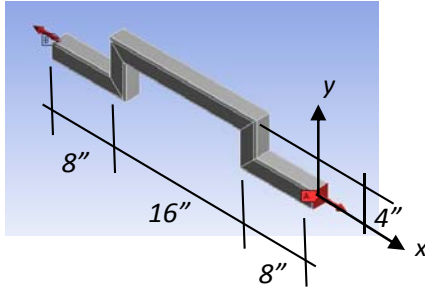


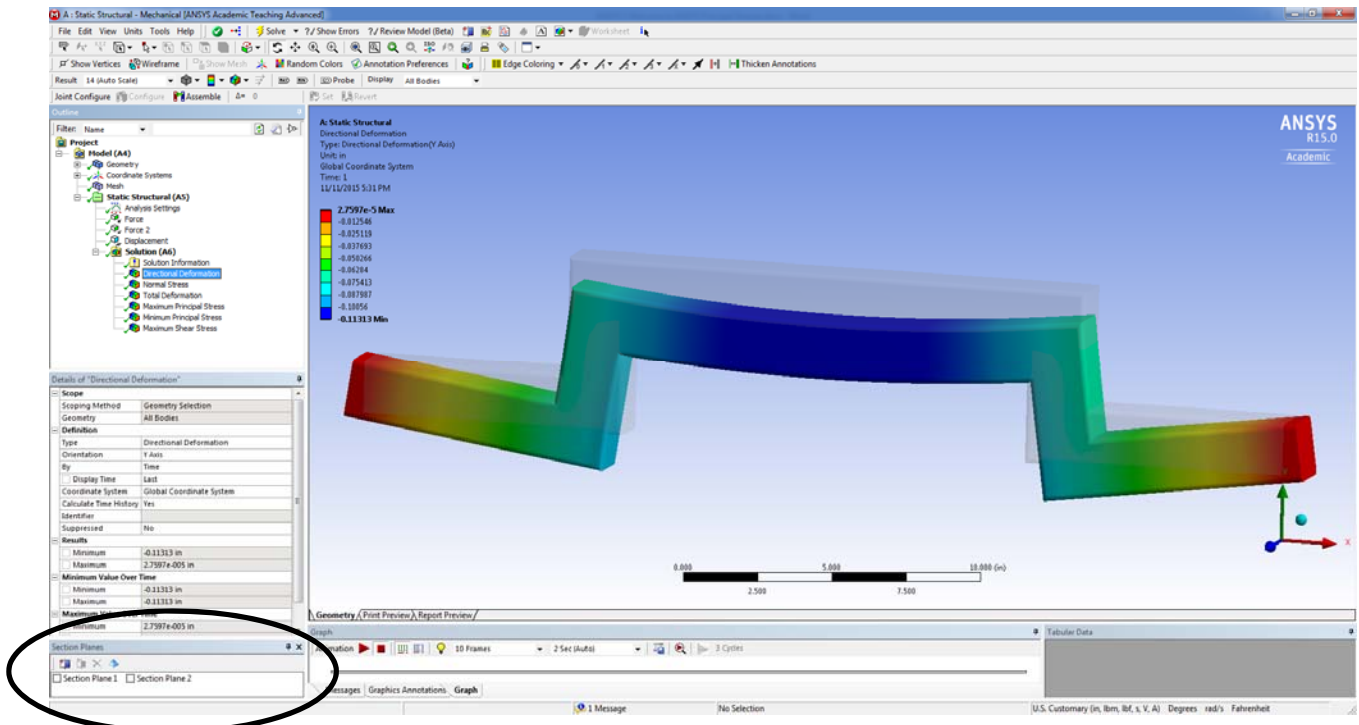
TUTORIAL 5: Principal Stresses and Maximum Shear Stress in Combined Load Problems

In this tutorial you will learn how to interpret principal stress results and maximum shear stress results in ANSYS. The first bar you will work with is the eccentric bar you drew in Tutorial 4. The bar had a 2"x2" square cross-section and was made out of 'Lafayette Steel'. The lengths of the bar segments were 8 inches, 16 inches and 8 inches in the x-direction. The vertical jog in the bar was 4 inches in the y-direction. An axial load of 10,000 lb. was applied to the bar in the x-direction. This axial load created both axial and bending normal stresses in the middle, 16" length of the bar and only axial normal stress in the 8" end sections.

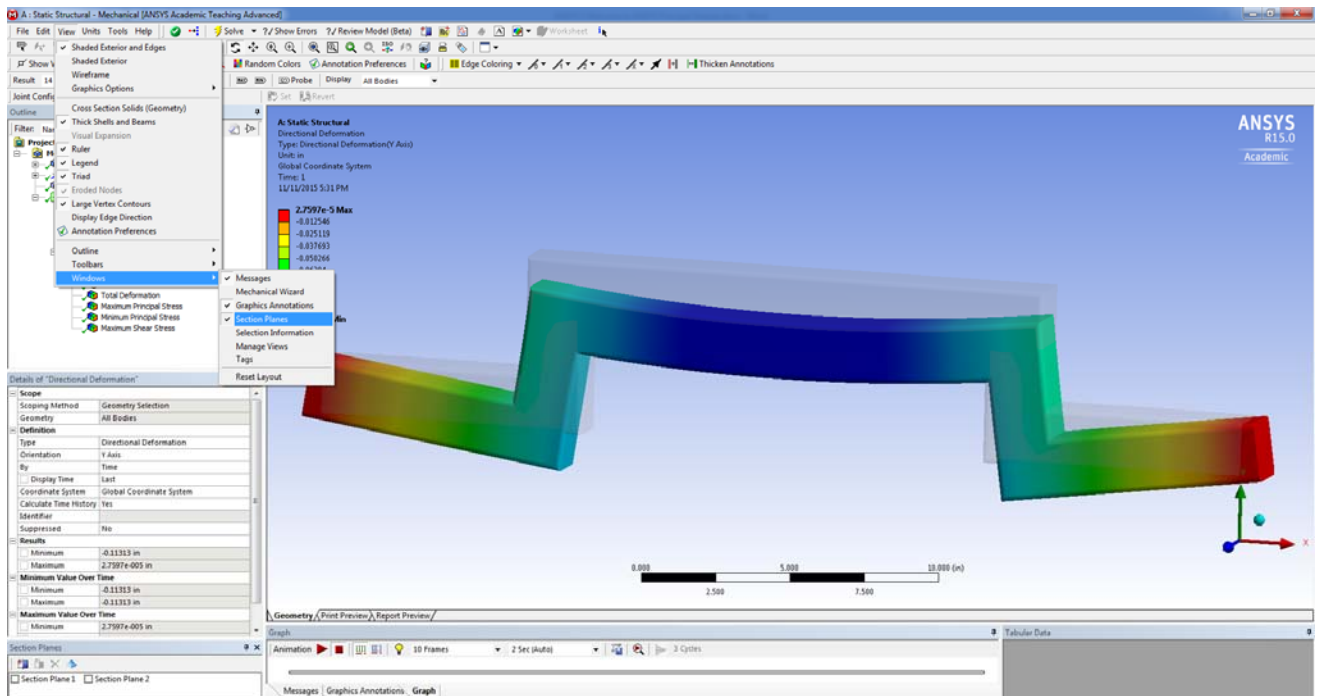


Modification of Tutorial 4 Model – Axial and Combined Bending/Axial Effects

Open your saved (hopefully!) Tutorial 4 Model in ANSYS and then open the **Mechanical** window to view the model.

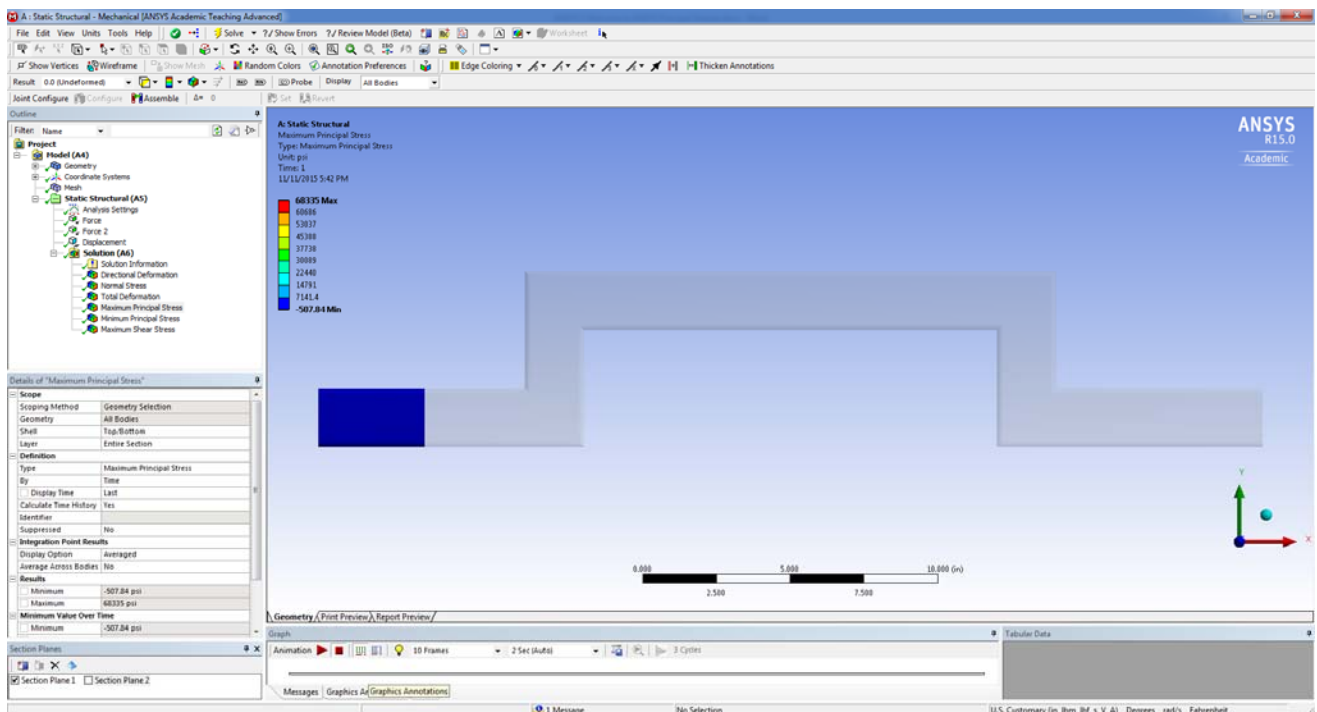


Add three additional **Result Viewers** to the Model: *Maximum Principal Stress*, *Minimum Principal Stress* and *Maximum Shear Stress*. **Solve** the model to get results for the new Viewers. Try to view the **Maximum Principle Stress** results on the **1st Section Plane**, which was cut through the 8" inch end portion of the bar. All of the available Section Planes are shown in the very bottom LHS window as shown in the view above. If you do not see this window then you can add it to the view by checking to turn on this window with **View->Windows->Section Plane** in the top toolbar. This toolbar option is shown in the next view. Using the checkbox for each Section Plane in this window, you can switch back and forth between the whole bar and each section plane cut created.



