Sprayer Assembly

Updated 10/26/2018

Introduction

[1]**SolidWorks®** **Assembly** models are simply a collection of two or more **Part** models put together in an organized fashion. Geometric relationships called **Mates** are applied to properly position the parts in relation to each other.

Assemblies can be joined with other assemblies to make larger assemblies.

Also, features such as holes, etc. that were not on the original **Part** can be added at the **Assembly** level.

You will find assembly models to be useful when working on projects in various engineering courses later.

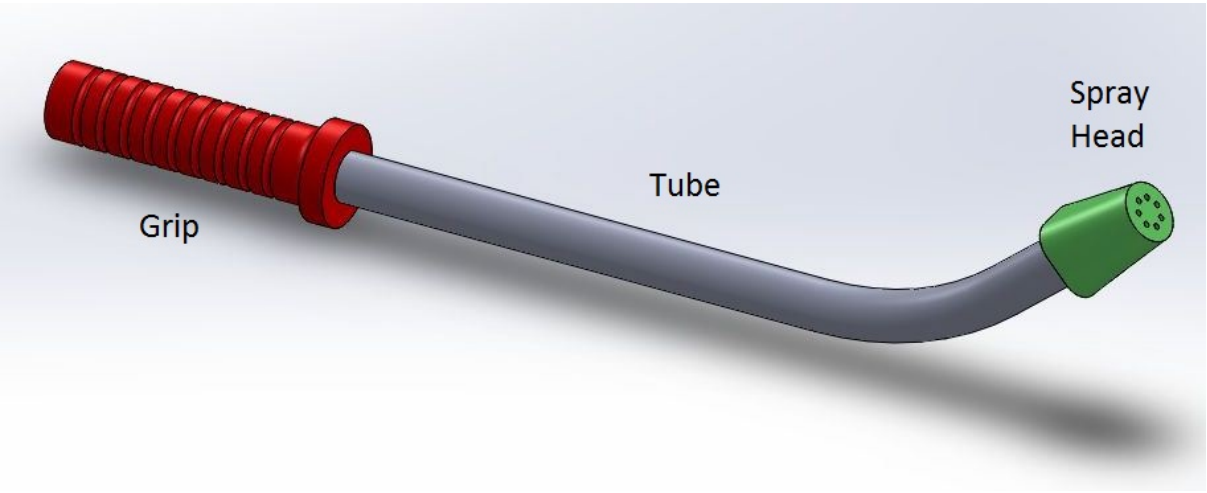
What you will be doing in this tutorial is to first complete each of the individual **Part** models that make up a *Sprayer Assembly*, and then put the parts together to form the assembly.

The three **Part** models that you will need to build are:

1. *Tube*
2. *Grip*
3. *Spray Head*

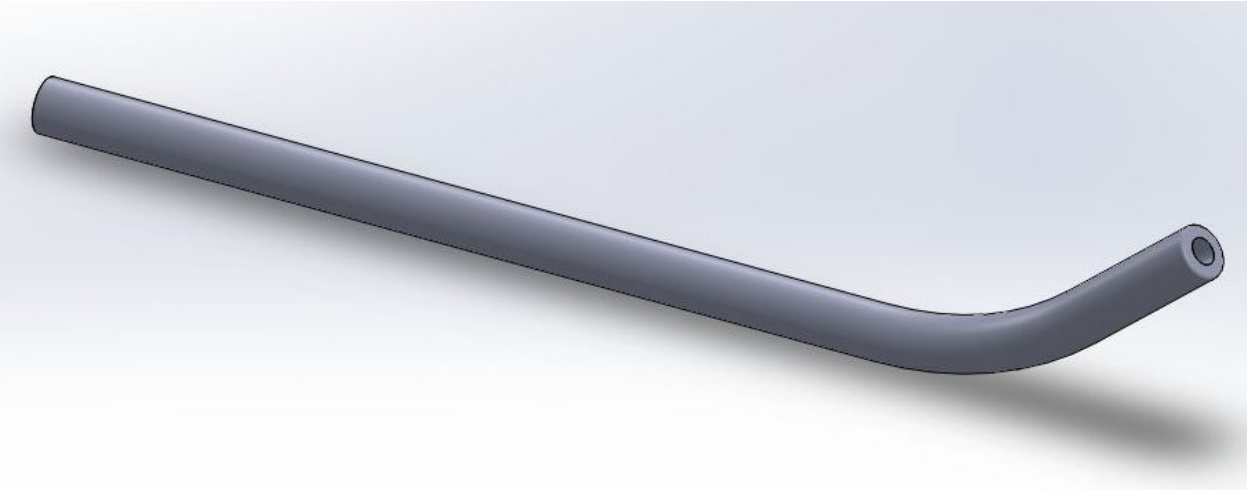
All three parts are sized in **IPS** units. Set decimal places for Length to **two** and Angle to **None**.

