

## Tutorial – Shaft Design

### *Situation*

The company you are interning for is manufacturing a shaft for use with a motor and gear. The customer has warned that the previously used shaft was ruined because the gear jammed and the torque from the motor snapped the shaft.

### *Objective*

The design team has created specifications for the shaft they feel is the best option. You are to take these specifications and analyze the shaft using SolidWorks and Cosmos so that the team can see possible problems.

## **Part 1 – Modeling Shaft**

### *Opening SolidWorks*

1. Start by opening SolidWorks 2000 using the **Start** Menu at the bottom left-hand corner of the screen.
2. Open a new drawing by selecting the **New Drawing Icon** on the left side of the toolbar.

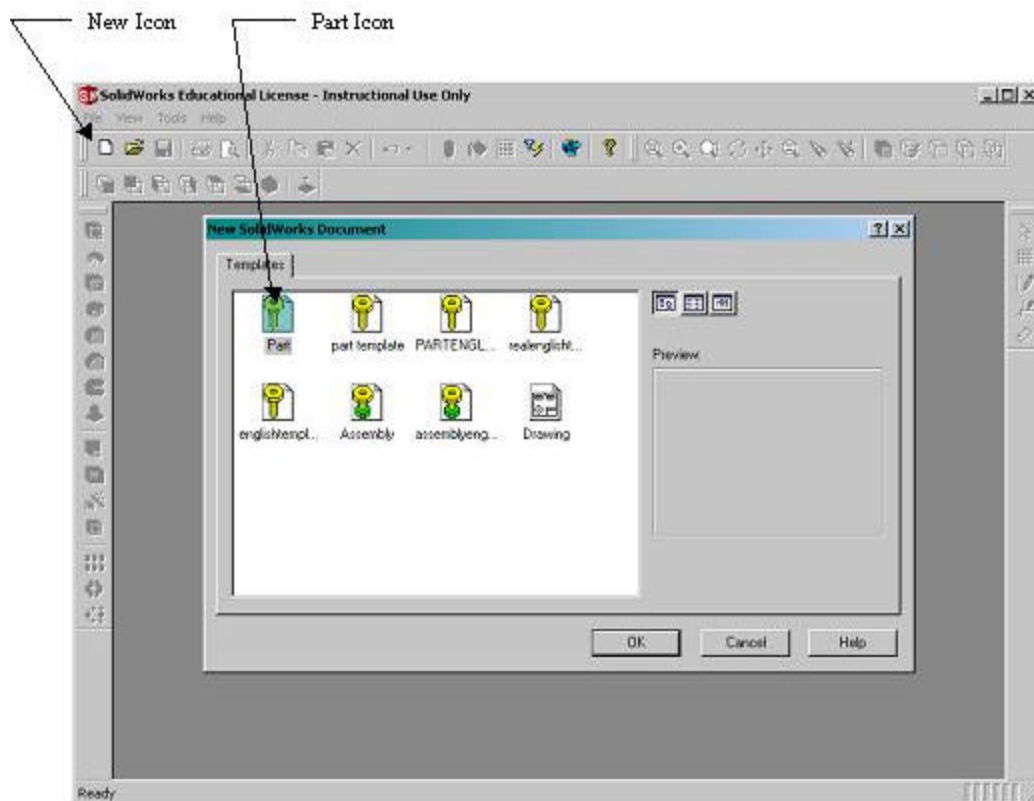


Figure 1.1

3. The **New SolidWorks Document** window will appear. Select the **Part** Icon and click **OK**.
4. To make sure the units are in metric, go to the **Tool/Options...** menu. Click on the **Document Properties** tab. Select **Units** from the tree on the left side of the window. Change the **Linear Units** to millimeters, which should be the default setting. Click **OK**.

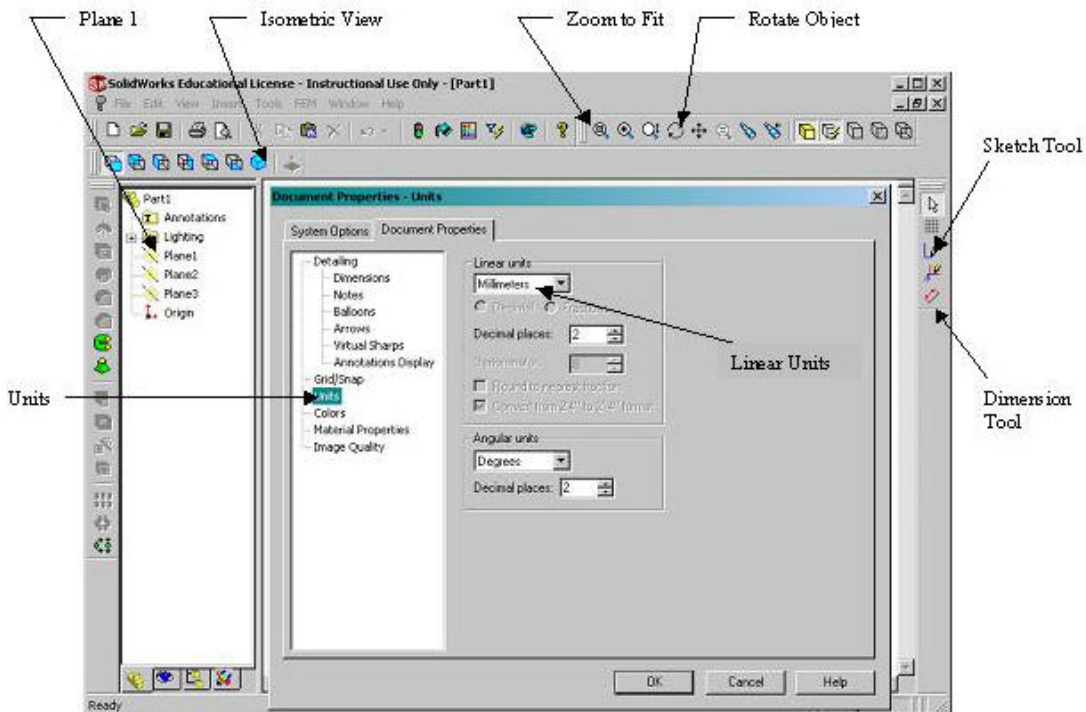


Figure 1.2

### *Drawing the Shaft*

5. Select the **Plane 1** line on the left side of the screen.
6. Click on the **Sketch** Icon on the right side of the screen.
7. Select the **Circle** Tool on the right side of the screen.