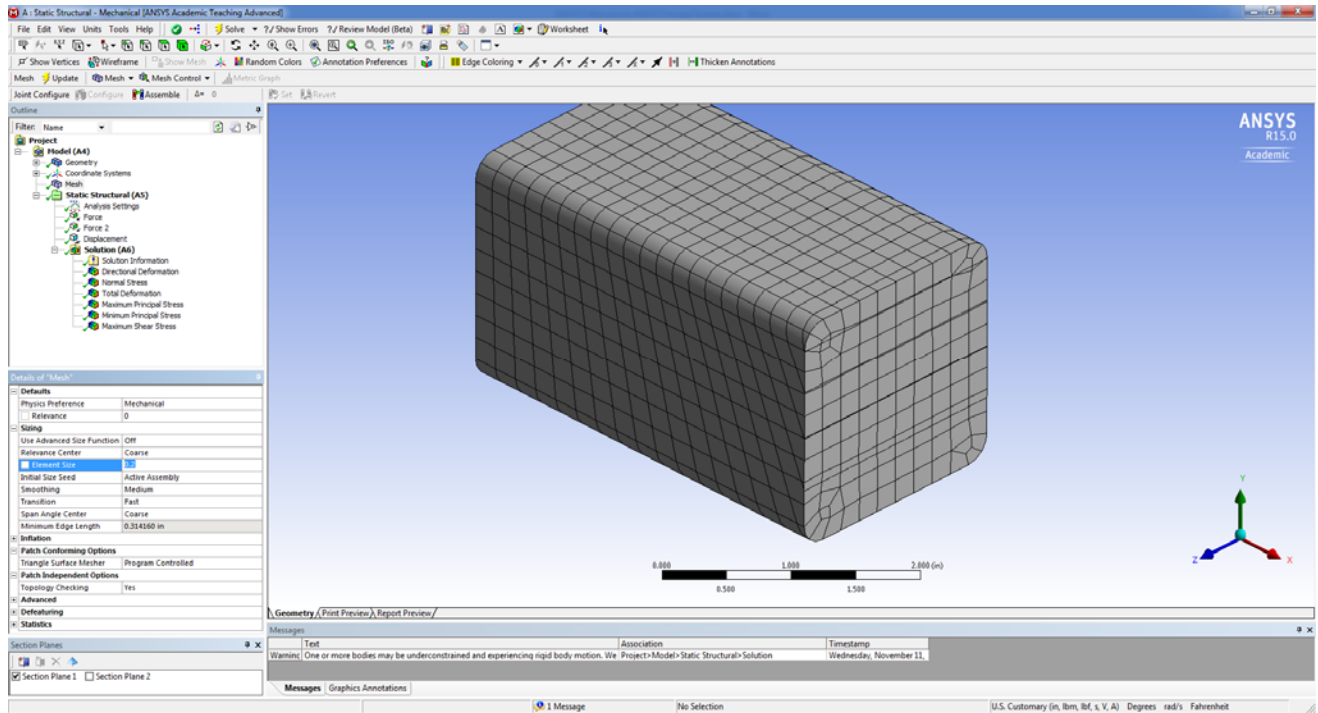
Maximum Principal Stress Results

The *Maximum Principal Stress* results provided by ANSYS corresponds with the principal stress, σ_1 , you calculate when determining a stress transformation of a state of stress at a specific point. Typically, by hand you also calculate, σ_2 , but this is provided by ANSYS through a separate viewer, which is the *Minimum Principal Stress* viewer.

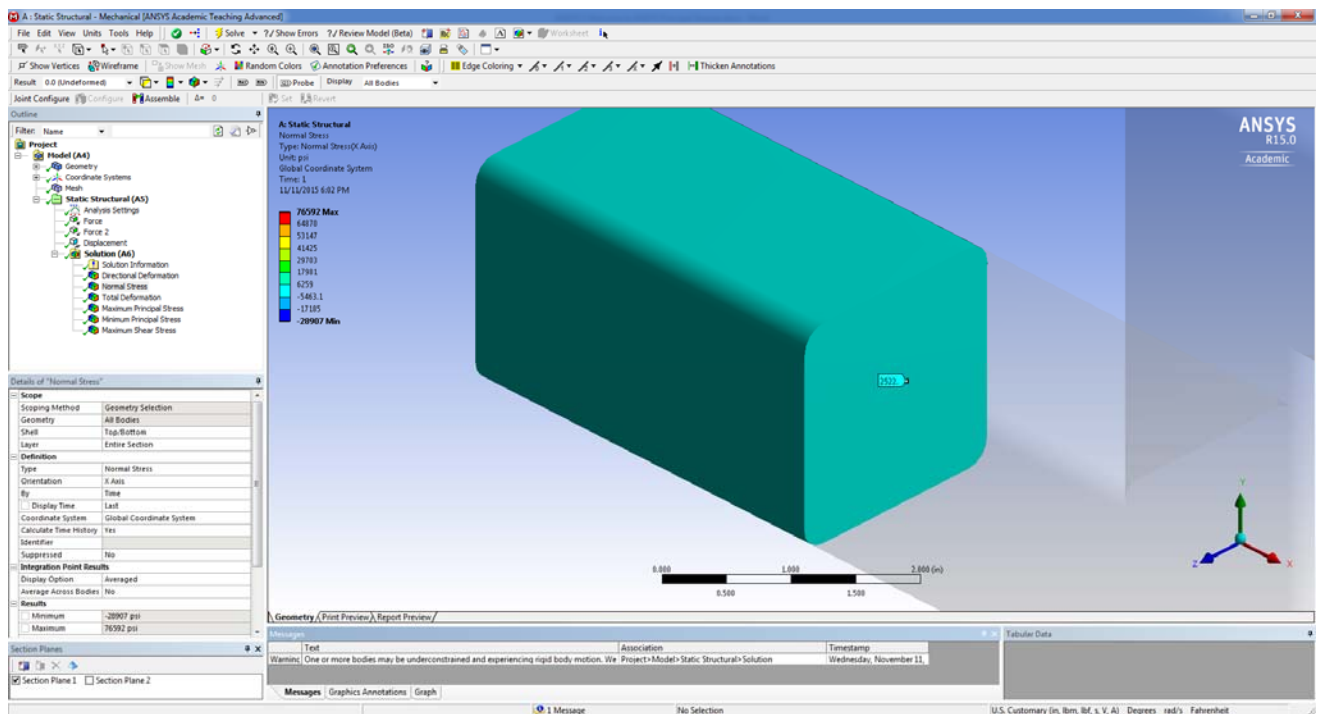


In Tutorial 4, the normal (axial) stress in the 8" end portions (non-eccentric portions) was approximately 2530 psi. The normal stress, which in theory is constant across the cross-section, however did vary across the cross-section from +/- 30 psi or so. This is due to using a Coarse mesh. Before continuing on, you should remesh the eccentric bar with a more refined mesh. This will provide more accurate results, although it will take a little longer to obtain the results. To remesh the bar, click on the *Mesh* leaf in the LHS tree. In the options boxes that appear in the lower LHS menu, type *0.2* in the **Element Size** box. Then right-click on the *Mesh* leaf and select **Clear Generated Data** and click OK in the popup dialog. Once this is done, right-click on the *Mesh* leaf and

select **Generate Mesh** to obtain a refined mesh for the bar (more, tinier elements), which will be similar to what is shown below. **Solve** your model again to update all of the results.



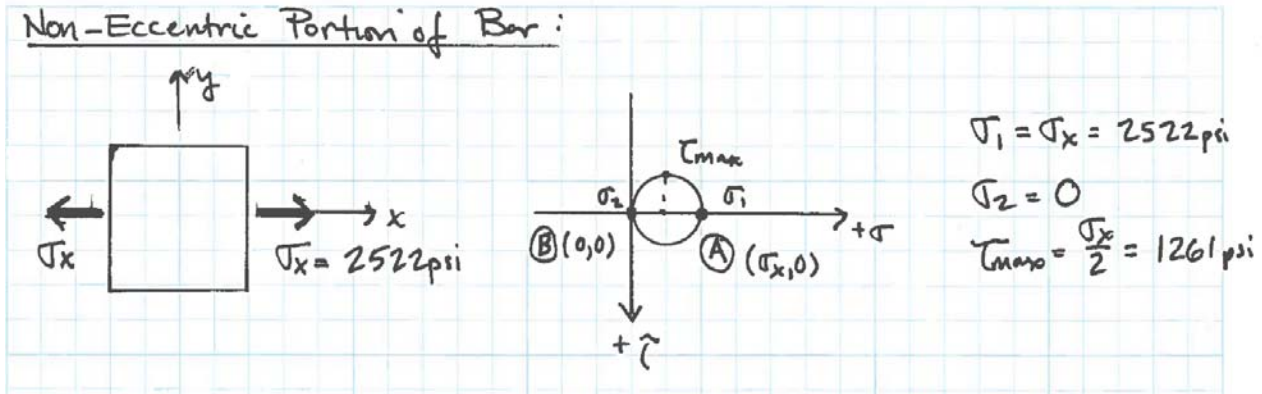
Now change to the **Normal Stress Viewer** and use *Probe* to look at the stress values on the section plane surface. The stresses now vary by less than ± 1 psi on the cross-section in general. The normal (axial) stress now is 2522 psi, which is a more accurate result of the actual normal stress in the bar than the previous value obtained with the coarser mesh.



