

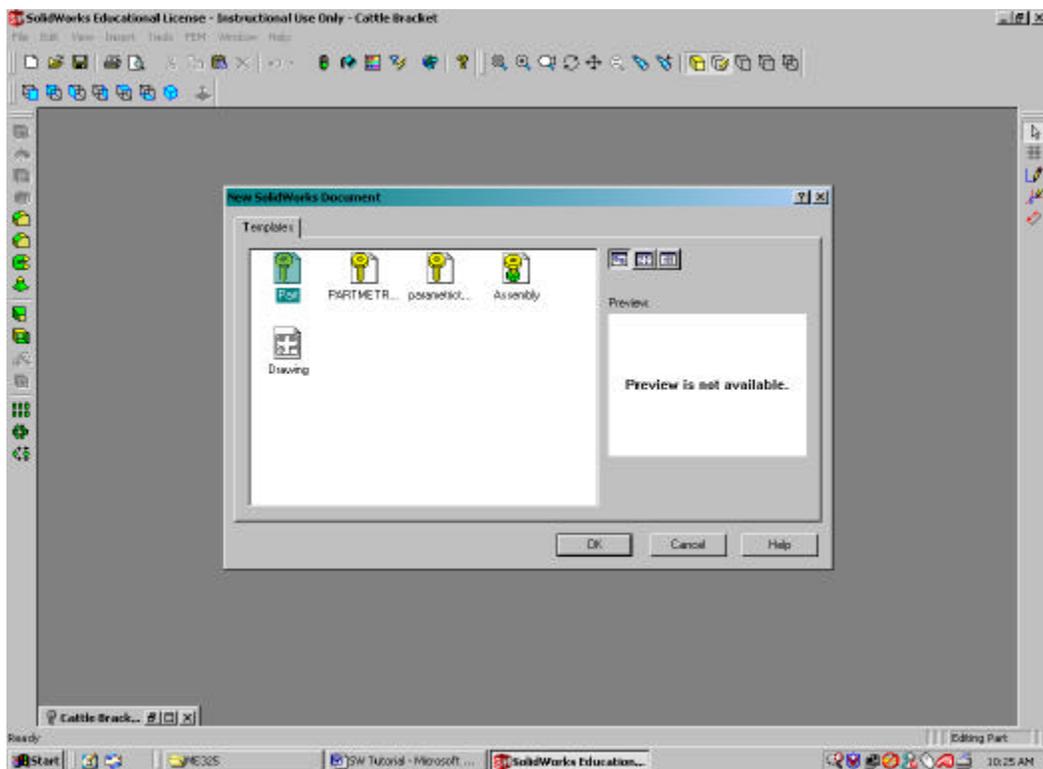
TUTORIAL 2

OBJECTIVE:

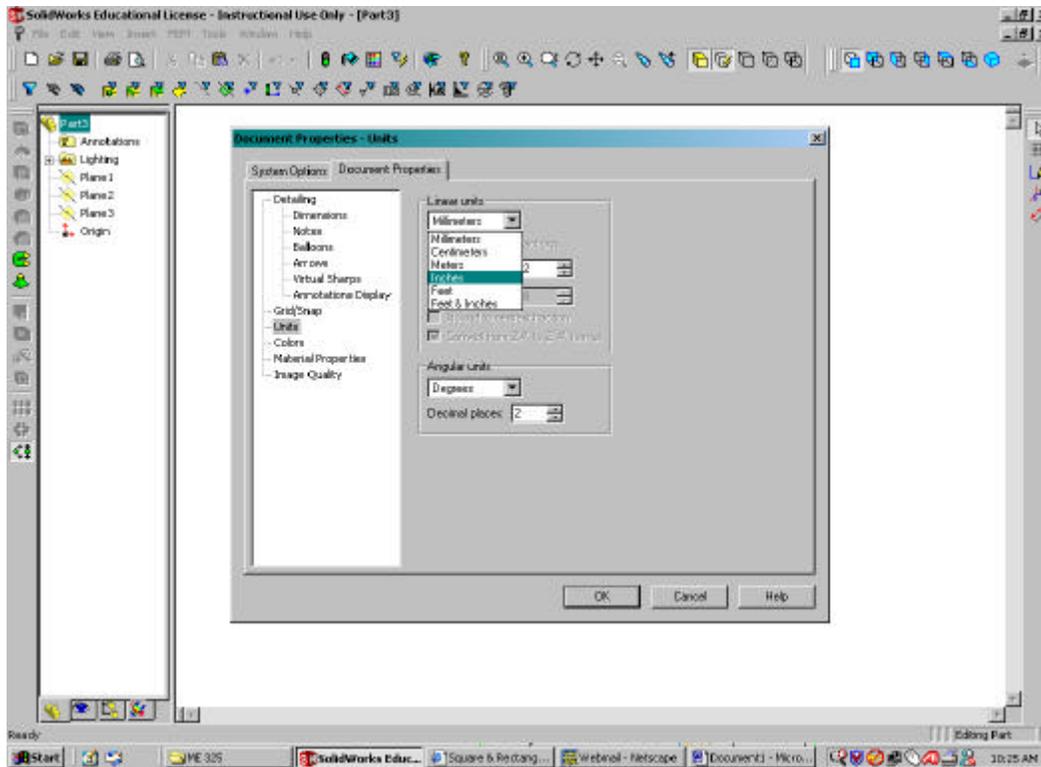
Use SolidWorks/COSMOS to model and analyze a cattle gate bracket that is subjected to a force of 100,000 lbs.

GETTING STARTED:

1. Open the SolidWorks program.
2. Open a new part file.
 - a. From the title bar select **File**
 - b. Then select **New**, which brings up the screen below.
 - c. From this screen select **Part**.



