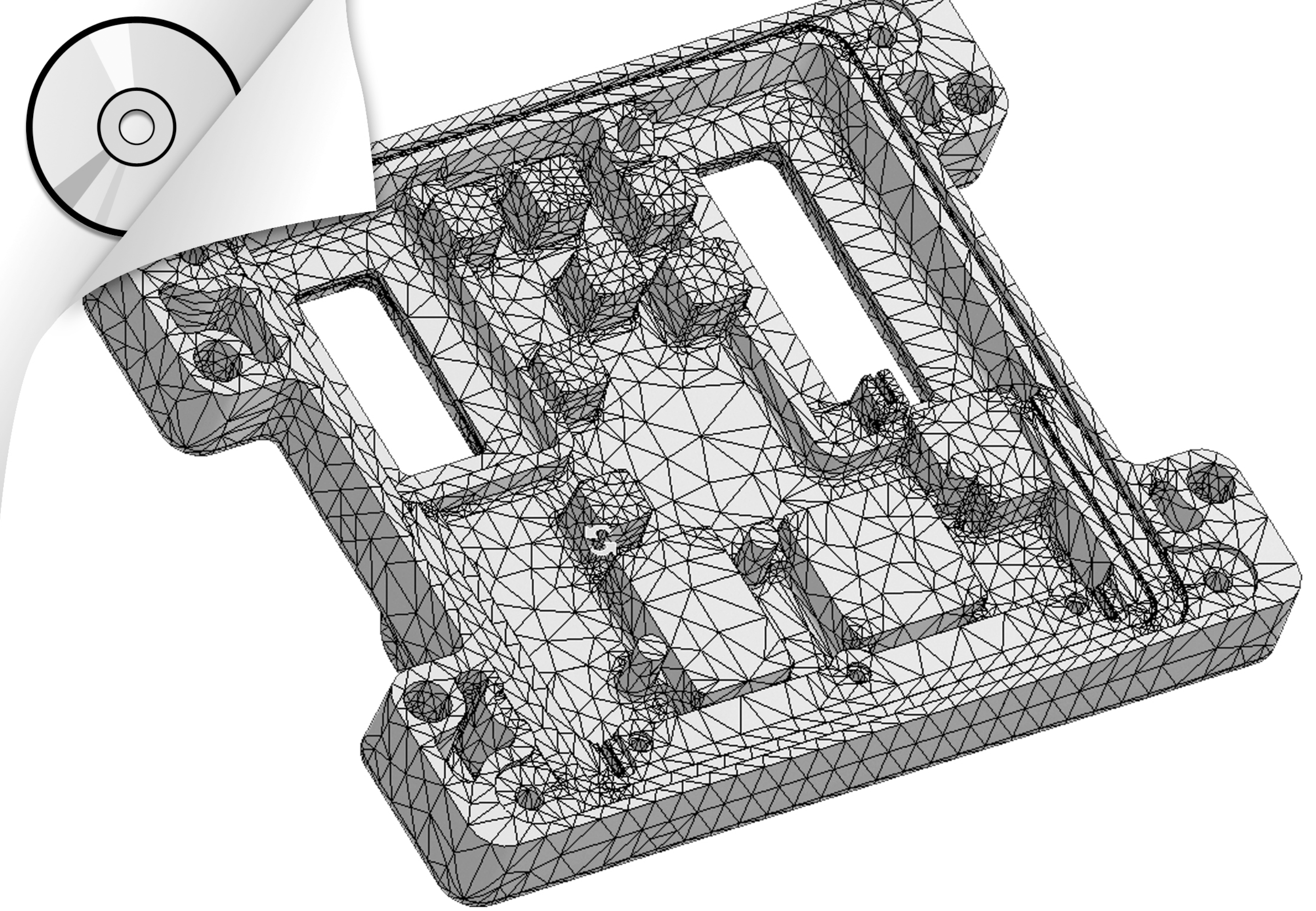
ANSYS® Workbench™ Software
Tutorial with Multimedia CD

RELEASE 11

INSIDE:



■ **Fereydoon Dadkhah**
Delphi Electronics & Safety

■ **Jack Zecher, P.E.**
Indiana Univ. - Purdue Univ. Indianapolis

SDC publications

Schroff Development Corporation

www.schroff.com
www.schroff-europe.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 to solve a problem

In general you follow the same steps to perform a finite element analysis. However, it is important to note that it is possible to use Workbench to perform a large number of different analysis types. These analysis types may include various material non-linearities, transient loads, rigid body dynamics, etc. which may require additional steps to be performed. The steps described below are aimed at solving a static, linear stress, heat transfer or free vibration analysis. Also remember that the following steps are performed in the **Simulation** application of Workbench.

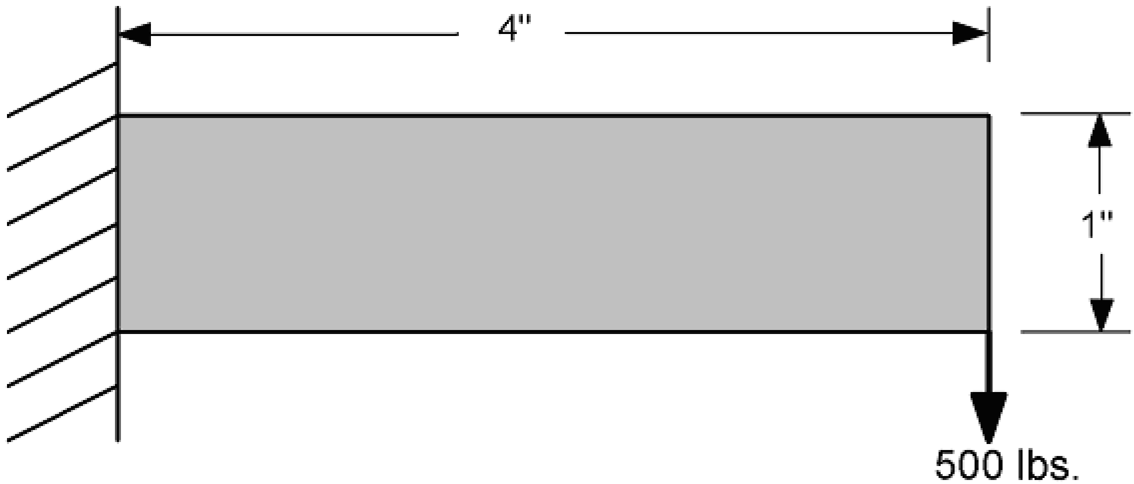
1. **Attach to geometry**
2. **Define/Assign material properties**
3. **Define the analysis type**
4. **Set loading and boundary conditions**
5. **Request results**
6. **Solve**
7. **Review the results**
8. **Generate a report**

Below is a brief explanation of each step.