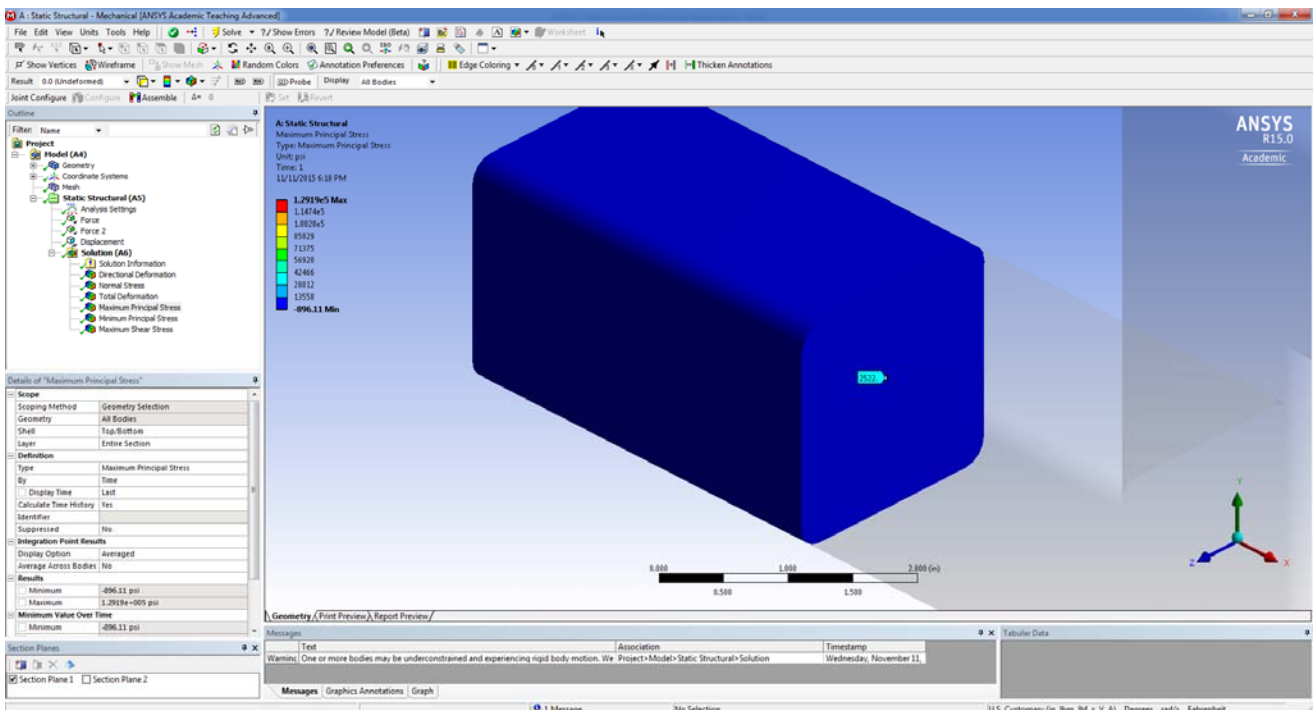
There is only normal (axial) stress acting on any of the stress elements located on this section plane cut. Therefore a 2D stress element can be used to capture the state of stress at any location on the cross-section. The 2D x-y stress element in shown in the sketch below. The Mohr's Circle for this 2D state of stress, which is called a plane stress state, can be drawn starting with points A and B as discussed in class. The Mohr's Circle

for the 2D x-y stress element is also shown in the sketch below. From the Mohr's Circle sketch you can obtain the values of the two principal stresses, σ_1 and σ_2 , as well as the value of the maximum shear stress, τ_{max} . These values can be used to represent the same state of stress as the original 2D x-y element and are very useful in determining the failure of a material at a particular location on a bar or element. You can directly compare these principal stress values and maximum shear stress value obtained using Mohr's Circle with the principal stress and maximum shear stress values that ANSYS provides.

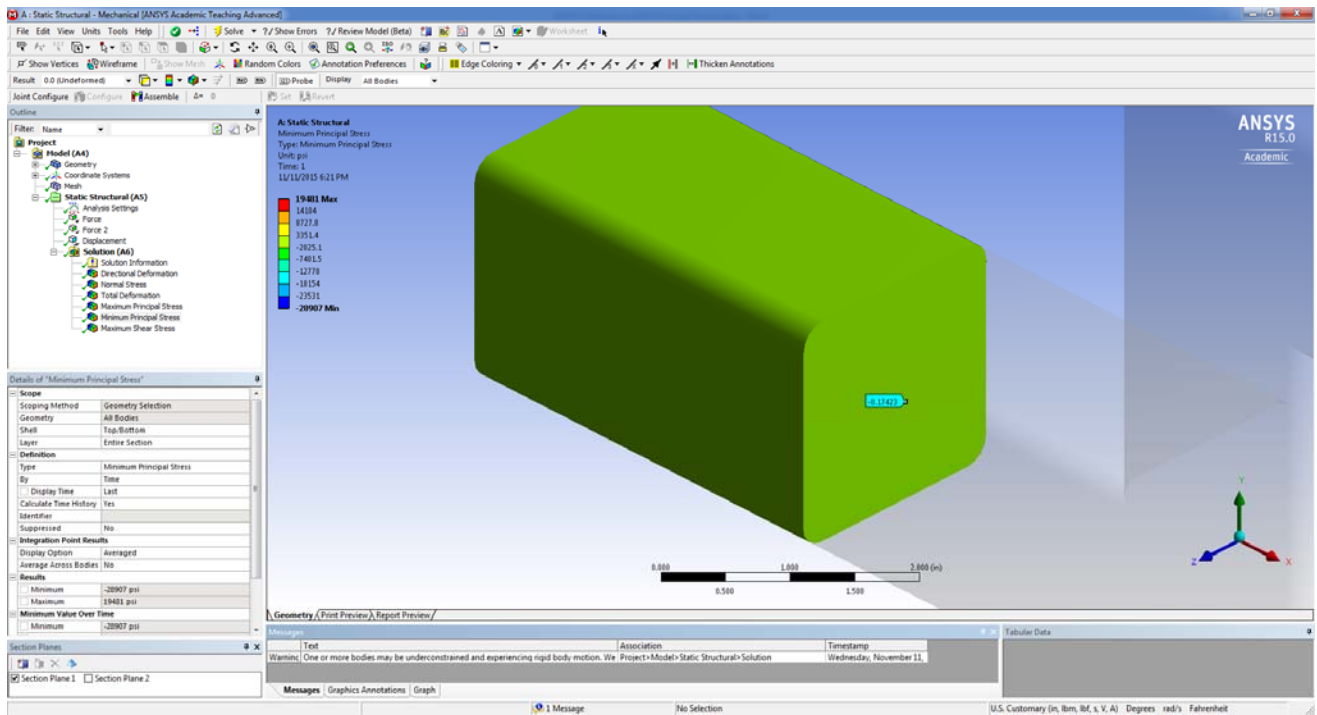
Figures showing the Stress Transformation of 2D x-y Stress Element in non-eccentric portion of the bar



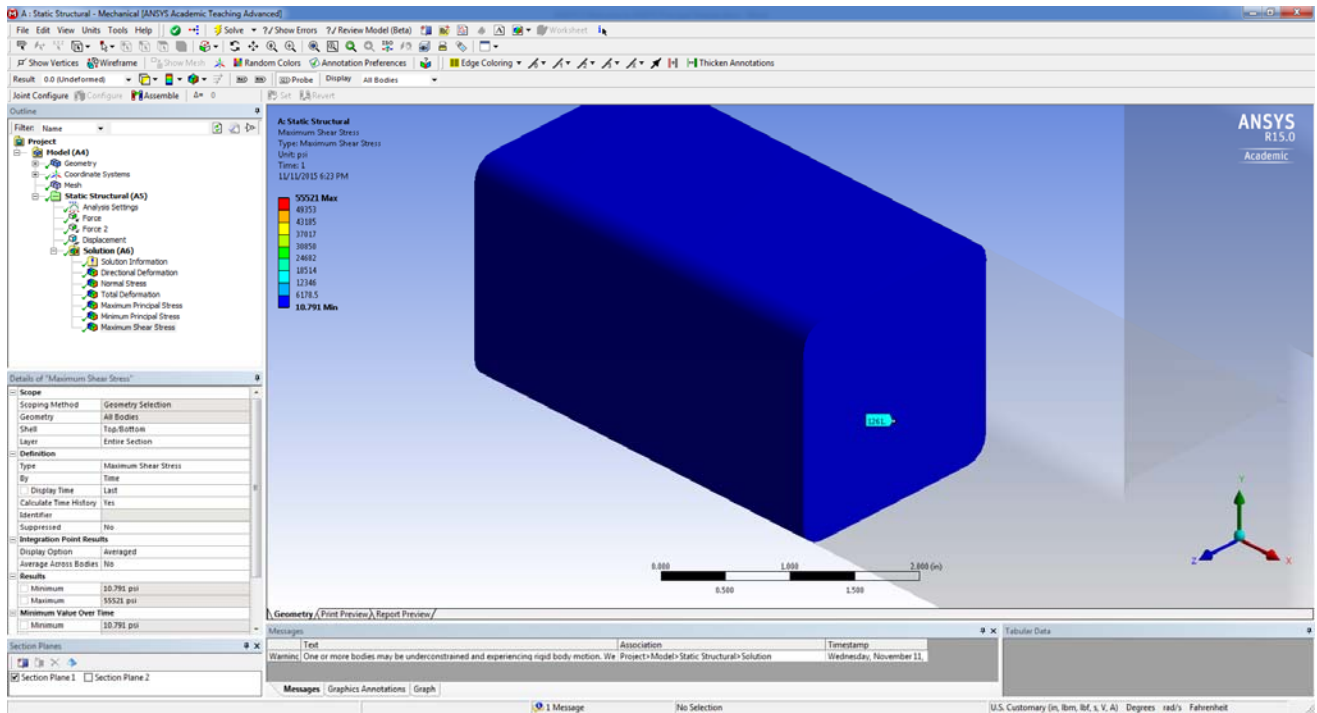
To start the comparing values view the **Maximum Principal Stress** results. Use *Probe* to determine the maximum principal stress, σ_1 , for any point on the cross-section at this section plane cut. The value of $\sigma_1 = 2522$ psi found by ANSYS as shown in the view below matches the Mohr's circle hand calculations result of $\sigma_1 = 2522$ psi.



