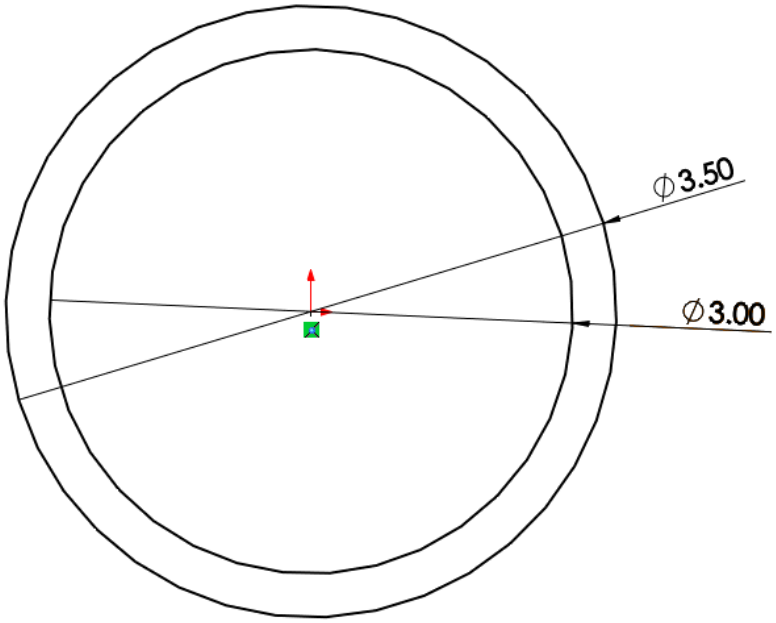
# SOLIDWIZE

## ONLINE SOLIDWORKS TRAINING

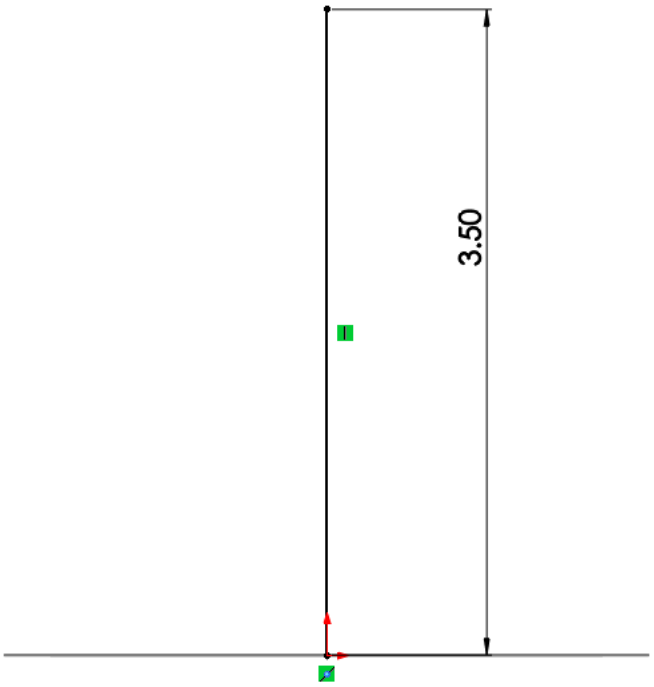
Simple Sweep: Coffee Mug

**Step 1: Creating the Base Profile and Sweep Path**

Using the **Inches** as the unit, create the following sketch on the **Top Plane**. Use the **Circle** tool centered on the origin.

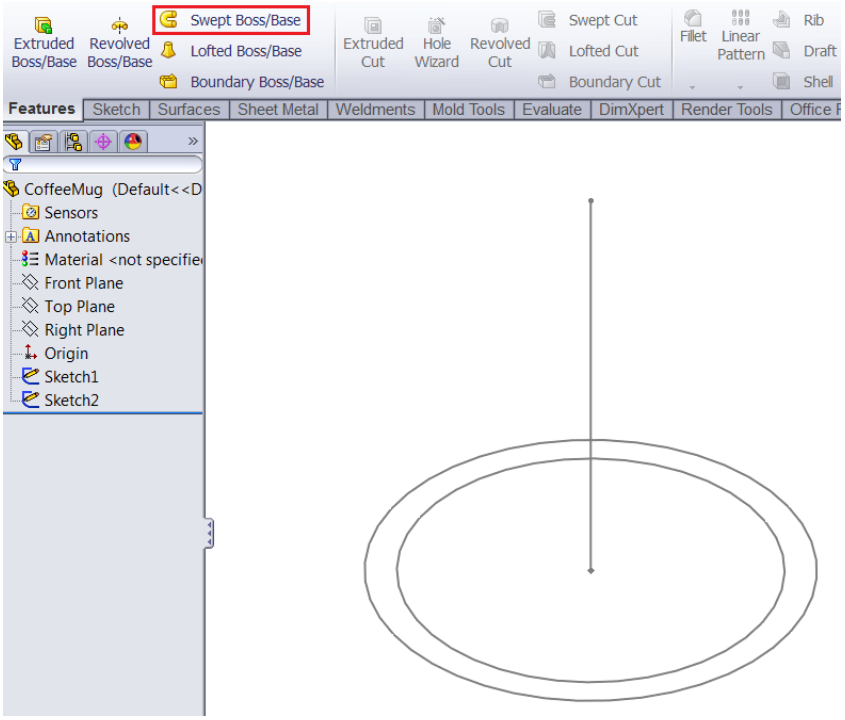



On the **Front Plane**, create the following sketch with the end-point coincident to the origin.

