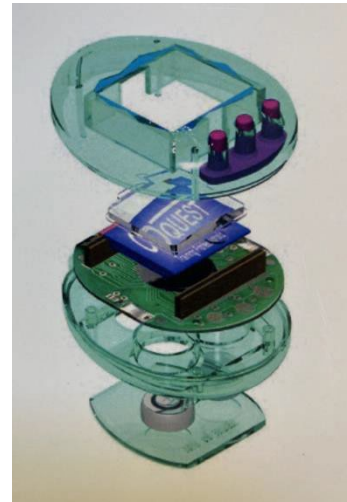


Lab Assignment #2: Intro to Part Modeling

ME 170

Overview

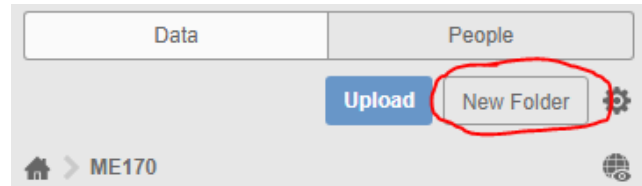
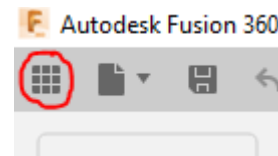
You will learn the basic operation of sketching in Fusion and how to capture “Design Intent” through use of ‘constraints’. You will then CAD model a few simple parts. These are components of a small electronic product.



Part I. Create a new folder called 'Lab 2'.

This file will be used to store your files for this lab.

- 1) Start Fusion, go into your ME 170 project, and create a new folder.
 - a. Start Autodesk Fusion 360, and make sure to log in using your Illinois email address.
 - b. Open the Data Panel by clicking the button in the top right.
 - c. Navigate to the ME 170 Project you made last lab by double clicking on the project.
 - d. In the top right of the Data Panel, select “New Folder.”
 - e. Enter “Lab 2” as the name for this folder, then double click to open.
Note: You can change the name of folders by right clicking on them and selecting the “Rename” option.
 - f. Once finished, you can click the Data Panel button again to close the Data Panel.

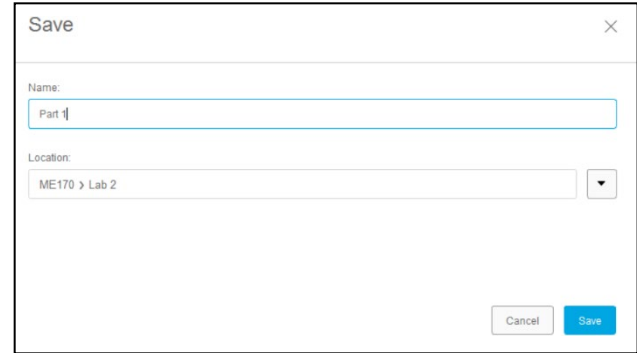


Part II Create a New Part.

Every part we will be making will follow the same initial steps to setup the file. Follow the directions below for each new part.

- 1) Create a new part within the correct folder.
 - a. Launching Fusion should open an untitled new part. If it does not, in the top left of your screen, select File > New Design (or press Ctrl-N).

- b. Make sure the file has mms as the primary units (if you need a refresher on how to change units, check Lab 1).
- c. Select File > Save (or press Ctrl-S).
- d. Enter the appropriate name (in this case Part 1), make sure it is saving to the appropriate location (ME 170 > Lab 2), and press Save.



Part III. Create Basic Sketches

In this exercise, you will begin modeling a small consumer product, an electronic keyfob. You will create part files and save them in your shared folder for grading.

Part 1

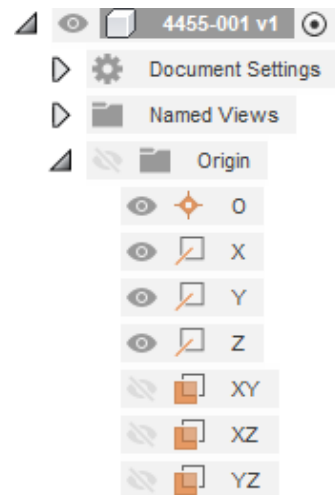
- 1) Begin a new sketch

- a) In the same part you just created (Part 1), select “Create Sketch” from the top toolbar, next to the word “Design.”



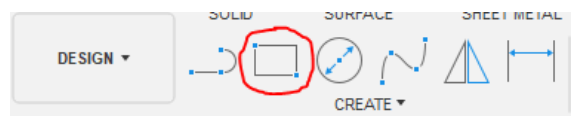
- b) You now need to select a plane to sketch on from the 3 principal planes (XY, XZ, and YZ). Remember, sketches are 2-D outlines that are converted into 3-D parts, so the sketch must be on a single plane. The principal axes are labeled on the View Cube in the top right of the screen. Select the XZ plane by clicking on the blue rectangle that lines up with the X and the Z axis.

Note: You can also click the arrow next to the Origin folder on the left side of the screen to see the principal geometry. After selecting “Create Sketch” you can directly click on one of the principal planes here to begin a sketch on it.



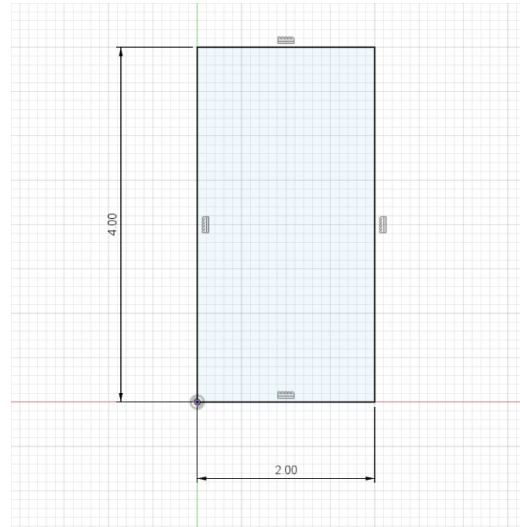
- 2) Sketch and dimension a rectangle

- a) In the “Create” Toolbar, select 2-Point Rectangle. This tool creates a rectangle using 2 points as the opposite corners.



