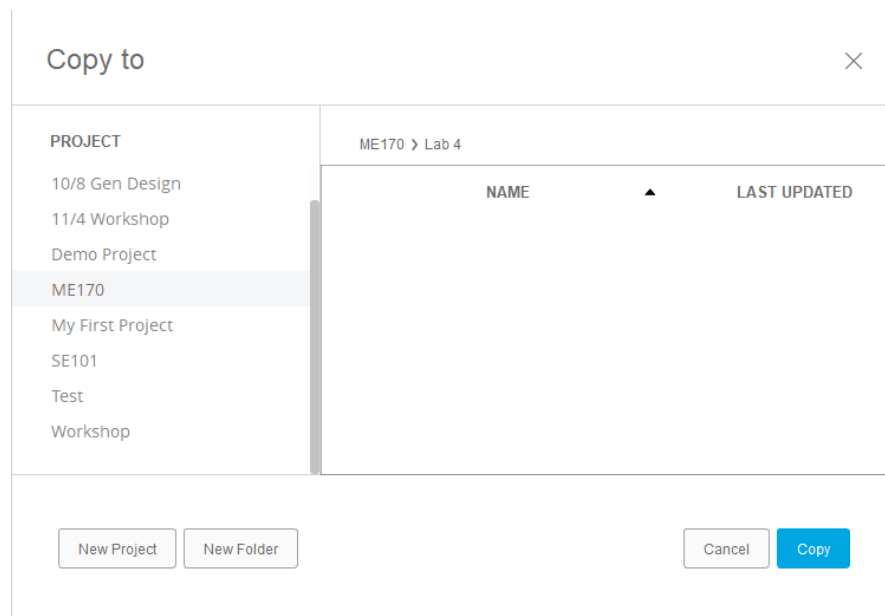


Lab Assignment #4: Loft & Sweep Features

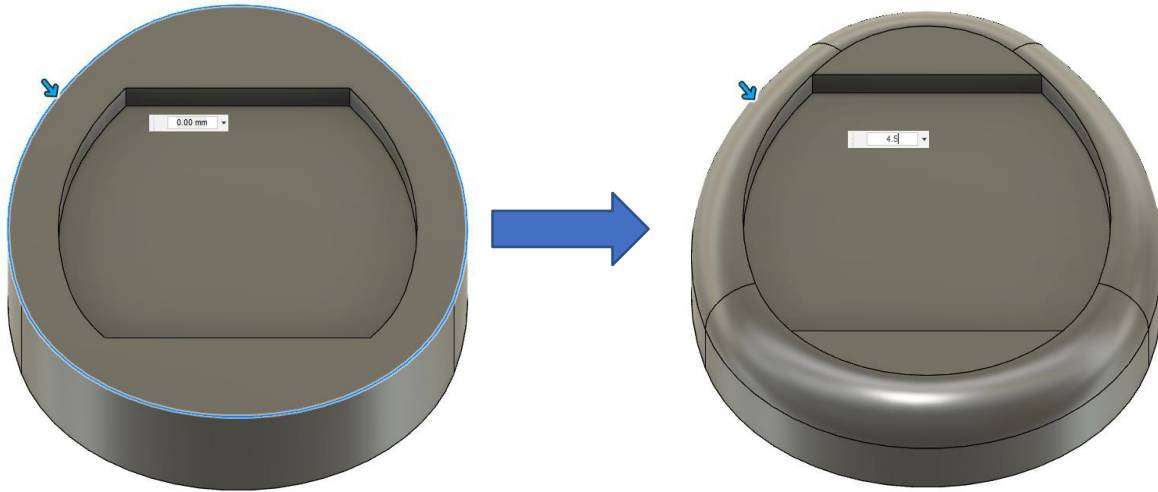
ME 170

Part I. Modify Part 4

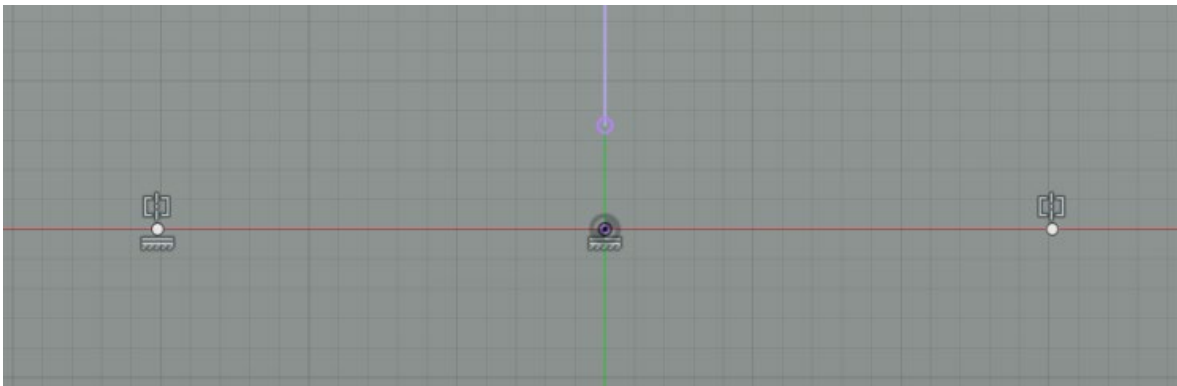
- 1) Create a new folder
 - a) Start Fusion and open the Data Panel
 - b) Navigate to your ME170 project and create a new folder named “Lab 4”
 - c) In your Lab 2 folder, right-click on Part 4, then select Copy
 - d) In the pop-up, set the Copy destination as your Lab 4 folder, and press Copy



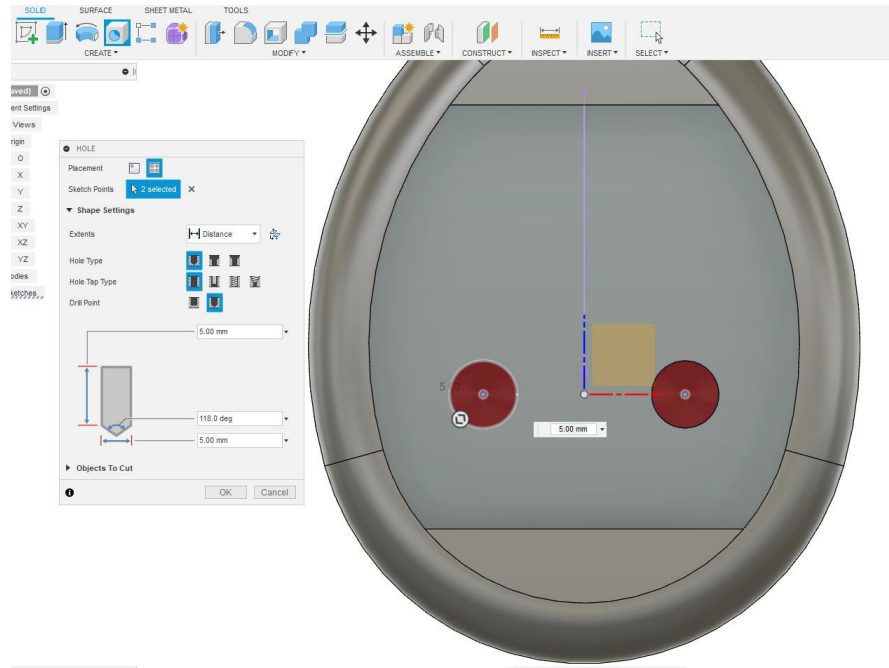
- e) Repeat this process for Part 5, 7, and 9
 - f) Double-click on Part 4 to open it
- 2) Add features to the part
 - a) Under the “Modify” tab, select the Fillet tool
 - b) Select the outer edge on the front face of the part and enter 4.5 mm as the fillet radius