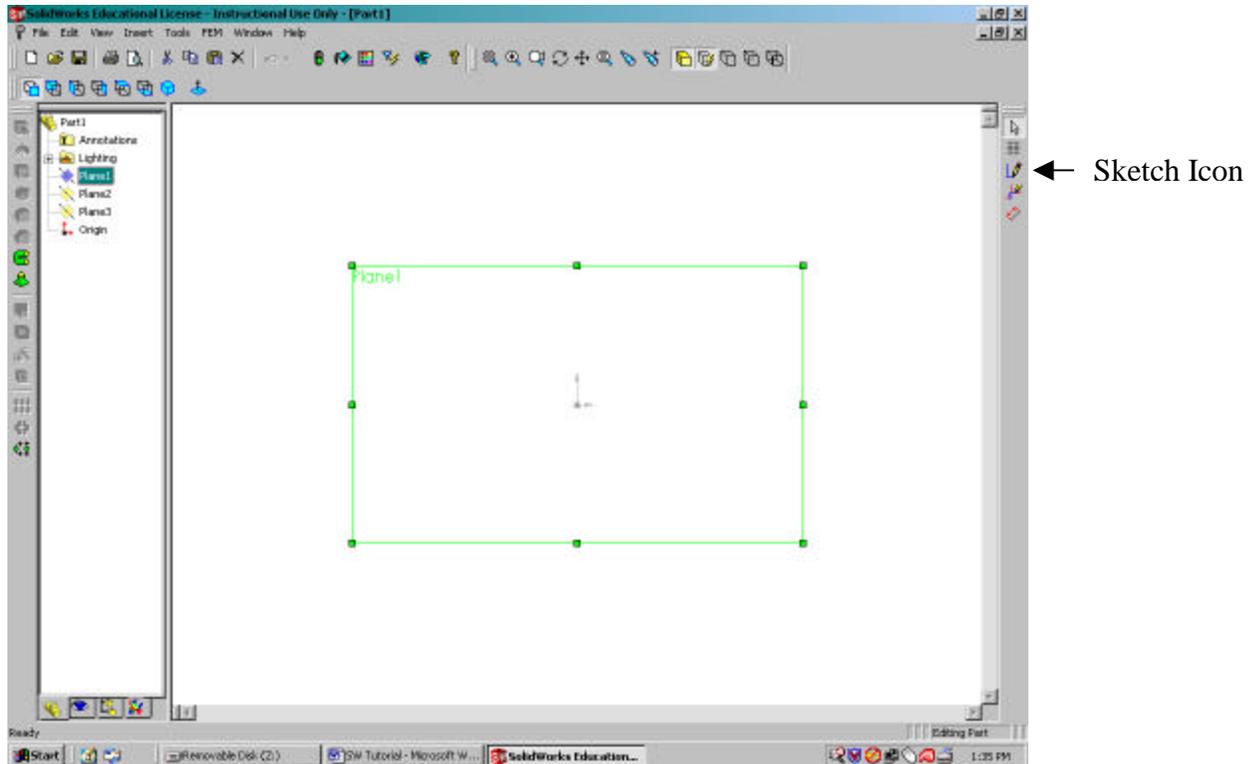
3. Change the units to English.
 - a. To do this, select **Tools** from the pull down menu bar.
 - b. Select **Options** from the bottom of this pull down menu.
 - c. In the “Systems Options” window, click on the **Documents Properties** tab.
 - d. Select the **Units** option from the tree on the left hand side.
 - e. Change the linear units to **Inches** in the pull down menu.
 - f. Click the **OK** button.



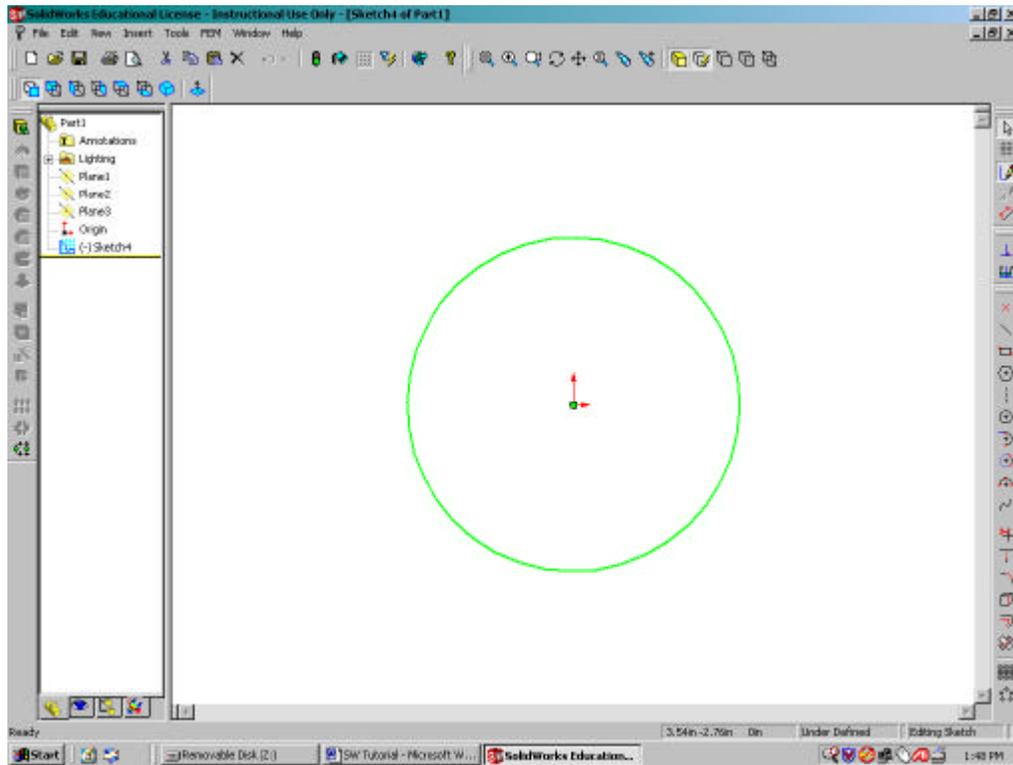
SKETCH THE BRACKET:

The bracket will be assembled from a number of parts, most of which are extruded circles. The details are outlined below.

1. Open a sketch on plane 1
 - a. Click on the **Plane 1** icon in the feature manager design tree along the left hand side of the screen. This will bring up an image of Plane 1.
 - b. Select the **Sketch** icon from the toolbar on the right hand side of the screen.

