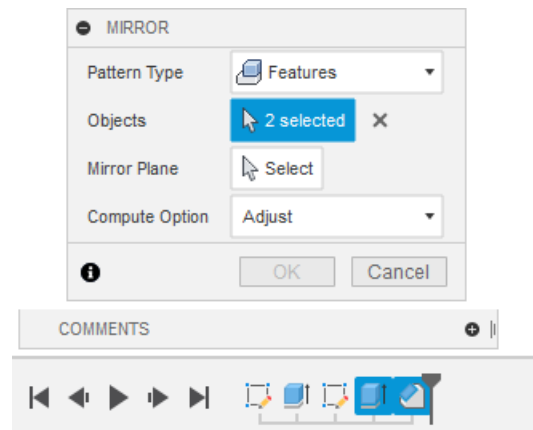


Lab Assignment #5: Patterns & Model Modification

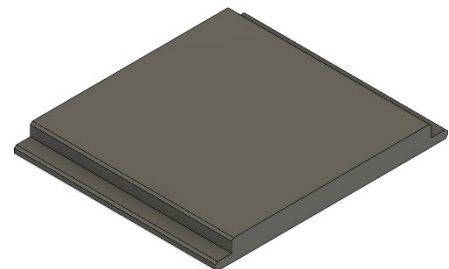
ME 170

Part I. Create Mirrored Features

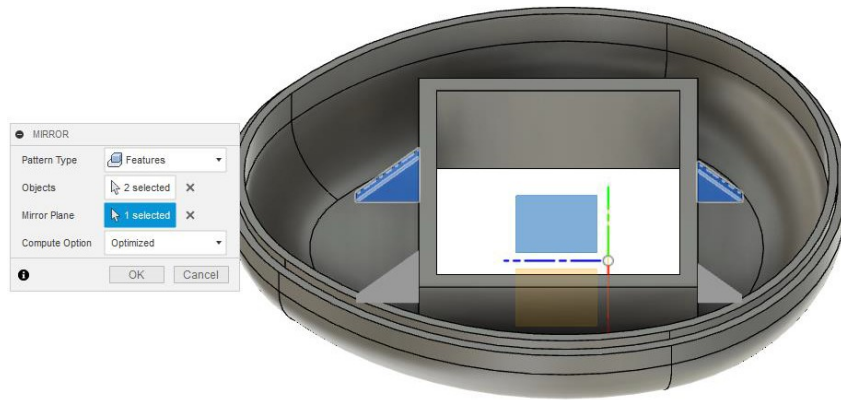
- 1) Create a new folder
 - a) Start Fusion and open the Data Panel
 - b) Navigate to your ME170 project and create a new folder named “Lab 6”
 - c) Make copies of Parts 2, 4, 5, and 8 in the new Lab 6 folder
 - d) Double-click on Part 2 to open it
- 2) Mirror the Cut in Part 2
 - a) Select the Mirror tool from the “Create” dropdown. This tool repeats inputs across a chosen plane. In this case we will be mirroring features, so change the “Pattern Type” from Faces to Features
 - b) For the Object, select the cut Extrusion and the Chamfer from the timeline in the bottom left of Fusion



- c) For the Mirror Plane, select the YZ plane, change the Compute Option from “Adjust” to “Optimized” and press OK to mirror the cut
 - d) Save the part (making sure it is in the Lab 6 folder) and open Part 5
- 3) Mirror the Ribs in Part 5
 - a) Select the Mirror tool from the “Create” dropdown



