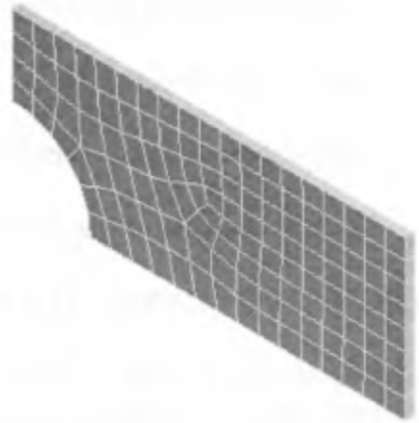# Chapter 4

ANSYS Mechanical I

4-1   OVERVIEW

ANSYS Workbench provides tools for the user to analyze the behavior of **Electric**, **Fluid**, **Magnetic**, **Mechanical**, **Thermal** systems and problems where more than one physical behavior is considered. In this Chapter we consider:

   ◆ Stress response of a plate with a central hole.

   ◆ Stress response of a plate with various FEM mesh densities.

   ◆ The use of convergence criteria for controlling solution accuracy.

4-2   INTRODUCTION

Evaluating the response of a mechanical part or system in Workbench involves accessing the Geometry, assigning the Materials, applying the Loadings and Displacement Boundary Conditions, Solving the system Equations, Reviewing/Reporting the Results, and Updating the model if desired. An outline of this process is shown below.

1.   **Create and Attach problem Geometry.**

2.   **Assign Materials.**

3.   **Establish Contact Conditions if applicable.**

4.   **Preview the FEM Mesh and set up Mesh Controls if desired.**

5.   **Apply Loadings and Displacement Boundary Conditions.**

6.   **Select Results to be computed and displayed.**

7.   **Solve the system governing equations.**

8.   **Review the Results.**

9.   **Set problem Parameters if desired.**

10.   **Create Reports of the response as appropriate.**

11.   **Update the CAD model if necessary.**

To develop confidence in the process we start in Tutorial 4A by solving a simple structural static mechanical response problem. In this problem we can check the maximum stress result separately by a hand calculation.

4-3   TUTORIAL 4A – PLATE WITH CENTRAL CIRCLUAR HOLE

In this tutorial we will use **ANSYS Static Structural Analysis** to compute the maximum deflection and stress in a thin **steel** plate with a central hole. Its dimensions are **1000 mm long, 400 mm high** and **10 mm thick**. The central circular hole is **200 mm in diameter** as shown in the figure below. The plate is loaded in the long direction by a tensile force of **100 kN**.
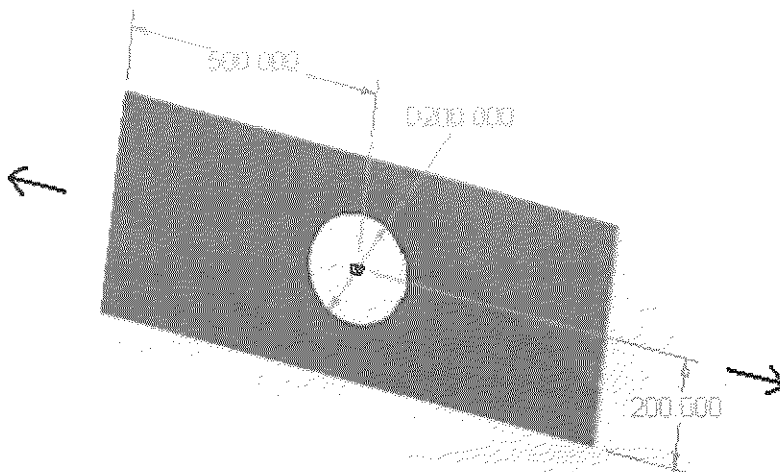

Figure 4-1 Thin plate with central hole; dimensions in mm.