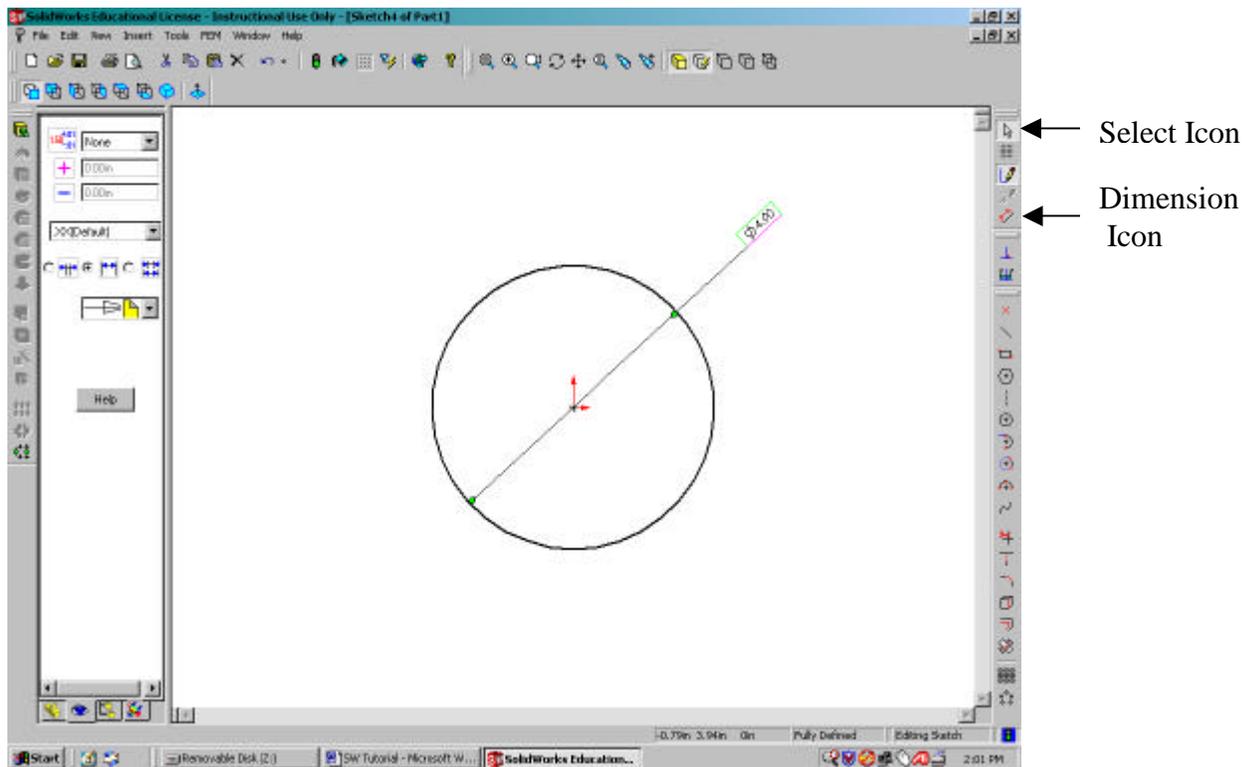


2. Sketch the base of the bracket.
 - a. Click on the **Circle** icon on Sketch toolbar.
 - b. Move the cursor to the origin in the center of the screen. Position the cursor over the origin until the pointer turns yellow. Click and drag the circle to any size.
 - c. You do not need to worry about the dimensions of the circle at this point, only that it is centered about the origin.



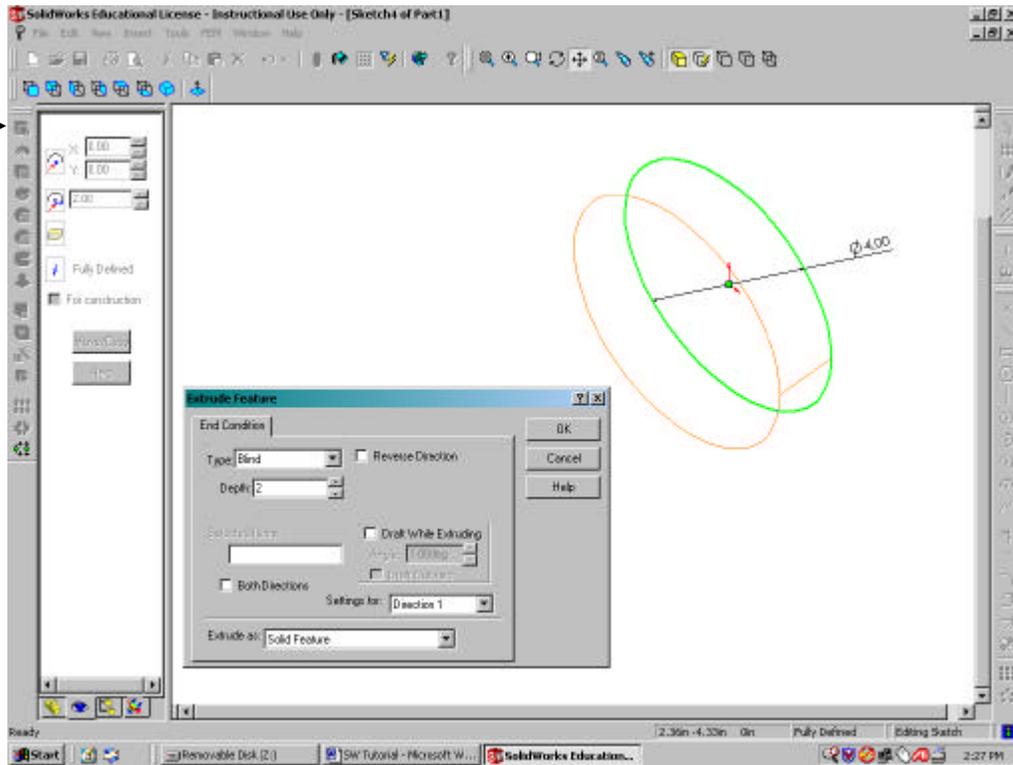
← Circle Icon

3. Dimension the circle.
 - a. Select the **Dimension** icon from the Sketch toolbar.
 - b. Click on the circumference of the circle. By moving the mouse around, you can see the different places that you can put the dimension text.
 - c. Click the mouse again to place the dimension text anywhere.
 - d. Click on the **Select** icon (pointer) on the Sketch toolbar.
 - e. Double click on the dimension text, which will bring up the **Modify** window.
 - f. Change the diameter of the circle to 4 inches, and then hit enter. You will notice that the circle has been updated.

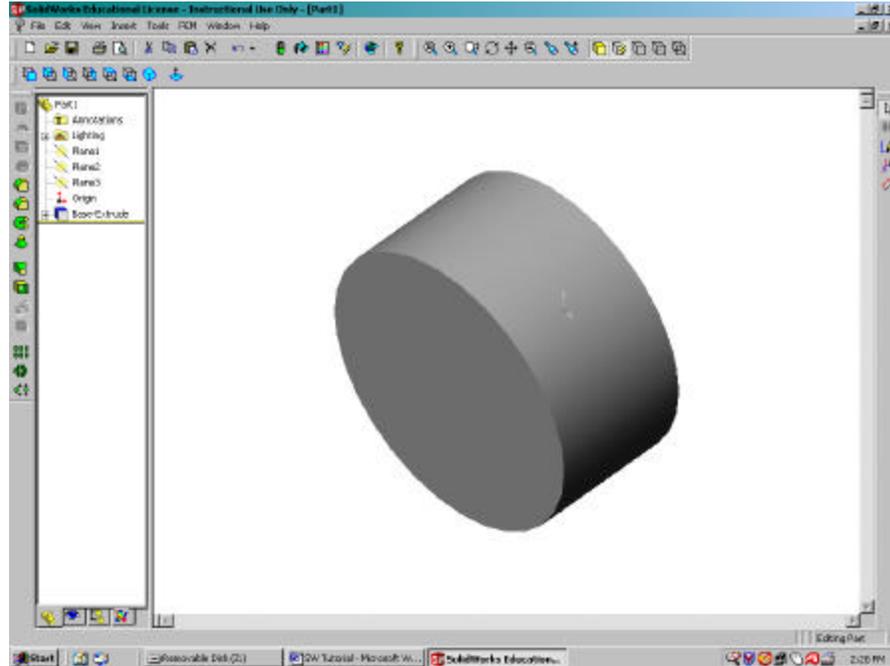


4. Extrude the circle.
 - a. With the pointer, select the circumference of the circle and then click on the **Extruded Boss/Base** icon on the left hand side of the screen in the Features toolbar.
 - b. The Extrude Feature window will open. **Change the depth to 2 inches.** You will notice in the background the preview of the extruded circle.
 - c. Click on **OK** to complete the solid cylinder extrusion.

