Difficulty: Beginner | Time: 20 minutes

As a Beginner Tutorial, it is assumed that you have little or no prior SOLIDWORKS experience.

The Quick Start Tutorial is part of an overall orientation to SOLIDWORKS and is intended to be used in conjunction the SOLIDWORKS Tip Sheet, and with an instructor or recording of an instructor to walk you through SOLIDWORKS.

After you start a new part for the first time, it may prompt you to select Units. The default is Metric (MKS), but all of our Tutorials are in American Standard (IPS). If you ever need to check Units, select the gear icon in the Top Menu: Document Properties tab \ Units.

You will create this assembly:
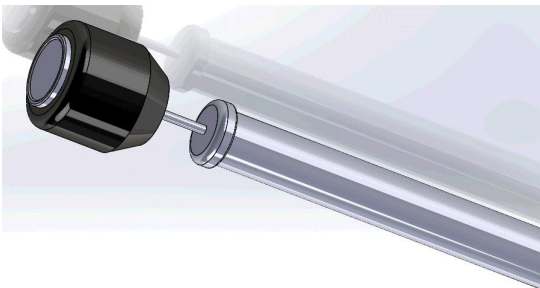

This lesson includes:

- An orientation to the menu, tool bars, Design Tree, and Property Manager and Display Manager
- Learn to start a New Part/Assembly, Sketch, Accept Changes, add 3D Features, create an Assembly, Mate, Save and Close.
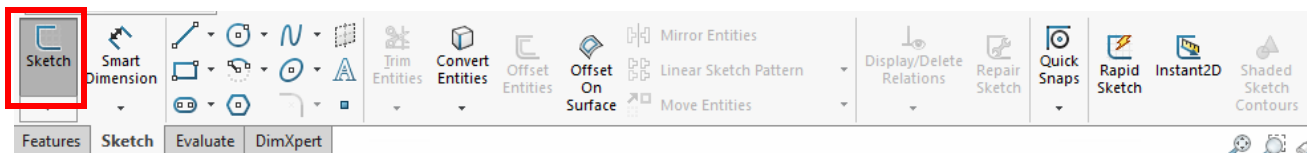
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Sketch a Throttle Knob

Every part begins with a 2D Sketch. This part will be a red aircraft throttle knob.

1. Create a New Part. In the top menu, select File \ New to open the New SOLIDWORKS Document window and select New Part.
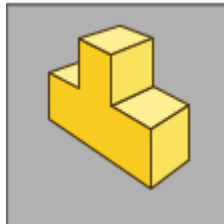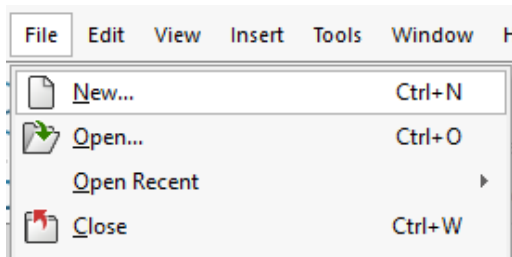
2. On the Sketch toolbar, select the Sketch tool in the top menu to turn on Sketch mode.

3. Select the Top Pane. Panes will outline in orange when you hover over them. When selected, all other panes are hidden and all sketches will be drawn on the selected pane.

