
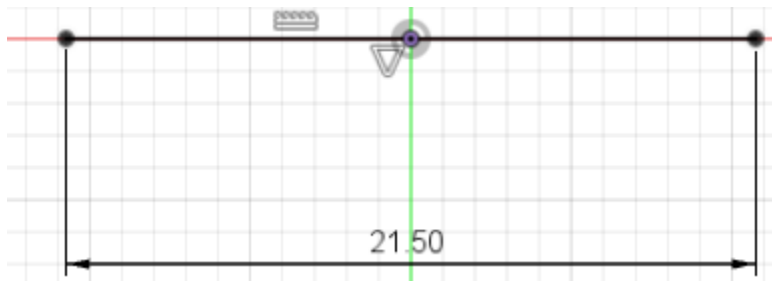


## **Lab Assignment #3: Multi-Step Part Modeling**

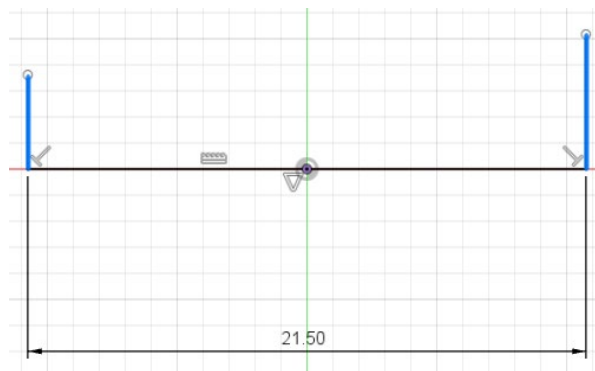
**ME 170**

### **Part I. Create a new lens part**

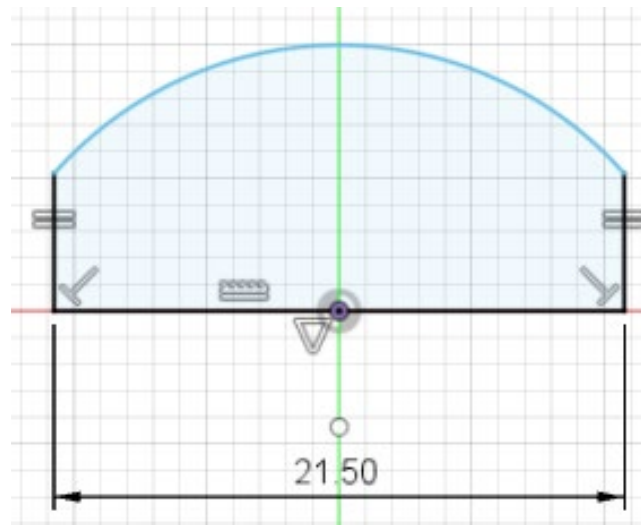
- 1) Open a new part and save it
  - a) Start Fusion and open the Data Panel
  - b) Navigate to your ME170 project and create a new folder named “Lab 3”
  - c) Save a new part file with mm units to this folder named “Part 6”
- 2) Begin the part
  - a) Create a new sketch on the XZ plane
    - i) If you do not remember how to differentiate the planes, check Lab 2
  - b) Sketch a horizontal line NOT beginning at the origin and dimension it with a length of 21.5 units
  - c) Under the Constraints tab, select the Midpoint constraint  and then select the line and the origin



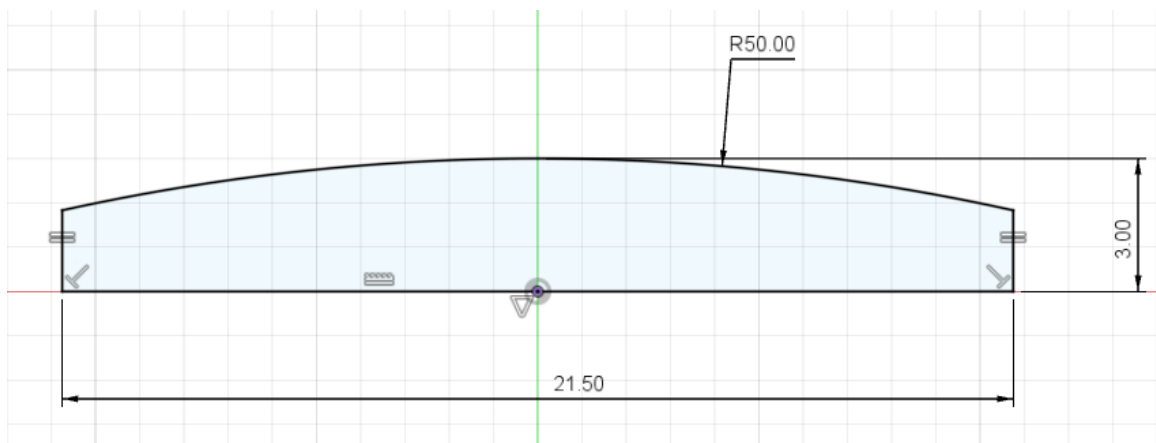
- d) Create two short, vertical lines at the ends of the horizontal line with no dimension



