

SOLIDWIZE

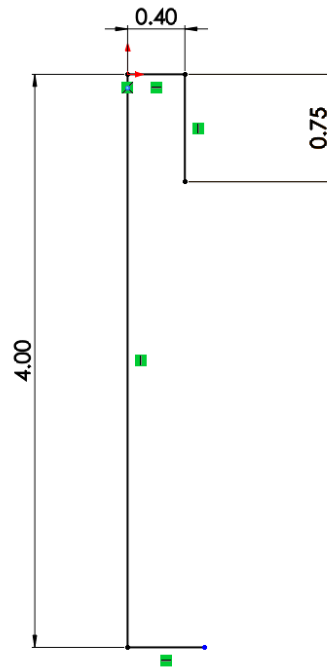
ONLINE SOLIDWORKS TRAINING

Simple Sweep: Head Scratcher

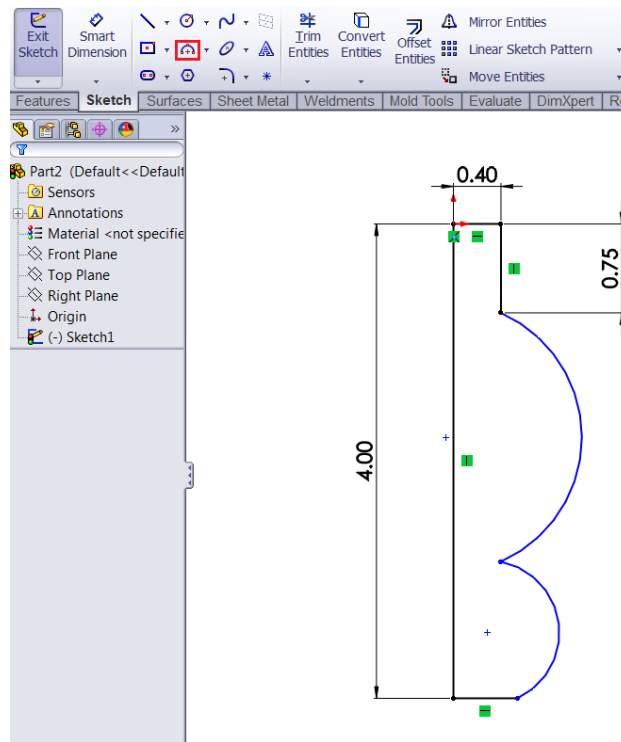


Step 1: Creating the Handle: Sketch

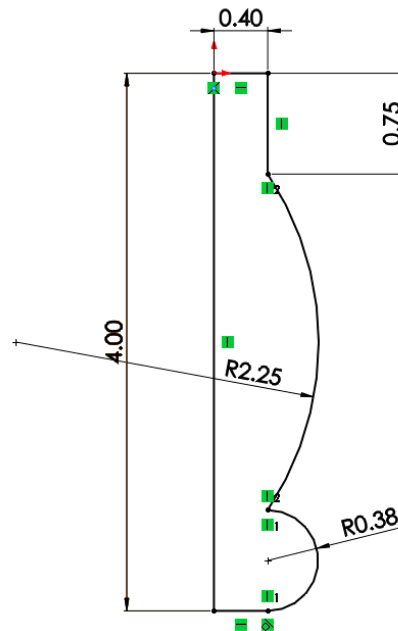
Using **Inches** as the unit create a sketch on the **Front plane**. Start with the sketch shown below:



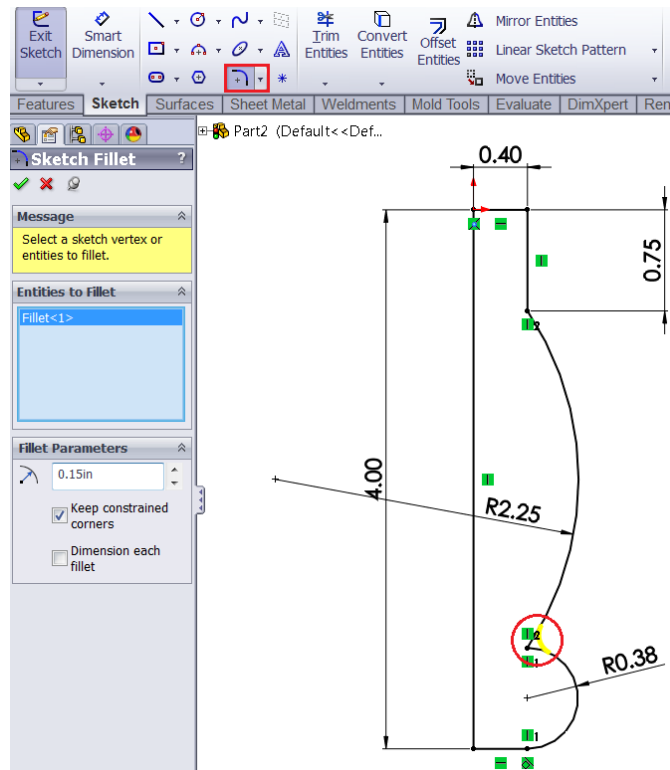
Create two **3-Point Arcs** using the **arc tool** found in the **Sketch** tab.



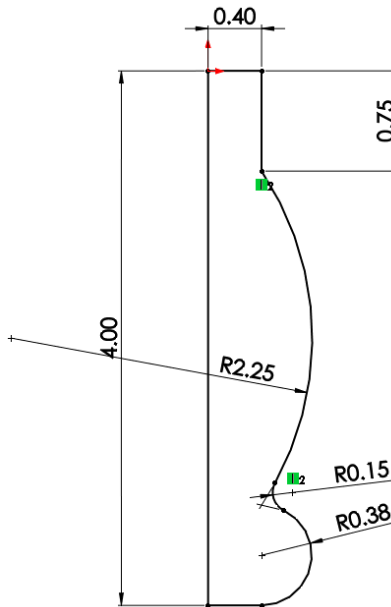
Add the following dimensions and relations to the sketch. Notice the tangent relation between the bottom horizontal line and the arc, as well as the Vertical relations between the arc end-points.



Add a **0.15in** radius **Sketch Fillet** to the highlighted vertex:

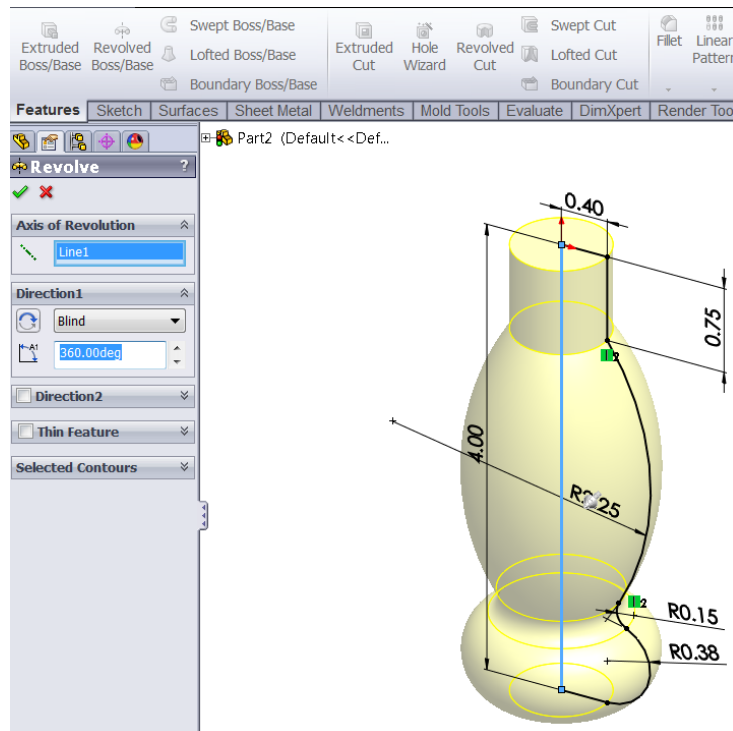


The resulting sketch should look like the following:



Step 2: Creating the Handle: Revolve

Using the **Revolved Boss/Bass** tool found under the **Features** tab, revolve the sketch using the vertical line (highlighted below) as the axis of revolution:

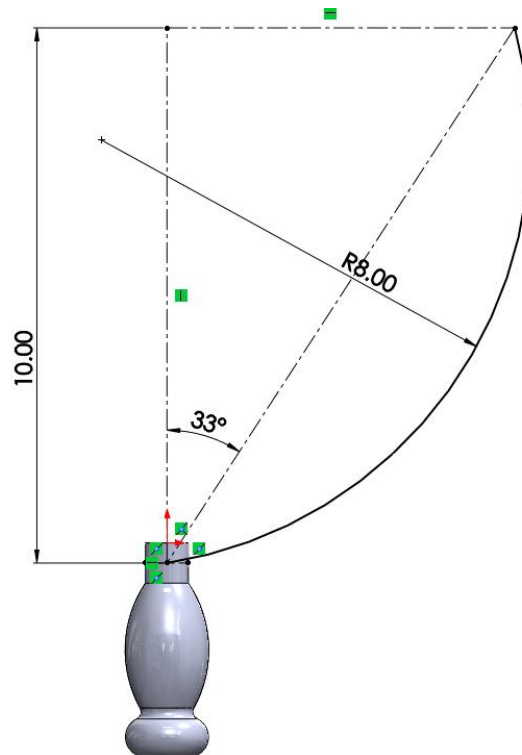


The revolved handle should look like the following:



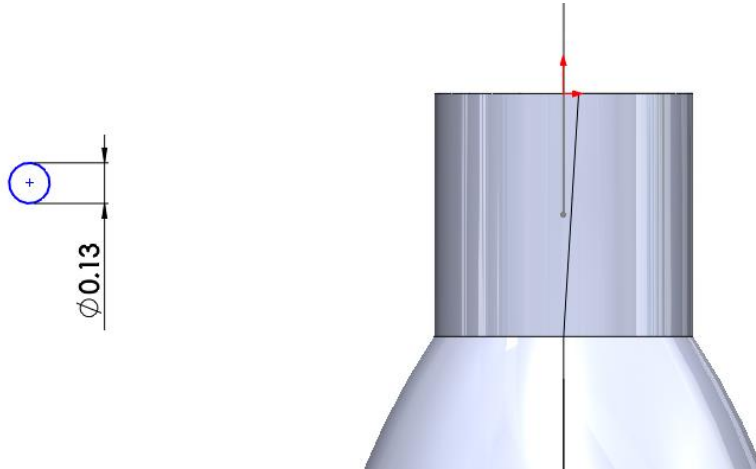
Step 3: Creating the Long Legs: Sweep Path

Create a sketch on the **Front plane**. Pay attention the dimensions and relations shown below. Hint: use a construction line running from the mid-points of the cylindrical face edges. The arc begins from the mid-point of the construction line.

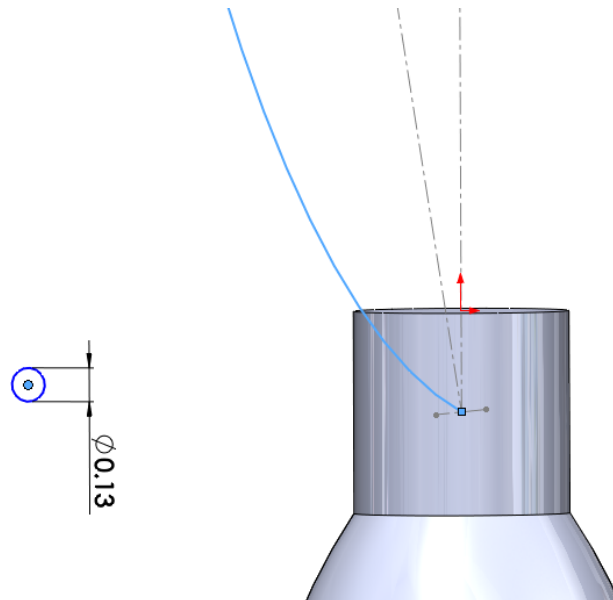
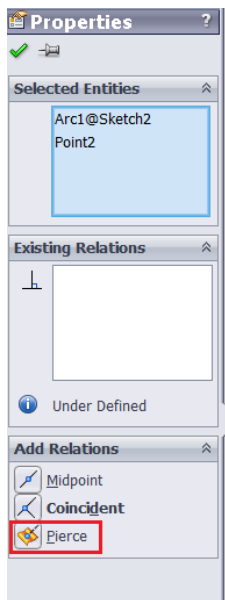


Step 4: Creating the Long Legs: Sweep Profile

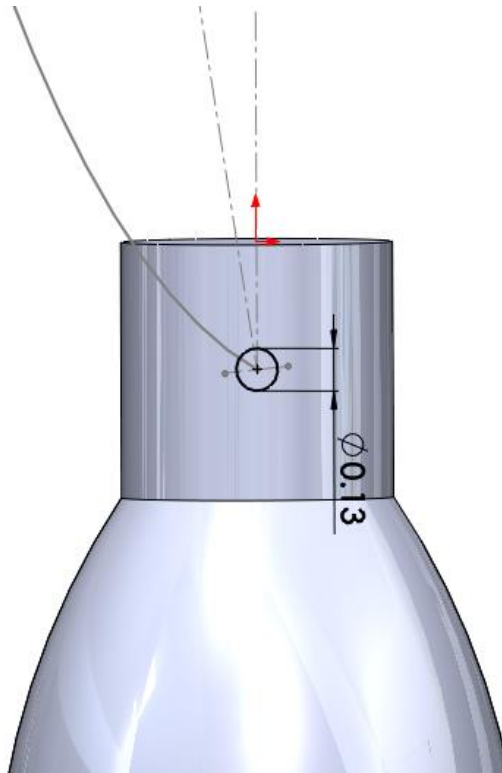
Create a sketch on the **Right plane**. As we will add a pierce relation later on, the sketch can be created anywhere in the display pane.



To add the **pierce** relation, select both the **Center** of the circle and the sketch of the sweep path created in **step 3**. To select both entities, hold **ctrl** while selecting. In the property manager, under “**Add Relations**,” select **Pierce**.

