
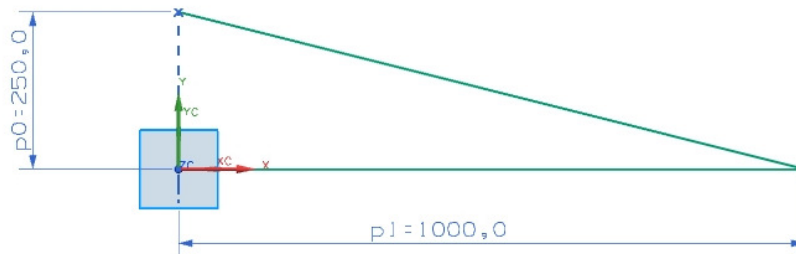


## Lab Objectives

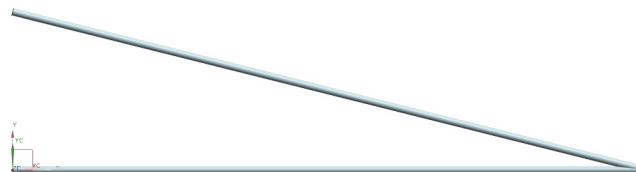
- Become familiar with Siemens NX finite element analysis using the NX Nastran solver.
- Perform deflection and stress analyses of planar truss structures.
- Use modeling and FEA tools to input the structure, perform the analysis and visualize the results.
- Check the FEA results with hand calculations.

## Lab Tasks

1. Start the NX CAD software. From the start menu, select “Start” → “All Programs” → “UGS NX 8.0” → “NX 8.0” or double click the  icon on the desktop.
2. Start a new CAD model by pressing the “New” button. In the “File New” dialog box make sure the “Model” tab is selected. Select the first row in the table (“Model”), input the correct filename and select the correct folder for the new model file. Then press “OK”.
3. Create a CAD model similar to the one shown in Figure 1. Keep solid and sketch objects on different layers. If a different design is created, be aware that each line represents one member. Be sure that all lines end at intended joint locations. Also be sure that the members of the structure form triangles, otherwise the structure, whose joints are assumed to be spherical joints in this FEA, will become a mechanism with some members able to move kinematically.



(a)



(b)

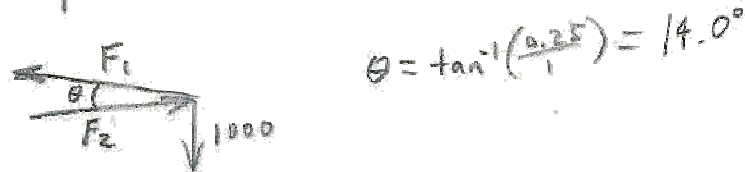
Figure 1. CAD Model of Truss Structure (a) sketch (b) solid (optional)

MAE 456 FINITE ELEMENT ANALYSIS  
LAB INSTRUCTIONS  
Tutorial 2

4. Perform rough hand calculations to anticipate the directions and order of magnitudes of the reaction forces and stresses that should be expected if the structure was mounted to a wall on the left and a downwards force of 1000 N was applied on the right and the members had a cross-section of 100 mm<sup>2</sup>. See Figure 2.



At point C



$$\theta = \tan^{-1}\left(\frac{0.25}{1}\right) = 14.0^\circ$$

$$\sum F_y = 0$$

$$F_1 \sin(14.0) - 1000 = 0$$

$$\boxed{F_1 = 4134 \text{ N}}$$

$$\sum F_x = 0$$

$$-F_1 \cos(14.0) + F_2 = 0$$

$$-(4134) \cos(14.0) + F_2 = 0$$

$$\boxed{F_2 = 4011 \text{ N}}$$

$$\sigma_1 = \frac{F_1}{A_1} = \frac{4134}{100 \times 10^{-6}} = \boxed{41 \times 10^6 \text{ Pa}}$$

$$\sigma_2 = \frac{F_2}{A_2} = \frac{4011}{100 \times 10^{-6}} = \boxed{40 \times 10^6 \text{ Pa}}$$