4. Select the Center Circle tool. You may need to select the drop down arrow next to the circle tool.
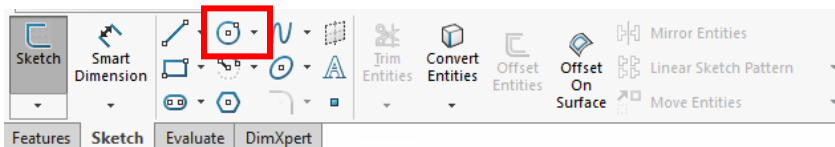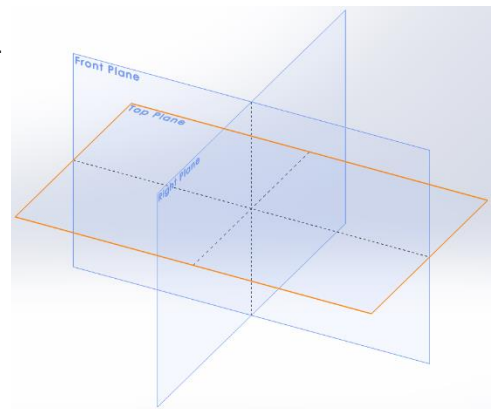
5. Draw a 1.5 inch Center Circle. Draw it first and then establish dimensions. Click, drag and then release. Notice this tool creates a circle from the center out and establishes a visible center point that you can use as a drawing reference. You can dimension the radius of the circle under Parameters in the Property Manager on the left side of the screen. If we want a 1.5 inch circle, the radius needs to be .75. Select the green checkmark at the top of the Property Manager to accept the dimensioned circle. Notice that the Design Tree tab becomes the active tab, when you are not editing a design
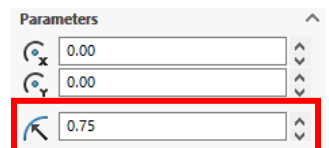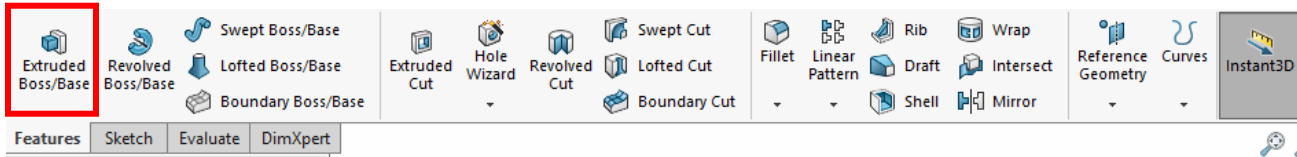
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element. If the circle needs to be edited further, simply click on the edge of it and the Property Manger becomes the active tab.

6. To begin adding 3D features to this sketch, Exit Sketch and select the Feature toolbar.
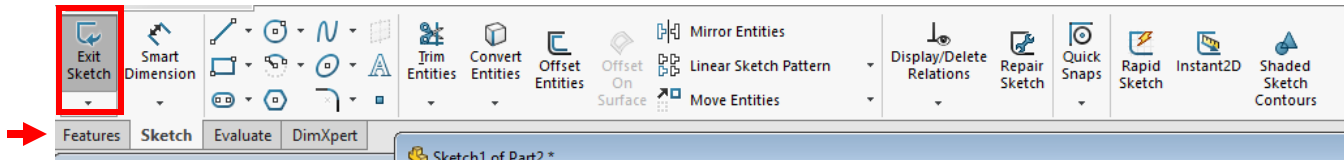

7. Extrude the circle to a depth of 1.5 inches. Select the circle and select the Extruded Boss/Base tool in the Feature toolbar.

8. You can set the height to 1.5 inches by either clicking and dragging the arrow that appears on the preview of the extruded circle or change the dimension in the Property Manager.


