ANSYS® Workbench™ Software

Tutorial with Multimedia CD

RELEASE 12

INSIDE:

Fereydoon Dadkhah
Delphi Electronics & Safety

Jack Zecher, P.E.
Indiana University-Purdue University Indianapolis

SDC publications
Schroff Development Corporation
www.schroff.com
www.sdcpublications.com

6

Introduction to Finite Element Simulation

Historically, finite element modeling tools were only capable of solving the simplest engineering problems which tended to reduce the problem to a manageable size and scope. These early FEA tools could generally solve steady-state, linear problems in two dimensions. The factors that forced these simplifications were lack of efficient computational techniques and the computing power to model more complex real-life problems.

As numerical computation techniques have advanced and computing power has increased, analysis tools have also advanced to solve more complex problems. A real-life engineering problem may involve different physics such as fluid flow, heat transfer, electromagnetism and other factors. The finite element method has been used to solve engineering problems in all of these areas successfully and the goal of most software developers is to include as much of the real-world in the simulation they perform as possible.

However, in many situations, use of simplifying assumptions such as symmetry, axisymmetry, plane stress, plane strain, etc., is still preferable to using a complete three dimensional model because of the efficiency they provide. These assumptions should be used if the problem being solved requires it. In other words, there is no need or justification to perform a full three dimensional analysis if symmetry is present in the problem being solved.

The ANSYS philosophy can be summarized as one that aims to simulate the complete real-life engineering problem. The simulation usually begins by using a three dimensional CAD model to construct a finite element mesh followed by imposing loads and boundary conditions and then computing the solution to the finite element problem.

6.1 Steps required for solving a problem

In performing any finite element analysis you must complete certain tasks which can be thought of as the steps required for completing the analysis. Regardless of what FEA tool is being used, these same tasks must be performed in order to complete the analysis. These tasks are listed below.

1. **Generate the mesh**
2. **Define/Assign material properties**
3. **Define the analysis type**
4. **Set loading and boundary conditions**
5. **Solve**
6. **Review the results**

These steps are the same as those discussed in Section 1.1, Figure 1.3, although in a slightly different order. In the past an FEA practitioner had to explicitly perform each step and ensure the proper flow of data from one step to the next. Today, it is important for those new to FEA to know these tasks in order to better understand how FEA works.

ANSYS Workbench version 12 combines the required steps into a complete Analysis System which appears in the Project Schematic area. The analysis system is part of a project maintained by Workbench. This approach allows Workbench to perform two very important tasks. First, Workbench is able to enforce certain rules such as the sequence of the analysis steps and secondly, Workbench can monitor changes to data at any step of the analysis and update the project to take these changes into account.

Consider the Static Structural analysis system. As was briefly discussed in Chapter 3, the analysis system appears as a number of cells arranged in a column and in order to perform the analysis, we proceed from the top of the column to the bottom – from cell A2 to cell A7 – and complete the tasks required in each cell.

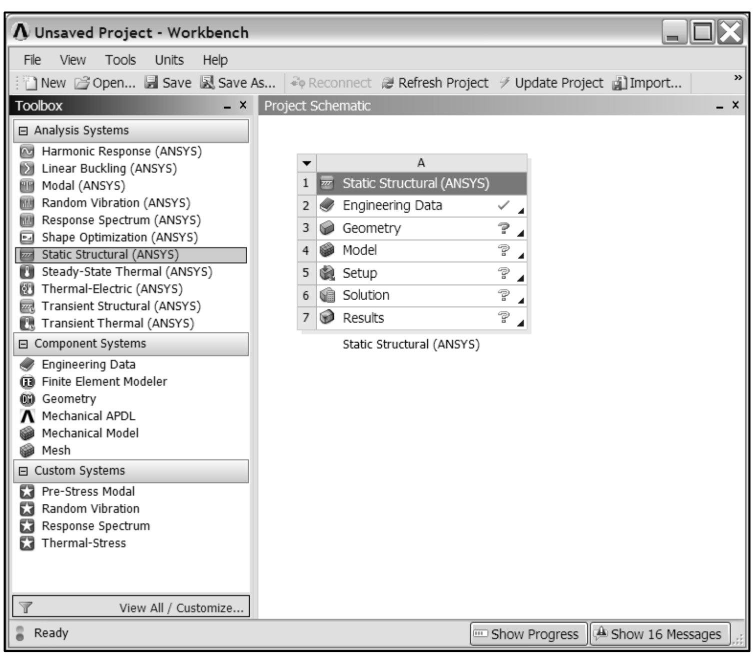


Figure 6.1 Static Structural system

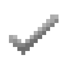
Some important aspects of the Static Structural analysis system are outlined below.

Engineering Data – This cell represents the material property definitions for the analysis. By default Workbench assumes that everything in our model is made of structural steel. Therefore, if the part in our analysis is made of a different material, we must change the material assigned to the part by Workbench. We will do this in a later tutorial.

Geometry – This cell represents the solid model of the part or assembly to be analyzed. The geometry can be imported from another CAD package or generated with DesignModeler.

Model, Setup, Solution and Results – By default, the data for all these cells is handled by the **Mechanical Application** (formerly **Simulation**). Therefore, double-clicking on any of these cells places the focus on the related object on the project tree in the Mechanical Application interface.

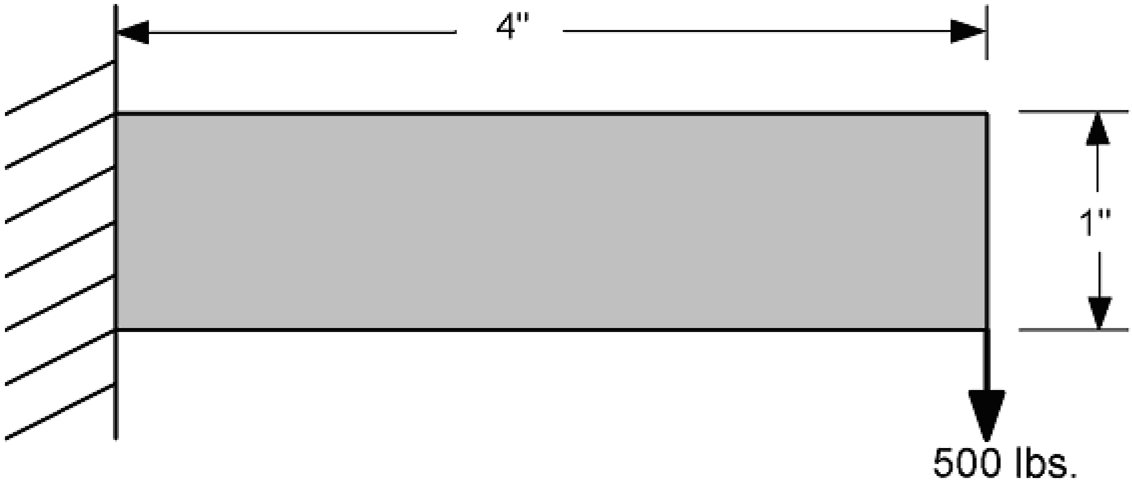
Status of the cells – The icons appearing on the right side of each cell indicate the status of that cell as described in the following table.

Icon	Meaning
	Attention is required. It may mean that no upstream data is available yet, e.g., no geometry has been assigned.
	Unfulfilled. An upstream step has not been completed. You will see the Attention Required icon upstream of the current cell. You must bring the upstream cell up to date before you can proceed.
	Refresh is required. This icon means that upstream data has changed. Choose the context menu to decide what action to take.
	Update is required. This icon means that changes within this cell have happened which require the output from this cell to be regenerated.
	Up to Date. Data from this cell is up-to date and can be provided to downstream cells.

The tutorial that follows provides step-by-step instructions for performing a structural analysis.

6.2 Tutorial 6_1 – 4”x1”x1” 3-D cantilevered beam

In this tutorial you will create a cantilevered beam, and perform a stress analysis after constraining it and loading it with a 500lb load. The beam is 4 inches long and has a 1 inch square cross-section.