1. **Attach to geometry**. In this step you identify the geometry (CAD) model that will be used in your simulation. The model may have been created in DesignModeler or in some other CAD tool.
2. **Define/Assign material properties**. In this step you specify the type of material each part of your model is made of. You can assign material types from a small database supplied with Workbench or if the material is not in the database you must define it using the **Engineering Materials** application.
3. **Define the analysis type**. In this step you set the type of analysis you will be performing such as Static Structural, Modal or Steady-state heat transfer.
4. **Set loading and boundary conditions**. In this step you specify how your model is constrained and what loads are acting on it.
5. **Request results**. In this step you specify the results quantities that you want to see once the problem has been solved.
6. **Solve**. In this step you request that Simulation solve the problem you have defined and compute the results you requested.
7. **Review the results**. In this step you review the analysis results that you requested in step 5.
8. **Generate a repor**t. This step is optional but is recommended. In this step you generate an HTML report which includes the inputs to your simulation, the results and any comments you want to add. You can publish the report or e-mail it in various formats.

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: *δ = (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 + μ))* = 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:

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

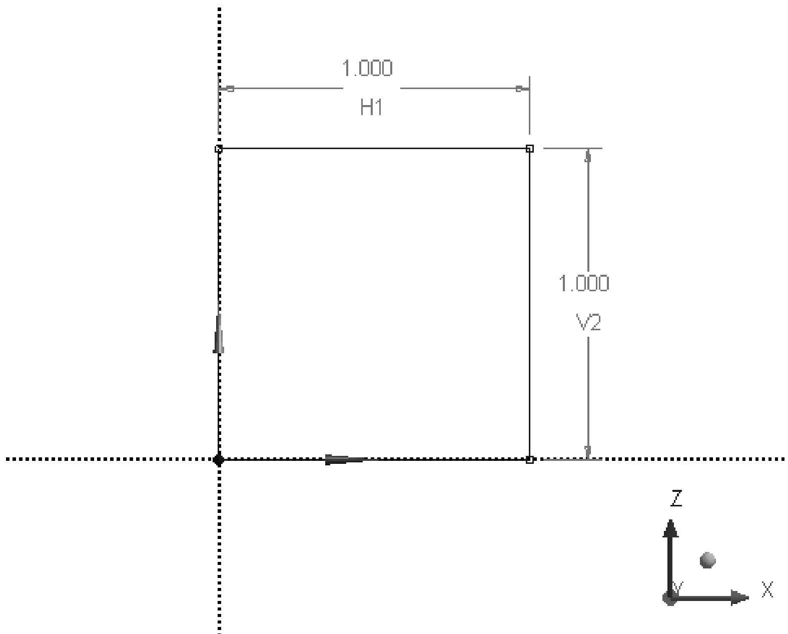
The maximum bending stress in the beam is expected to occur at the top and bottom of the beam where it contacts the wall. Again, from strength of materials, we would expect a maximum bending stress value of:

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

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

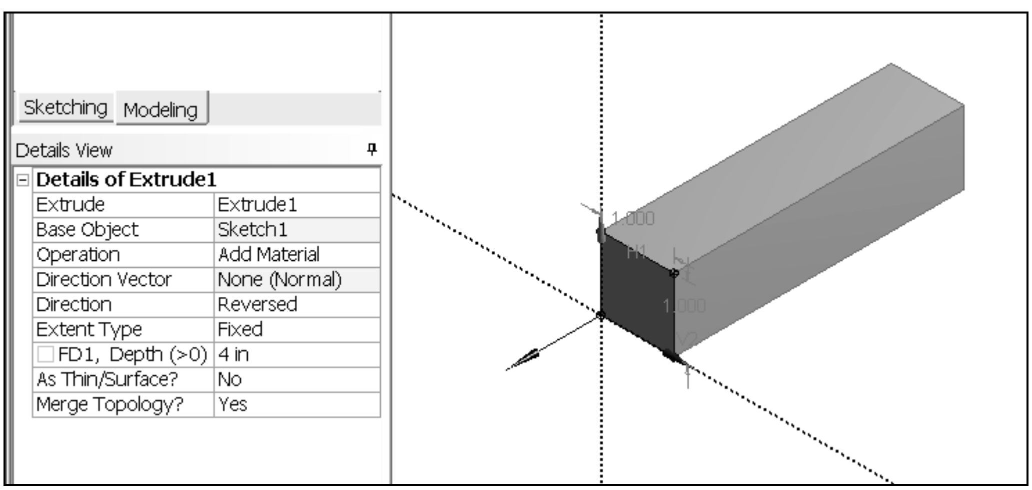
Step 1 – Start a new DesignModeler database and select **Inch** in the unit selection dialog box.

Step 2 – On the **XZPlane** construct a 1”x1” square section.

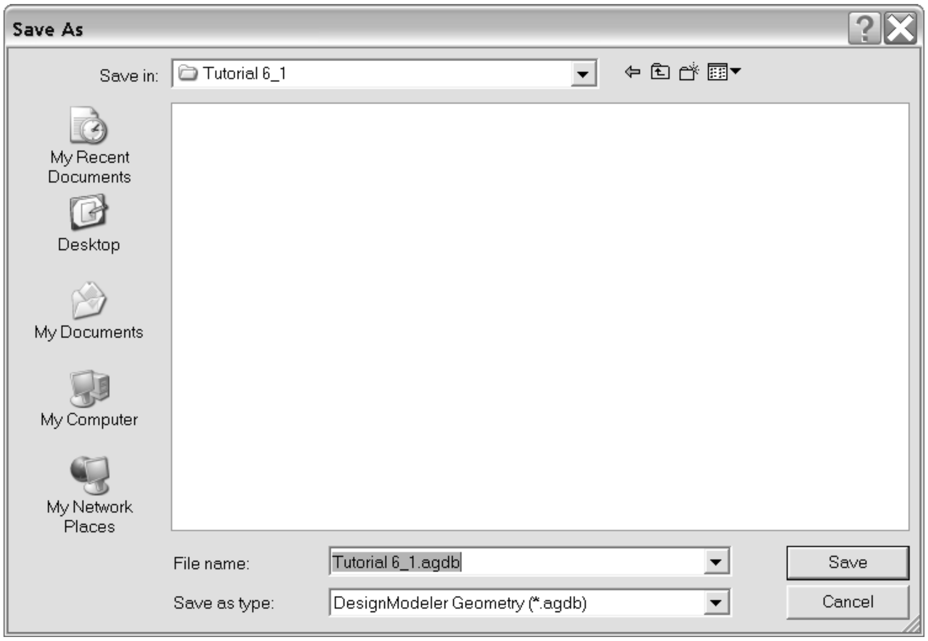


Step 3 – Create a beam by extruding the section 4 inches in the Y direction.

