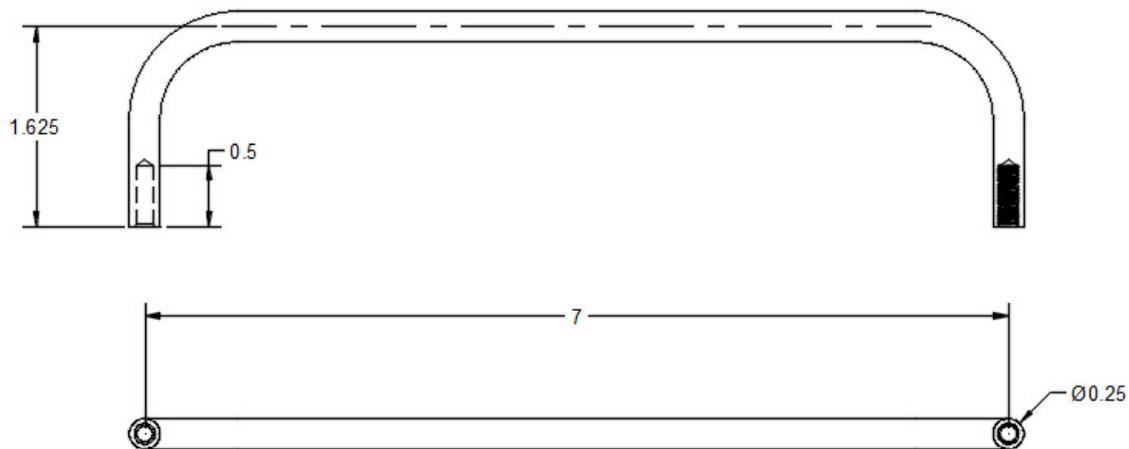


In this lesson you will learn about the **Sweep** tool. Sweep is similar in function to the Extrude and Revolve tools, which all create 3-D objects from 2-D sketches. Extrude extends a sketch in a linear direction. Revolve extends a sketch by rotating it around an axis. Sweep extends a sketch by pulling it along a path. If the path is a straight line, then the result is the same as an extrude. However, if the path is curved, then the 3-D object will be curved.

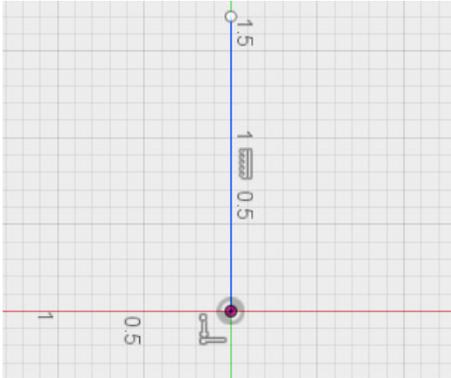


You will be creating a drawer handle like the one illustrated to the left. The procedure requires the drawing of two sketches: 1) a sketch of the path for the sweep and 2) a sketch of the cross section of the handle, which is a circle. The image below provides the dimensions of the handle. Note that the ends of the handle have threaded holes to accommodate screws.