Figure 2. Hand Calculations

MAE 456 FINITE ELEMENT ANALYSIS  
LAB INSTRUCTIONS  
Tutorial 2

5. In NX, select “File” → “New” to start a new finite element model:
  - a) In the “File New” dialog box select the “Simulation” tab.
  - b) Select the first row (“NX Nastran FEM”) in the table.
  - c) Enter the desired filename and folder, and press “OK”.
  - d) In the “New FEM” dialog box make sure that “Associate to part” is checked.
  - e) If a solid shape was created in Step 3, make sure that “Create idealized part” is also checked.
  - f) Click on “Geometry Options...,” make sure that “Sketch Curves” is checked, and press “OK”.
  - g) Press “OK” for the “New FEM” dialog.
6. If the idealized part was created in the previous step, use the “Window” menu to switch to the CAD model for the idealized part (e.g., fem1\_i.prt). The idealized CAD model is a simplified CAD model of the actual part. Use “Wave geometry linker” to copy the solid or just the sketch from the CAD model from Step 3 into the idealized CAD model. If the entire solid was copied, then a new 3D line should be created to represent the centerlines of the truss members. In the end, it should look similar to the original sketch, as shown in Figure 2.

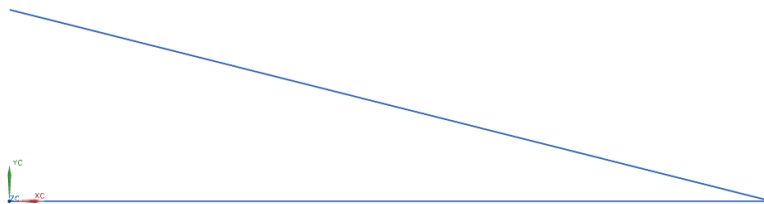


Figure 2. Idealization of the Solid Model

7. A mesh of rod elements will be created in the fem file. If necessary, first use the “Window” menu to switch to the fem file (e.g., fem1.fem). Select “1-D mesh” from the meshing drop-down buttons (see Figure 3). In the element “Type” selection drop-down select “CROD”. Select both lines. Make sure that “Default Element” is set to “Number” and the value is set to 1. Make sure that “Merge Nodes” is enabled. Press “OK”.

MAE 456 FINITE ELEMENT ANALYSIS  
LAB INSTRUCTIONS  
Tutorial 2


8. Assign material and cross-sectional properties to the elements.
  - a) Expand the “1 D Collectors” folder in the “Simulation Navigator” in order to see the collections of 1-D meshes. Expand the “Crod Collector(1)” to see the list of meshes (there should only be one).
  - b) Double-click on “Crod Collector(1)” to edit the properties for the meshes under the collector.
  - c) In the “Mesh Collector” dialog press the “Modify Selected” button ()
  - d) In the “Prod” dialog press the “Choose material” button. Select “Aluminum\_2014” and press “OK”. Note that you cannot inherit a material from a curve in a CAD model.
  - e) Enter a cross-sectional area of 100 mm<sup>2</sup> in the “Area” box of the “Prod” dialog. Enter 1667 mm<sup>4</sup> for the “Torsional Constant”(J) and 7.07 mm for the “Torsional Stress Coefficient” (c). ( $\tau = cT/J$  where  $\tau$  is the computed shear stress and T is the torque on the rod.)
  - f) Press “OK” to accept the new property values.
  - g) Press “OK” to accept the changes to the Mesh Collector.



Figure 3. Element Drop-down Buttons.

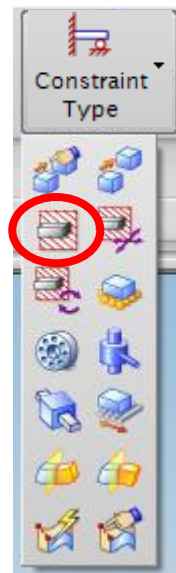


Figure 4. Constraint Type Drop-down Buttons.



Figure 5. Load Type Drop-down Buttons.