Now view the **Minimum Principal Stress** results. Use *Probe* to determine that the minimum principal stress, σ_2 , is very close to 0 psi at any point on this section cut. The value of $\sigma_2 = 0$ psi found by ANSYS as shown in the view below matches the Mohr's circle hand calculations result of $\sigma_2 = 0$ psi.



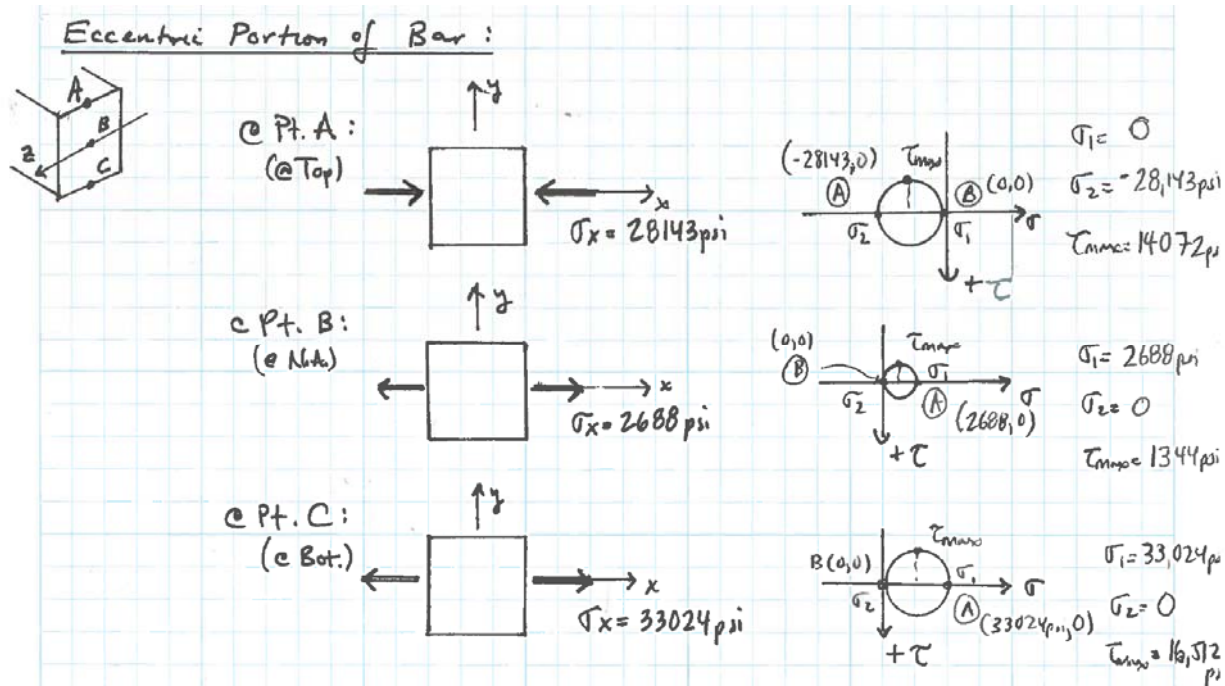
Now view the **Maximum Shear Stress** results. Use *Probe* to determine that $\tau_{\max} = 1261$ psi (or very close) at any point on this section cut. The value of $\tau_{\max} = 1261$ psi found by ANSYS as shown in the view below matches the Mohr's circle hand calculations result of $\tau_{\max} = 1261$ psi.



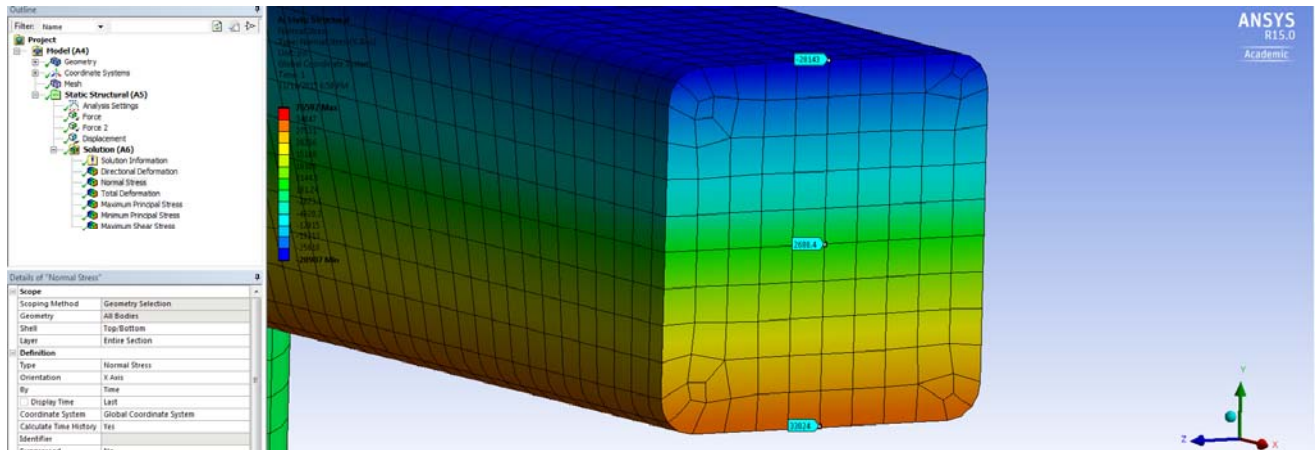
Now change to the **2nd Section Plane** that was setup to cut through the middle eccentric portion of the bar by checking the 2nd Section Plane in the Section Plane window and unchecking the 1st Plane. Repeat looking at the three results views and compare the ANSYS Results obtained with the results obtained by hand for specific 2D stress elements using Mohr's Circle. In the eccentric portion of the bar, the normal stress varies from the top to the bottom of the cross-section due to the combined bending and axial normal stresses acting on the cross-section. Therefore you can look at points of stress at several different locations through the depth of the cross-section. In the Mohr's Circle hand calculations presented below, three 2D x-y stress elements are considered:

one at the top of the cross-section, one at mid-depth of the cross-section and one at the bottom of the cross-section.

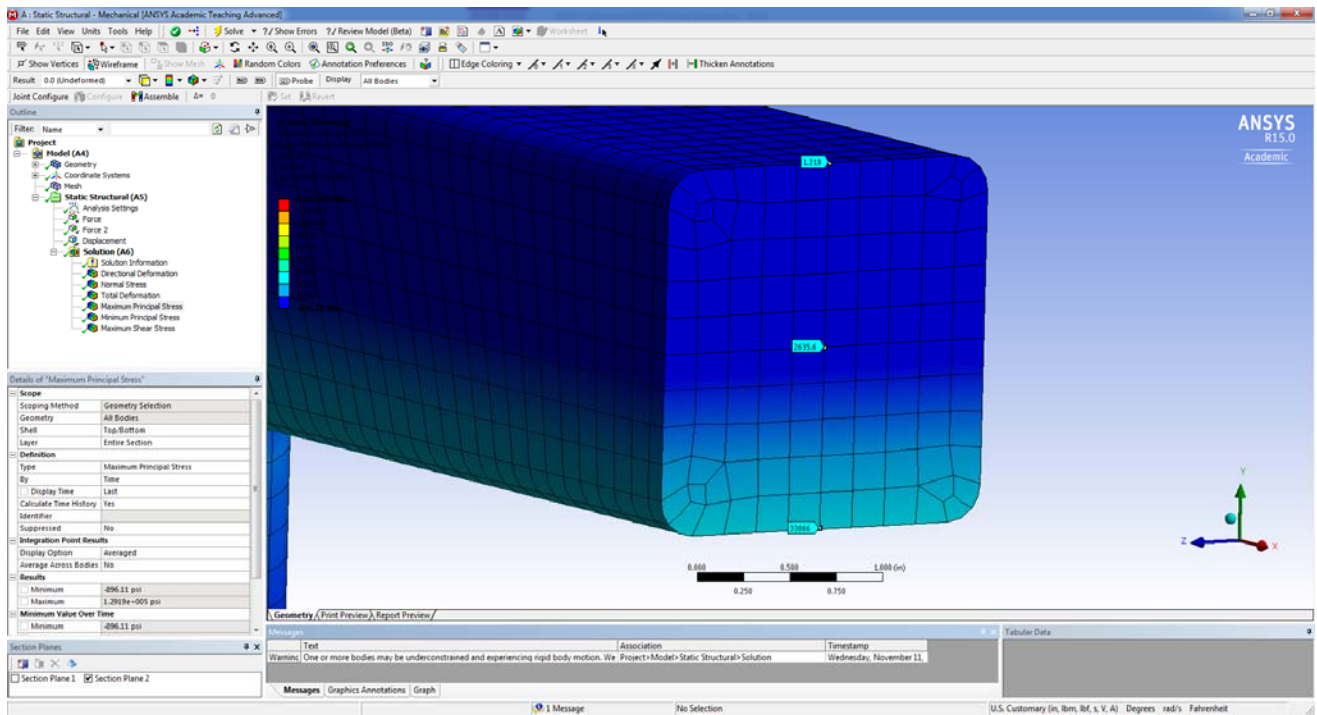
Figures showing the Stress Transformation of 2D x-y Stress Element in non-eccentric portion of the bar