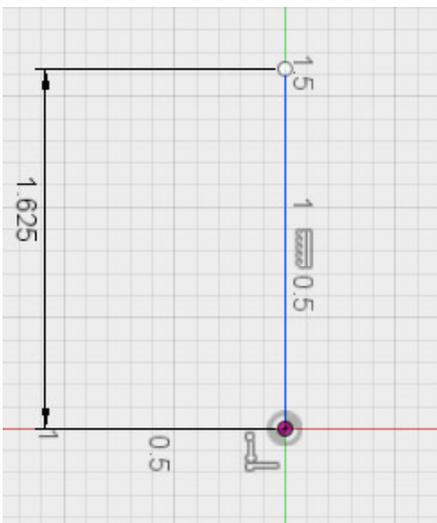


1. Open Fusion 360
2. Create a new sketch on the top plane

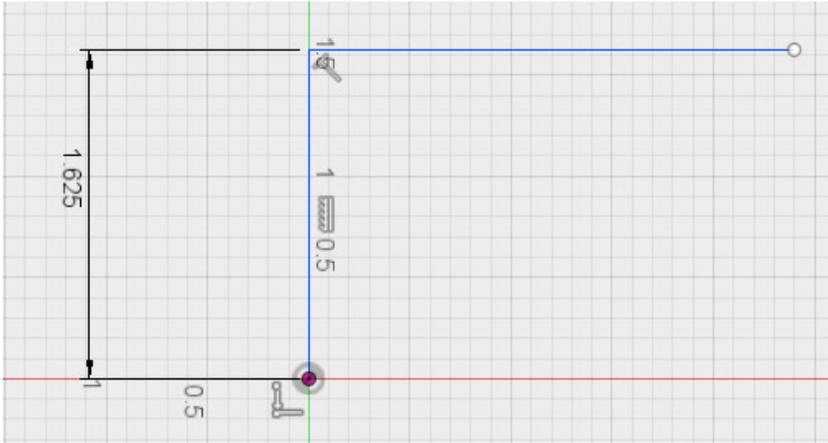
- Now begin drawing the sweep path of the handle. Select the **Line** tool. Draw a vertical line as illustrated below. The bottom end-point of the line should be at the center point of the top plane. The length is not important as you will use the Sketch Dimension tool later. But make sure the line is vertical (it should have the vertical constraint symbol after you finish drawing it).



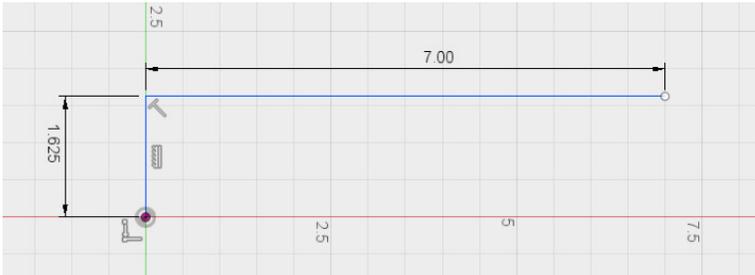
- Use the **Sketch Dimension** tool to make the length of the line 1.625 inches.



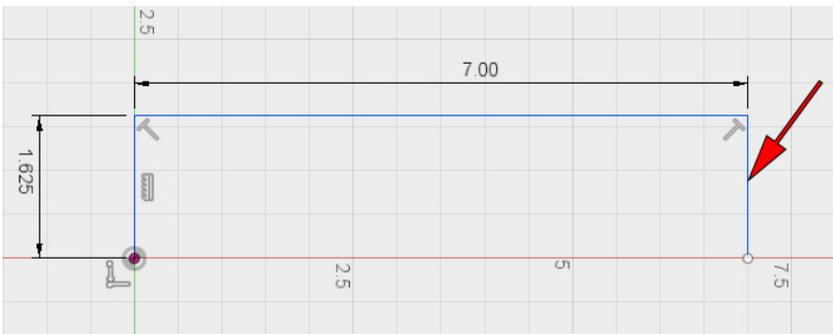
- 5. Draw a horizontal line like the one illustrated below. You will set the length in the next step.



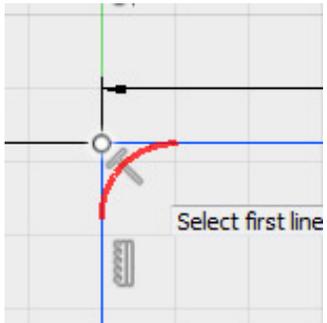
- 6. Use the **Sketch Dimension** tool to make the horizontal line 7.00 inches long.