After these **Parts** are joined together as a new **Assembly** model, you will have this *Sprayer Assembly*:



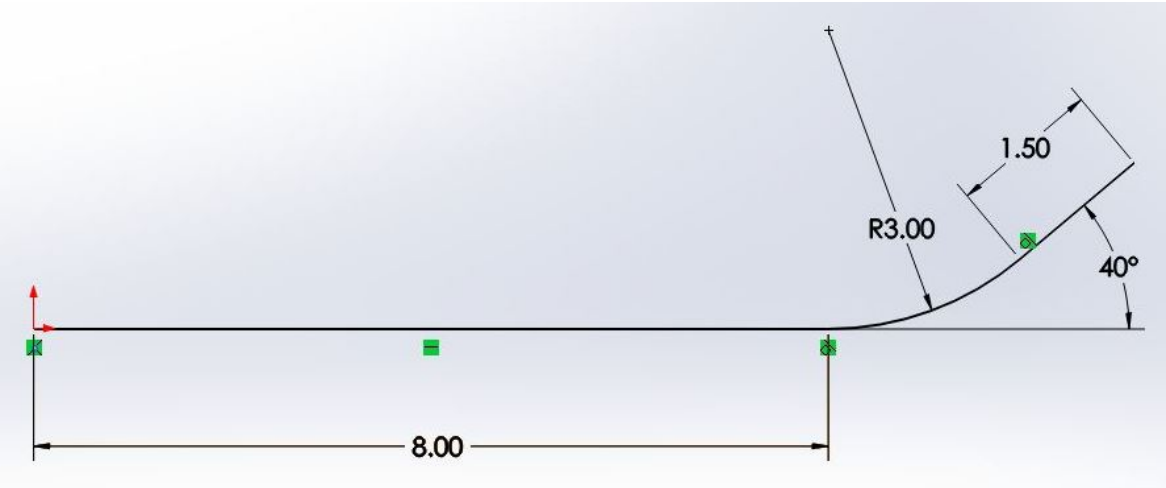
[1]**SolidWorks** is a registered trademark of Dessault Systémes SolidWorks Corporation. Screen shots from SolidWorks are used throughout this tutorial.

1. Tube



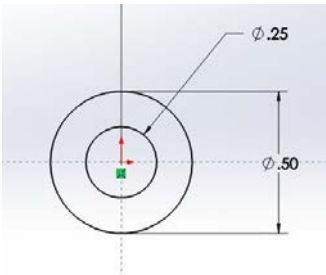
This tube model will be created using the **Swept Boss/Base** feature.

Begin by sketching a simple one-plane **path**, as shown below, on the **Front Plane**.



Exit the **path** sketch and then select the **Right Plane** to begin a new sketch (for the **profile**).

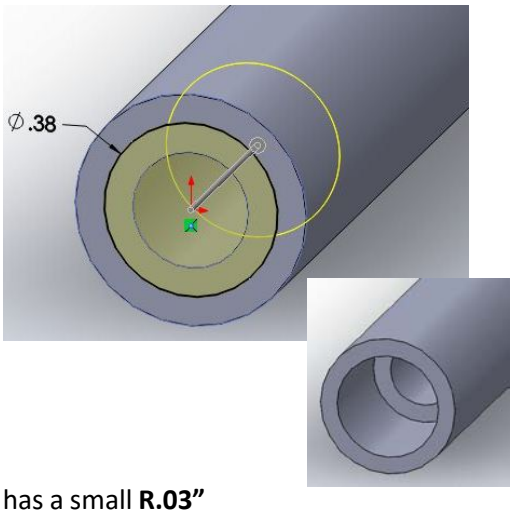
Create the **profile** sketch (as shown to the right) and then **Exit** the sketch.

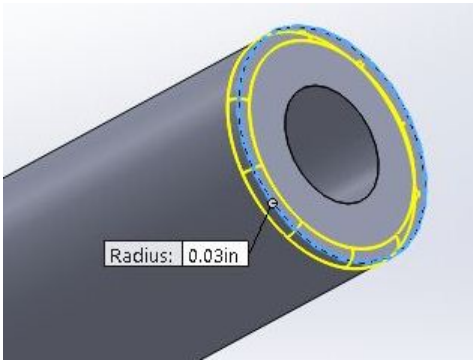


From the Features group, use the **Swept Boss/Base** tool to sweep the **profile** along the **path**, creating the tube.

On the left end of the tube, where the *Grip* goes, there is a *counterbore* (which is a blind-depth second hole following the first hole, but larger in diameter).

To create the counterbore, sketch and dimension a **.38** diameter circle on the end of the tube and use **Extruded Cut** for a blind depth of **.5**





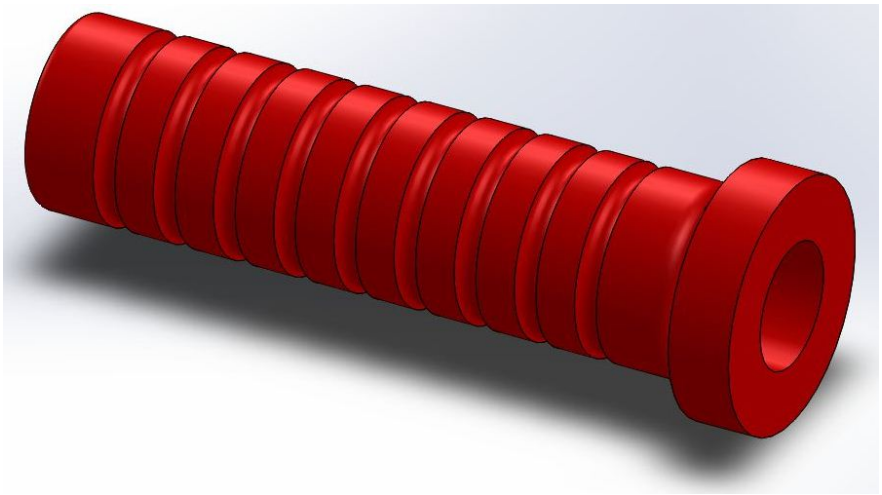
The right end of the tube has a small **R.03"** round on the outside diameter. Use the **Fillet** Feature to add it.