9. A throttle knob is typically black in an aircraft to differentiate from other knobs. You can change this knob to black before adding other features. Right click on the Boss-Extrude1 in the Design Tree, select the down arrow next to the Appearance tool and select the silver sphere. Select the black color swatch on the Property Manager and select the
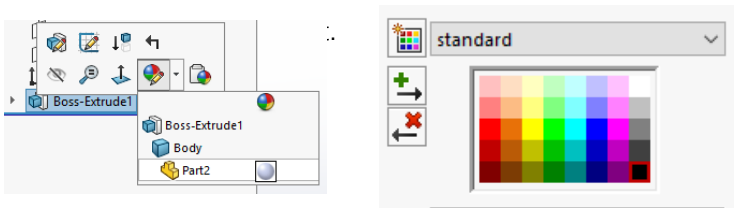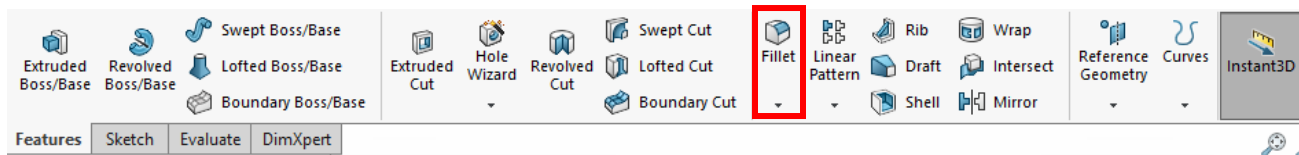
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10. Add a Filet to the top edge of the cylinder and a Chamfer to the bottom edge. Filet and Chamfer are similar tools and both accessible from the down arrow next to the Filet tool on the toolbar.
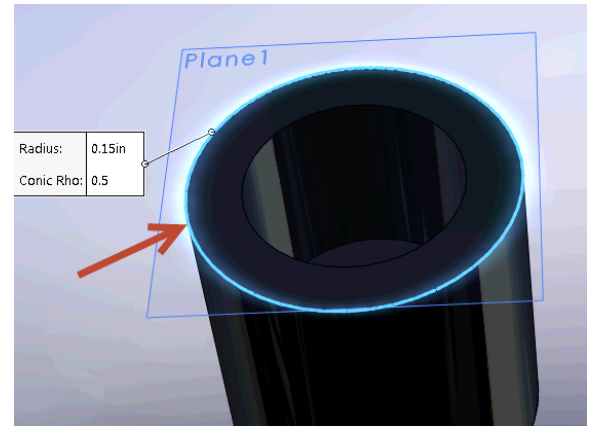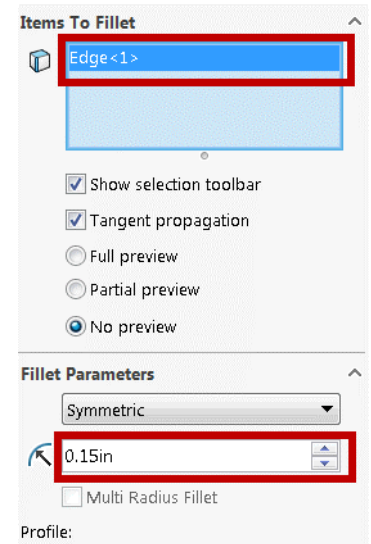

11. Select the top edge of the cylinder.
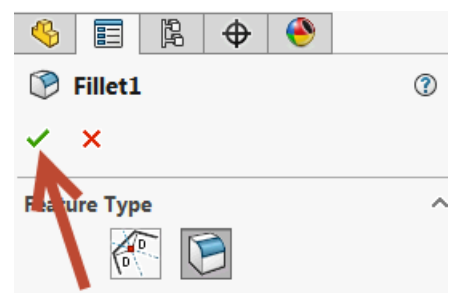
12. Select the Filet tool.

13. In the Fillet feature manager once you select the outside edge it will now be listed in the Items to Fillet listing.
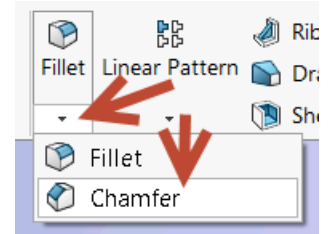
14. In the Fillet Parameters section input 1.5in.

15. Select the green check mark at the top of the Fillet feature manager to accept the new feature

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16. Rotate (hold down the middle mouse button) the part around so that you can see the bottom of the part.
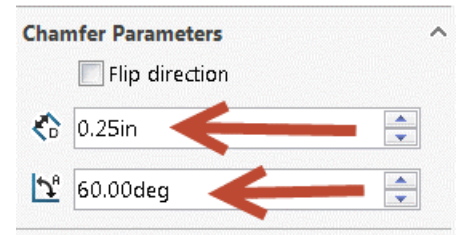
17. Select the Chamfer Tool.