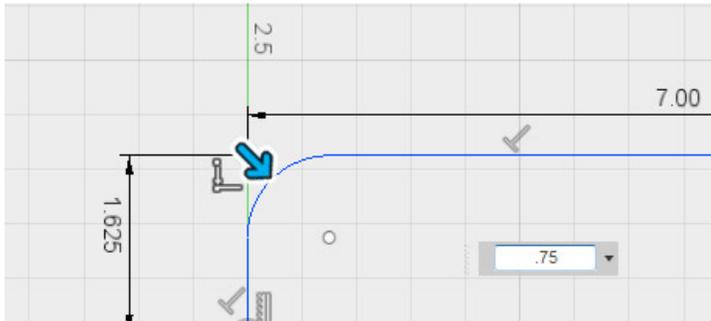
- 7. Draw a vertical line like the one illustrated below. The top end of the line should be coincident with the end-point of the horizontal line and the bottom end should be coincident with the horizontal axis line (red line). You now have the general outline of the sweep path for the handle. However, the path must be curved at each intersection of a vertical line with the horizontal line.



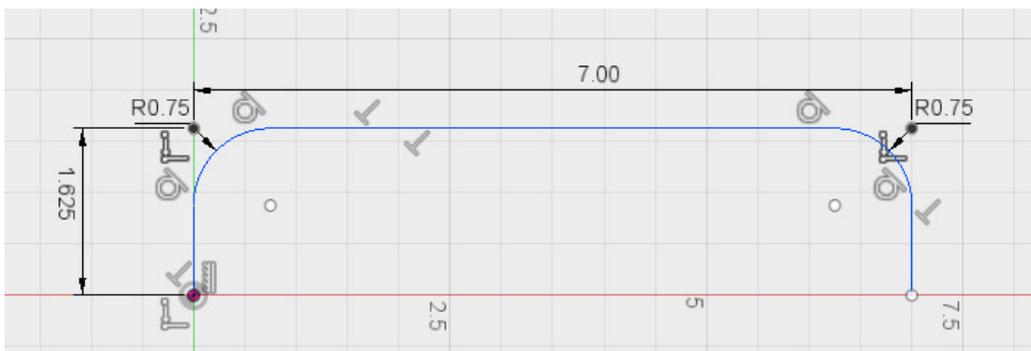
8. Select the **Fillet** tool in the Sketch menu. Move the mouse towards one corner of the sketch until a red curve appears as illustrated below. Then click on the mouse.



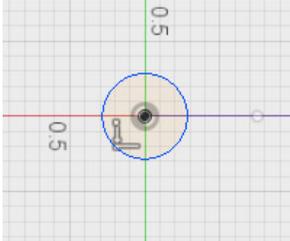
9. A dimension box should appear on screen as illustrated below. Enter a value of 0.75 to make the radius of the fillet 0.75 inches. Then press the Enter key to set that value. Repeat this procedure for the other corner.



10. Your sketch should now look similar to the one illustrated below. You have completed the shape of the sweep path for the handle. Click on the **Stop Sketch** button.



11. Create a new sketch on the front plane. Use the **Center Diameter Circle** tool in the Sketch menu to draw a circle centered on the center point of the front plane as illustrated below. The size does not matter.



12. Use the **Sketch Dimension** tool to set the diameter of the circle to 0.25 inch. You have now completed the profile of the handle. Click on the **Stop Sketch** button.



13. Select the **Sweep** tool in the Create menu. Click on the circle you created in steps 11 and 12 to select it as the Profile to sweep. It should now look like the image below. In the SWEEP dialog box click on the Path Select button. Then click on the line that represents the sweep path (the line with red arrow in illustration below).