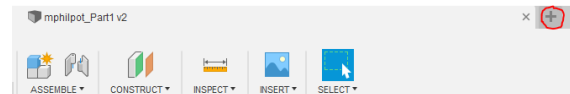
- b) Select the origin as your first point and enter 4 as the first dimension by typing it (as the first dimension box should already be highlighted). Then press Tab on the keyboard to switch to the other dimension and enter 2. Once both dimensions have been added, press Enter on the keyboard to create the shape.

Note: If either dimension is incorrect it can be modified by double-clicking on it. If no dimension appears, one can be added using the “Sketch Dimension” tool in the Create toolbar (or with hotkey ‘d’).



- c) Press “Finish Sketch” in the right side of the top toolbar to leave Sketch Mode.

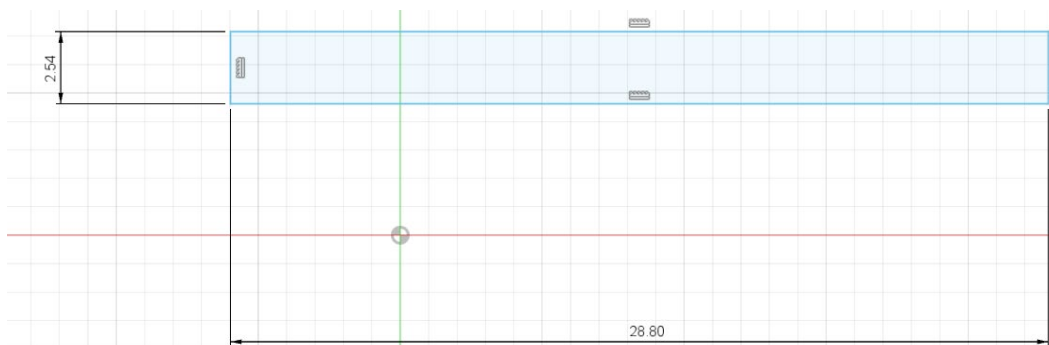
- 3) Save the part and open a new part



- a) Either select File > Save or press Ctrl-S to save the part. In the “Version Description” window that appears, type in “Sketch created” as a reminder that this version of the part was when the sketch was added.
- b) In the top right of the screen, click the + next to the name of this part to open a new part named “Untitled” which can be used as second part in thislab.

Part 2

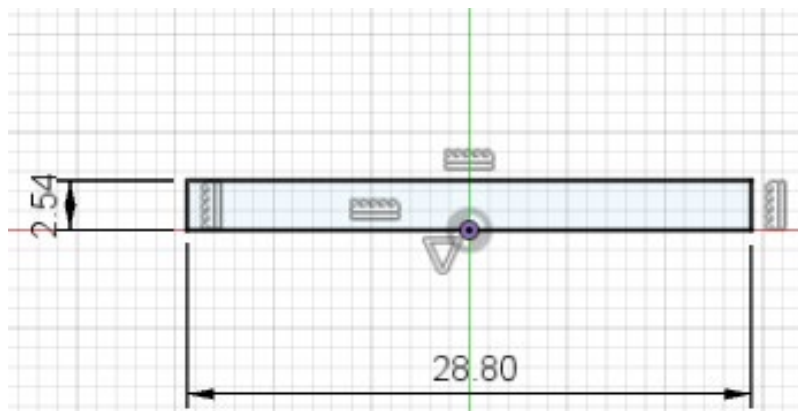
- 1) Set the units as Millimeters. Save the new file as “Part 2” in the Lab 2 folder using the same process as before.
- 2) Create a new sketch
 - a) “Create sketch” on the XZ plane.
 - b) Like in Part 1, use the Rectangle tool to make a rectangle, but this time do not make it start at the origin (choose anywhere else). Make the height 2.54 mm and the width 28.8 mm



- c) From the “Constraints” section of the top toolbar, select the Midpoint Constraint.



This tool centers an object on a chosen point. With the tool active, select the bottom line of the rectangle, then select the origin.

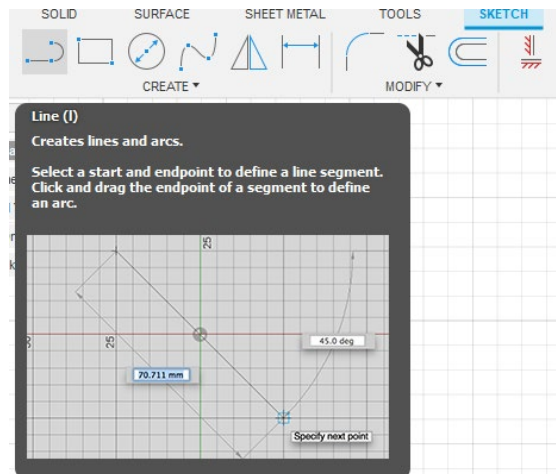


- 3) Press Finish Sketch, save this part, and open a new part using the same method as Part 1.

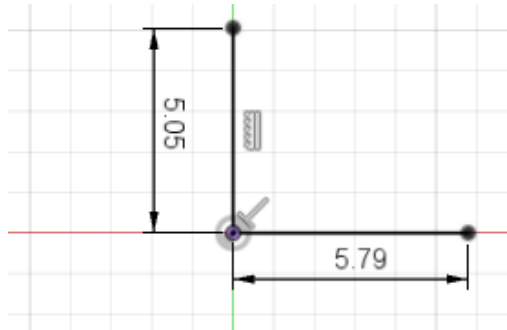
Part 3

- 1) Set the units as Millimeters. Save the new file as “Part 3” in the Lab 2 folder.
- 2) Create a new sketch
 - a) “Create sketch” on the YZ plane
 - b) Select the Line tool from the top toolbar.

