

E91: DYNAMICS
Lab 2: SolidWorks Fundamentals

Introduction

The goal of this lab is for you to familiarize yourself with the Fundamentals of SolidWorks. In certain parts of this lab, you will be referred to the “Beginner’s Guide to SolidWorks 2006” by Alejandro Reyes, available on the course website.

Task 1: Pressure Plate

For this task you will go through SolidWorks’ On-line Tutorial – Lesson 0. This lesson will guide you through the generation of the part model and drawing shown in Figure 1.

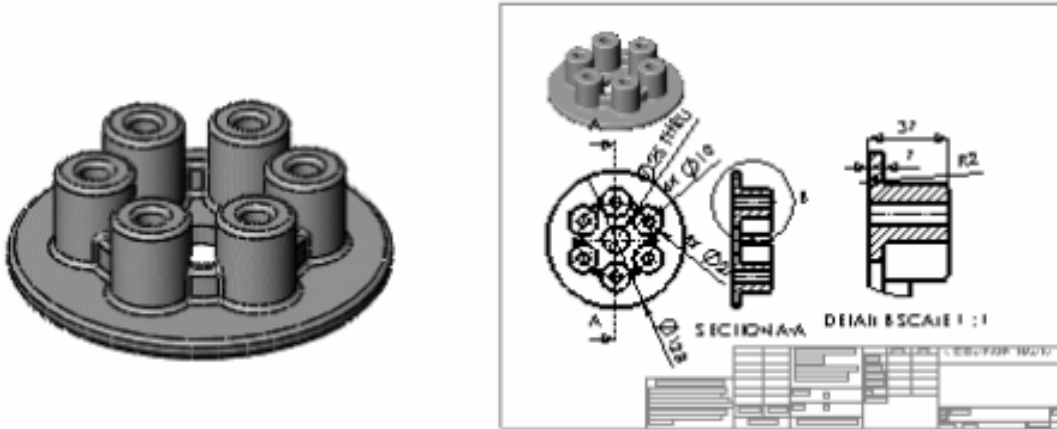


Figure 1. Part for Task 1

Launch the SolidWorks application and go to the *Help* Menu and select *SolidWorks Tutorials*. This will launch the *On-line Tutorial* on the right side of the screen. Select the *30 Minute Lesson* and complete this lesson.

In some versions of SolidWorks there is a bug which results in crashing the application when you have to use the section view when drawing. Instead of creating a section view, create an alternate view: left, right, top, or bottom (these are equivalent since the part is symmetric). Make a detailed view of an area around the cylinders in this view. Use these two views as the ones to add the center-lines to.

Task 2: The Housing

For this task you will go through the tutorial in the “Beginner’s Guide to SolidWorks 2006” by Alejandro Reyes. This will be a separate drawing file from Task 1.

Task 3: Part Model

Complete *Lesson 1* in the SolidWorks Tutorials. This lesson will familiarize you with the techniques used to generate part models.

Task 4: Creating U-Joint Parts

In this portion of the lab, I will take you through the steps required to generate parts and assemble them into the U-Joint shown in Figure 2. Please save each part into a separate drawing file.

Housing

Let’s begin with the housing whose schematic is shown in Figure 3(a). In a new SolidWorks drawing file, create the sketch shown in Figure 3(b) on the Right Plane. The dimensions in Figure 3(b) are in inches. Next extrude the part such that the thickness from the center plane is 1.5 inches. In other words, in the PropertyManager under

Direction 1 select Mid Plane and enter 1.5” for the dimensions. After extrusion, your part should now look like the one shown in Figure 4.

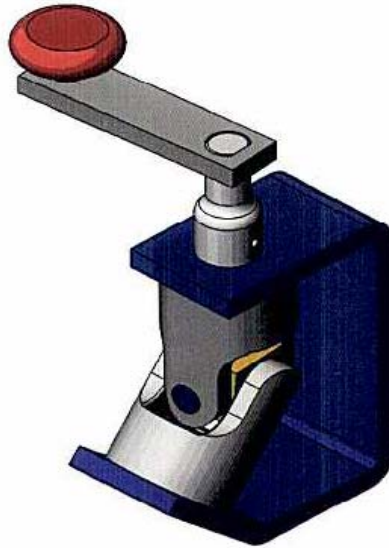
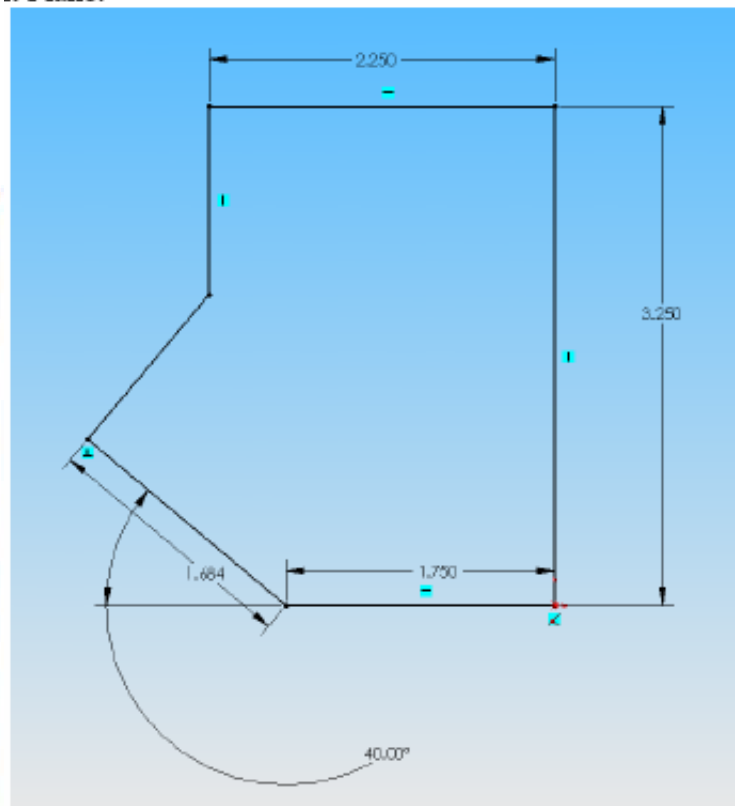


Figure 2. U-joint.



(a)

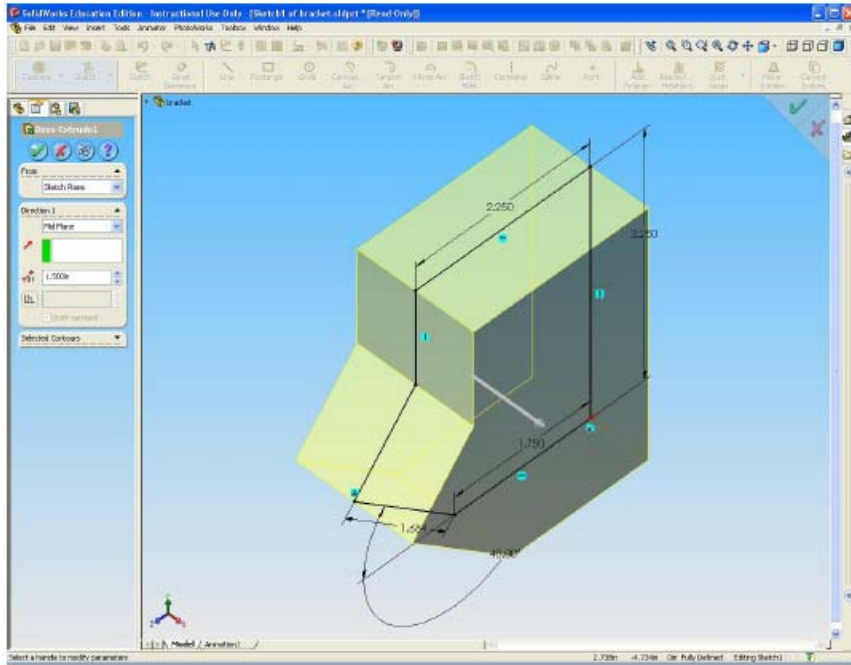


(b)

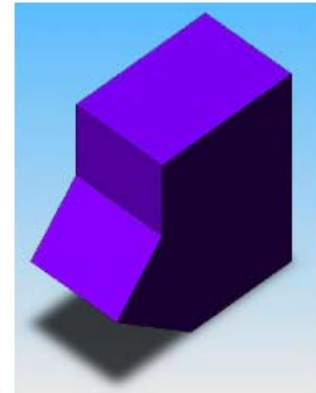
Figure 3. (a) U-joint for Task 3. (b) Initial sketch for the part shown in Figure 2(a).

To hollow out the part, select the 4 faces shown in Figure 5(a) to shell such that the remaining thickness of the part is 0.25 inches. The shelled out part is shown in Figure 5(b). Next, fillet the 3 edges by first selecting the edges shown in Figure 6(a), each with radius 0.125 inches. The filleted part is shown in Figure 6(b).

To drill the hole on the top face of the bracket create sketch 2 on the top face of the bracket. Use the Cut-Extrude tool to cut the hole. See Figure 7. This completes the U-joint bracket.

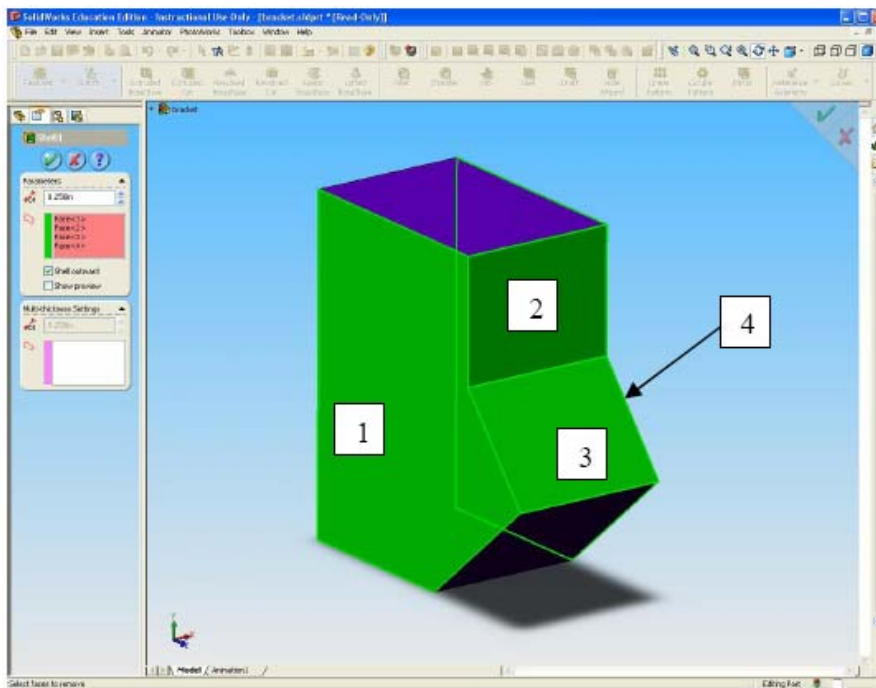


(a)

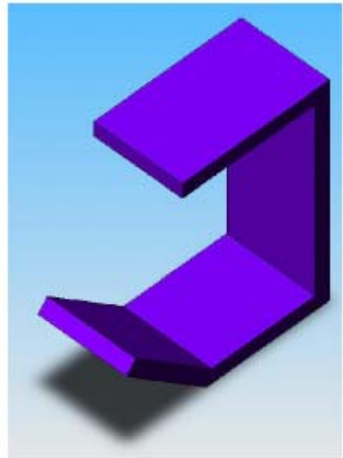


(b)

Figure 4. (a) Extruding the boss. (b) Extruded boss.

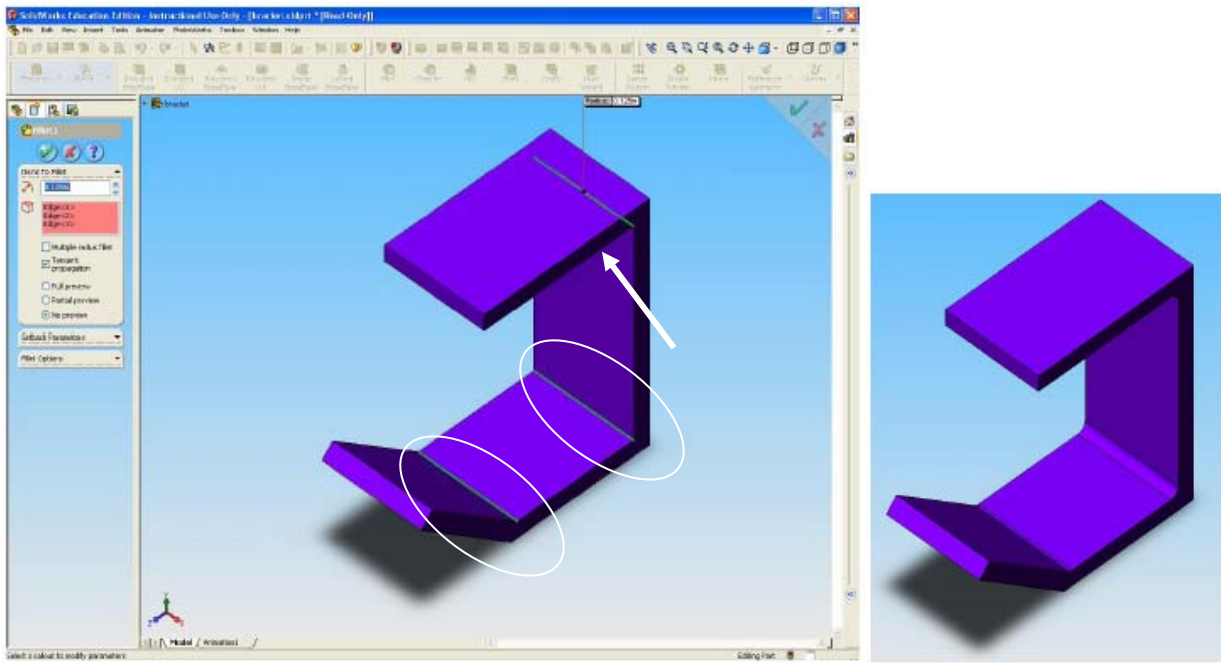


(a)



(b)

Figure 5. (a) Faces to be shelled. (b) Shelled part.