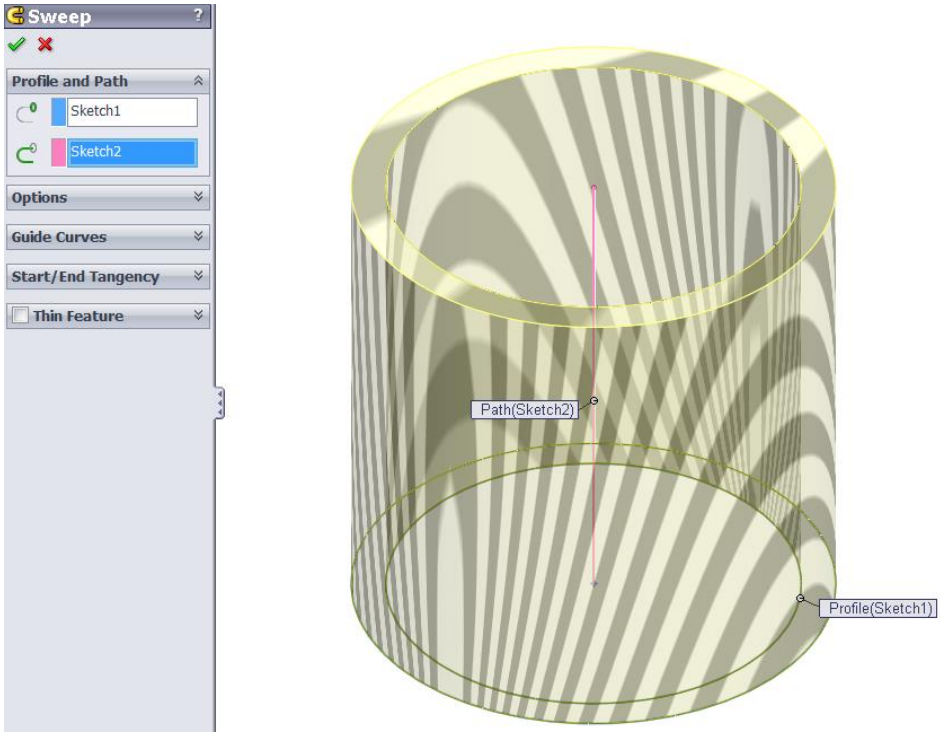


**Step 2: Sweeping the Profile**

Exit the sketch. Select the **Swept Boss/Bass** tool from the **Features** tab.




Use **Sketch1** as the sweep profile and **Sketch2** as the swept path as shown below:

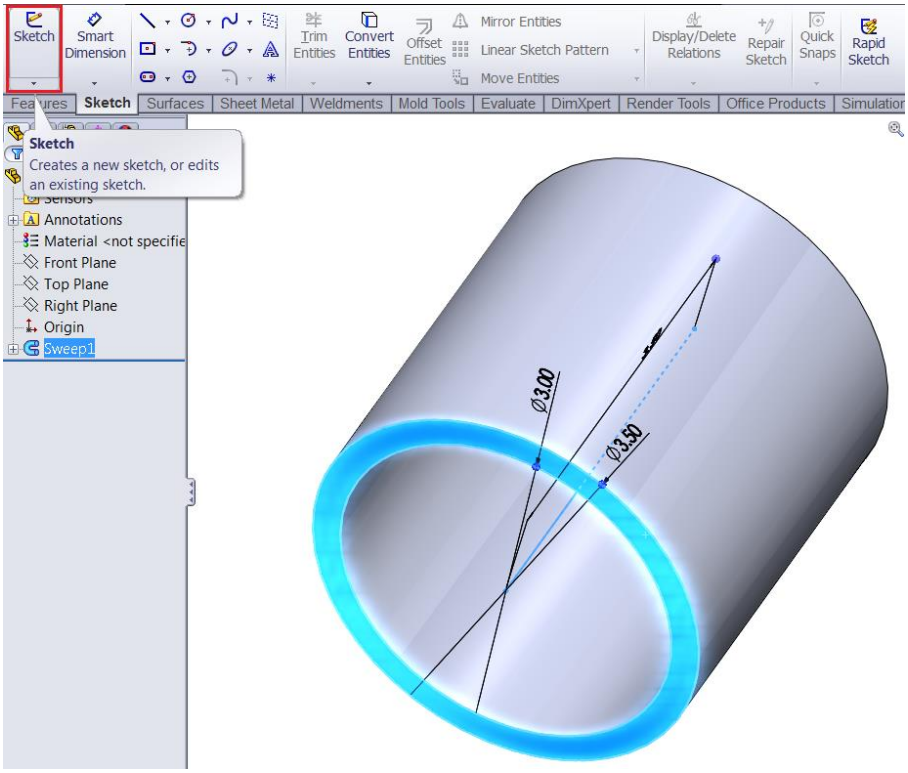


The result should look like this:

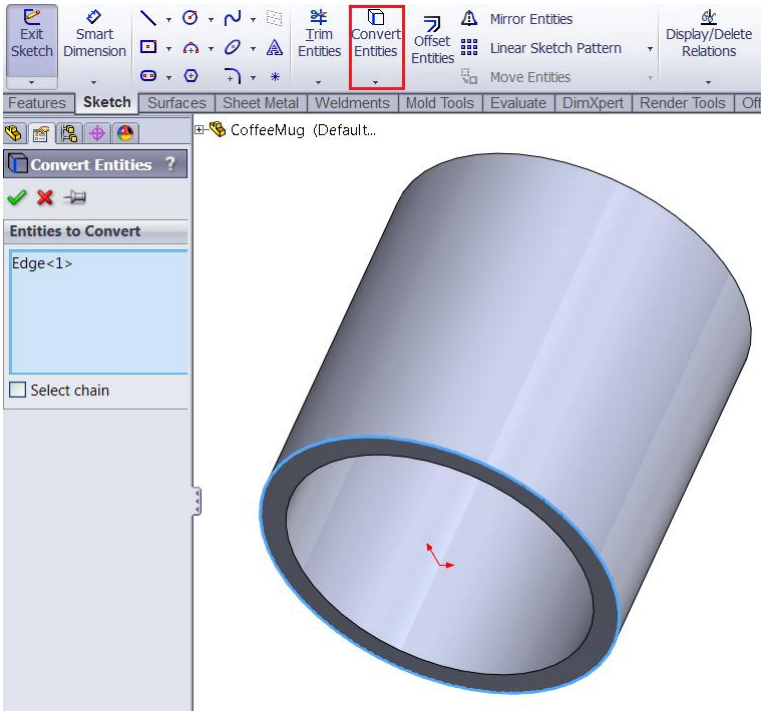
**Note:** This feature could have be created using an extrusion or a revolve tool. When creating parts, you will often have a variety of ways to create it.