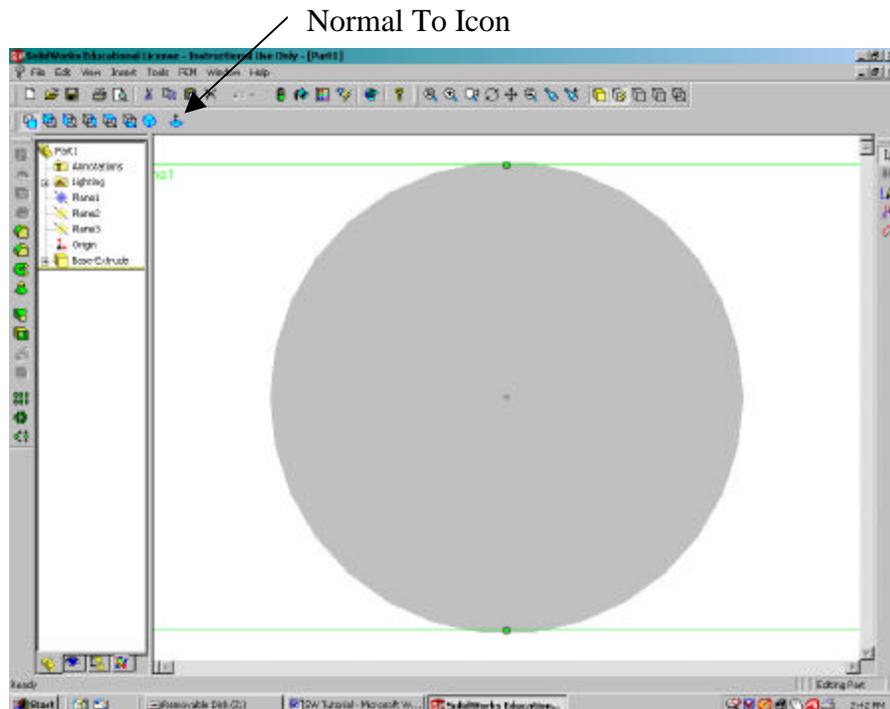
Extrude →



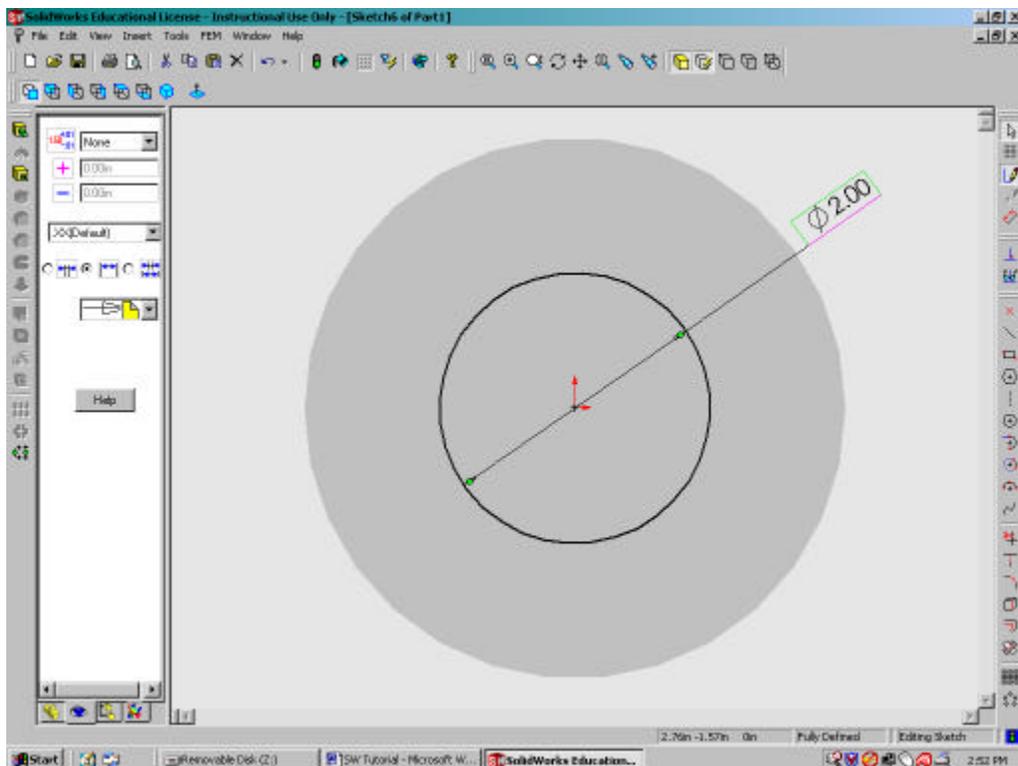
The extruded circle will look like the solid shown below.



5. Sketch the bracket extension.
 - a. Select **Plane 1** from the FeatureManager Design Tree. An image of Plane 1 will be shown on the solid cylinder.
 - b. Select the **Normal To** icon from the Standard Views toolbar. Your part should now be in the following position.