(a) (b)
 Figure 6. (a) Edges shown by the white circles and white arrow. (b) Part with the desired edges filleted.

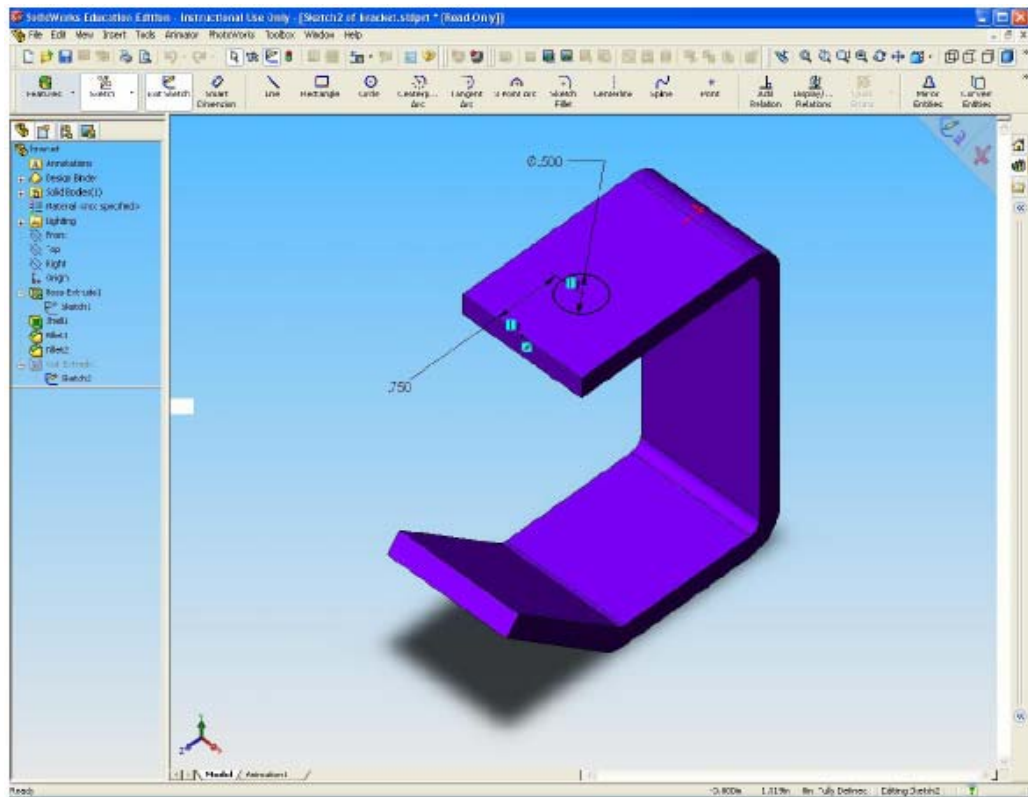


Figure 7(a).

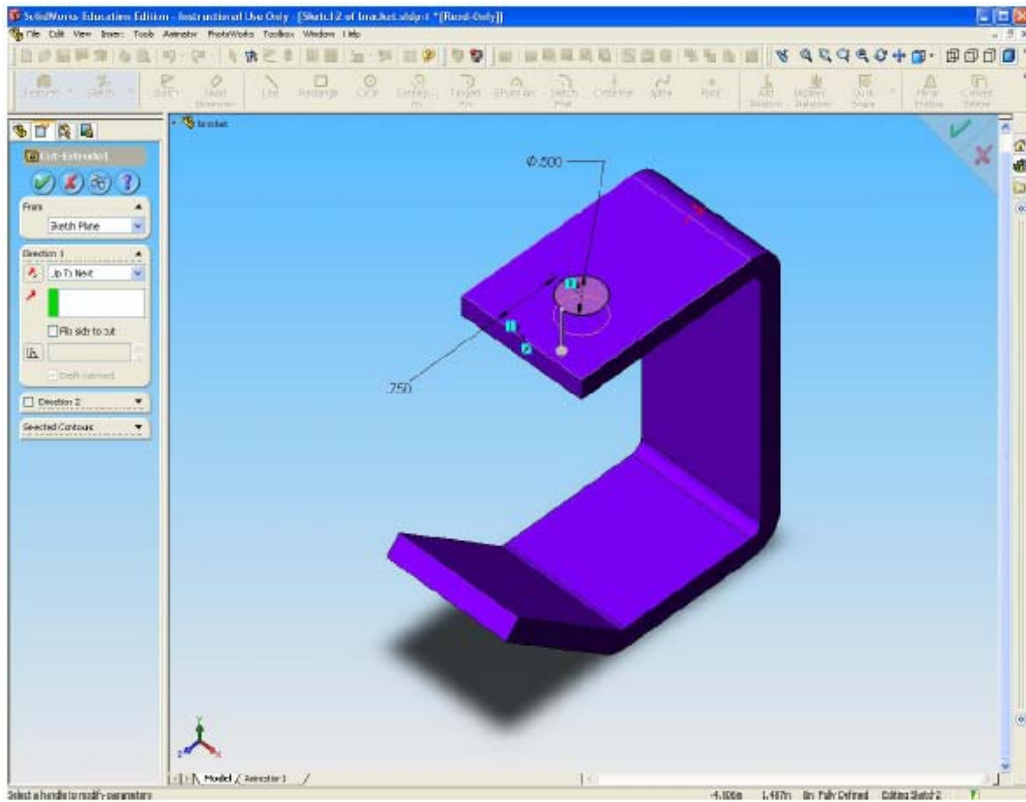
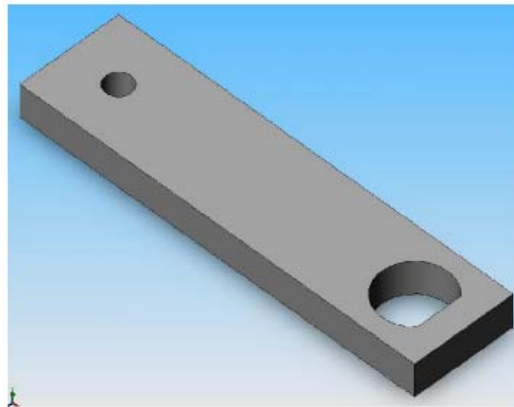
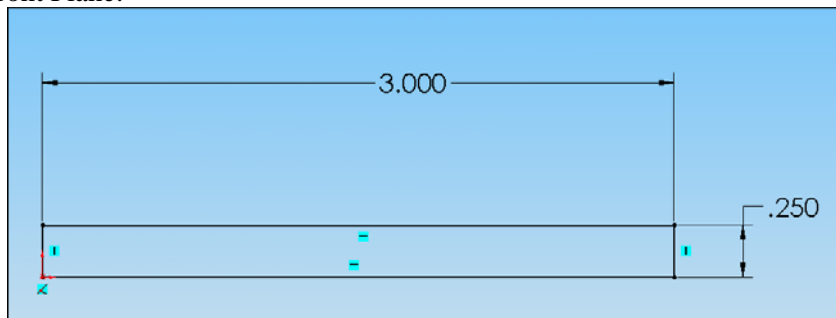


Figure 7(b).

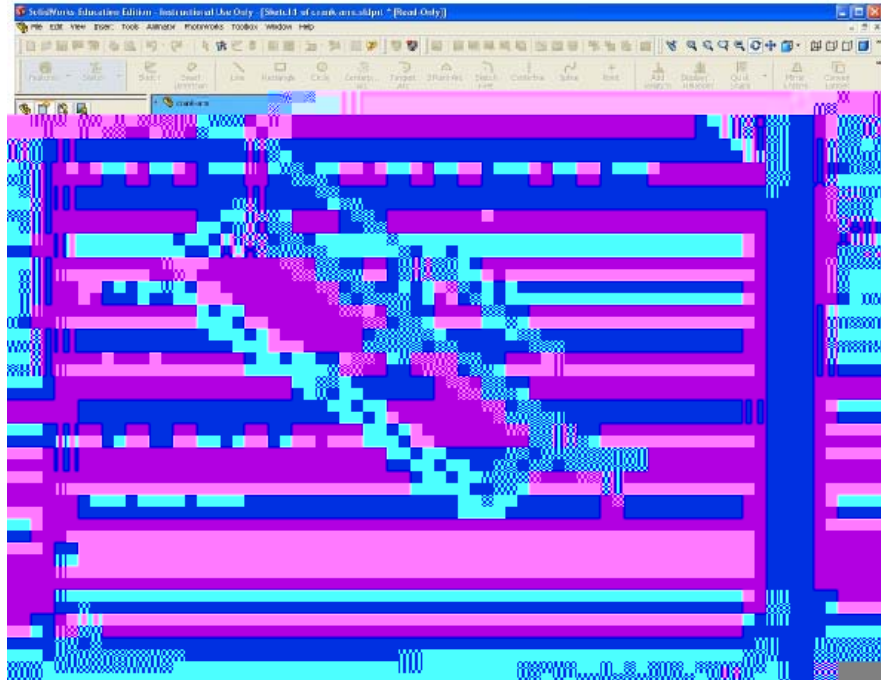
Crank-Arm



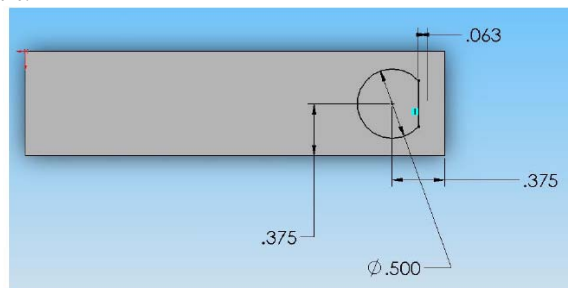
Create sketch 1 on Front Plane:



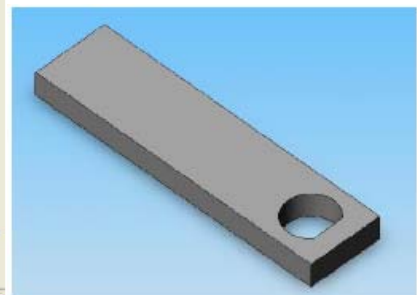
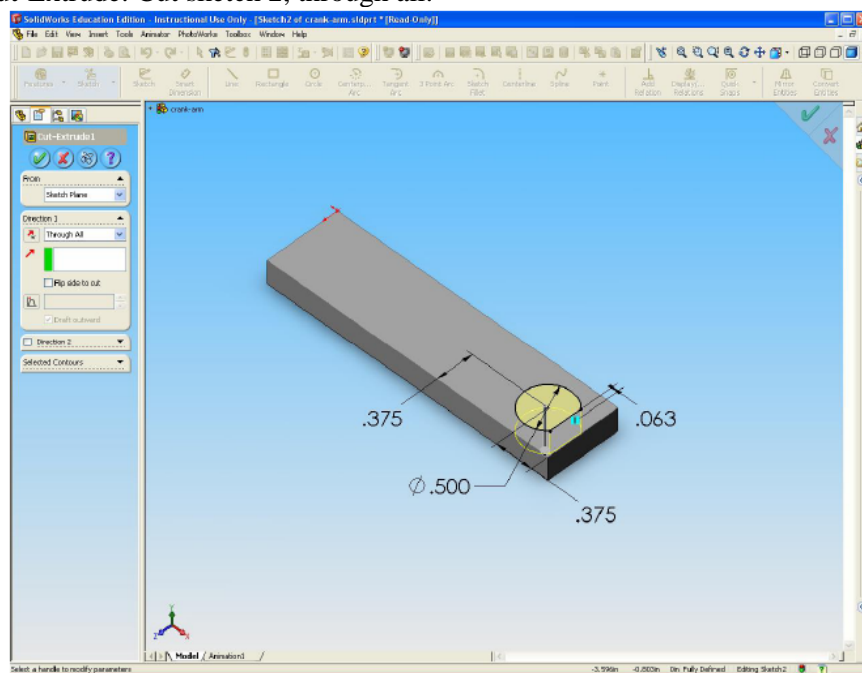
Extrude sketch 1 with Blind, distance = 0.750.