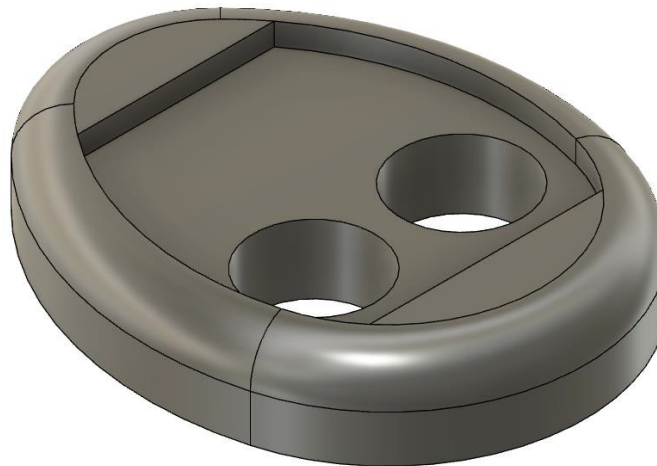
- c) Start a new sketch on the indented face
- d) Under the “Create” tab, select the Point tool and use it to place 2 points, one to the right of the origin and one to the left
- e) Use the Horizontal/Vertical Constraint on one point and the origin to make sure it lies on the horizontal axis
- f) Use the Symmetric Constraint to make the 2 points symmetric about the vertical (Z) axis



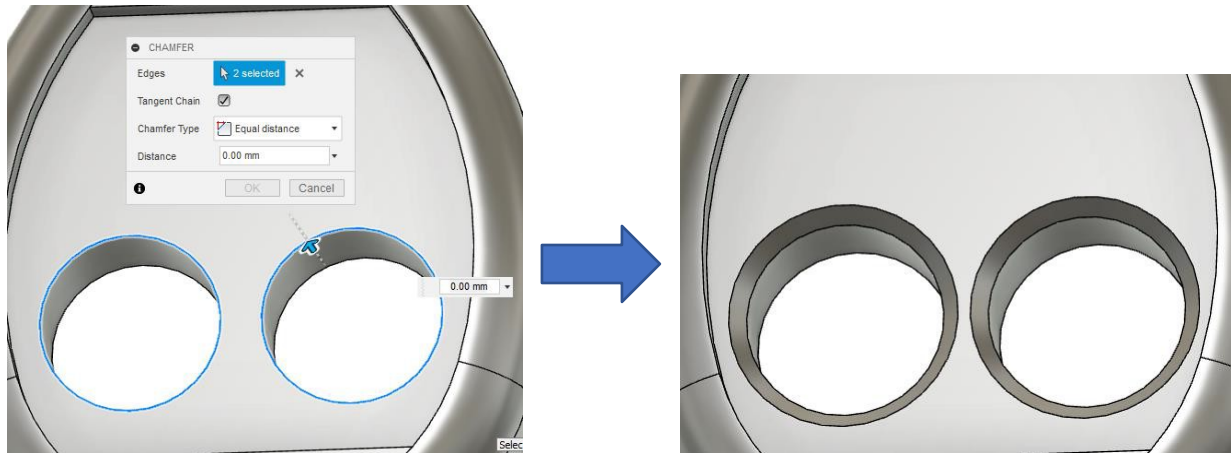
- g) Dimension the points to be 15 mm apart and press Finish Sketch
- h) Under the “Create” tab, select the Hole tool, then select both points you just made



- i) Enter 12.25 as the diameter of the hole and under “Extents” change Distance to All. Then press OK to create the holes



- j) Under the Modify dropdown, select the Chamfer tool. Using it, select the top edges of the holes, enter 1 mm as the distance, and press OK

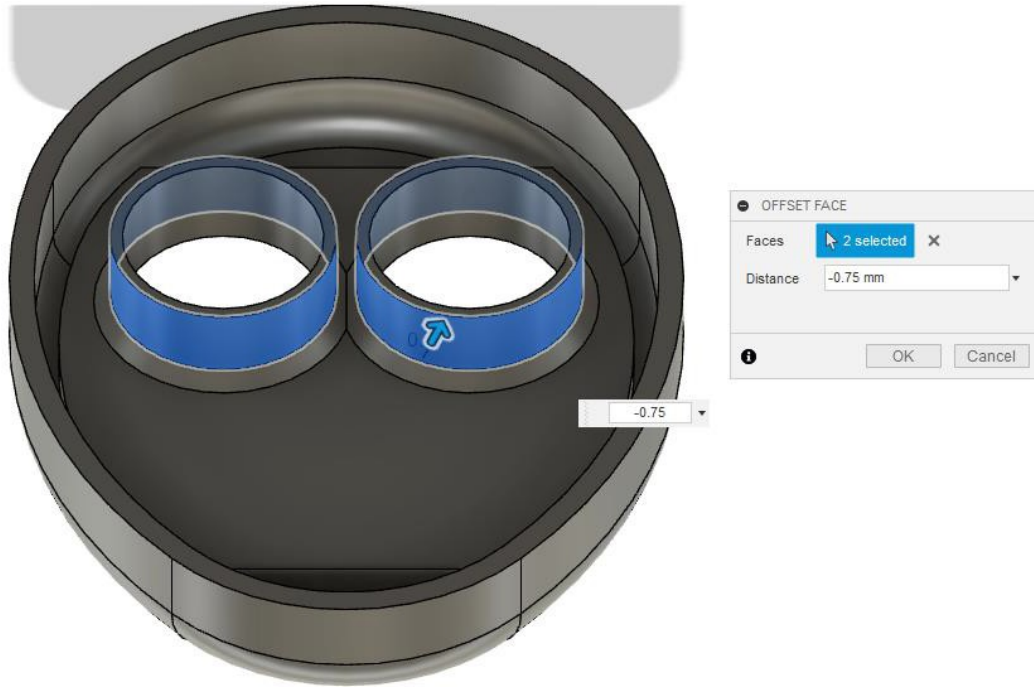


3) Hollow out the part and add more features

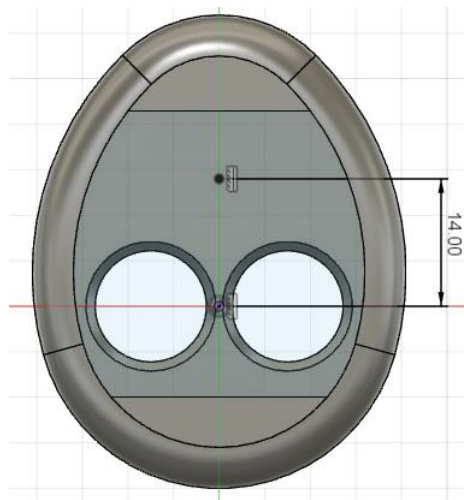
- a) Under the “Modify” tab, select the Shell  tool. Click on the bottom face of the part and enter 1.5 mm as the shell thickness.