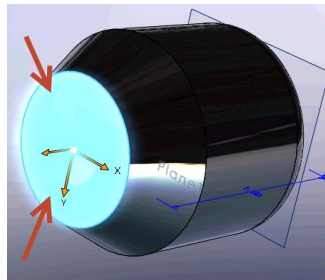18. In the Chamfer Type section select Angle Distance.

19. Select the outside the edge of the component

20. In the Chamfer Parameters section input .25in for the distance and 60.00deg for the angle.

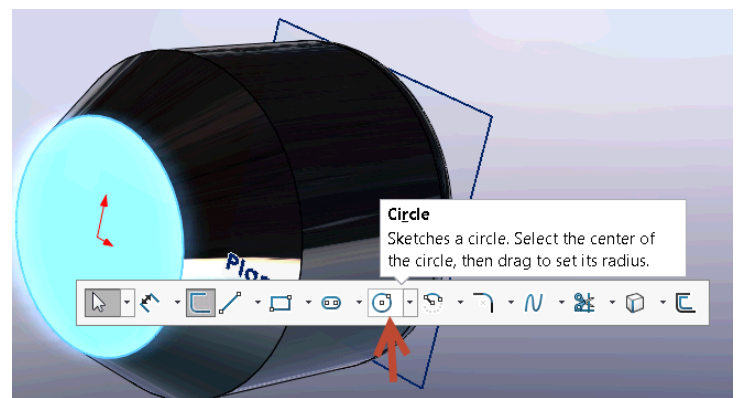21. Select the green check mark at the top of the Chamfer feature manager to accept the new feature.

22. Select the bottom face (it will highlight).

23. From the Command Manager select Sketch tab then select Insert Sketch.

24. Select the S on your keyboard and then select Circle

Circle
Sketches a circle. Select the center of the circle, then drag to set its radius.
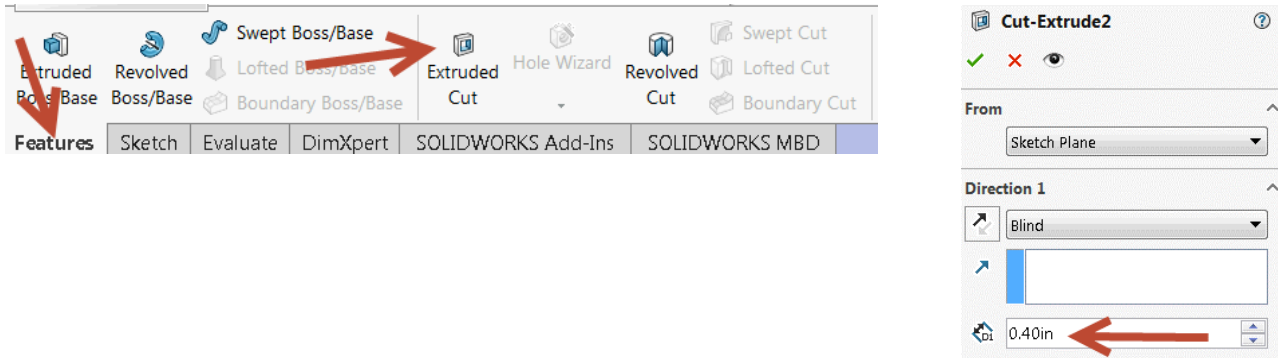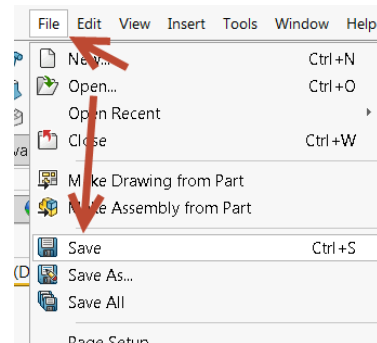
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25. With the mouse pointer select the RED origin and in the Parameter Manager insert .37 for the radius
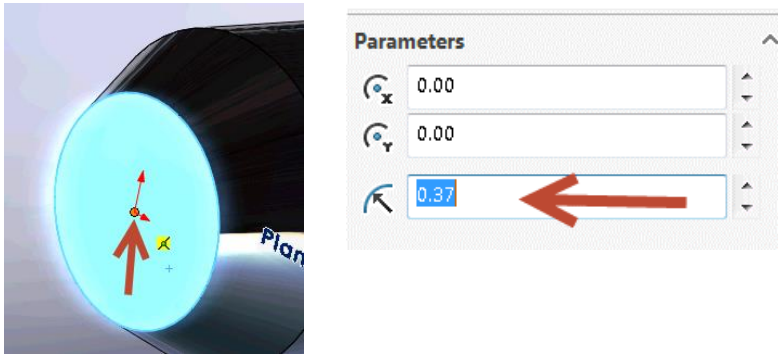
26. From the Command Manager select the Features tab and then select Cut Extrude, and enter 0.40 for the depth of cut in the Direction 1 dimension field. Select the green check mark to accept the feature.