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**.

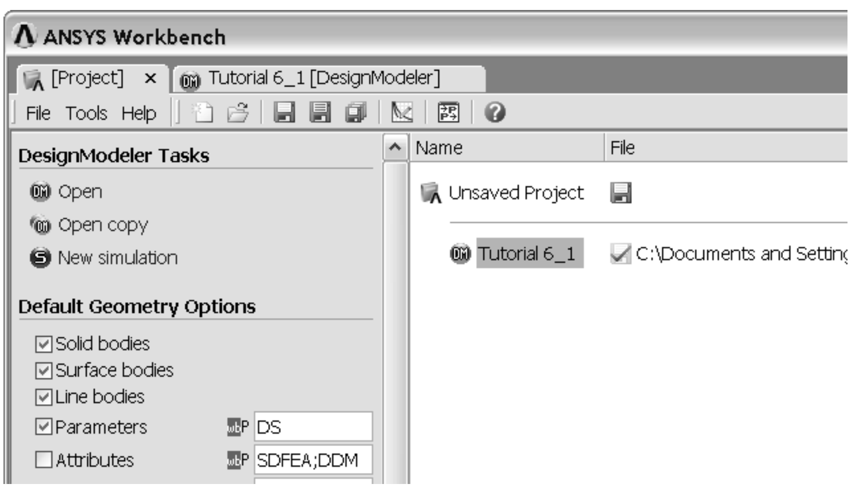


Step 4 – Save the solid model. Click on the **Save** (Floppy disk) icon, browse to a desired location and save the file as ***Tutorial 6_1***.



Step 5 – Begin a simulation using the solid model of the beam.

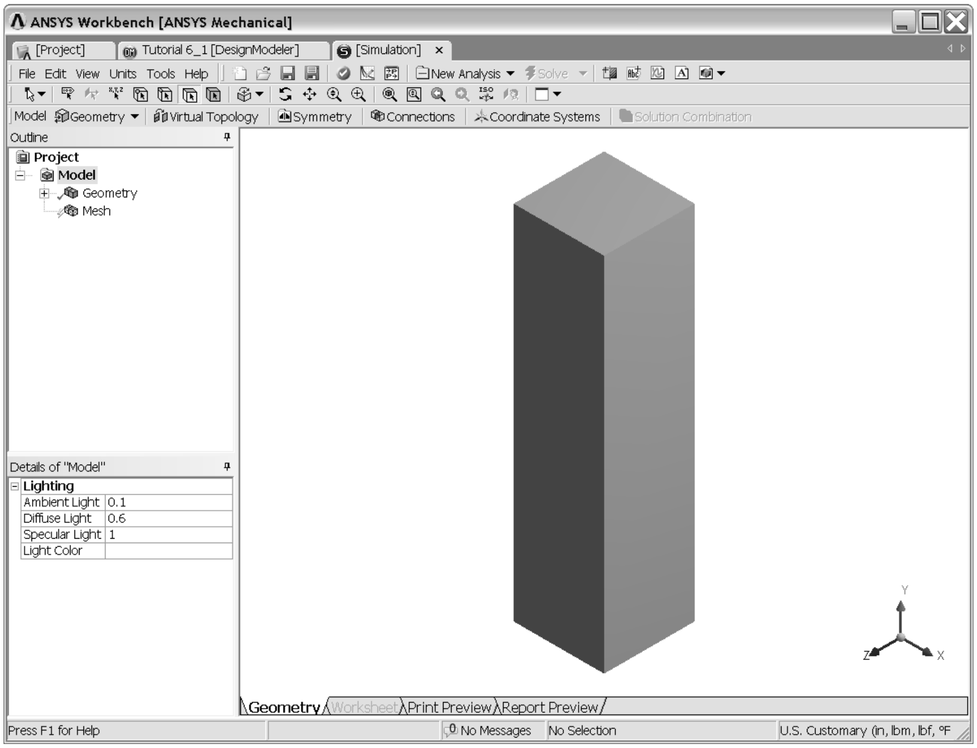
A. Go to the project page by clicking on the project tab.
B. Select the solid model (Tutorial 6_1.agdb) and click on **New Simulation**.



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

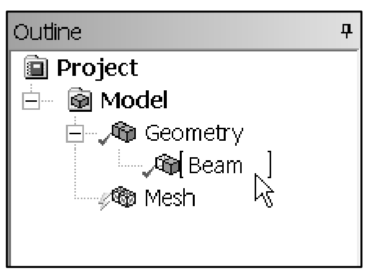
Note: If you see the **Map of Analysis Types** window pane on the right hand side, click on the **X** at the top of the window pane to close it. See section 3.6.2 for steps to prevent the map from being displayed when the Simulation application starts.

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.



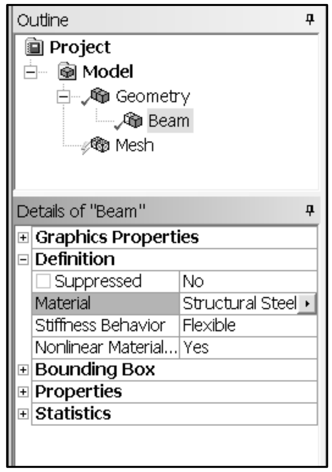
Step 5 (continued) – Rename the solid model to **Beam**.

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



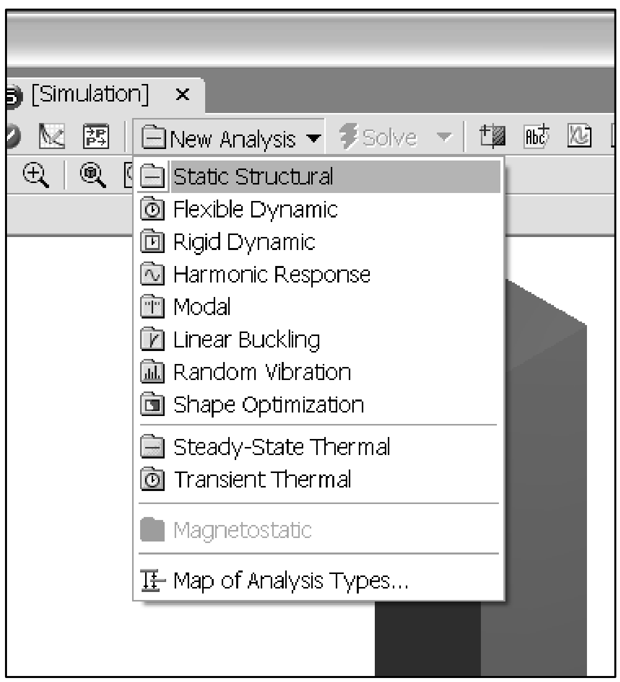
Step 6 – Set the material type. By default, the Simulation application sets all materials in the model to be Structural Steel. In this step we simply check that the material type has been set.