- b) Under the “Modify” dropdown, select the Offset Face tool, select the outside of both holes, type -0.75 as the dimension, and press OK



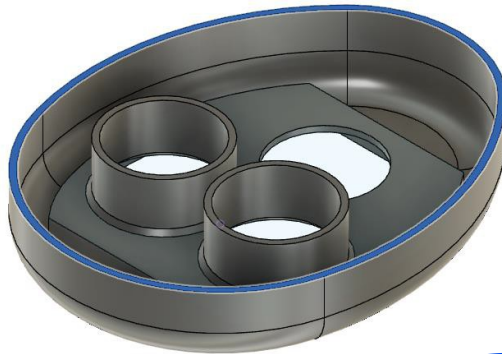
- c) Start a new sketch on the indented face on the top side of the part
- d) Create a point and dimension it so that it is 14 mm above the origin. Then add a Horizontal/Vertical constraint to the point and the origin so the point is directly above it.



- e) Press Finish Sketch and use the Hole tool to make a hole at that point with diameter 12.5 mm, setting the “Extents” to All.



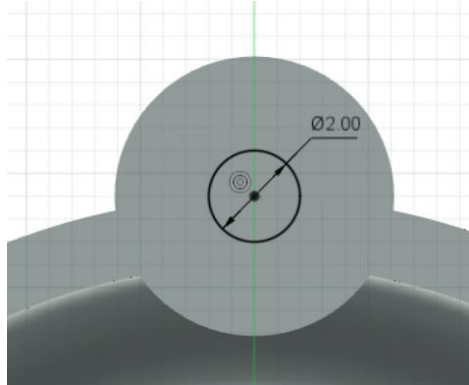
- f) Start a new sketch on the outer ring of the bottom of the part.



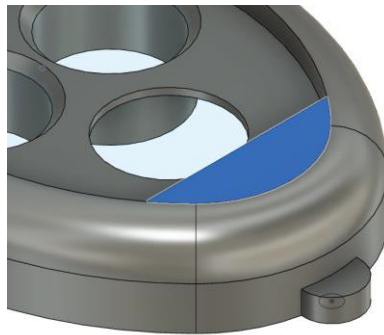
- g) Create a circle with diameter 6.13 mm at the top point of the smaller arc
- h) Press Finish Sketch and Extrude the circle (make sure to select the whole profile, it may take multiple clicks) for -2.5 mm. Set the “Operation” as Join and press OK.



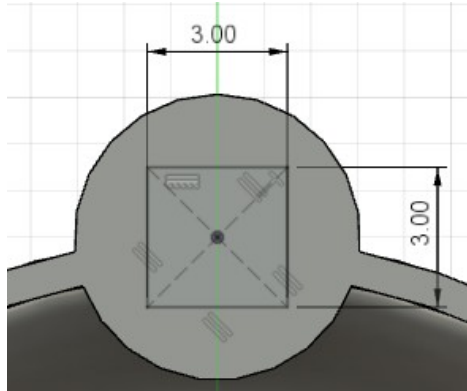
- 4) Add a lofted cut
- a) Start a new sketch on the outer ring of the bottom of the part
 - b) Create a circle with the same center as the circle you just extruded, giving this one a diameter of 2 mm. Then press Finish Sketch



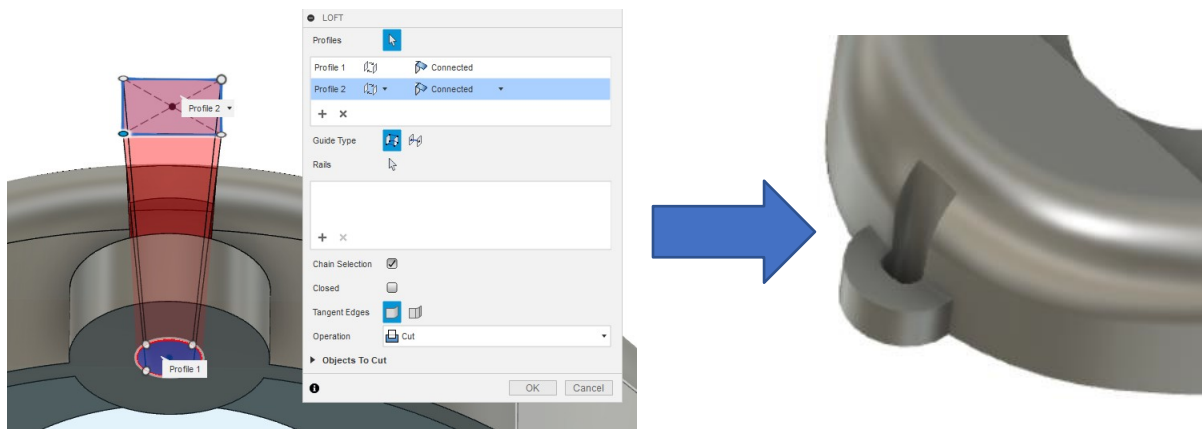
- c) Start a new sketch on this face:



- d) Under the Create dropdown, select Project/Include and then the Project tool (hotkey p). Use it by clicking on the center point of the circle in the other sketch, then pressing OK to include that center point in this sketch.
- e) Use the Center Rectangle tool (Create > Rectangle > Center Rectangle) to sketch a rectangle centered at the projected point, with both side lengths set as 3 mm. Then press Finish Sketch.