27. In the top menus, select File \ Save, name your part "Throttle Knob" and close the part.

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Create Assembly

Now, assemble the parts into a assembly. In the interests of time, the rest of the parts have been modeled for you. You will have to use the ZOOM and Rotate to manipulate the assembly components as you mate them into position).

1. Create a New Assembly. In the top menu, select File \ New to open the New SOLIDWORKS Document window and select Assembly. Select Ok.
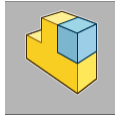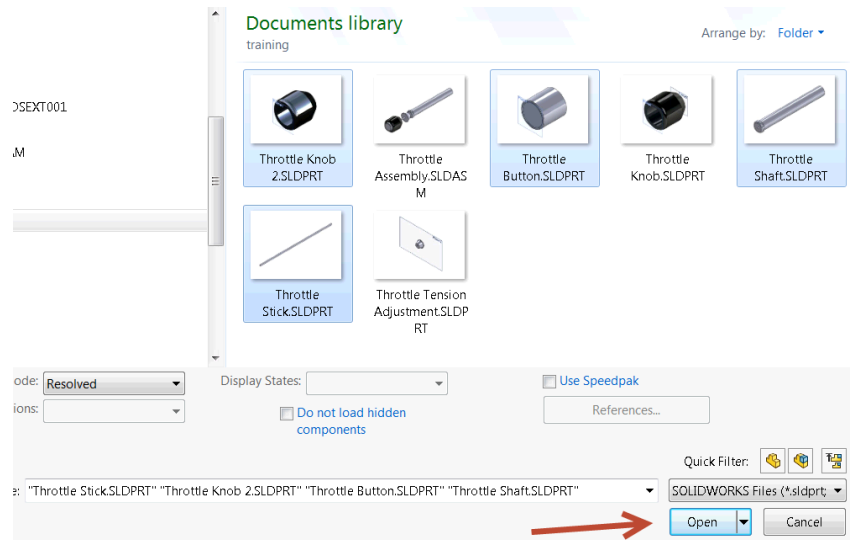
2. The Begin Assembly manager will appear and the Open file dialog box.
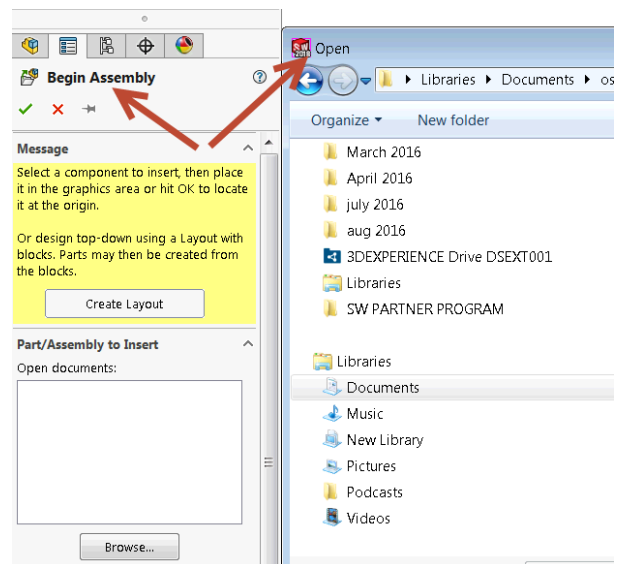
3. Hold down the Ctrl key on your keyboard and select the following components in order
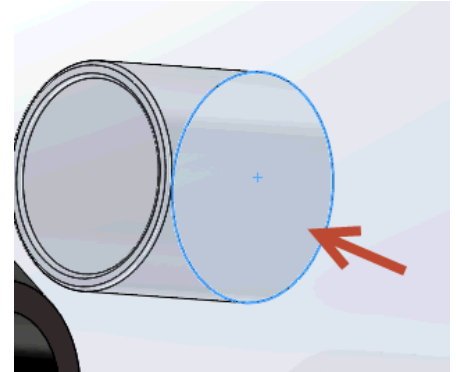
   a. Throttle Button.sldprt
   b. Throttle Knob.sldprt
   c. Throttle Shaft.sldprt
   d. Throttle Stick.sldprt