Switch to **Trimetric View** when finished.

No **material** needs to be assigned to this model. Do not change the **color** from the default **SolidWorks** color.

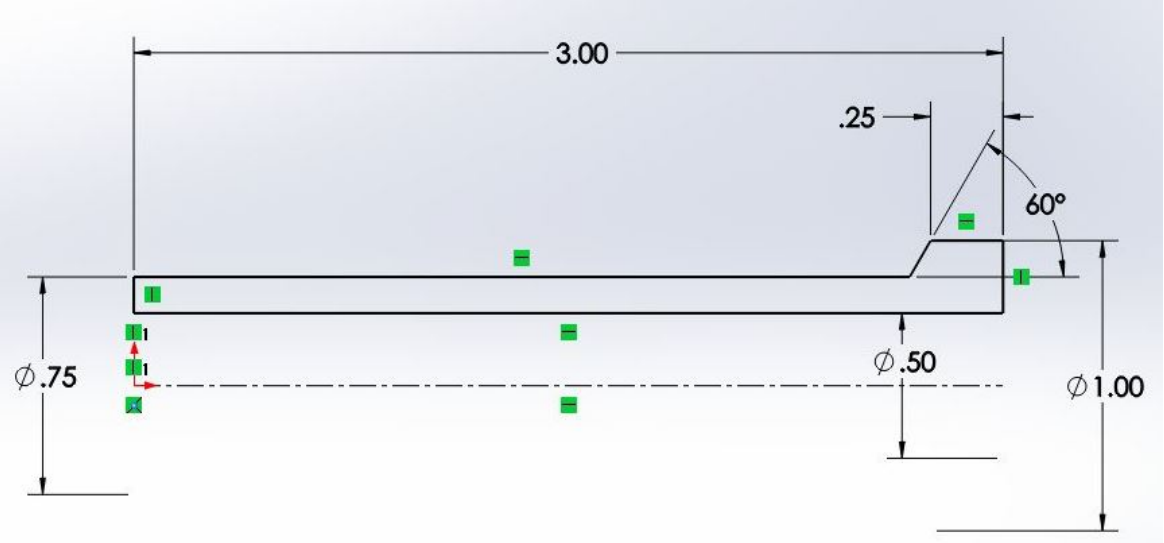
Save the *Tube* part file.

2. Grip



The basic structure of this model will be built using the **Revolved Boss/Base** feature (other features will be added later).

Begin by sketching the top half of the longitudinal cross section, as shown below, on the **Front Plane**.



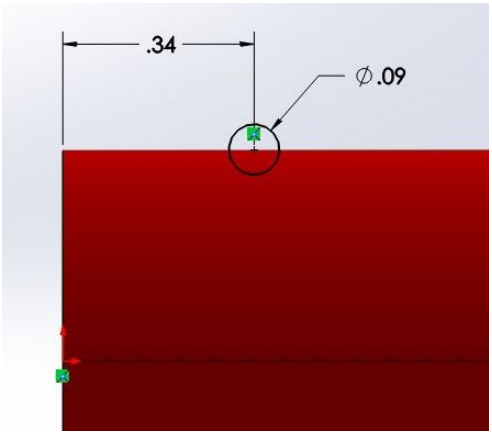
Note that for the sketch to be fully defined, the left end of the cross section must be aligned with the origin either by adding a vertical relation between the origin and a point or by sketching a vertical centerline up from the origin and coincident or colinear with the left end.

The *Grip* is sized using its *diameter* dimensions (not *radii*), so use **Smart Dimension** to add the *diameter* dimensions shown, using the method of dimensioning across a centerline (the horizontal one).

From the Features group, use the **Revolved Boss/Base** tool to create the body of the grip.

Notice the set of grooves on the outside of the grip. You will need to create one groove and then copy it.

The sketch for the first groove is shown here. The sketch is on the **Front Plane** and the center of the small circle is coincident with the top edge of the grip, as shown.

