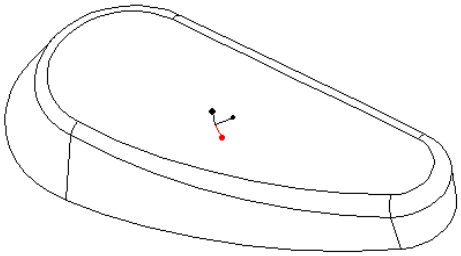
Dimensional information for the “lobes” before any rounds are added: (front view, the 2.00 is a thickness dimension) are shown below.
Hint: Model the upper lobes with different radial values and after the “parallel” blend has been created, modify the radius to be equal.



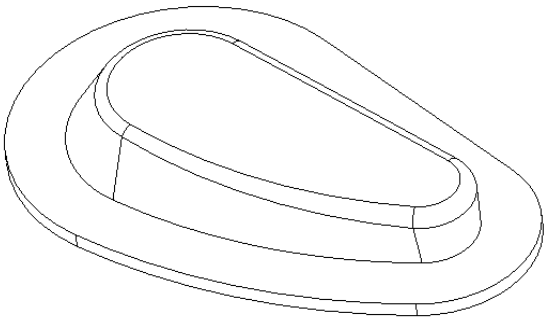




After upper round (R .5) has been added:

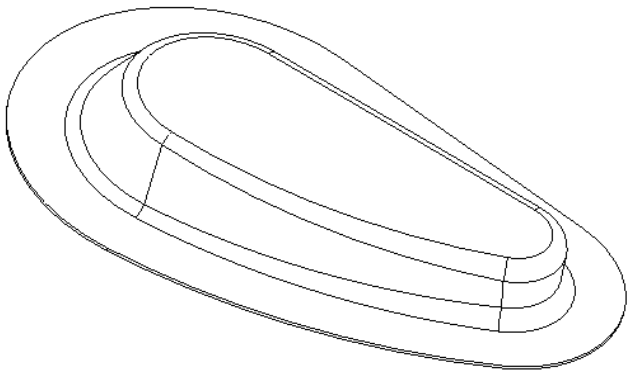


The outside edge of the flange is a constant 2 inches away (use *Offset Edge* function in sketcher) from the edge of the lobes with a thickness of .1. The part is still fully solid at this point. In prior releases of Pro/E, the flange would have to be created oversize (thickness) to allow us to shell the part, as shown below.



This is not the case with Wildfire or Creo. We can shell a surface that is .1 in deep with a shell thickness of .1 and not have the feature fail.

After inclusion of lower round (R .5) and shelling the part to a constant thickness of .1:



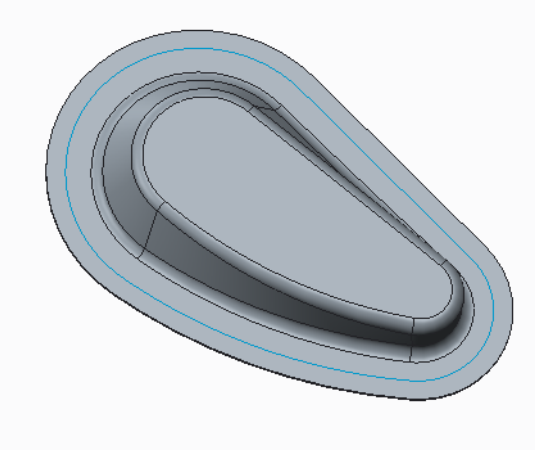
After adding Datum geometry to locate holes: The holes (.375 + 1/32 inch clearance for diameter) will be on a line that is at a constant .75 inch distance away from the outer edge of the flange. The last hole that is created (patterned) should be located along a line (datum) that intersects the two center axis that was used to locate the lobes.