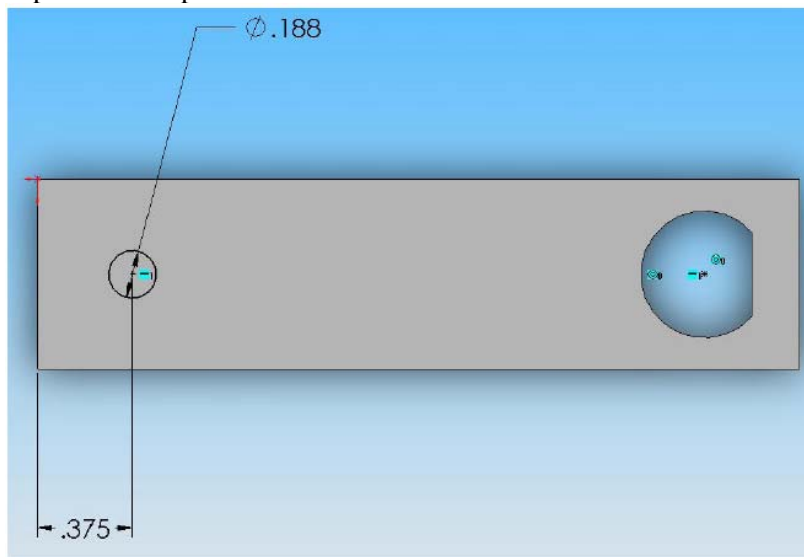
Create sketch 2 on top face of part:



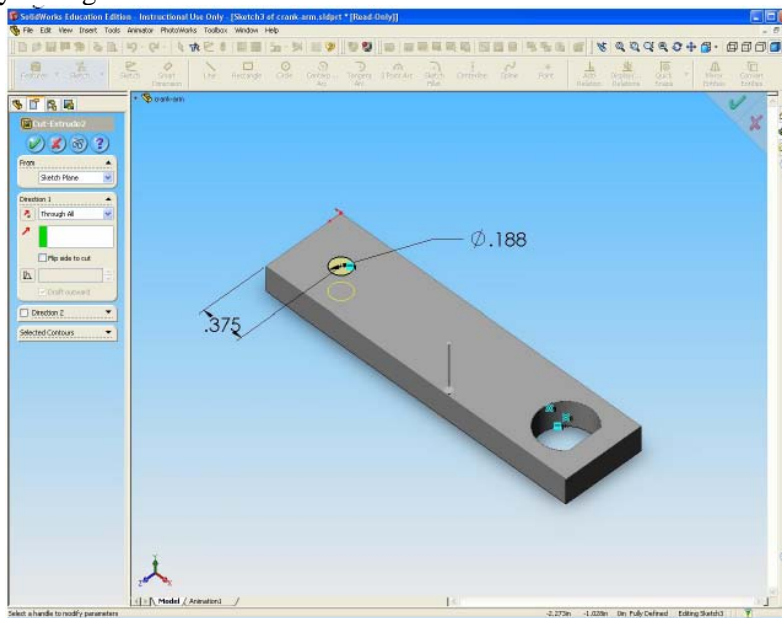
Cut-Extrude: Cut sketch 2, through all.



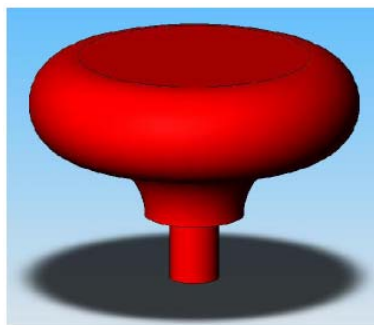
Create sketch 3 on the top face of the part.



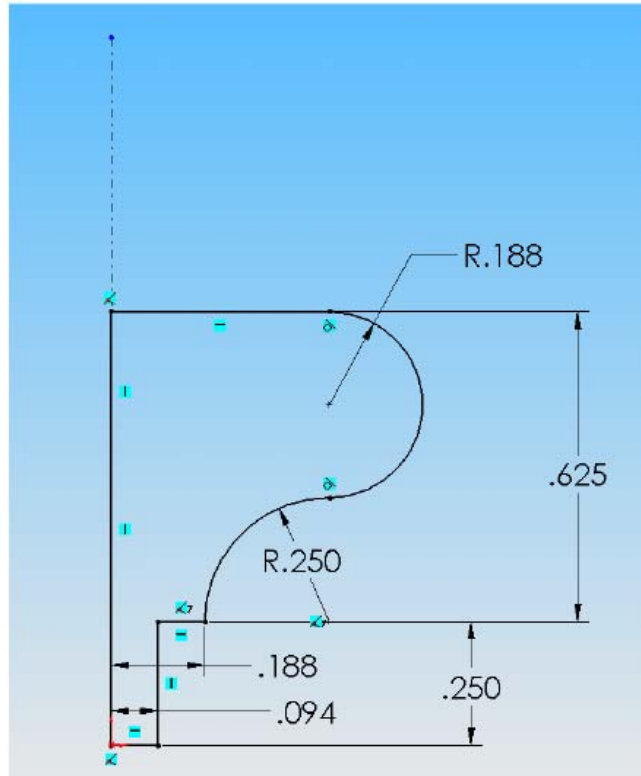
Cut sketch 3 all the way through. This finishes the crank-arm.



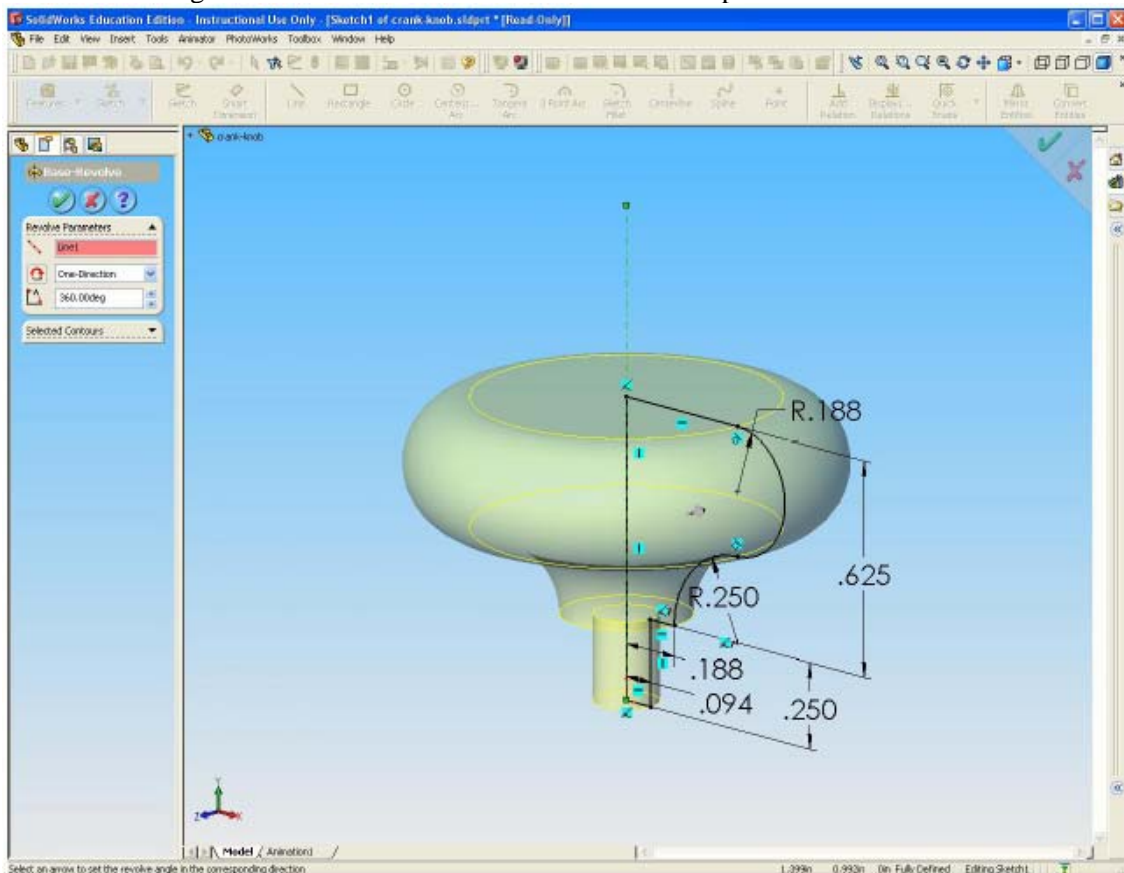
Crank Knob



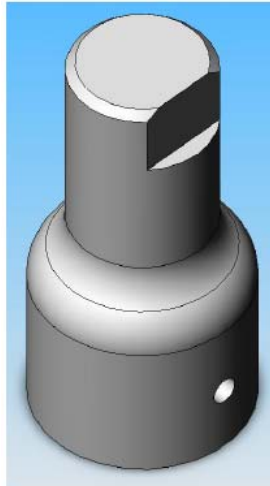
Create Sketch 1 on the Front Plane (don't forget to include the centerline).



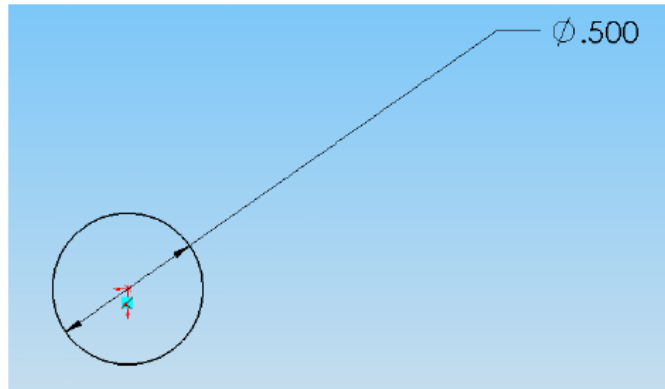
Revolve Sketch 1 360 degrees use the Base Revolve action. This completes the crank knob.



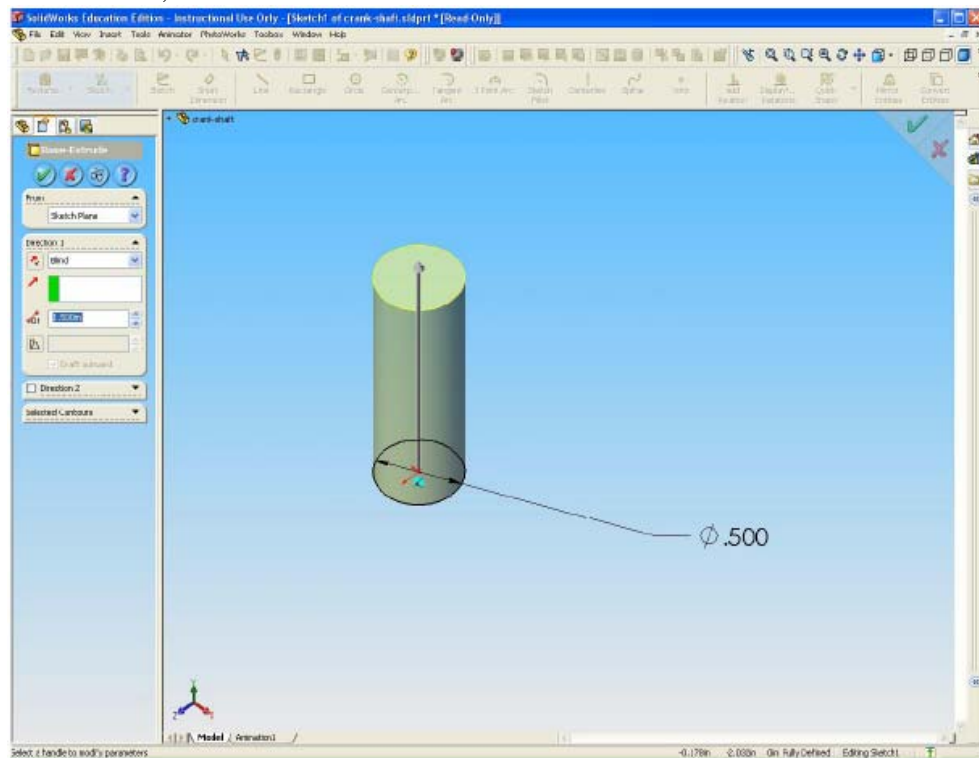
Crank Shaft



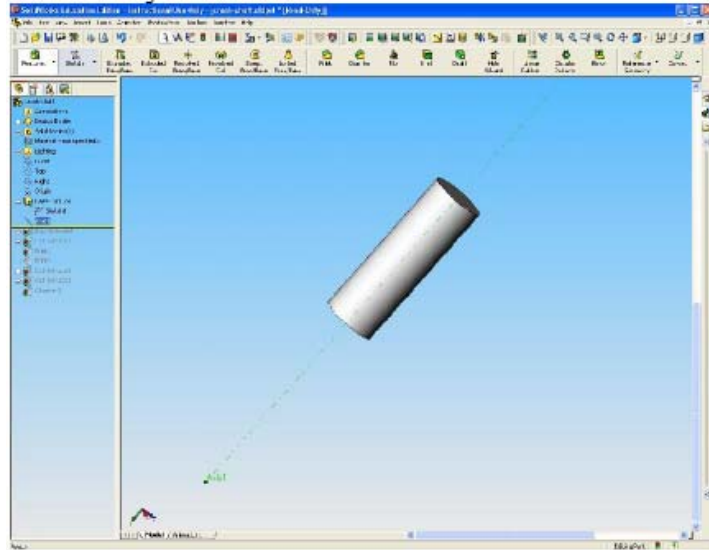
Create Sketch 1 on the Top Plane.



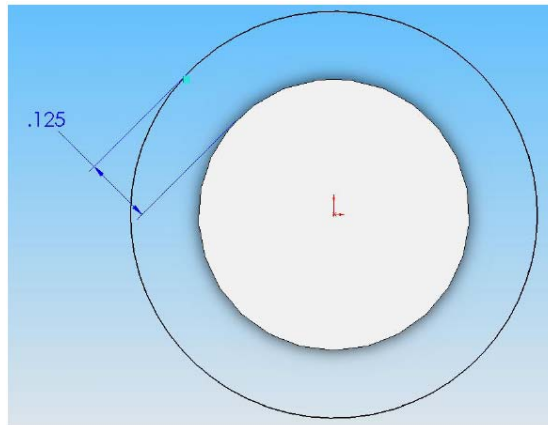
Extrude Sketch 1 with Blind, Distance 1.5 inches.