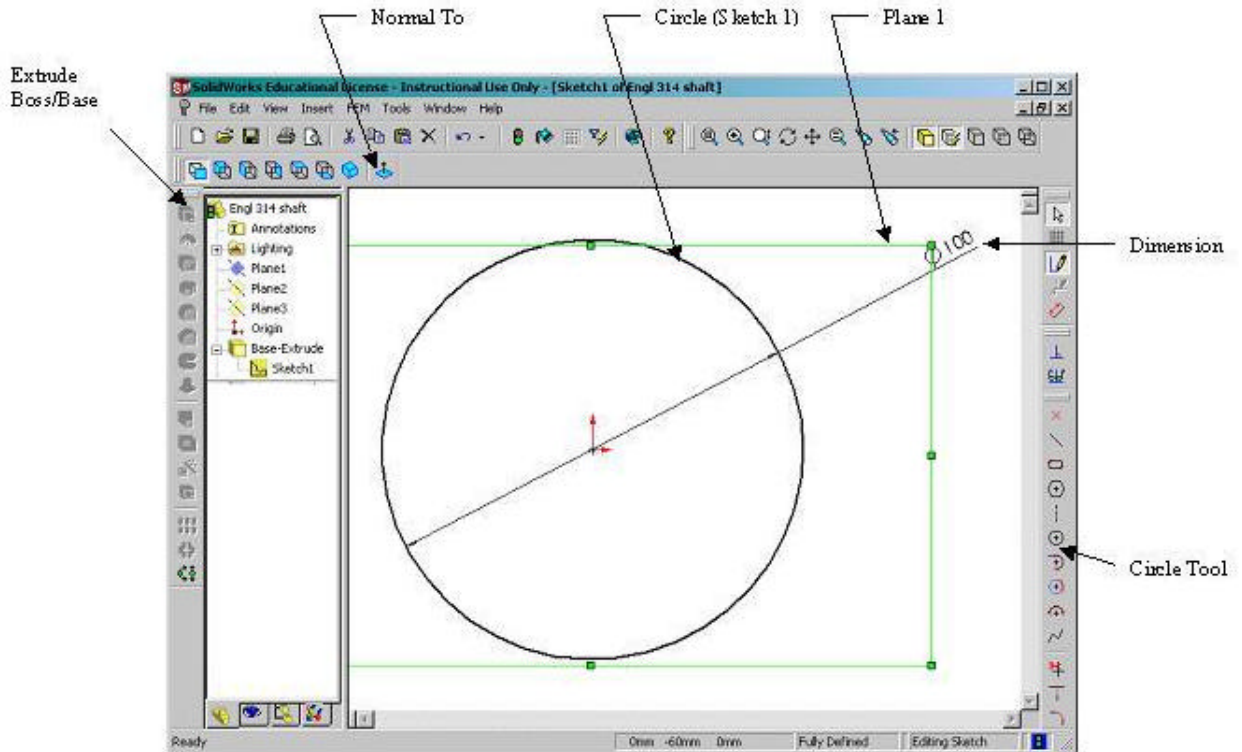


Figure 1.3

**Hint:** For different uses, the mouse becomes different objects related to the use. When used as a sketch tool, the mouse will become an arrow accompanied by the object that is to be sketched (ex: when sketching a circle the icon is accompanied by a circle). Perhaps the most helpful part of this is locating a part of the model. When using the select icon a different picture will appear for a line, face, point and vertex. Take the time to learn them and it will help you in the future.

8. Put the cursor over the origin, where all three axis and planes intersect. The pencil on the screen will turn yellow when you are at the origin.
9. Click and hold the mouse button to build the circle. Make the circle of an arbitrary diameter by dragging the mouse. The diameter will be changed in the next step. Let go of the mouse button.
10. Click on the **Dimension** Tool (Refer to Figure 1.2) on the right side of the screen.
11. The circle should now appear blue. Click on the circle to set the diameter. After you click on the circle, click anywhere on the screen to place the dimension. Once the circle is fully constrained, it will turn black.
12. Switch to the **Select** Tool by clicking on the arrow on the upper right-hand side of the screen.

13. Double-click on the dimension that was just placed. Set this dimension to 100mm.
14. Click on the circle and then the **Extruded Boss/Base** Button on the left-hand side of the screen.
15. Pick **Blind** from the **Type** pull-down menu. Set the **Depth** to 300 mm. Click on the **Reverse Direction** checkmark. Click **OK**.

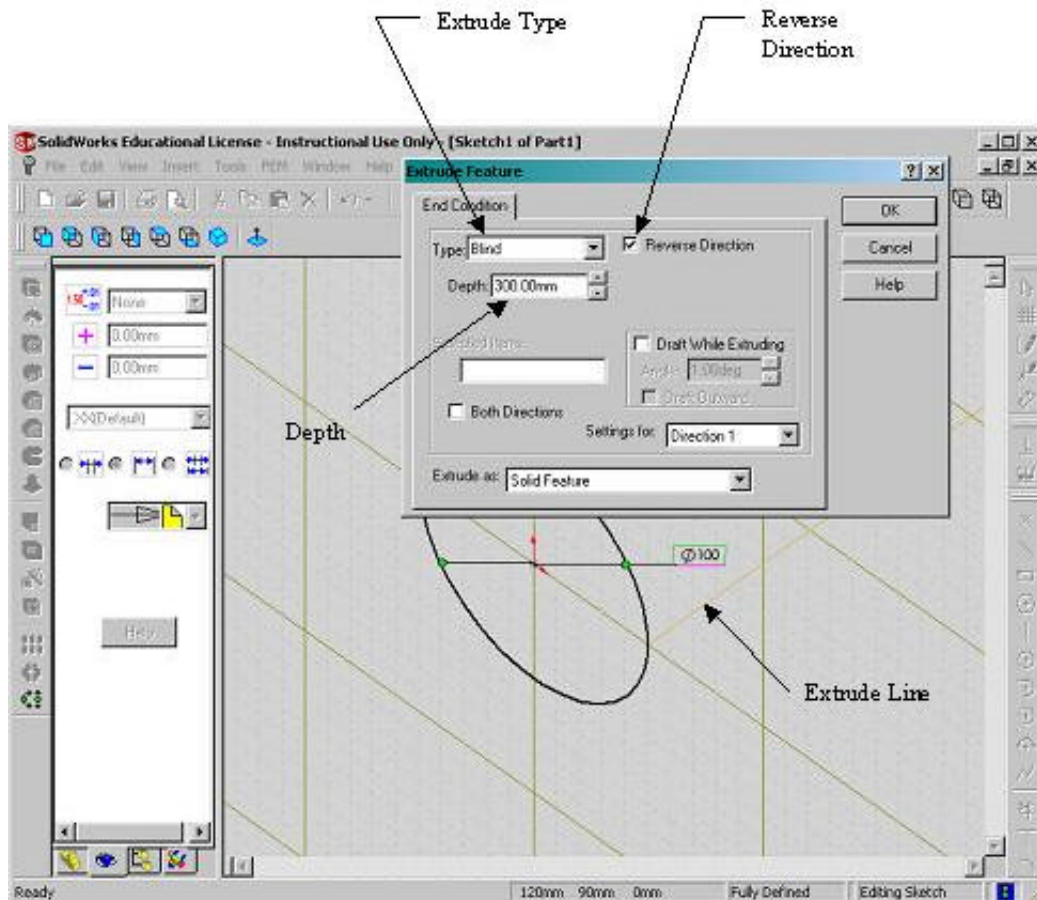


Figure 1.4

16. Click the **Isometric** Button and the **Zoom to Fit** Button (Refer to Figure 1.2) to see the extruded shaft.

**Hint:** Simply hit the “F” key to quickly zoom to fit. Also, if the mouse has a middle button that rolls, it can be used as a quick zoom tool by placing the icon somewhere in the model window, clicking the middle button, and rolling it one way or the another.

17. Click on the end surface of the tube closest to the viewer. To ensure you are selecting a surface, a white flag will appear. The surface should turn green when selected. Then click the **Normal To** Button (Refer to Figure 1.3) followed by the **Sketch** Button.
18. Create another circle by repeating *Steps 5-13*. Set the diameter of this circle to 75 mm.
19. Click on the circle and then the **Extruded Boss/Base** Button. Set this extrusion to be a blind depth of 200 mm. For this extrusion, ***do not reverse the direction***.
20. Click the **Isometric** Button and the **Zoom to Fit** Button to make sure the shaft extruded in the proper direction.