**Step 3: Extruding the Base**

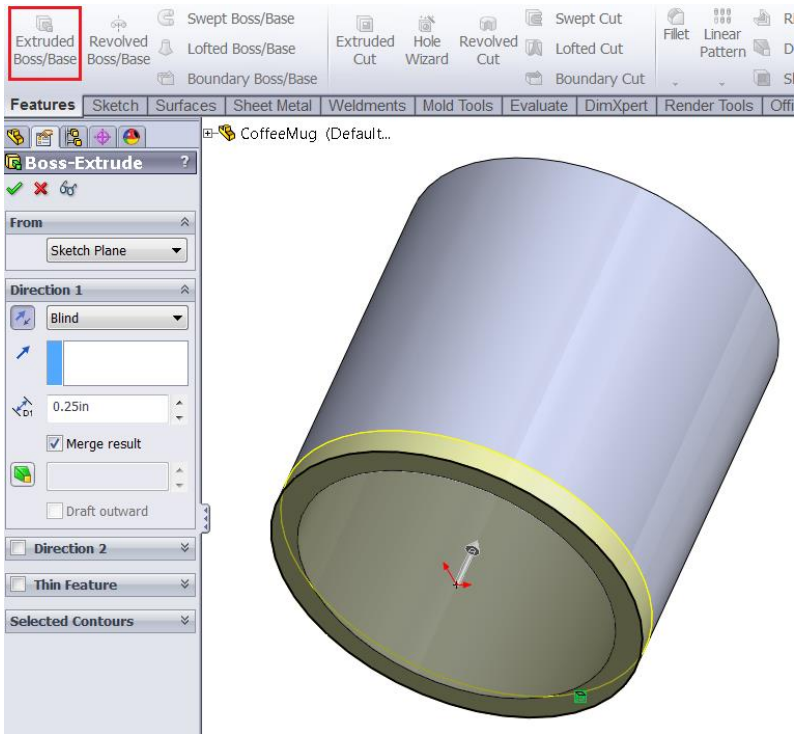
Create a sketch on the base of the swept feature. This will be the base of the mug. To create the sketch, select the bottom face and click **Sketch** in the **Sketch** tab.



Using the **Convert Entities** tool, convert the outer circle of the base as shown below:

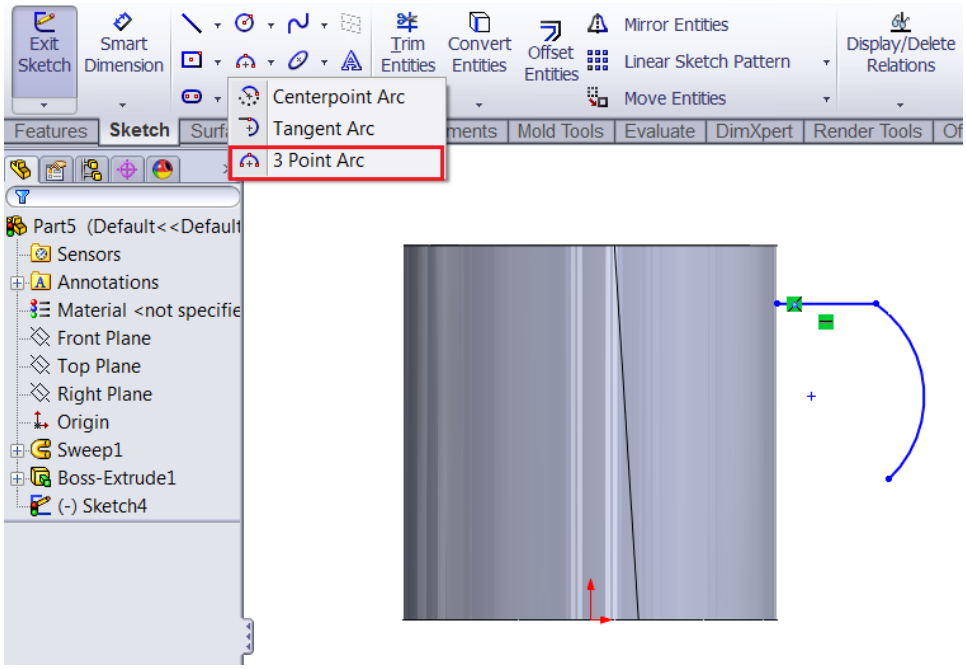



Using the **Extruded Boss/Bass** tool under the **Features** tab, extrude the sketch **0.25in** into the cylinder. Make sure the extrusion direction is correct.

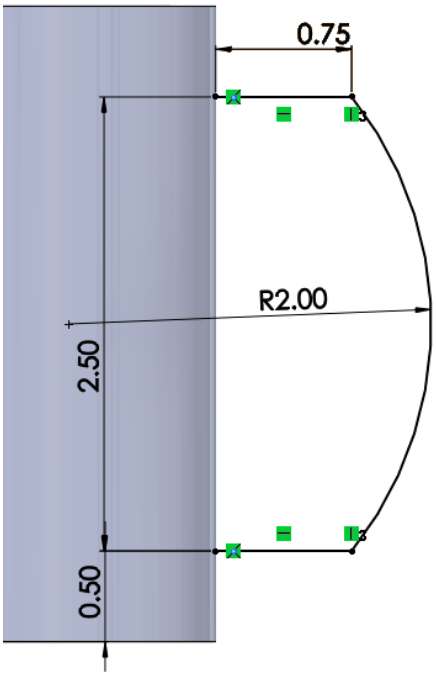


**Step 4: Creating the Handle: Sweep Path**

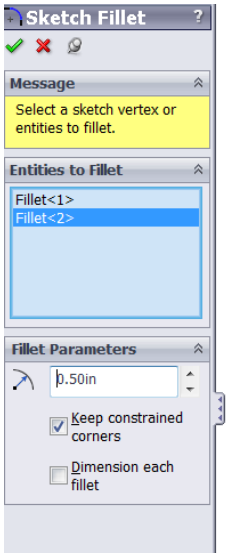
On the **Front Plane**, create a sketch as shown below. To create the **3 Point Arc,** use the **3 Point Arc** tool found in the **Sketch** tab as shown below:

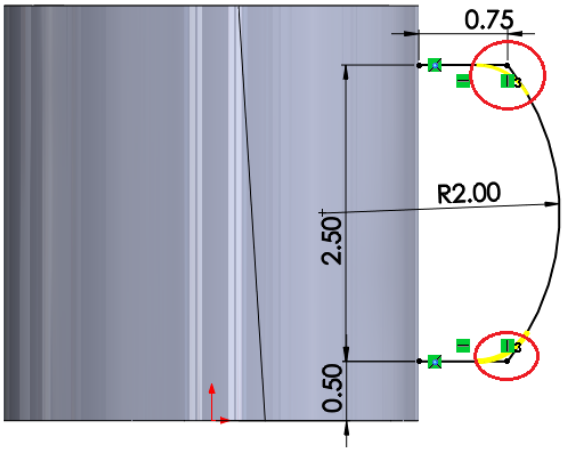



Complete the sketch as shown below. Pay attention to the dimensions and relations shown:



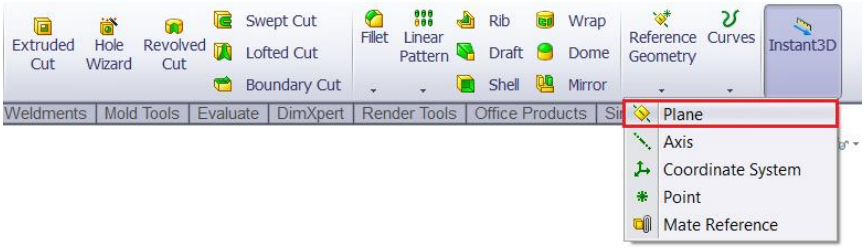
Using the **Sketch Fillet** tool, create fillets at the top and bottom corners where the horizontal line intersects the arc. Use a fillet radius of **0.50in.**

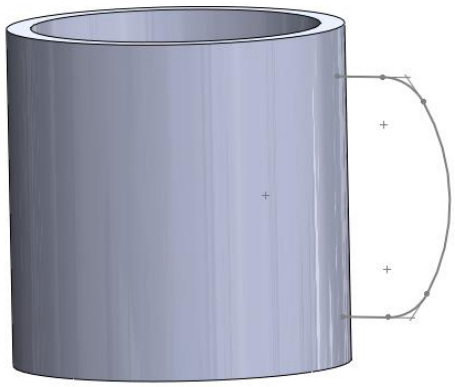


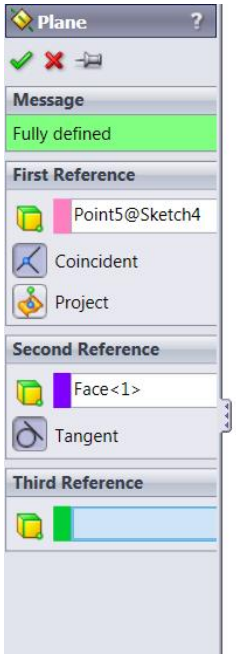

**Step 5: Creating the Handle: Sweep Profile**

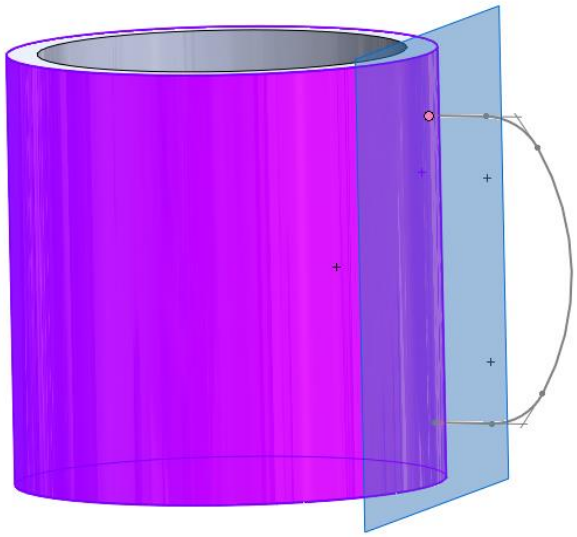
To create the profile, you have to create a plane normal to the path. To create plane, select **Reference Geometry > Plane** under the **Features** tab as shown below.