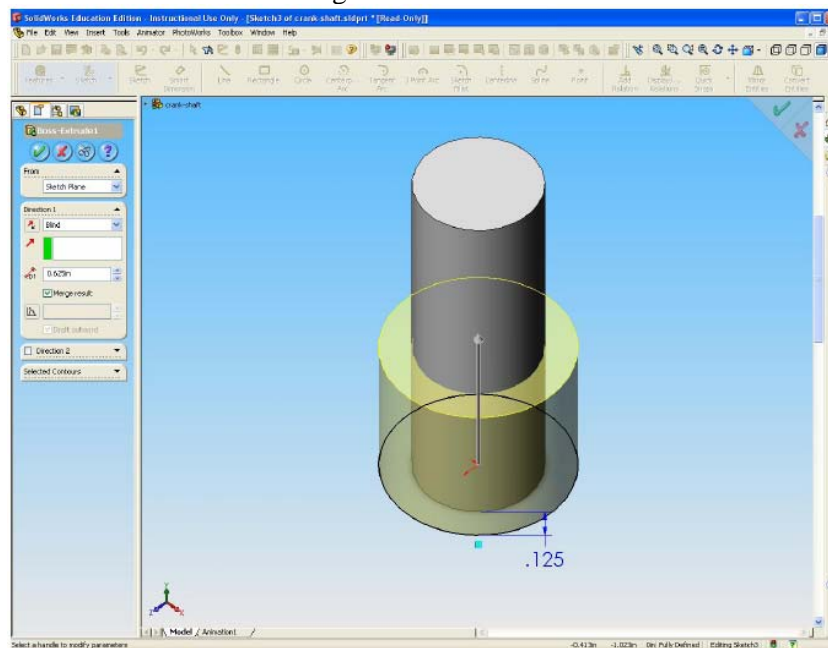
To insert a vertical axis through the center of the cylinder, from the *Insert* menu > *Reference Geometry* > *Axis*.



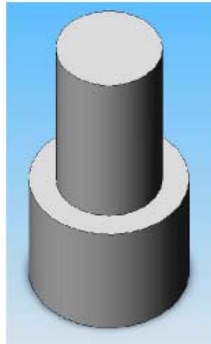
Create Sketch 2 on the Top Plane.



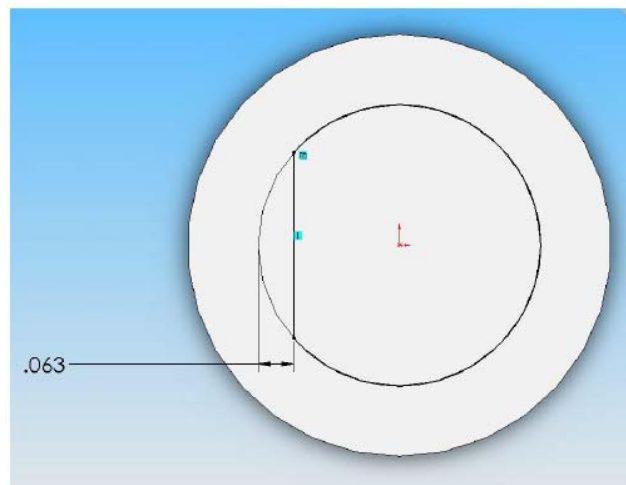
Extrude Sketch 2 with distance = 0.625" and merge the result.



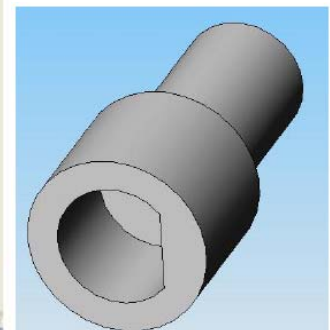
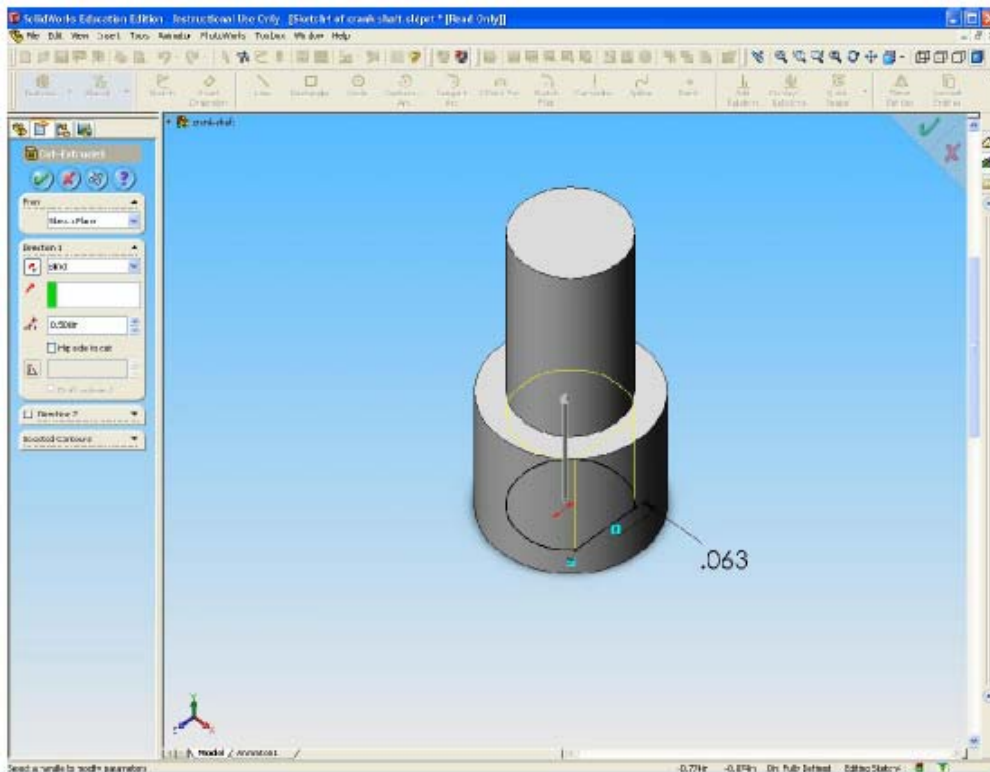
This is what the part should look like at this point.



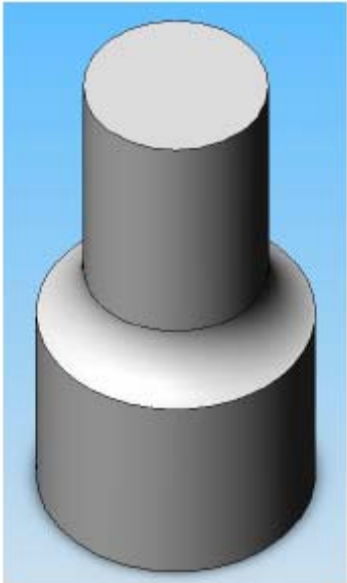
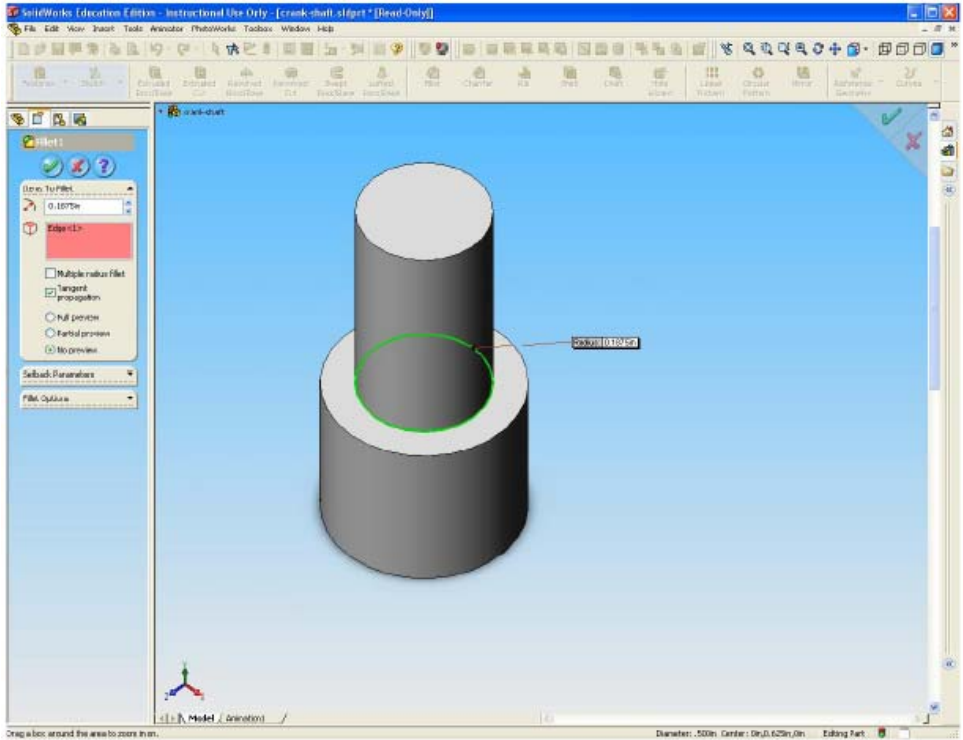
Create Sketch 3 on the Top Plane.



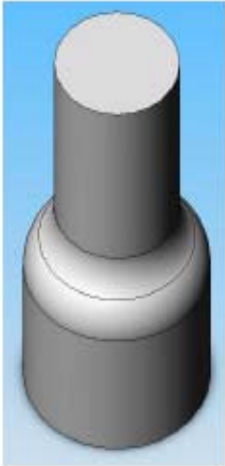
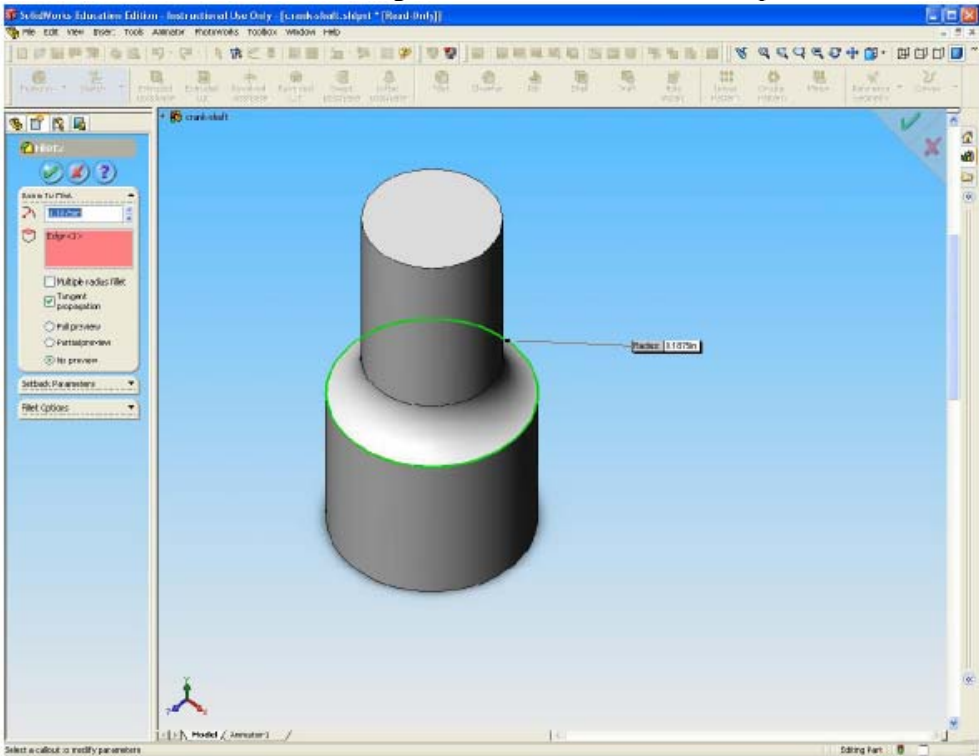
Cut/Extrude Sketch 3, Blind, Distance = 0.5". After the cut, the part should look like the image on the right.



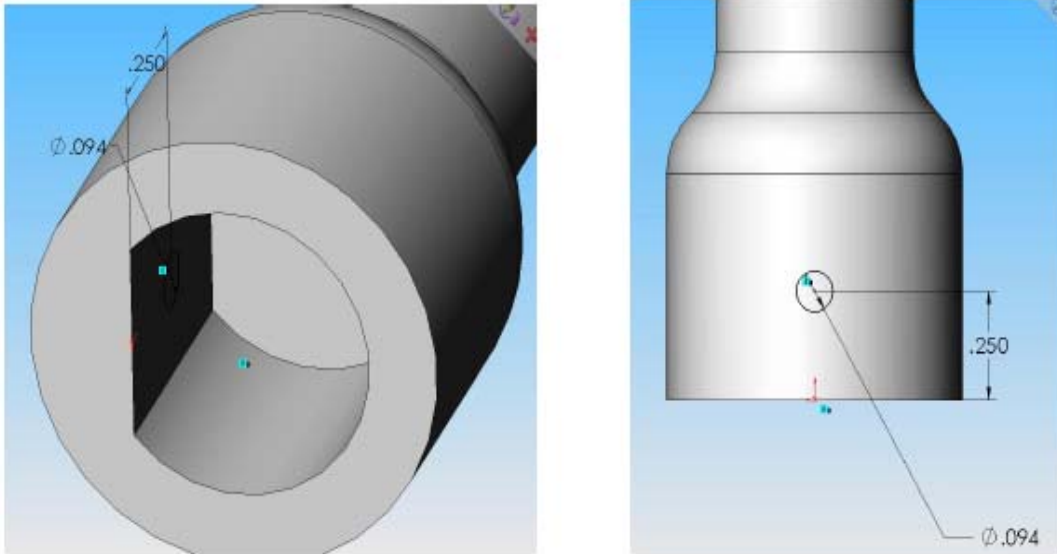
Next, create Fillet 1 on the selected edge with radius = 0.1875". The filleted part is shown to the right.