**Hint:** Mistakes in SolidWorks are much easier to correct than mistakes in other modeling programs. Place the mouse icon over the bar located in the parts menu on the left side of the screen. The mouse icon will turn into a hand. Drag the bar back to the step where the mistake was made. Everything that was done after this step will disappear from the model screen, but don't worry. Correct your mistake and then drag the bar back to the last step. SolidWorks automatically corrects all in-between steps.

21. Pick the end surface of the smaller shaft that is visible to the user. Then click on the **Normal To** Button followed by the **Sketch** Button.
22. Create another circle by repeating *Steps 5-13*. Set the diameter of this circle to 150 mm.
23. Click on the newest circle and then click the **Extruded Boss/Base** Button. For this part, ***reverse the direction*** and use a blind extrusion with a depth of 100 mm.
24. Click the **Isometric** Button and the **Zoom to Fit** Button to make sure the shaft extruded in the proper direction.
25. Click on the end surface of the large section just extruded. Click on **Normal To** Button and then the **Sketch** Button
26. Create another circle by repeating *Steps 5-13*. Set the diameter of the circle to 50 mm.

27. Click on this circle and then the **Extruded Cut** button. For this feature, ***do not reverse the direction***. Change the **Type** to **Through All**.

**Hint:** The Through All option is used when a cut is desired to go through all pieces perpendicular to the sketch.

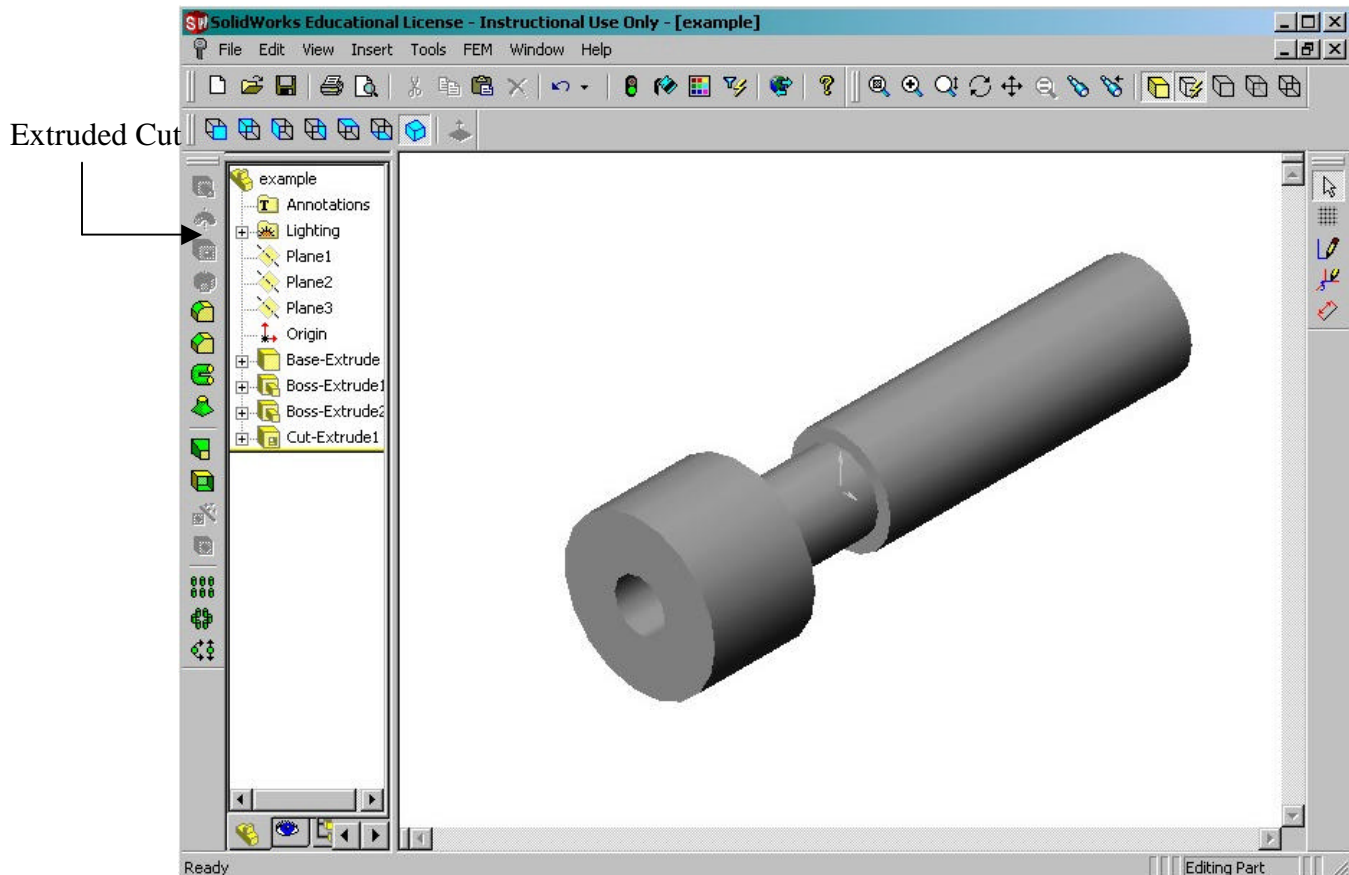


Figure 1.5

**Hint:** If you like the use of split screens in modeling programs like AutoCAD, you can do the same thing in SolidWorks. In the model window, below the close (X) button and above the scroll up arrow there is a bar. There is another one at the bottom of the model window by the left scroll arrow. These are the split screen boxes; by clicking them you can create two or four views.

28. Use either the **Middle Mouse Button** or the **Rotate View** Tool on the top toolbar to rotate the view manually. This can be used to make sure the hole went all the way through. After you have made sure the hole went all the way through, click the **Isometric** Button.

## Part 2 – FEM

1. Go to the **FEM/Study...** menu.
2. Click **Add...** Name the study whatever you would like as long as two studies aren't labeled the same. For this example we will use the name "**DEMO**". Pick **Static Analysis Type**. Pick **Solid Mesh Type**. Click **OK**. Click **OK** again.
3. Go to **FEM/Material...** For this tutorial, we will be using **Stainless Steel (ferritic)**. Pick this material under the **Material Name Menu** and click **OK**.