First we need a **solid model of the plate,** and this can be created with **DesignModeler** or **another solid modeling system.** Since the central horizontal and vertical axes of the plate are on planes of plate geometry symmetry as well as loading symmetry, we need analyze only a quadrant of the part to obtain the stress distribution.

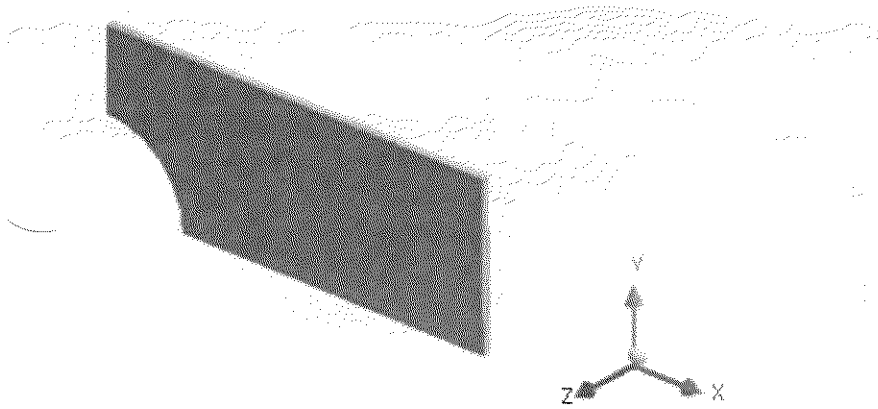Use your solid modeler to trim the model to a quadrant as shown next.

Figure 4-2 Quadrant of plate.

We follow the steps outlined above skipping those not needed in this tutorial.

In working through this and all the other tutorials keep in mind that your **results may vary slightly** due to sight difference in generated meshes.

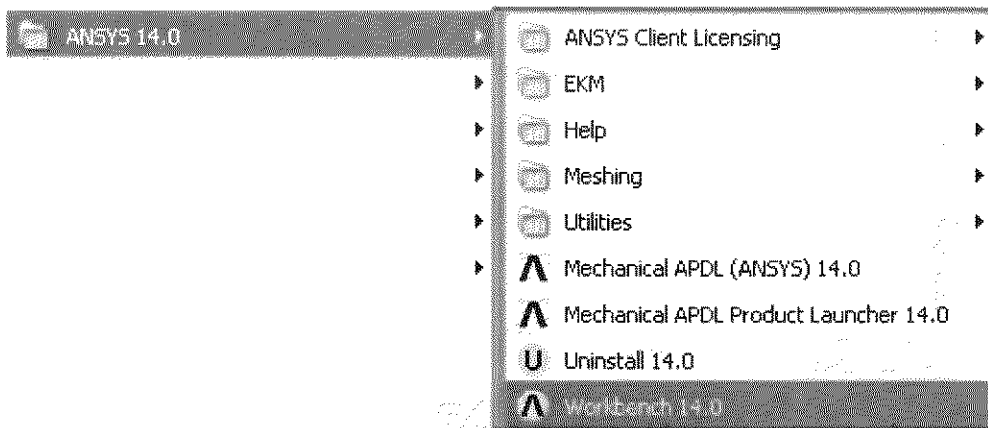1.   **START UP: >** Start **ANSYS Workbench,** begin a **new Project.**

Figure 4-3 Starting ANSYS Workbench in Windows.

