

## Lab 2: Pro-ENGINEER Constraints, Loads, and Analysis

Erik Phillips,  
Hiroshi Yamaguchi  
Spring 2009

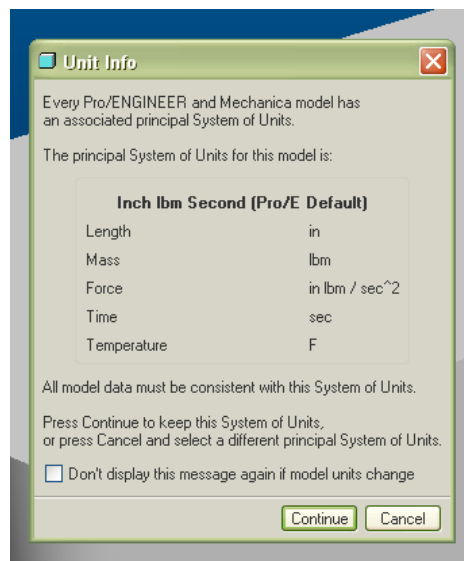
### Part 1: Adding Loads and Constraints

\*\*At the time of writing of these instructions, the Mechanica package is only installed on HAC lab computers 1-17, and 97-107. You'll need to use one of these machines to complete this lab.

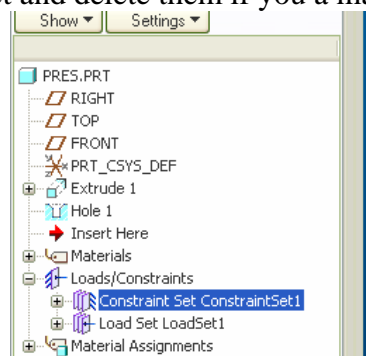
In this lab, we're going to anchor one side of the part you created in lab 1 with a 'displacement constraint'. Then, a load (a force) will be applied to the other side. After assigning a material to your part, we'll run Pro/E's FEM system and look at a visual plot of the stress. Finally, the resulting values will be compared to the stress concentration plots.

It is important that the loading is applied in the same directions as the stress concentration plots!

- Open up your part. Set the working directory where everything will be saved in the same manner as in lab 1.
- From the top menu, click **Applications**, and then **Mechanica**.
- A box will appear listing the current units. Make sure to write your specific units down!



\*\*As you complete the next few steps, the loads and constraints that are created will appear in the left side menu. You can select and delete them if you make a mistake:

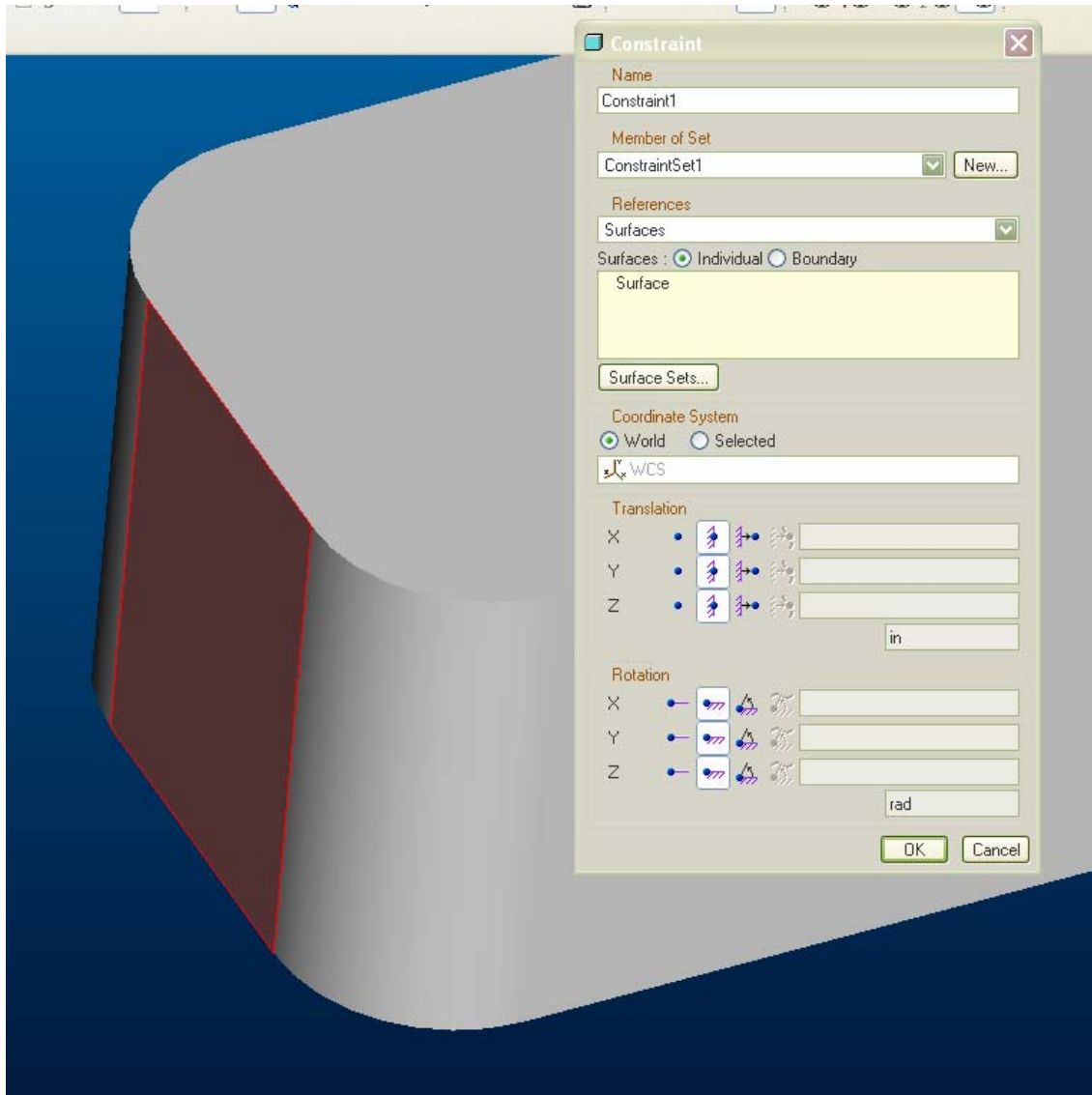


### A: Create Surface Constraints

- In the Mechanics mode, click on the 'New Displacement Constraint' icon, located on the right toolbar.



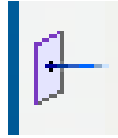
- The following 'Constraint' box appears:



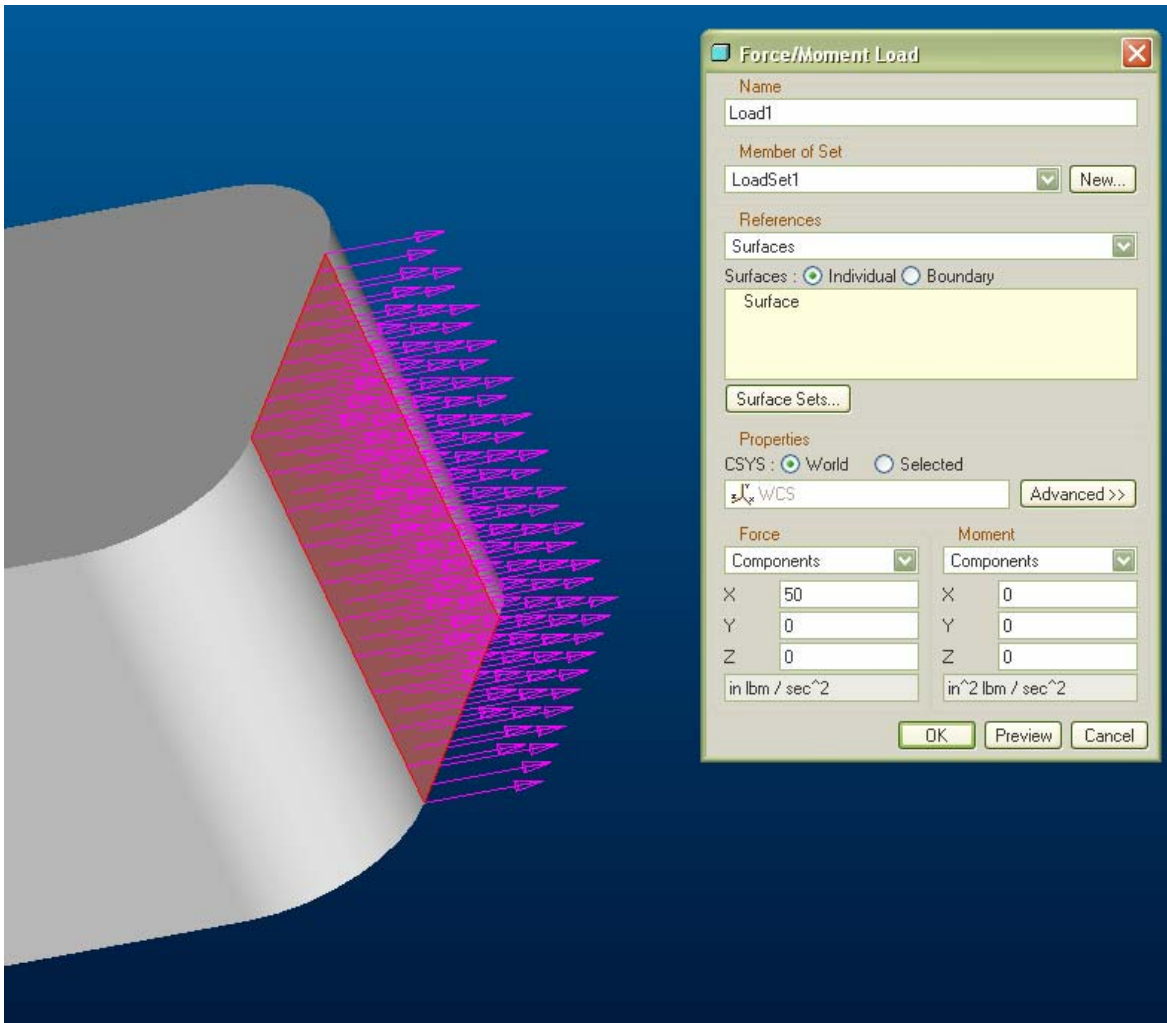
- The only thing to do here is to click the surface that you would like to anchor. Select one of the ends of your part – it should be one of the surfaces that has a force drawn on it in the lab 1 stress concentration plots. After it is selected, press **OK**.