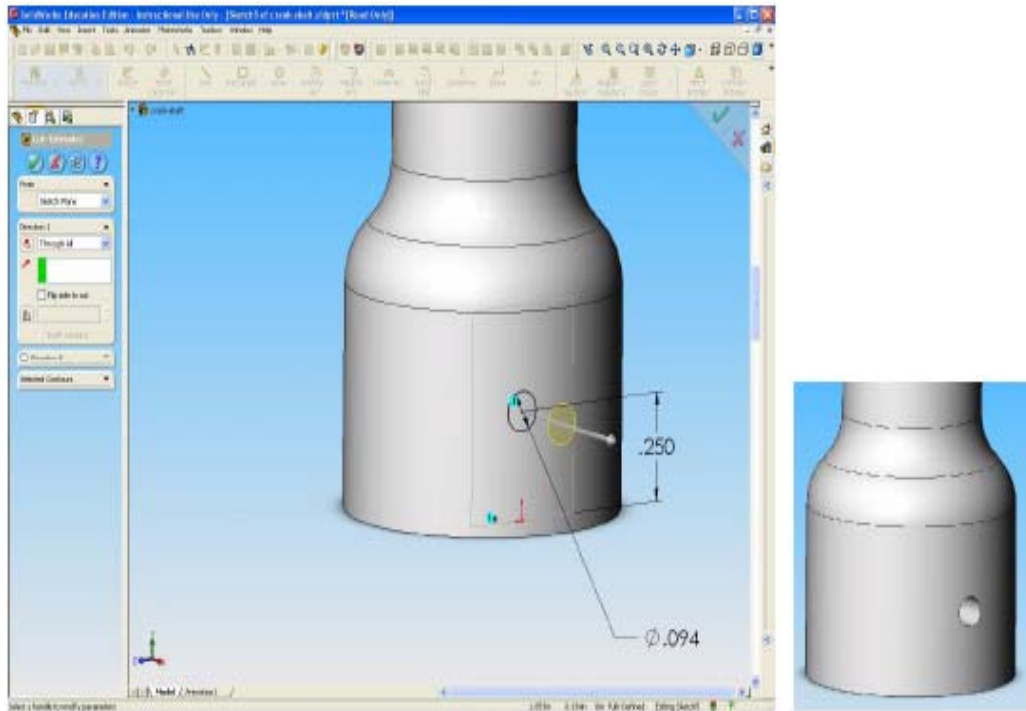
Then, create Fillet 2 on the selected edge with radius = 0.1875". The part after Fillet 2 is shown to the right.



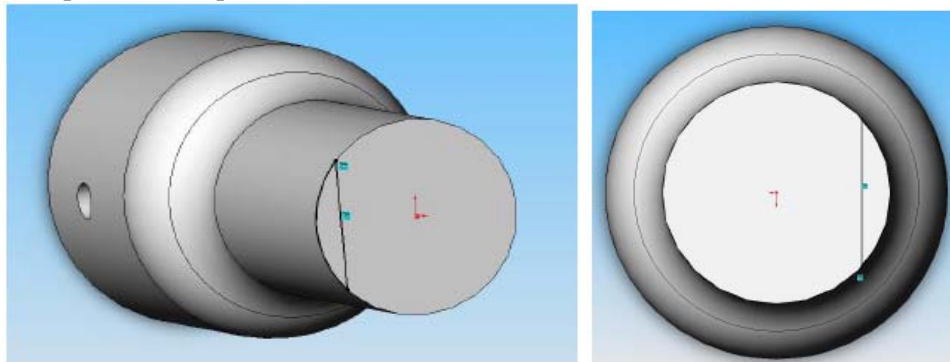
Create Sketch 4 on the inside flat face of the bottom cut-out portion of the part:



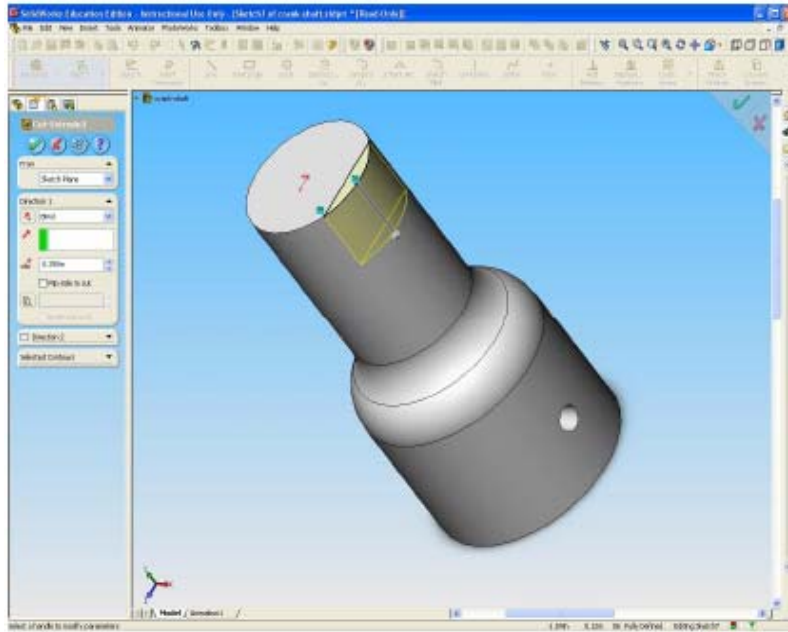
Cut through Sketch 4 to obtain:



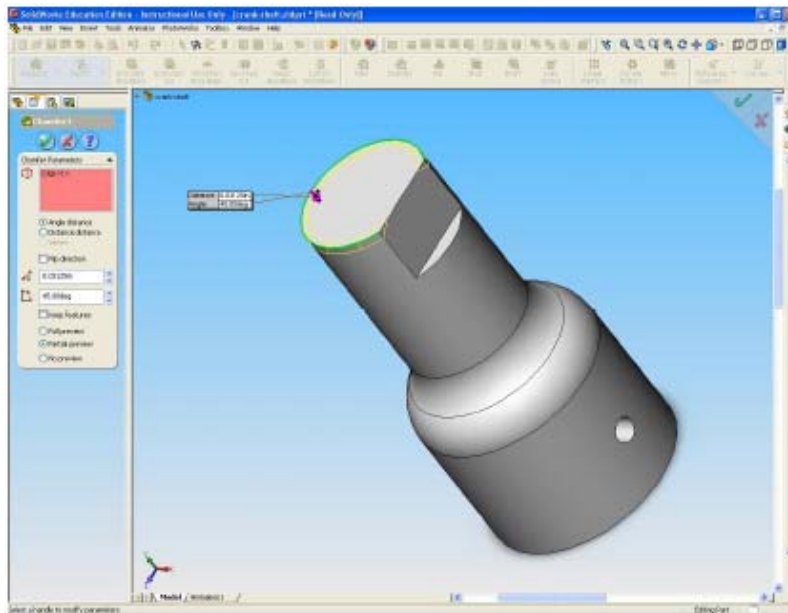
Create Sketch 5 on top face of the part.



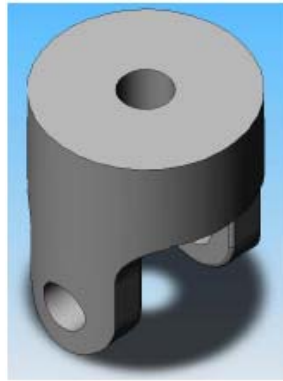
Cut out Sketch 5, Blind, Distance = 0.250”.



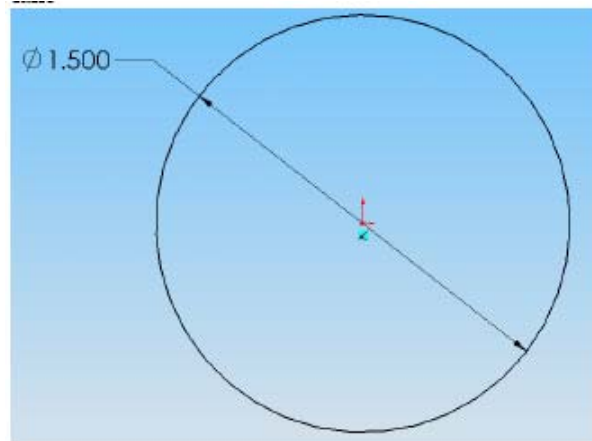
Create chamfer on selected edge with distance of 0.03125” and angle of 45 degrees. This completes the crankshaft.



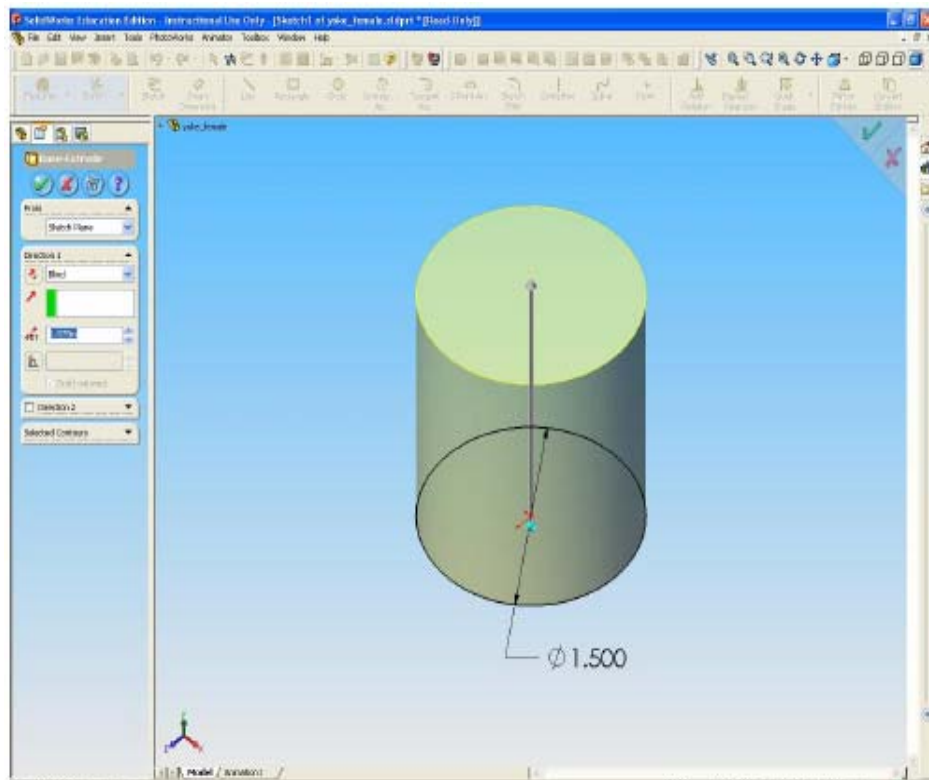
Yoke (Female)



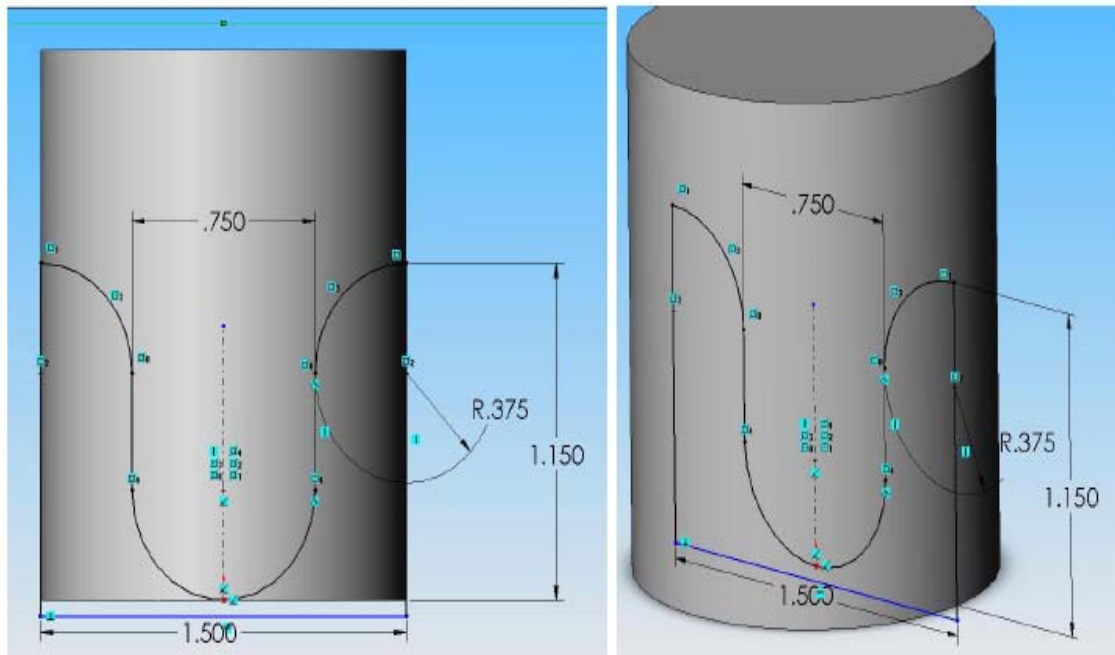
Create Sketch 1 on Top Plane.