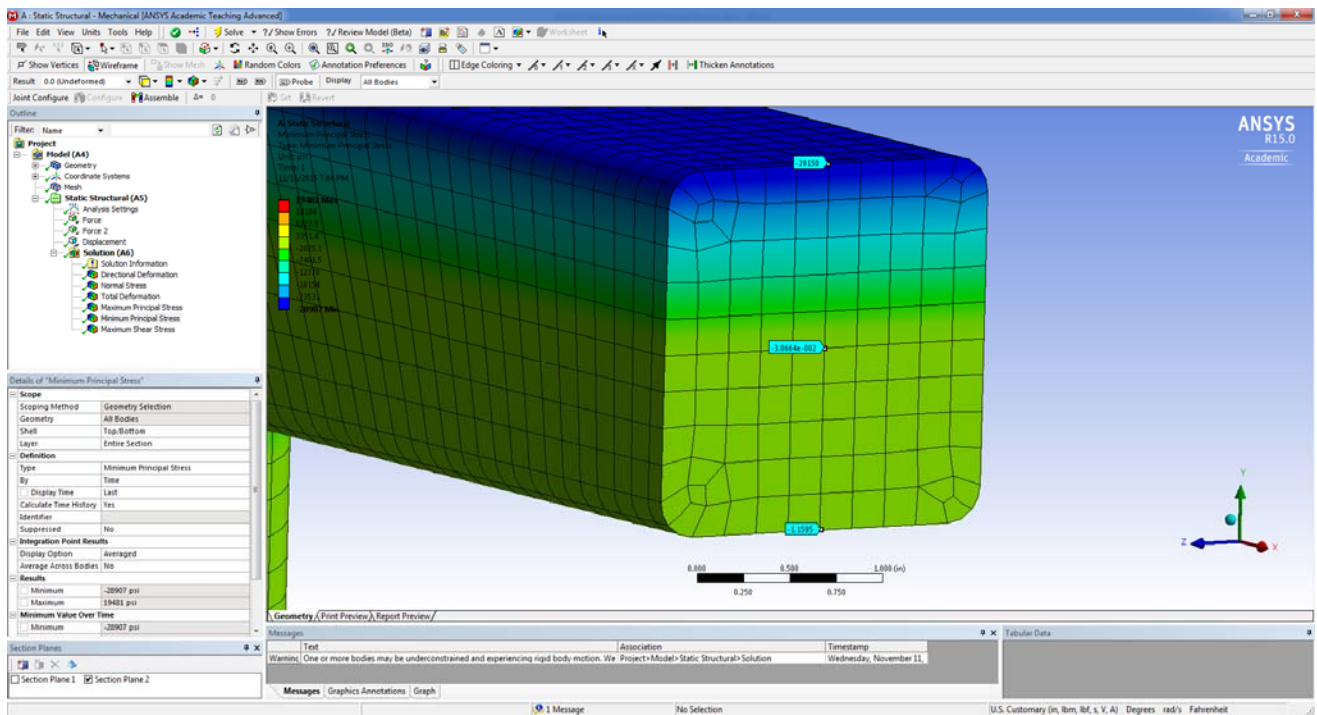
Change to the **Normal Stress** Viewer you created and viewed as part of Tutorial 4. Change to the **Wireframe** option on the top toolbar to view the mesh on the cross-section. This will help in selecting points using *Probe*. Use *Probe* to find three normal stress values on this cross-section: one at the top, one mid-depth, and one at the bottom. You can use these normal stress values, which are in the x-axis direction, to calculate by hand the principal stresses, σ_1 and σ_2 , and the maximum shear stress, τ_{max} , for each 2D state of x-y stress using Mohr's Circle or the stress transformation equations. There is no shear stress acting on the section plane cross-section so each 2D x-y stress element only has a normal (tensile or compressive) stress in the x-axis direction.



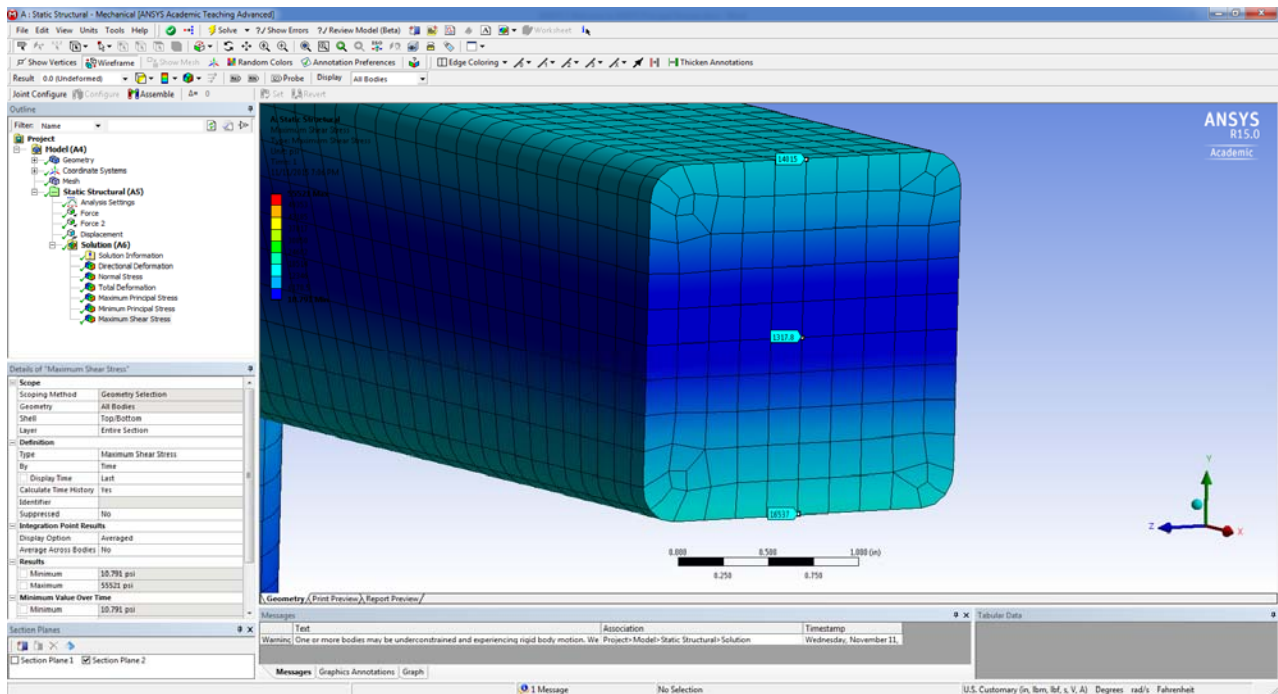
View the **Maximum Principal Stress** results for the Section Plane cut through the eccentric portion of the bar. Use *Probe* to determine the maximum principal stress, σ_1 , for approximately the same three points (A, B, and C) on the cross-section at this section plane cut. Check these ANSYS values against the values obtained by hand using Mohr's Circle for the maximum principal stress, σ_1 . The ANSYS results should be pretty close to those shown below in the view.



Now view the **Minimum Principal Stress** results. Use *Probe* to determine whether the three values of minimum principal stress, σ_2 , for each of the three points (A, B, and C) are also very close to those obtained by hand using Mohr's Circle.



Now view the **Maximum Shear Stress** results. Use *Probe* to determine that the three values of maximum shear, τ_{max} , at Points A, B, and C are very close to those obtained by hand using Mohr's Circle.



In Engineering Design, the results that we seek from ANSYS are the Maximum Principal Stress, σ_1 , the Minimum Principal Stress, σ_2 , and the Maximum Shear Stress, τ_{\max} . We prefer these results over the standard Normal Stress in the x-axis direction, Normal Stress in the y-axis direction, Shear Stress in the XY Plane, Shear Stress in the XZ plane, etc. Most Failure Theories use σ_1 , σ_2 , and/or τ_{\max} , so being able to obtain these results directly from the ANSYS is something you want to get good at.

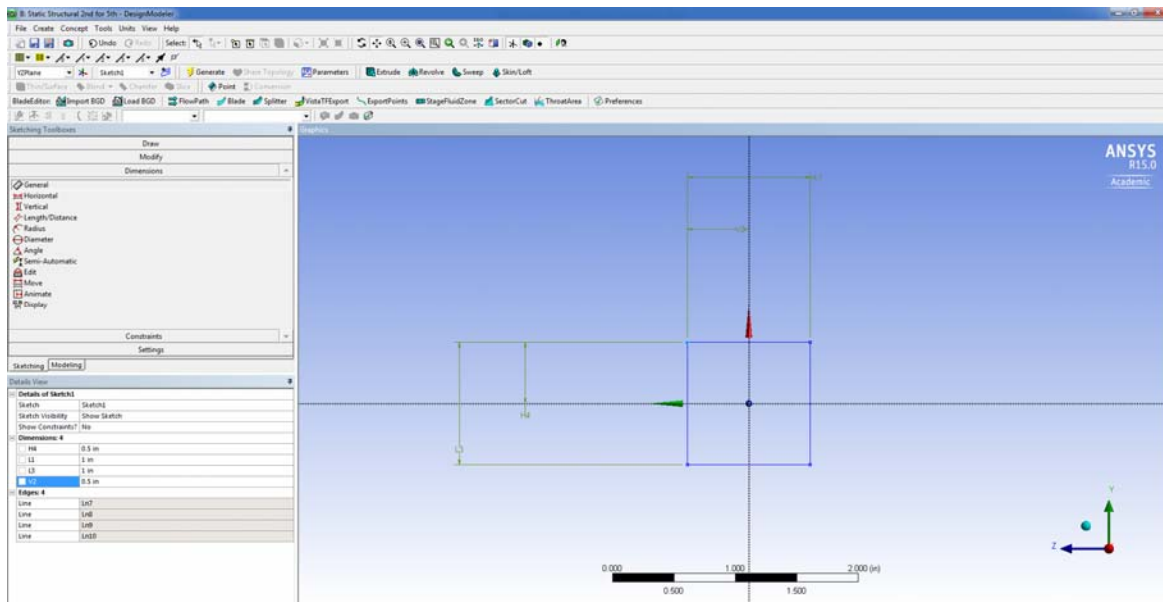
The 2nd PART of TUTORIAL – Combined Bending/Shear Effects