Double click Geometry under **Toolbox > Component Systems** to initiate the geometry object in the **Project Schematic.**
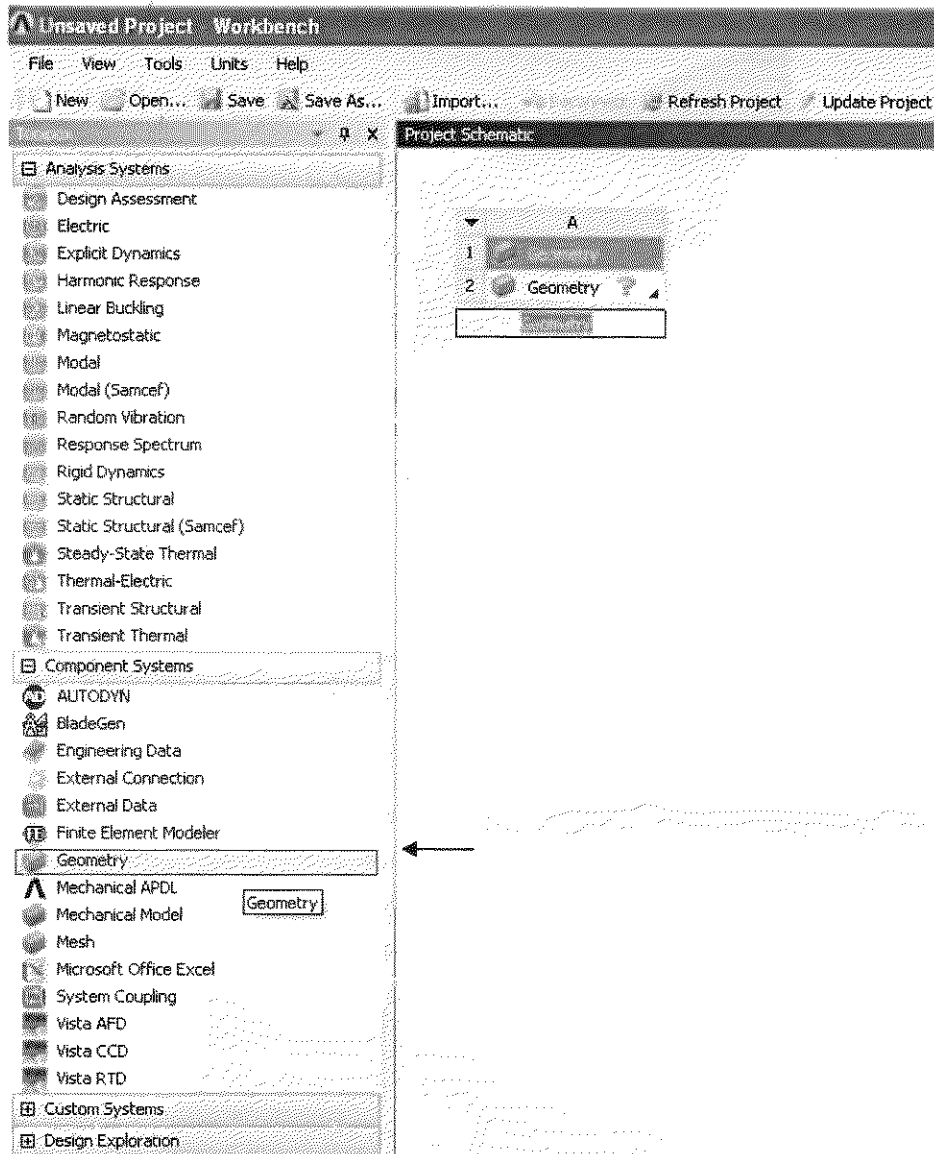
See the next figure.

Figure 4-4 Project Schematic details.

The question mark indicates that **cell A-2** is incomplete.

2.   Select the **small blue triangle** for additional information. Click anywhere in the schematic to close the information box.

Select a folder in a convenient location on your storage device and **use Save As ...** to **name** the project **T4A**. The project title, T4A, is displayed on the header as shown below and the **Workbench** project file **T4A.wbpj** and folder **T4A_files** are created in the selected workspace. Take a minute to find and verify this.

There are a number of ways to create and access geometry for your project. We discuss several, **A., B., C., D.,** and **E.** in what follows.