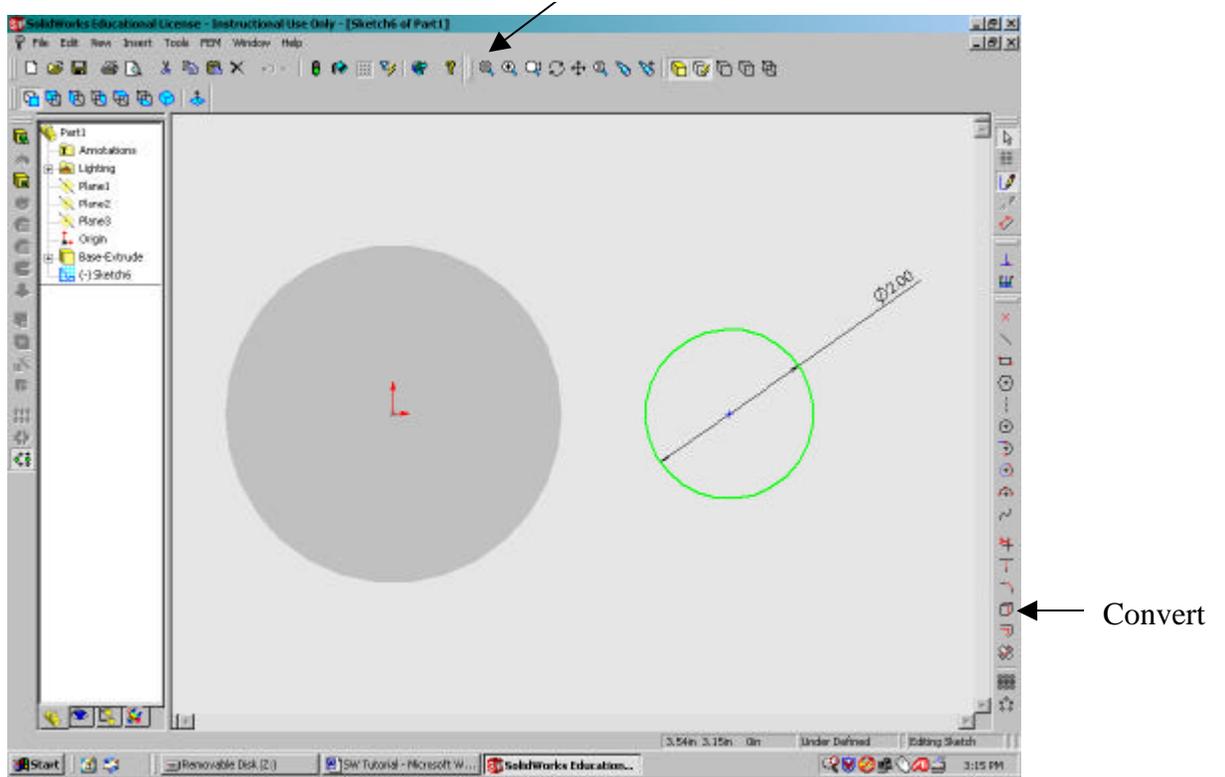
- c. Click on the **Sketch** icon in the Sketch toolbar. This will bring up the rest of the options in the Sketch toolbar.
- d. Select the **Circle** icon.
- e. Sketch another circle about the origin by clicking on the origin and dragging. Once again, do not worry about the dimensions at this point.
- f. Click on the **Dimension** icon in the Sketch toolbar and move the pointer to the circumference of the circle that you just drew.
- g. Click on the circumference of the circle and place the dimension text anywhere you please.
- h. Next, click on the **Select** icon (pointer) in the Sketch toolbar and then double click on the dimension text to change the dimensions of the circle.
- i. In the **Modify** window **change the diameter of this circle to 2 inches.**
- j. Click on enter. The following is what you should see.



- k. Next, using the pointer, select the circle that you just sketched so that it is highlighted.
- l. Choose the **Move/Copy** button from the PropertyManager frame on the left side of the screen.
- m. Choose the **No solve move** button from the PropertyManager frame and then in the **Translate** box, change the Dx to 4 inches.

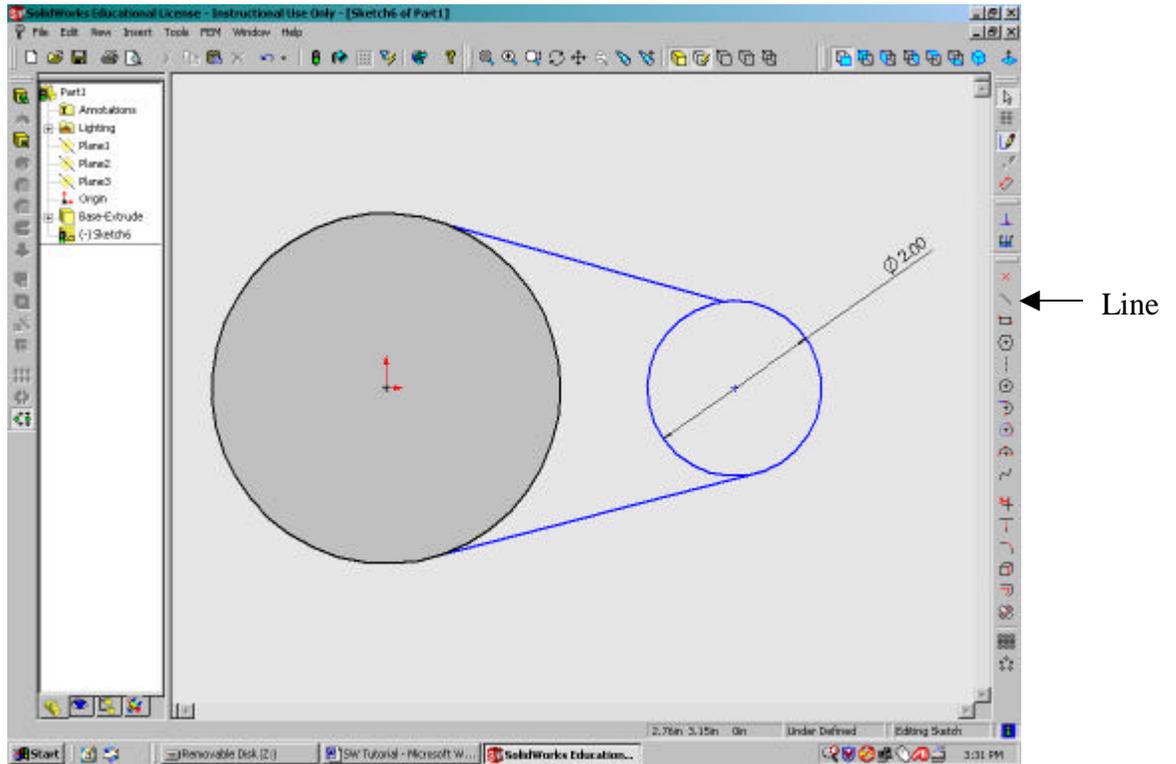
- n. To complete this move, click the **Apply** button in the same frame. Now you can zoom to fit, and should see the following screen.