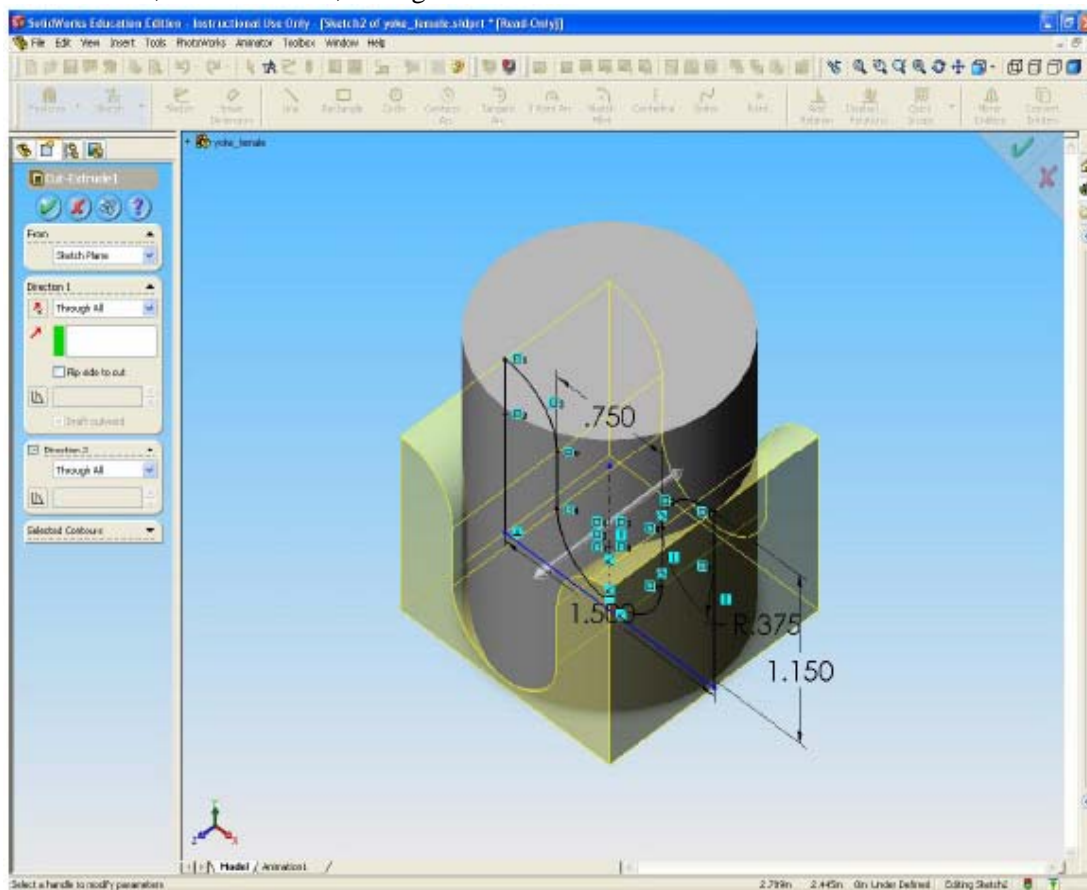
Extrude Sketch 1, distance = 1.875".



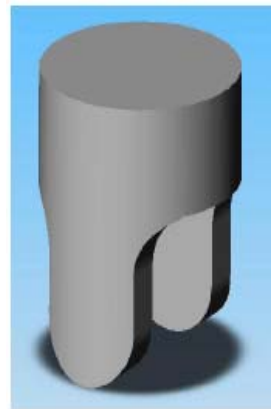
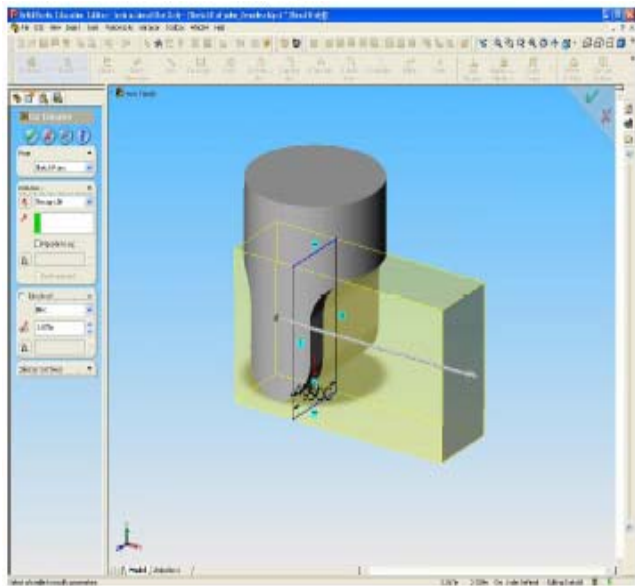
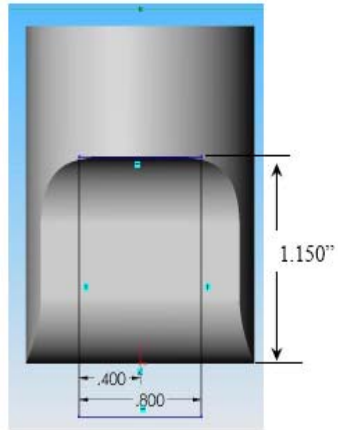
Create Sketch 2 on Front Plane.



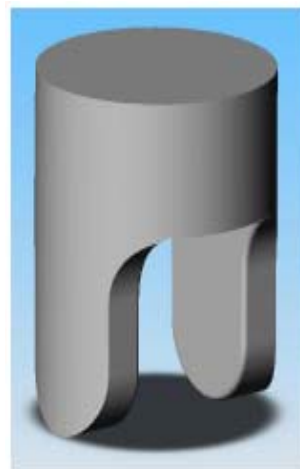
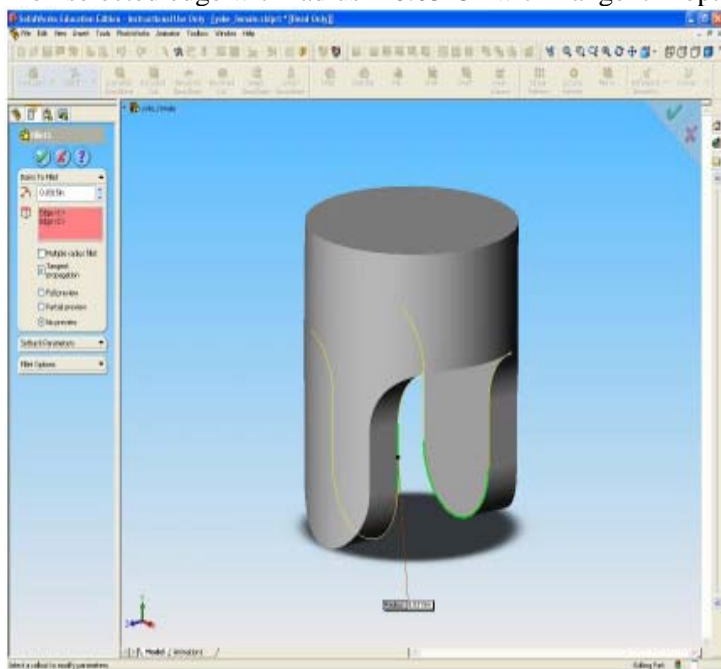
Cut Sketch 2, two directions, through all.



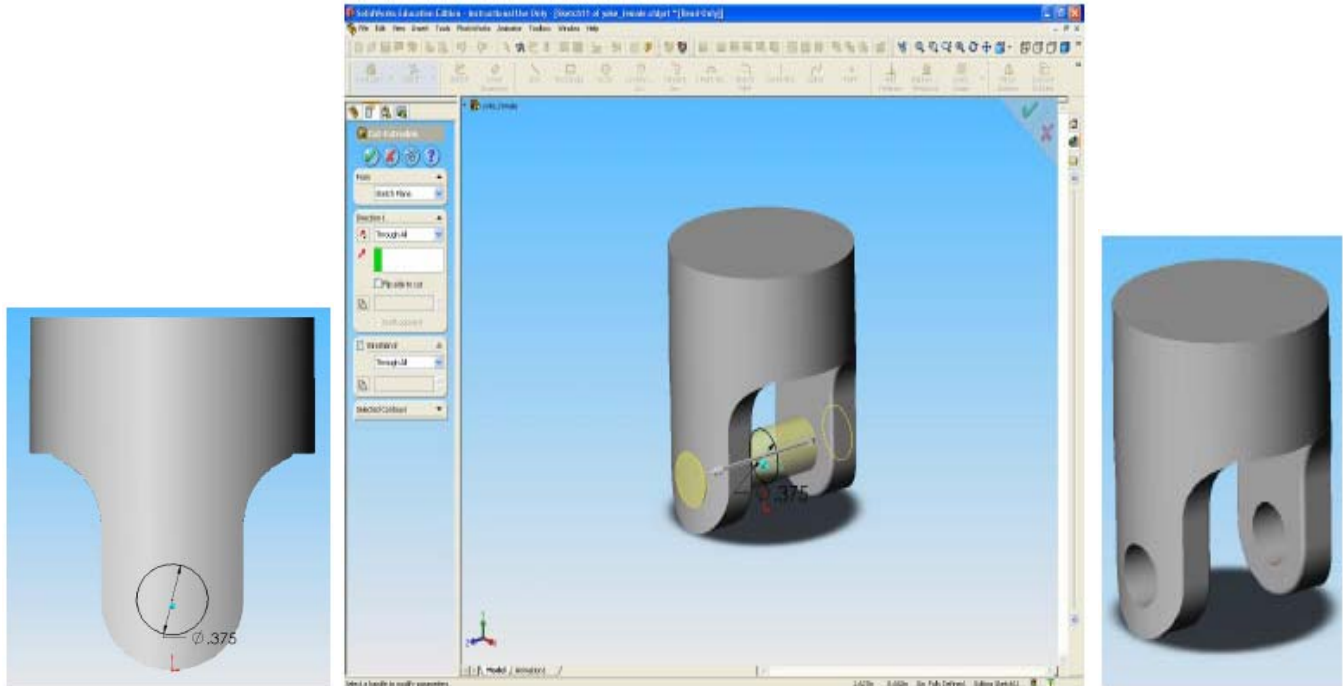
Create Sketch 3 on Right Plane, cut Sketch 3, two directions, through all.



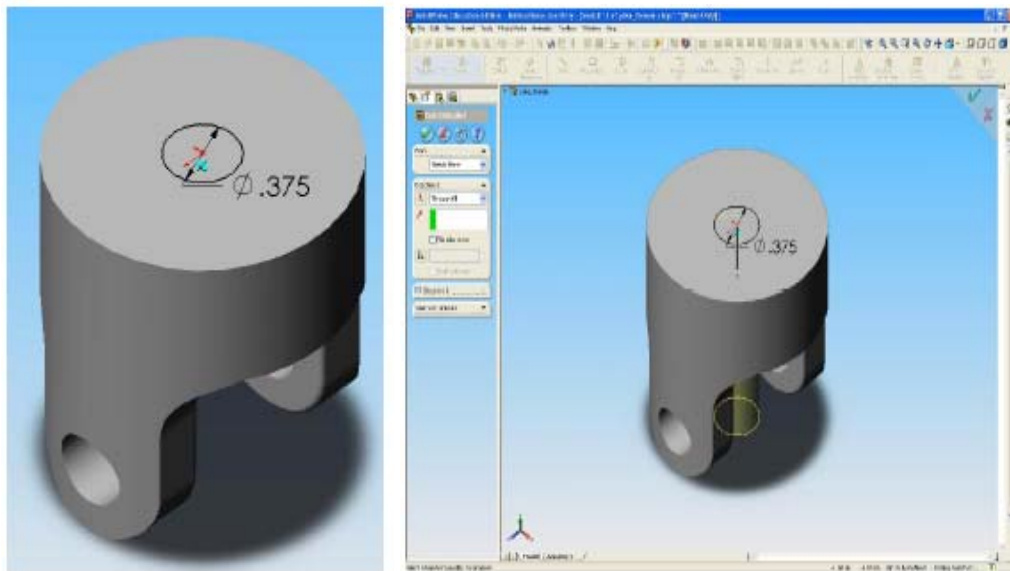
Create Fillet 1 on selected edge with radius = 0.0315" with Tangent Propagation.



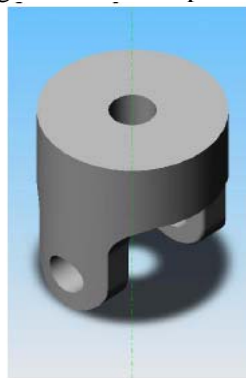
Create Sketch 4 on Front Plane. Cut Sketch 4, two dimensions, through all.



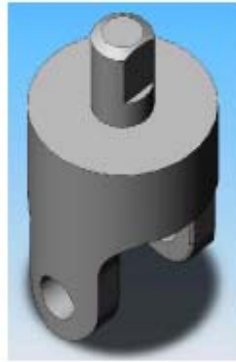
Create Sketch 5 on top face of part. Cut Sketch 5 through all.



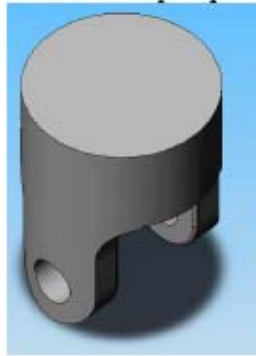
Insert Reference Geometry, select Axis, through the center of part. This finishes the female end of the yoke.