A. >> Create New Geometry in **Design Modeler: Double click cell A2** to start **Design Modeler.** Select problem **length units** and proceed to create geometry as discussed in Chapters 1 – 3. Save your DM work when you are finished.
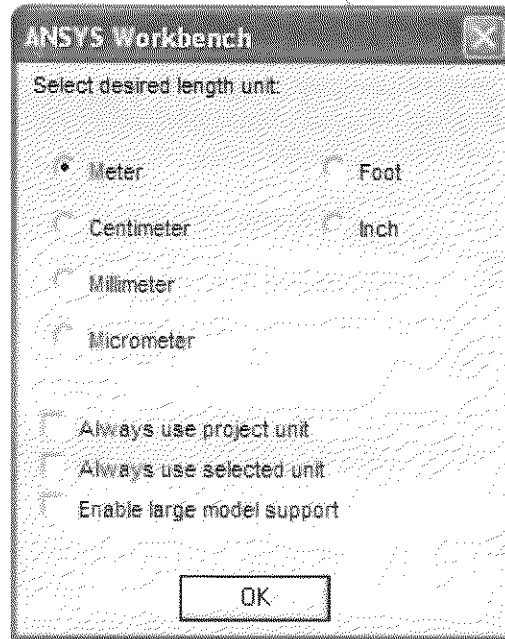

Figure 4-5 A >> Start new geometry in DesignModeler and set units.

B. >> Access Previously Created Geometry in **Design Modeler: Double click cell A2** to start **Design Modeler.**

File > Load DesignModeler Database (See the next figure.)
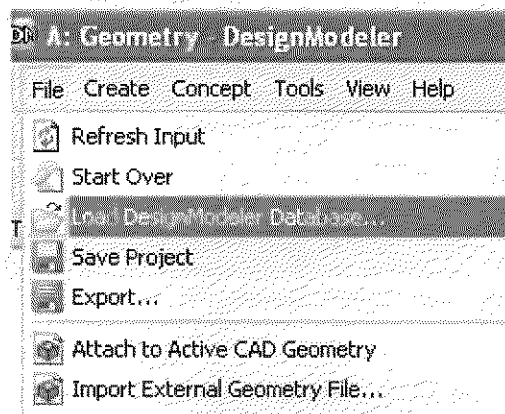

Figure 4-6 B >> Loading existing DM file.

C. >> Access Geometry Previously Created and stored using **another solid modeler: (CATIA, Pro/ENGINEER, SolidWorks, etc)   Double click cell A2** to start **Design Modeler.**

File > Import External Geometry File (Options include **IGES** and **STEP** formats)
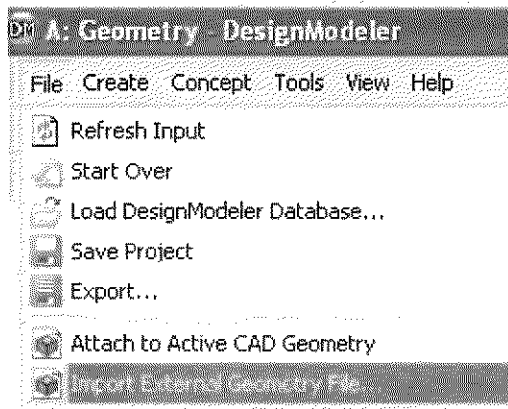(See the next figure.)

Figure 4-7 BC >> Importing from an alternate format.

This process starts the alternate solid modeler, loads that modeler's file, transfers the geometry to DesignModeler, and closes the alternate solid modeler.

D. >> Access Geometry in Active CAD system: Double click cell A2 to start **Design Modeler.**

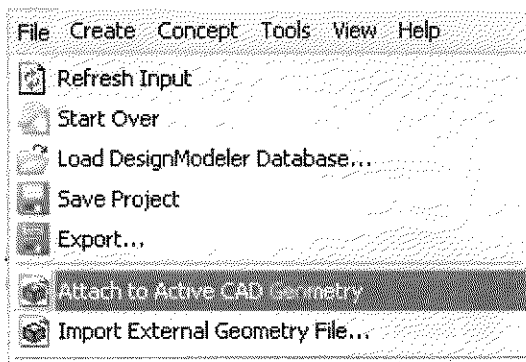**File > Attach to Active CAD Geometry**

Figure 4-8 D >> Attach to Active CAD Geometry.