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



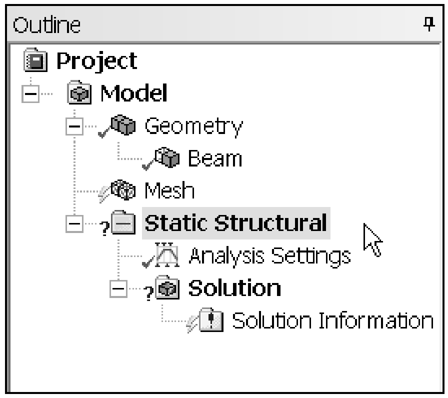
See section 3.5 in chapter 3 for instructions on adding new material types to your project.

Step 7 – Define the analysis type. Click on the **New Analysis** pull-down menu and select **Static Structural**.



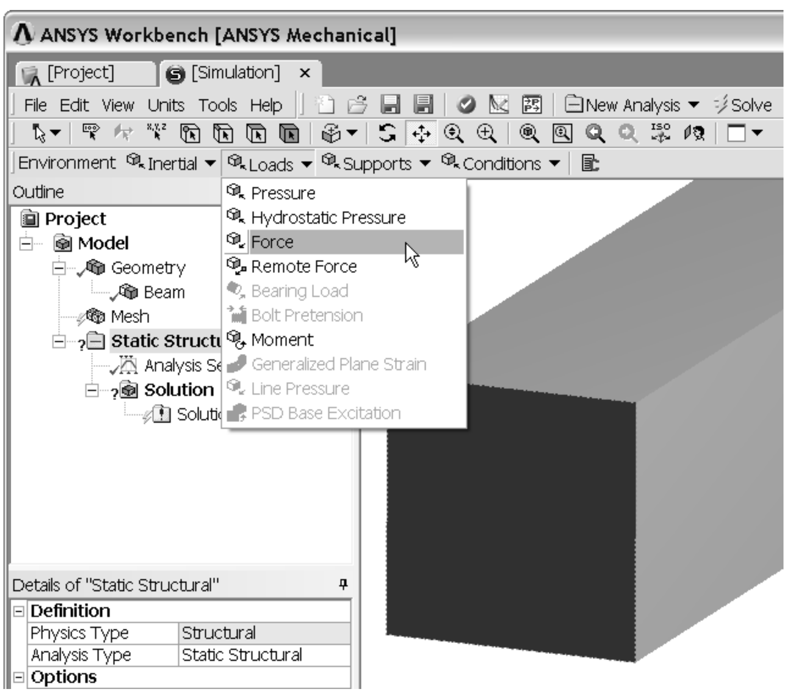
Step 7 (continued) – The **Static Structural** folder and the **Solution** folder are added to the **Model** in the **Outline** pane.

Note the question mark icons next to the Static Structural folder and the Solution folder. The reason for the question marks is that we have not yet defined any loading, boundary conditions or desired results for this analysis. Once these quantities are specified, the questions marks will be replaced by green check marks or other appropriate icons.



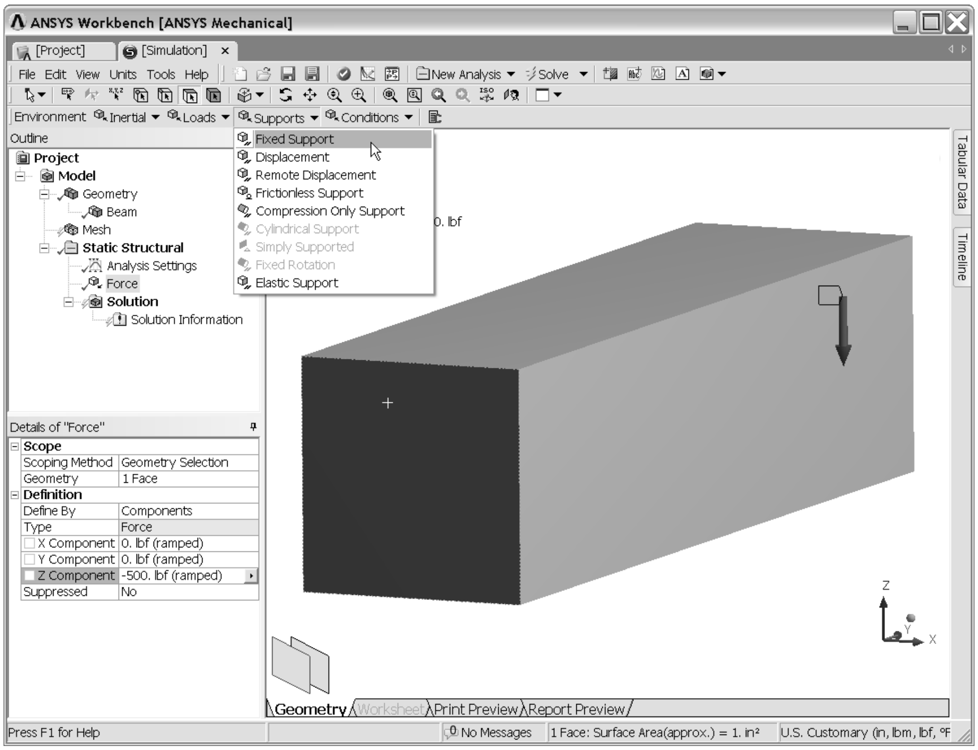
Step 7 (continued) – Set the load. The loading for this analysis consists of a 500lbf force in the –Z direction.

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.