In General, menu picks for:

Datum Curve….
Sketch Tool (Icon) (use offset edge)

Datum Planes….
Datum Plane Tool (Icon) select references etc

Datum Points….
Datum Point Tool (Icon). References are curve and plane

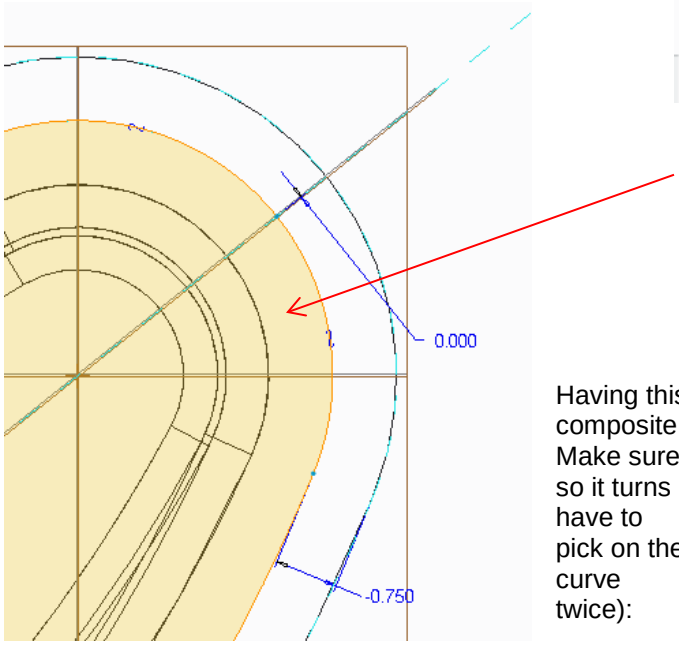


Create the offset edge datum curve first, then create a composite curve as instructed on the next page.

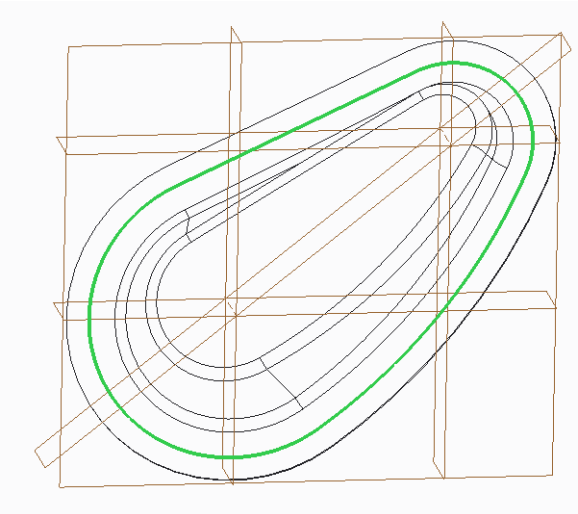
NEW: Composite curve

We will need to place equally spaced datum point on this curve to locate the holes.
You cannot just stick a datum point anywhere on the curve and pattern it around the curve using a ratio, for a lot of reasons.
We really need a “composite” curve that is made up of the individual elements. Not only that, but this composite curve must “start” at a specific location so we can offset points from that curve start point.

First “break” the original curve at the appropriate location by using the “divide” button in sketcher. You will of course need a reference for the locating dimension. This should come off a datum plane that is at the appropriate angle.



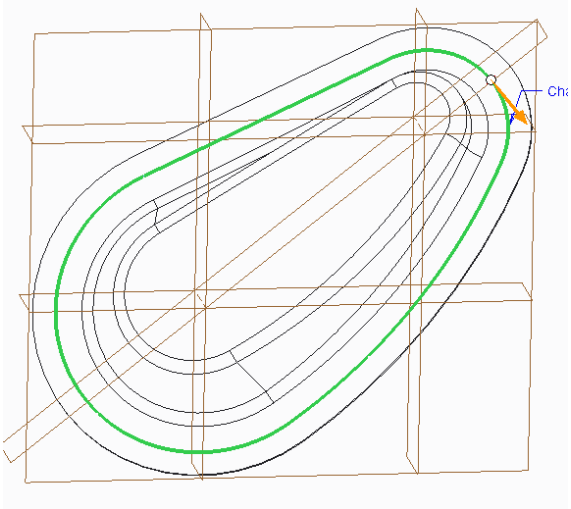
Having this “break” in the curve set, we can now build a new composite curve.
Make sure you are not in sketcher and select the curve set so it turns bright green as shown below (you will probably have to pick on the curve twice):



Select the Copy and then the Paste icon from the Operations region under the Model tab.

If you get a dialog box asking about a sketch plane, you have a problem with the original curve set. Return to the definition of the original curve and “trim” all the ends of the four entities with each other.

Drag the arrow around the curve until it rests at the point where we broke the original as shown.