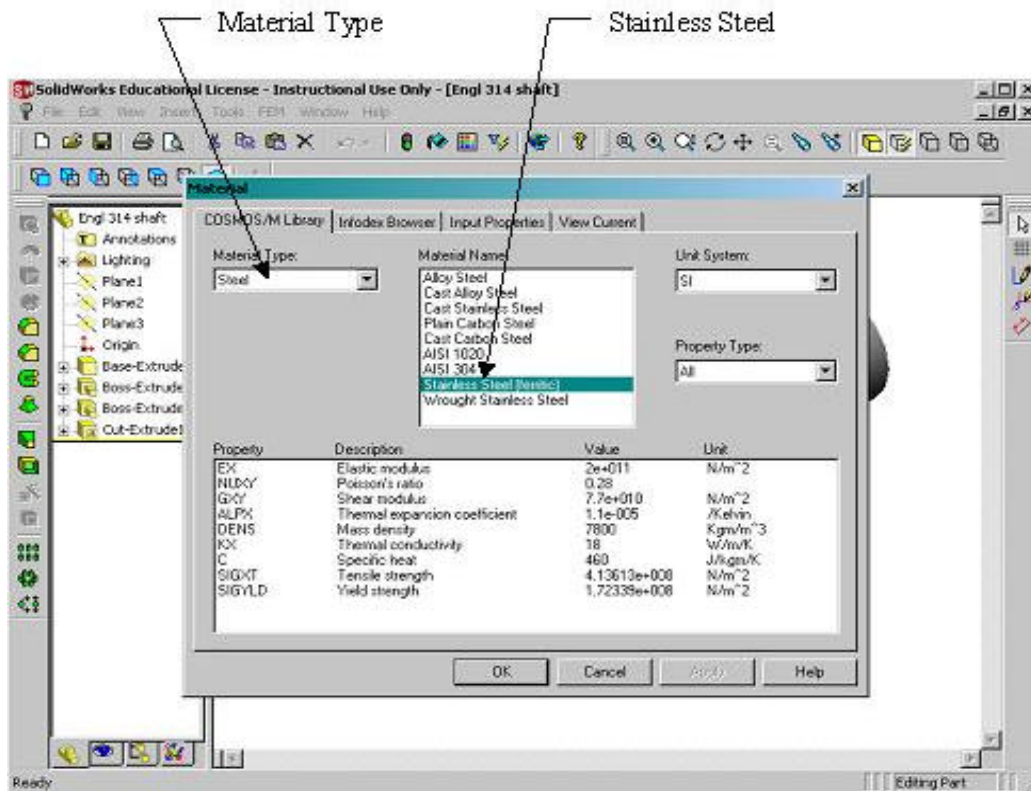


Figure 2.1

4. Go to **FEM/Mesh/Create...** Use default settings for this tutorial. If a higher degree of accuracy is necessary for the analysis, the slider can be moved to a finer setting. If a lesser degree of accuracy is needed, the slider can be moved to a coarser setting. The coarser the setting, the faster the analysis will be completed.
5. Go to **Insert/Reference Geometry/Axis...** This will be used to find the centerline of the shaft.

6. Pick **Cylindrical/Conical Surface** and select the outside of the largest part of the shaft. A centerline should appear down the middle of this portion of the shaft. Click **OK**.
7. Click on the new axis, **Axis1**, using the tree on the left side of the screen (refer to Figure 2.2). Then, while holding down the control key, select the outer surface of the largest shaft (the same one selected in the previous few steps).

**Hint:** Just as in Windows, pressing and holding the Ctrl key will allow you to choose multiple objects at one time.

8. Go to **FEM/Insert/Force...**. A menu will appear for adding forces and torques to the model.
9. Make sure the **Type** is on **Apply Torque**. Use the default **Uniform Distribution**. The **Torque** value should be **15 N·m**. Click **OK**. Purple arrows should now show the force being applied to the shaft.

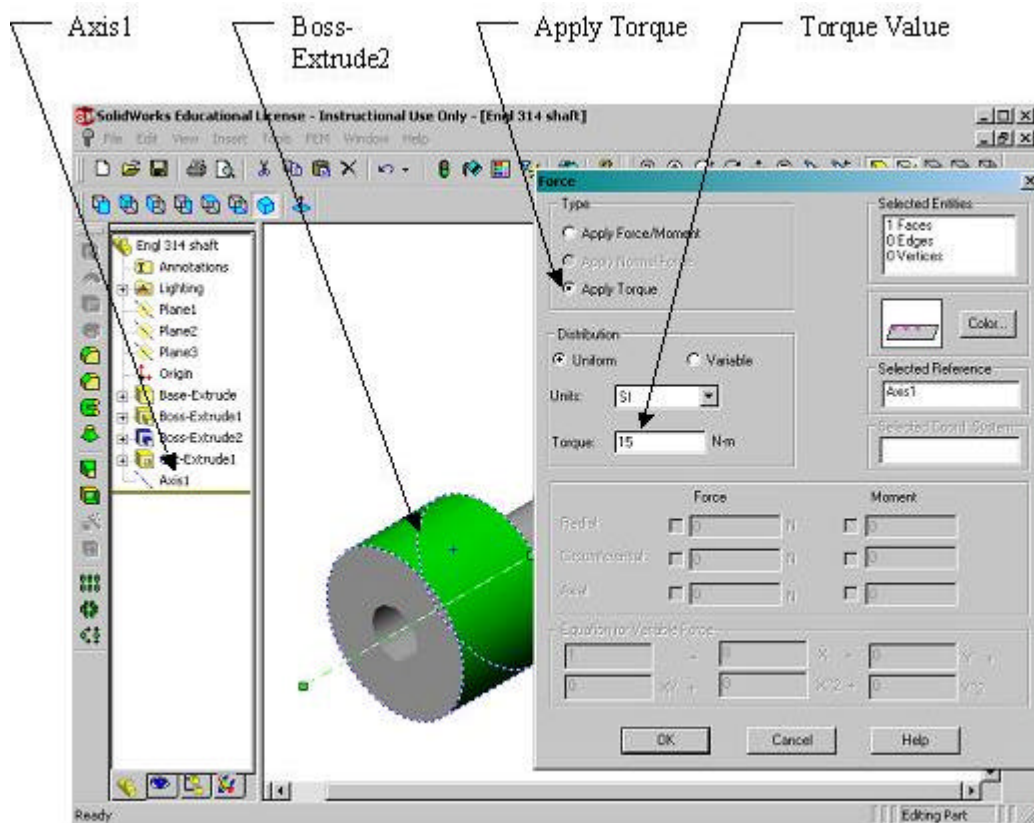


Figure 2.2

10. Use either the **Middle Mouse Button** or the **Rotate View Tool** on the top toolbar to rotate the view to look at the end opposite of the forces just applied.



**Hint:** Instead of clicking on the Rotate Tool the same thing can be done by putting the item mouse icon in the model screen, pressing and holding the middle mouse key and dragging it around.