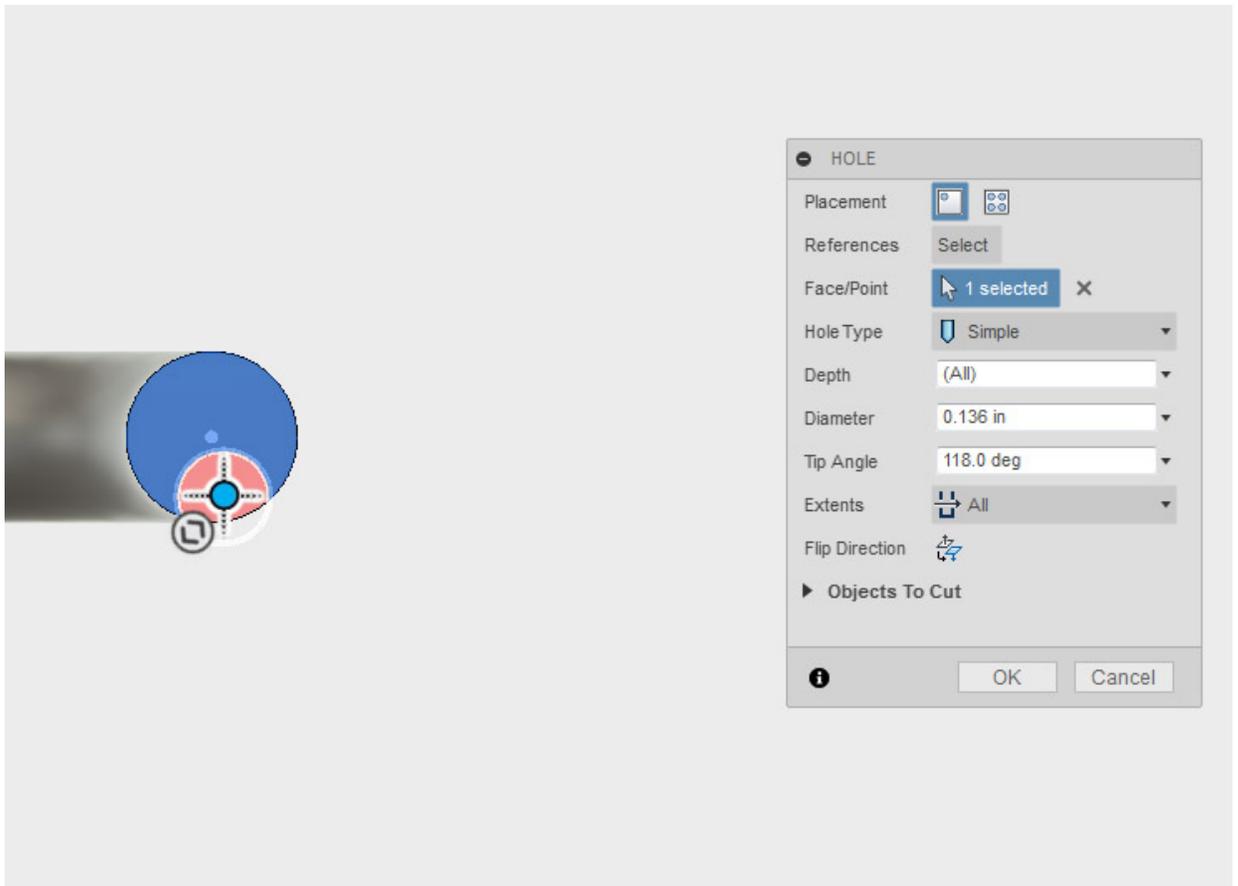
14. You should now have the 3-D object on your screen as illustrated below. You have created a 3-D object with the Sweep tool! Click on the **OK** button in the SWEEP dialog box to complete the operation.