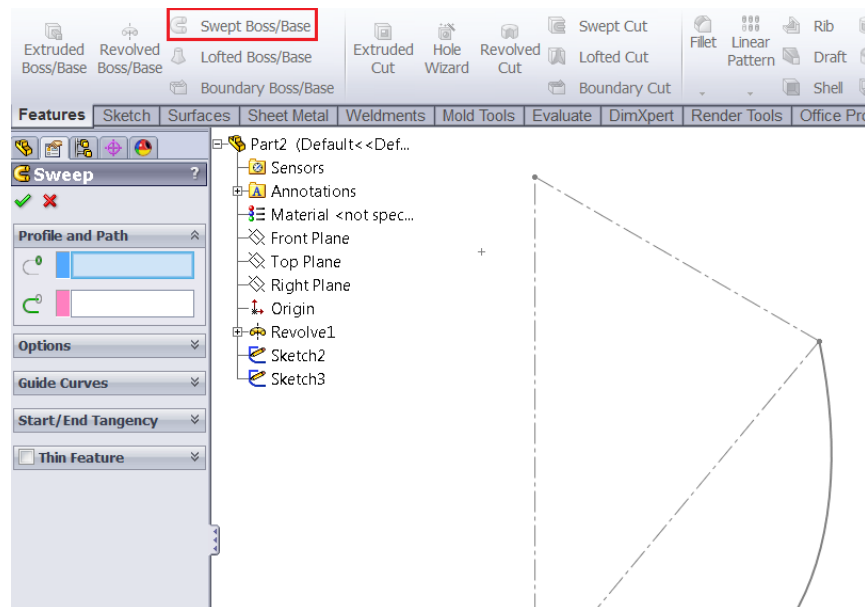


The resulting sketch should look like the following:

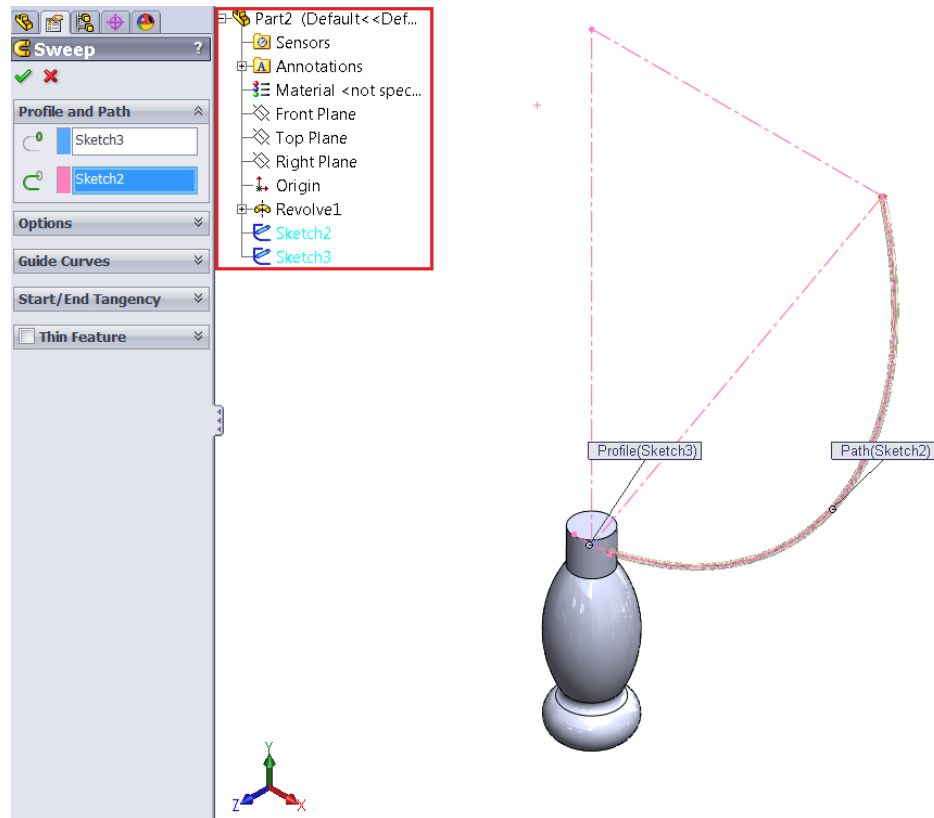


Step 5: Creating the Long Legs: Sweeping the Leg

Select the **Swept Boss/Bass** tool found under the **Features** tab. You must exit the sketch first if have not yet done so.



Expand the **Design Tree (boxed below)** by clicking on the “-” sign highlighted below. Select **Sketch3** as the sweep profile and **Sketch2** as the sweep path.

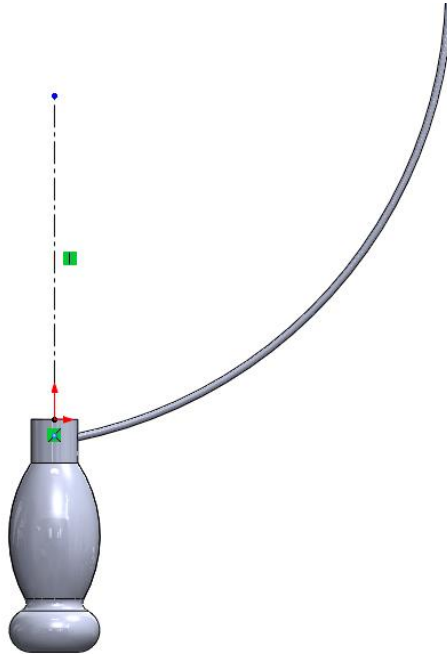


The result should look like the following:

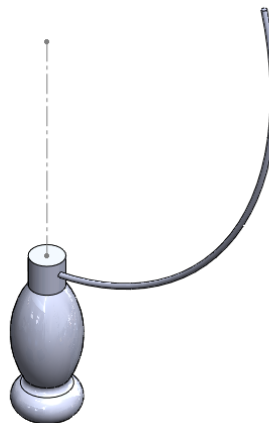
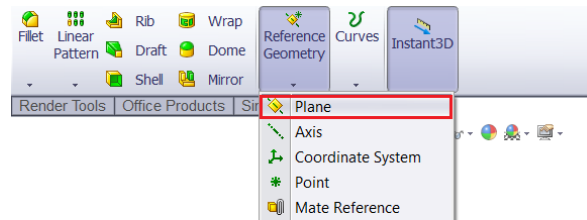


Step 6: Creating Reference Planes

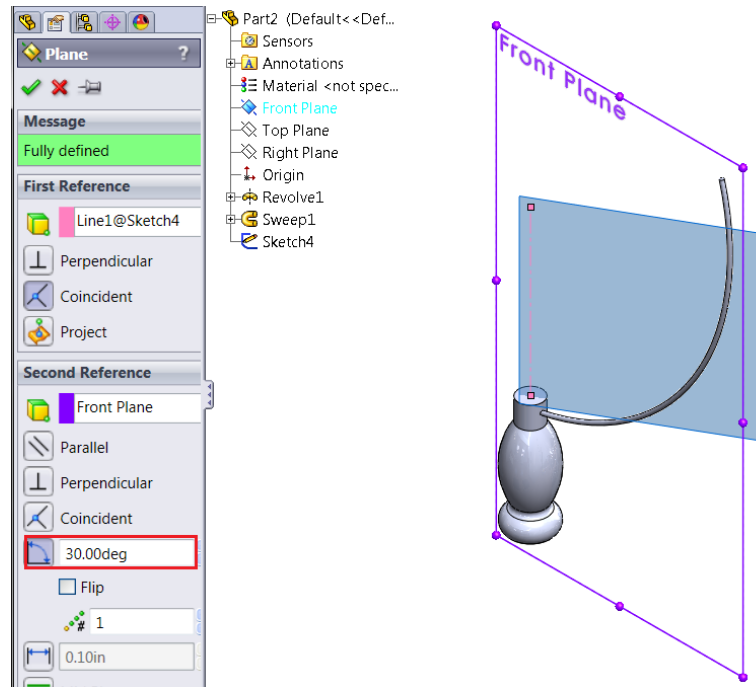
Since we want the shorter legs to be offset from the longer legs by 30 degrees, we will have to create a reference plane. First, create a sketch on the **Front plane** as shown below. The length of the construction line does not matter.