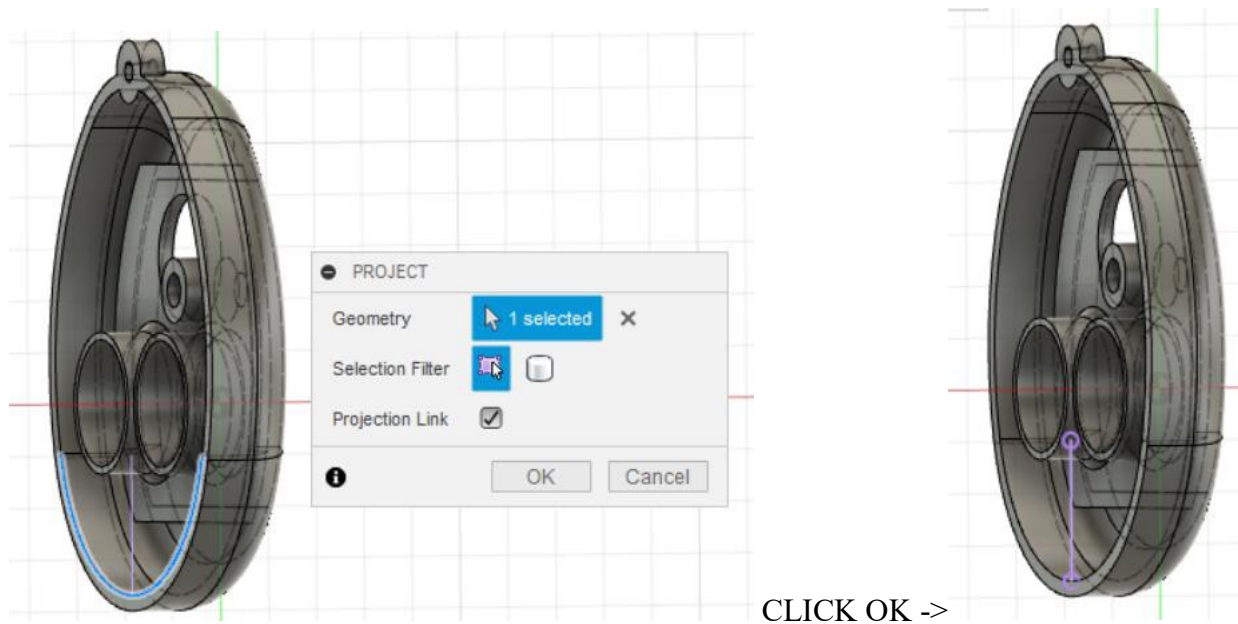


- f) Select the Loft tool from the Create dropdown. This tool smoothly transitions one profile into another. Select the circle and the square profile by clicking on them once the loft tool is open. This should set “Operation” to Cut, but if it does not you can set it yourself. Press OK to create the lofted cut.

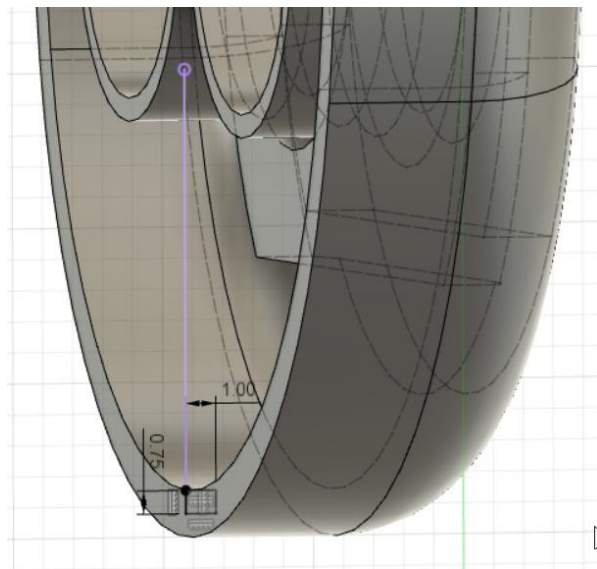


5) Use a Sweep to add a groove

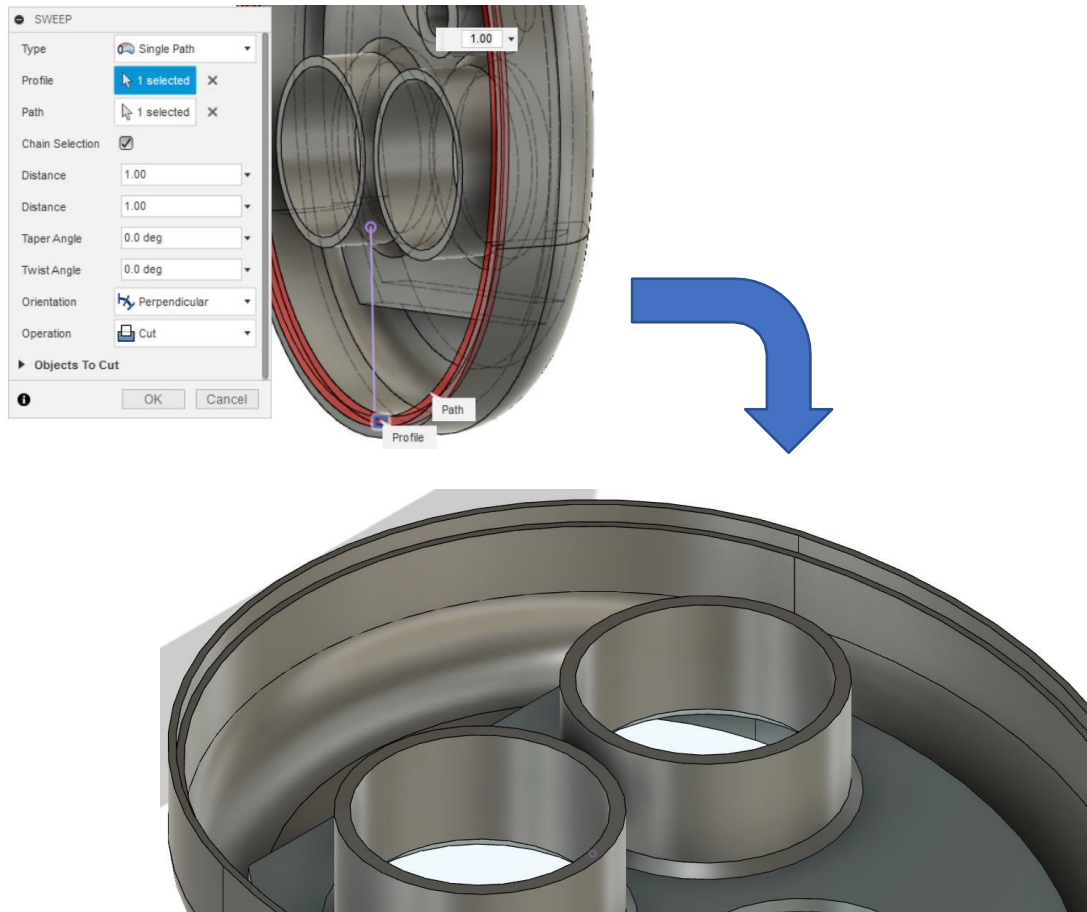
- Start a new sketch on the YZ plane
- Rotate the camera slightly and use the Project tool to project the following edge (after the projection use the View Cube to look at the sketch plane straight on again):



- c) Starting at the end point of the projected line, use the 2-Point Rectangle tool to create a rectangle with the following dimensions, then press Finish Sketch:



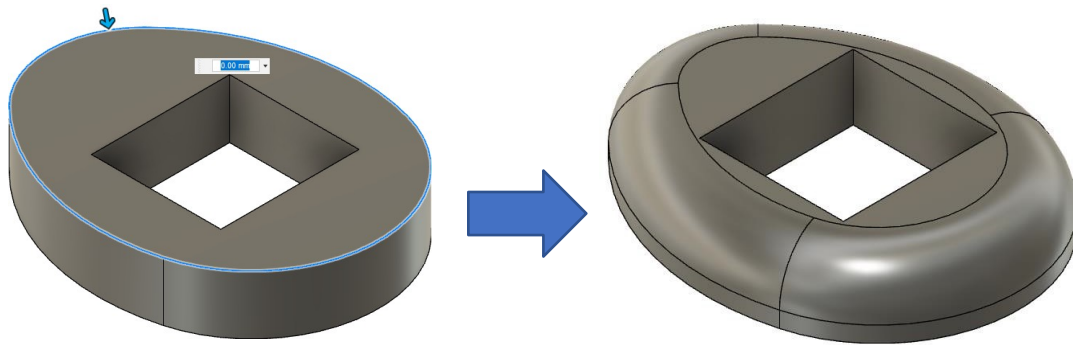
- d) Select the Sweep tool from the Create dropdown. This tool extrudes a profile along a given path. Select the new rectangle as the Profile (if clicking on the rectangular profile doesn't work, try a lasso selection – i.e. dragging a box around the profile) and for the Path select the inner edge (that was projected in the sketch). Then press OK to create the Sweep.



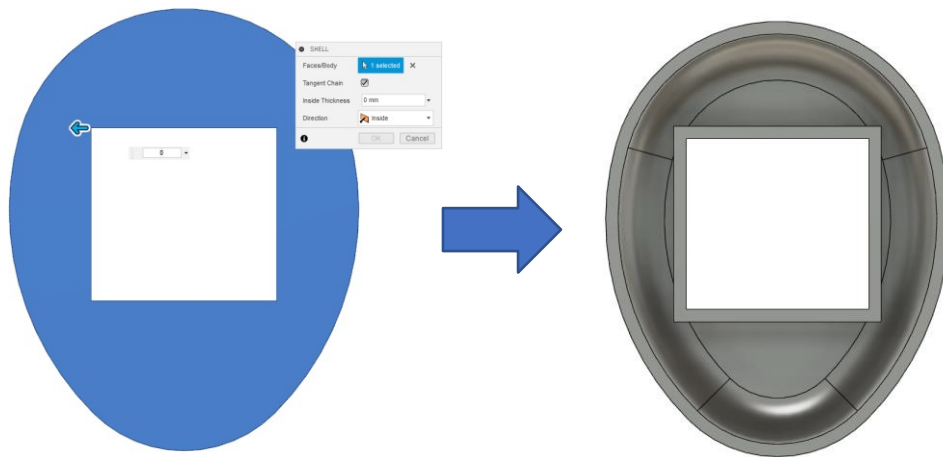
6) Make sure to Save and then close the part.

Part II. Modify Part 5

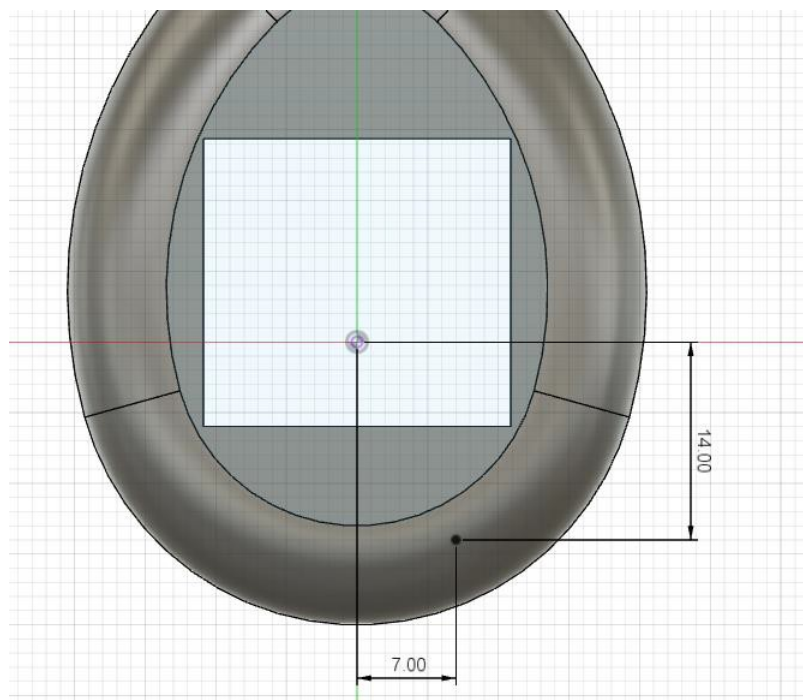
- 1) Open Part 5, making sure it is saved in your Lab 4 folder
- 2) Add features to the part
 - a) Fillet the outer edge with a fillet radius of 7 mm



b) Shell the part, selecting the bottom face and a thickness of 1.5 mm



c) Start a new sketch on the top (flat) face of the part. Create a new point, dimension it to the origin as follows, and press Finish Sketch:

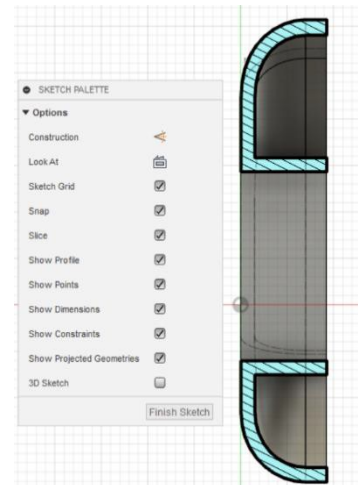
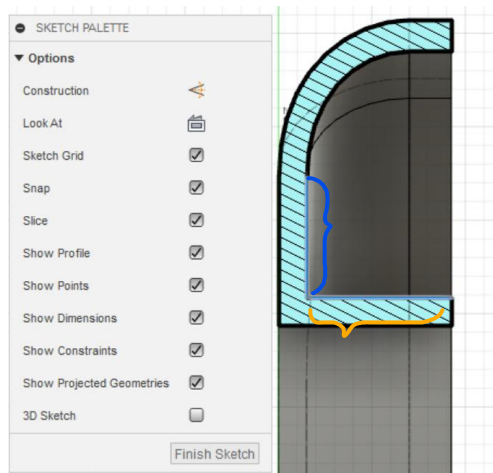


- d) Create a Hole at that point with diameter 3.9 mm and “Extents” set as All

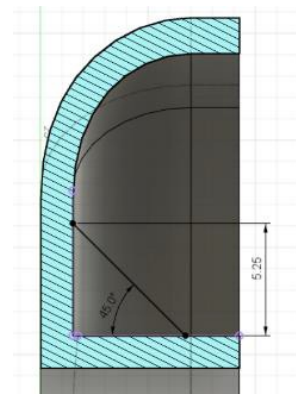


- e) Under the Construct dropdown, select Offset Plane, choose the YZ plane, and enter -6 mm to offset the new plane by
- f) Create a new sketch on the new plane and in the Sketch Palette, turn on Slice