Zoom To Fit



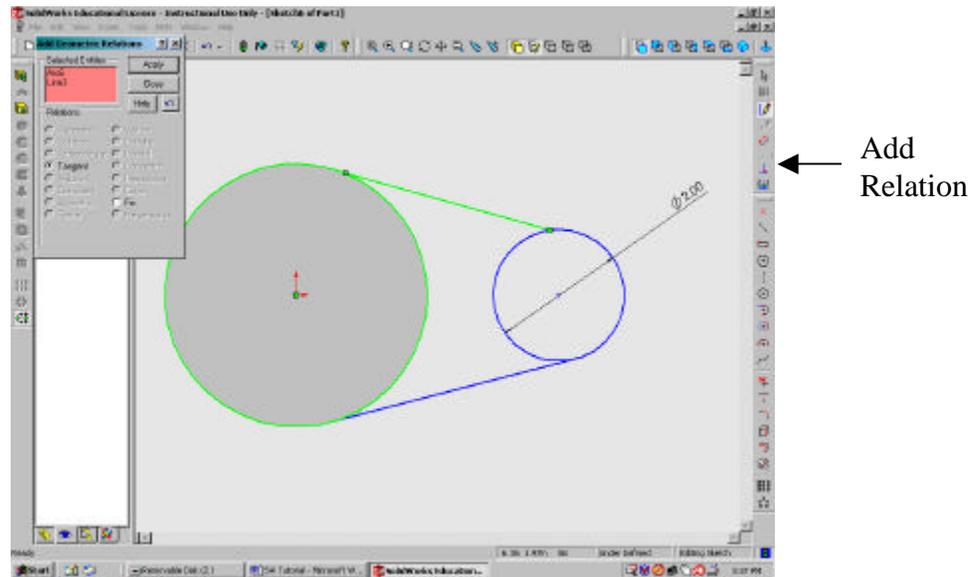
- o. Select the face of the solid cylinder and click on the **Convert Entities** icon on the Sketch Toolbar. This should convert the circumference of our cylinder into a circle on the sketch plane, which will stand out in black.

- p. Select the **line** tool and draw a line which connects the edges of both circles. When done the sketch should look something like the following.

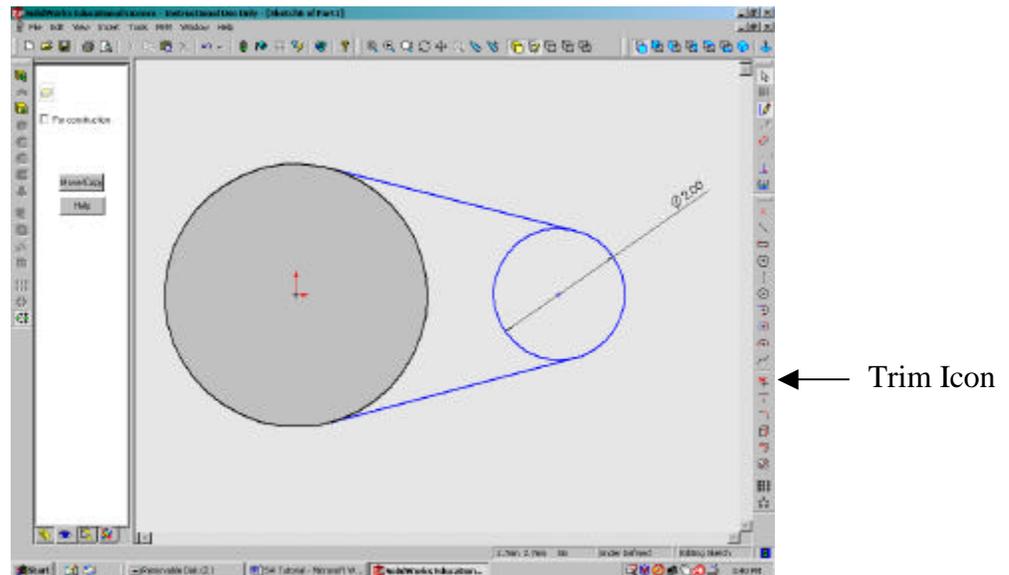


- q. Make the lines tangent to the two circles. Using the pointer **select one of the lines and one of the circles**. (Hold the control key on the keyboard down to select them both at once)

- r. Then click on the **Add Relation** icon. This should bring up the following screen.

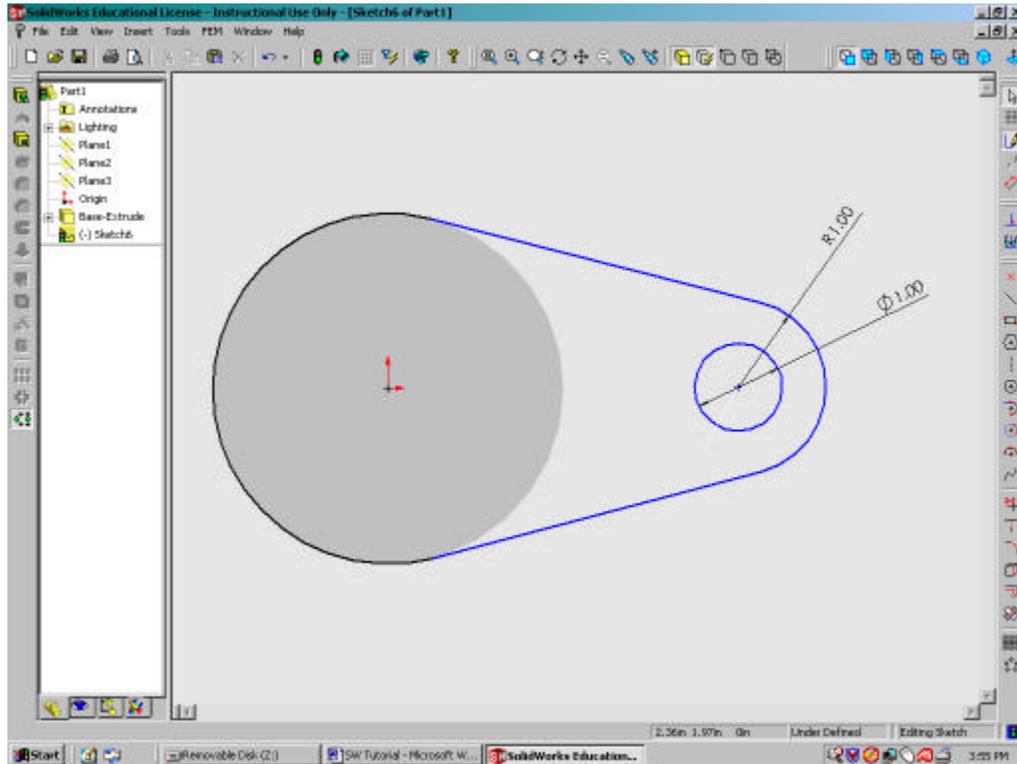


- s. As you can see, the **Add Geometric Relations** window comes up and probably has the tangent option already selected.
- t. Click **Apply**.
- u. Repeat these steps for the other three instances where the lines are supposed to be tangent to the circles. The sketch should now look like the following.