Extension to a 3D Bar Bending Problem – Create a NEW ANSYS Project and Bar Model

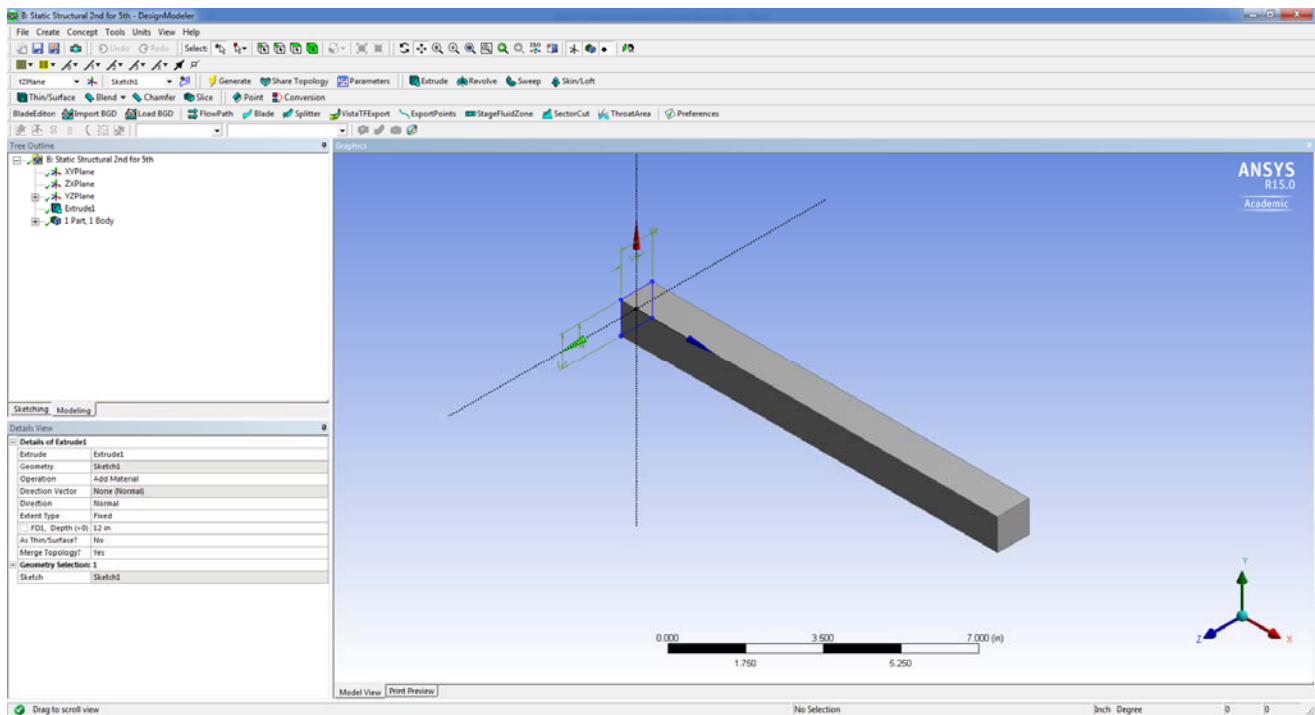
The previous eccentric bar only involved 2D x-y stress elements subject to a normal stress, σ_x . There were no τ_{xy} shear stresses acting on the 2D stress elements at any of the points examined. To consider how to find the principal stresses and maximum shear stress in ANSYS from more complex stress elements, you will create a new 3D rectangular bar model that is subjected to an 800 lb force at one end. The bar with a 1” x 1” cross-section is fixed at its other end. The material is ‘Lafayette Steel’.

Initial Project Space Setup

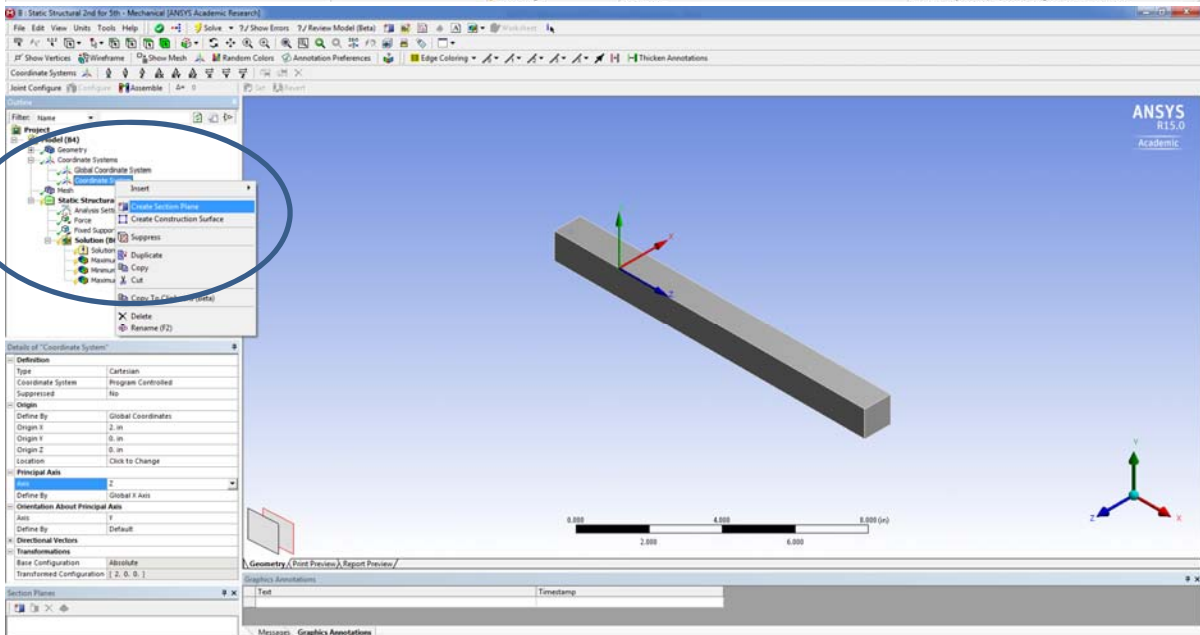
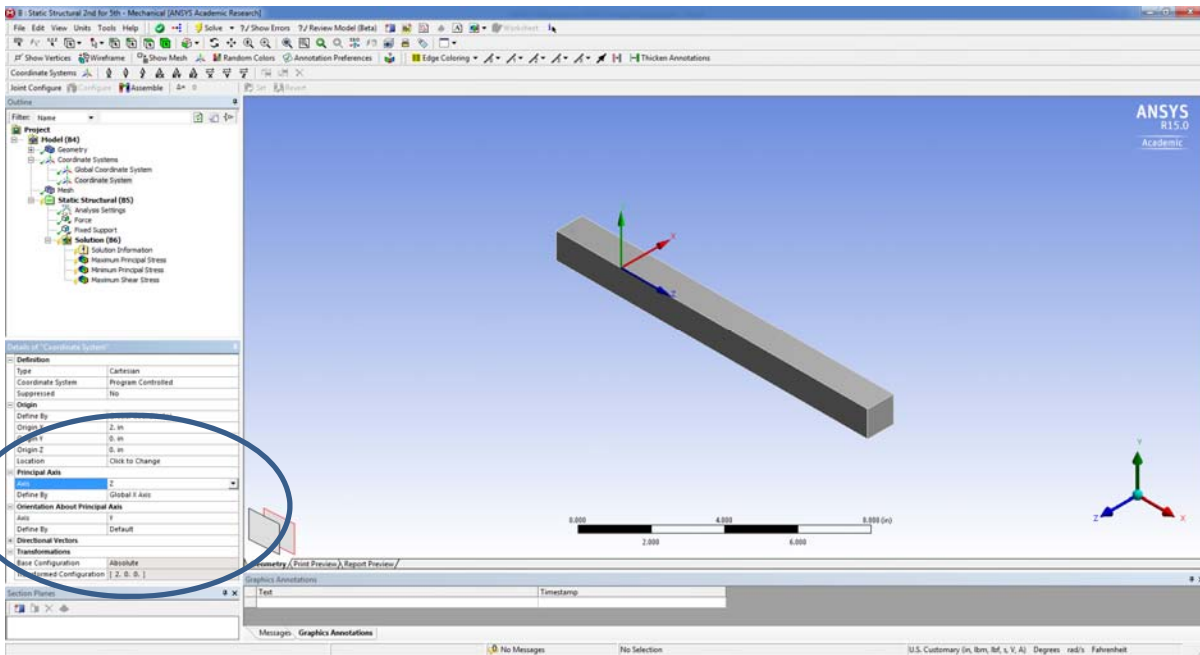
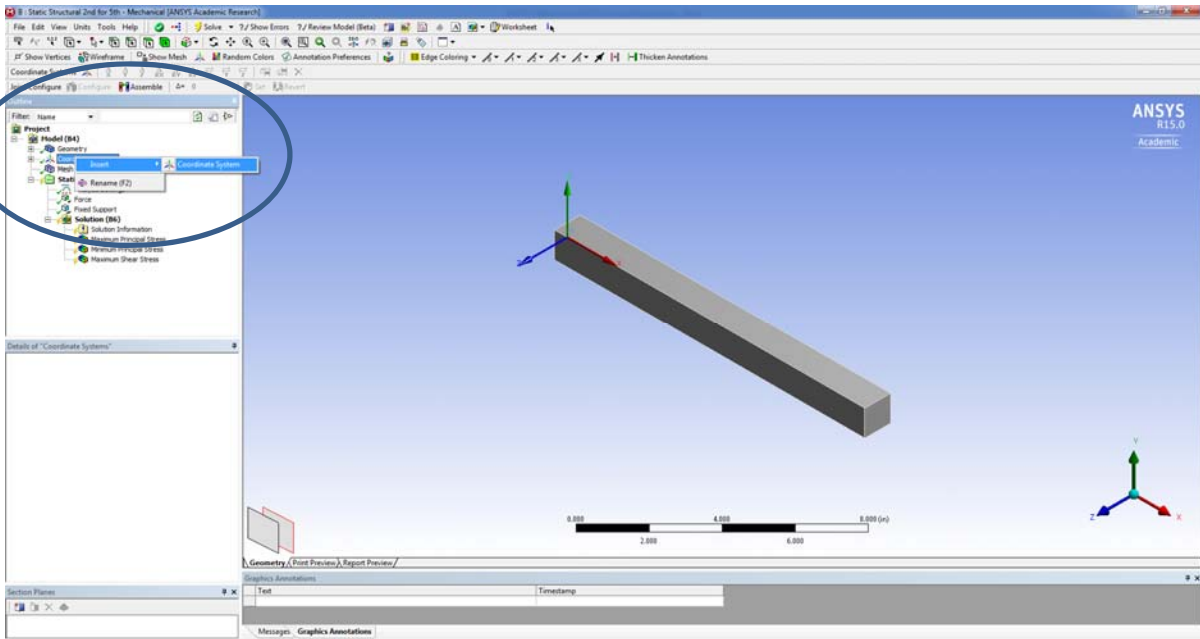
Set up a **Static Structural** project in a new **ANSYS** environment. Change the material by *Importing* in ‘Lafayette Steel’ and set it as the *Default Material* for the model, set the *Units* to US Customary and start up the *Sketching* environment with the *Units* in Inches and with the *Auto Constraints* turned On. Sketch out a rectangle on the **YZ Plane**. Make sure the *Units* are in Inches. Add two dimension lines to set the width and height of the rectangle to be 1 inch each and also add the two dimension lines that help center the square cross-section on the YZ axes.