MAE 456 FINITE ELEMENT ANALYSIS  
LAB INSTRUCTIONS  
Tutorial 2

9. Create a new simulation.

- a) Right-click on the finite element model (“fem1.fem”) in the Simulation Navigator and select “New Simulation...” in the drop-down menu.
- b) In the “New Part File” dialog, select the first row (“NX Nastran Sim”), type in a different file name and select a different directory if desired, and press “OK”.
- c) In the “New Simulation” dialog, make sure that the finite element model to which the simulation will apply is correct (“fem1”) and press “OK”.
- d) In the “Create Solution” dialog, verify that “NX Nastran” is the solver, “Structural” is the analysis type, and “SESTATIC 101 – Single Constraint” is the solution type before pressing “OK”.

10. Create restraints.

- a) Click on the “Constraint Type” button on the toolbar and select the “Fixed Constraint” shown circled in Figure 4.
- b) With the “Fixed Constraint” dialog showing, select the nodes at the left ends of the two rod elements (see Figure 6.). The nodes will be difficult to pick because it will try to pick the elements. When this happens set the selection filter to “Node” or hover the mouse over the node, press the right mouse button, select “Select from List...”, and select in the drop down menu. It will show a list of things you can pick. Select the node from this list.
- c) Press “OK” in the “Fixed Constraint” dialog.

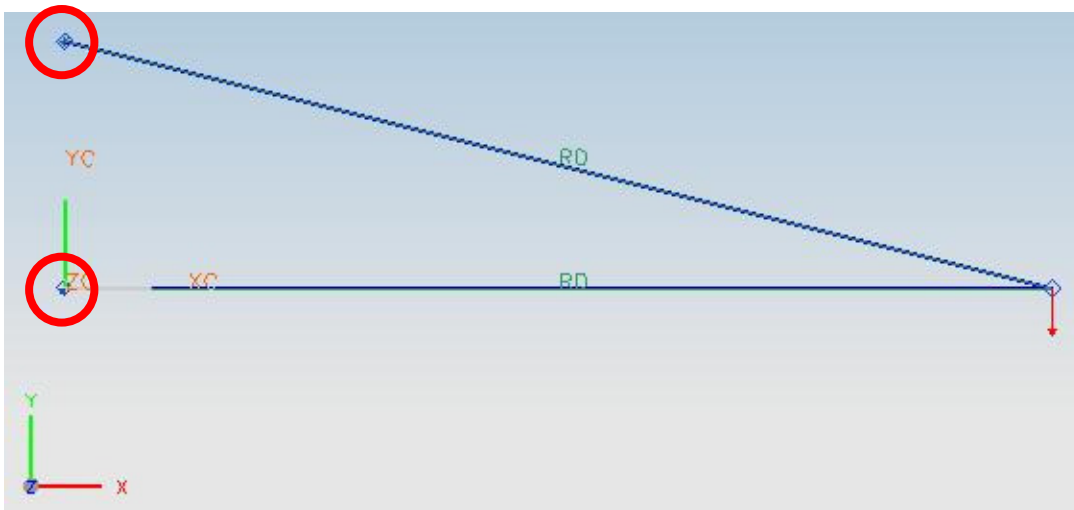



Figure 6. Two Rod Elements with Restraints on Left and Load on Right.

MAE 456 FINITE ELEMENT ANALYSIS  
LAB INSTRUCTIONS  
Tutorial 2

11. Create loads.

- a) Click on the “Load Type” button on the toolbar and select the “Force” shown circled in Figure 5.
- b) With the “Force” dialog showing, select the node at the right end of the two rod elements (see Figure 6.). Again, set the filter to “Node” or use “Select from List...” if the node is difficult to pick.
- c) Type in 1000 N for the force magnitude.
- d) To set the direction of the force, press the “Inferred Vector” drop-down and select the  $-y$  direction.
- e) Press “OK” in the “Force” dialog.