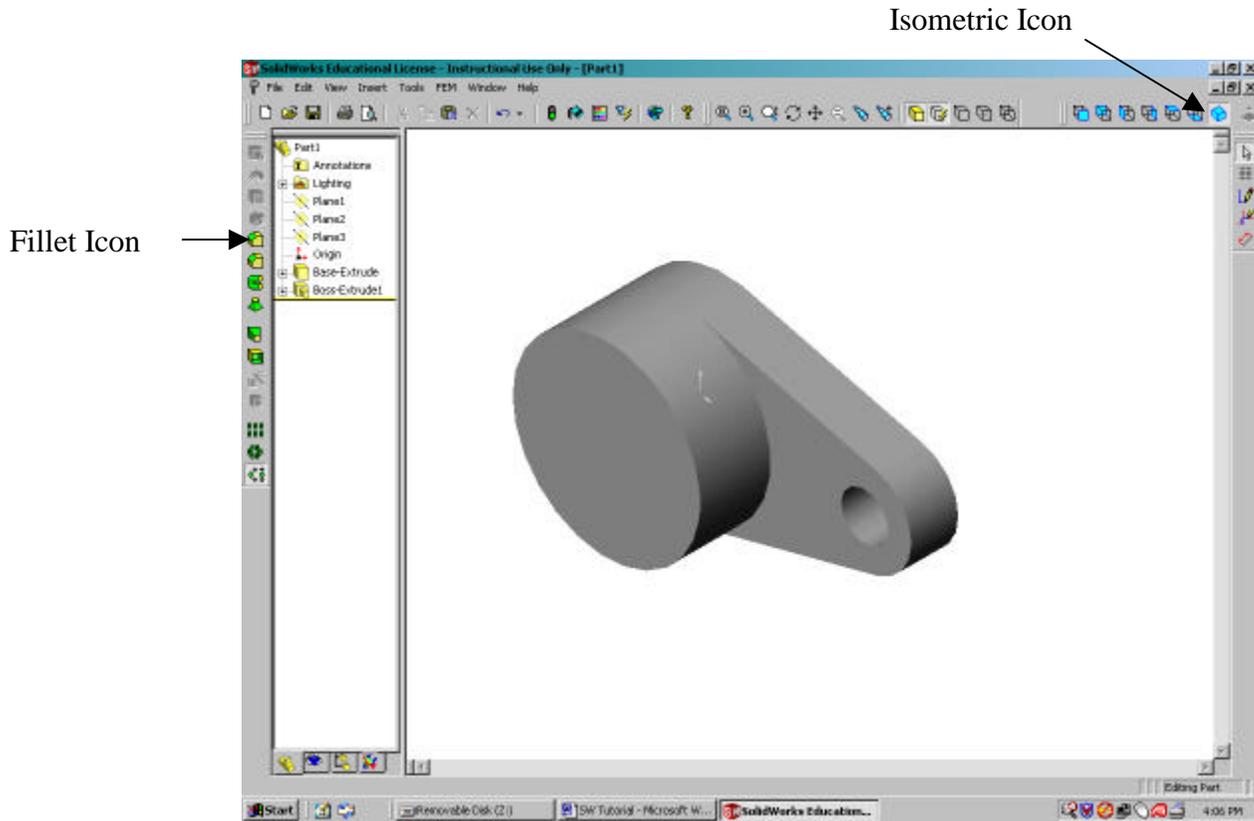


- v. Trim the interior sections of the circles. Select the **Trim** tool from the **Sketch Toolbar**. Simply select the individual arcs that you want to remove.

6. Sketch the hole in the bracket.
 - a. Select the **Circle** icon from the Sketch Toolbar. This circle and the last circle should have the same centers.
 - b. **Dimension** the circle so that **the diameter is 1in**. Now the part should look like the following.



7. Extrude this feature.
 - a. Select the **Isometric** icon from the views toolbar.
 - b. Select the **Extruded Boss/Base** icon.
 - c. **Select the “Both Directions”** option.
 - d. Change the depth to 0.5in.
 - e. Using the “Settings for:” pull down menu, select Direction 2.
 - f. Change the depth for this direction to 0.5in.
 - g. Click **OK** and you should see the following.

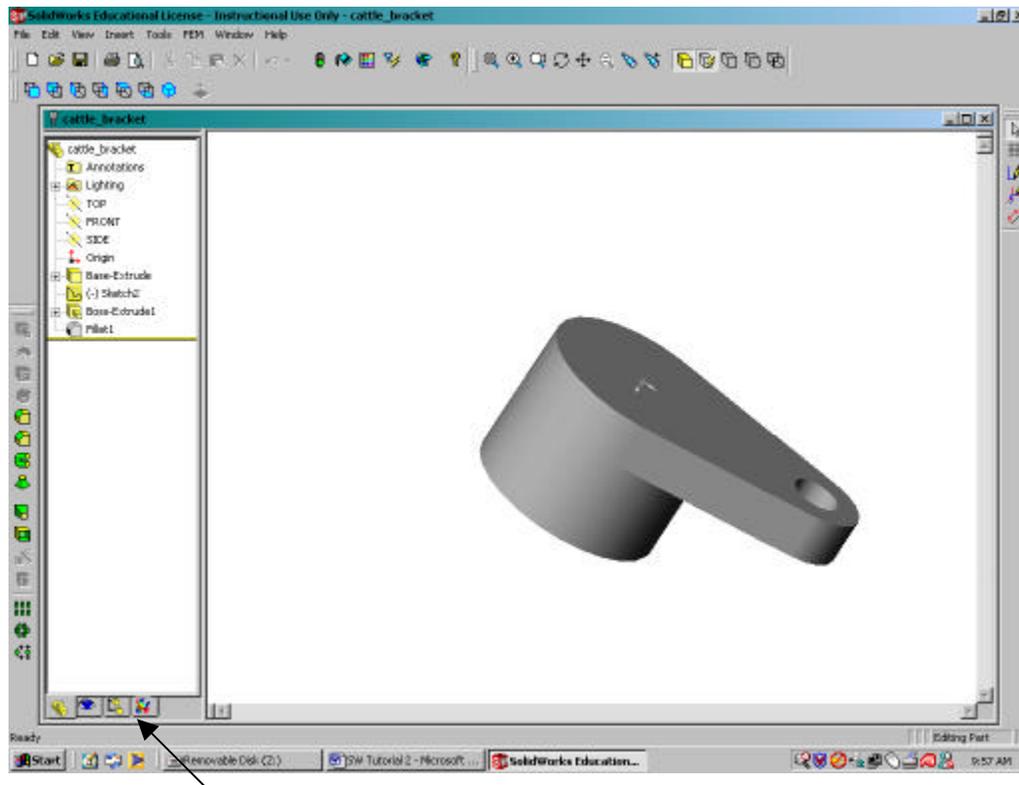


8. Fillet the edge where the cylinder and the bracket extension intercept.
 - a. Position the pointer over this intersection. When the arrow is accompanied by a vertical line, the program is selecting the line of intersection (the pointer is usually accompanied by a “flag” shape which indicates selecting a surface).
 - b. Click on the **Fillet** icon.
 - c. Change the radius to 0.25in.
 - d. Click **OK**.