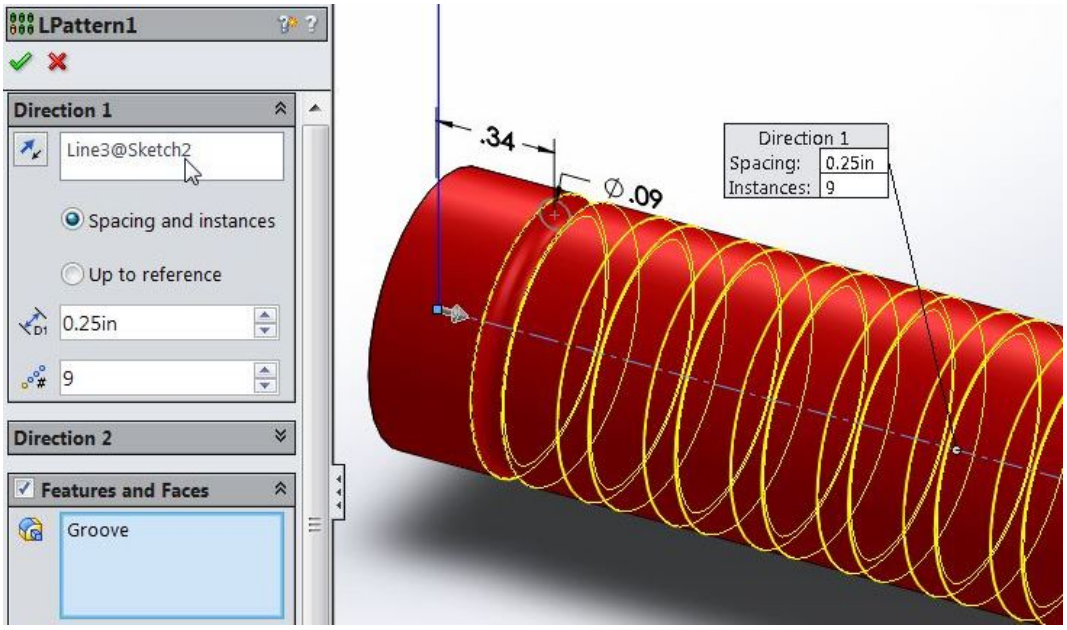


A new horizontal centerline is needed, beginning at the origin and extending to the right.

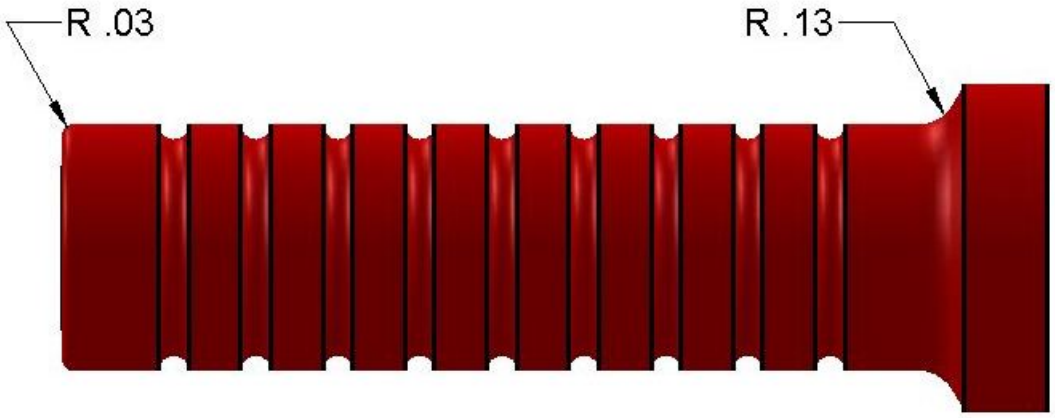
Use the **Revolved Cut** Feature to revolve the circle around the centerline and cut the groove. In the Design Tree, rename this feature: *Groove*.

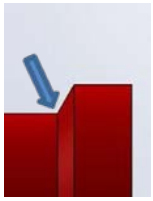
The grip geometry contains no straight lines to select as the **Pattern Direction** of the linear pattern. A reference axis could be created but that isn't necessary because the **centerline** within the groove sketch can be used for that purpose. The sketch must be **visible** however. In the Design Tree, click on the ▸ symbol to the left of *Groove* to cause the sketch name to be displayed below it. Now click on the sketch name and select **Show** from the menu.

To copy the groove, select **Linear Pattern** from the Features group and enter the values shown in the Property Manager below.



Finally, a *fillet* and a *round* must be added to the model. Use the **Fillet** tool from the Features group to add them to the proper corners as shown in the figures below.





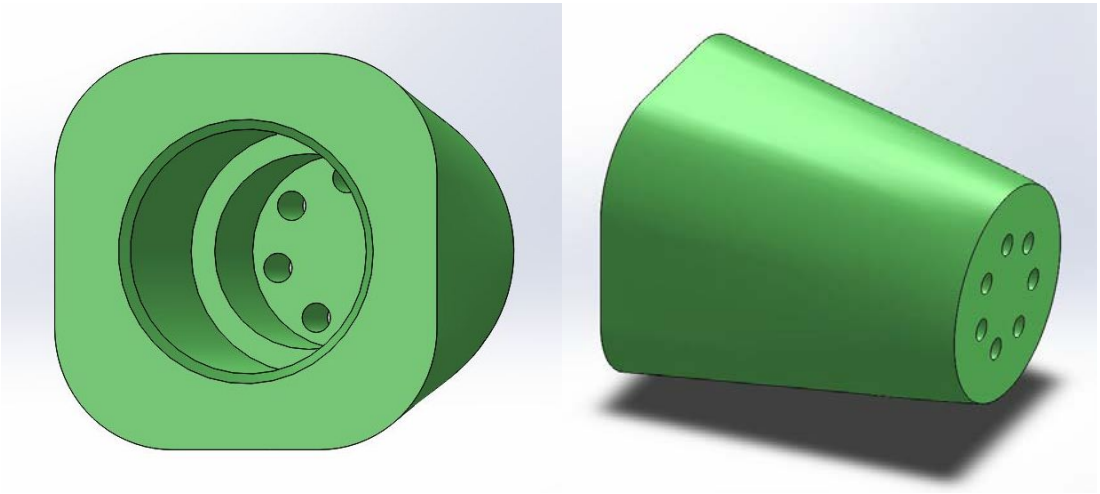
Switch to **Trimetric View** when finished.