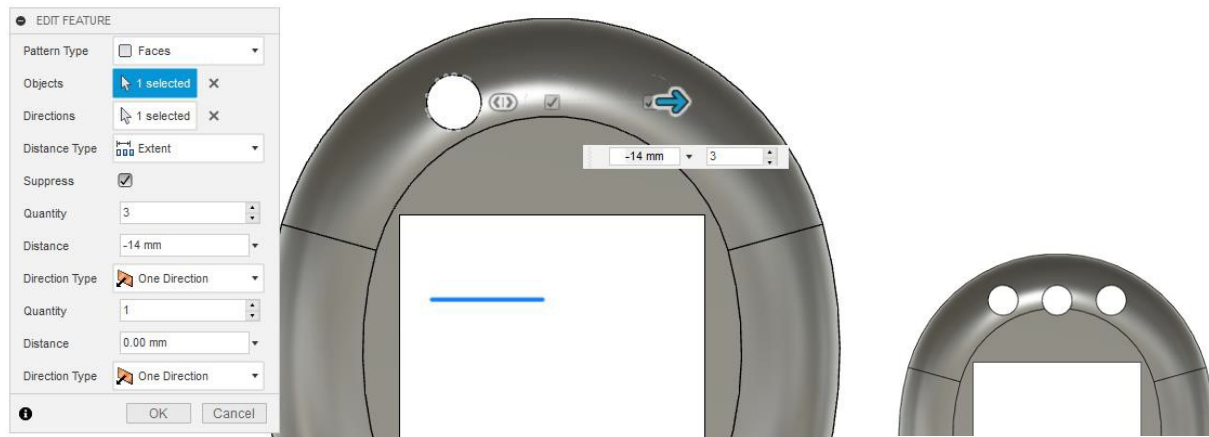
- b) For the Object, select both Rib features from the timeline at the bottom-left of Fusion
- c) For the Mirror Plane, select the YZ plane (if this plane does not bisect the part parallel to the ribs, create a Construction Midplane between the sides of the rectangular cutout and use that)



- d) Press OK to create the Mirror and save the part

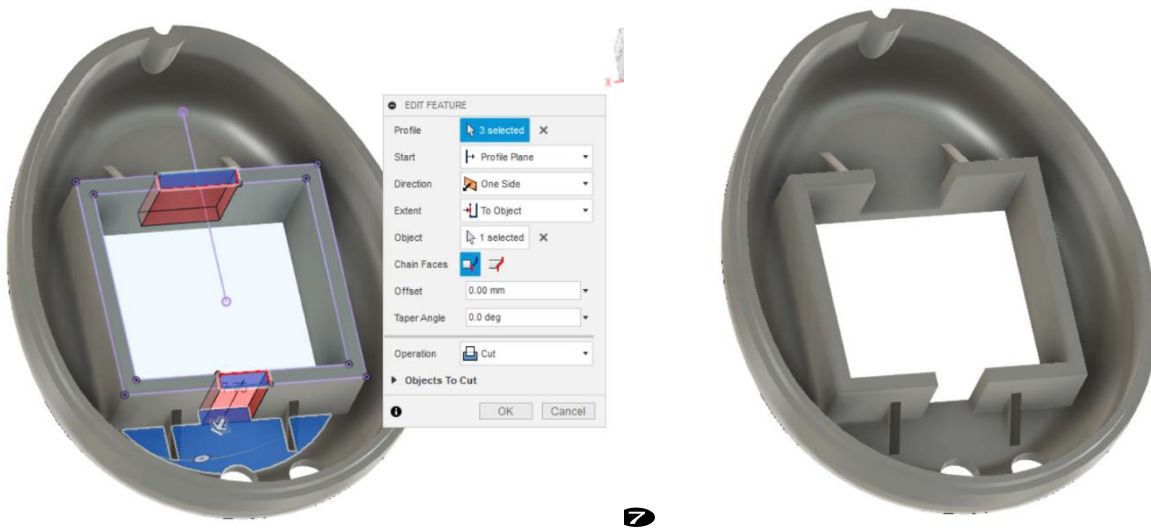
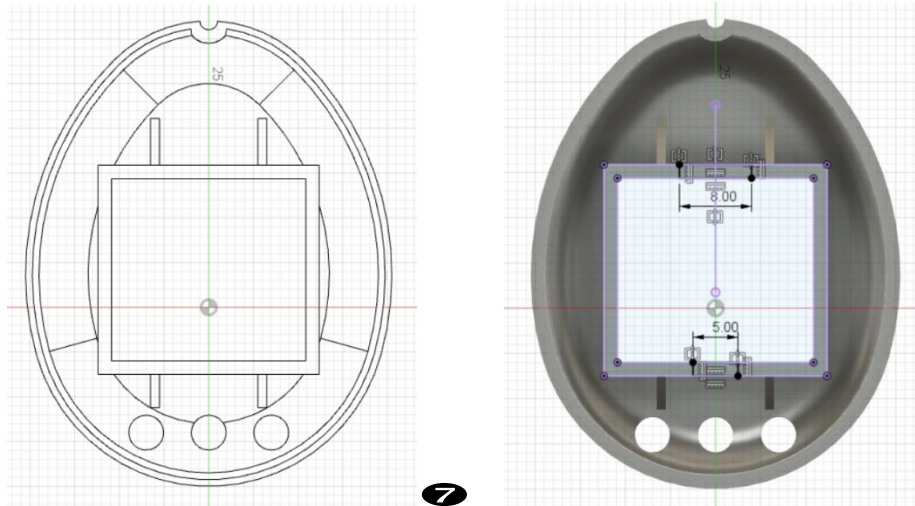
Part II. Create Patterned Features