12. Solve the Finite Element problem.

- a) In the Simulation Navigator right-click on “Solution 1” and select “Model Setup Check” in the drop-down menu. A text window will open up indicating whether there are any errors in the setup of the problem. If there are errors or warnings, go back to previous steps to resolve the error. Otherwise just close the window.
- b) In the Simulation Navigator right-click on “Solution 1” and select “Solve...” in the drop-down menu.
- c) In the “Solve” dialog click, turn on the calculation of beam forces. (Normally stresses, strains, deflections, and reaction forces are computed, but not beam forces.) Click on “Edit Solution Attributes.” Then select the “Case Control” tab and click on the “Modify Selected” button () next to “Output Requests.” In the “Structural Output Requests” dialog box, click on the “Force” tab and click on “Enable FORCE Request” so that it is turned on. Then click on “OK” to close the “Structural Output Requests” and “OK” again to close the “Solution” dialog box. Click “OK” in the “Solve” dialog box to start the solving.
- d) Wait for the solution to be calculated. After it is finished, the *Results* item should appear at the bottom of the Solution Navigator as shown in Figure 7. All of the windows that popped up can then be closed.

13. View the results.

- a) In the Simulation Navigator double-click on the results item to bring up the “Post-Processing Navigator” shown in Figure 8. If this gives an error, it is likely that your structure is a mechanism. To test this try fixing nodes with fix constraints to see if the problem goes away.

# MAE 456 FINITE ELEMENT ANALYSIS LAB INSTRUCTIONS Tutorial 2

- b) Expand “Solution 1” and double-click on “Displacement – Nodal” to show the results as shown in Figure 9. The areas in red have high displacements while the areas shown blue have small or no displacements.
- c) To show the undeformed shape at the same time as the deformed results, double-click on “Post View 1” in the “Post-Processing Navigator.” In the “Post View” dialog, shown in Figure 10, check “Show undeformed model”.
- d) To view the stresses, double-click on “Stress – Element-Nodal” in the “Post-Processing Navigator”. The stresses in the two elements should appear as two solid colors as shown in Figure 11. If they do not, expand “Stress – Element-Nodal” and double-click on “Von-Mises”.

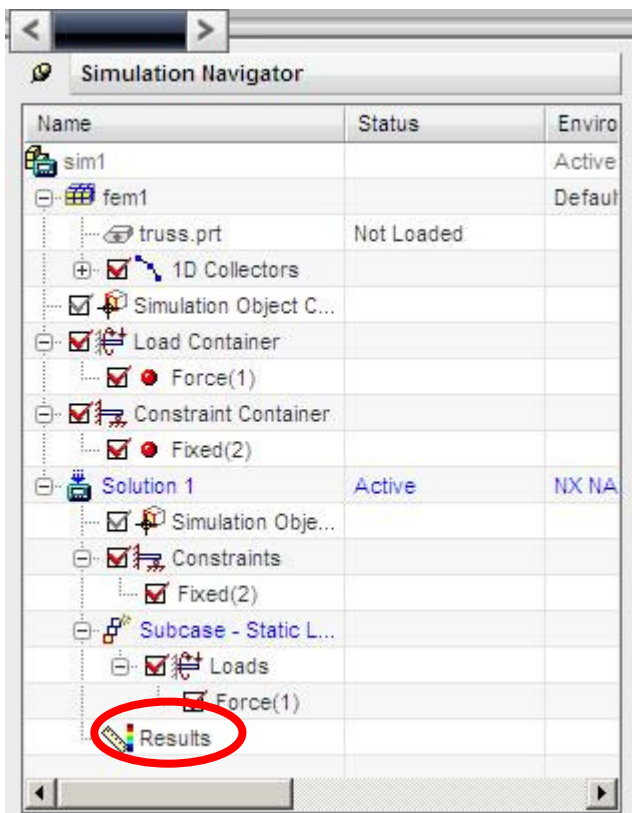


Figure 7. Solution Navigator with Results Computed.