4. Select the Open button and select the green check mark at the top of the Begin Assembly manager. SOLIDWORKS will begin inserting components into the assembly space using the order on which the components were selected.

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5. Using your mouse place the components were you want them to go and select the left mouse button (LMB) to drop them in-place


6. Select the Mate icon to start putting the components in the assembly together. Select the bottom face of the Throttle Button and the inside bottom face of the Throttle Knob. Select the Mate type Coincident. This mate will make sure the bottom Throttle Button will mate the bottom Throttle Knob. Select the green check mark to accept and mate the parts.
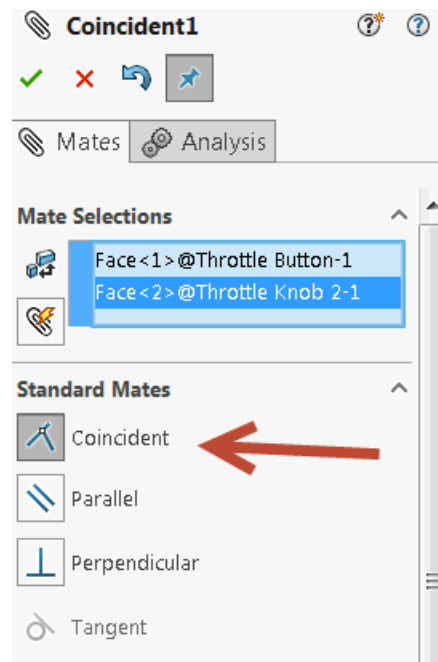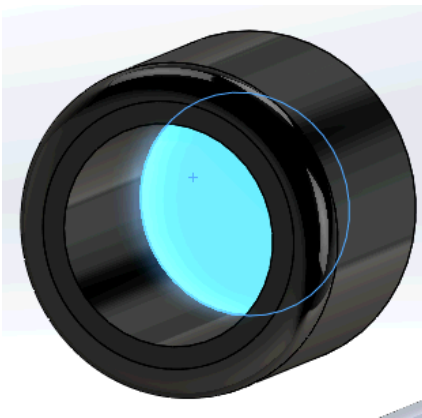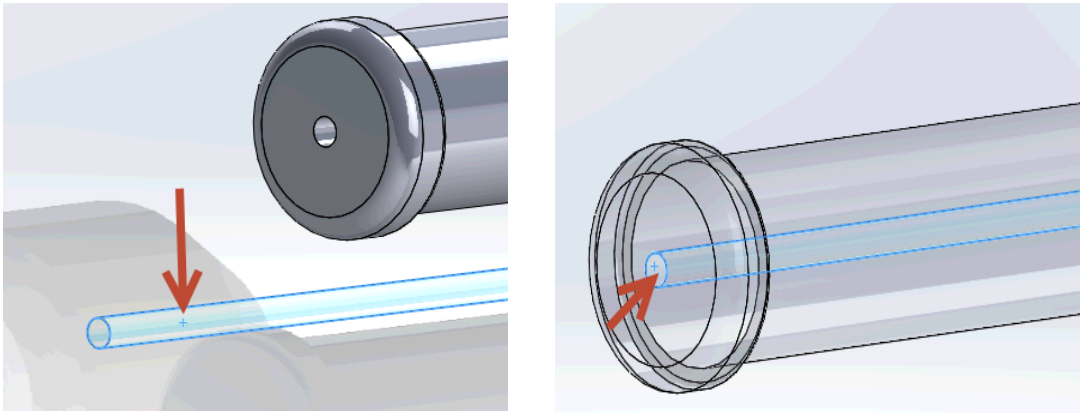