- 1) Pattern the existing Hole
 - a) Select the Rectangular Pattern tool from the “Create” tab
 - b) Select the Hole feature (from the timeline at the bottom) as the Object, set the Direction as the X axis, set the Distance as -14 mm, and set the Quantity as 3

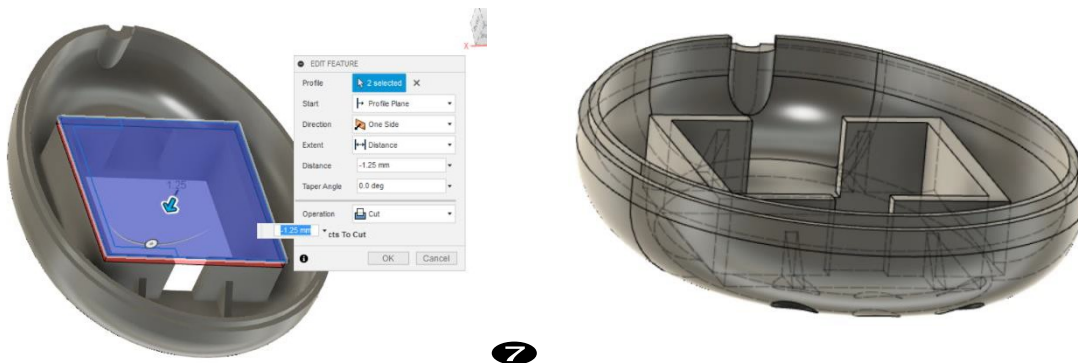


- c) Press OK to create the Pattern

d) Create slots in Part 5 as cuts dimensioned and constrained as shown below:



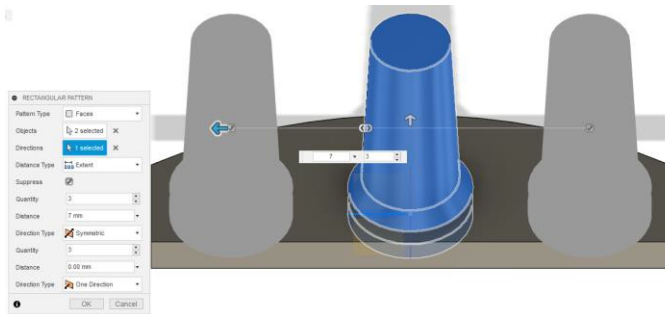
e) Reduce the height of the central rectangular protrusion by 1.25mm



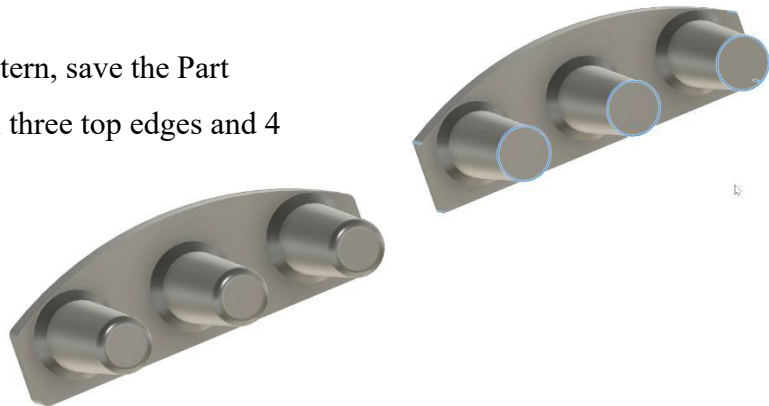
f) Save the part, and close it. Open Part 8

2) Pattern the Revolutions

- From the “Create” tab, select the Rectangular Pattern tool
- Select both Revolves in the part for the Object, select the Y-axis for Direction, enter 7mm as the Distance, and change Direction Type to Symmetric (Quantity should default to 3, otherwise change it to 3)