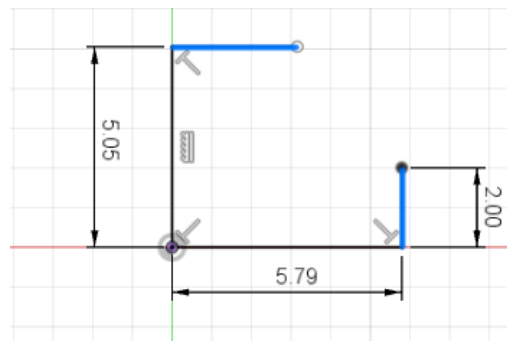
This tool creates a straight line between chosen start and end points. Once you have chosen the starting point, you can type to enter the length of the line and use the mouse to determine direction.



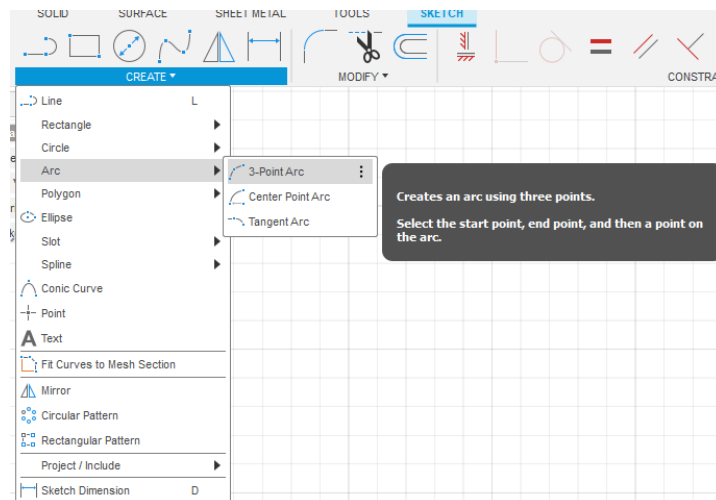
- c) Starting at the origin, use the Line tool to create the following two lines



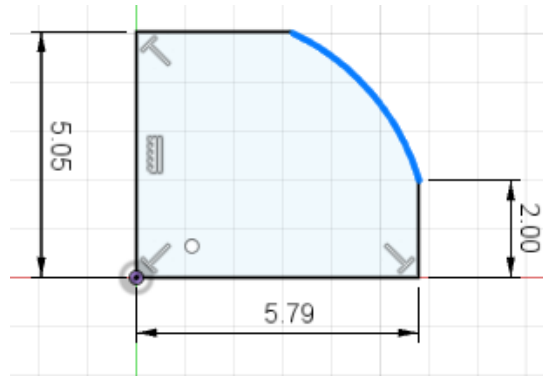
- d) Add the following two lines, one of specified length, one of unspecified length.




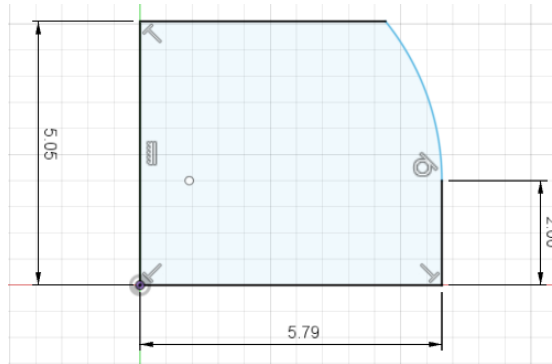
- e) Open the “Create” dropdown, hover over the “Arc” section, and select the 3-Point Arc tool. This tool creates a curved arc between two points, with the third point determining the extent of the curve.




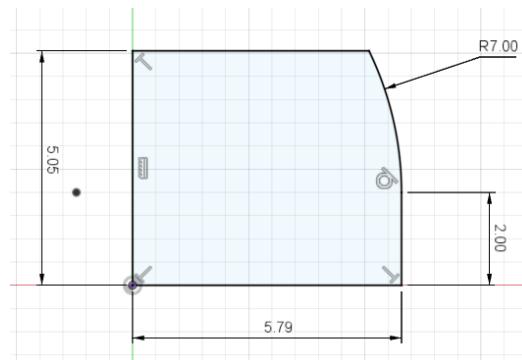
- f) Using the 3-Point Arc tool, connect the two newest lines, selecting a random third point.



- g) From the “Constraints” section of the top toolbar, select the Tangent Constraint . This tool makes a curve tangent to a line or vice-versa. With the tool active, select the Arc and then the line on the right.



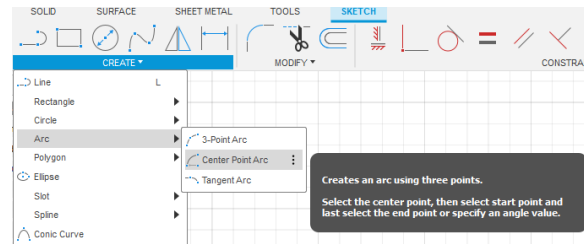
- h) From the “Create” section of the top toolbar, select the Sketch Dimension tool  (hotkey d). This tool allows you to modify the dimensions of existing geometry. With the tool active, select the arc, then left click elsewhere to place the dimension. Type 7 and press Enter to set the dimension as 7 mm.




- 3) Press Finish Sketch, save this part, and open a new part.