11. Select the end surface and go to **FEM/Insert/Restraints...**

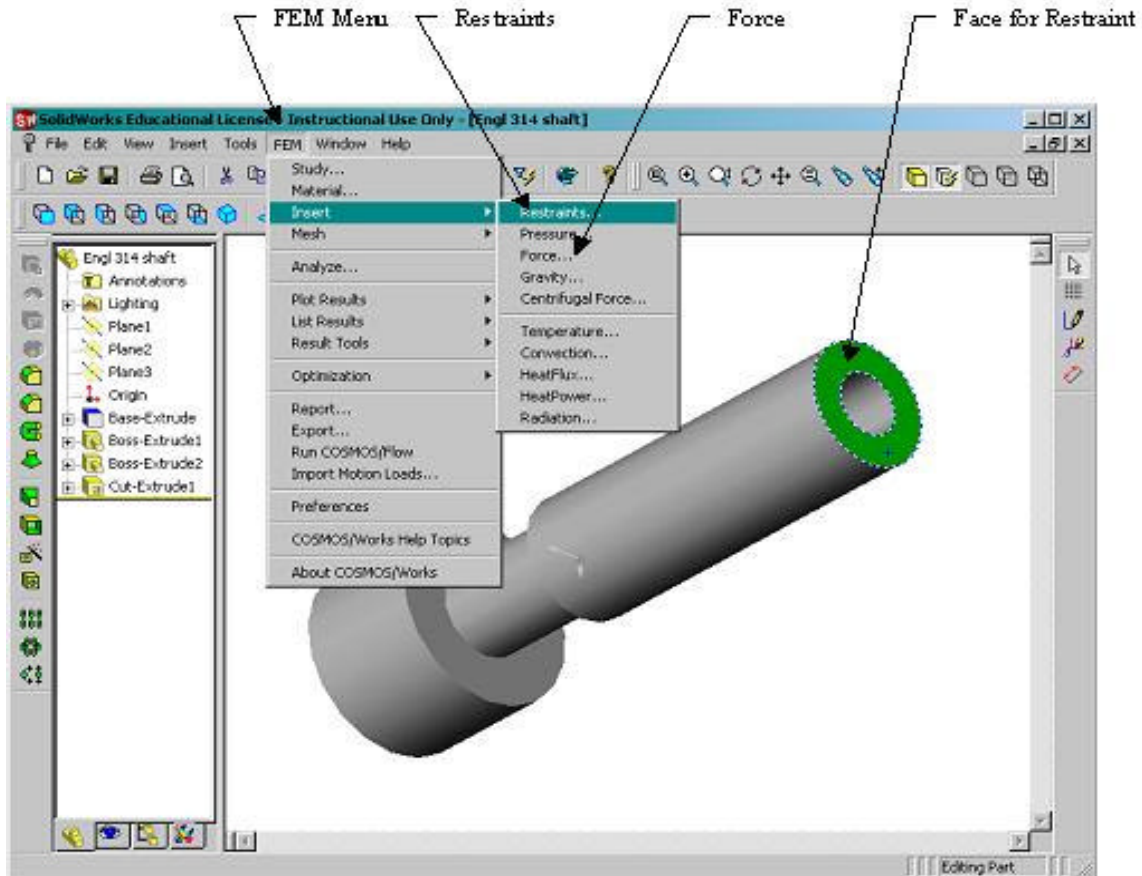


Figure 2.3

12. Pick **Immovable** from the choices and click **OK**.

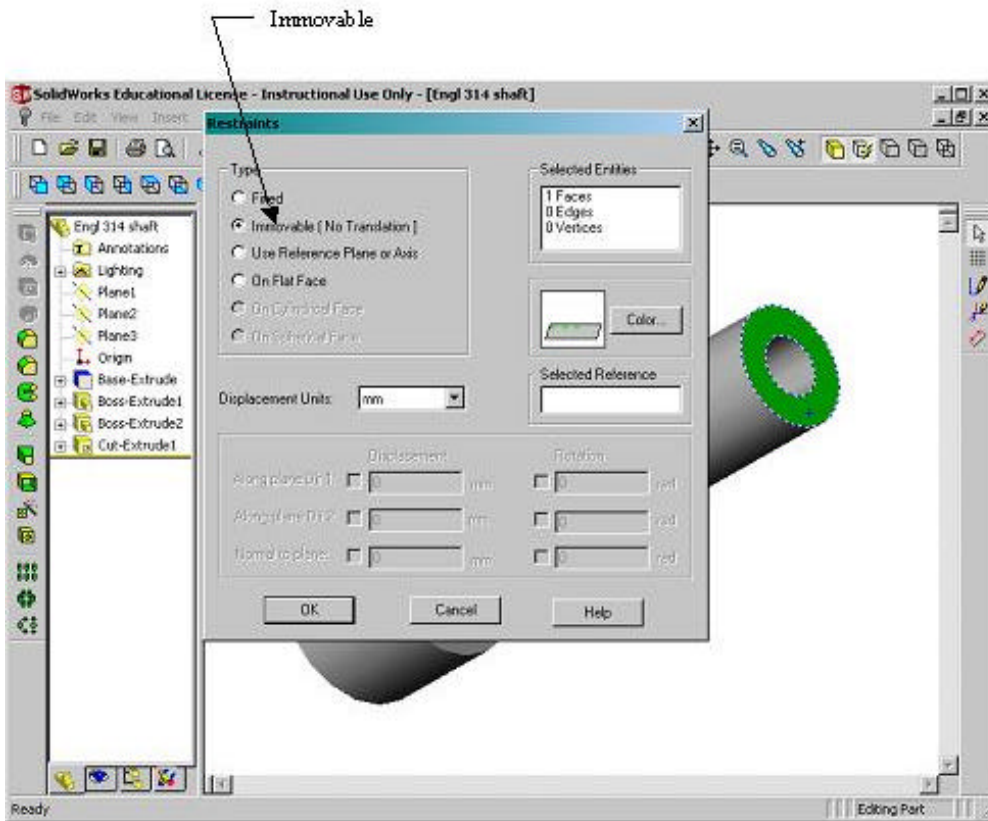
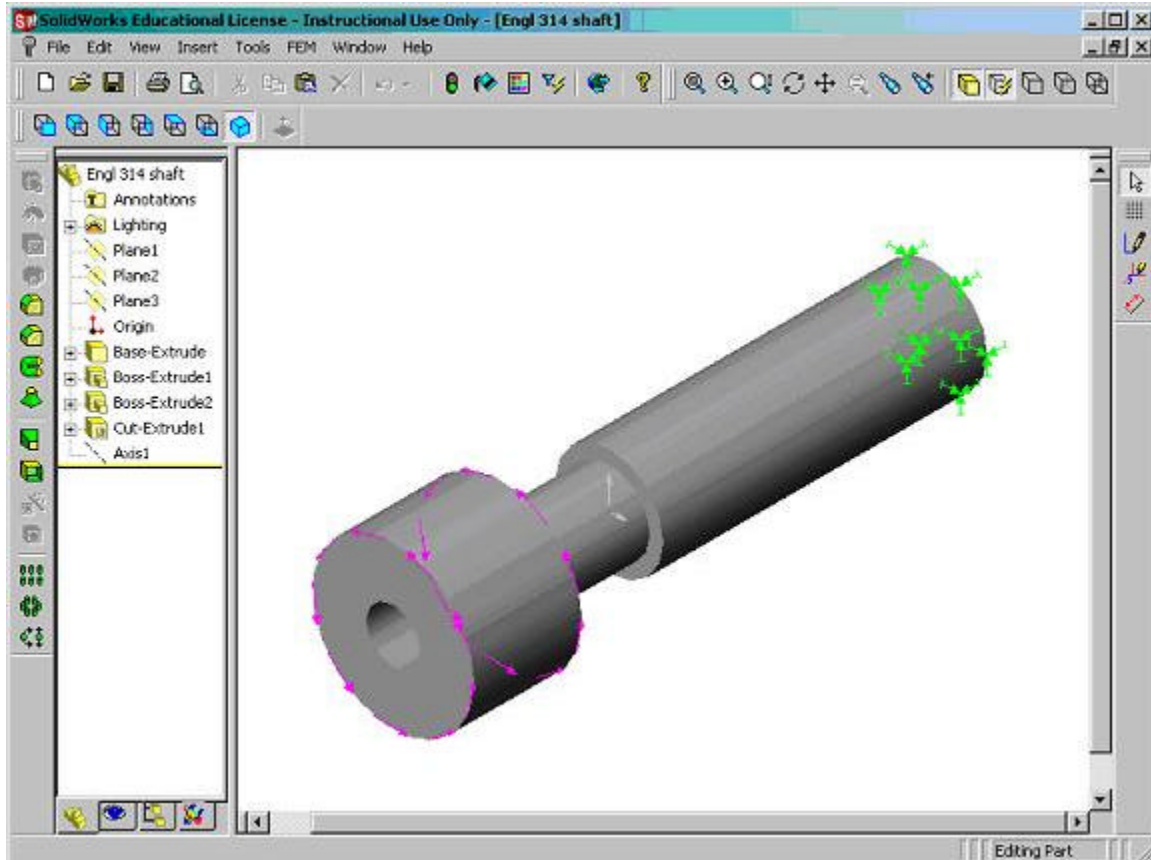


Figure 2.4

- The additions to the model are now complete. The model should have the following forces on it.



*Figure 2.5*

- To start the analysis, go to **FEM/Analyze....**
- A pop-up menu will appear. From this menu pick the study that you would like to analyze. For this example it is "**DEMO**". Then click **OK**. After a short time you should receive a pop-up menu saying the analysis is complete. Click **OK**.