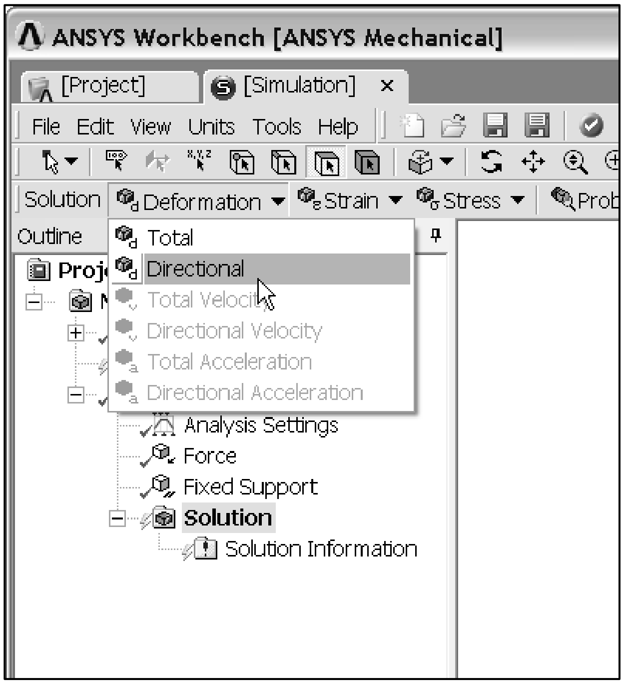


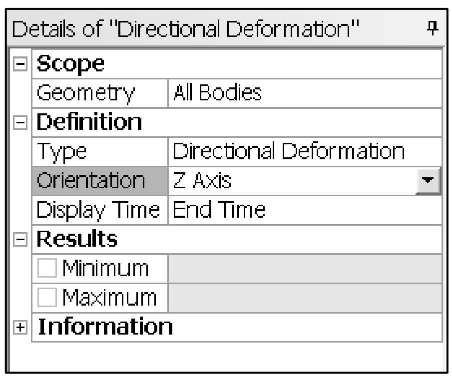
Step 8 – Set The boundary conditions. The boundary condition for this analysis consists of a fixed support at one end of the beam.

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.



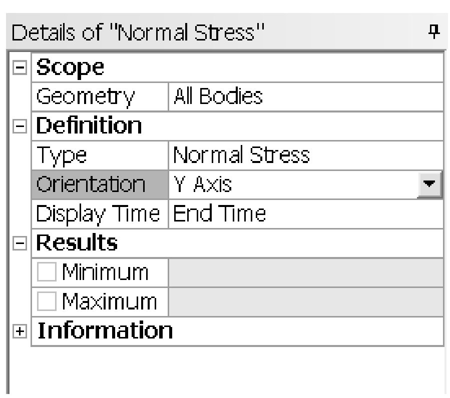
Step 9 – Request results. First request deformation.
A. Select the **Solution** folder.
B. In the Solution toolbar click on **Deformation** and select **Deformation-Directional**.
C. In the **Details** pane change the **Orientation** to **Z Axis**.



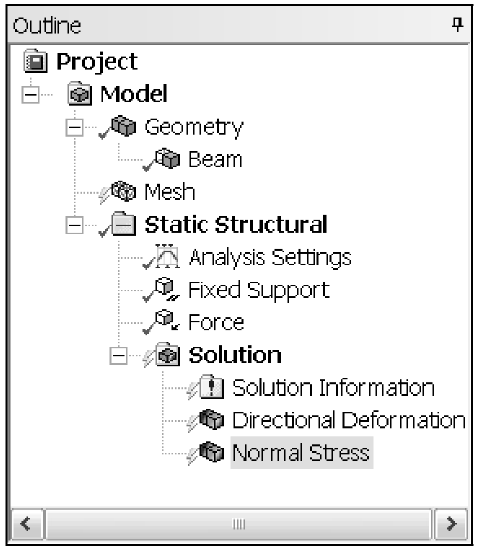


Step 9 (continued) – Request normal stress

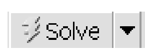
D. Click on **Stress** in the toolbar and select **Stress-Normal**.
E. In the **Details** pane change the Orientation to **Y Axis**.



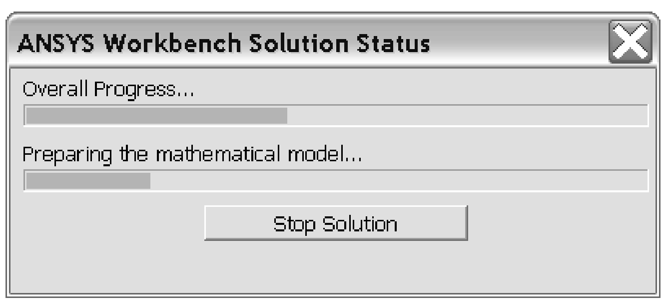
The Project Outline pane now appears as shown below. The question mark next to **Static Structural** folder has been replaced by a green check mark indicating that loads and boundary conditions sufficient to solve this simulation haven been specified. The question mark next to the Solution folder is replaced by a yellow lightning bolt. This icon indicates that the solution has not been calculated yet.



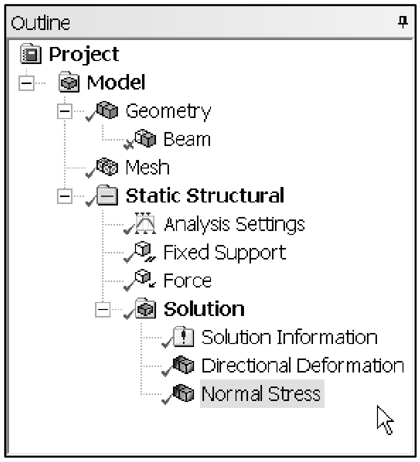
Step 10 – Solve.
Click on the Solve icon in the toolbar.



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

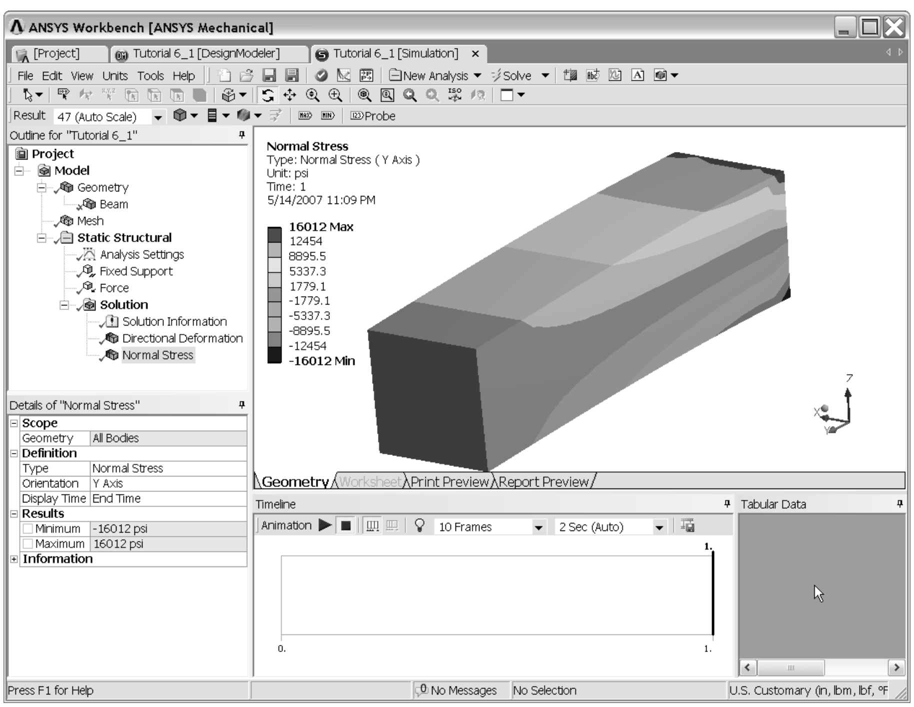


Step 10 (Continued) – Once the solution is successfully completed, green check marks will be placed next to the solution quantities you had requested.