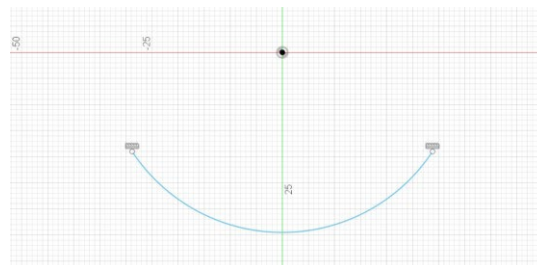
Part 4

- 1) Save the new file as “Part 4” in the Lab 2 folder. Make the file units mm.
- 2) Start a new sketch on the XZ plane.
 - a) Select the dropdown arrow on the Create tab, hover over the Arc option, and select Center Point Arc. This tool requires 3 points to make an arc: the first determines the center and the next two selected are the start and end points

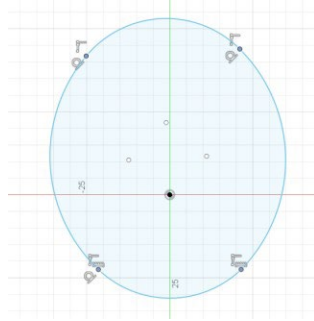


- b) Use the Center Point Arc tool to make an arc centered at the origin, aligning the start and end points. Use the grid to help create the arc reasonably to scale, the radius will ultimately be 20 mm. It will help later when you add dimensions if you are not too far out!


- c) Under the “Constraints” tab, select the Horizontal/Vertical Constraint  and then select the start and end points of the arc. This constraint ensures the points are of equal height



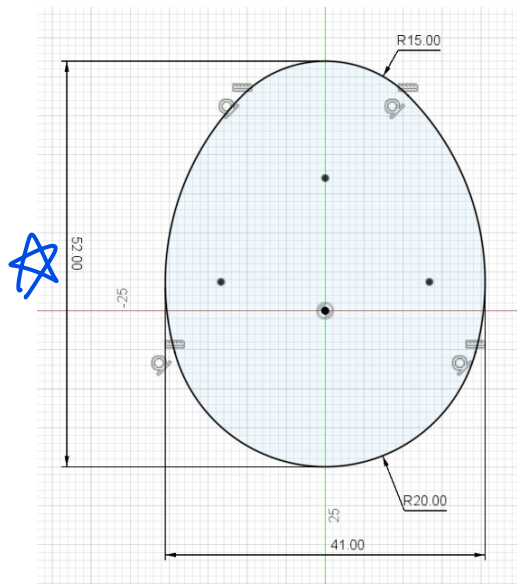
- d) Select the Tangent Arc tool from Create > Arc and make 3 new arcs to complete the following shape:



- e) Use the Horizontal/Vertical Constraint on the top two points, then select the Tangent

Constraint  and use it by selecting the last arc created and the first arc. This should ensure all 4 arcs are tangent at the start and endpoints.

- f) Use the dimension tool to enter the following dimensions:




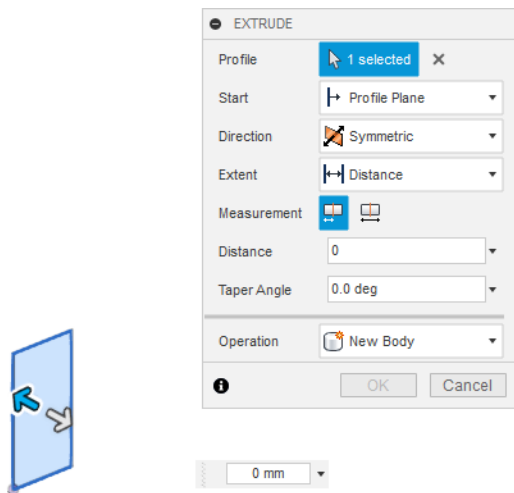
- i) Note: To dimension to the sides of arcs, select the Dimension tool but before selecting the arc, right-click somewhere on the canvas but away from the sketch itself and select “Pick Circle/Arc Tangent”. Then select the two arcs and place the dimensions as shown in the above sketch.
- 3) Press Finish Sketch, save this part, and go back to Part 1. The Part 1 tab should still be open, but if it is not, press File > Open.

Part IV. Transition 2D Sketches into 3D Parts

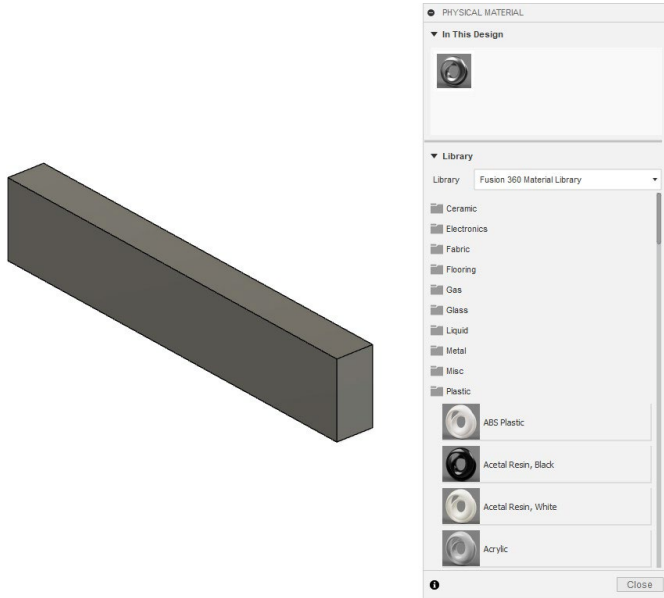
Part 1

1) Extrude the rectangle and select a material

- In the Part 1 file, select the Extrude tool  from the “Create” toolbar.
- The rectangle profile should be automatically selected (if it is not, click inside the rectangle to select it). In the Extrusion pop-up window, in the “Direction” section, change One Side to Symmetric.