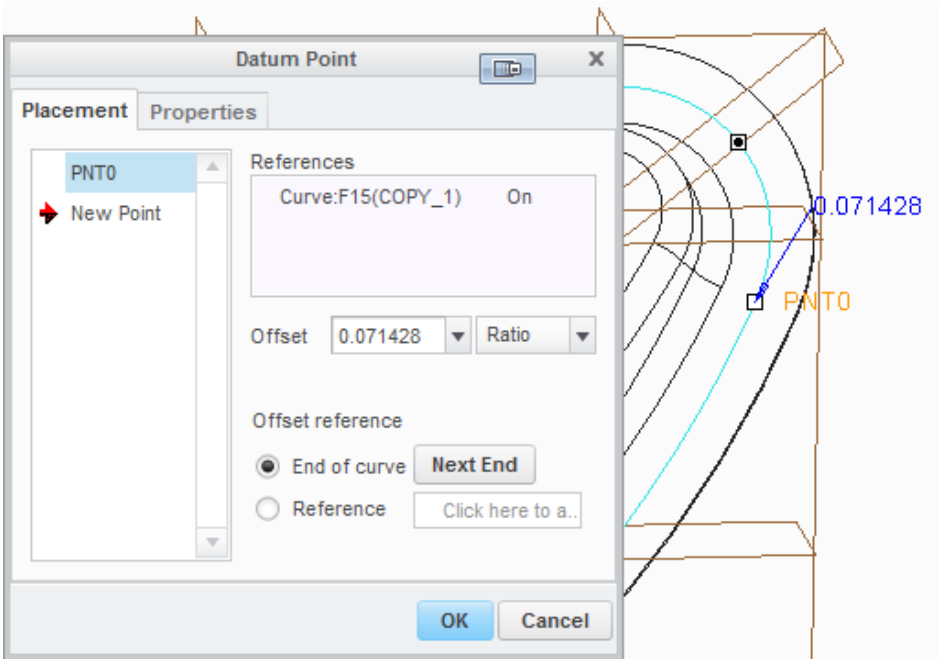
Exit the feature, and hide the original sketch so it is not selected by mistake when placing the points.

With the composite curve selected, create a datum point. Since the curve starts at the correct location, position this point on the curve with an offset of .071428 (1/14) of the curve length.

Make sure this point is located at 1/14 of the way around the entire curve, not just the first segment of the curve. You may have to use the query select (right click) to select the entire curve.

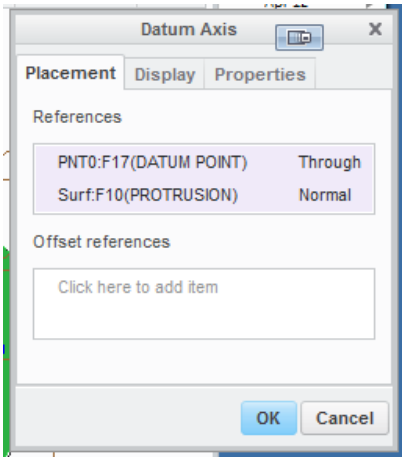
Now pattern this point around the curve (14 copies), using a Dimensional pattern.

Make **sure** your last point is exactly .071428 from the first point. Always **check** your work. Do not assume the system has accomplished the task correctly!



Note that we cannot directly create holes using the datum points for reference, therefore create:

Datum Axis (through the points)
Datum Axis Tool (Icon) using the point and horizontal plane as references.
Pattern this axis using a Reference pattern.



Creating the hole, use the hole wizard (Hole Tool icon)
References are the datum axis (coaxial) and the top surface of the flange of the pan.

Pattern this hole using a Reference pattern. Reference patterns of the axis and hole are required.

Required Documentation:

Create a B size drawing (next page) showing the front view of the part in the "normal" orientation (No Hidden, Tangent lines as Phantom), with all the holes dimensioned. Use the Standard B size title box for Penn State Erie.

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The scale is .375 for all views. This is necessary to allow checking of the geometry by overlaying an answer key, as most dimensions are **not shown**.

Note that the dimensions have been switched to symbolic representation so as to not allow the student to simply use linear dimensions to place the holes. Your dimensions will of course be numeric values and ***NOT*** symbolics.

All dimensions are taken from 0,0. that we will define as the center of the **larger (lower) lobe**. Make sure you also locate the center of the "upper" lobe.

Your drawing will have the holes located with actual ordinate dimensions shown to 2 places, no tolerance required. The linear dimension of the overall height (2.10) in the sectional views should also be set to two places. The hole should be specified to four places.