No **material** needs to be assigned to this model.

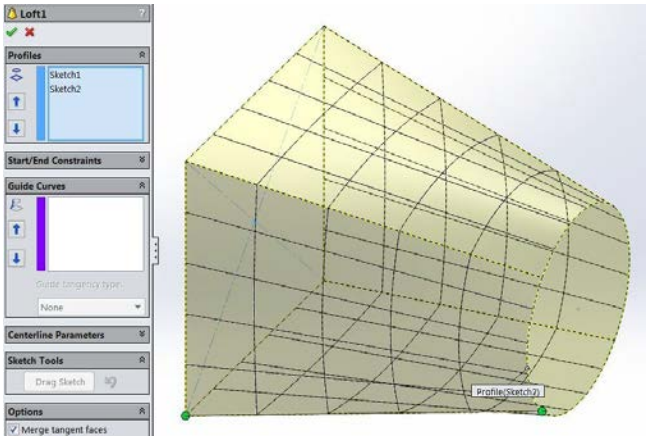
Change the **color** of the grip to some shade of **red**.

Save the *Grip* part file.

3. Spray Head



The **Lofted Boss/Base** tool will be used to create the main body of this model.

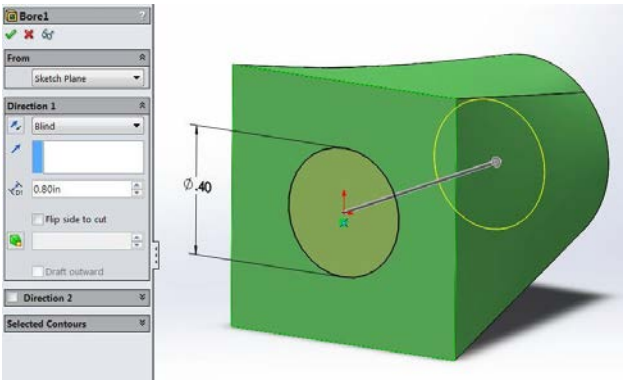


Begin by selecting the **Right Plane** and use the **Center Rectangle** tool to sketch a **.81”** square, centered on the origin.

Exit that sketch and then create a **Reference Plane** with an Offset Distance of **1.00** in front of the **Right Plane**.

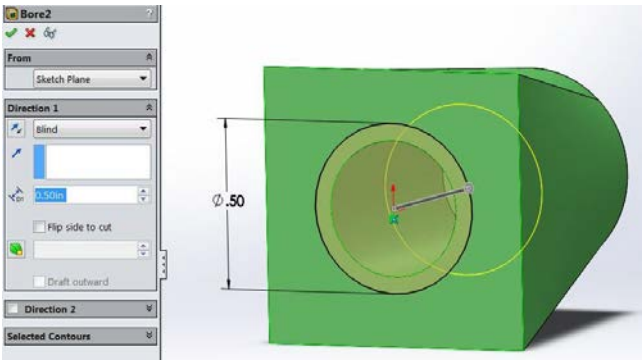
Sketch a **.6”** diameter circle on the new plane and then exit that sketch.

Use the **Lofted Boss/Base** feature to transform the two sketches into a lofted solid.



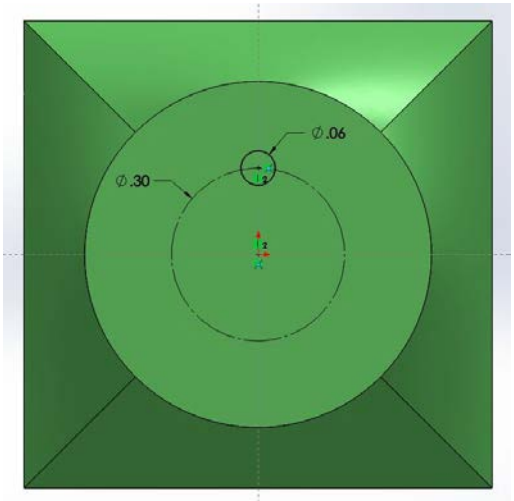
Rotate the part and click on the face of the square and then sketch a **.4”** diameter circle on it.

Use **Extruded Cut** set for a **Blind** depth of **.8”** to create the first hole.



Click on the face of the square again and sketch a **.5"** diameter circle on it.