E. >> Start Workbench from your **alternate Solid Modeler.** An example of this using **Pro/ENGINEER** is shown below. Note the Pro/E icon is shown in cell A2 in the Workbench Project Schematic.

Double click cell A2 to start **DesignModeler**, then use **Generate** Generate to attach the Pro/E geometry.
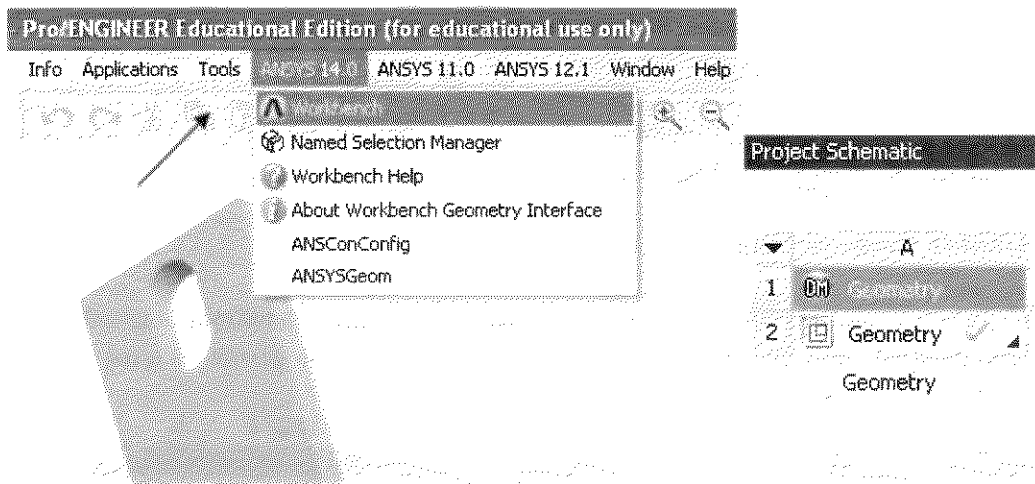
Figure 4-9 E >> Starting Workbench from Pro/ENGINEER.

3. Getting **back to Tutorial 4A**, with the geometry attached, **Double Click Toolbox > Analysis Systems > Static Structural** to add the analysis to the **Project Schematic.** (Or **drag** it from the Analysis Systems column to the Schematic.)
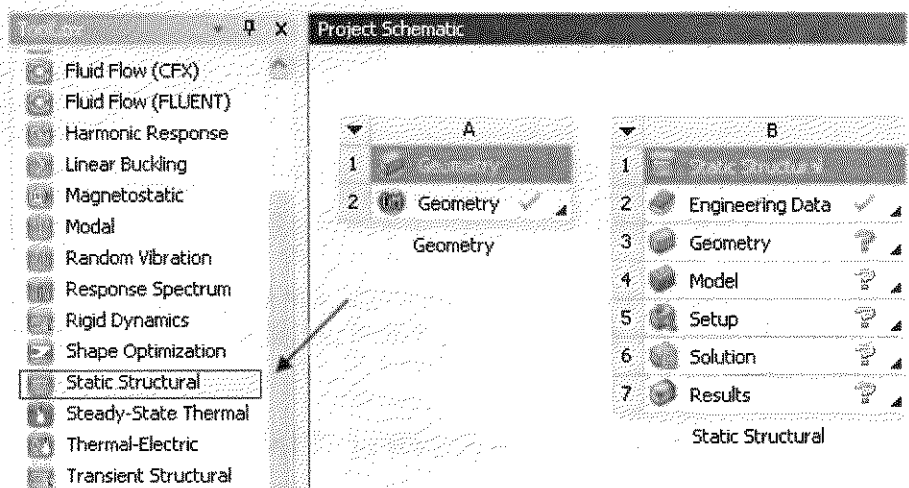
Figure 4-10 Adding Static Structural Analysis from the Toolbox.