Manual calculation of the deflection at the end of the cantilevered beam can be done using the following formula introduced in strength of materials courses: $\delta = (PL^3)/(3EI)$, along with an additional term that takes into account the deflection due to shear $(6PL)/(5AG)$. Although the deflection due to shear for the cantilevered beam is small, we will include it here for sake of completeness[1]. Since the beam is made of structural steel, we will be using a modulus of elasticity *E* = 29,007,557psi, and determining the shear modulus, from the formula $G = E/(2\,(1+\mu)\,)$ = 11,156,753psi (based on the values stored in the ANSYS database for structural steel). Substituting these values into the deflection equation yields an expected deflection of:

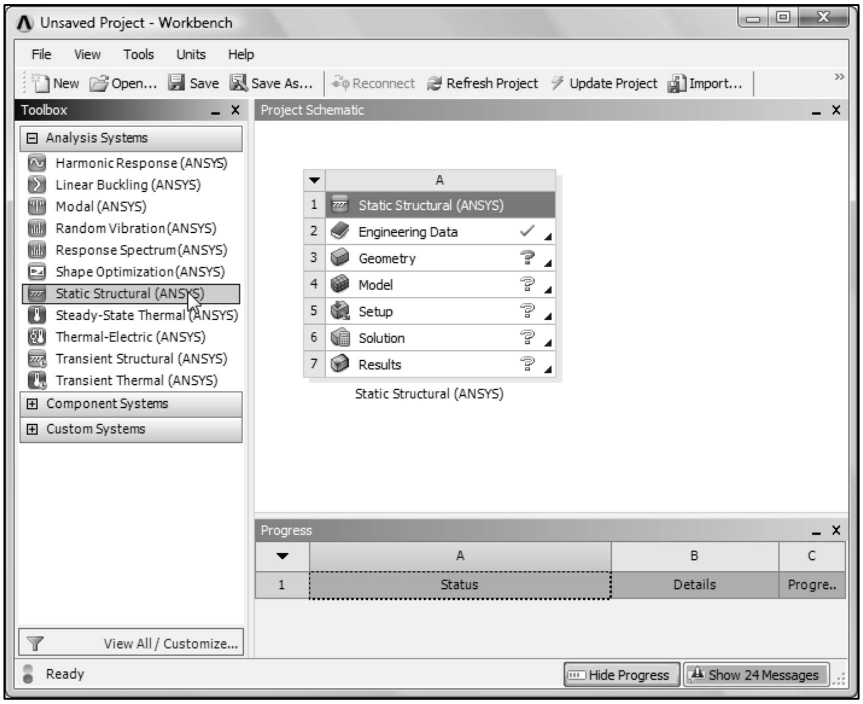
$$\delta = \frac{PL^3}{3EI} + \frac{6PL}{5AG} = \frac{(500lb)(4in)^3}{3(29.01e6\,psi)(\frac{(1in)(1in)^3}{12})} + \frac{6(500lb)(4in)}{5(1in)^2(11.16e6\,psi)} = 0.004627in$$

The maximum bending stress in the beam is expected to occur at the top and bottom of the beam where it contacts the wall. Again, from strength of materials, we would expect a maximum bending stress value of:

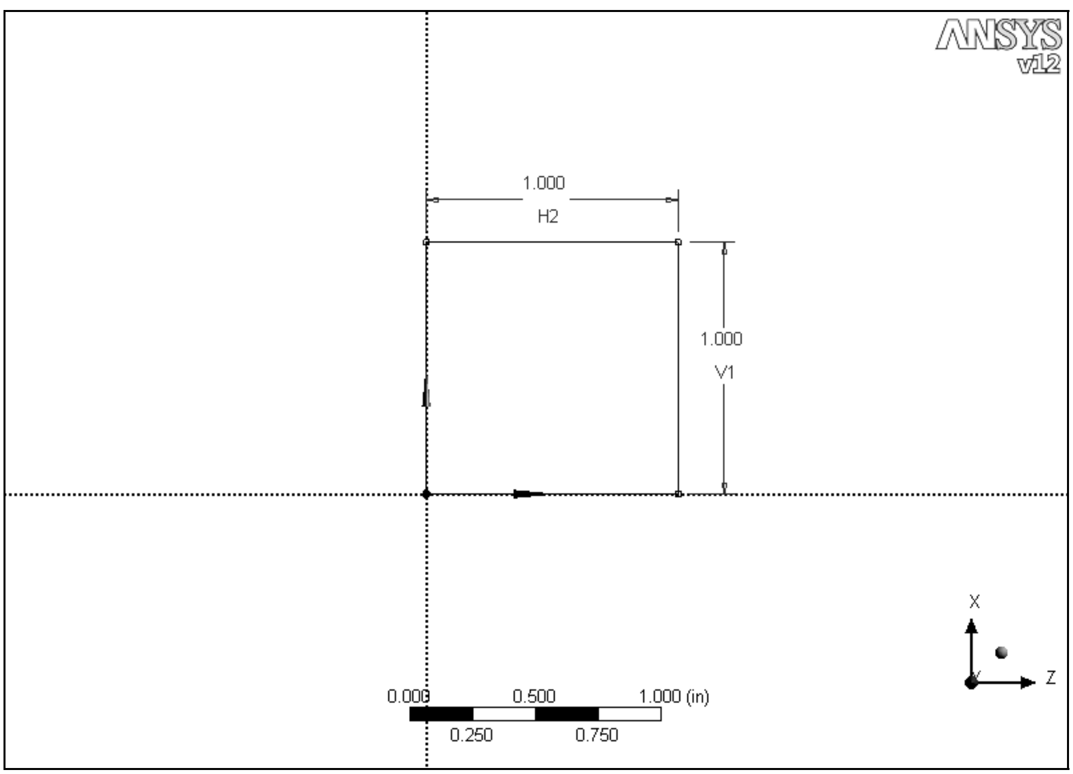
$$\sigma = \frac{Mc}{I} = \frac{(500lb)(4in)(0.5in)}{\frac{(1in)(1in)^3}{12}} = 12{,}000\,psi$$

We will use these two values to compare with our finite element results.

Step 1 – Start Workbench then start a new **Static Structural** analysis project by double-clicking on the icon in the Toolbox. Once the **Static Structural** System appears in the Project Schematic area, double-click on **Geometry** to start a new **DesignModeler** session. Select **Inch** as the unit system.

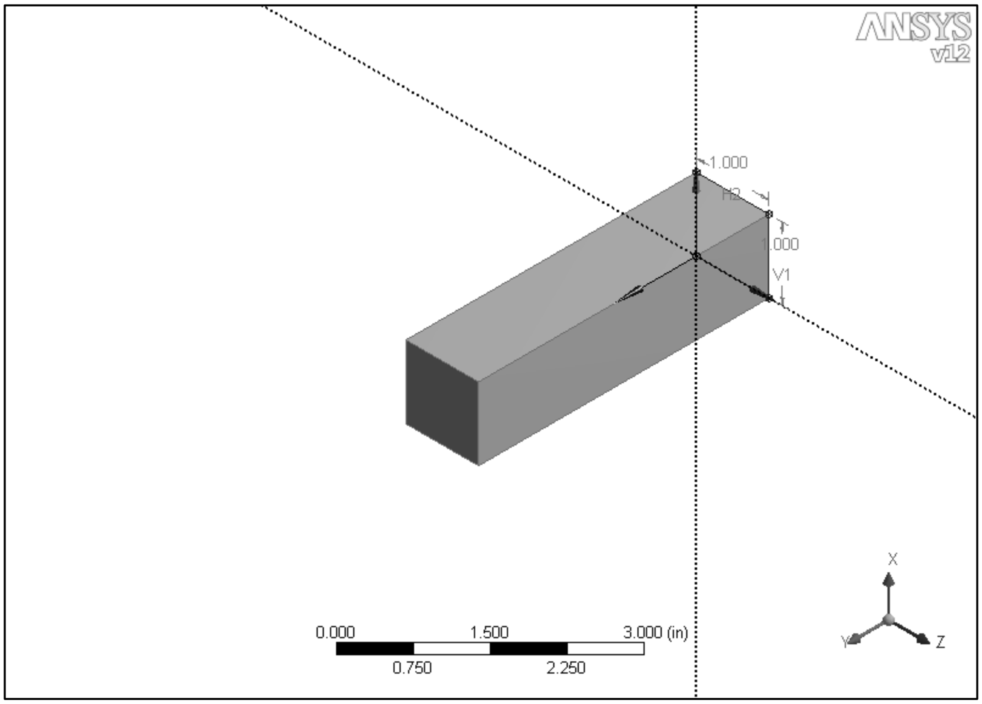


Step 2 – On the **XZPlane** draw a 1″x1″ square section.