### Part 3 – Analyzing the Results

**Note:** Do not use the Middle Mouse Button or the **Rotate View** Tool after this point. Doing so causes forces to appear all over your screen.

1. You can look at the results in a graphic form. Go to **FEM/Plot Results/Displacement...** For this study, you are going to use the default settings of **Displacement Units** in **mm** and **Reaction Force Units** in **Newtons**. Click **OK**. This will give you a color-coded graph of displacement.

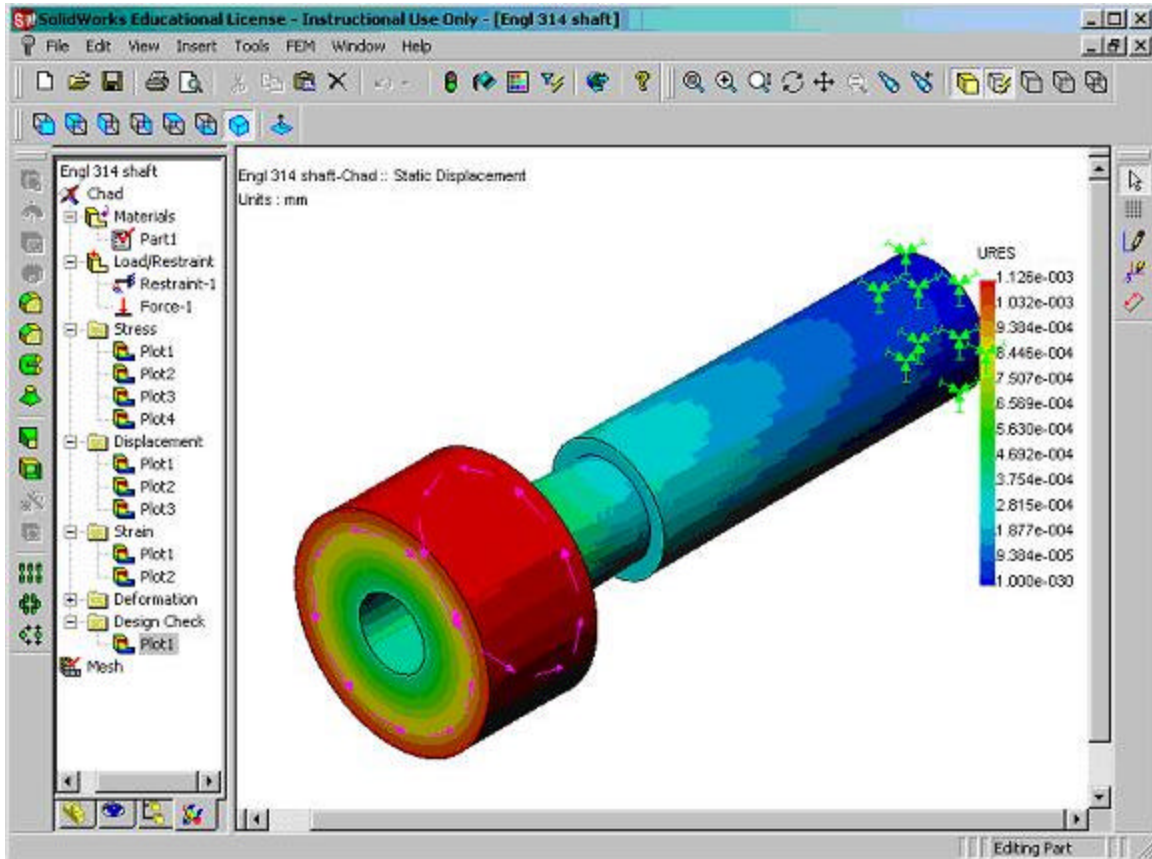


Figure 3.1

2. You can also look at a plot of the stresses on the object. To view this, go to **FEM/Plot Results/Stress...** For this example leave the Stress Units in the default setting of  $\text{N/m}^2$ . Click **OK**. A color-coded plot of the Von Mises stresses will appear.

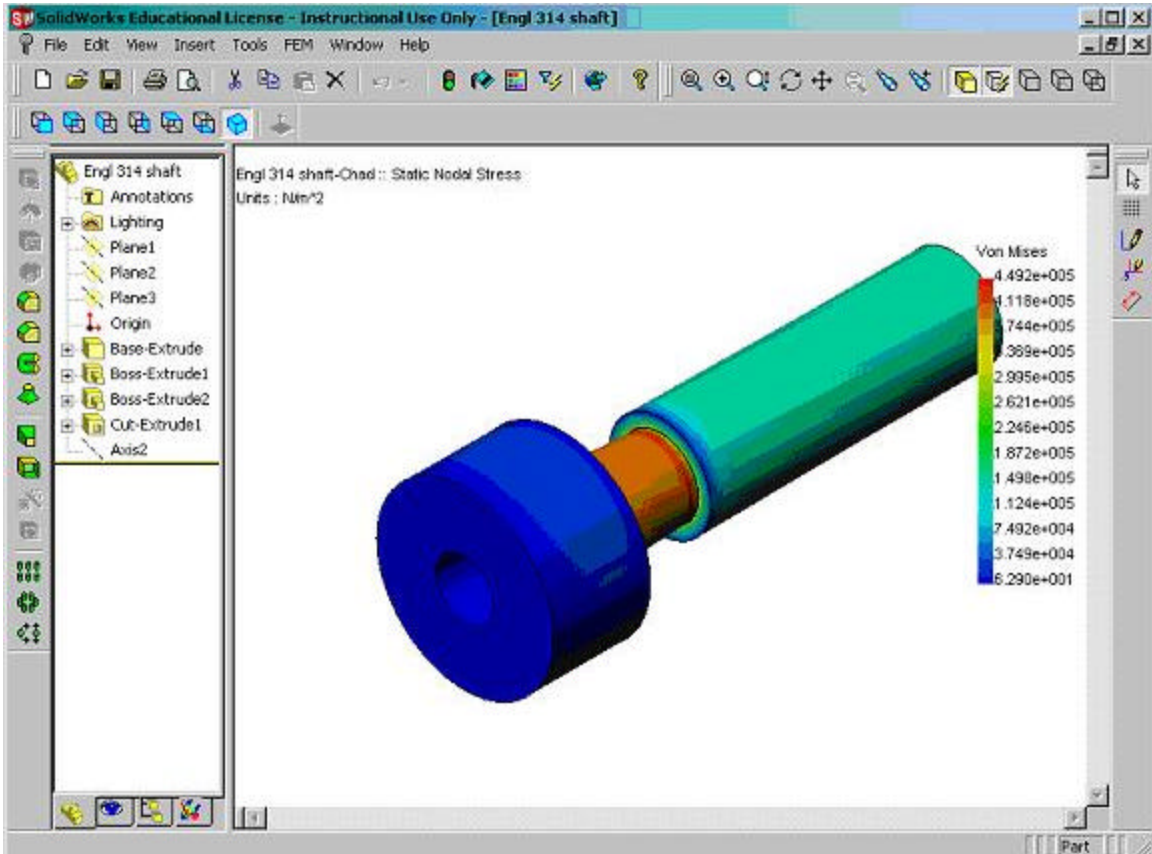
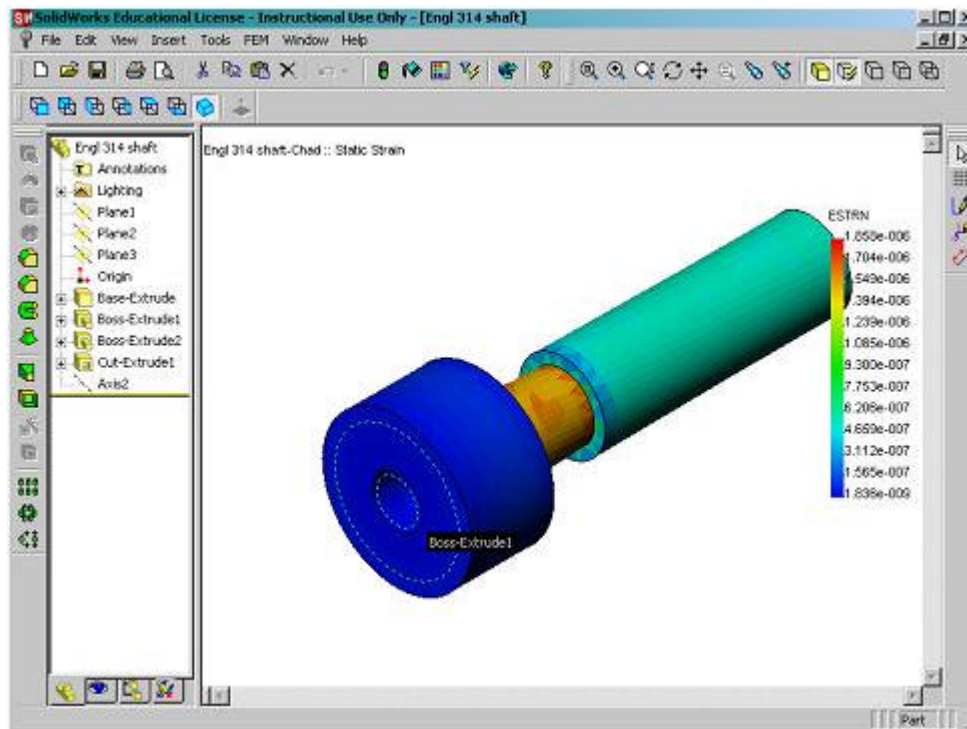