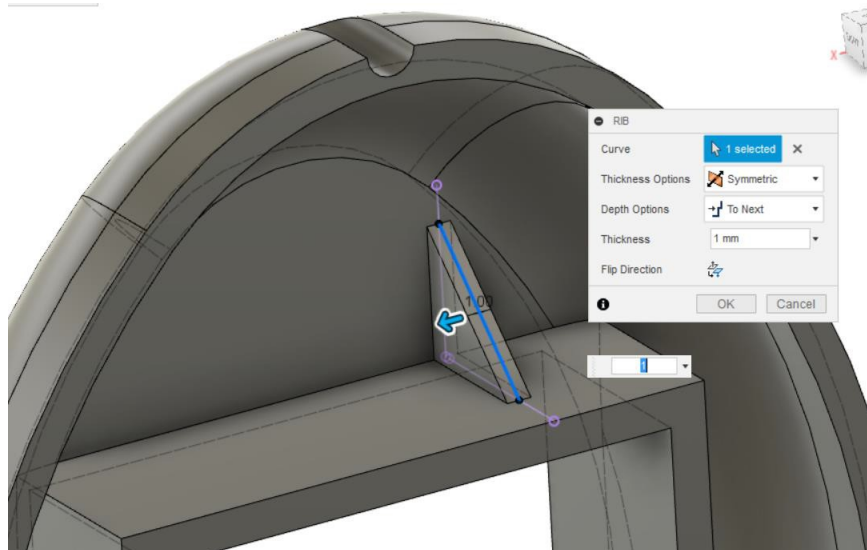
- g) Use the Project tool to include the highlighted borders in this sketch:



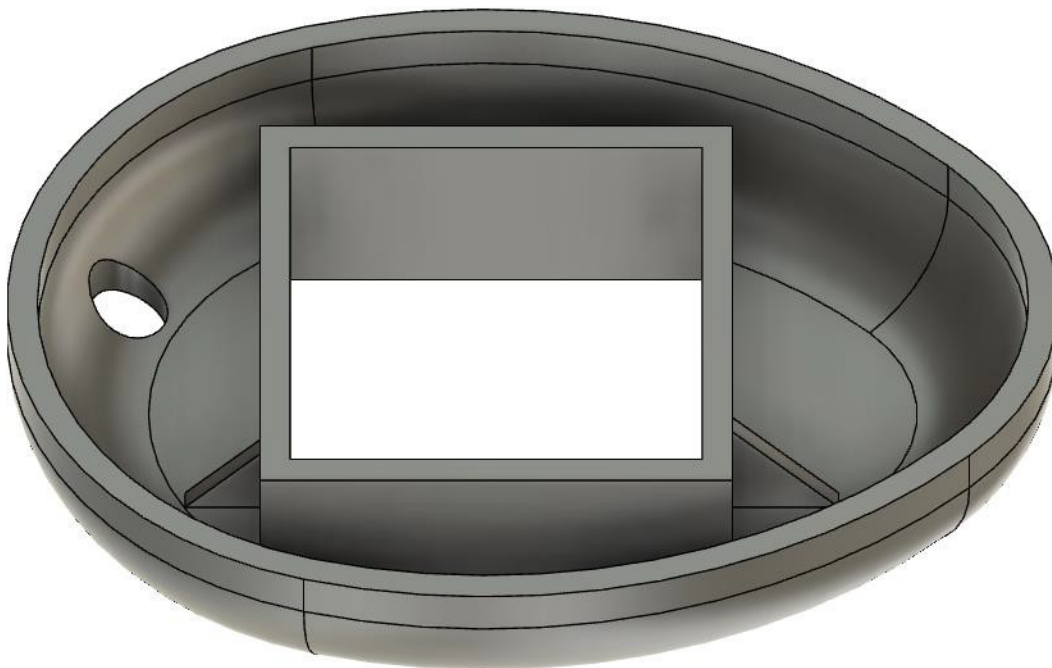
- h) Add a line creating a triangle with height 5.25 mm, then press Finish Sketch



- i) Under the Create dropdown select the Rib tool. Select the line you just created, enter 1 mm as the rib thickness and press OK.

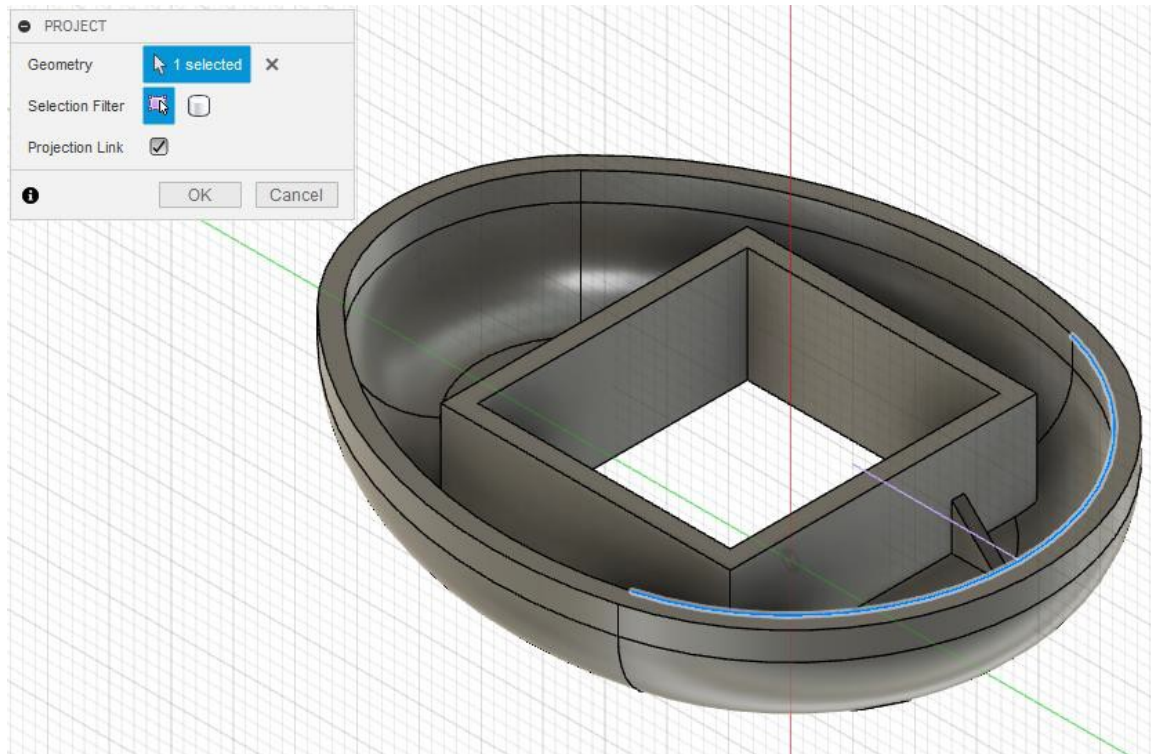


- j) Create a rib on the other side of the rectangular cutout using the same exact process but with two key difference
- i) Instead of 5.25 mm, set the height of the line to 3.7 mm
 - ii) When using the Rib tool, select the “Flip Direction” option