The 3D bar model consists of solid bar with a length of 12". Use **Extrude** and then **Generate** to create the 3D bar part.



Now open the **Mechanical** window. Change the **Mesh** to have a *Element Size* of 0.1 in and then right-click on **Mesh** to *Generate Mesh*. Add a load of 800 lbs in the $-y$ -axis direction to the near end and add a fixed support to the far end of the beam. At this point you should have a 3D bar problem ready to analyze. Add the *Maximum Principal Stress*, *Minimum Principal Stress*, and *Maximum Shear Stress Viewers*. Before continuing you should create a new Coordinate System that is located 2 inches from the support in the x -axis direction with the Z axis specified as the plane cut using the next three steps. This is the same process you used in Tutorial 4.

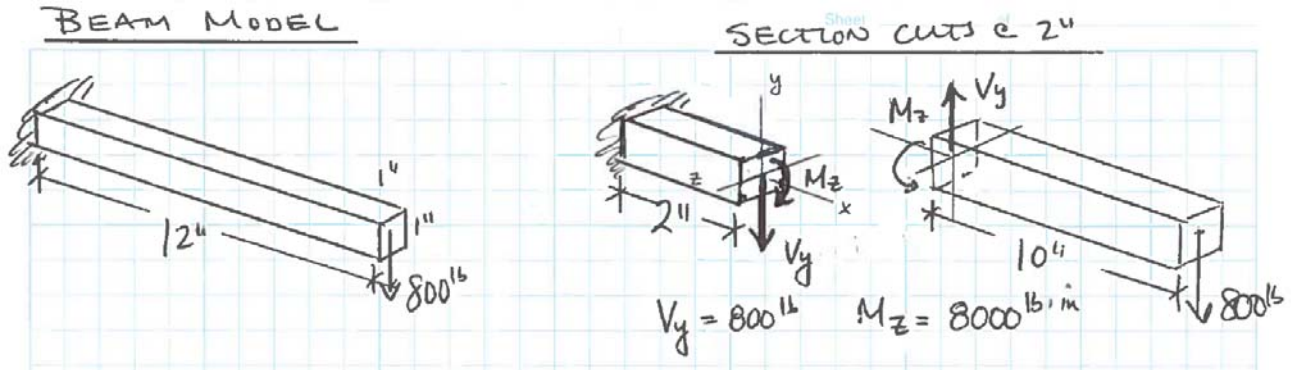


Solve the model and View the **Maximum Principal Stress** results. If you get an *Assignment error* at this point, you need to assign 'Lafayette Steel' as the material for the model either as the default in Property Data or to the Bar element in the **Mechanical** window.

Hand Calcs to determine the Internal Force and Moment at a section cut on the bar

This 3D bar is subjected to a shear force at its free end that creates a shear force in the negative Y axis direction and a negative bending moment about the Z-axis at the section cut location taken 2" from the support. This section cut location and the five specific points (A, B, C, D, and E) defined are shown in the figure provided below.

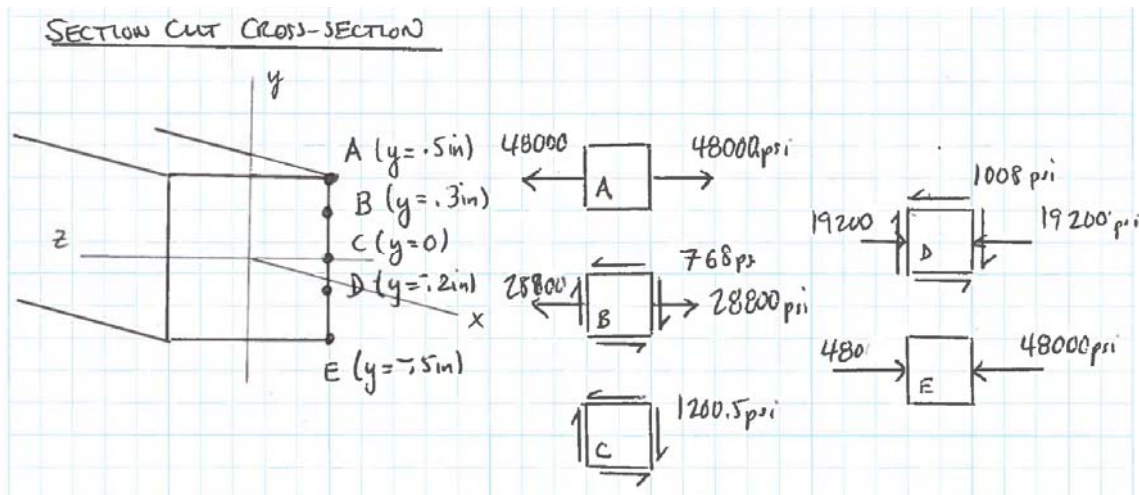
Figure showing beam section cut and internal forces and moments acting on the face of the section cut



Hand Calcs to determine the 2D x-y Stress Elements for the 3D Bar Model

Determine the normal stress due to bending and the shear stress due to the shear force acting on each point and show these on a 2D x-y stress element at Point A, B, C, D and E. Compare your hand stress calculations with the 2D stress elements shown below. For the cross-section, $I_z = 0.8333 \text{ in}^4$, $Q_{NA@C} = .125 \text{ in}^3$ and $t = 1 \text{ inch}$. You will need to consider different Qs for Point B and D in order to calculate the shear stress at those levels in the cross-section. In your bending stress calcs you will also need to consider the depth location (y) at each point location.

Figures showing the five 2D x-y Stress Elements in the 3D Bar with Moment and Shear Force



Hand Calcs to find Principal Stresses and Maximum Shear Stress

On your own, use Mohr's Circle or the stress transformation equations to find the Principal Stresses (σ_1 and σ_2) and the Maximum Shear Stress (τ_{max}) for each of the 2D x-y stress elements at Point A, B, C, D, and E in the beam cross-section. You can check the values you get for the principal stresses and for the maximum shear stress using the MecMovies Stress Transformation App M12.19.

Maximum Principal Stress

View the **Maximum Principal Stress** results for the Section Plane cut through the 3D bar at the 2" location. Change to the **Wireframe** option on the top Toolbar to view the mesh points and to help select Probe points. Use *Probe* to determine the maximum principal stress, σ_1 , for approximately the same five points (A, B, C, D, and E) on the cross-section at this section plane cut. These are the same point locations as run in the hand calculations. Check these ANSYS values against the values obtained from Mohr's Circle for the maximum principal stress, σ_1 . These should be pretty close as shown below.