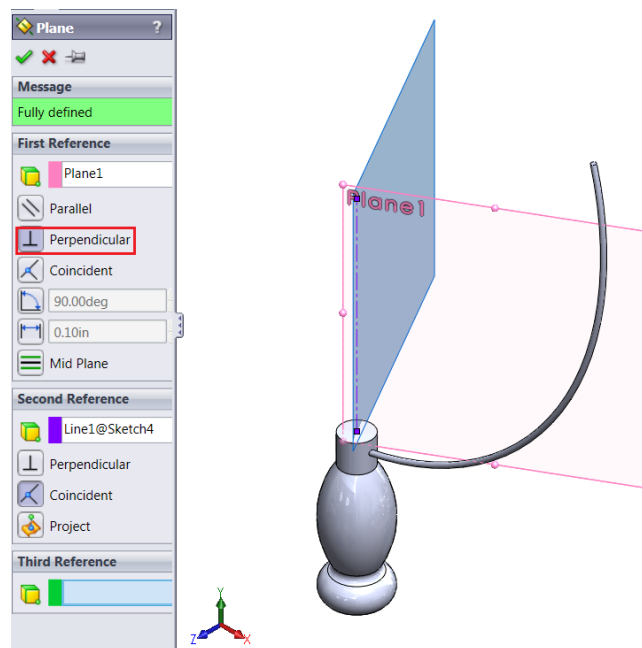
Then under the **Features** tab, select **Reference Geometry>Plane**:



Using the **Design tree** (refer to step 5), select the construction line previously created, and the front plane. Under the **Front plane** reference, use the **“Angle”** selection of **30.00deg**.

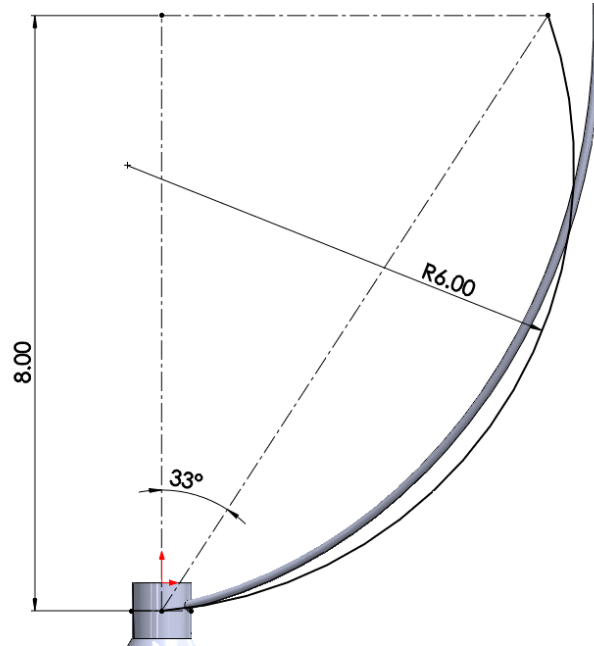


Now we will need to create a plane normal to the 30 degree plane just created. On this plane, we will create the profile of the sweep. Add a second reference plane using the construction line and the previously created plane as the references. Select **“Perpendicular”** under the **Plane1** box.



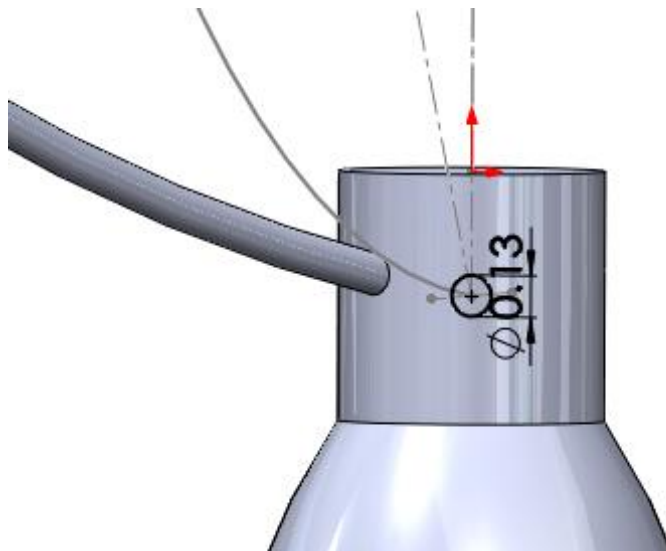
Step 7: Creating the Short Legs: Sweep Path

Similar to **step 3**, create the following sketch on the 30 degree plane created in the previous step:



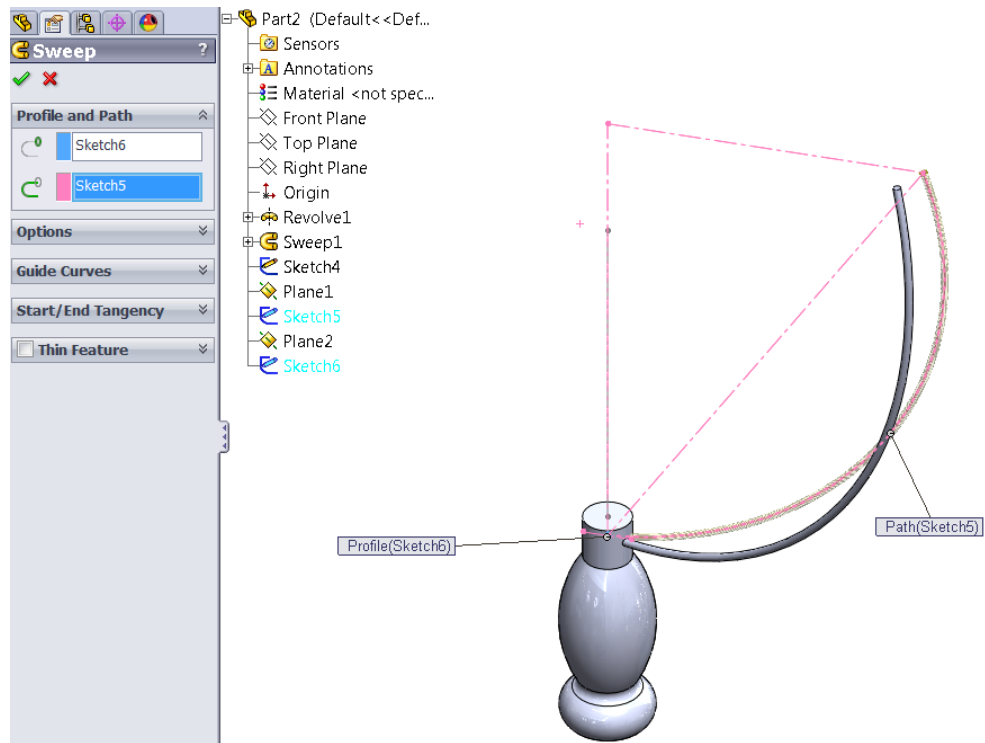
Step 8: Creating the Short Legs: Sweep Profile

As in **step 4**, create the profile and add the pierce relation between the center of the circle and the sweep path.

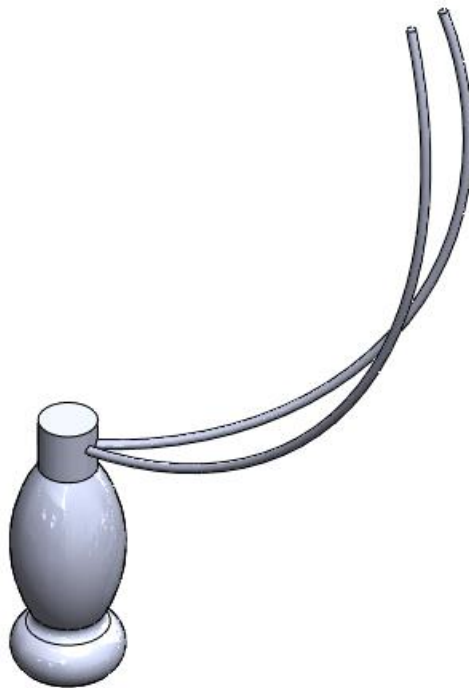


Step 9: Creating the Short Legs: Sweeping the Leg

Using the profile and path created in the previous 2 steps, sweep the short leg.