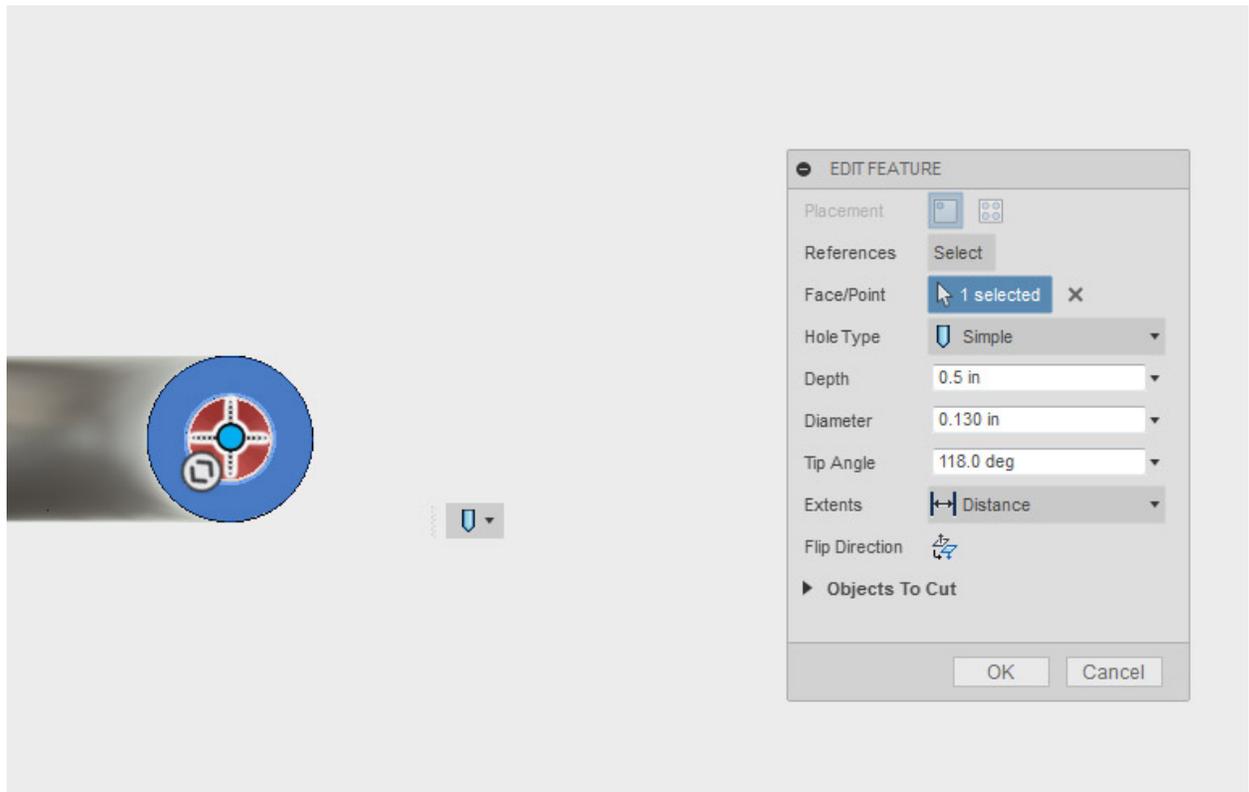
15. Keep your view as it is in the illustration above (Front view). If you were to change the view to "Home" it would look like the illustration below. You now have a handle but without the holes in the ends, which accommodate screws for attaching the handle to a drawer. Now you will work on making the required holes with threads to fit screws.



16. Zoom your view in so that the end of the handle looks something like the one below (without the hole appearing). Select the **Hole** tool in the Create menu. Click on the circle face to select it for the hole procedure. You should now have the hole tool appearing in the circle (the pink colored circle). Grab the middle of the hole-circle with the mouse and move it around until you can see the center point of the end of the handle (the light blue dot). Move the hole with the mouse until it snaps on to the center point. This fixes the position of the hole. In the Diameter slot of the HOLE dialog box, enter a value of 0.136 but do not press the Enter key afterward (you are not finished with the dialog yet).



17. Click in the Depth slot and enter a value of 0.5 (the hole must be 0.5 inches deep). Now you are finished with the dialog. Click the **OK** button.



18. Repeat the procedures outlined in steps 16 and 17 to bore a hole in the other handle end.
19. You are almost ready to cut the threads for the holes. First you need to apply a chamfer to the edges of the holes. Select the **Chamfer** tool in the Modify menu. Then click on the edge of one of the holes to select it. A CHAMFER dialog box will appear. In the Distance slot set a value of 0.02 . Then click the **OK** button. Repeat this step for the second hole.



20. Your holes should be chamfered as illustrated below.



21. Select the **Thread** tool in the Create menu. A THREAD dialog box will appear. Click with the mouse on the inside surface of the hole to select it as the face to thread. Make sure that both the Modeled and Full Length boxes have check marks (click in them if they don't). In the Thread drop-down, select **ANSI Unified Screw Threads**. In Size select **0.164(#8) in**. The Designation should be **8-32 UNC**. Class should be **3B** and Direction should be **Right hand**. When all of these are set, click on the **OK** button. You now have a threaded hole that will fit a # 8 screw with 32 threads per inch. Repeat this procedure for the other hole to complete this lesson!