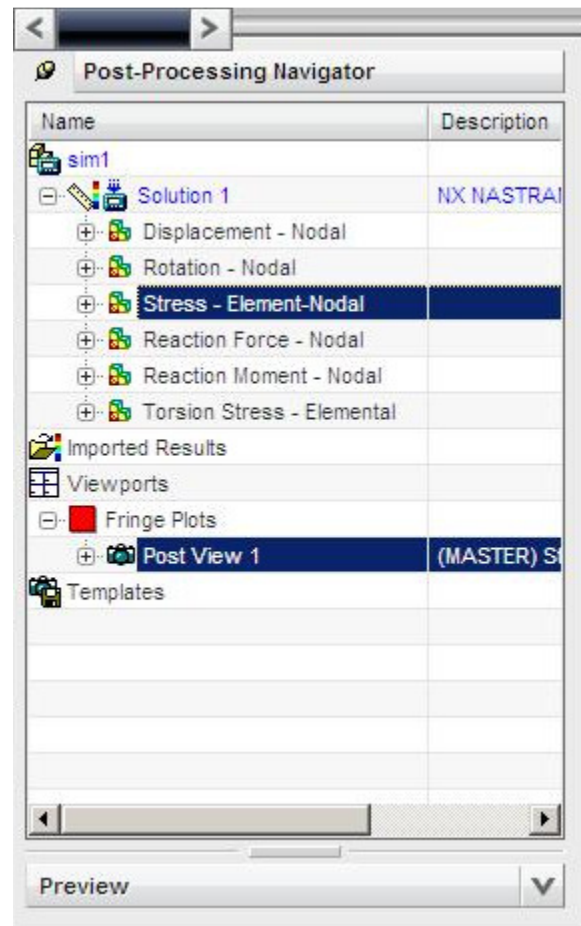


Figure 8. Post-Processing Navigator.