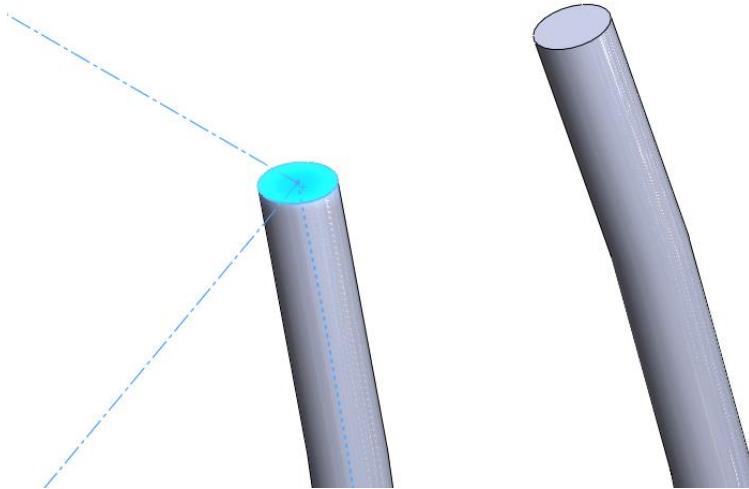


The resulting sweep should look like the following:

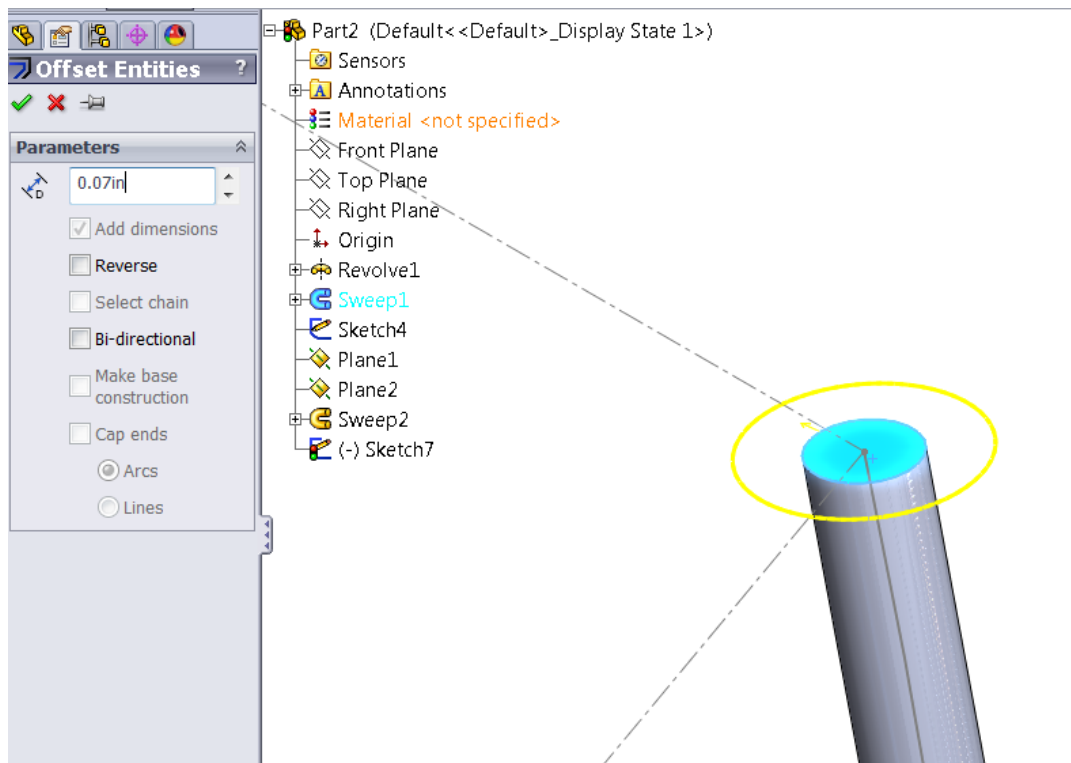


Step 9: Creating the Spherical Ends

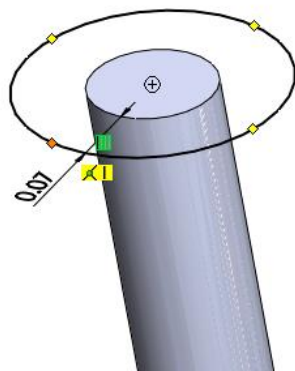
Create a sketch on either of the end-phases of the two swept legs.



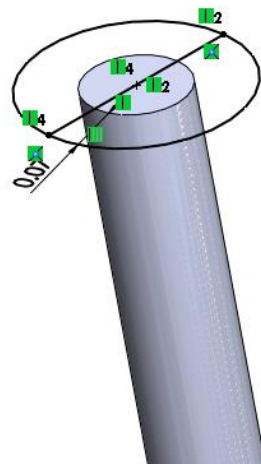
Using **Offset Entities**, found in the **Sketch** tab, offset the circular edge a distance of **0.07in**.



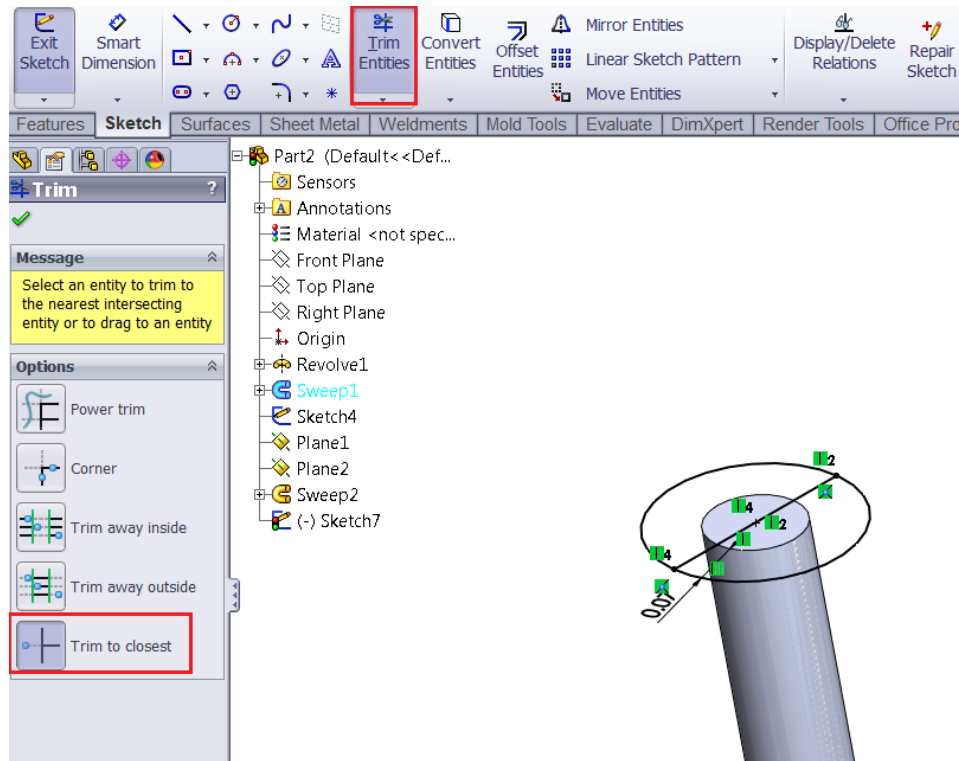
Add a construction line running along the diagonal of the circle. The orientation does not matter as long as it is the diagonal. If you place the cursor over the circle, yellow diamonds will appear at the horizontal and vertical positions of the circle. Placing the construction line between the two vertical or horizontal diamonds will create a diagonal.