*B: Create A New Surface Load*

- Click on the 'new force/moment load' icon, at the top of the right toolbar



- The following 'Force/Moment Load' box appears:



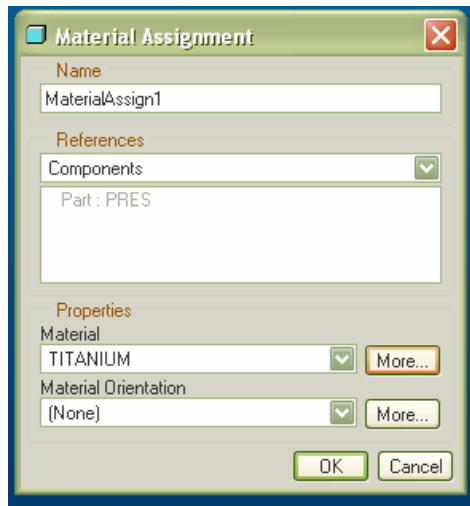
- Choose the surface **opposite** to the one where the displacement was applied. After selecting the surface, you can specify the direction and magnitude by filling in the boxes below 'Force'. For this lab, you want to supply a tensile force, coming out perpendicular to the selected surface. In the picture above, X was the correct direction, but depending on how you made your part, it may be Y or Z. Entering a negative value switches between tension and compression. You can press the **preview** button for an image of which direction the force is going in. The choice of magnitude is completely up to you. Press **OK** when finished.

### C: Define Material Properties

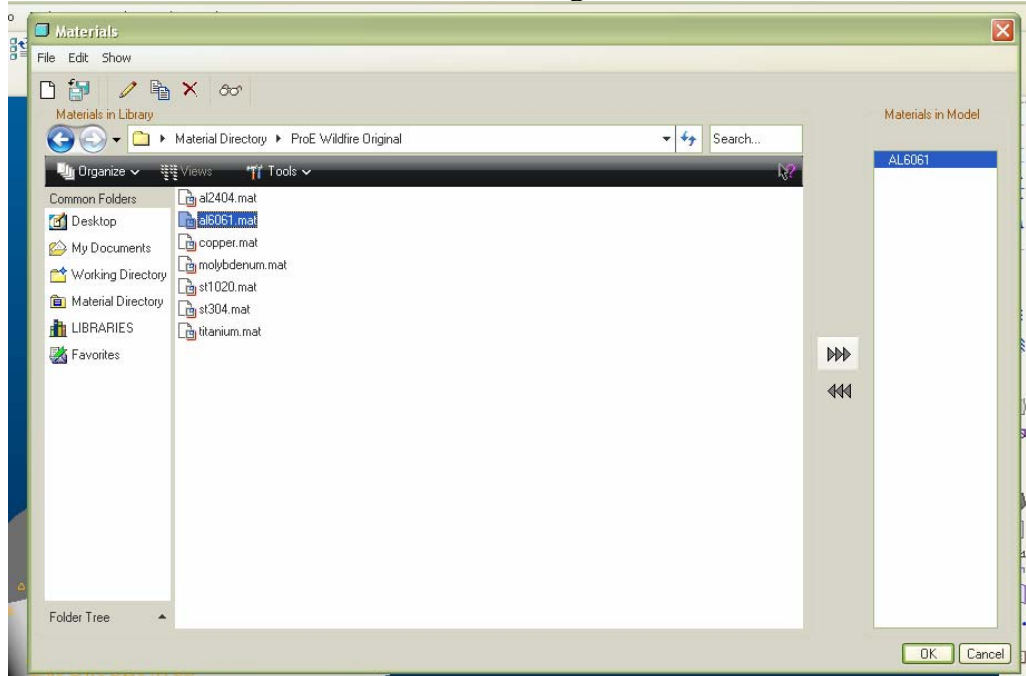
- Now it's time to assign material properties to the part. The icon is on the right toolbar.