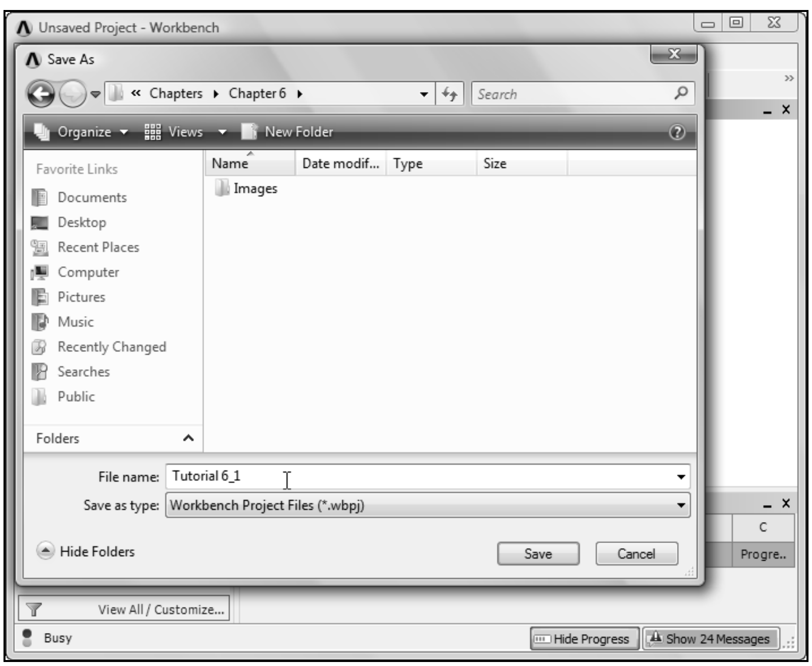


Step 3 – Create a beam by extruding the section 4 inches in the Y direction.

A. Switch to Isometric view by clicking on the **ISO** icon in the toolbar.
B. In the **Details View** switch the **Direction** to **Reversed**.
C. Enter ***4*** for **FD1, Depth(>0)**.
D. Click **Generate**.

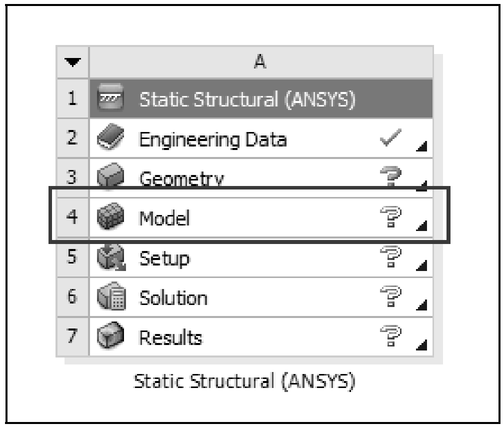


Step 4 – Save the project. Click on the **Save** (diskette) icon, browse to a desired location and save the file as ***Tutorial 6_1***.

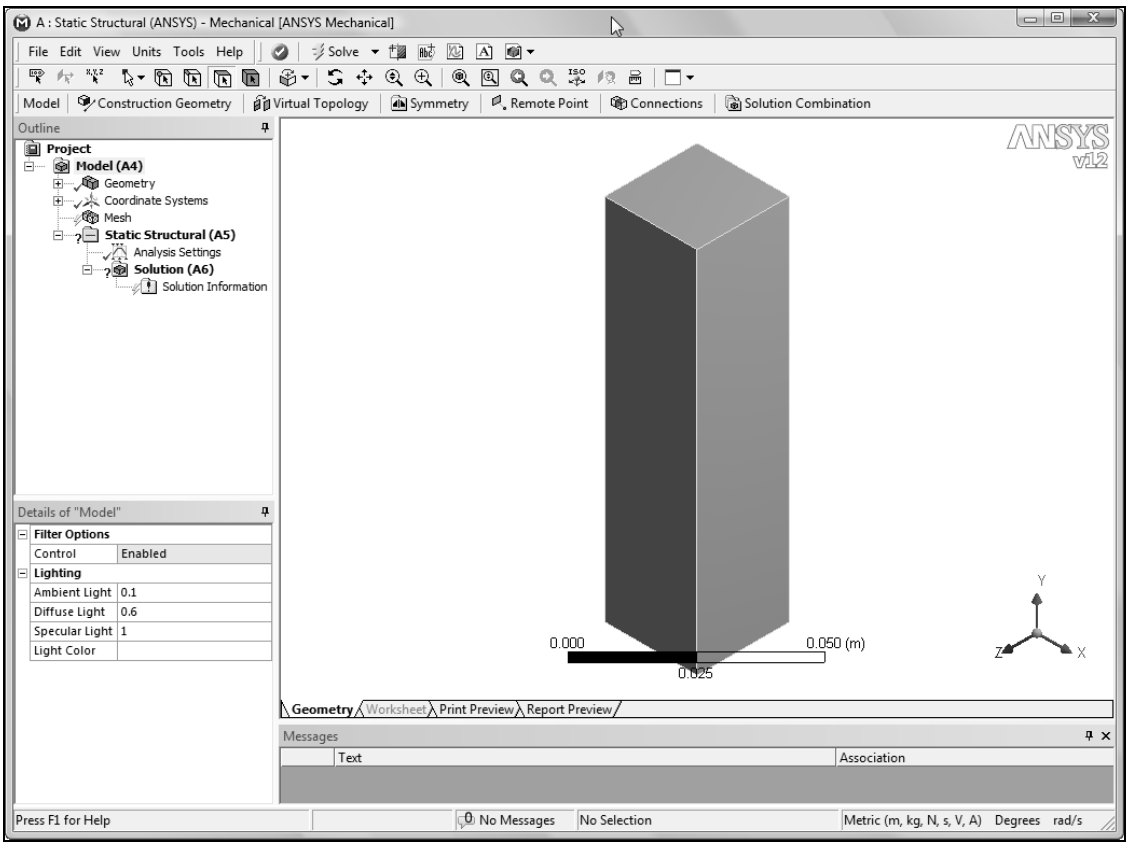


At this point a green checkmark appears next to **Geometry** in the **Static Structural** system in the **Project Schematic** area. Since we will not be making any changes to the beam geometry, we can minimize the **DesignModeler** window or if we wish we can close the application.

Step 5 – Begin a simulation using the solid model of the beam. In the Project Schematic area double-click on **Model**.



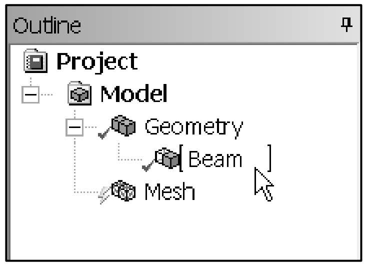
Step 5 (continued) – The simulation window will open and the part file will be brought in and displayed as shown.



Check the bottom right portion of the Simulation window to make sure the units are set correctly. In this tutorial the working units are "**US Customary (in,lbm,lbf,F,s)**". If a different system is displayed, use the **Units** pull down menu to change working units.

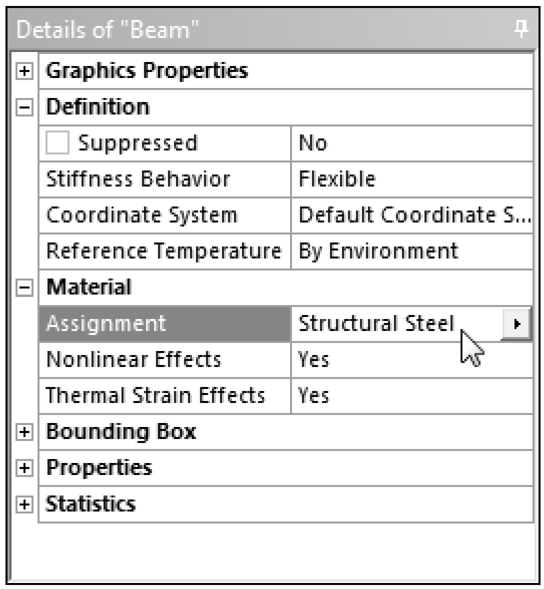
Step 5 (continued) – Rename the solid model to **Beam**.