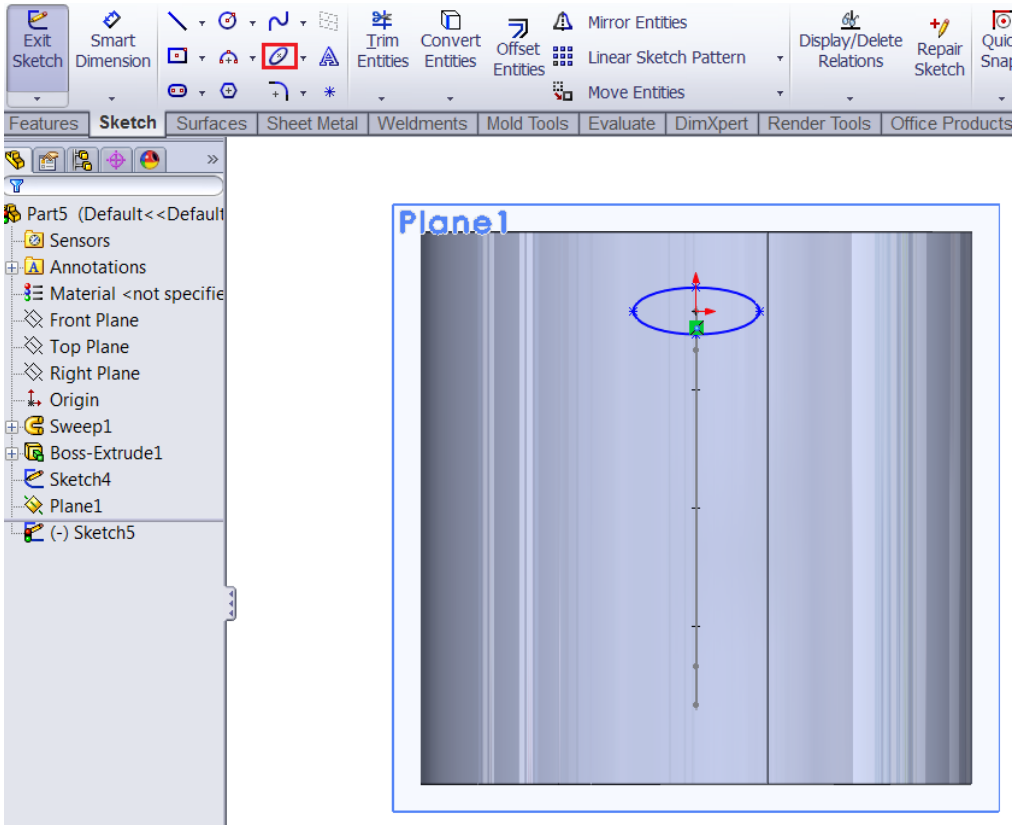



Use the end point of the path and the cylindrical surface as the references for the plane:

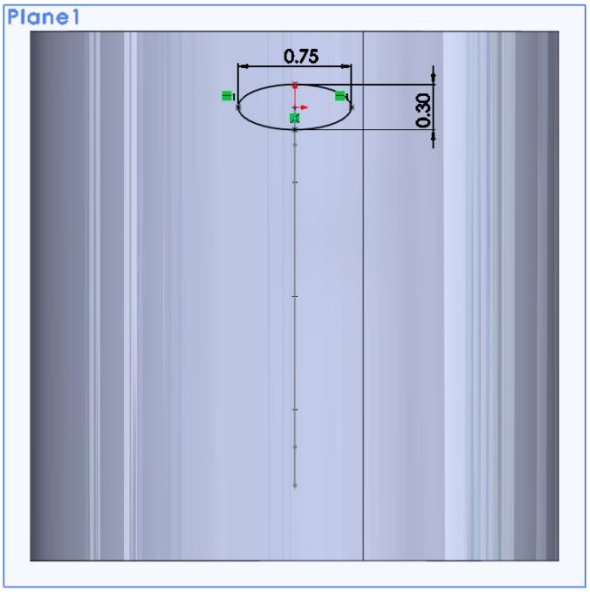


Create a sketch on the plane. Sketch and **ellipse** using the **ellipse** tool highlighted below:

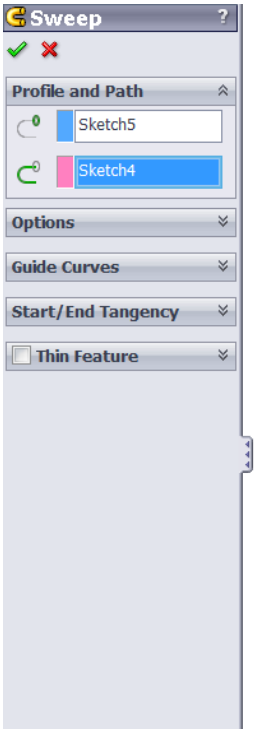


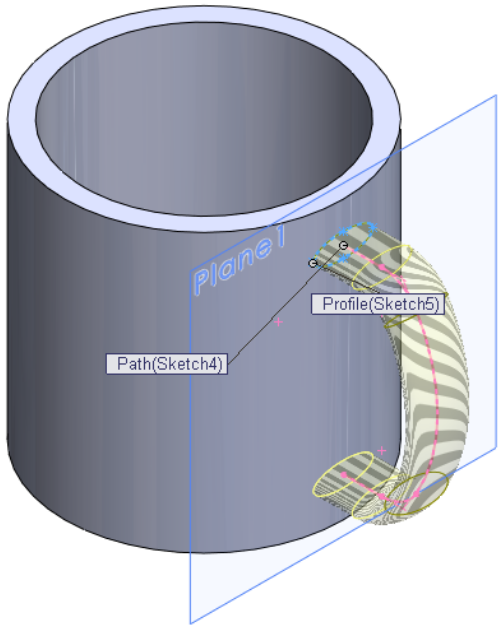
Add dimensions and relations to the ellipse as shown below. Note the horizontal relation between the two ends of the major axis of the ellipse:



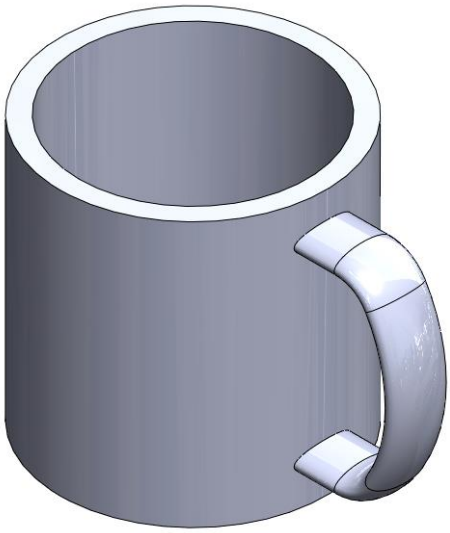

**Step 6: Creating the Handle: Sweep**

Create a sweep using **Sketch5** as the profile and **Sketch 4** as the path.