MAE 456 FINITE ELEMENT ANALYSIS  
LAB INSTRUCTIONS  
Tutorial 2

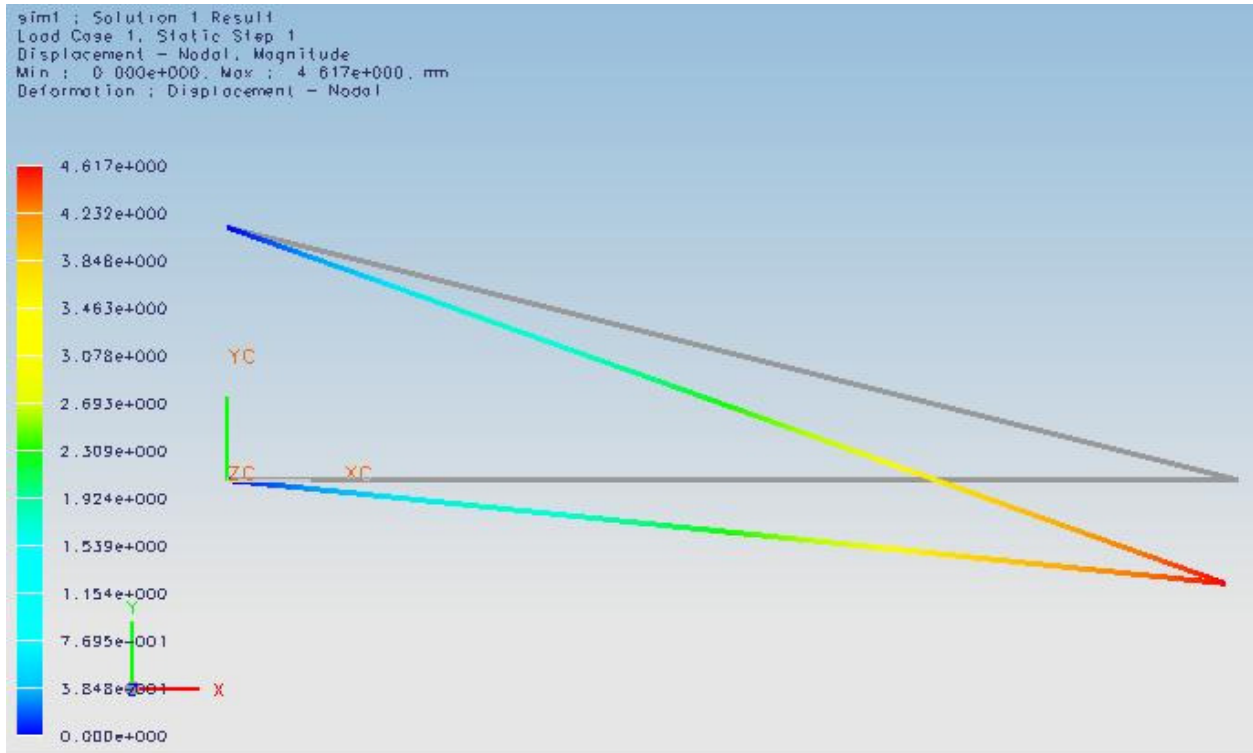


Figure 9. Displacement Results

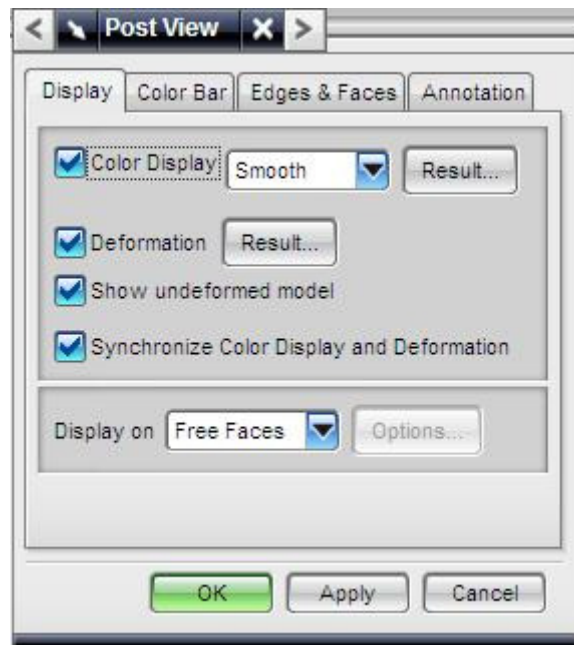


Figure 10. Post View Dialog



# MAE 456 FINITE ELEMENT ANALYSIS LAB INSTRUCTIONS Tutorial 2

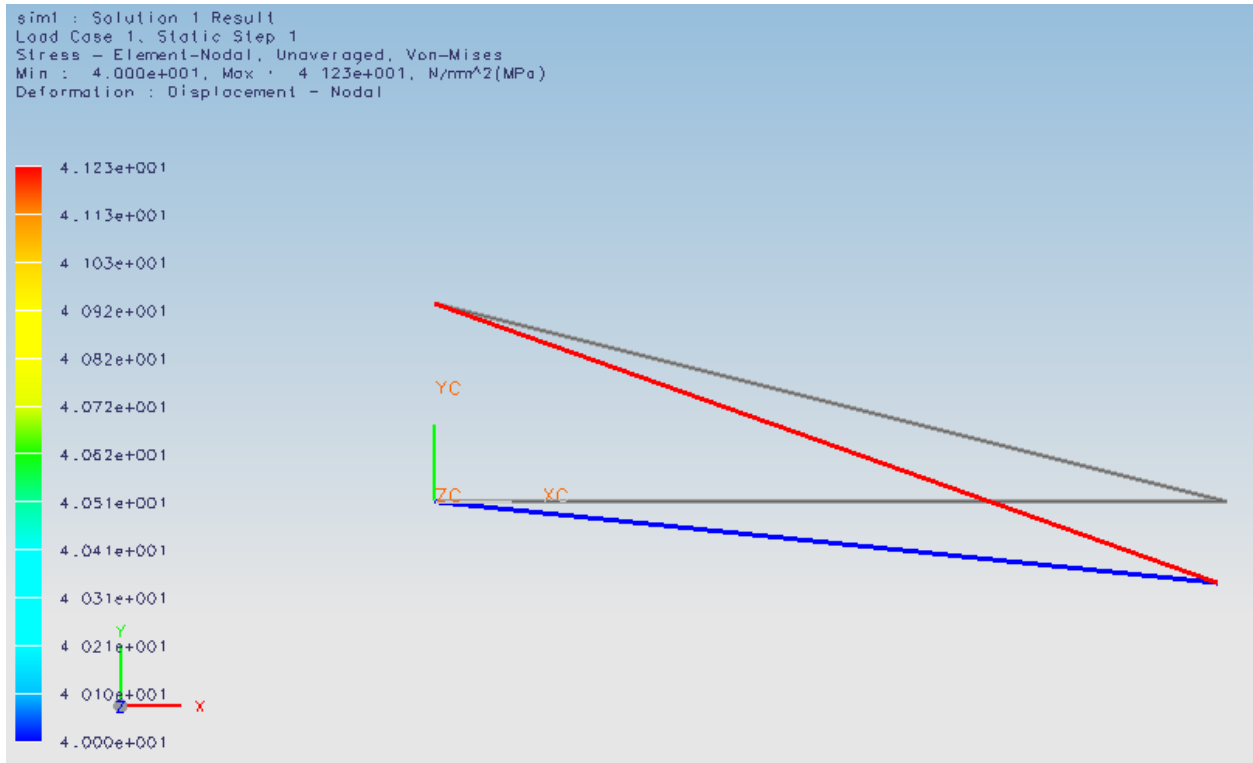


Figure 11. Stress Output

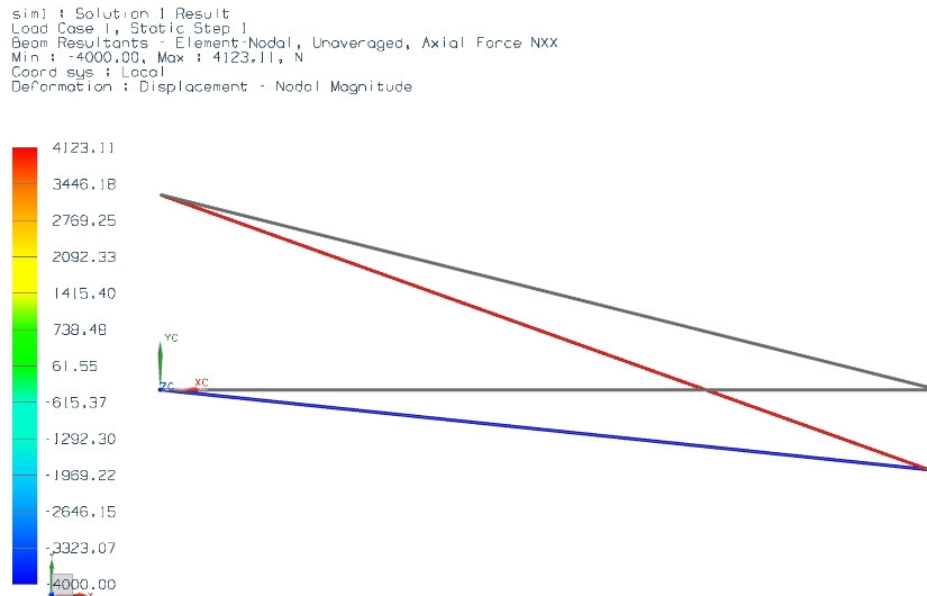



Figure 12. Force Output

MAE 456 FINITE ELEMENT ANALYSIS  
LAB INSTRUCTIONS  
Tutorial 2

14. Examine the results to verify that the data was entered correctly and that the calculations were performed correctly.
- a) Looking at the restraint locations, we can see that there were no deflections of the structure at the restraints. If there were deflections we would need to re-examine our input for the restraints.
  - b) Looking at the deflections near the applied forces, we see that the deflection is downwards. This implies that the force was entered on the correct node and the direction of the force is correct.
  - c) Compare the FEA results with the rough hand calculations from Figure 2:
    - a. In the hand calculations, the force in the top bar should be 4134 N in tension and the force in the bottom bar should be 4011 N in compression. This corresponds closely with the results shown in Figure 12, which show values of 4123 N for the top bar and 4000 N for the bottom bar. Note that the FEA software displays tension as positive and compression as negative values.
    - b. In the hand calculations, the stress in the top bar should be 41 MPa and the stress in the bottom bar should be 40 MPa. This corresponds exactly with the results shown in Figure 11 (which are in MPa).
    - c. Another possible check is to look at the reaction forces. Figure 13 shows the magnitudes of the reaction forces from the FEA. Note, however, that the software interpolates the reaction forces between the nodes of the element. This is not physically correct for this situation; however, we can get the force at the node using the “Identify Results” tool. Turn on the display of the “Post-Processing” tool bar and select . Moving your mouse over a node will make the software display the value at the node in the status bar. Clicking on the node will cause the value to be displayed at the node, as shown in Figure 13. Observe that the analytical and FEA reaction forces match very closely.
15. Make some conclusions from the results. In this case, since the highest stress is 41 MPa and this is a factor of 10 less than the 410 MPa yield strength of 2014 Aluminum in the T6 condition, the structure will not fail with these boundary conditions and, in fact, has a safety factor of 10. However, note that this analysis does not consider failure through buckling, which will be considered in Lab Assignment 2.