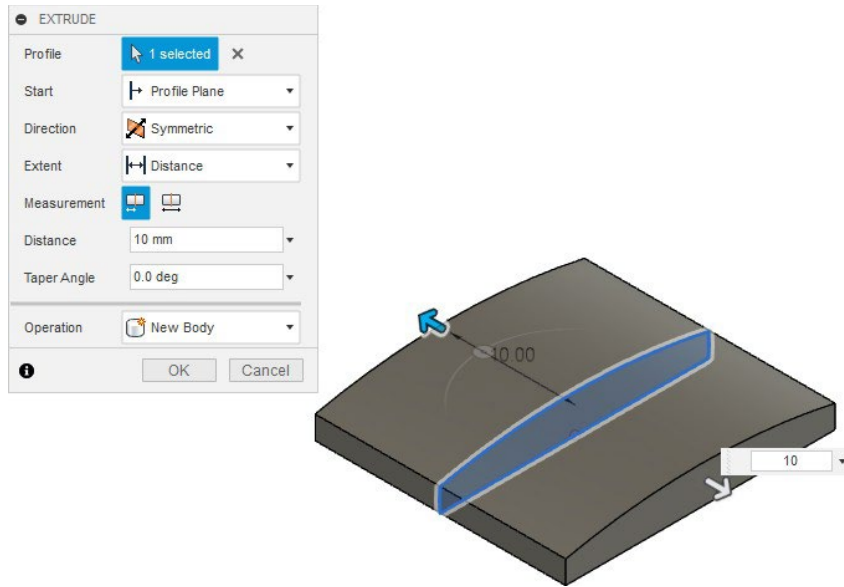
- e) Under the Constraints tab, select the Equal constraint and then select both vertical lines to give them equal length
  - i) Make sure not to select the parallel constraint
  - ii) If an error messages appears saying the sketch is over-constrained, hit Esc on the keyboard, reselect the Equal constraint, and select the lines in the opposite order
- f) From Create > Arc select the 3-Point Arc tool. Select the ends of the two vertical lines as the start and end point of the arc, then select a third point above and between the ends to complete the arc



- g) Use the Sketch Dimension tool to dimension the height of the shape as 3
  - i) To dimension to the top of the arc, before selecting the arc, right-click and select “Pick Circle/Arc Tangent”
- h) Dimension the arc itself to have radius 50




- i) Select Finish Sketch and Extrude the profile. Under “Direction” change One-Side to Symmetric, and enter 10 as the distance (due to symmetry the total depth becomes 20)

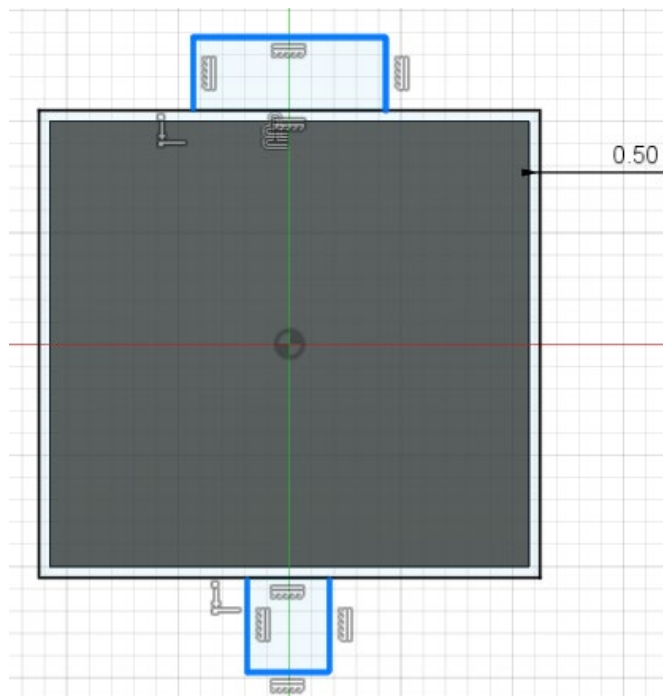



3) Add another element to the part

- a) Create a sketch on the bottom (flat) face of the newly extruded part


- b) Select the Offset tool , choose the rectangular border, and enter 0.5 mm as the offset distance

- c) Sketch one smaller rectangle directly on top and one directly below this rectangle



d) Use the Symmetric Constraint  to constrain the vertical sides of the new rectangles to be symmetric about the vertical axis

i) Select the tool, then select both vertical sides of one rectangle, then select the Y-axis from the Origin folder under the Browser on the left. Repeat for the other rectangle

e) Use the Trim tool  under the Modify tab to remove the overlapping sides of the smaller rectangles with the larger, offset rectangle (see pictures below)