A. In the **Outline** pane expand the **Geometry** object.
B. Click the RMB on **Solid** and select Rename.
C. Type ***Beam.***



Step 6 – Set the material type. By default, the Simulation application sets all materials in the model to be Structural Steel. In this step we simply check that the material type has been set.

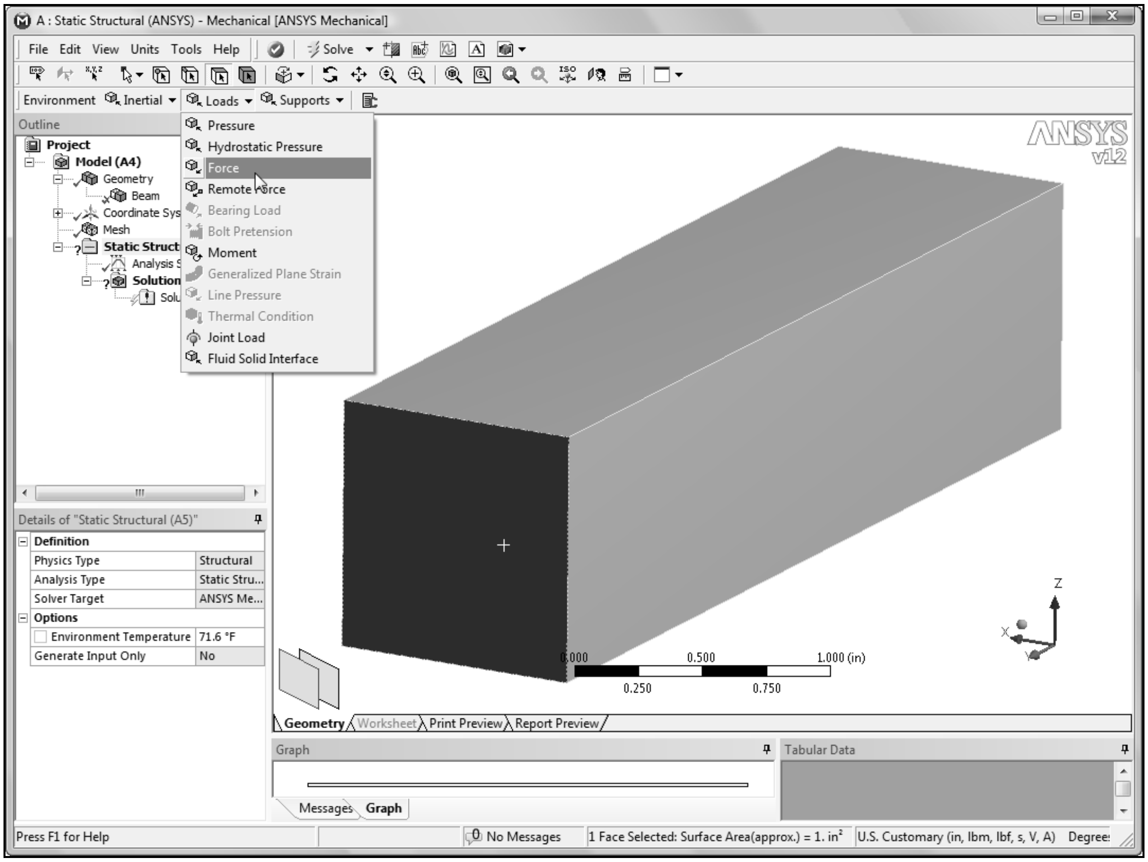
Select **Beam** in the **Outline** pane and look at the **Material** setting in the **Details** pane below.



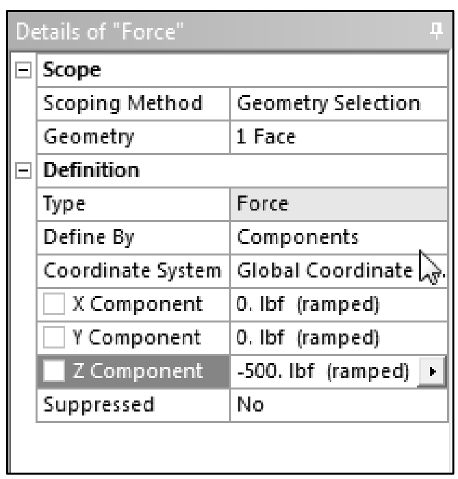
See Section 3.2 in Chapter 3 for instructions on adding new material types to your project.

Step 7 – Set the load. The loading for this analysis consists of a 500lbf force in the –Z direction.

A. Select the **Static Structural** folder in the Outline view. The Environment toolbar is displayed.
B. Orient the beam by rotating it so that the Z axis is in the vertical direction and one end of the beam is visible as shown. The Y-axis should point towards you.
C. Change the selection mode to **Face** by clicking on the face icon in the graphics toolbar (called the **Selection** toolbar in DesignModeler)  and then select the visible end face.
D. Click on **Loads** in the toolbar and select **Force** from the menu.
E. In the **Details** pane, change **Define By** to **Components** if it is not already set.
F. Enter ***-500*** for the Z component and ***0*** for the X and Y components.

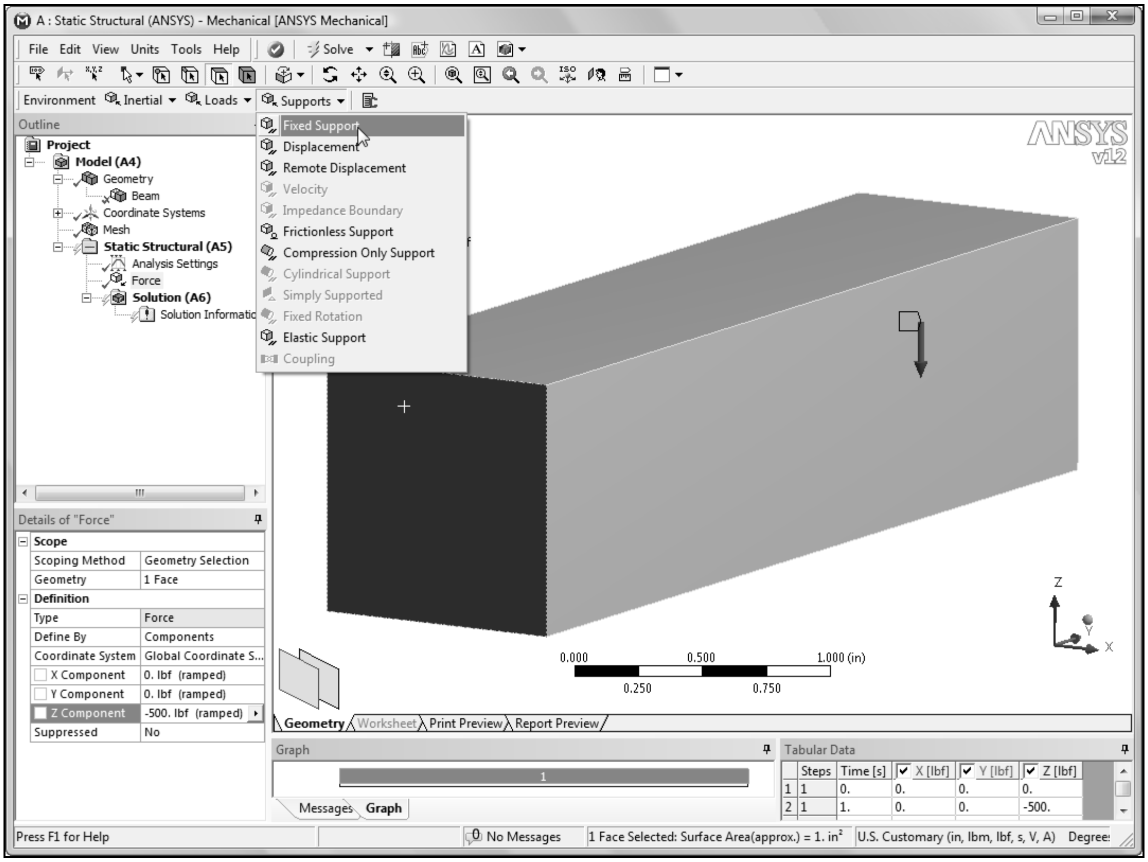


Step 7 – (continued) View of the Detail window pane.