# MAE 456 FINITE ELEMENT ANALYSIS LAB INSTRUCTIONS Tutorial 2

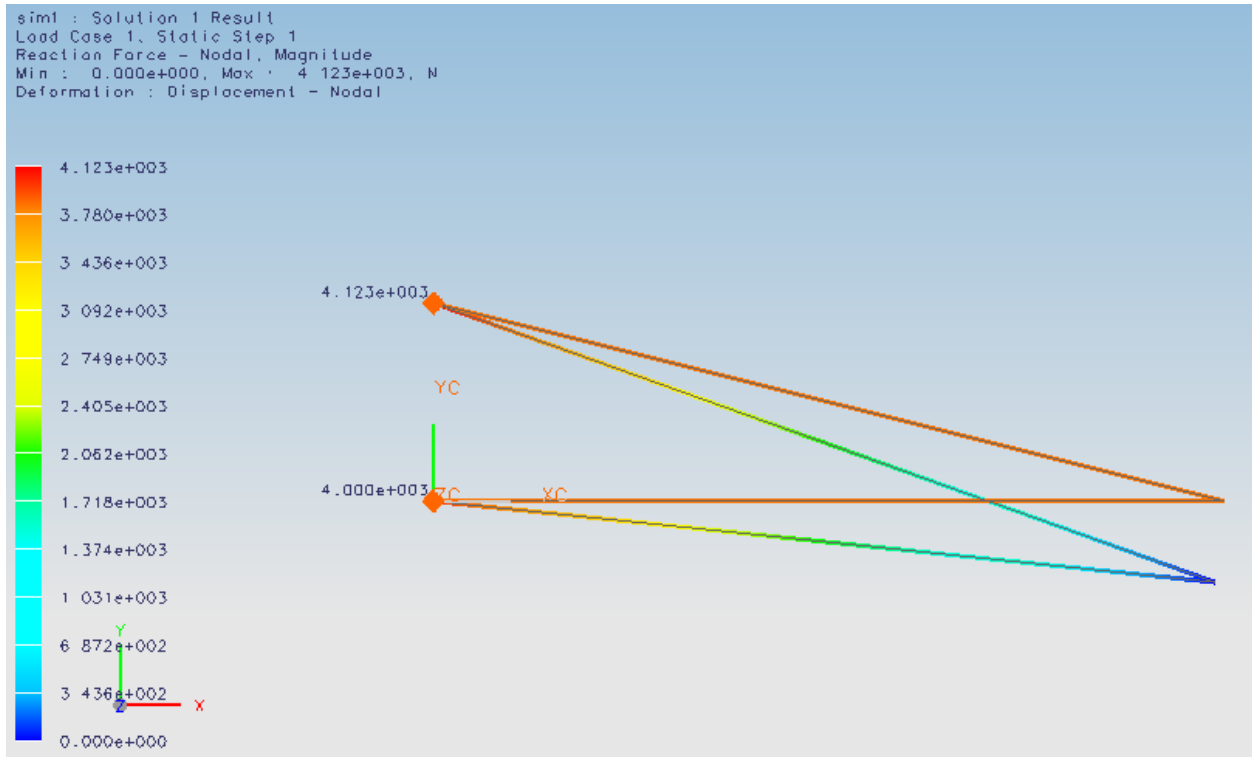


Figure 13. Reaction Forces from FEA