Figure 3.2

- Likewise, the strains on the object can also be plotted. To do this, go to **FEM/Plot Results/Strains...** For this study, all the default settings will be used. Click **OK**. A color-coded plot of the ESTRN strains will then appear.



*Figure 3.3*

- Numerical values of the displacement, stresses, and strains can be viewed by going to **FEM/List Results/(Displacement..., Stress..., or Strains...)**.

5. Go to **FEM/Result Tools/Animate...**

**Hint:** The changes in displacement and stress can be animated using the procedure described in Step 5, but first you must bring up the appropriate plot. Refer to Steps 1 or 2.

6. A pop-up menu should appear. You can adjust the number of **Frames** and the **Speed**, as you desire. For this case they will be left at the default settings of 5 Frames and a Speed setting of 50. Press the **Play** button to see the animation. When you are done viewing the animation, press the **Stop** button, and close the pop-up menu.

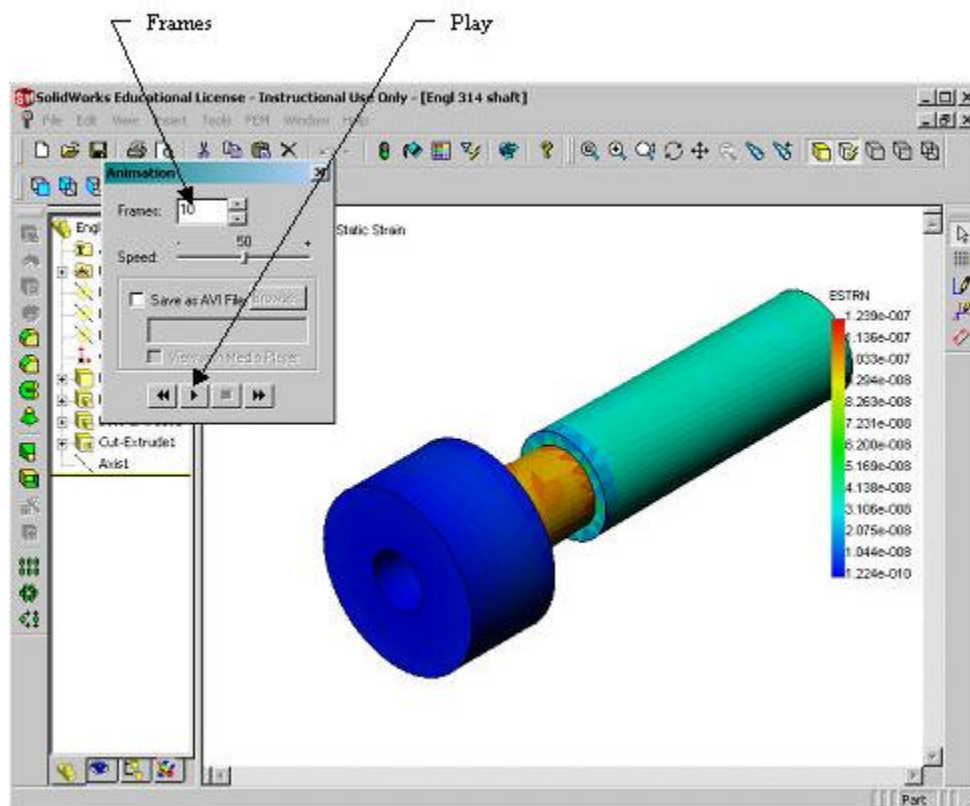
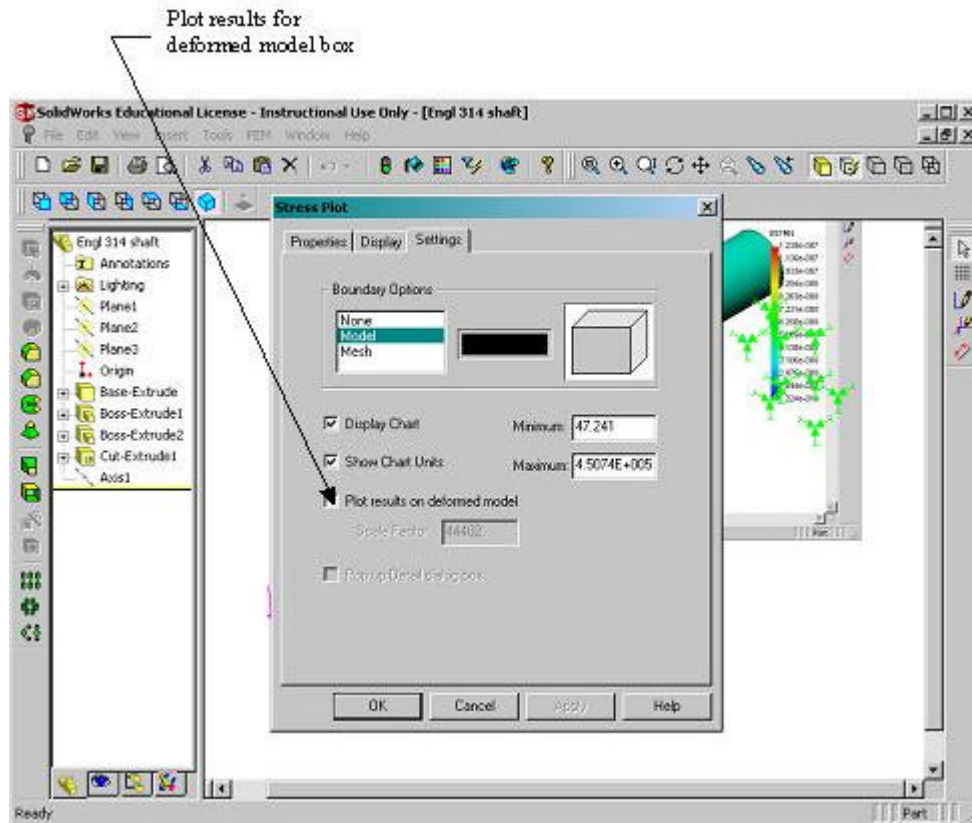


Figure 3.4

**Hint:** For a better view of the process, increase the number of frames to ten.

7. You can find the values of displacement, stresses, and strains at specific points using the following procedure. Go to **FEM/Plot Results/Stress...** Click on the **Settings** tab. Uncheck **Plot Results on deformed model**. Click **OK**.