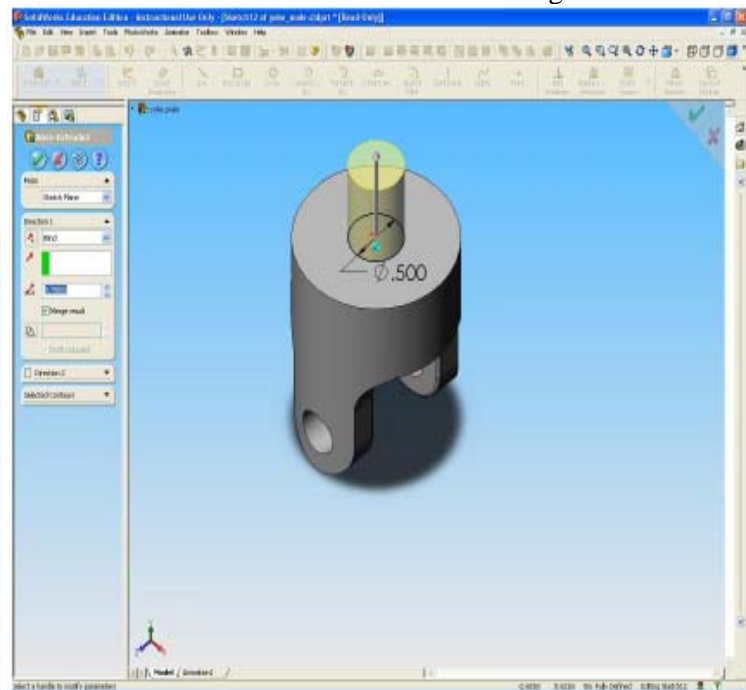
Yoke – Male



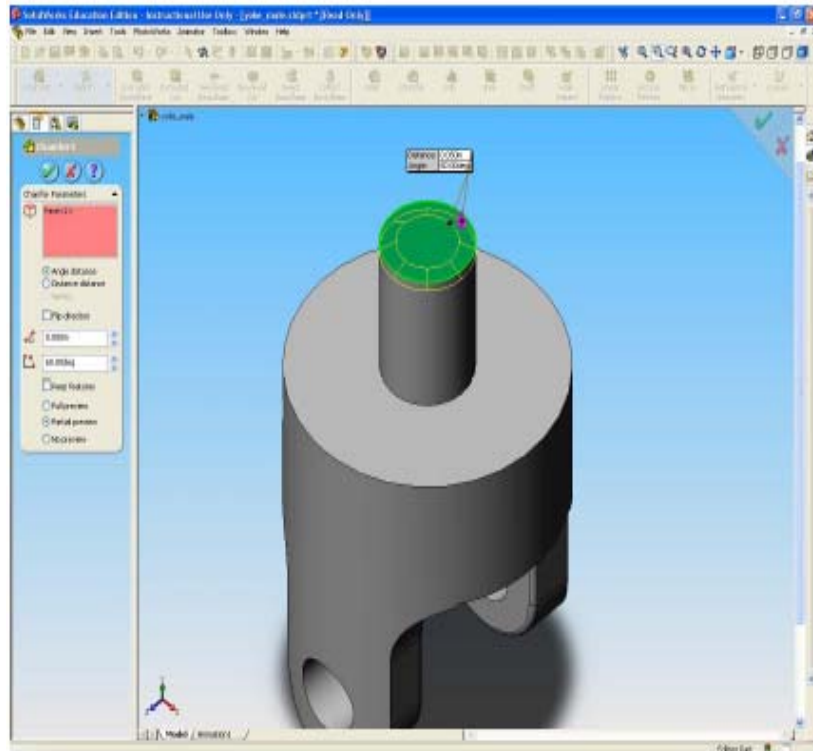
Follow the same directions for the Yoke – Female part up to and including the instructions for Fillet 1.



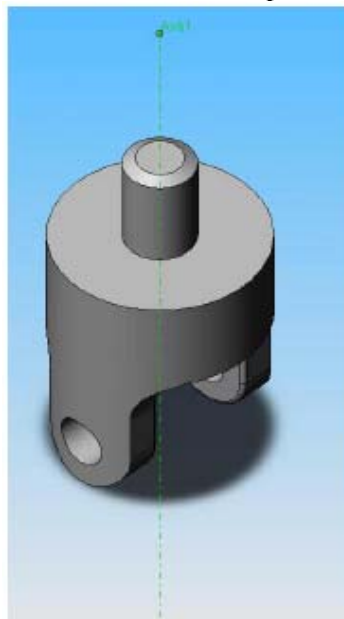
Create new sketch on top face of part. Extrude sketch to distance = 0.750" and merge the result.



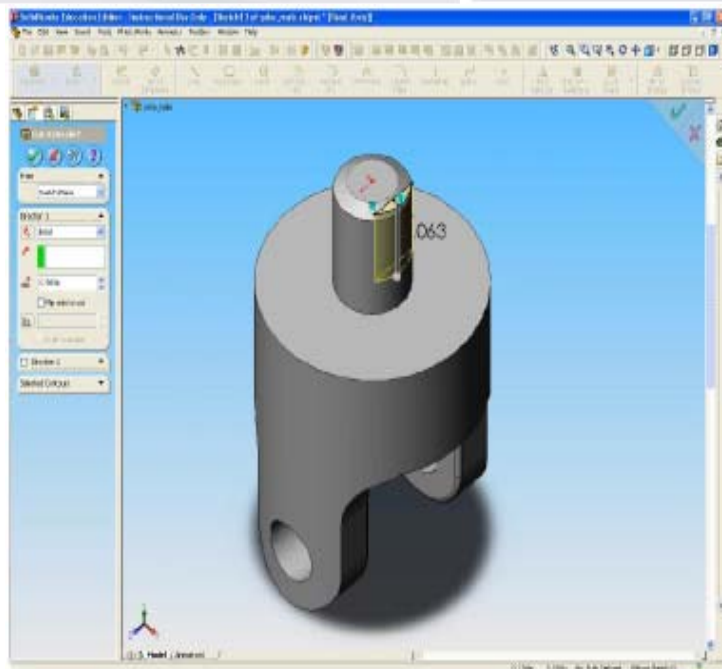
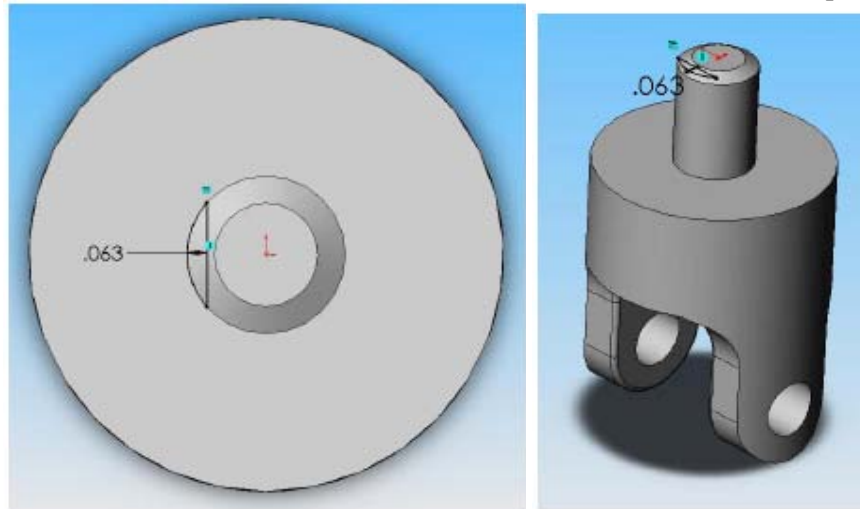
Create Chamfer on selected face, distance = 0.050", angle = 60 degrees.



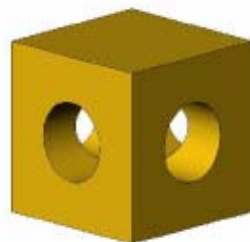
Insert Reference Geometry, axis along vertical dimension of the part.



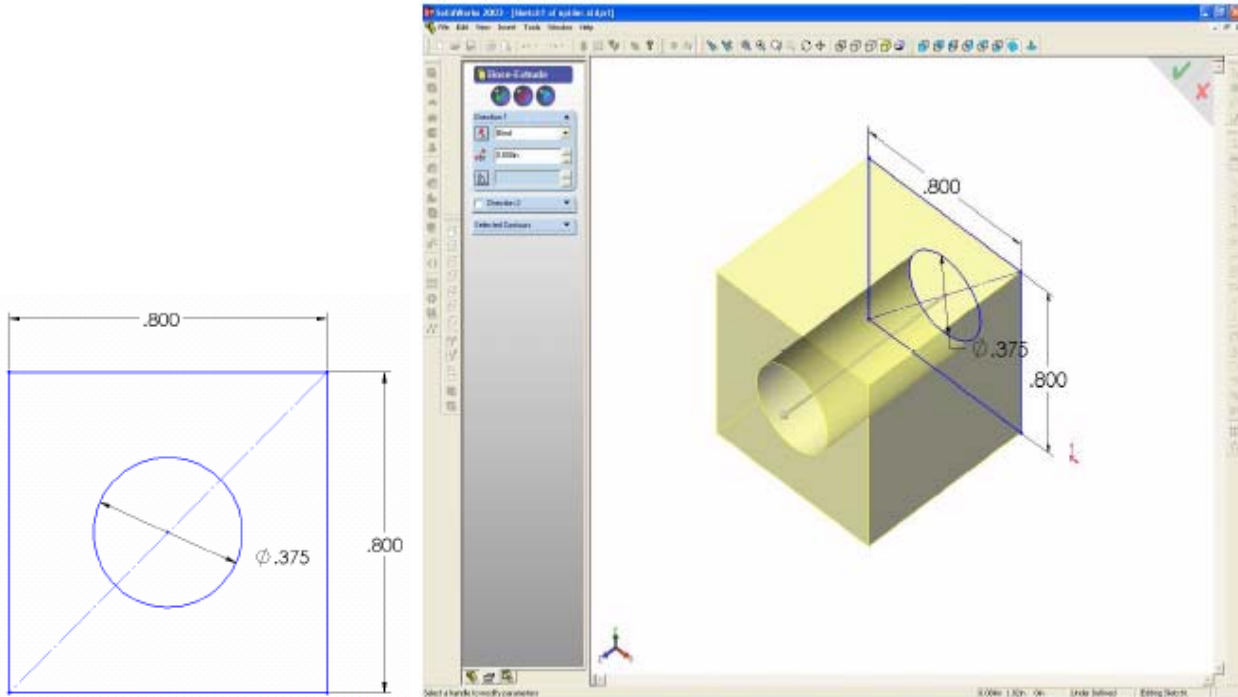
Create the next sketch on top face of part. Cut sketch, Blind, distance = 0.500". This completes the part.



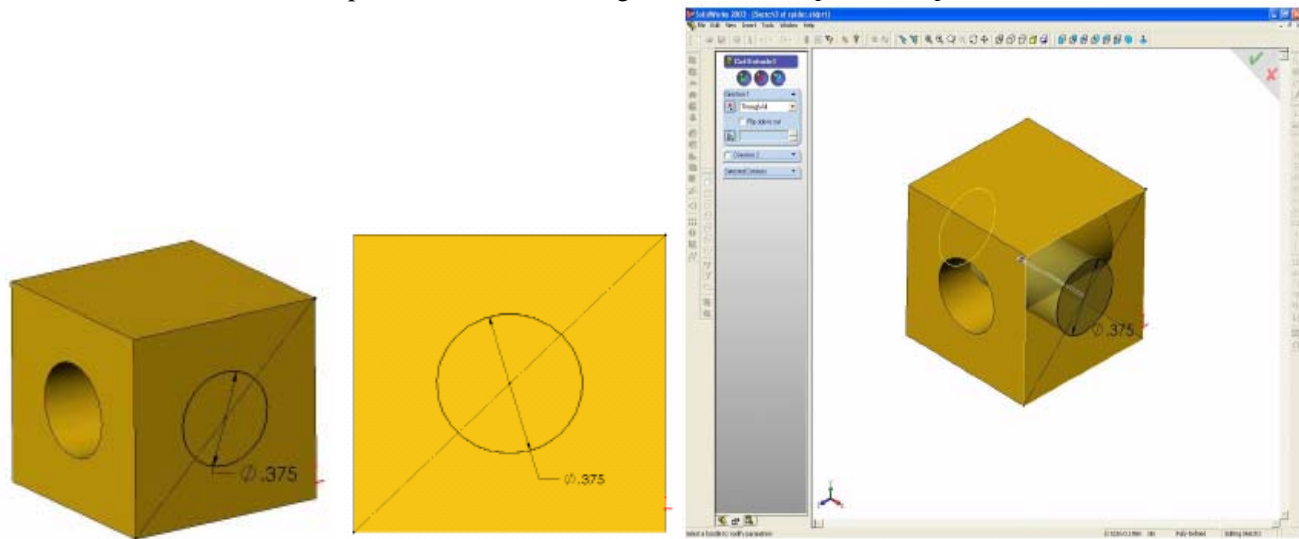
Spider



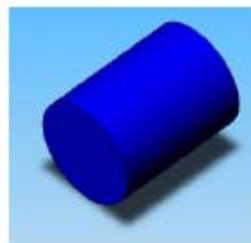
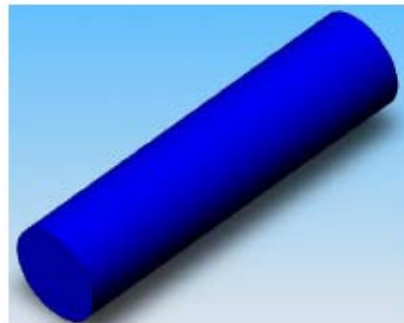
Create Sketch 1 on Front Plane. Extrude Sketch 1, distance = 0.800”.