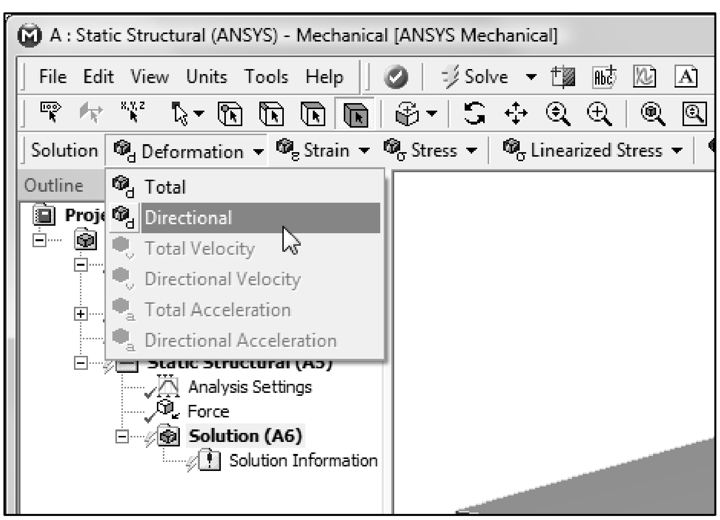
Step 8 – Set The boundary conditions. The boundary condition for this analysis consists of a fixed support at one end of the beam.

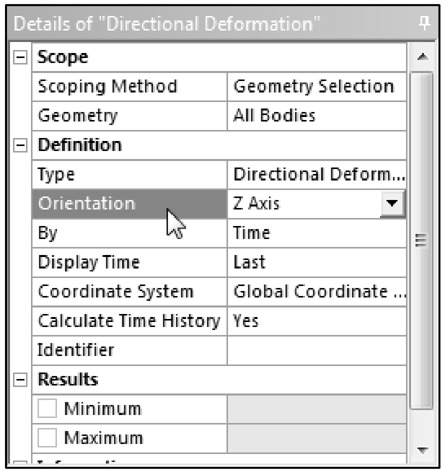
A. Orient the beam by rotating it so that the Z axis is in the vertical direction and the opposite end of the beam (opposite end on which you imposed the force in step 7) is visible. The Y axis will point away from you.
B. Select the visible end face of the beam.
C. Select the **Static Structural** folder.
D. Click on **Supports** in the toolbar and select **Fixed Support**.
E. Click on **Apply** in the **Details** window pane.



Step 9 – Request results.

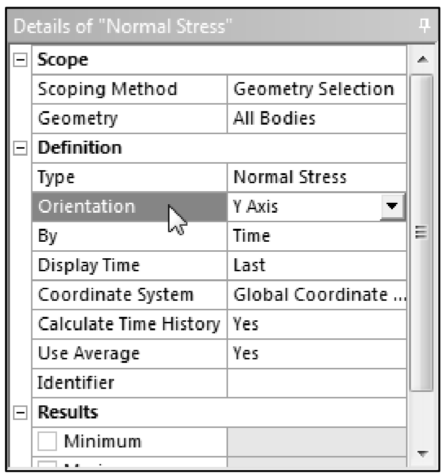
A. Select the **Solution** folder.
B. In the toolbar change the selection filter to Body and click on the beam to select it.
C. In the Solution toolbar click on **Deformation** and select **Deformation-Directional**.
D. In the **Details** pane change the **Orientation** to **Z Axis**.



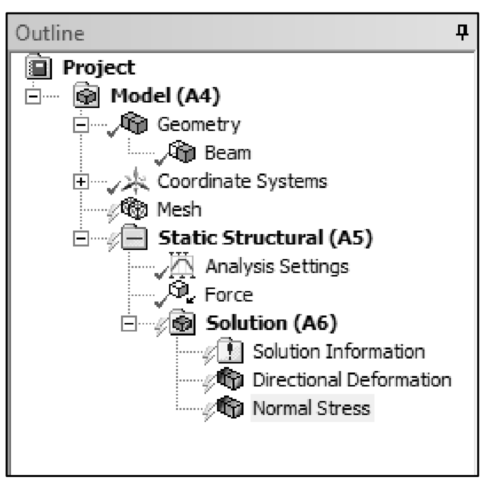


Step 9 (continued) – Request normal stress

E. Click on **Stress** in the toolbar and select **Stress-Normal**.
F. In the **Details** pane change the Orientation to **Y Axis**.

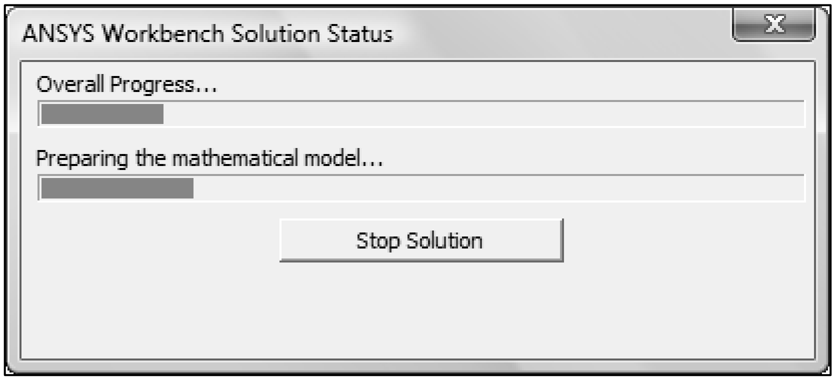


The Project Outline pane now appears as shown below. The question mark next to **Static Structural** folder has been replaced by a yellow lightning bolt indicating that solution has not been computed yet but all the necessary inputs have been set.

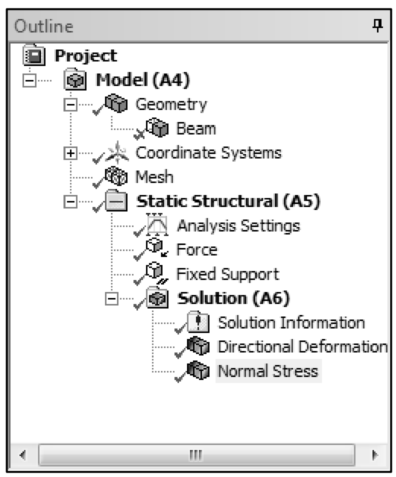


Step 10 – Solve. Click on the 

icon in the toolbar.
The solution status window will appear and display the various stages of the solution process. The solution is complete when this window disappears.

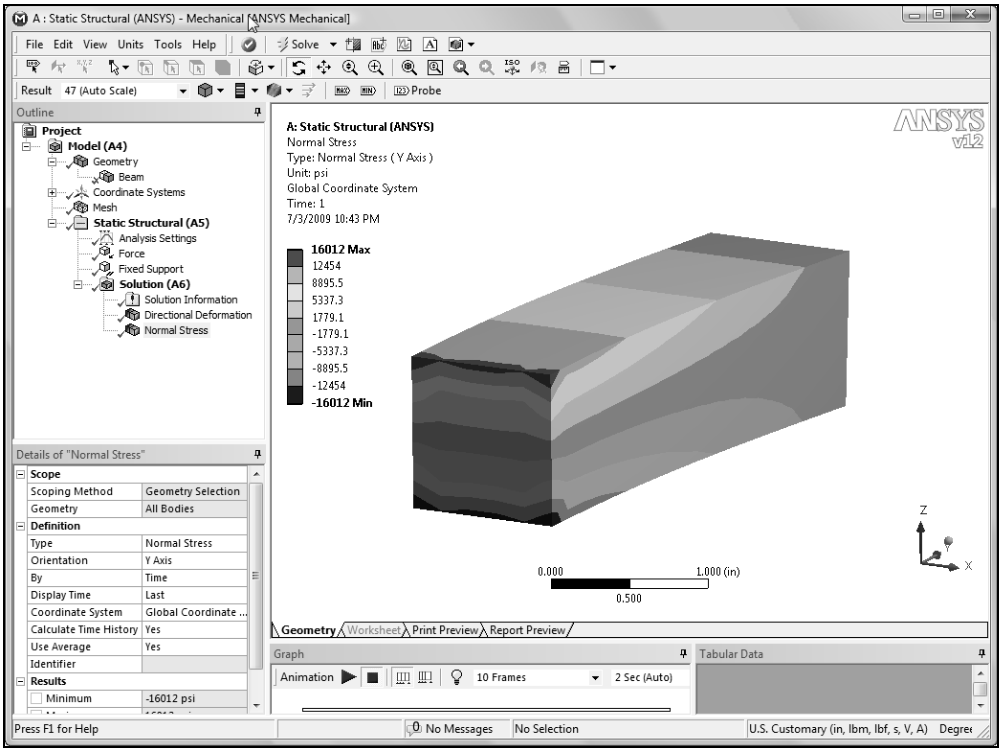


Step 10 (Continued) – Once the solution is successfully completed, green check marks will be placed next to the solution quantities you had requested.