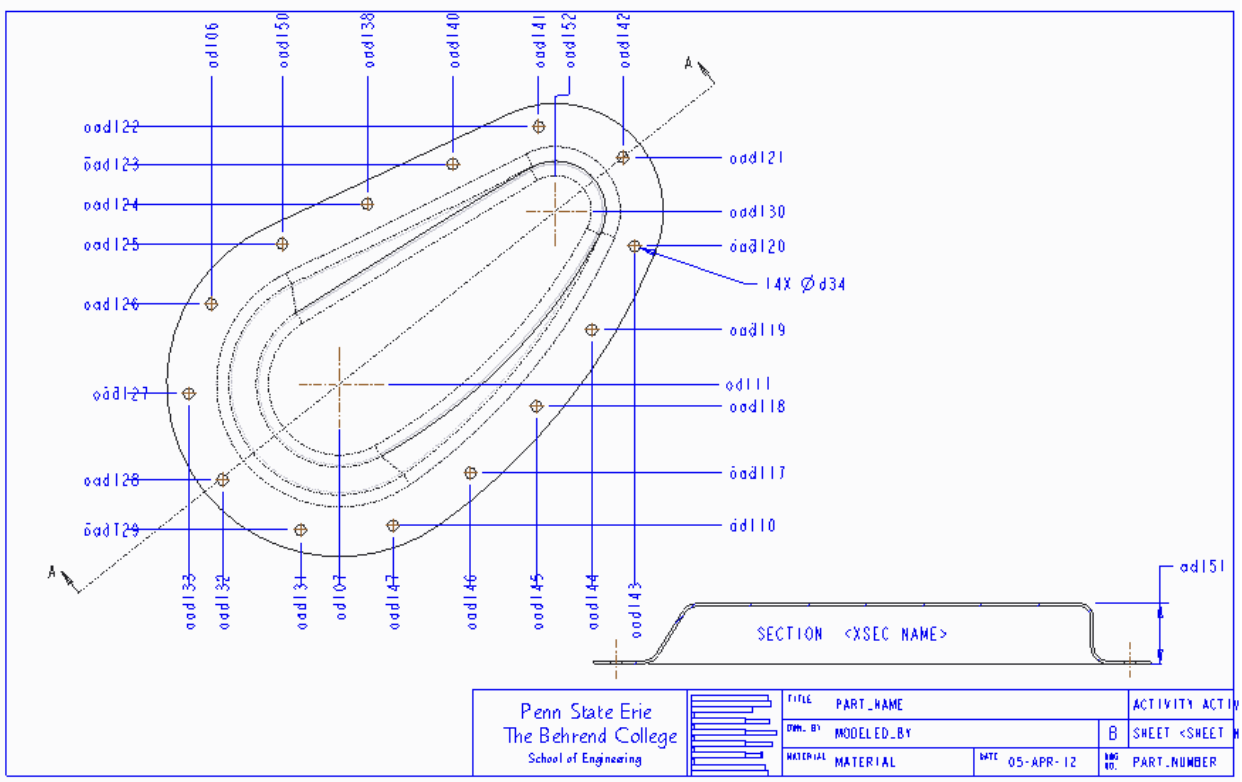
Create baseline Ordinate horizontal and vertical dimensions from the center of the larger lobe. Add additional Ordinate dimensions by using an Attachment Type of "On Entity". Make sure you select the .000 value and not the center line other wise you will be adding another .000.

To align dimensions, select them then Align Dimensions. You might have to jog select dimensions for readability.

Cross sections are easily created at the part level using the View Manager. Create the sectional view by creating a General View that is oriented by Geometry References, not by Name.

The sectional view should also be shown with No Hidden lines. You might have to have the label SECTION A-A created as a note, depending on how you generated the cross sectional view, as we do not want a scale on the drawing.

Print this on **A** size paper (set the sheet size to A when plotting, full plot). Title is OIL PAN, Material is 16 GA STEEL.



Select the Tools icon and pick Feature List and **print** this information.

The instructor will be looking (checking) for:

Incorrect pan geometry	Incorrect location of holes	Dimensions not aligned
Missing dimensions	Incorrect sectional view	Two and four place decimals
Printed on A size paper	B size title block	

Note: Level 7 continues in Lab 2b !!