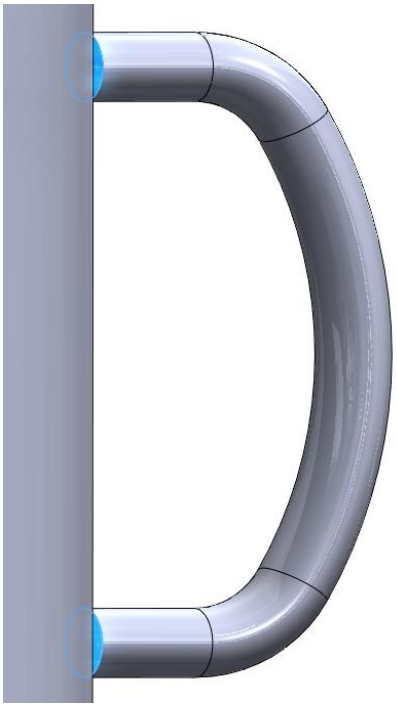
The result should look like the following



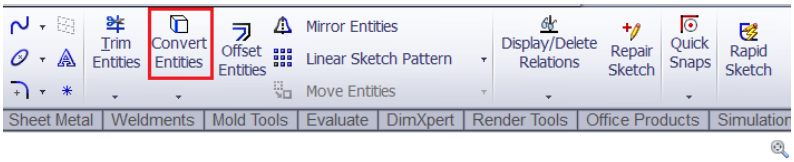
**Step 7: Creating the Handle: Clean-up Extrusion**

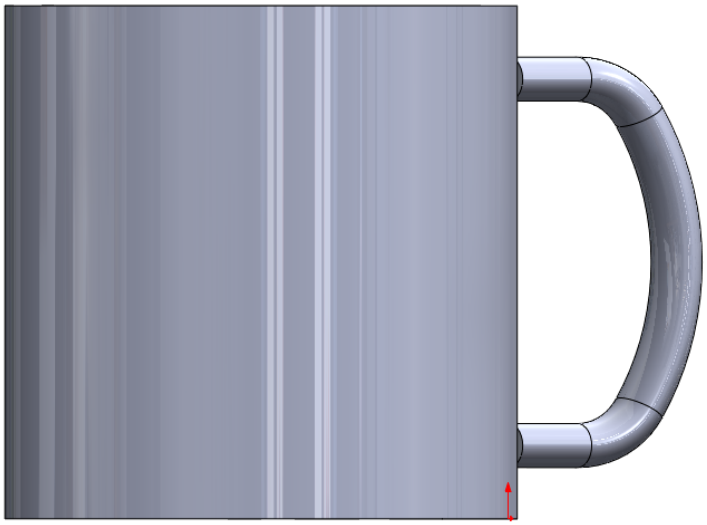
If you look carefully, since we sketched the swept profile from a plane, the end faces of the handle are actually tangent to the cylindrical surface of the mug which leaves a gap and results in a multi-body part. We want the sweep profile to go all the way against the cylindrical surface.

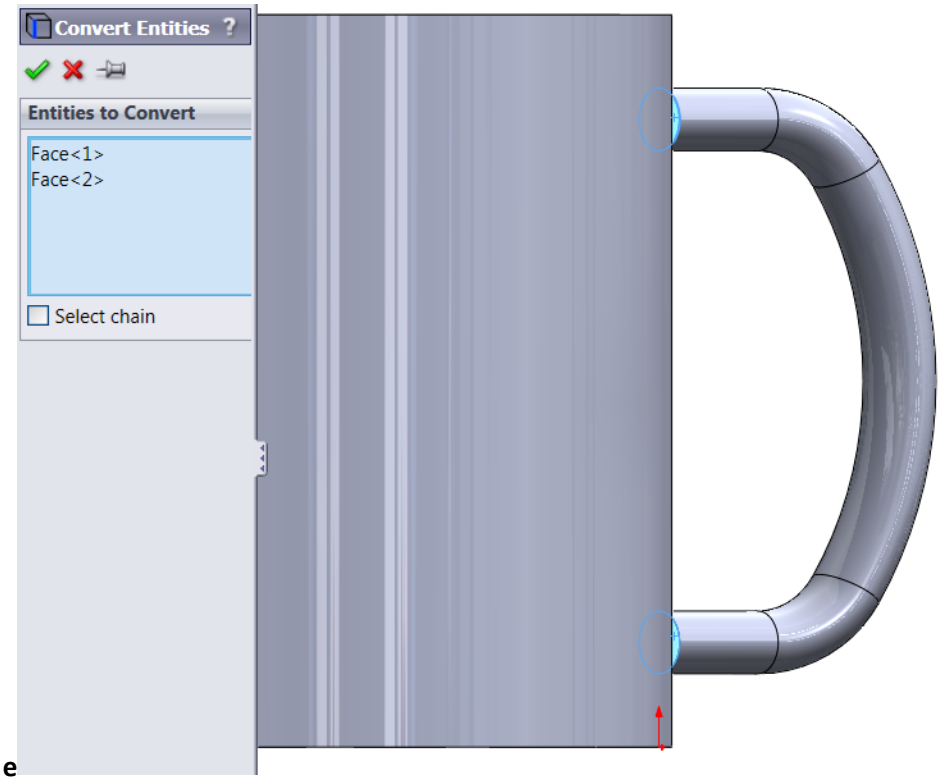
Create a sketch on either of the highlighted faces below:



Using the **Convert Entities** tool, convert the two ellipses into the sketch.