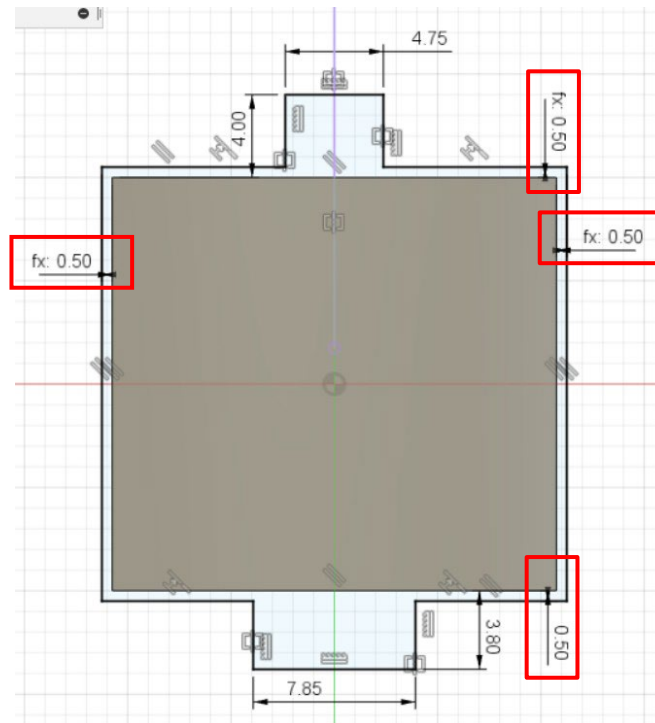
i) Make sure to Trim twice as the Trim tool only removes one layer of lines at a time

f) After trimming the overlapping part, you will notice that the “offset” constraint is released. You have to add them back to make your sketch full-constrained.

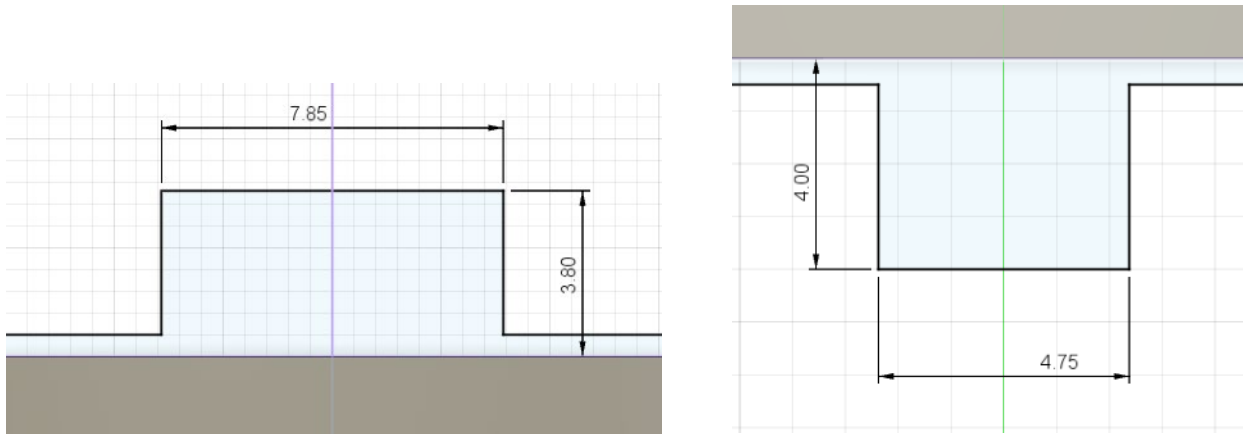
i) Add a dimension of 0.5mm to one of the edges.

ii) For the other three edges, when adding dimension, click dimension 0.5mm (from previous step) instead of typing manually. This could make all four dimension linked and same to each other.

iii) You should see “fx” on your dimension if you do it correctly. If you want to change the offset distance, you only have to modify one dimension instead of four.

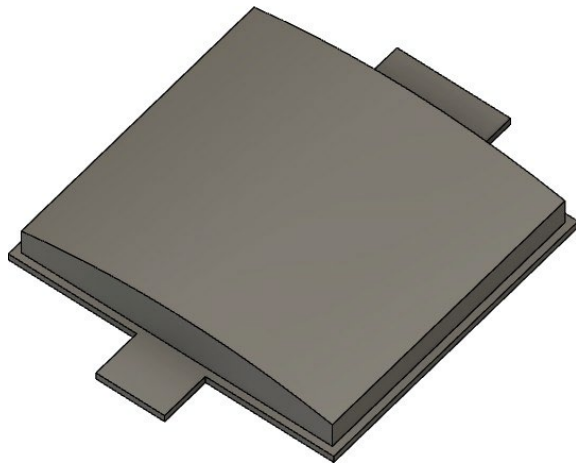


g) Give the two rectangles the following dimensions:



h) Select “Finish Sketch” and Extrude the profile 0.5 mm

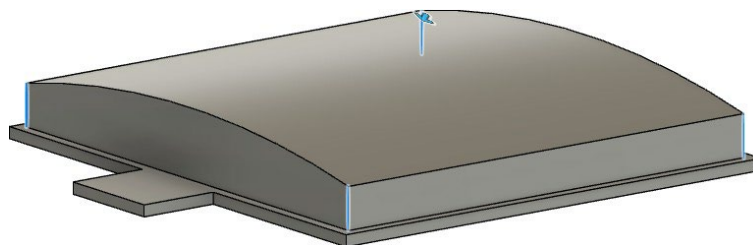
i) Make sure to select the entire profile to extrude, this may take multiple selections