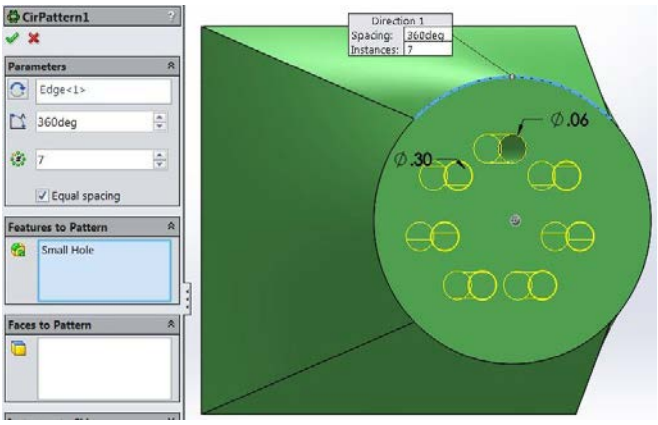
Use **Extruded Cut** set for a **Blind** depth of **.5"** to create the counterbore.



The round face at the small end of the loft has a circular pattern of small holes. Switch to the **Right** view and click on the round face.

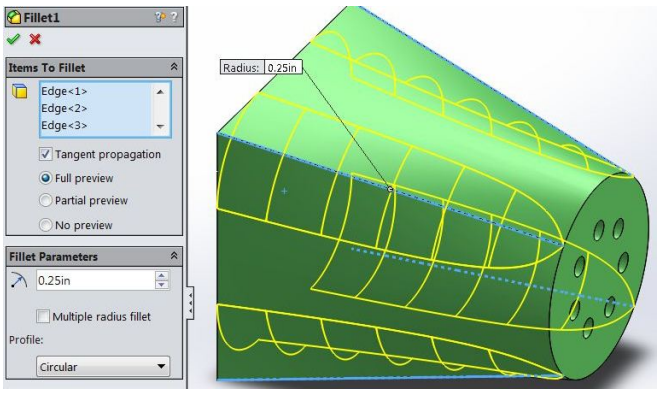
First, sketch the **Ø.3** reference bolt circle on it. Be sure to check the ☑ **For construction** box in the Property Manager.

Now sketch the small, **Ø.06** regular circle centered on the top quadrant of the bolt circle and do an **Extruded Cut**, Through All, to make the first small hole. In the Design Tree, rename this extruded cut: *Small Hole*.

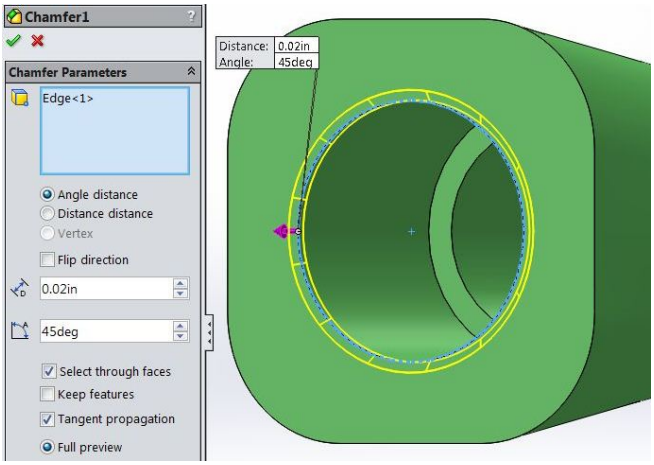


Use the **Circular Pattern** tool from the Features group to create the ring of **7** small holes, equally spaced.

The outer **circular edge** at the front end of the part is a convenient selection for the **Pattern Axis** since the center of that circle locates the axis. Be sure to select the **edge** (in the picture, the selected portion is highlighted in blue). Do not select the flat round **face** on the front. (However a **cylindrical face** sharing the pattern axis, such as inside the large hole or counterbore on the back side, would work.)



Notice that the four edges running the length of the loft are rounded. Use the **Fillet** tool from the Features group to add **R .25"** rounds to them all at once with one use of the tool.



The last feature needed to complete the model is a **.02" x 45° Chamfer** on the edge of the counterbore, as shown.

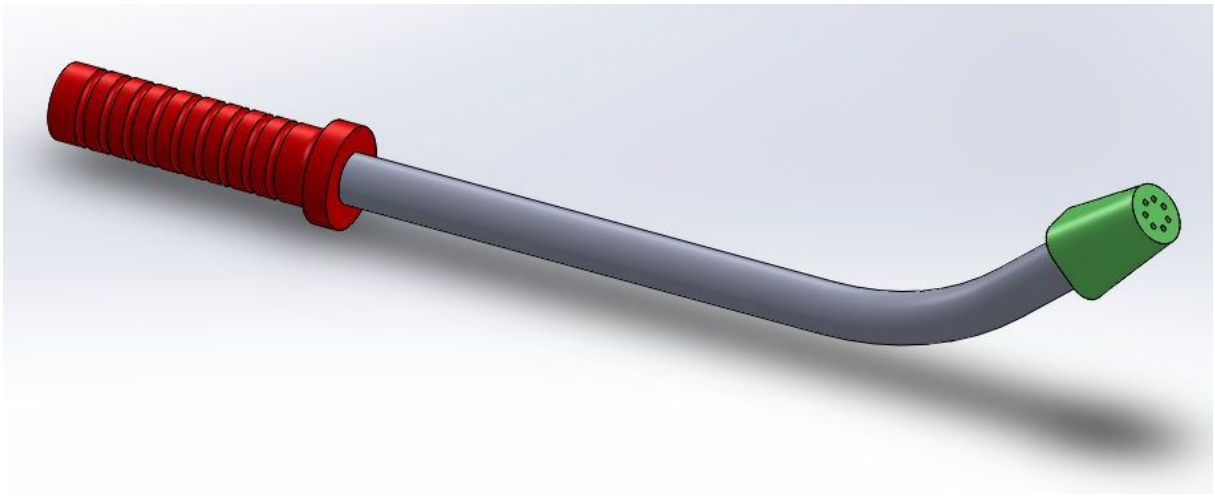
Switch to **Trimetric View** when finished.