8. Click the **Feature Manager Plot Tree Icon**.

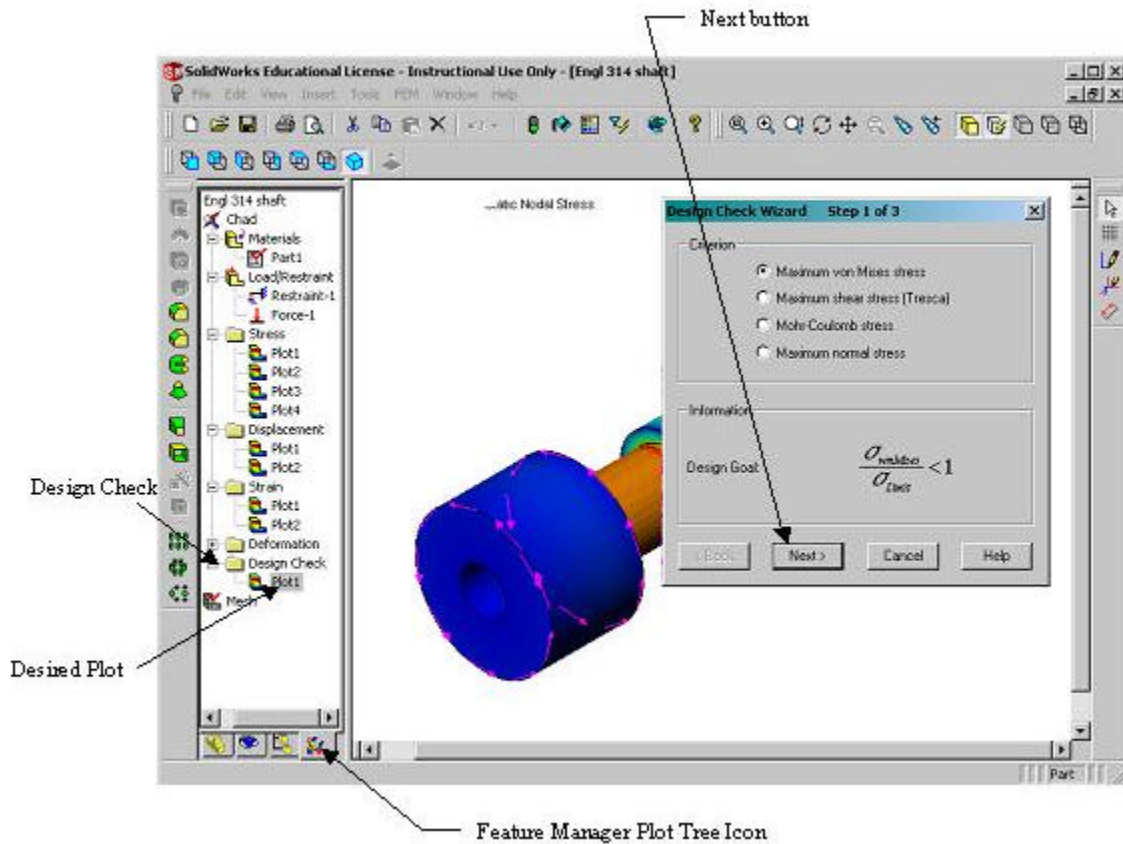


Figure 3.6

9. Click the + by **Design Check** in the tree on the left side of the screen.
10. Double-click **Plot 1**. A pop-up menu labeled **Design Check Wizard** should appear. Click **Next**. A new window should appear, click **Next** again. A new window should appear, click **Finish**.

11. Go to **FEM/Result Tools/Probe...** A **Probe** pop-up menu should appear.
12. Click on any point you want to know the value of the stress. This value will be displayed on the part as well as in a pop-up menu. You can continue to click wherever you need to find values.

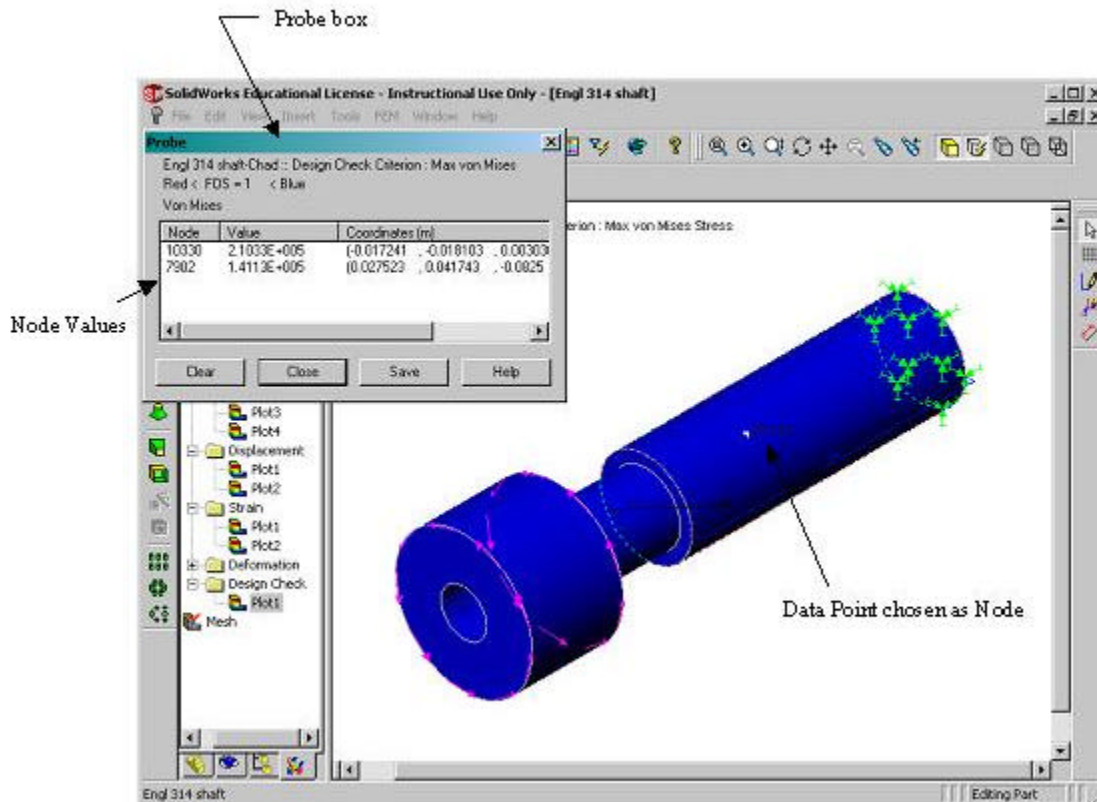


Figure 3.7