7. Next select the outside face Throttle Button and the inner face of the Throttle knob select the Mate type Concentric. This mate will make sure that both components are Concentric inline down the axis. Select the green check mark.
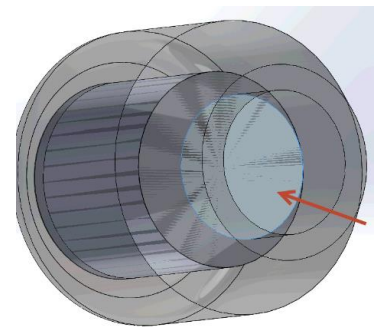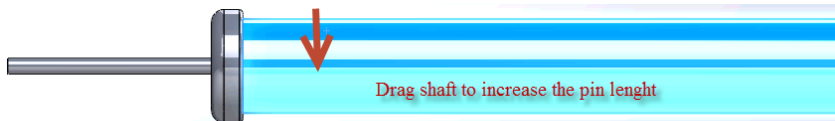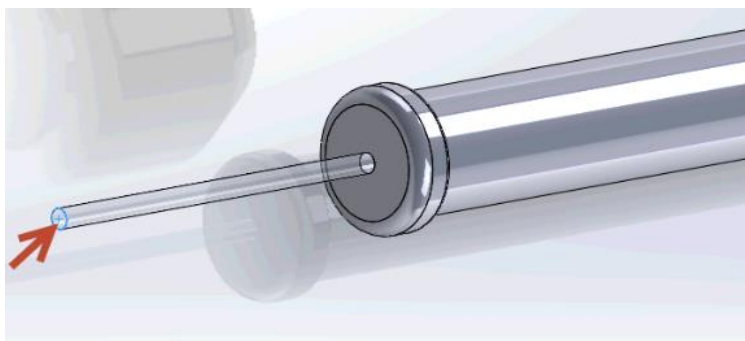
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8. Next select the outer face of the Throttle Pin and the Inner face of the cut-extrude down the middle of the Throttle Shaft. Select the Mate type Concentric. You should see that the Throttle Pin is now in line with the Throttle Shaft. Select the green check mark to accept.

9. With the mouse pointer select and hold down the left mouse button (LMB) and drag the component allowing the Throttle Pin to be more exposed for applying the next mate.


Drag shaft to increase the pin lenght

10. Select the top face of the Throttle Pin and the bottom inner face of the Throttle Knob and select the Mate type Coincident. Select the green check mark to accept.

11. Select the green check mark in the mate fly-out.

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12. Select the outside face of the Throttle Pin and the Inside face of the Throttle Knob select the Mate type Concentric
    This mate will make sure that both components are Concentric inline down the axis.
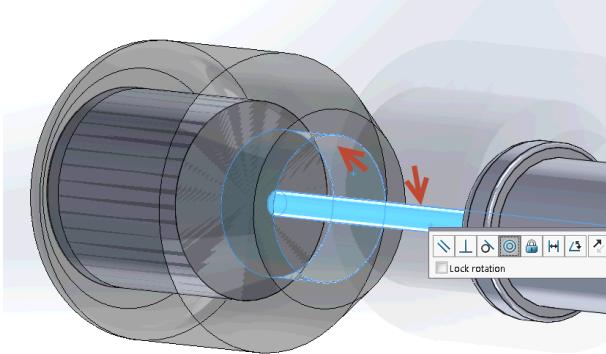
13. Your assembly should look like this. Select File \ Save in the top menu.