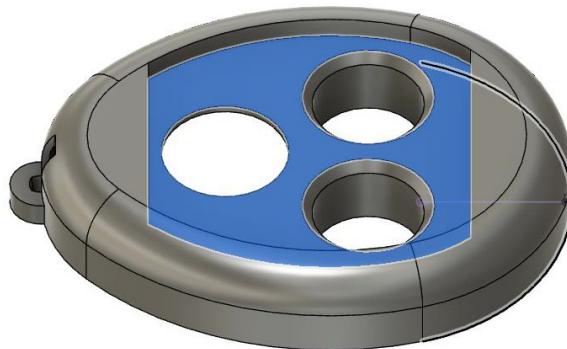


- Press OK to create the Pattern, save the Part
- Add R0.5 mm fillets to all three top edges and 4 bottom corners

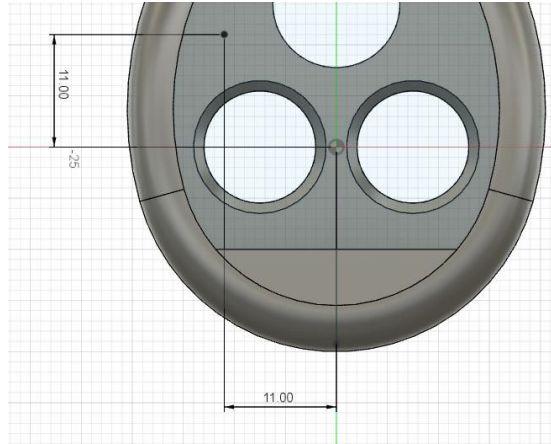


Part III. Modify the Timeline

- Open Part 4: Create a new Hole and Modify it
 - Start a new Sketch on the top, indented face of the part (shown below)



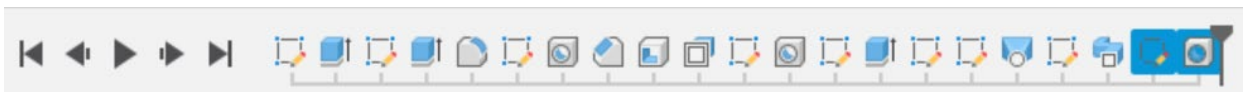
- Place a point (Create > Point) and constrain it so that it is 11 mm away from the origin in both X and Z directions. Then press Finish Sketch



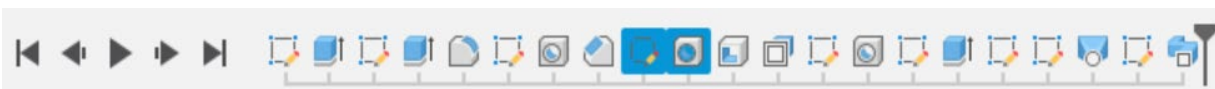
- c) Use the Hole tool from the “Create” tab to make a Hole at the point with a diameter of 3.3 mm with Extent set to All. Turn the part over to observe the hole you just created.



- d) In the timeline at the bottom-left of Fusion, Select the Hole feature, then hold down Shift and select the Sketch it is based on (selecting both objects)



- e) Drag them along the timeline until they precede the Shell feature, and drop them there

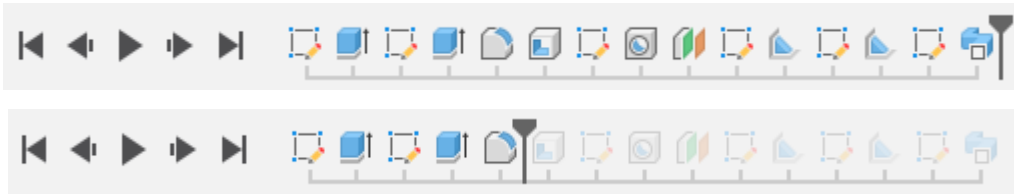


Notice how the part changes (specifically the underside). Consider why it does this.