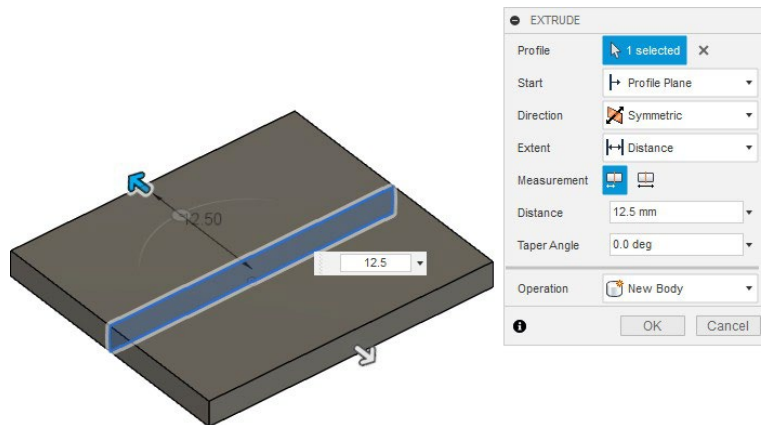
- Type 10.75 mm as the Distance and press OK on the pop-up to create the Extrusion.
- Open the “Modify” dropdown and select Physical Material. Click on the Plastic folder to open it and scroll down to Rubber. Click and drag Rubber onto the rectangular prism to change the material to Rubber. This changes the appearance and the physical properties.



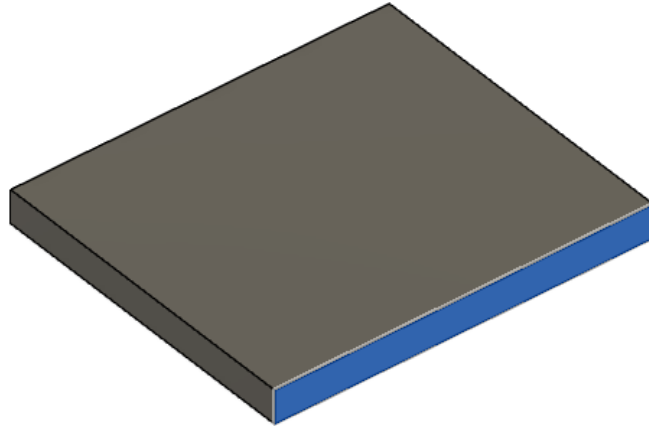
- 2) Save the part with Version Description “Extrusion Added” and close the file. Open Part 2.

Part 2

- 1) Extrude the profile symmetrically for a Distance of 12.5 mm.



- 2) Create a secondary sketch
 - a) Start a new sketch, but instead of selecting one of the principal planes, select the face of the rectangular prism that is parallel to the sketch profile (XZPlane)



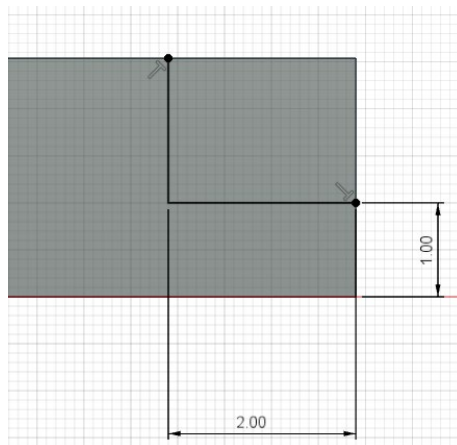
(this face)

- b) In the top right corner, create two Lines that form a rectangle with the border



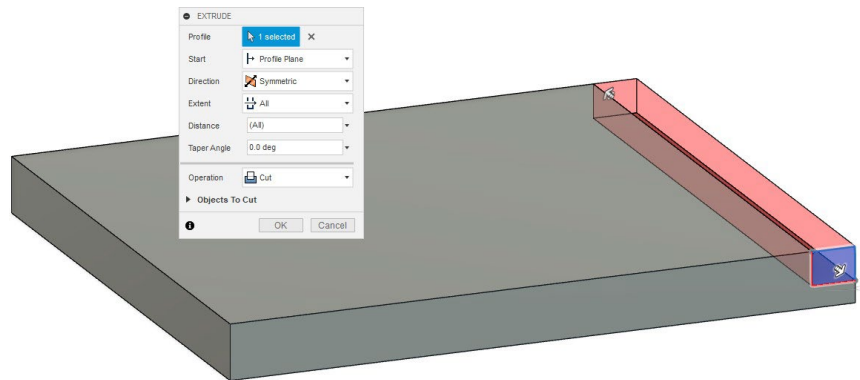
- c) Use the Sketch Dimension tool to dimension the two lines as follows

Note: to create the 1 mm dimension, you must select first the horizontal line and then the bottom border of the object instead of just the length of a line

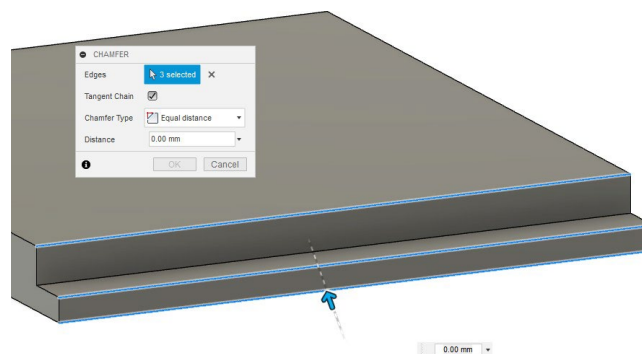


- d) Select “Finish Sketch.”
- 3) Use the new sketch to cut out a portion of the model
- a) Select the Extrude tool and choose the newly sketched rectangle as the Profile

- b) Set the Direction to Symmetric and change the Extent from Distance to All. The Operation should automatically switch to Cut.