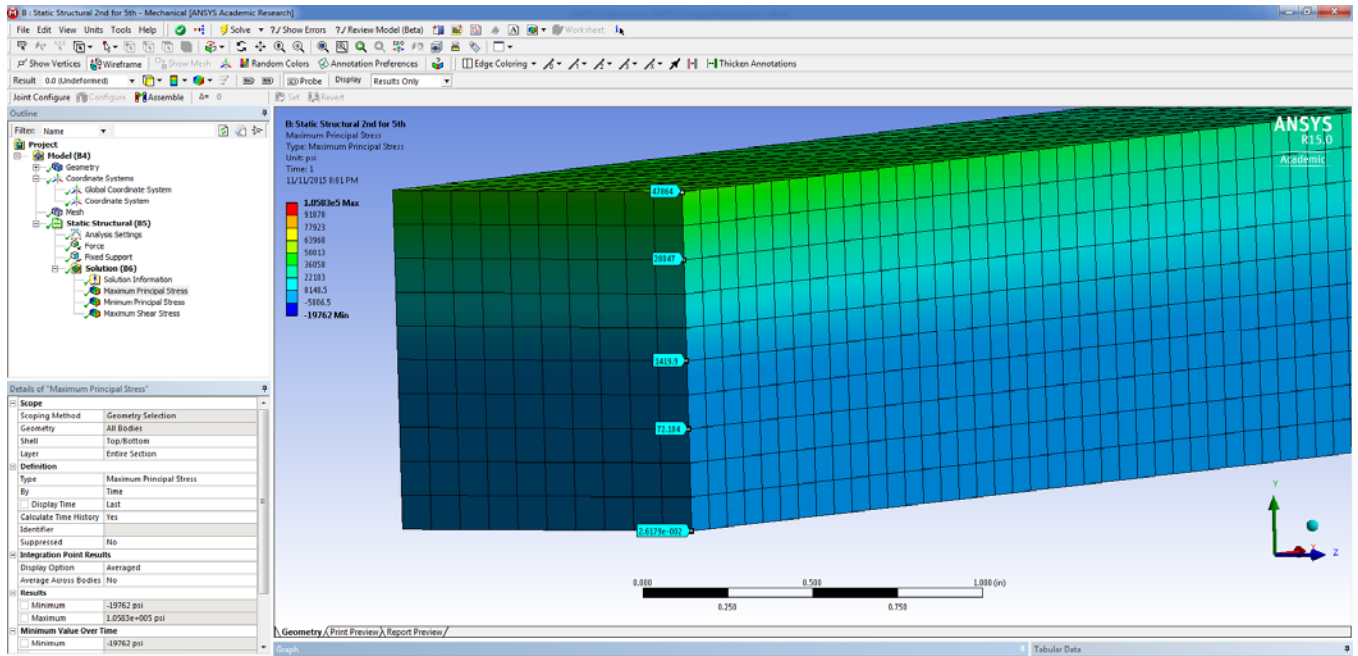
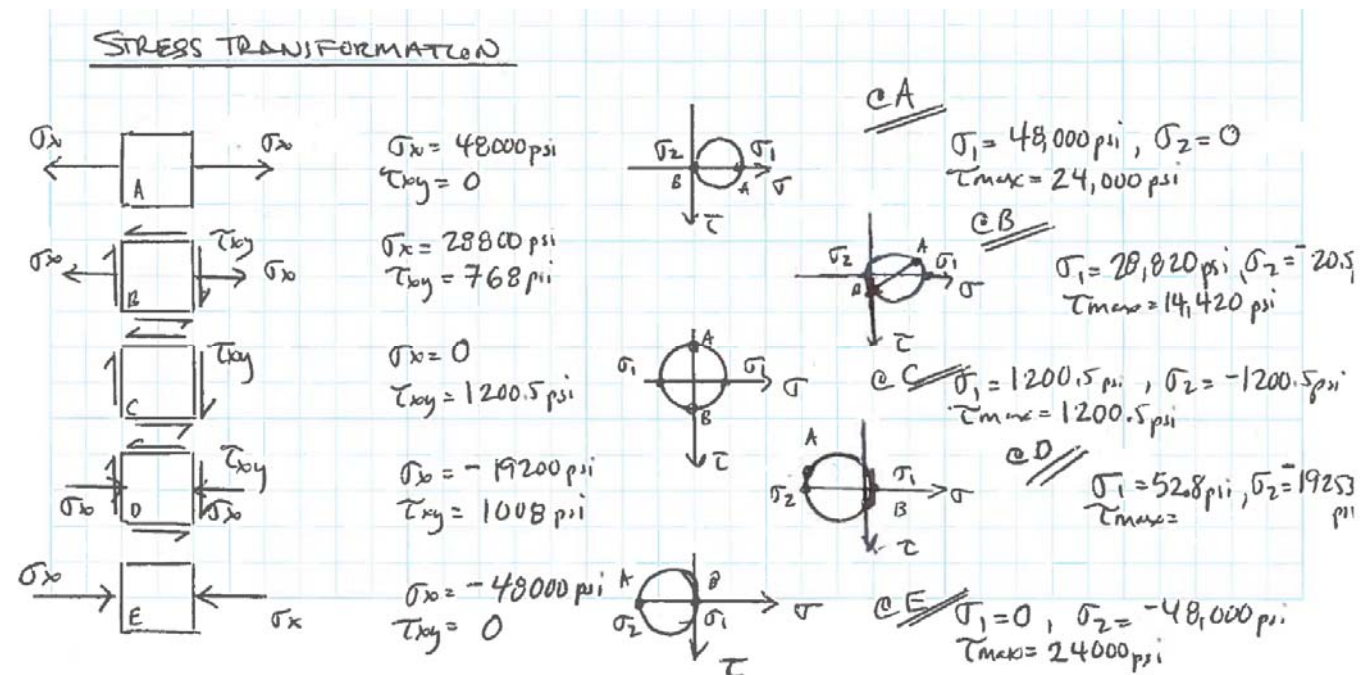
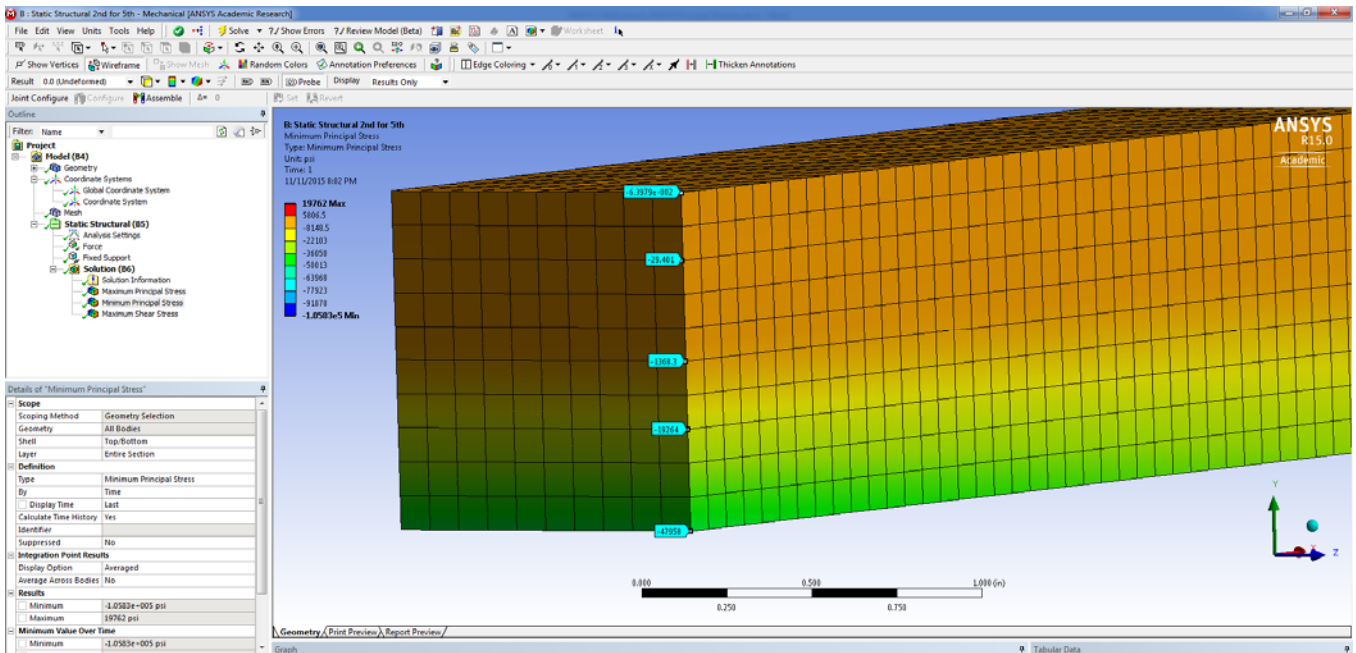


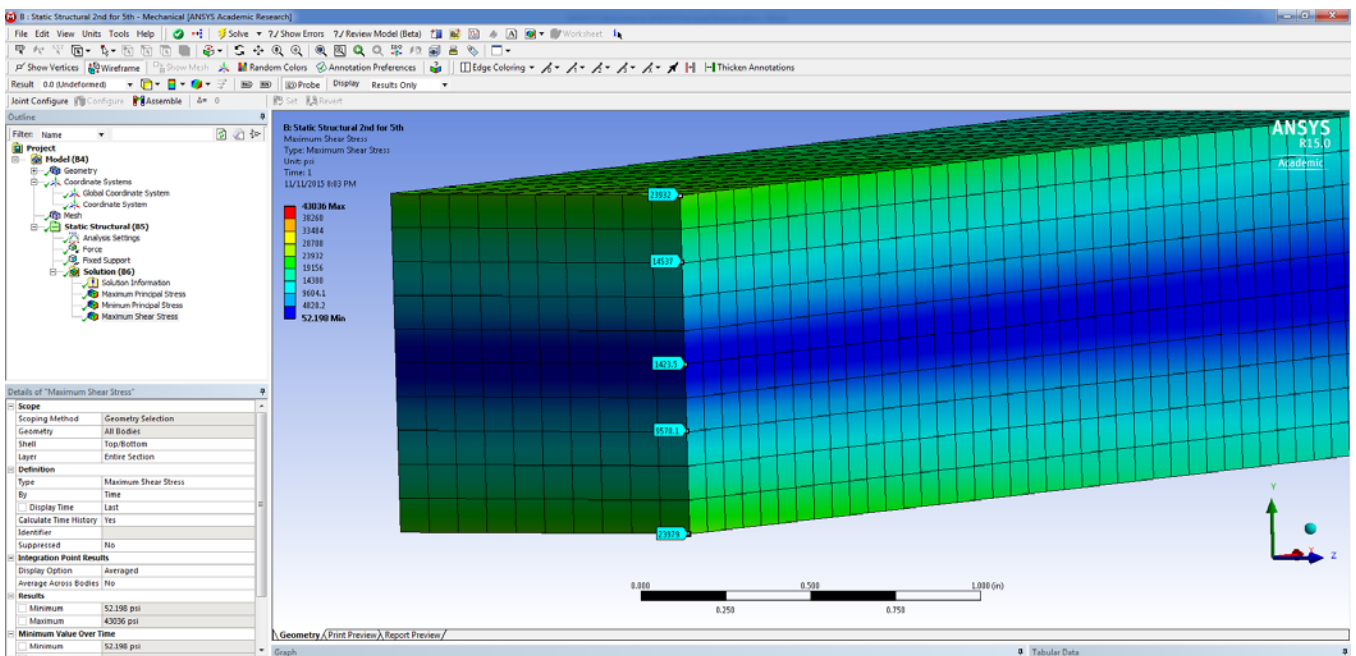
Figure showing the Results of Transforming the five 2D x-y Stress Elements Using Mohr's Circle



Now view the **Minimum Principal Stress** results. Use *Probe* to determine the three values of minimum principal stress, σ_2 , for each of the five points (A, B, C, D and E). Check that these values are also very close to those obtained using Mohr's Circle.



Now view the **Maximum Shear Stress** results. Use *Probe* to determine the five values of maximum shear, τ_{max} , at Points A, B, C, D and E. Check that these are very close to those obtained using Mohr's Circle.



If you are getting values of ANSY that match well with those from Mohr's Circle you are in good shape to move onto ANSYS Homework 5. In the future you will learn how to set up specific points in the mesh to get specific results for the selected points to make a closer comparison, but setting up the selection/results is a real pain! So for now we will rely on the somewhat careful selection of Probe values to help us compare the results between ANSYS and hand calculations.

Save your file and work and fill out the Online Survey using the link provided by email for Tutorial 5.