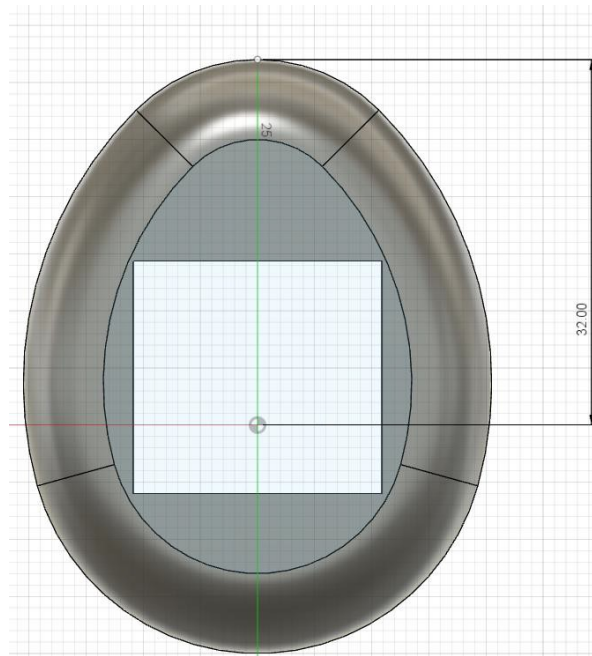


- f) Save the part and Open Part 5
2) Create a Hole early in the part history

- a) Click and drag the timeline marker to before the Shell was created



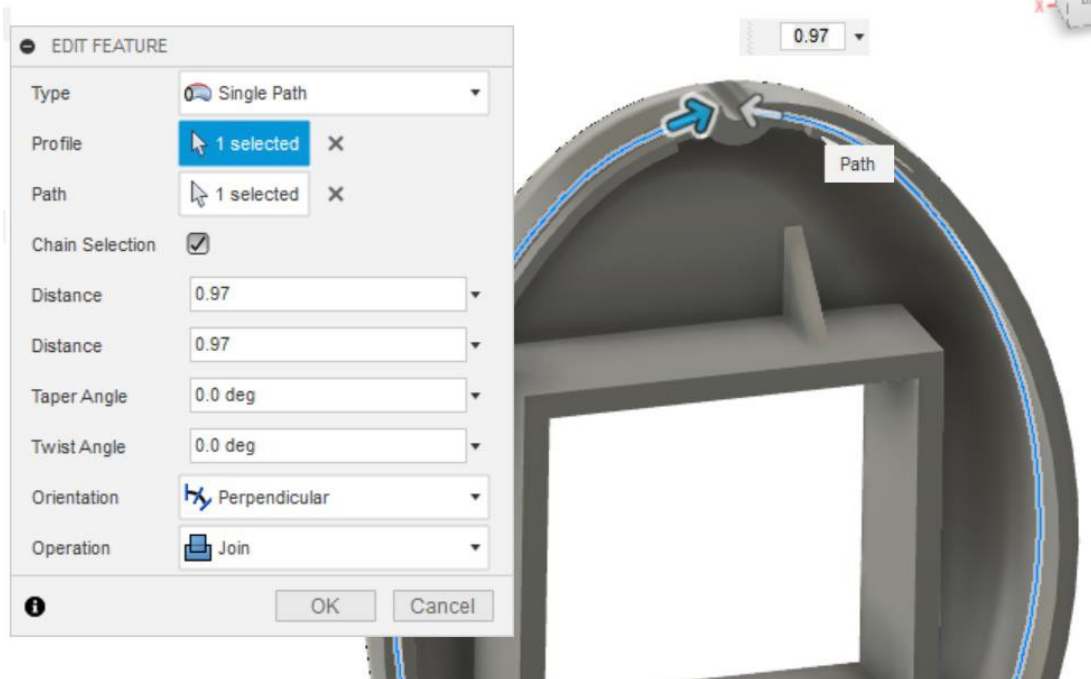
- b) Start a Sketch on the top face of the part
- c) Place a point (Create > Point) and constrain it so that it is 32 mm away from the origin in Z direction. Then press Finish Sketch



- d) Use the Hole tool from the “Create” tab to make a Hole at the point with a diameter of 2 mm with Extent set to All
- e) Move the timeline back to the end of the part, and notice how at the top of the part, the swept lip is not cut by the hole. This is because of the ordering in the timeline



- f) Double click on the Sweep in the timeline at the bottom to adjust it for a larger gap. Change both distances to 0.97 to create 97% of a full sweep. Press OK and the gap in the sweep will be larger.



- g) Save the Part.

Part IV. Submission Requirements

- a. Make sure all your files are saved in the correct folder and that your ME170 folder is shared with your TA.
- b. Go to ME170 Blackboard website and the CAD LAB Assignments content area. Click directly on the “CAD LAB 5” 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.