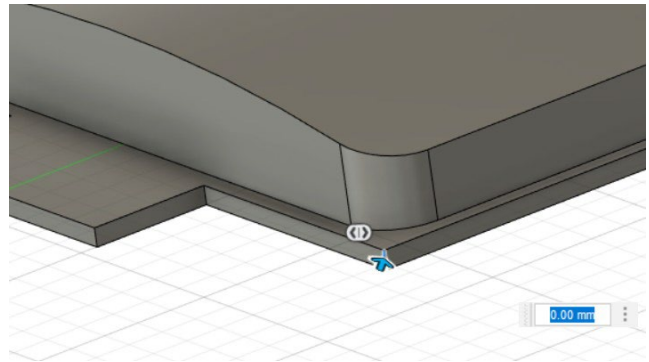
4) Add fillets to the part

a) Under the Modify tab, select the Fillet tool

b) Select the 4 large vertical edges of the part and enter 1.5 mm for the fillet radius



- c) Repeat the process for the 4 smaller vertical edges directly below existing fillets, this time with a radius of 2 mm



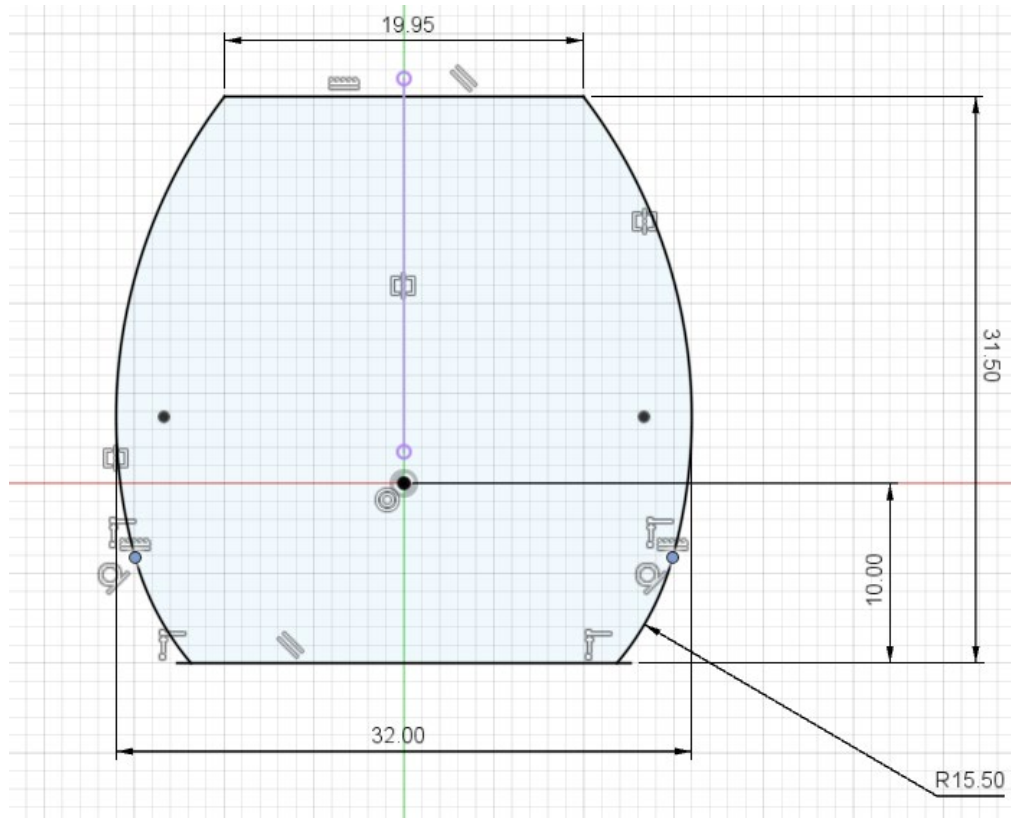
- d) On the bottom smaller rectangular addition, fillet both vertical edges 2.375 mm (half the length of the rectangle)
- e) On the larger rectangular addition, fillet both vertical edges 2 mm



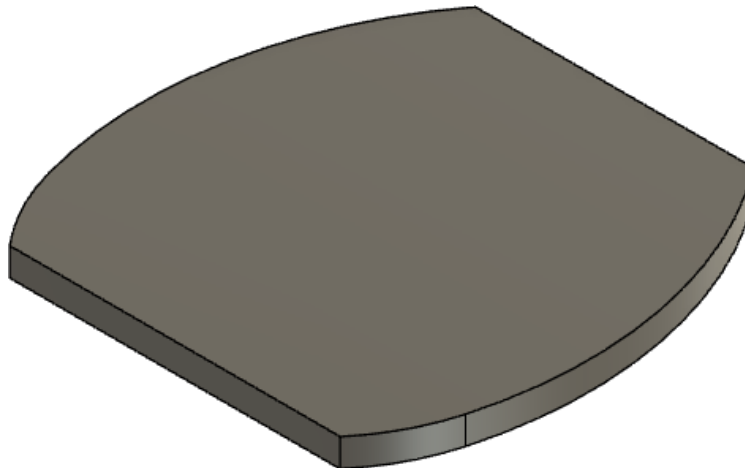
- 5) Make sure to save the part and then close it