COSMOS ANALYSIS:

Determine the stresses at the base of the cylinder when there is a load applied to the hole.

- 1) Click on the “COSMOS” tab on the bottom left corner of the screen, it looks like a rainbow colored wrench.



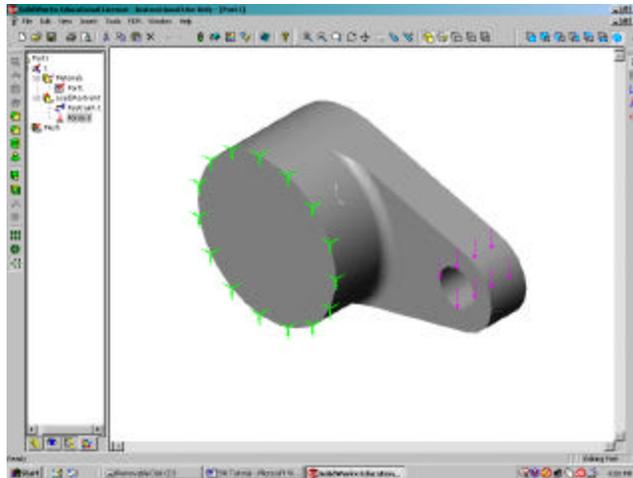
Cosmos Tab

- 2) Create a study.
 - a. Right click on your part name in the COSMOS design tree.
 - b. Left click on **Study**.
 - c. Click **Add**.
 - d. Type in a name for the analysis and click **OK**, followed by another **OK**.
- 3) Select the materials for your part.
 - a. Right click on **Materials**.
 - b. Left click on **Apply to all ...**,
 - c. Select a material from the list and click **OK**.
- 4) Creating a mesh of the part.
 - a. Right click on **Mesh**.
 - b. Left click on **Create...**, you can change the size of the mesh from this window, although the default size is usually sufficient.
 - c. Click **OK**.

- 5) Displaying the mesh.
 - a. Right click on **Mesh**.
 - b. Left click on **Show...**, this step can be skipped but you will not be able to see how the program meshed your part, so it is probably a good idea.
 - c. After seeing the mesh you should hide it before continuing.

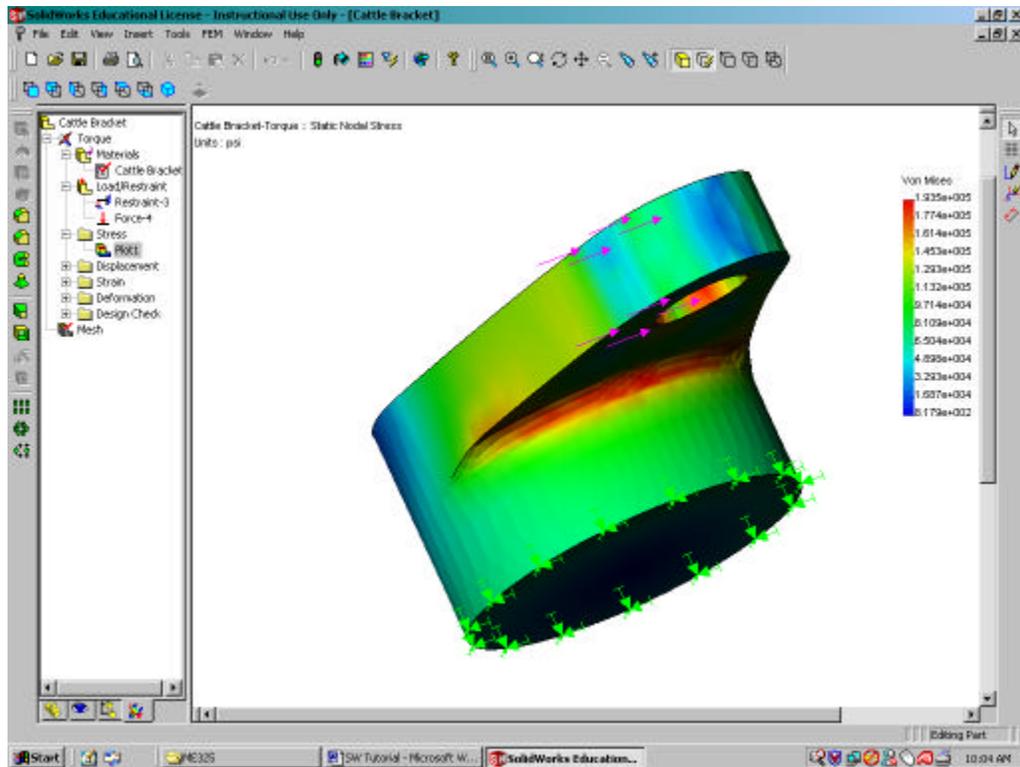
- 6) Fix the bottom face of the bracket.
 - a. Select the bottom face of the cylinder.
 - b. **Right click on Load/Restraint**.
 - c. Select **Restraint...**, make sure that the Fixed button is selected and click **OK**.

- 7) Apply a load to the hole.
 - a. Select the inside surface of the hole.
 - b. Right click **Load/Restraint**.
 - c. Select **Force...** ,Using the **Units** pull-down menu, change the units to English.
 - d. Check the box next to “along Plane Dir 2”, and enter a value of $-100,000$.
 - e. Click **OK**. Your screen should now look similar to the picture below.



- 8) Run the analysis
 - a. Right click on the name of your study and left click **Run**.

- 9) Open the plot for stress. If you rotate the object so that it is similar to the one below, you will see that the maximum stress is concentrated about the hole and the fillet.



- 10) Change units of stress to psi.
- Right click on the **stress plot** icon.
 - Click on **edit definition**.
 - Click on the **properties** tab.
 - Change the units as necessary.

- 11) Use the probe feature to find stresses along the base of the circle. To be able to use the probe feature you need to change the plot style so that it does not show deformation.
- Right click on the **stress plot** icon.
 - Click on **edit definition**.
 - Click on the **settings** tab.
 - De-select the option for “plot results on deformed model”, and click **OK**.
 - Right click on the **stress plot** icon.
 - Select **probe**. You can now select any point on the surface of the model and the stress value will show up in the probe box.