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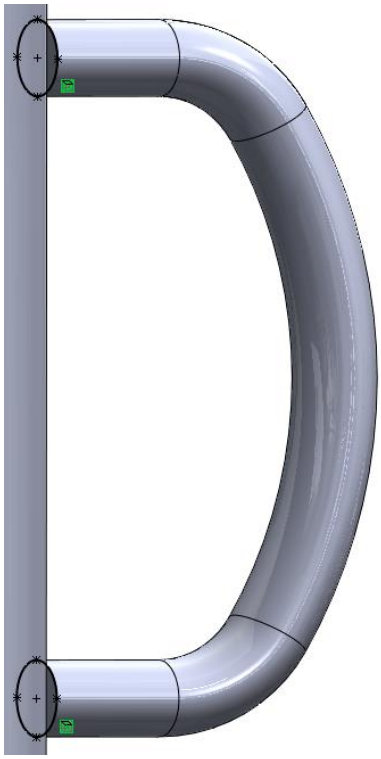
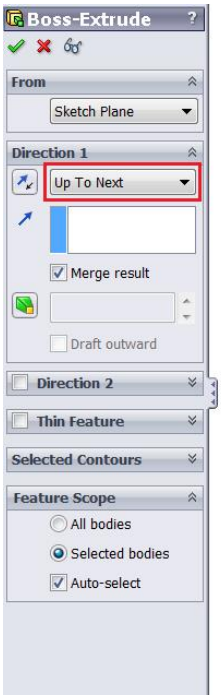
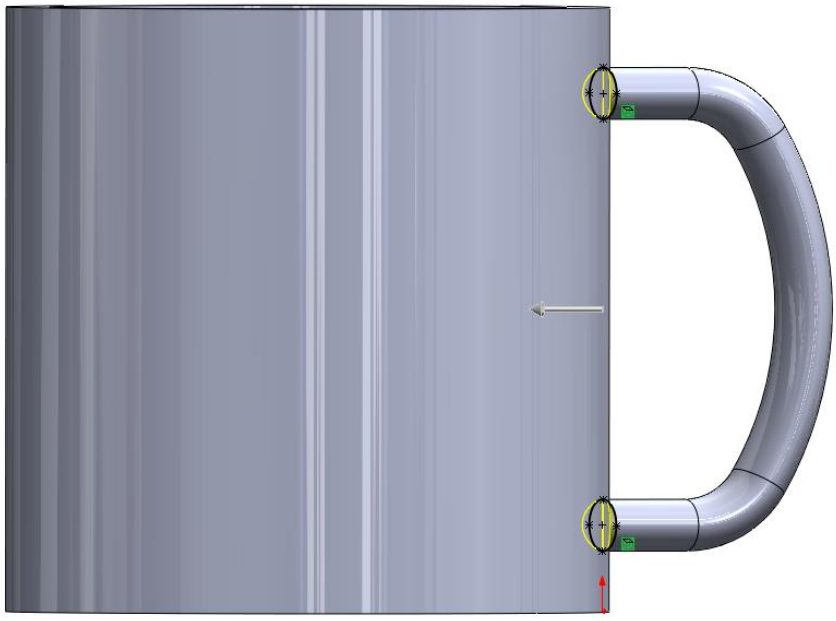
The sketch should look like the following:

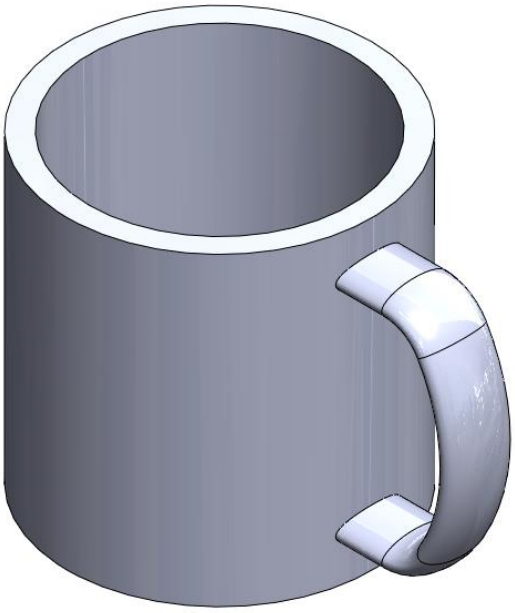


Using the **Extruded Boss/Bass** tool, extrude the sketch up to the cylindrical surface. Use the **End Condition "Up To Next"** as shown below:



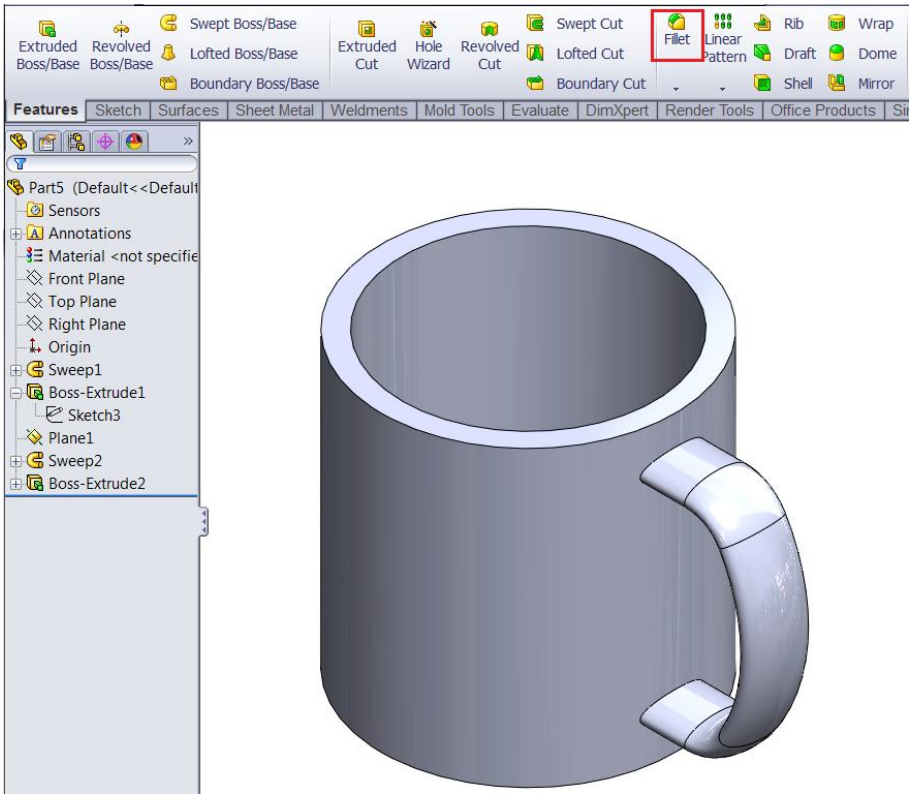



The result should look like the following:

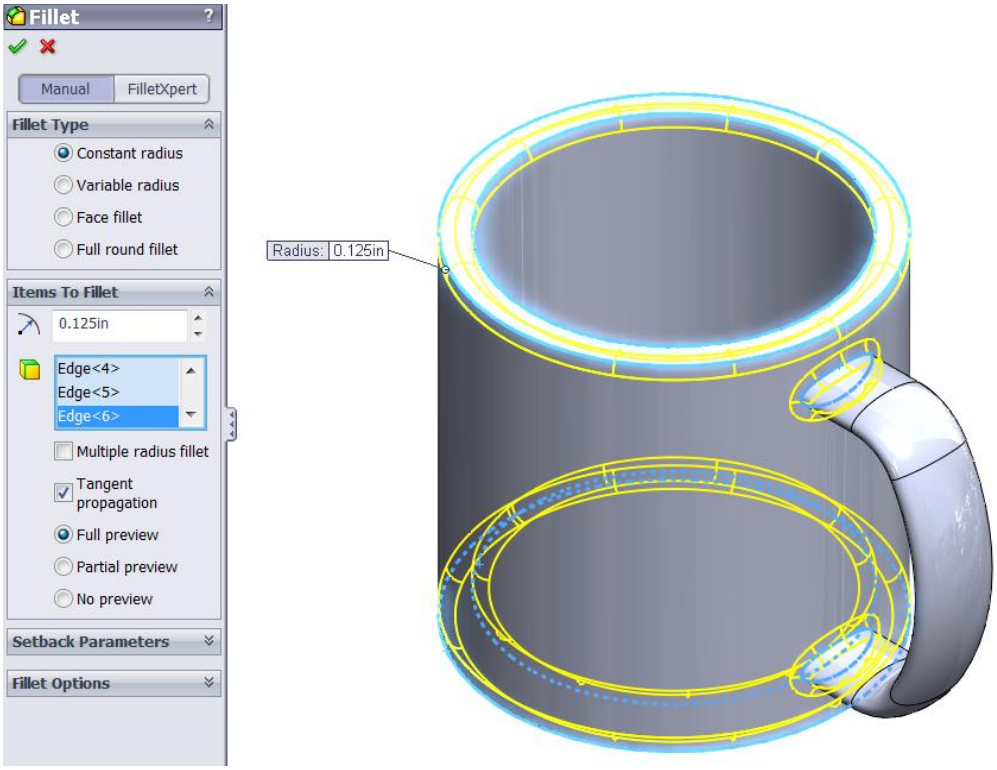


**Step 8: Adding Fillets**

Select the **Fillet** tool found under the **Features tab**.



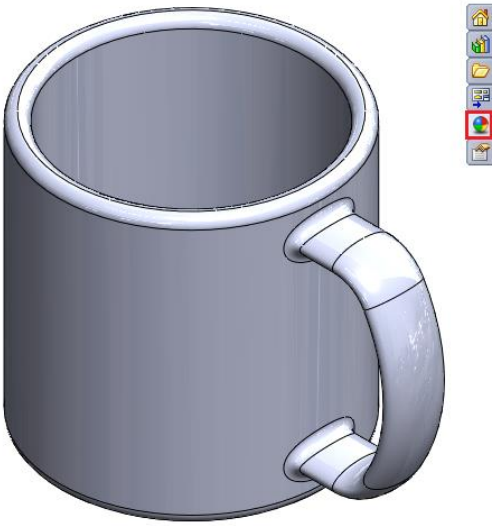
Add **Fillets** to all edges of the mug including the bottom inner edge. Use a fillet radius of **0.125in.**



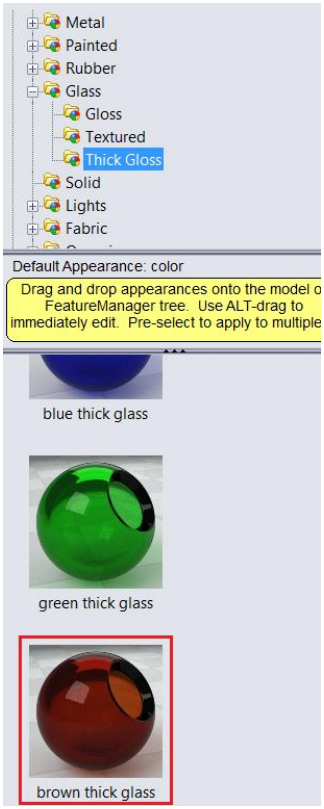

The result should look like the following:

**Step 9: Adding Appearances**

To add an appearance to the part, select the **Appearance** tab from the menu located on the right of the display pane:

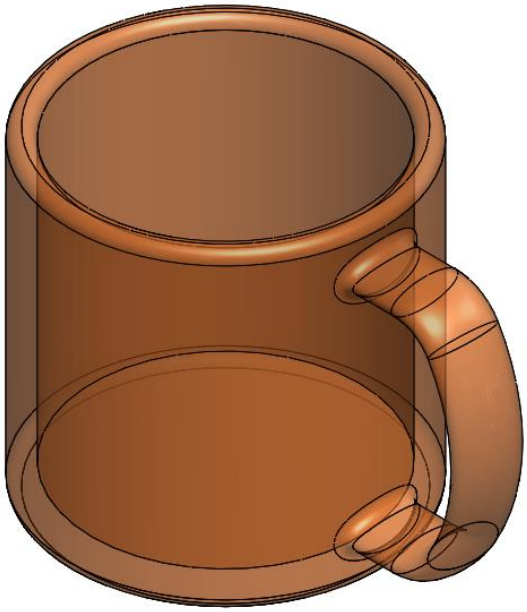


Select **Appearances>Glass>Thick Gloss** and select **Brown Thick Glass.**



To add the appearance, simply drag the **Brown Thick Glass** image onto the display pane. By dragging it right onto the display pane (the blank area around your part), the appearance is automatically applied to the entire part. If you wanted to apply the appearance only to a particular face, feature, or body, drag the appearance onto the desired face and select the desired options from the popup menu.

The resulting part should look like this:



**Step 10: Save and Exit**

Save the part as **Simple_Sweep_Mug.sldprt** and exit the part.