## Part II. Create a new battery cover part

- 1) Open a new part and save it
  - a) If a new part file did not automatically open, select File > New Design
  - b) Save the new part file with mm units to your Lab 3 folder with name “Part 7”
- 2) Create the part
  - a) Create a new sketch on the XY plane
  - b) Using the same process as Part 4 in Lab 2, create the following sketch:
    - i) General approach: First create 3 arcs (bottom, left, right) with tangent constraints , then create 2 horizontal lines, then trim dimension, then trim (see Lab 2 for specifics)



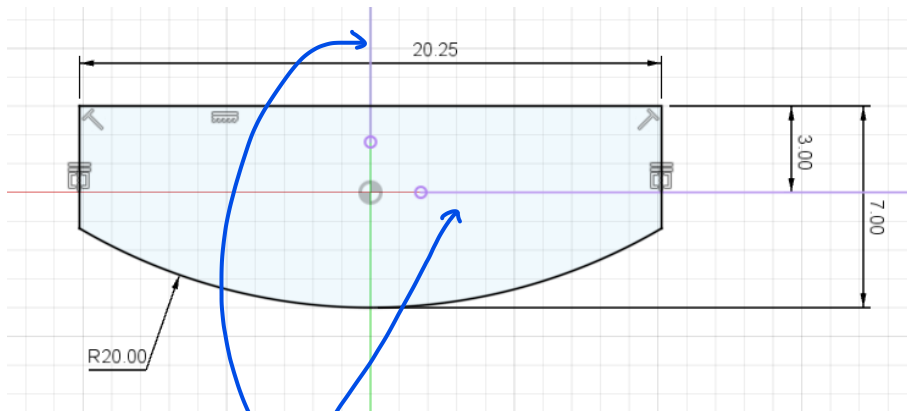
c) Press Finish Sketch and Extrude the profile for 2 mm.



3) Make sure to save the part and then close it

### Part III. Create a new button part

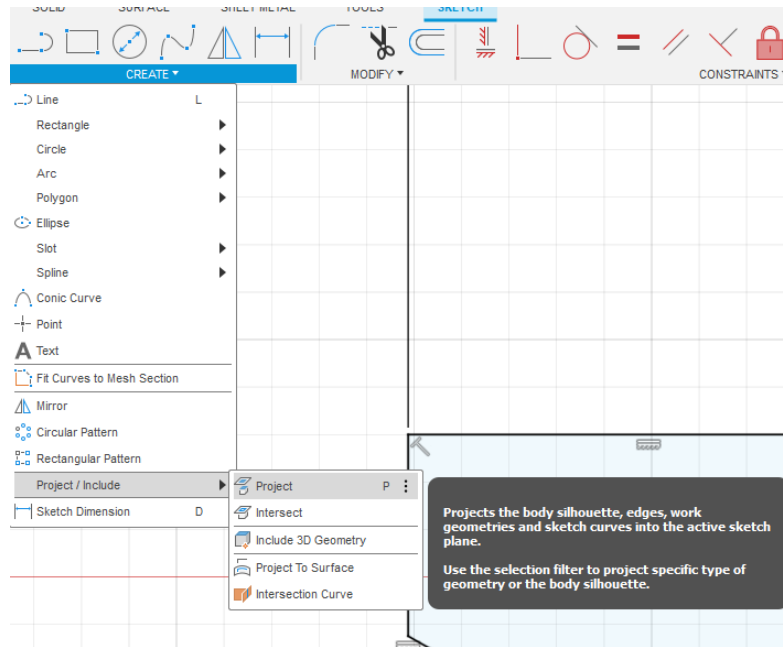
- 1) Open a new part and save it
  - a) If a new part file did not automatically open, select File > New Design
  - b) Save the new part file with mm units to your Lab 3 folder with name “Part 8”
- 2) Begin the part
  - a) Create a new sketch on the YZ plane
  - b) Sketch the following profile:



- i) This profile is very similar to Part I of this assignment. A few key distinctions:
  - (1) The Midpoint constraint can be replaced by a Symmetric constraint about the vertical axis on the smaller vertical lines
  - (2) Use the **Project tool** (analogous to tracing) under the **Create dropdown** on the two **main axes** to easily dimension to them. With the tool selected, select the geometry to trace onto the sketch (in this case the axes) and then press OK