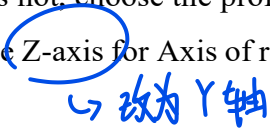
- c) Select OK or press Enter on the keyboard to complete the operation. Change the view to notice how the part has been modified.
- 4) Add Chamfers
- a) From the “Modify” Dropdown, select the Chamfer tool. This tool creates a bevel on edges.
- b) Select the following three edges:

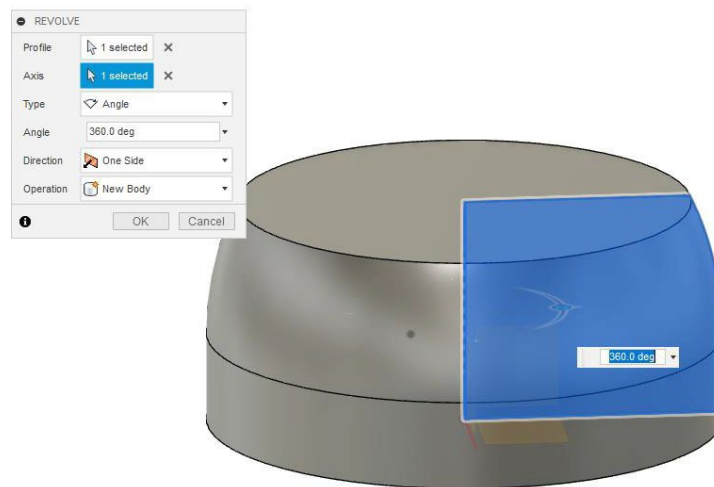


- c) Type 0.25 mm as the Distance and press Enter to create the feature.
- 5) Save and close the file. Open Part 3.

Part 3

- 1) Revolve the profile

- a) Select the Revolve tool  from the “Create” section of the toolbar. This tool traces a profile around a central axis.
- b) The profile should automatically be selected. If it is not, choose the profile enclosed by the sketch. Select the vertical line that runs along the Z-axis for Axis of rotation, causing the part to revolve around its side.

- i) Note: you can also choose the Z-axis itself as the Axis. This requires opening the Origin folder in the Browser, on the left side of the screen.



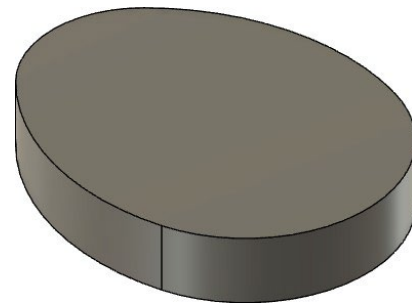
- c) Press OK to finish the feature.
- 2) Save and close the file. Open Part 4.

Part 4


- 1) Extrude the profile 9.25 mm, leaving the other setting unchanged

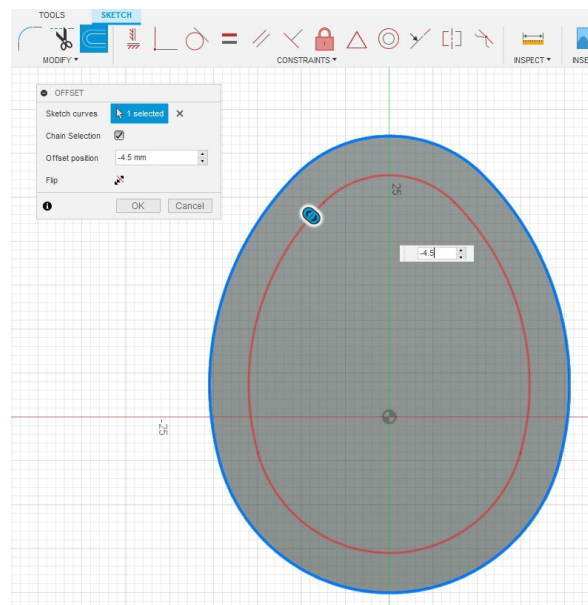
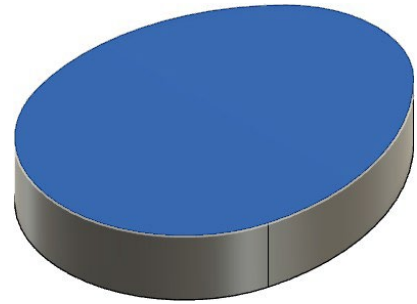


- 2) Make a copy of the part
- a) Open the Data Panel by clicking the Show Data Panel in the top left of the screen
- b) Navigate to your Lab 2 folder and right-click on the part named “Part 4”
- c) Select “Copy” from the list and when a pop-up appears select the “Copy” option

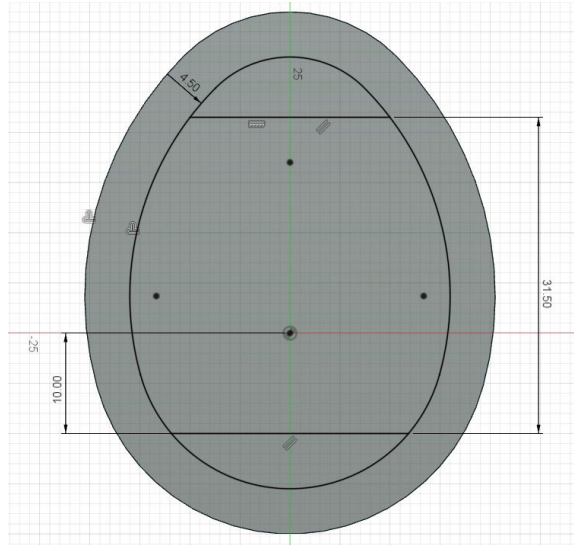



- d) Once a copy has been made, right-click on the copy, select “Rename” and enter “Part 5” as the name
- e) Close the Data Panel
- 3) Continue working in Part 4
 - a) Start a new sketch on the flat surface of the extruded part
 - b) Under the “Modify” section of the top toolbar, select

the Offset tool . This tool makes new sketches a set distance away from existing geometry, but keeps the shape. With the tool open, select the outline of the oval-shape and enter -4.5 as the offset value.