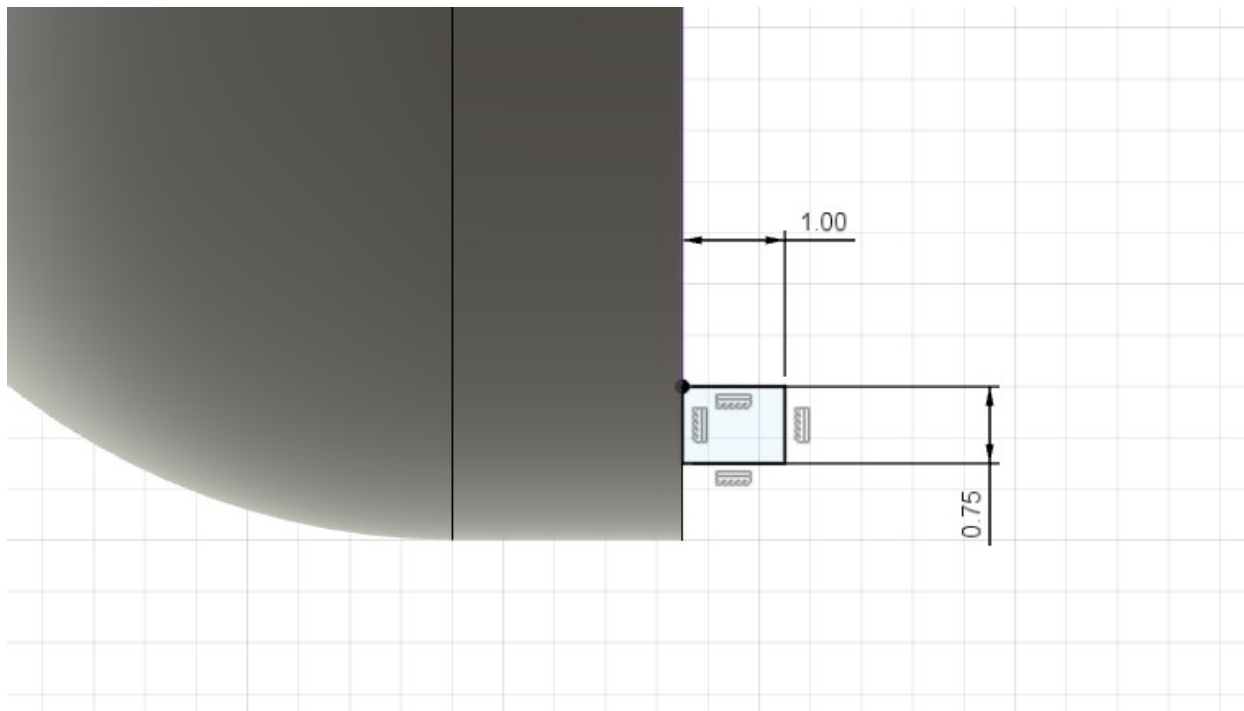


- 3) Use a Sweep to create a Lip
- a) Create a new sketch on the YZ plane

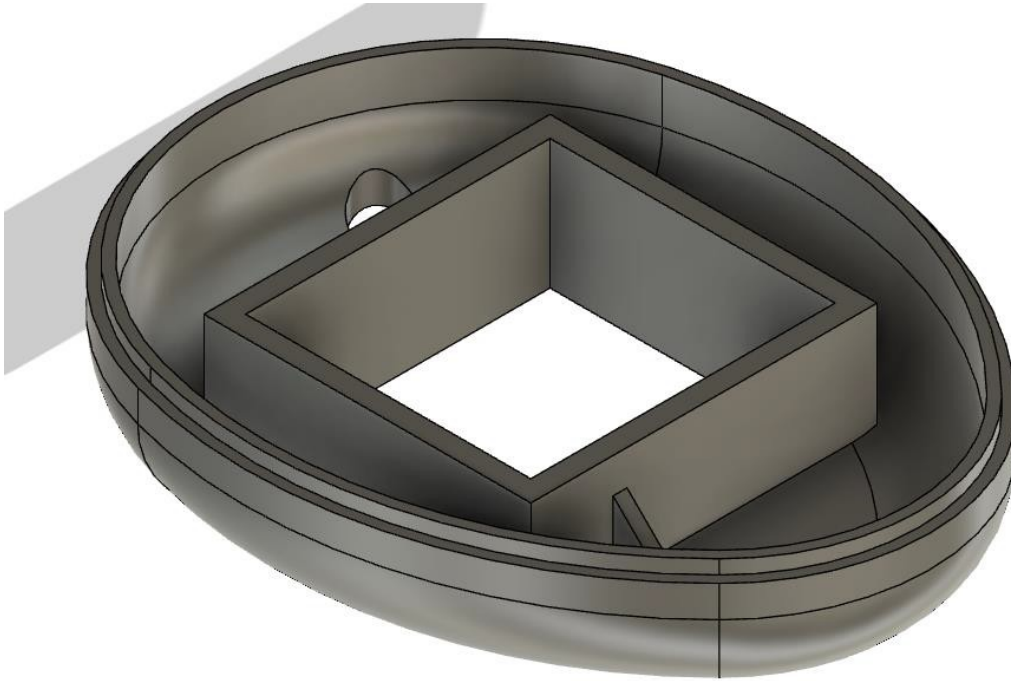
- b) Use the Project tool to include the highlighted edge in the sketch, then return to the YZ plane



- c) Starting from the end of the projected line, create a rectangle with the following dimensions and then press Finish Sketch



- d) Use a Sweep (under the Create dropdown) to sweep this profile along the inner edge.
This time the “Operation” should be a Join.