投影 Z, Y 轴

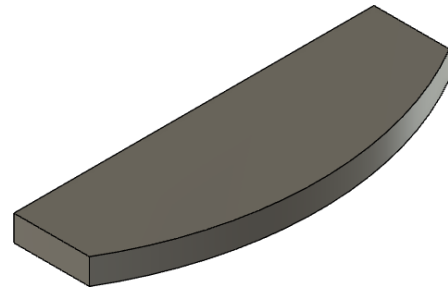


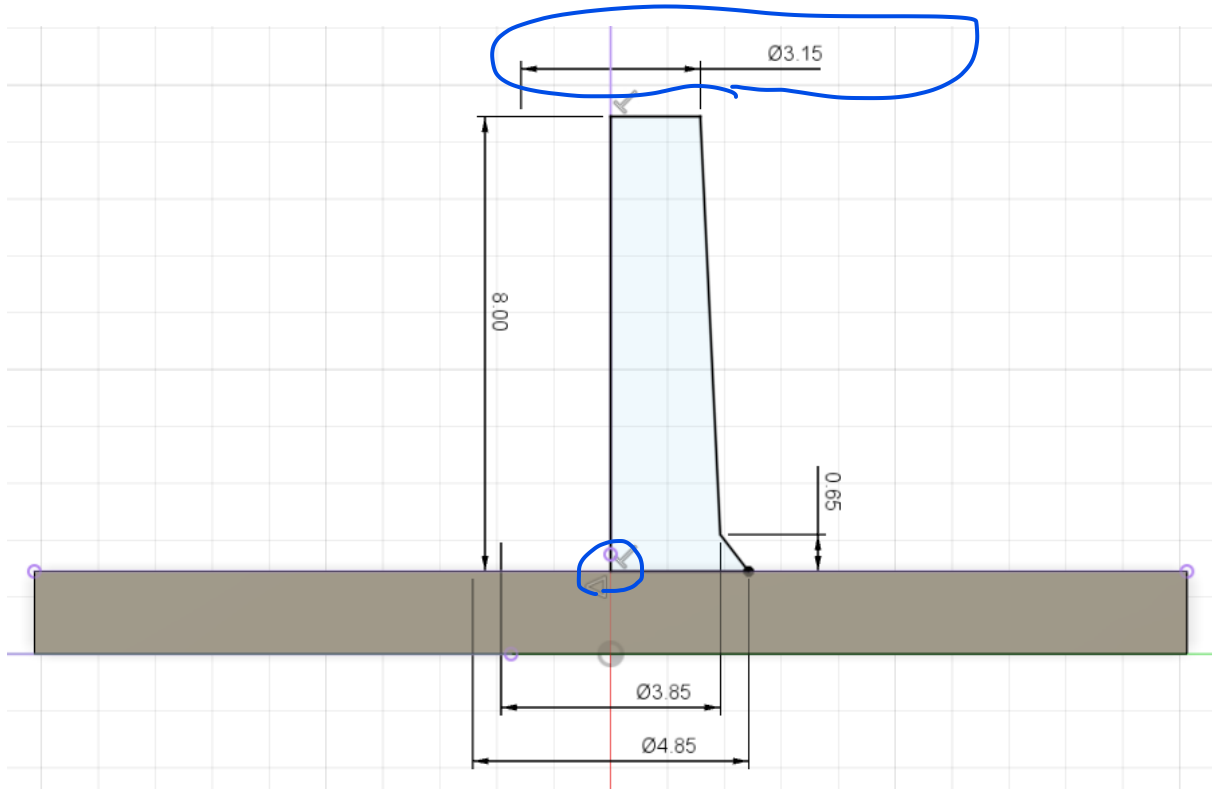


c) Press “Finish Sketch” and Extrude the profile for 1.45 mm

3) Add another feature

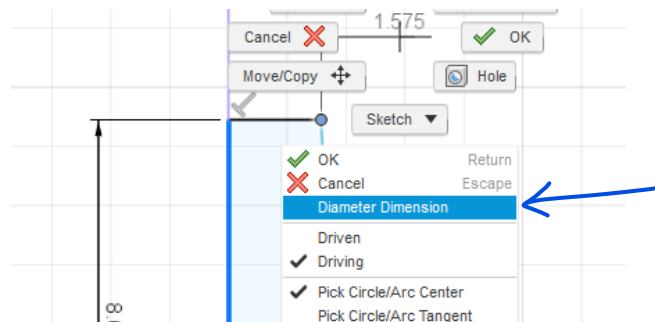
- a) Create a new sketch on the XY plane
- b) Sketch the following profile:





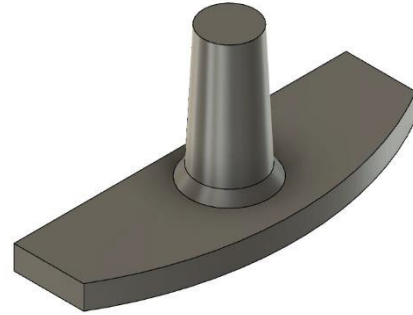
i) A few suggestions:

- (1) The view of the **XY** plane has been rotated using the View Cube in the top right of the screen
- (2) Project the principal axes and rectangular outline before starting
- (3) The dimensions to the vertical (X) axis are diameter dimensions because this sketch will be revolved. To create a diameter dimension, choose the point and the axis of rotation, then right-click, select “Diameter Dimension”, and left-click to place it