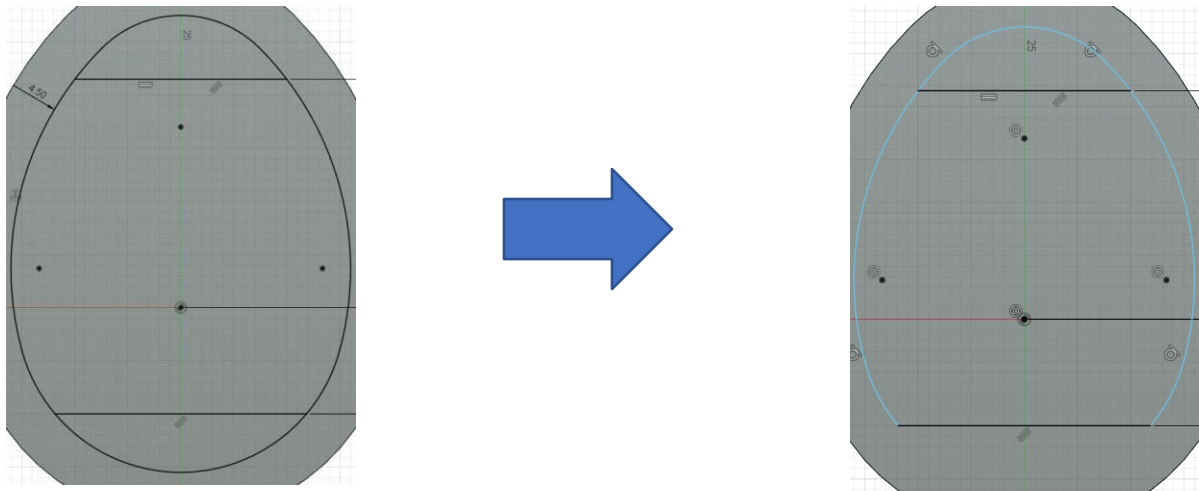


- c) Sketch two horizontal lines that start and end along the new shape and use the Sketch Dimension tool to dimension them as follows:

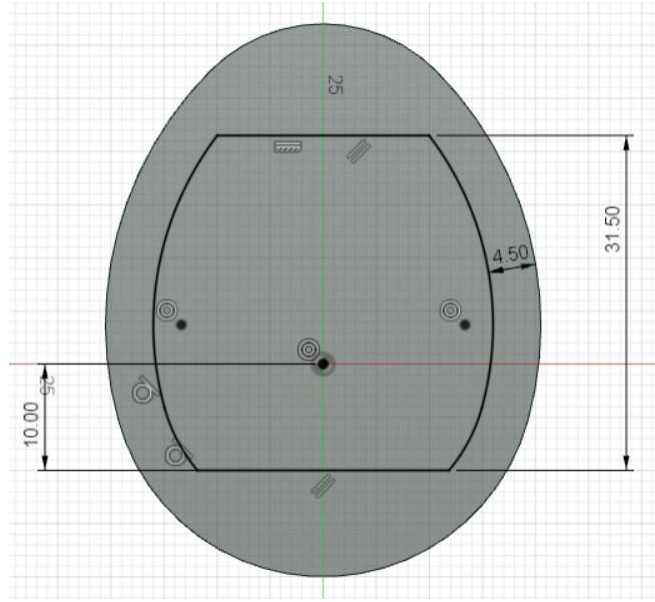


- d) Under the “Modify” section of the top toolbar, select the **Trim tool**  and then click to delete the top and bottom portions of the oval-shape that are outside the boundary created by the horizontal lines. This should require 4 trims (1 for the bottom and 3 for the top since the top is made of 3 different segments)

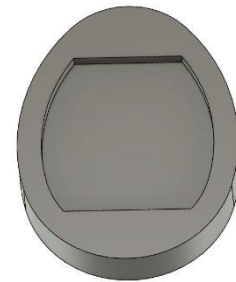
Example of one trim:



- e) Add back the 4.5mm offset dimension by using the dimension tool and selecting the two arcs and make sure the sketch is fully constrained (i.e. no blue lines, all black)




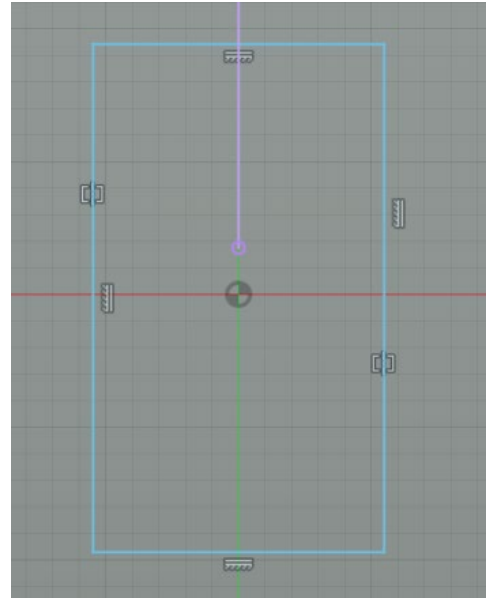
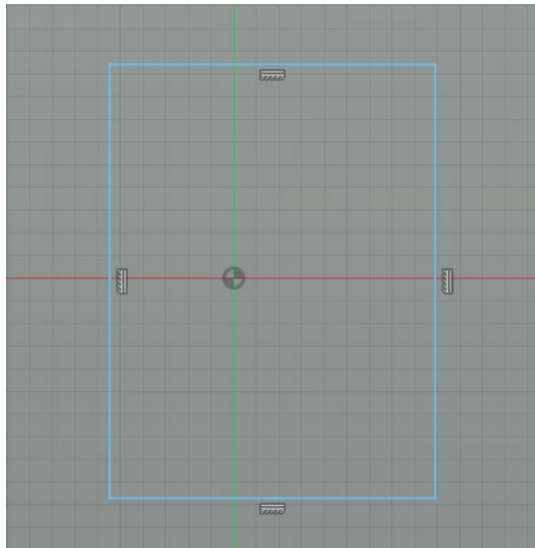
- f) Select “Finish Sketch” then select Extrude and choose the newly created profile. Enter -2.25 as the depth, and finish the extrusion by selecting “OK”