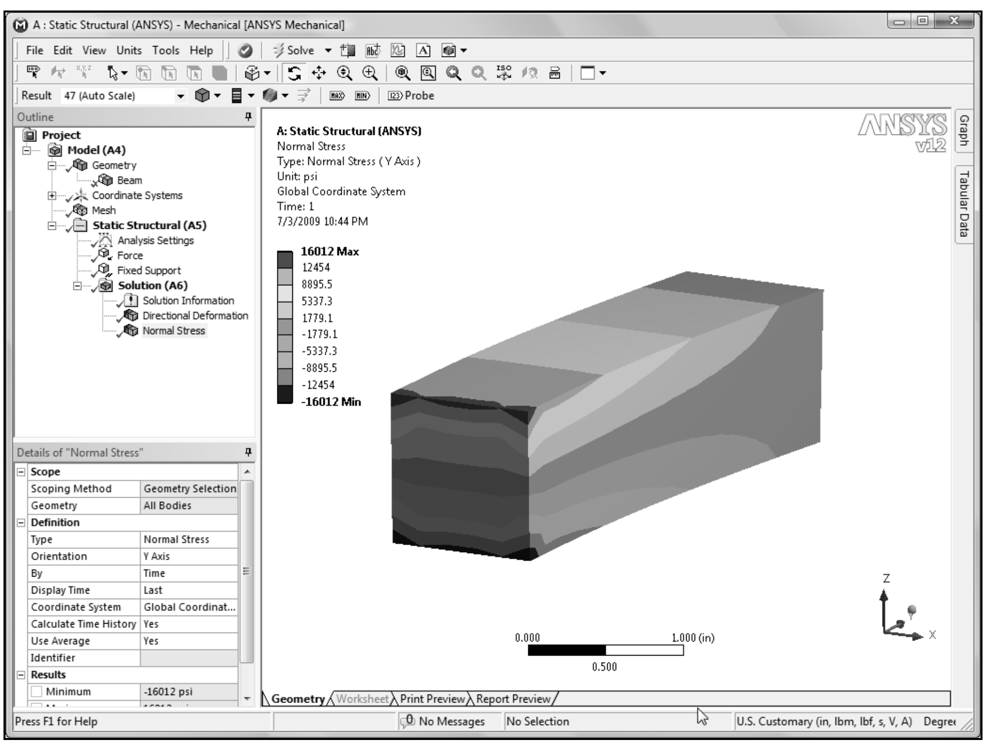
Step 11 – Save the analysis. From the File menu select **Save Project...** The project and the analysis data will be saved under the project name we provided previously.

Step 12 – Review the results. When the solution is complete, click on one of the solution quantities you requested such as **Directional Deformation** or **Normal Stress** to display those quantities as contour plots. For example, the figure below shows the contour plot of the normal stress in the Y direction superimposed on the deformed shape of the beam. Note that since in most cases deflections and deformations are too small to be perceptible, these quantities are automatically scaled (exaggerated) to make them easily visible. In this way you can quickly determine if you have set the loads and boundary conditions correctly. The scale factor can be modified from the **Result** toolbar and set to various values including 1.0 (True Scale).

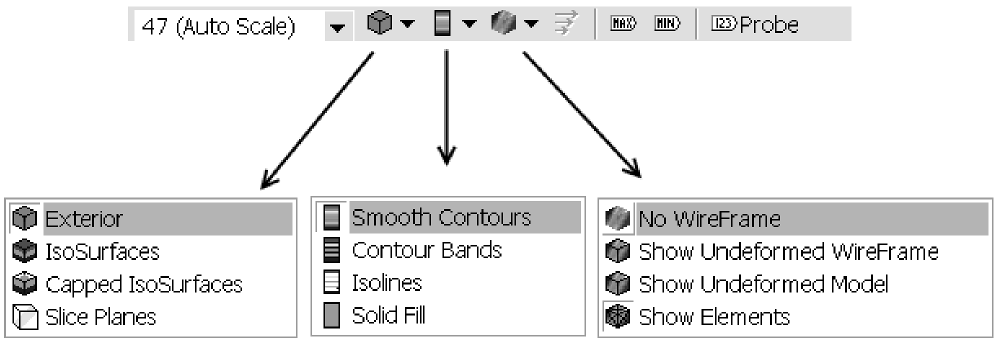


Step 12 (continued) – Enlarge the displayed results area.

By default, when displaying graphical results, a considerable portion of the Workbench window is occupied by window panes such as the **Timeline** and **Tabular Data** window panes. These window panes can be unpinned (collapsed) in order to enlarge the graphical results display as described in Section 3.4.3, Window Manager Features. To unpin the **Timeline** and **Tabular Data** Window panes, click on the push-pins located at the top right hand corner of each pane. The resulting display is shown below.

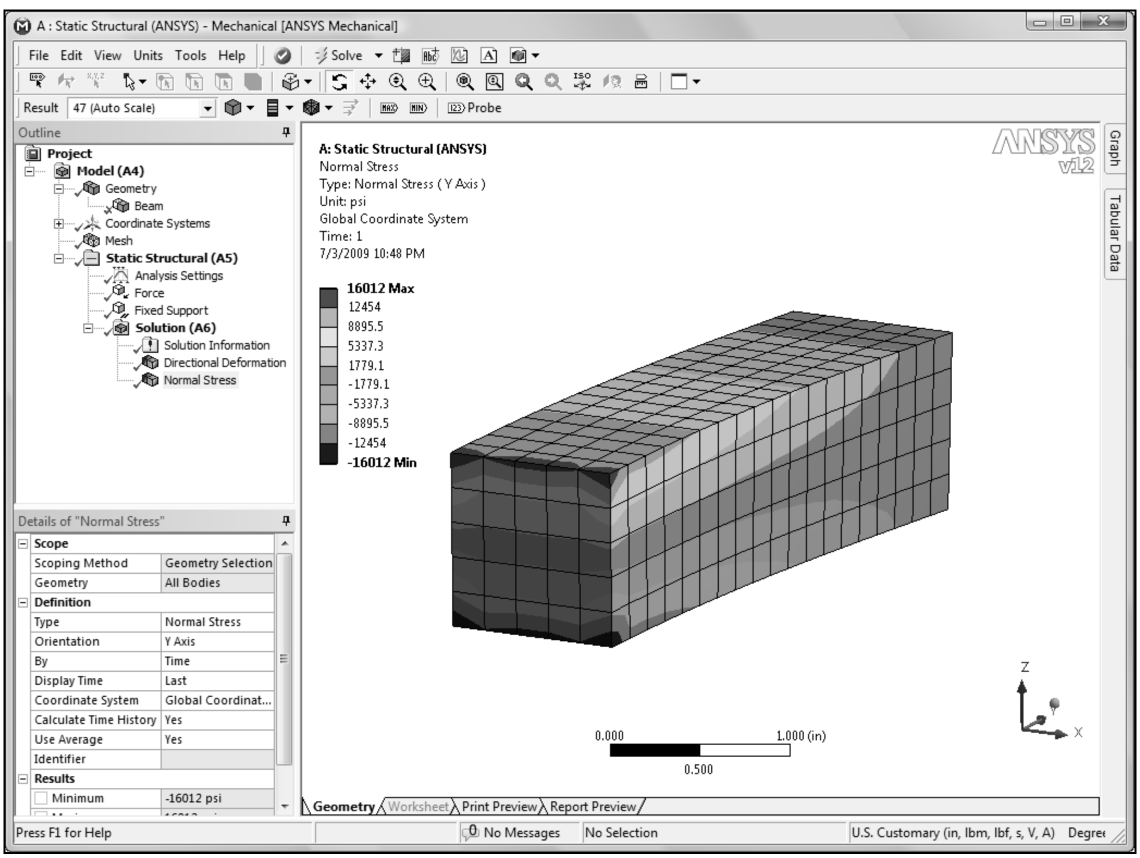


Step 12 (continued) – The displayed contour plots can be modified in several ways in order to better represent the results of the analysis. The modifications are performed from three pull-down menus as shown below.