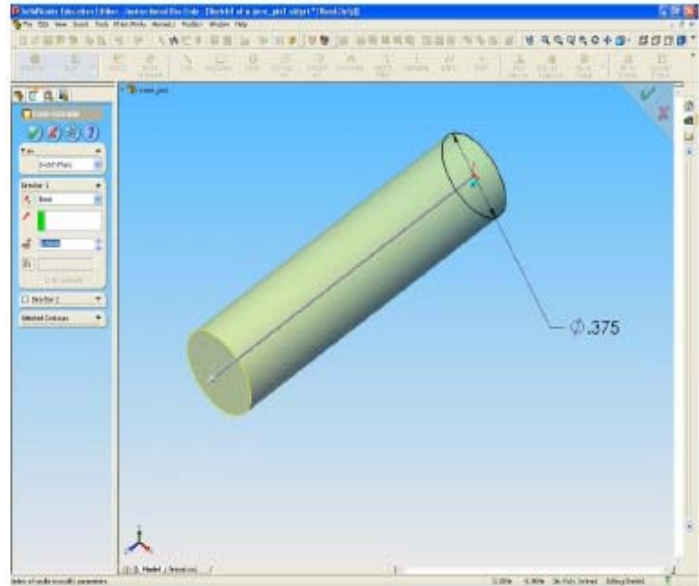
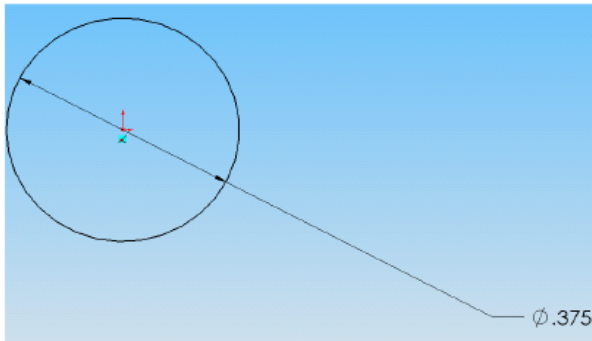
Create Sketch 2 on side face of part. Cut sketch, through all. This completes the part.



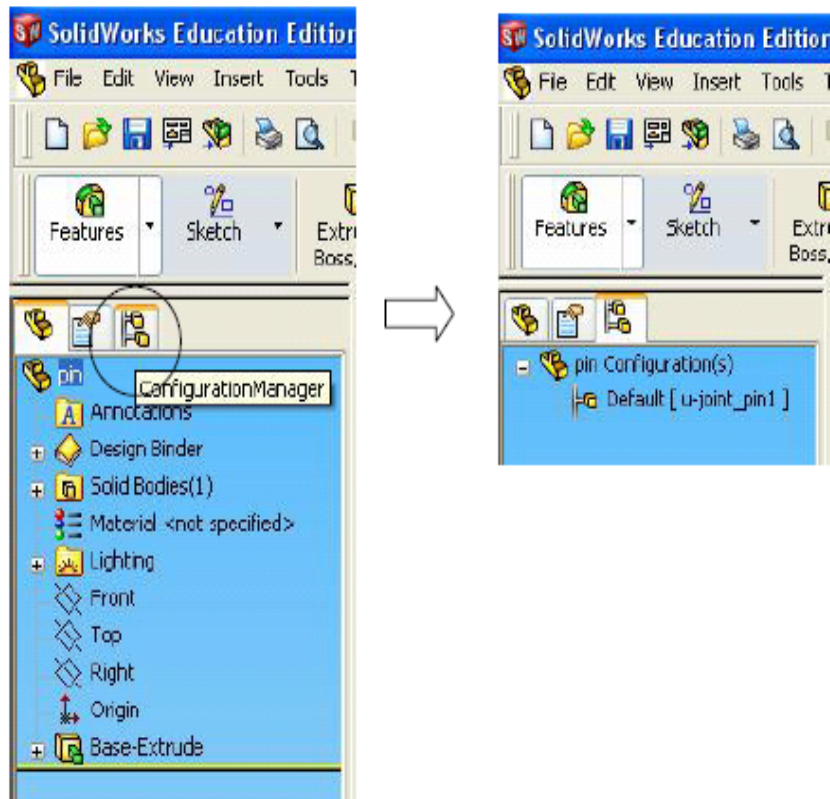
Pin



Create Sketch 1 on Front Plane. Extrude Sketch 1, Blind, distance = 1.500". This completes the part.

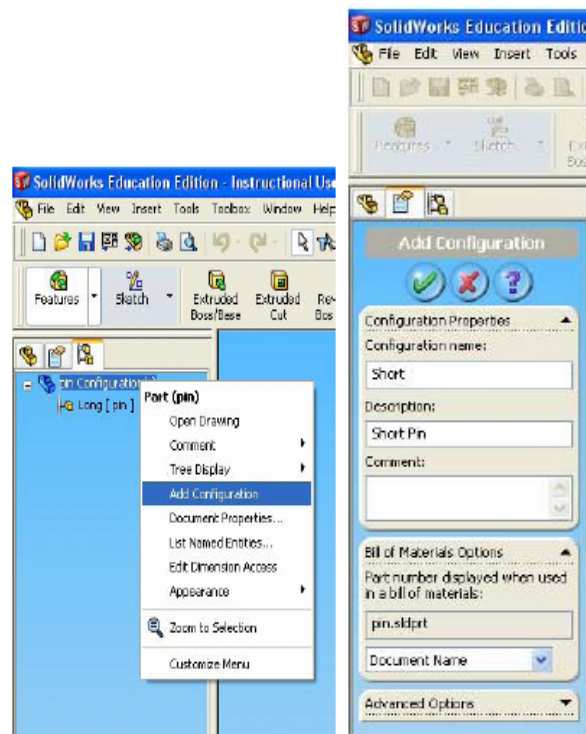


To finish Task 4, in the same file as the previous part, click on *ConfigurationManager* tab that is on top of *FeatureManager Design Tree*. Right-click on *Default* configuration, then select *Properties*. Rename configuration name and description as shown.

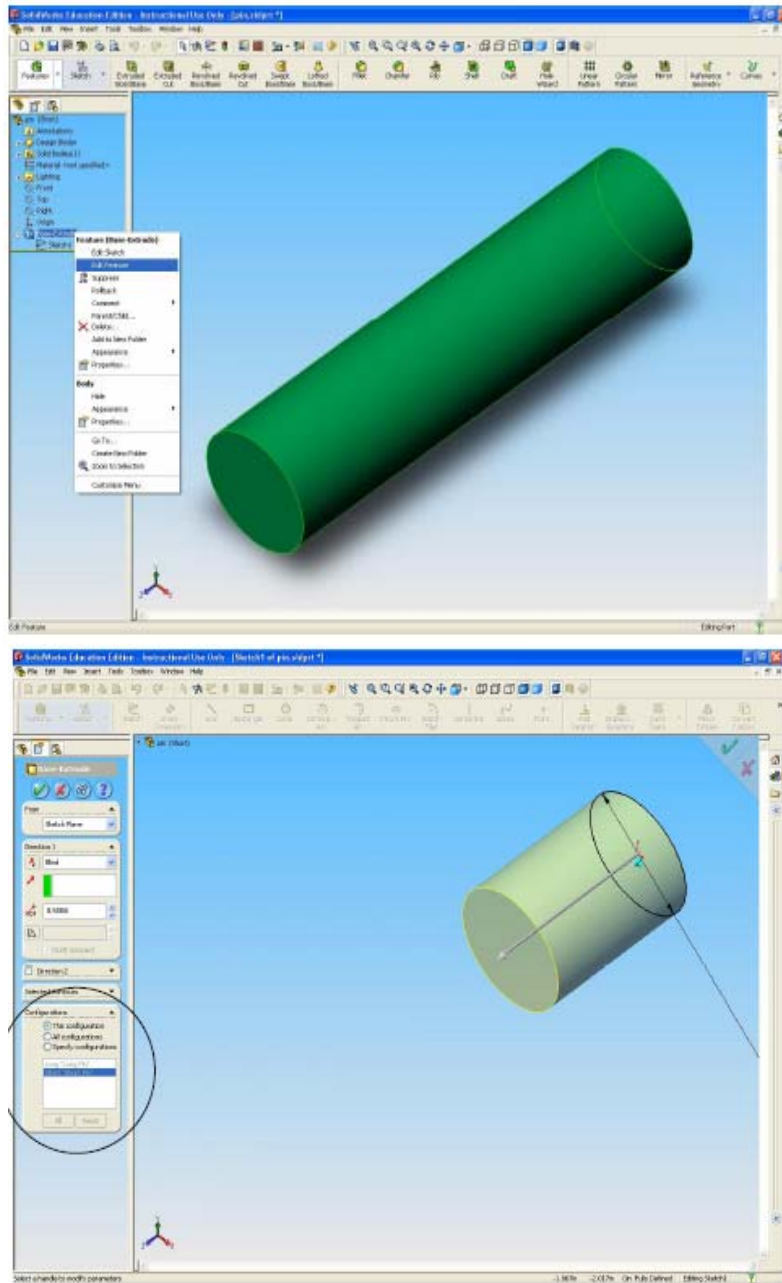




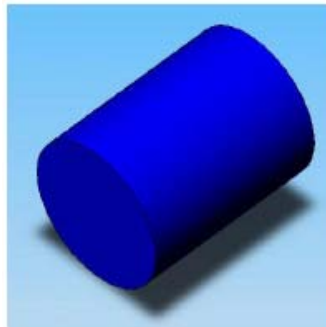
Right-click at top of *ConfigurationManager* and select *Add Configuration* and provide a name and description for the configuration file.



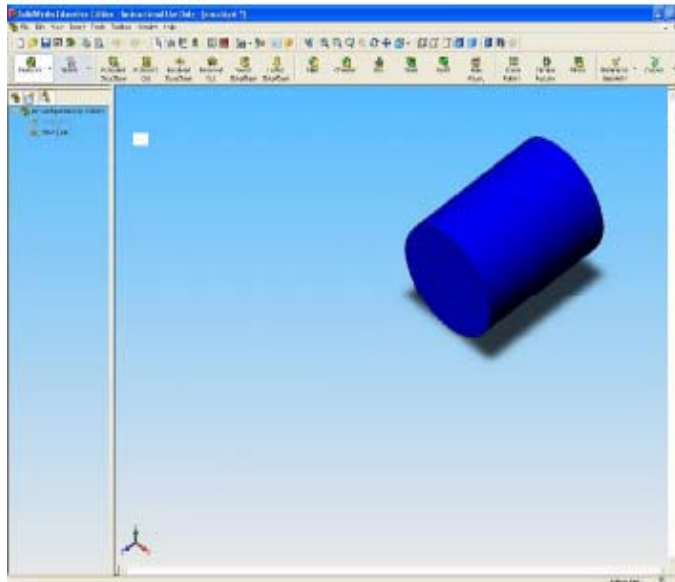
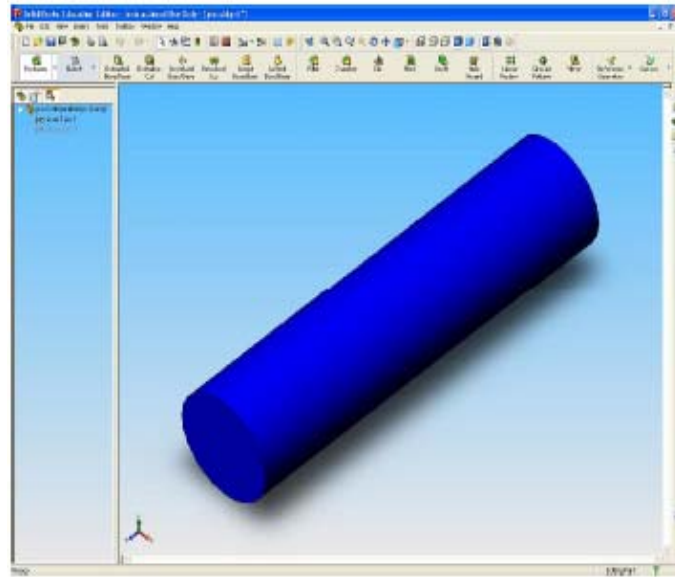
Go back to the *FeatureManager Design Tree* and right-click on the Base-Extrude Feature and select *Edit Feature*. Modify the extrude distance to distance = 0.500". In the configuration tab in feature definition, select to apply this change to "This configuration" (short).



After the feature modification, your part should look like this.



Go back to ConfigurationManager. Right click and select Show configuration for Long and Short pins to confirm proper configurations created. This completes Task 4.



Task 5: SolidWorks Assemblies.

The purpose of this task is to familiarize you with Assemblies in SolidWorks and generate the subassembly model for the U-Joint Crank and the assembly model for the U-Joint created in the previous task. From the *Help* menu, first complete Lesson 2 from the On-line Tutorial.

For the second part of this task, you must complete Lessons 14 and 15 from the SolidWorks 2005 Training Manual available on the course website.

Write-up

For your write-up, please provide the following:

1. All drawing files for every part you have generated in SolidWorks, clearly labeled with all the appropriate final dimensions.
2. A summary on the analysis of the assembly. (Maximum 1 page.)

3. Show me your working physical simulation.
4. Short answers to the following questions: (Maximum 1 page.)
 - a. What was easy to do, what was hard to do?
 - b. What was intuitive, what was not intuitive about the user interface?
 - c. What are the advantages and disadvantages that you can think of?
 - d. Suggestions for improvements: this could be in terms of the user interface, how things are organized, etc.