No **material** needs to be assigned to this model.

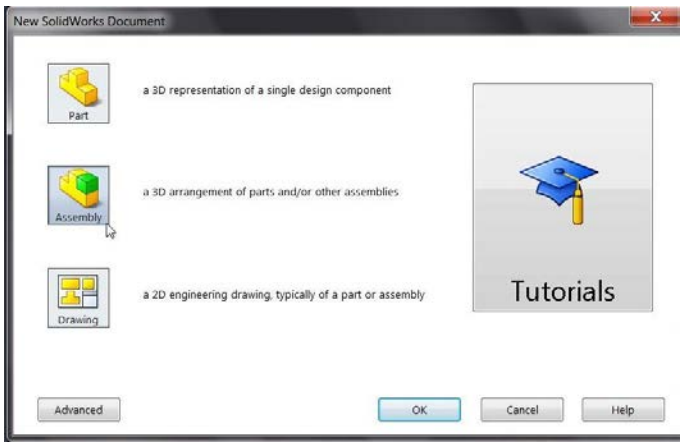
Change the **color** of the spray head to some shade of **green**.

Save the *Spray Head* part file.

4. Sprayer Assembly

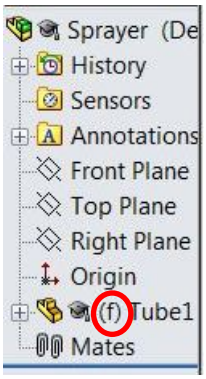


Start a new assembly by clicking on the **New** icon and choosing **Assembly** as the document type.



To build an assembly, you start by bringing in the base part of the assembly and fixing it in place at the origin of assembly space. Then, copies of the other parts are brought in and **Mates** are added to properly orient them in relation to the other parts.

In the **Begin Assembly** Property Manager, the **Part/Assembly to Insert** box needs to have your **Tube** listed as the Part to insert first. If it's not already listed in the **Open documents** window, click the **Browse** button to locate it. Now, and this goes for the **first part** of any new assembly, do not click randomly in the Graphics Area but click on the **OK** ✓ check mark. This fixes the tube at the **origin** of the assembly.

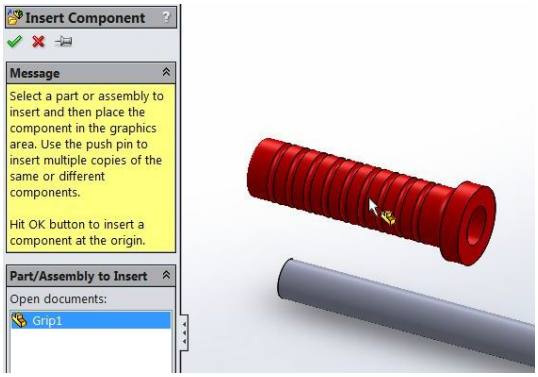


In the Design Tree, parts are added to the list whenever you bring them into the assembly. Note that your tube includes (f) in its listing, which indicates that it is **fixed** and cannot be moved. If you need to later move a fixed part, right-click on its name and select **Float** from the popup menu. The (f) notation will then change to (-) and you can then drag the part around.

If **Units** are not already set to **IPS** do so now.

Note that the tabs in the Command Manager area for Assemblies differ from the tabs that you are used to seeing when building Part models.

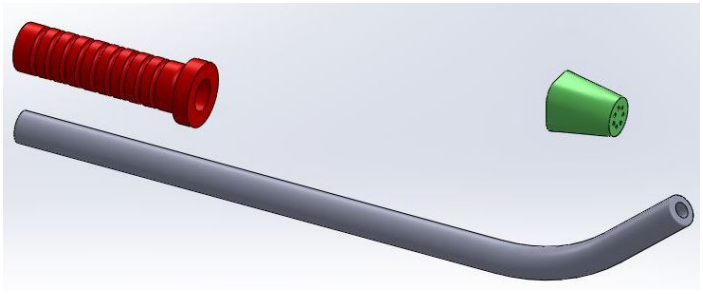
The **Assembly** tab in the Command Manager should already be selected. Click on the **Insert Components** tool to bring in a copy of your **Grip**.



Don't click **OK** which would lock the part onto the origin but instead, move the grip somewhere into the Graphics Area and click to temporarily park it.

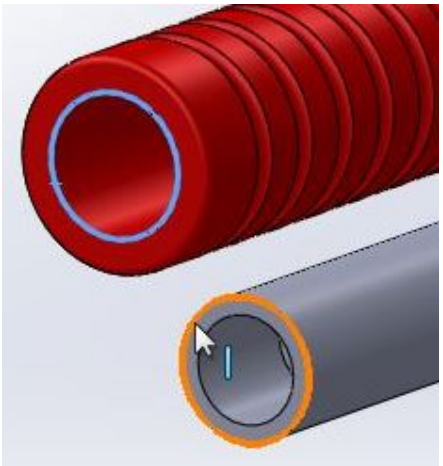
In a similar fashion, insert a copy of your **Spray Head**.