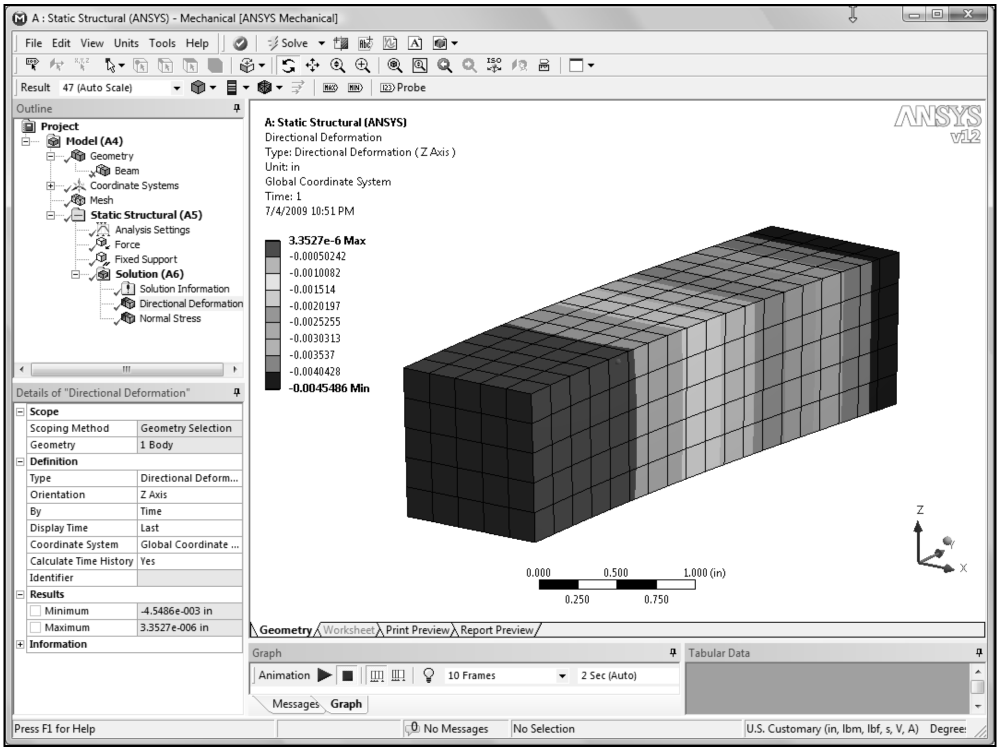


The previous display is set to display **Exterior**, **Contour bands** and **No WireFrame**.

Step 12 (continued) – Select **Contours Bands** and **Show Elements** from the pull down menus. The resulting display is shown below.

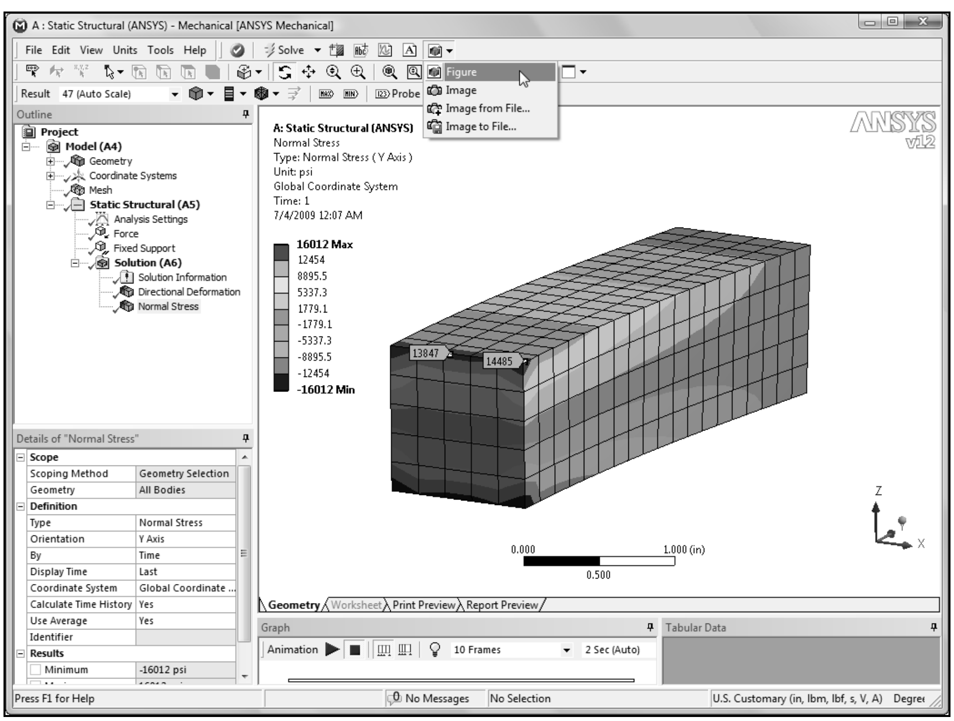


Step 12 (continued) – Click on **Directional Deformation**. The resulting display is shown below.



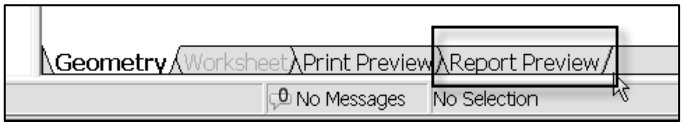
Step 12 (continued) – Note that the maximum and minimum values displayed are the range values for the contours. For example, on the **Normal Stress** plot, the red contours represent stress values ranging from -16,012. PSI to +16,012. PSI. In order to get a more precise value, use the Probe tool:

A. Rotate the beam and position it as shown below.
B. Click on **Probe** in the toolbar.
C. Move the cursor over the contoured part and observe the contour values at the position of the cursor.
D. Click the LMB to place a tag indicating the value at that location.



To remove a tag, click on the **Label** icon in the toolbar, select the tag to be deleted and press the Delete key on the keyboard.

Step 13 – Generate a report. A simple HTML report can be automatically generated by simply clicking on the **Report Preview** tab.



The report generated in this way includes basic information about the analysis such as the type of analysis, material properties of the materials used, boundary conditions, etc.

Step 13 (continued) – The table of contents of the default report is shown below. The entries in the table of contents are hyperlinked to the location of the information in the report.

Contents

- **Model**
  - Geometry
    - Beam
  - Mesh
  - **Static Structural**
    - Analysis Settings
    - Loads
    - Solution
      - Solution Information
      - Results
- **Material Data**
  - Structural Steel