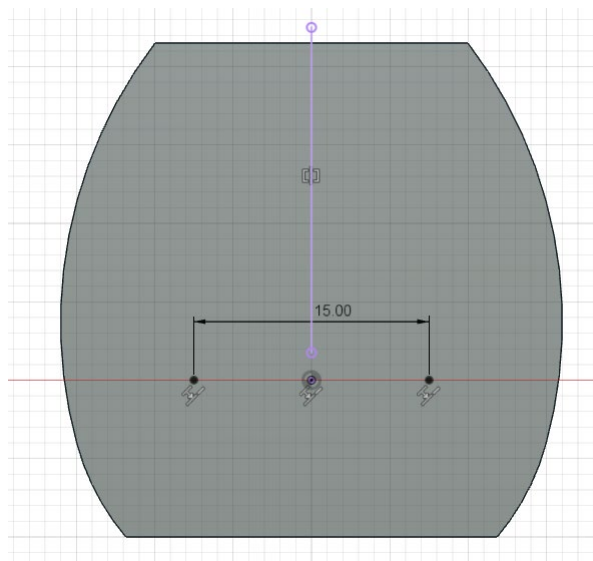


- e) Save the part and close it. Open Part 7

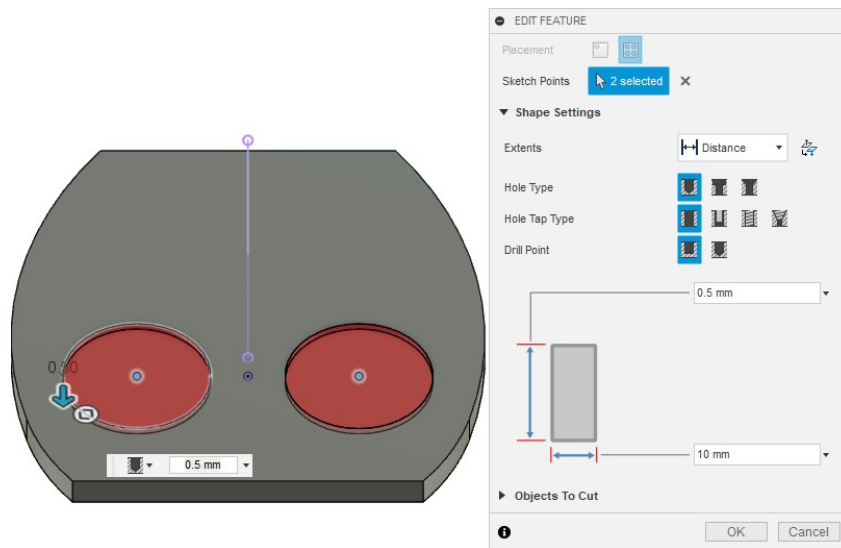
Part III. Modify Part 7

1) Create Holes

- a) Start a sketch on the Front face of the part
b) Create two points, symmetric about the Y axis, that lie on the X axis and are 15 mm apart



c) Finish Sketch and Create a Hole at both points with the following specifications



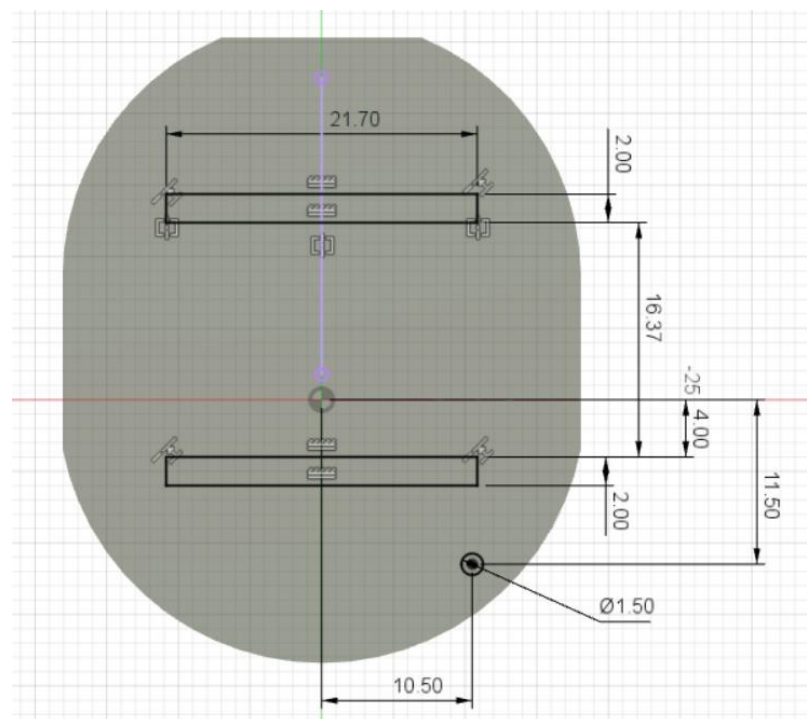
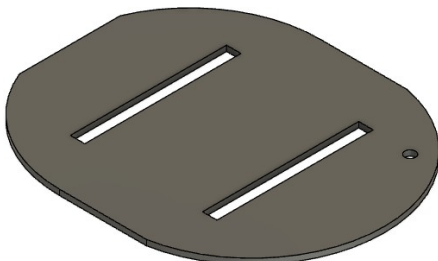
i) Note: make sure to change the Drill Point type to Flat

d) Press OK to make the Holes, then Save and close the part. Open Part 9.

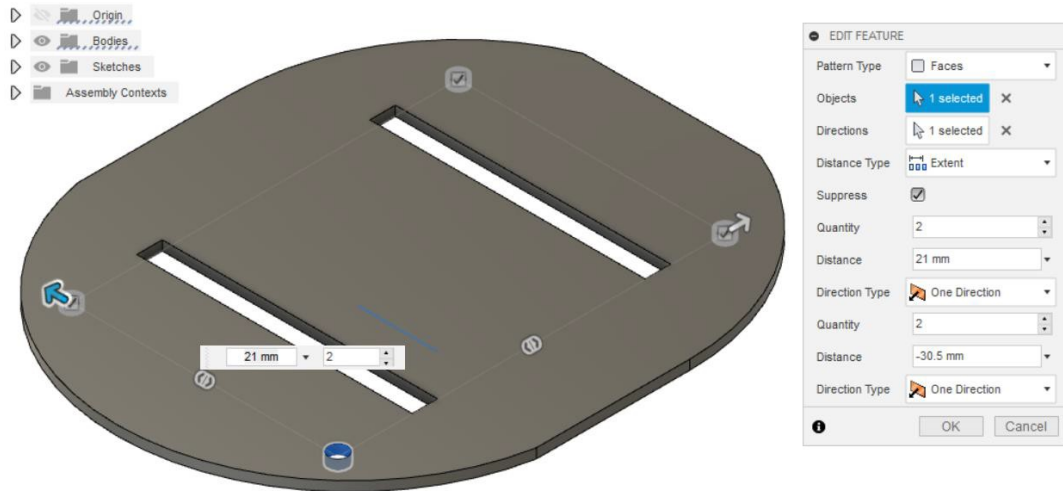
Part IV Modify Part 9

1) Create a cutout

- Start a sketch on the Top face of the part
- Sketch the following geometry. Note: the rectangles are identical, and the circle is constrained to the origin
- Finish Sketch, and Extrude all 3 profiles as a cut through the body



- d) Pattern the hole as below, to create 4 holes as shown. Under “Create”, find “Pattern-Rectangular Pattern”. Select the hole and the correct direction (one of the axis):



Part V. Submission Requirements

- Make sure all your files are saved in the correct folder and that your ME170 folder is shared with your TA.
- Go to ME170 Blackboard website and the CAD LAB Assignments content area. Click directly on the “CAD LAB 4” 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.