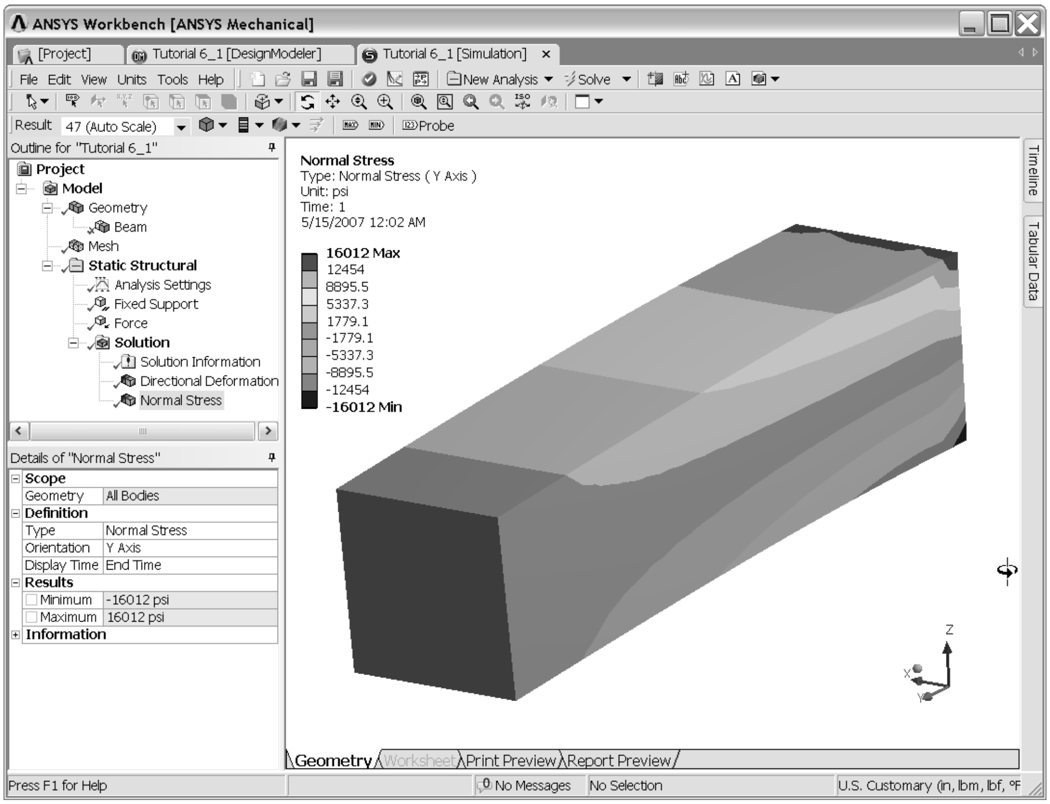


Step 11 – Save the analysis. Click on the floppy disk icon in the toolbar and click the **Save** button in the dialog box that appears next. The file name will be automatically entered as **Tutorial 6_1.dsdb** and the directory will be the same one you selected when you saved the DesignModeler file.

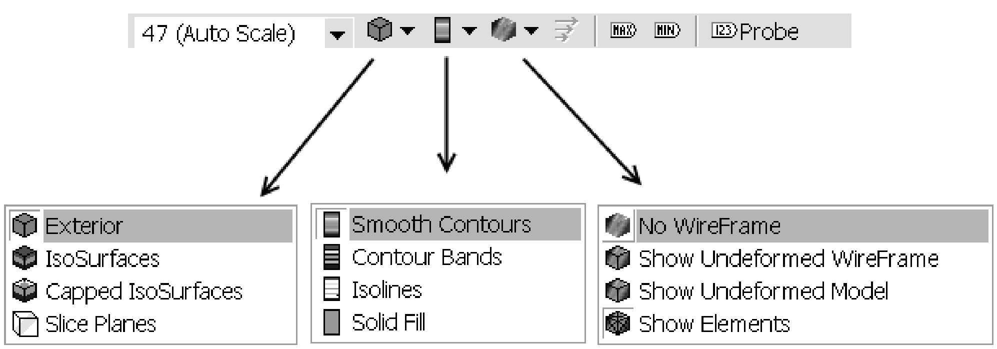
Step 12 – Review the results. When the solution is complete, click on one of the solution quantities you requested such as **Directional Deformation** or **Normal Stress** to display those quantities as contour plots. For example, the figure below shows the contour plot of the normal stress in the Y 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.

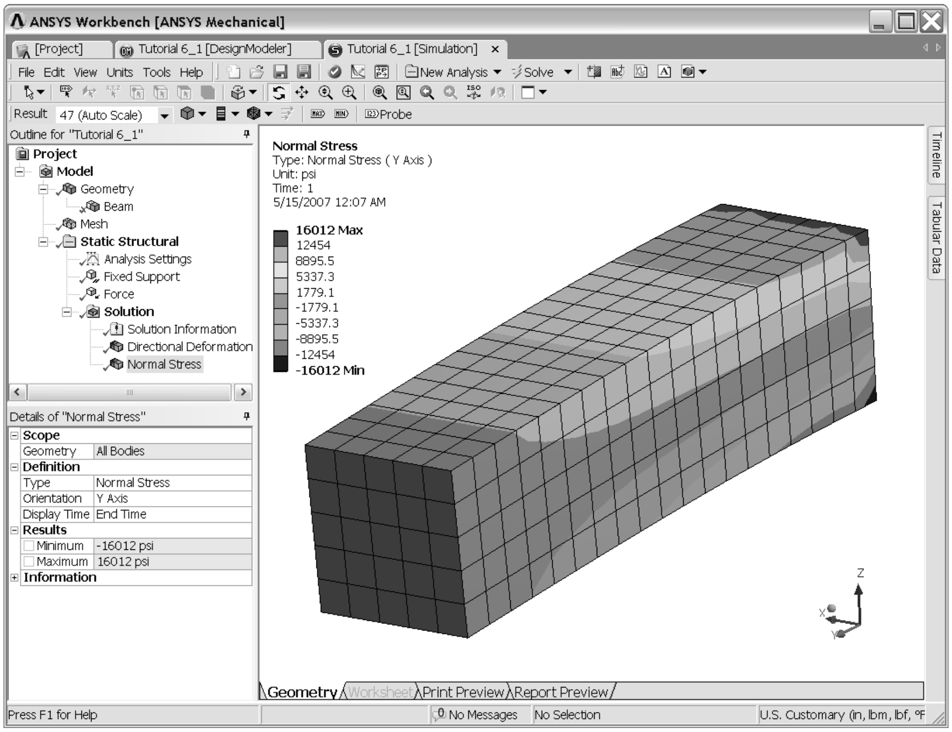


Step 12 (continued) – The displayed contour plots can be modified in several ways in order to better represent the results of the analysis. The modifications are performed from three pull-down menus as shown below.