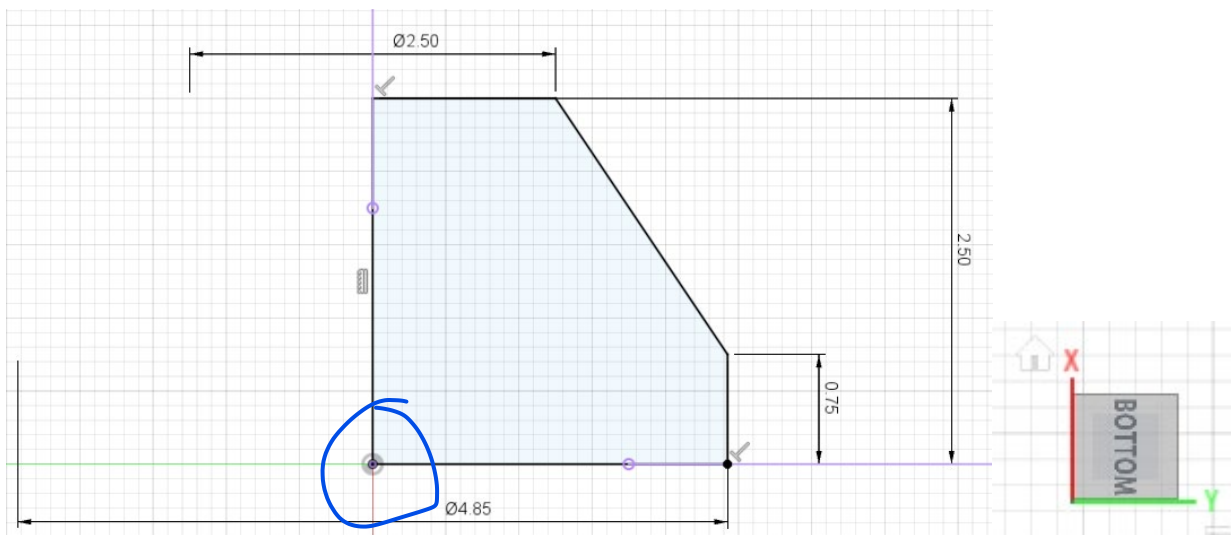
- (4) Do not forget to draw the bottom border of the profile

- c) Select Finish Sketch and select the Revolve tool under the Create tab. Make sure the entire profile is selected, and choose the X-axis as the axis of revolution



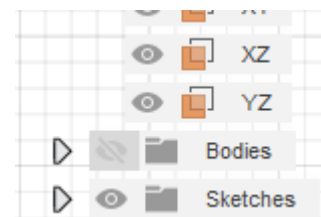
4) Add another feature

- a) Create a new sketch on the XY plane  
b) Sketch the following profile when the View Cube is set as follows:

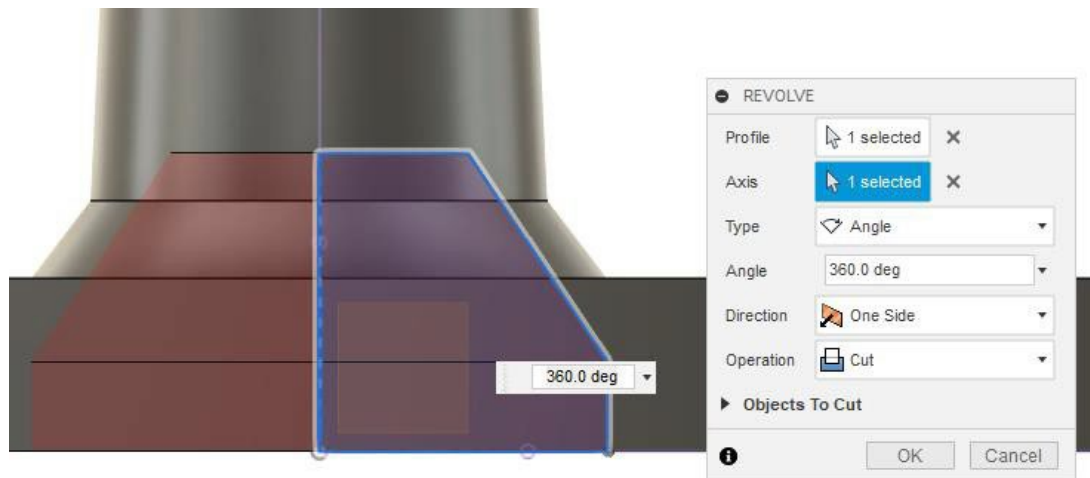


i) A few notes:

- (1) To hide the previously made features for a cleaner sketch plane, select the eye button next to the Bodies tab in the Browser on the left side of the screen  
(2) Make sure to project the principal axes and make sure your sketch is oriented the same way as the given sketch (using the View Cube)