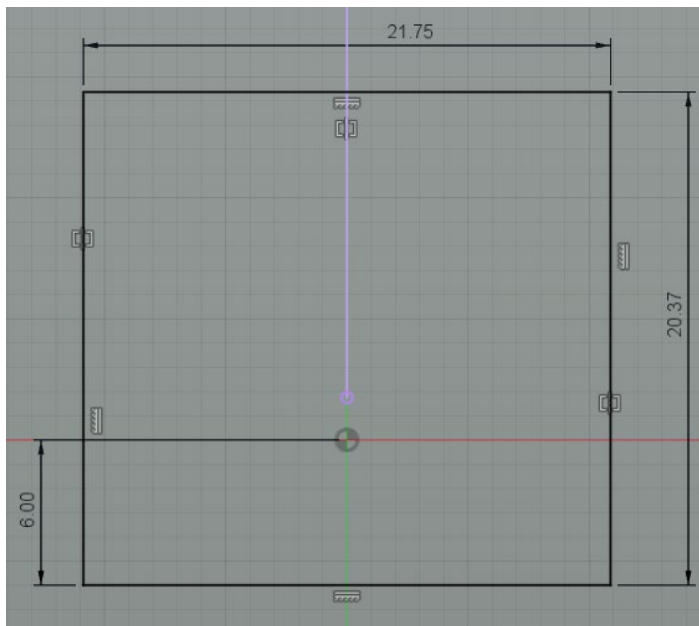
- 4) Save and close the file. Open Part 5 (the newly made copy).

Part 5

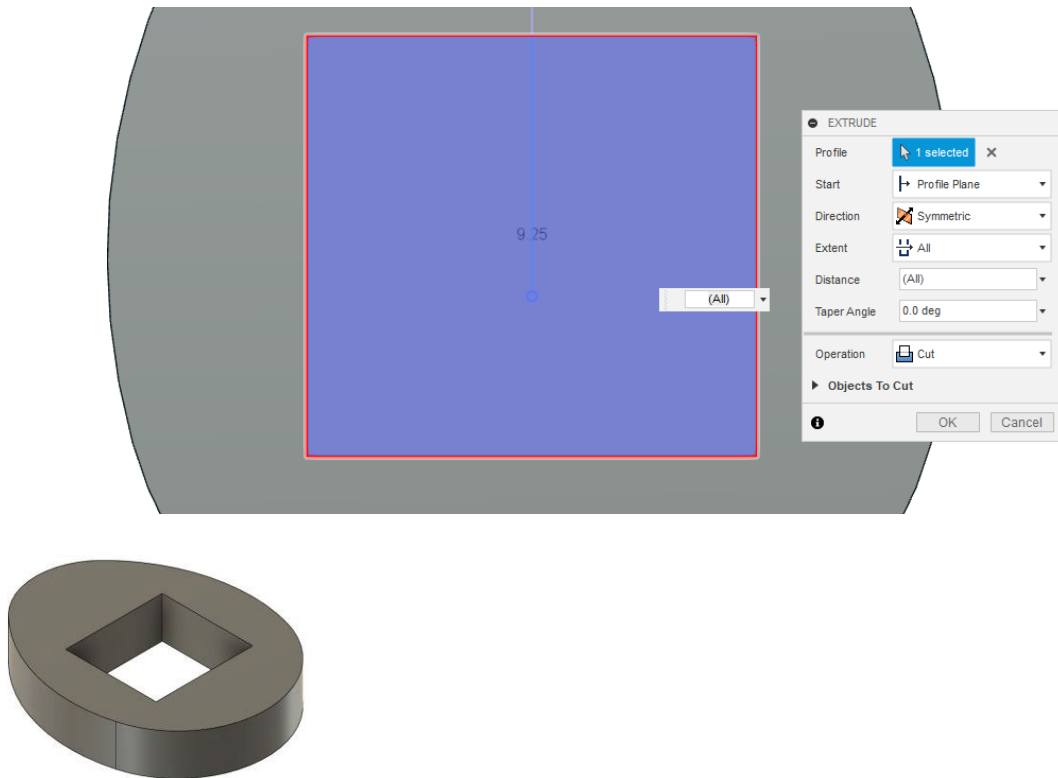
- 1) Make a rectangular cut in the part
 - a) Start a new sketch on the front (flat) face of the body, parallel to the XZ plane
 - b) Using the Rectangle tool, draw a rectangle that surrounds the origin
 - c) From the “Constraints” tab, select the Symmetric Constraint . This constraint requires you to select 3 things, and it makes the first two things you select symmetric around the third. In this case, first select the two vertical sides of the rectangle and then select the vertical (Z) axis from the origin tab under the browser on the left.



d) Dimension the rectangle as follows:



- e) Press Finish Sketch and select the Extrude tool
- f) Select the rectangular profile to extrude, under “Direction” change One-Side to Symmetric and under “Extent” change Distance to All. This should change the Operation to Cut. Press OK to finish the extrusion.



g) Save the file and close it.

Part V. Submission Requirements

- Make sure that all your sketches are fully constrained i.e. no blue lines, all black.
- 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 2” assignment title and use the “Write Submission” button to type “Ready for Grading”. Please do not go back and change your files in the Fusion Lab 2 folder. You can copy files into Lab 3, but other than this, leave Lab 2 files as they are.