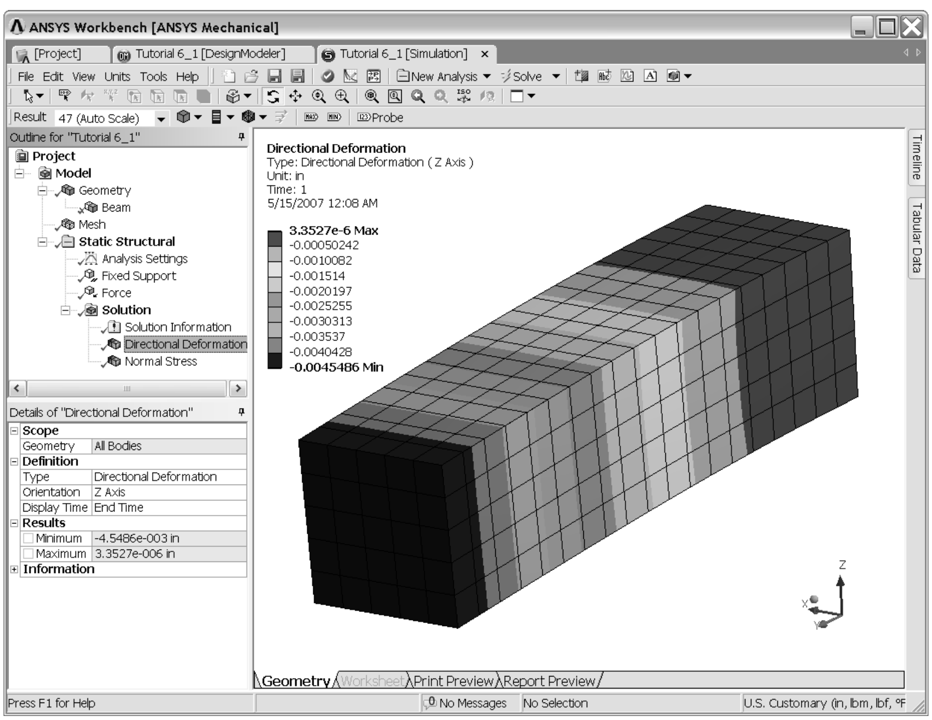


The previous display is set to display **Exterior**, **Contour bands** and **No WireFrame**.

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

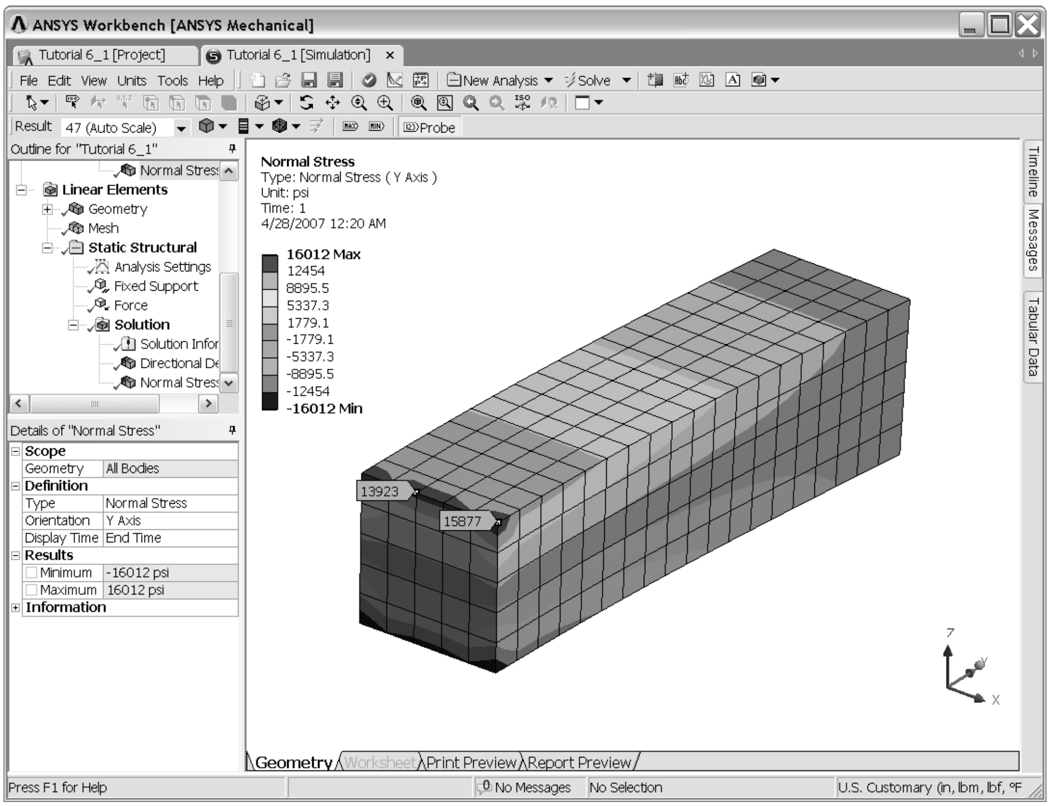


Step 12 (continued) – Click on **Directional Deformation**. The resulting display is shown below.