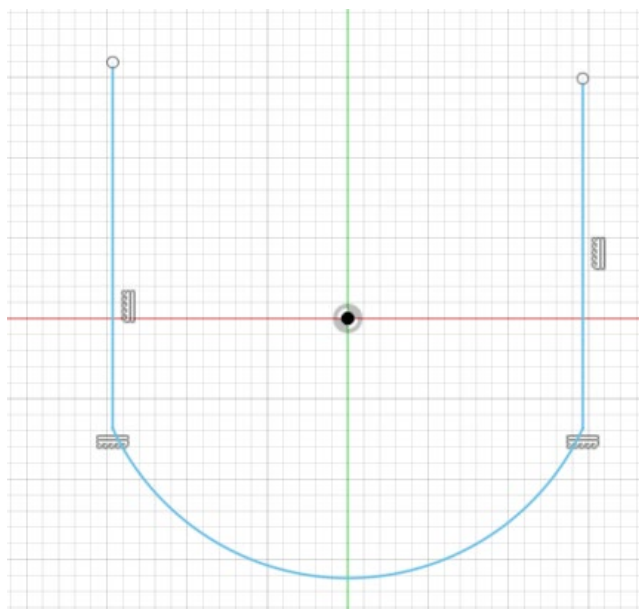
- c) Select Finish Sketch and then the Revolve tool. Make sure the full profile is being revolved and choose the X-axis as the axis of revolution. Make sure the “Operation” option is set to Cut and press OK to complete the revolve



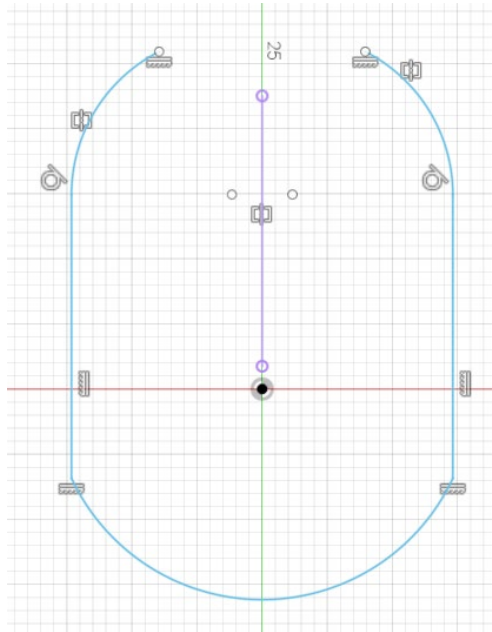
5) Make sure to save the part and then close it

#### Part IV. Create the circuit board part

- 1) Open a new part and save it
  - a) If a new part file did not automatically open, select File > New Design
  - b) Save the new part file with mm units to your Lab 3 folder with name “Part 9”
- 2) Create the first sketch
  - a) Create a new sketch on the XY plane
  - b) Sketch a Center Arc centered at the origin, then use a Horizontal/Vertical Constraint to make the endpoints set to the same height
  - c) Add vertical lines to the ends of the arc



- d) Create a Tangent Arc at the end of each of the vertical lines, then add a Symmetric Constraint so that the arcs are symmetric about the central (Y) axis, and a Horizontal/Vertical Constraint to the end points of the arcs

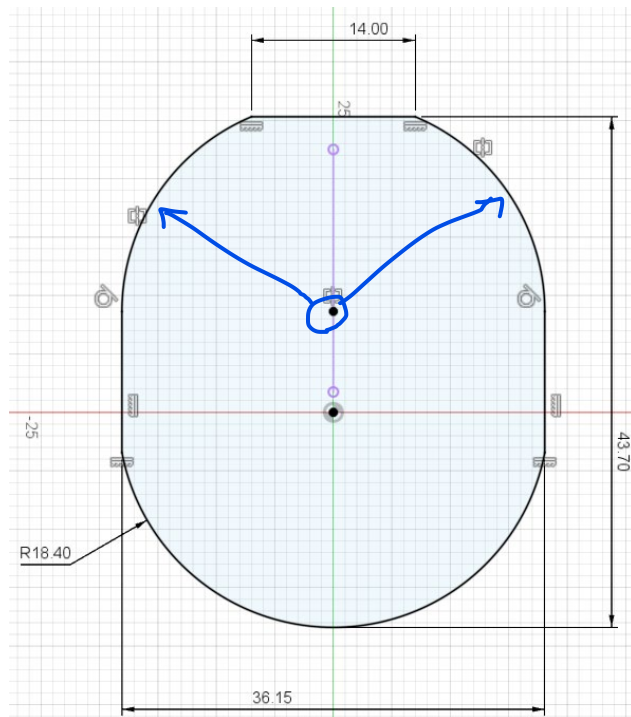


- e) Connect the ends of the arcs with a horizontal line, then add a Coincident Constraint

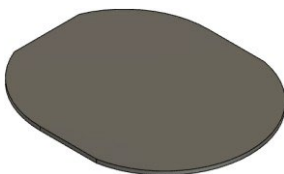


between one of the arc's center point and the Y axis

- f) Dimension the sketch as follows:



- g) Select Finish Sketch and Extrude the profile 0.76 mm



- 3) Save and close the file.

## **Part V. Submission Requirements**

- a. Make sure all your files are saved in the correct folder and that your ME170 folder is shared with both your TA.
- b. Go to ME170 Blackboard website and the CAD LAB Assignments content area. Click directly on the “CAD LAB 3” assignment title and use the “Write Submission” button to type “Ready for Grading”. As before, please do not go back and change your files in the Fusion Lab folder.