- The following box appears:



From the first menu, select the **More** button to the right of the material.



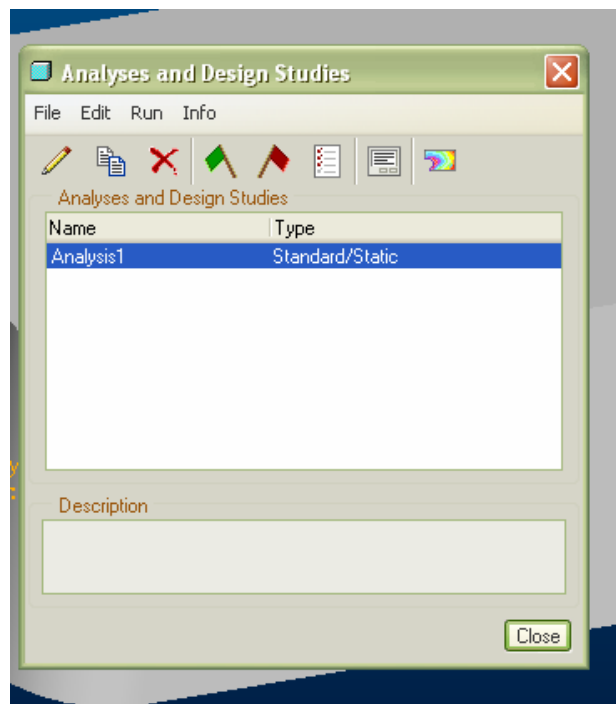
In the next menu, click the **ProE Wildfire Original** folder: this gets you to the above image. Choose *any* material you'd like from the list, press the right arrow ( >>> ) to assign it to your part, and then **OK** your way out of both of the boxes. A little tag will now appear on your part.

## Part 2: FEM Analysis

### *D: Running the Analysis*

At this point, we've locked the part in place, applied a load, and given the part physical properties. Now we're going to let Pro/E calculate the stress throughout the part.

- From the top menu, under **Analysis**, select **Mechanica Analysis/Studies**.
- The 'Analysis and Design Studies' box will appear. Under **File**, select '**New Static...**'
- The 'Static Analysis Definition' box will appear. Keep the settings as they are: press **OK**.
- This puts us back at the 'Analysis and Design Studies' box. With Analysis1 highlighted, press the green flag icon, or go under **Run** and hit **Start**:



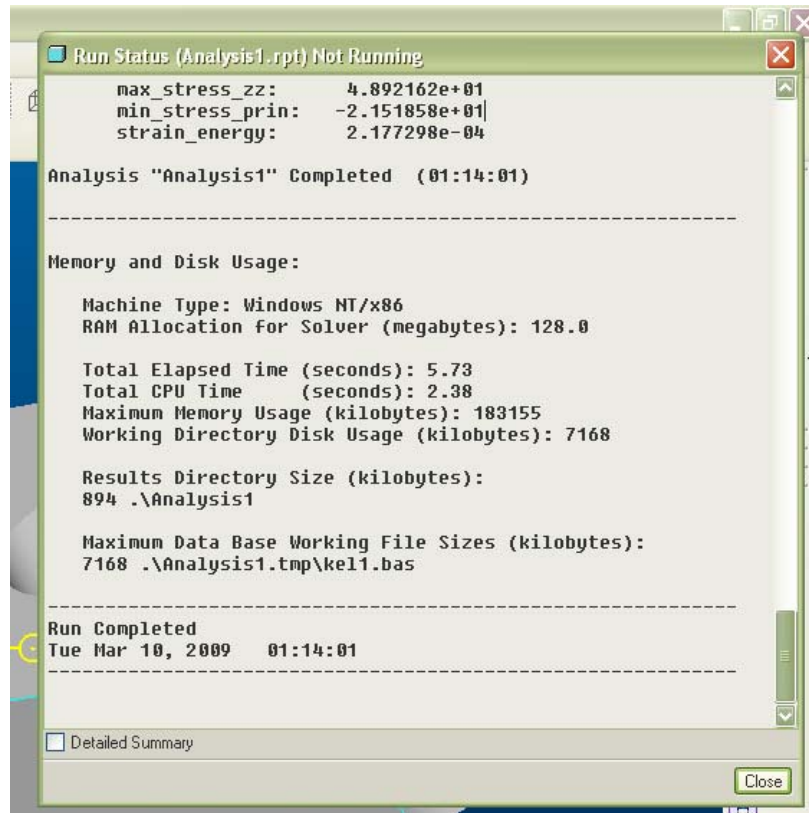
- If Pro/E offers to run diagnostics, say **Yes**.

The FEM analysis will now start. Depending on the individual computer and the size of your part, this could take anywhere from a few seconds to a few minutes.

- While the FEM solver is running, we can view it's progress by selecting the 'Display Study Status' icon, at the top of the 'Analysis and Design Studies' box:

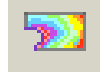


This creates the following log of Pro/E's actions. Wait until it reaches 'Run Completed' at the bottom of the log before moving on.

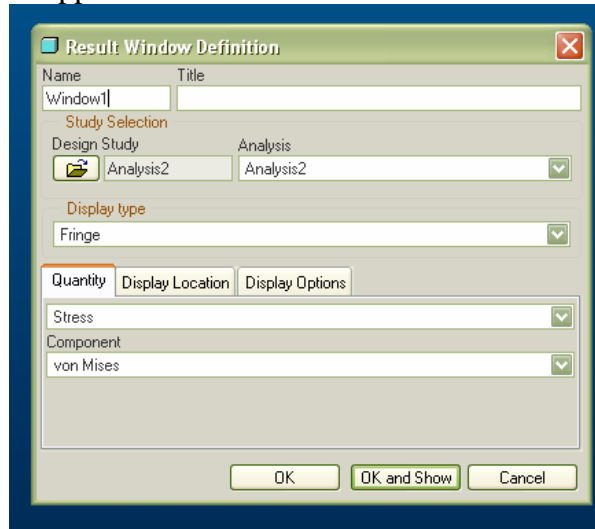


## E: Viewing Results

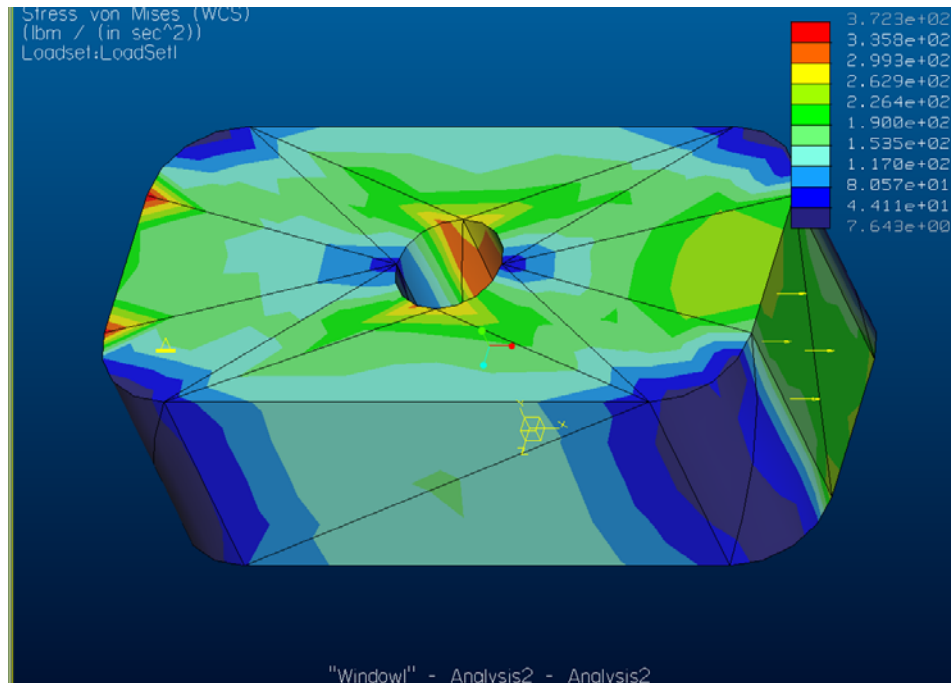
- Now it's time to view the results. Back at the 'Analysis and Design Studies' box, click on the 'view results' icon:



The following window will appear:



- Select the **Display Options** tab. Under this will be a series of boxes: select the **Show Element Edges** box, and then press **OK and Show**. A new full-screen window will appear.



See the triangular grid? That's the finite element mesh. Depending on how big your piece is, you may have more or less lines than the sample here. Pro/E uses an adaptive mesh that alters element size for the best results.

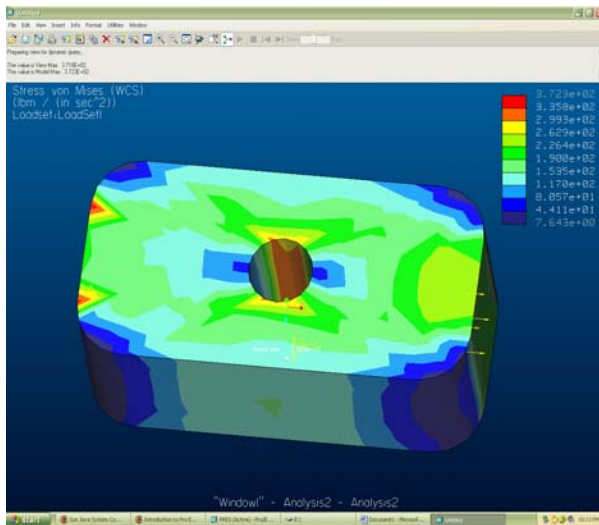
The colors show the von Mises contours; they create a ‘map’ of the different stress levels, with the key on the top right of the screen.

- Find the maximum stress value by selecting **Info** from the top of the results window, and then **Model Max**.
- Write down the value of the max stress and note its location.
- Also, record the units: they are listed on the top left of the screen.

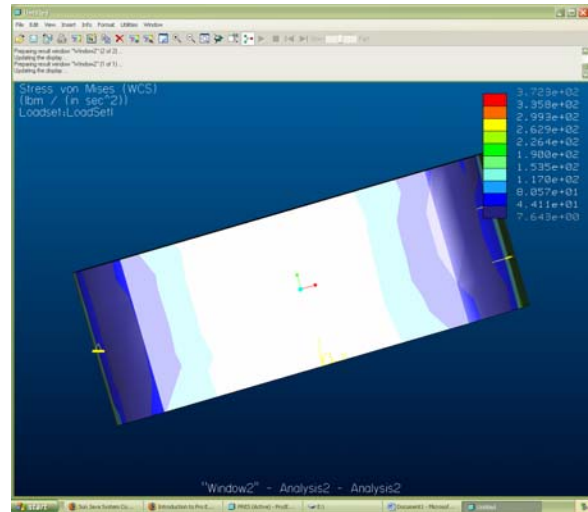
The lab requirements ask for a color printout of the finite element mesh, and the von Mises contours.

- Rotate the image to make sure that the stress concentration notch or hole on your part is clearly visible.

Good



Bad



Print out the ‘good’ image above.

### *F: Calculating Stress Concentration*

- Compute the Stress Concentration Factor, using the max stress on your part (which should be in the notch or hole). If for some reason the max stress does not appear, or doesn’t appear in the notch or hole, use the key on the plot to estimate the maximum stress at the notch or hole. Compare this value to what is predicted by the plots in lab 1.

## Summary

For the second lab, you will need to turn in four things:

1. Provide a drawing that shows the part dimensions and the applied forces. This can be a simple but neat sketch, showing the  $r$ ,  $h$ ,  $H$ , depth, etc values from lab one, as well as the location and magnitude of the forces.
2. The color printout of your part with the von Mises contours and FEM grid. No black and white!
3. A calculation of SCF, and a comparison with the expected values along with your thoughts on any differences.
4. A few comments and thoughts on this lab.

Extra Credit:

- (5.) Provide a second color printout of the von Mises contours of your part with double the thickness. How does this compare to the original part?