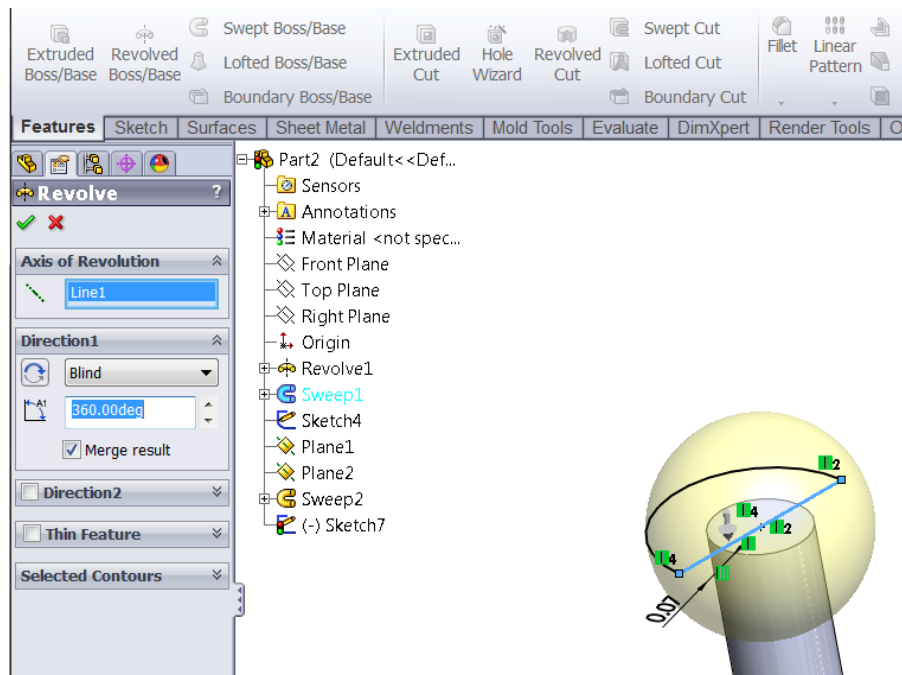
The resulting sketch should look like this:



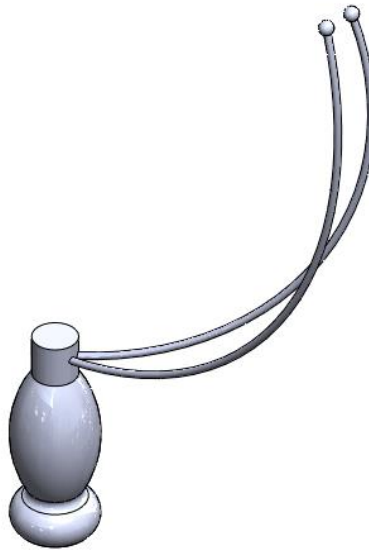
Trim away one of the semi circles as we only need one side for the revolution of the spheres. Use the **Trim Entities** tool found under the **Sketch** tab. Select the “**Trim to closest**” as the trim type:



Then using the **Revolved Boss/Bass** tool, revolve the semicircle along the diagonal as shown below:

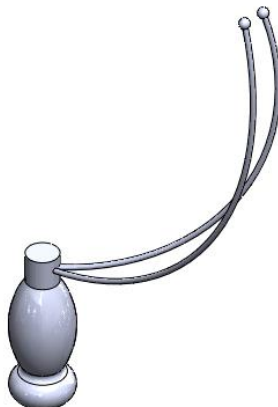
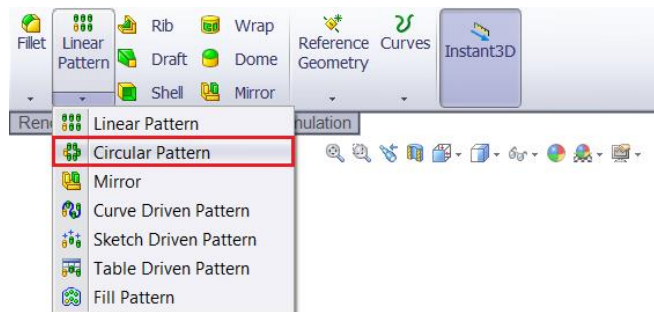


Repeat this process for the other sweep feature as well. The following results should look like the following:

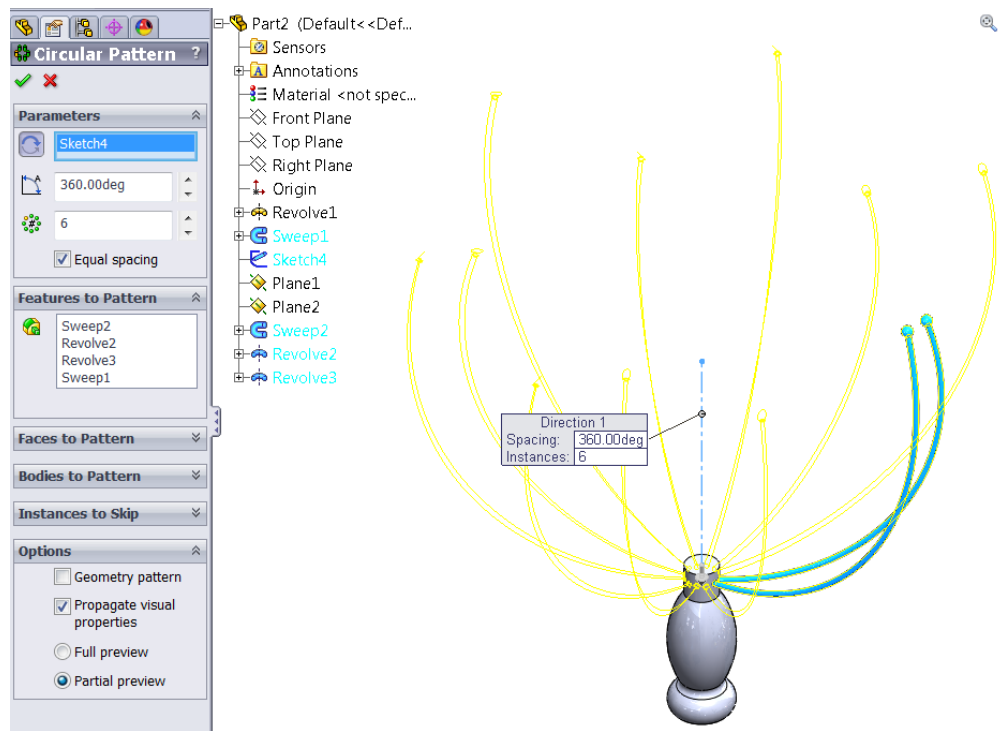


Step 10: Circular Pattern

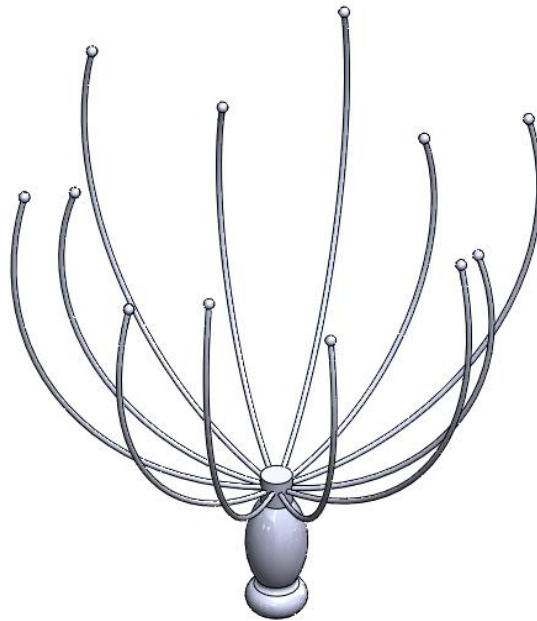
Instead of creating more sweeps and revolves for each legs, we will use the **Circular Pattern** tool, found under drop down menu of the “**Linear Pattern**” tool in the **Features** tab.



Using the **Circular Pattern** tool, under “**Features to Pattern**” select the two sweeps as well as the two revolved spheres. Select the construction line as the axis of rotation and set the number of instances to **6**. Also make sure that the box “**Equal spacing**” has been checked.



The resulting pattern should look like the following:

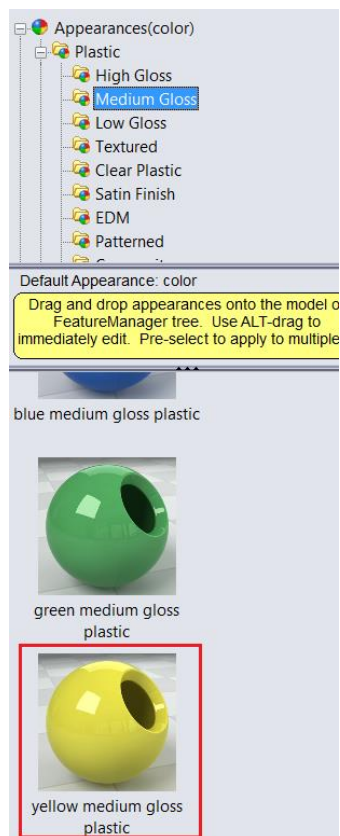


Step 11: Adding Appearances

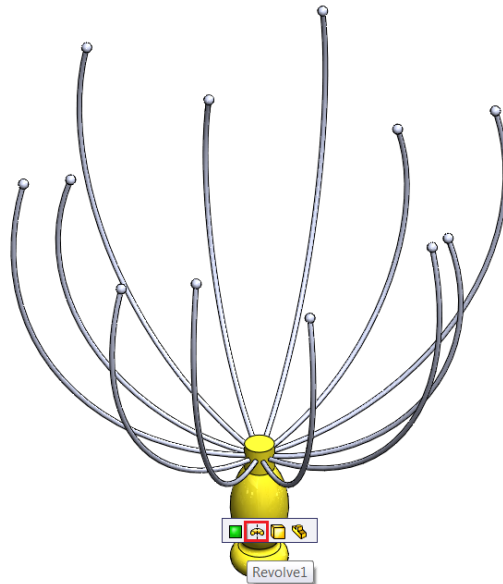
To add an appearance to the part, select the **Appearance** tab from the menu located on the right of the display pane:



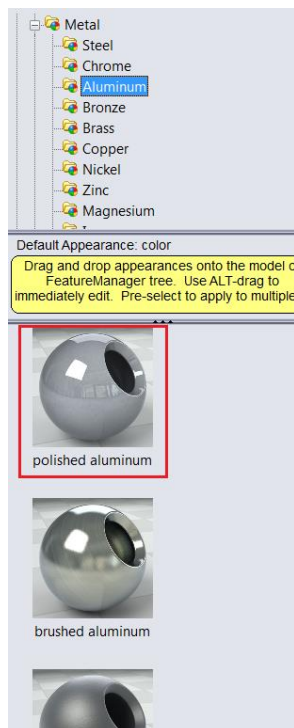
Select **Appearances>Plastic>Medium Gloss** and select **Yellow Medium Gloss Plastic**.



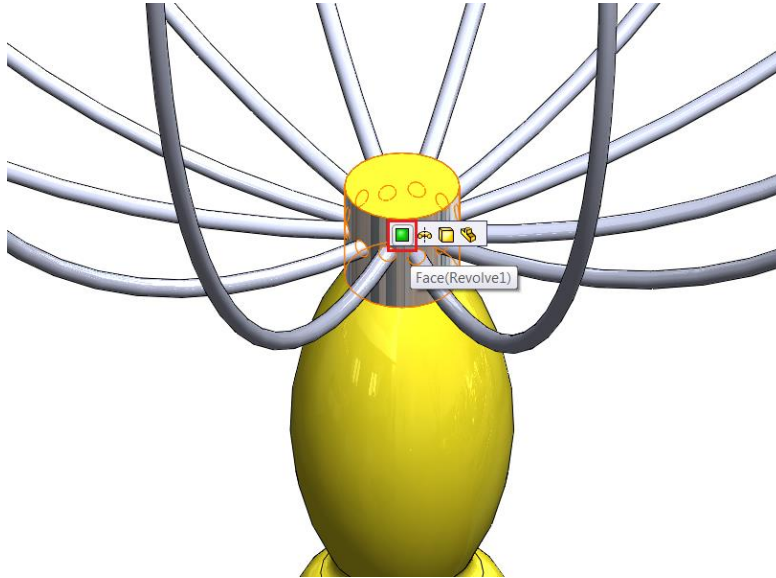
To add the appearance to the handle only, drag the color image onto the handle. A small menu next to the cursor will appear with several options for applying the appearance. Select the second option, **“Revolve1.”** This should apply the appearance to the entire handle as shown:



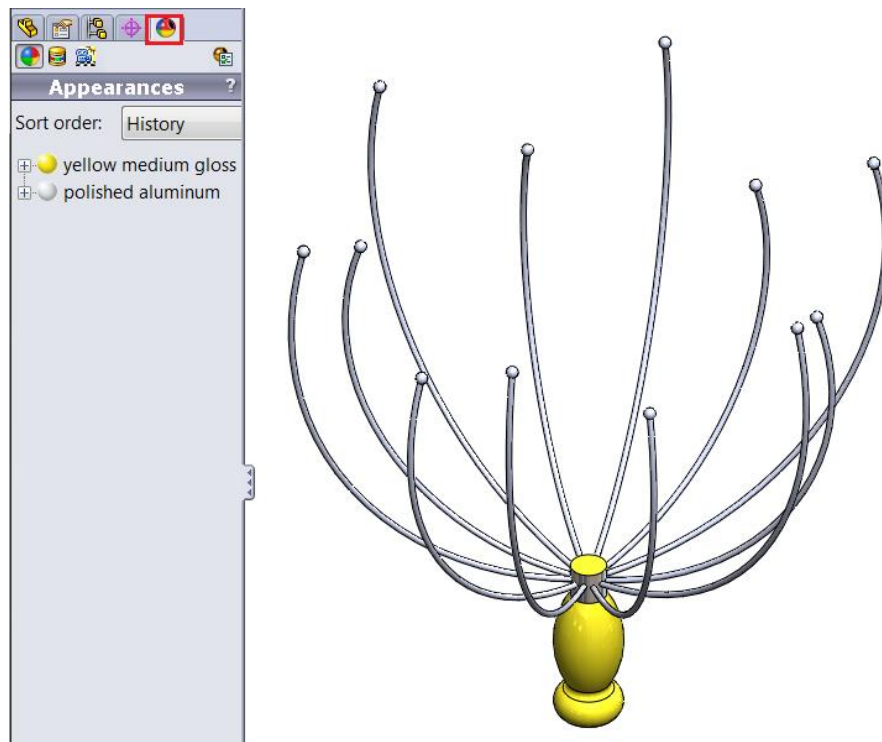
Now we want to add an aluminum appearance to the cap of the handle. To do so, select **Appearance>Metal>Aluminum>Polished Aluminum** from the **appearance tab**:



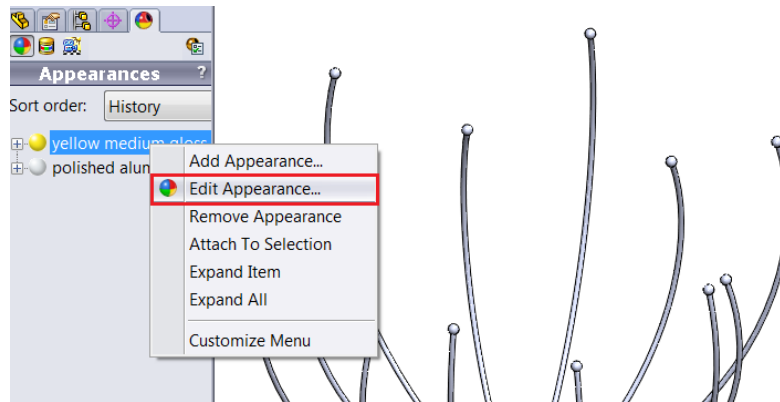
To add the appearance to the handle cap only, drag the color image onto the cylindrical portion of the cap. A small menu next to the cursor will appear with several options for applying the appearance. Select the first option, **"Face (Revolve1)."** This should apply the appearance to only the cap as shown:



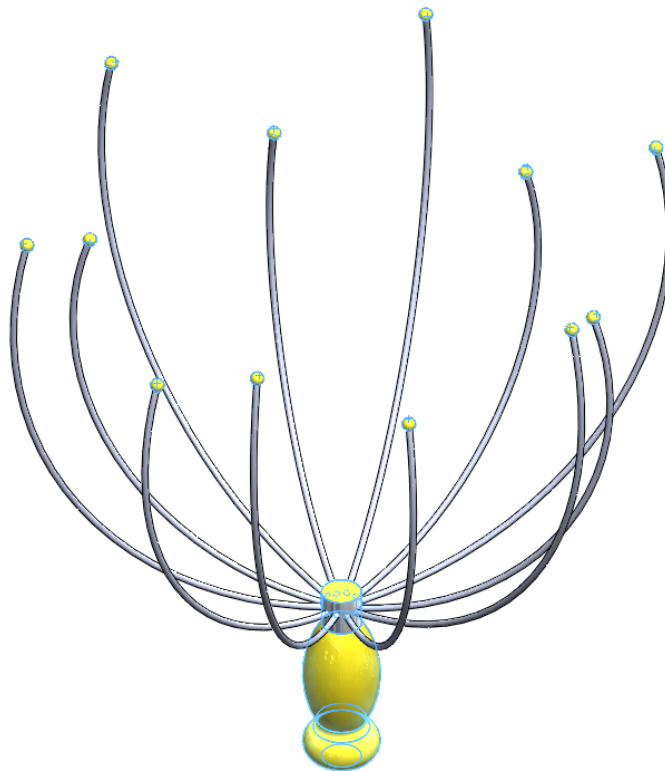
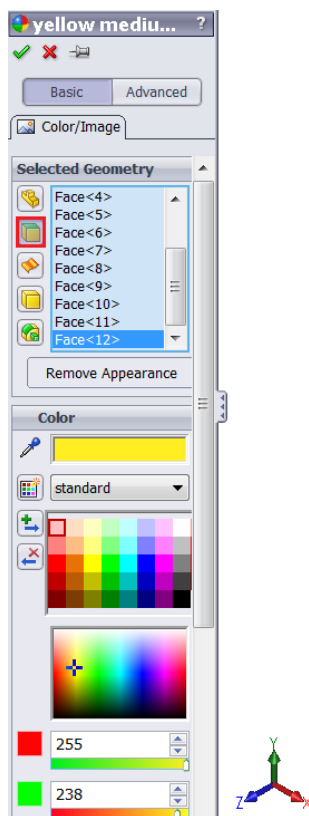
Now, instead of dragging and dropping the appearance onto each arm, we can edit the current appearance to include them as well. To do so, select the **appearance** tab from the **property manager** on the left of the display pane:



Right click the appearance “**Yellow medium gloss**” and select **Edit appearance**:

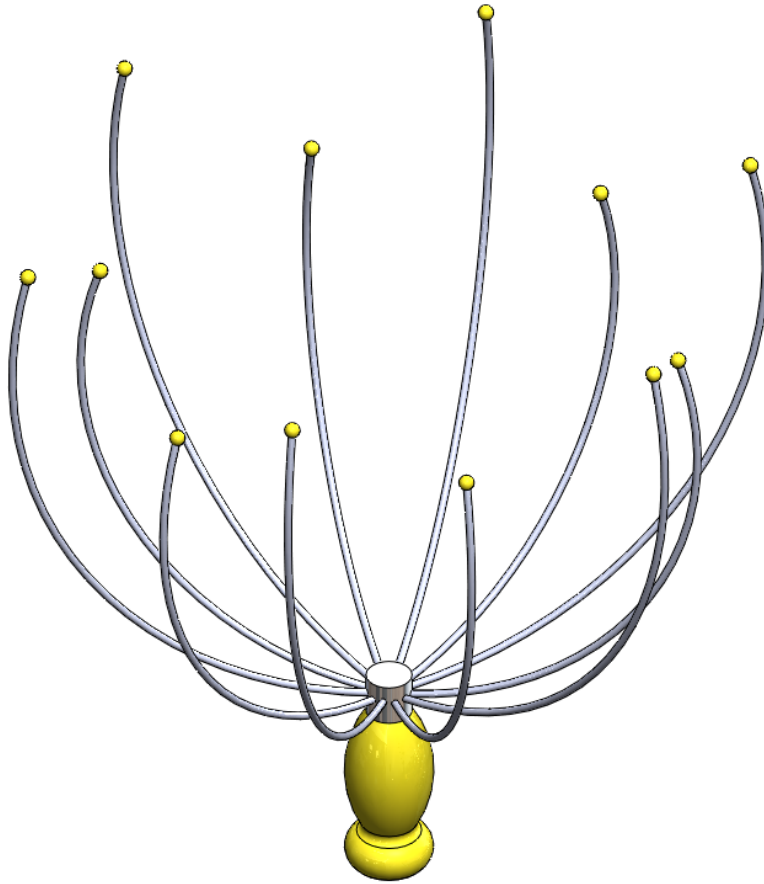


In the property manager, select the area under “**Selected Geometry.**” This area should be highlighted blue. Next to the box is again a list of options for picking which geometries are included. These include, from top to bottom: part, face, surface, body, and feature. Make sure the **Face** selection is chosen (as highlighted). Then select all the round spheres at the end of the sweeps.



Repeat this process for the top surface of the cap and the legs. Make the top surface of the cap and the legs **Polished Aluminum**.

The resulting part should look like this:



Step 10: Save and Exit

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