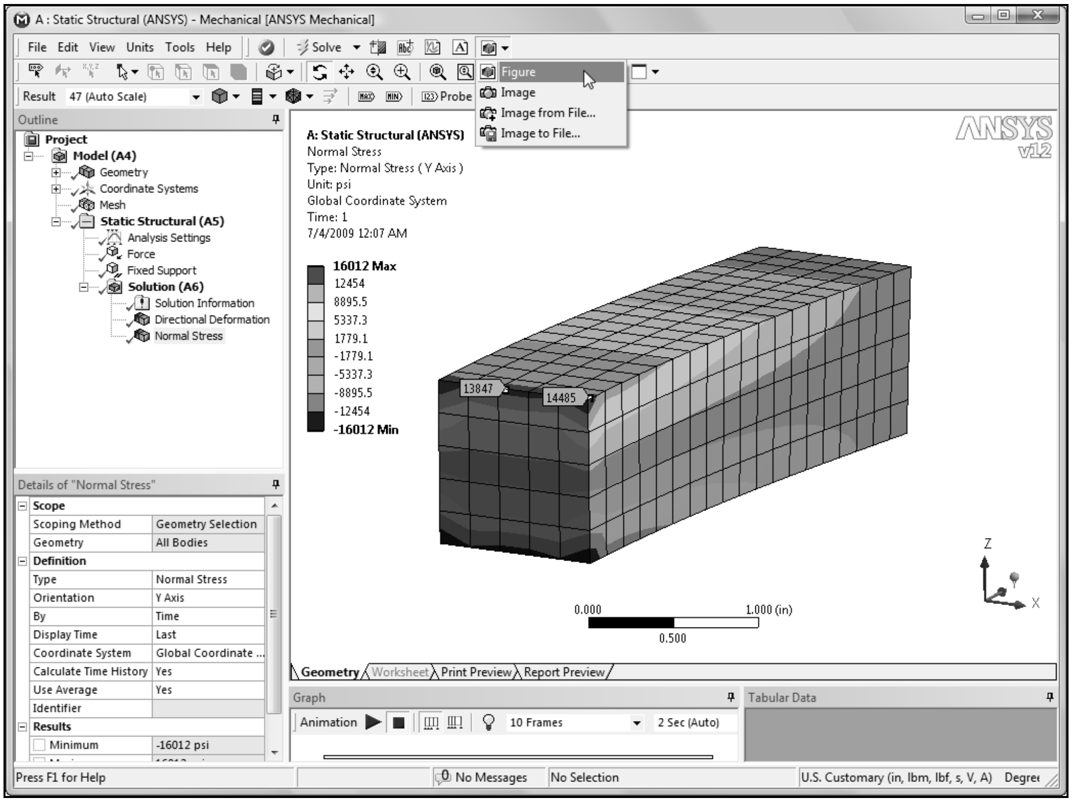
Step 13 (continued) – The default report includes the calculated results in tabular form but deformed shapes and contour plots are not automatically included in the report. An example of tabular results is shown to the right.

TABLE 11
Model (A4) > Static Structural (A5) > Solution (A6) > Results

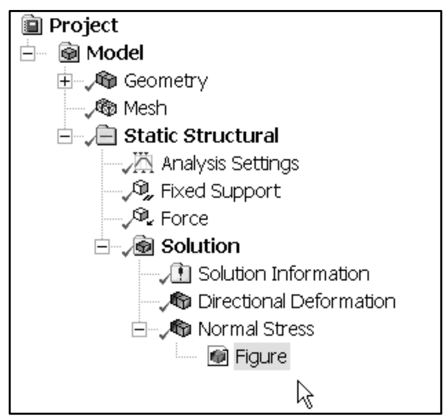
Object Name	*Directional Deformation*	*Normal Stress*
State	Solved	
Scope		
Scoping Method	Geometry Selection	
Geometry	All Bodies	
Definition		
Type	Directional Deformation	Normal Stress
Orientation	Z Axis	Y Axis
By	Time	
Display Time	Last	
Coordinate System	Global Coordinate System	
Calculate Time History	Yes	
Identifier		
Use Average		Yes
Results		
Minimum	-4.5486e-003 in	-16012 psi
Maximum	3.3527e-006 in	16012 psi
Information		
Time	1. s	
Load Step	1	
Substep	1	
Iteration Number	1	

Step 14 – Include contour plot of normal stress in the report.

A. In the Project tree click on **Normal Stress**.
B. Click on the **Geometry** tab at the bottom of the window.
C. Adjust the figure as desired by zooming, rotating, etc.
D. From the Figures pull down menu in the toolbar select **Figure**.



Step 14 (continued) – A figure of the contour plot of normal stress has been inserted in the project tree under **Normal Stress**. The figure can be renamed using the **RMB** and you can also add text that appears as caption with the figure.



Step 15 – Regenerate the report. Click on the **Report Preview** tab as before to regenerate an updated report which includes the contour plot figure.

Note: Figures inserted in the project tree are updated with the latest results every time a new report is generated. Therefore, they always reflect the latest conditions of the analysis. In order to include a static picture, select **Image** in Step 14, sub step D.

Step 16 – Exit Workbench. You will be prompted to save the project (Tutorial 6_1) since it was modified since the last time it was saved.

Summary

As we can see by comparing our manual calculations with the values calculated by ANSYS, the displacements are fairly close. However, the stress values do not have very good correlation.

	Manual Calculation	ANSYS Results	% difference
Max. Deflection	0.004627 in.	0.0045486 in	1.694%
Max. Bending Stress	12,000psi	13,923psi 15,877psi	16.03% 32.31%

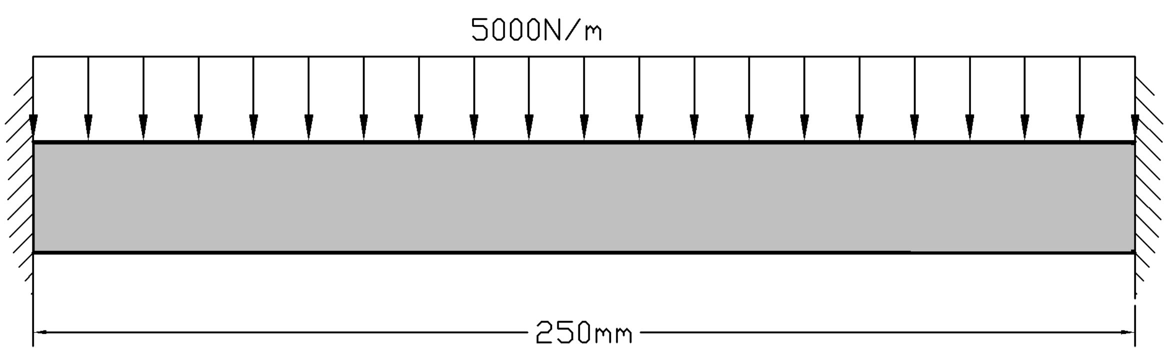
The primary reason for the poor correlation of stress values was due to the boundary condition occurring at the same location as our maximum stress values. However, our goal during this chapter was to become familiar with the steps needed to get a very simply job to run, and not to be concerned with the accuracy of the results. During the next chapters we will investigate several different techniques that can be used to improve the results of our analysis.

Reference

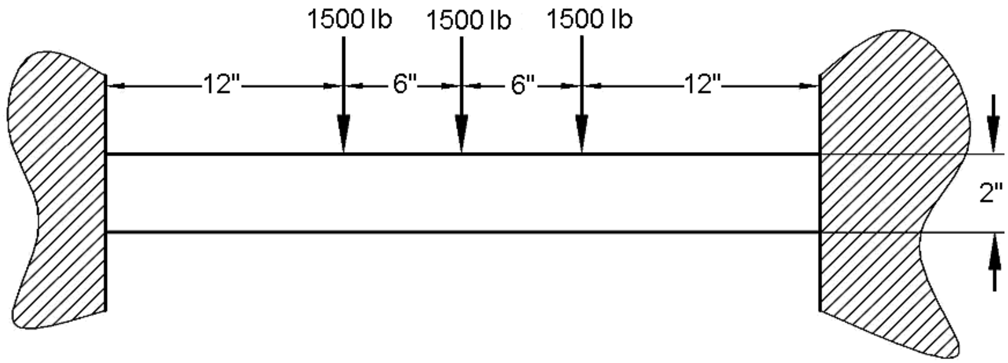
[1] Popov, E. P., Mechanics of Materials, 2nd Edition, Prentice-Hall, Englewood Cliffs NJ, 1976

Exercises:

1. Use ANSYS Workbench to build a finite element model of the beam shown below, that has a 20mm x 20mm cross section and is made from structural steel. Determine the maximum deflection and bending stress in the beam. Then compare these values with those that you manually calculate using strength of material formulas from your textbooks (you do not have to take into account the deflection due to shear, as was done in the example problem in this chapter).



2. Use ANSYS Workbench to model and analyze the 36 inch long beam shown below. Imprint lines on your model of the beam, to use as edges on which to apply the 1500 lbs. loads. The beam is made from 1/2" thick, structural steel. Determine the deflection and normal (bending) stress at the center of the beam. Then compare these values with those that you manually calculate using strength of material formulas from your textbooks (you do not have to take into account the deflection due to shear, as was done in the example problem in this chapter).



3. Use ANSYS Workbench to build a finite element model of the cantilevered beam shown below, that is made from structural steel. Determine the maximum deflection and bending stress in the beam. Then compare these values with those that you manually calculate using strength of material formulas from your textbooks (you do not have to take into account the deflection due to shear, as was done in the example problem in this chapter).