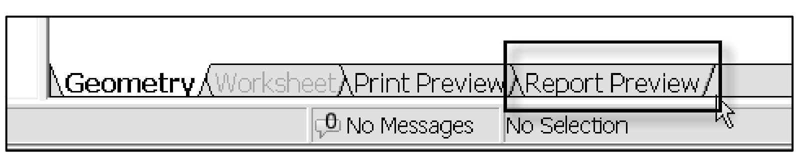
Step 12 (continued) – Note that the maximum and minimum values displayed are the range values for the contours. For example, on the **Normal Stress** plot, the red contours represent stress values ranging from 12,454. PSI to 16,012. PSI. In order to get a more precise value, use the Probe tool:

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

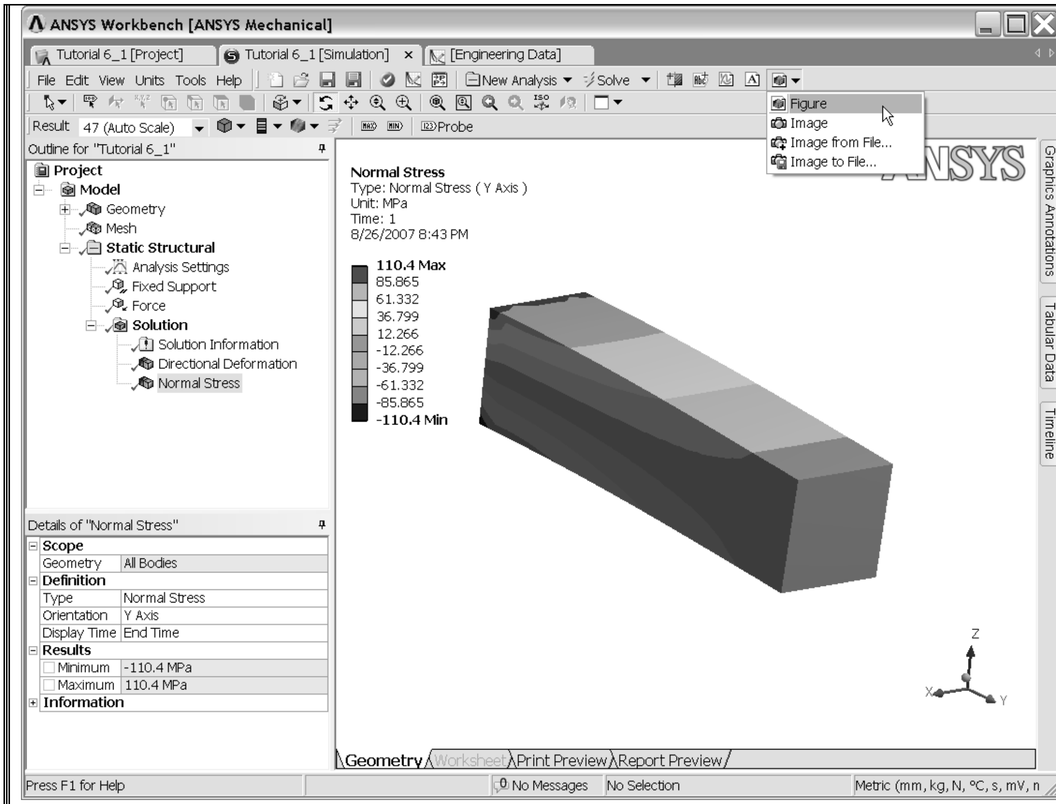
Step 13 (continued) – The default report includes the calculated results in tabular form but deformed shapes and contour plots are not automatically included in the report. Below is an example of tabular results.

TABLE 10
Model > Static Structural > Solution > Results

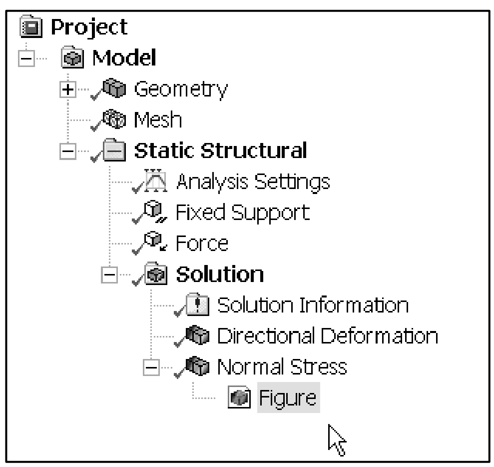
Object Name	*Directional Deformation*	*Normal Stress*
State	Solved	
Scope		
Geometry	All Bodies	
Definition		
Type	Directional Deformation	Normal Stress
Orientation	Z Axis	Y Axis
Display Time	End Time	
Results		
Minimum	-0.11553 mm	-110.4 MPa
Maximum	8.5159e-005 mm	110.4 MPa
Information		
Time	1. s	
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 – Save the project as ***Tutorial 6_1***.

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%

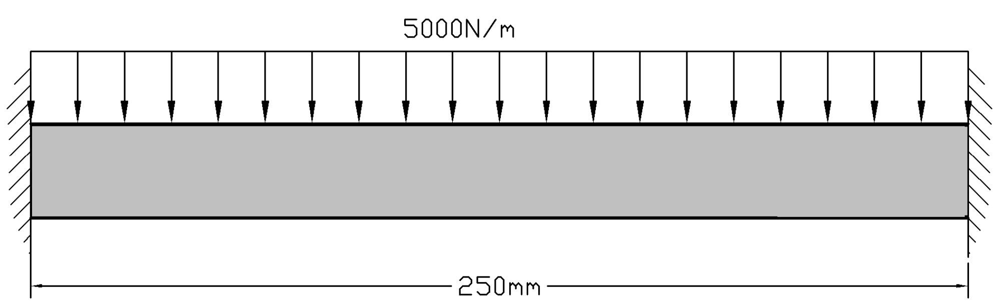
The primary reason for the poor correlation of stress values was due to the boundary condition occurring at the same location as our maximum stress values. However, our goal during this chapter was to become familiar with the steps needed to get a very simply job to run, and not to be concerned with 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

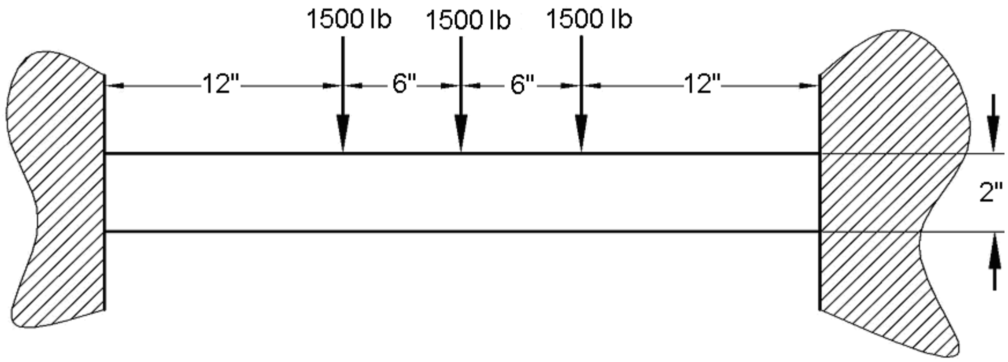
[1] Popov, E. P., Mechanics of Materials, 2^{nd} Edition, Prentice-Hall, Englewood Cliffs NJ, 1976

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).