Since the grip and spray head are not fixed, you can click on them and drag them around in the Graphics Area.

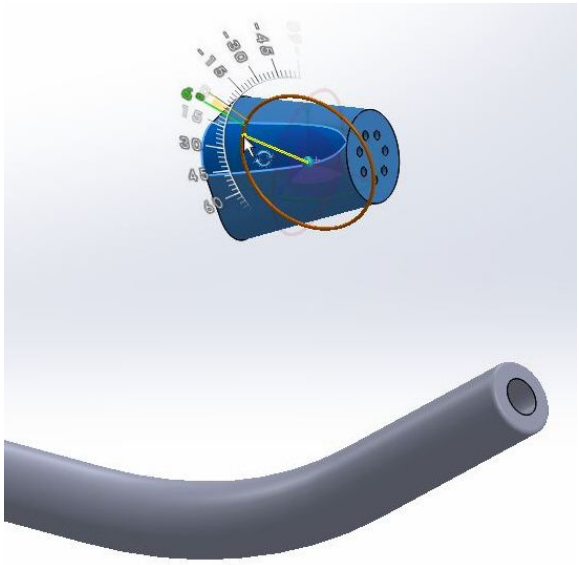


Notice on your drawings that the outside diameter (O.D.) of the tube is exactly the same size as the inside diameter (I.D.) of the grip. This being the case, you can use just **one mate** to properly locate the grip on the tube.

Activate the **Mate** tool and click on the circular edge at the left-hand end of the grip I.D. and then on the corresponding O.D. edge at the end of the tube. A **Coincident** mate should automatically get applied. This will also make the parts concentric without having to add an additional mate.

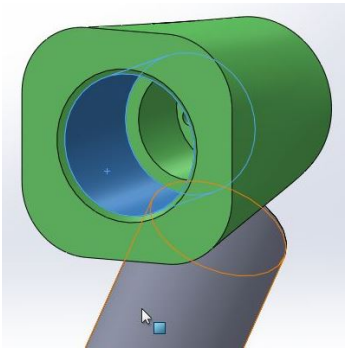


In assemblies, there is a tool available to rotate parts so that you can get them oriented into the approximate position needed before adding mates. To demonstrate, right-click on the spray head in the Graphics Area and select **Move with Triad** from the menu. The triad has three rings and three vectors that you can drag with your cursor to reposition the selected part.

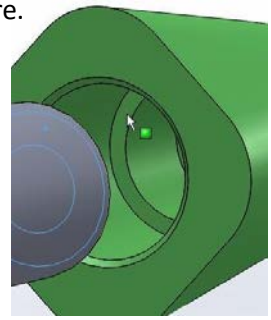


Click on the triad's ring that's parallel to the Front Plane of the spray head, as shown in the screen shot to the right, and drag around on the ring to rotate the spray head so that it is generally angled upward like the right-hand end of the tube. Click in the white space of the Graphics Area to exit the triad tool.

The turned-up right-hand end of the tube fits inside the counterbore of the spray head and extends all the way in until the end of the tube touches the bottom of the counterbore. Although the tube O.D. and counterbore I.D. are identical, you can't add a concentric and a coincident mate in one operation as was done with the grip, because the tube has a **round** on the O.D. at this end.

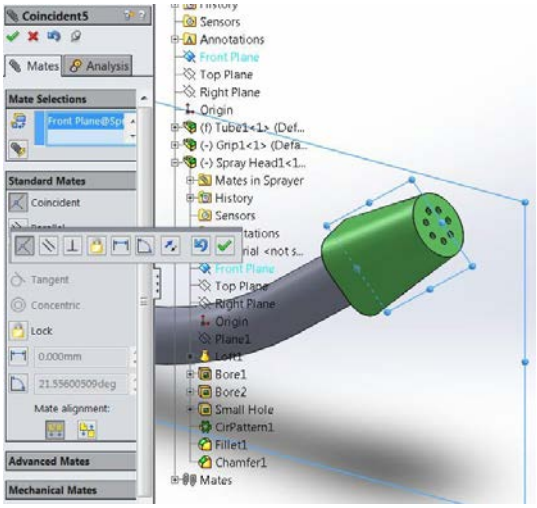


Add a **Concentric** mate between the **outside diameter** of the tube near its right end and the **inside diameter** of the counterbore.



Add a **Coincident** mate between the right end face of the tube and the bottom face of the counterbore. Be careful to select *faces*, not *edges*. To access the faces, you may have to move the spray head away from the tube, rotate the view, and zoom in closer.

The last thing that needs to be done is to properly align the square end of the spray head.



Using the **Mate** tool again, select the assembly's **Front Plane** from the Design Tree (the tree will have moved into the Graphics Area and you may need to click on its plus sign to expand the list).

For the second mate selection, select the spray head's **Front Plane** from the Design Tree.

Make the two planes **Coincident** and click **OK** ✓

Switch to **Trimetric View** when finished.

Save the *Sprayer Assembly* file.