4. To **share** the geometry, **Left click** on **DM Geometry** in **cell A2** and **drag** it to **Static Structural cell B3.**
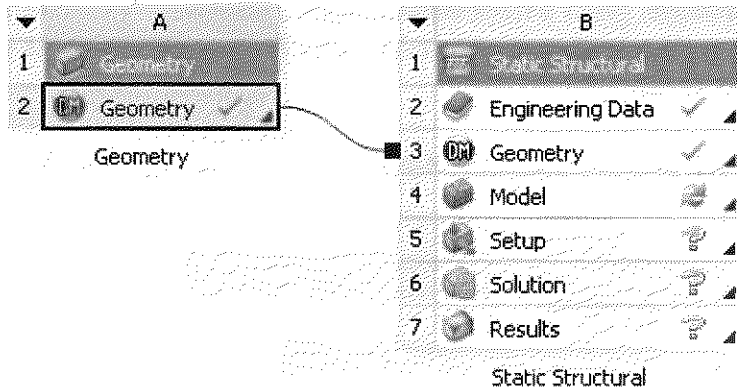
Figure 4-11 Sharing geometry.

5.   Double Click on **Model** in **cell B4**.

The Workbench display now shows the **Static Structural Analysis** that is associated with this **Project**, and the tree structure on the left contains project items that include **Model**, **Geometry**, and **Mesh**. See the figure below.
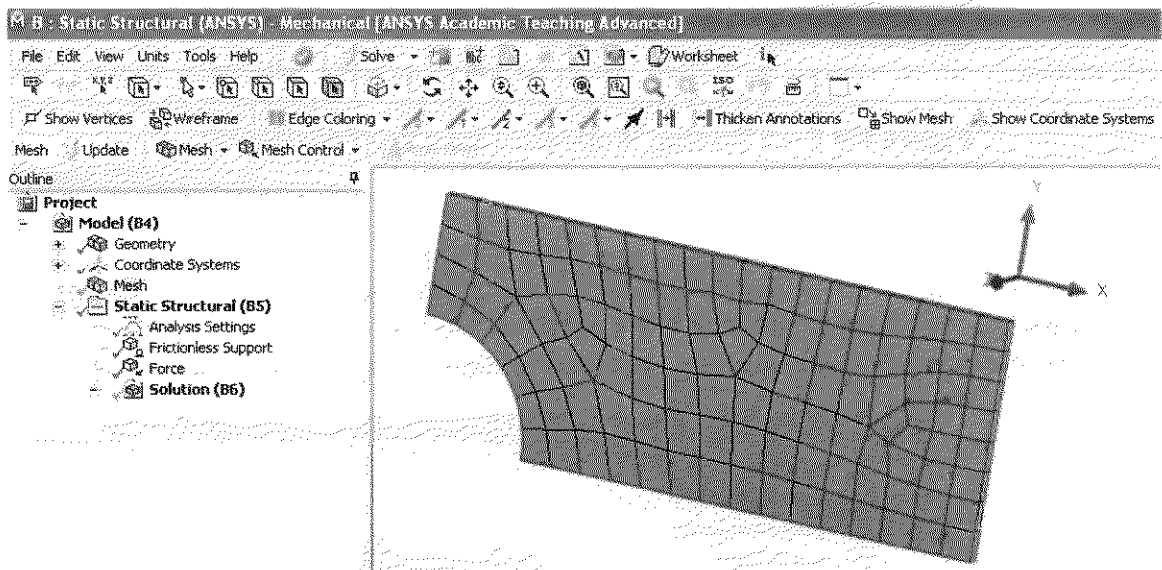

Figure 4-12 Plate quadrant in Static Structural Mechanical module.

Since the geometry was created using mm, the length units for the simulation should be mm also. **Check the units** using the **Units** pull-down menu. **Units > Metric (mm, kg, N, s, mV, mA)**

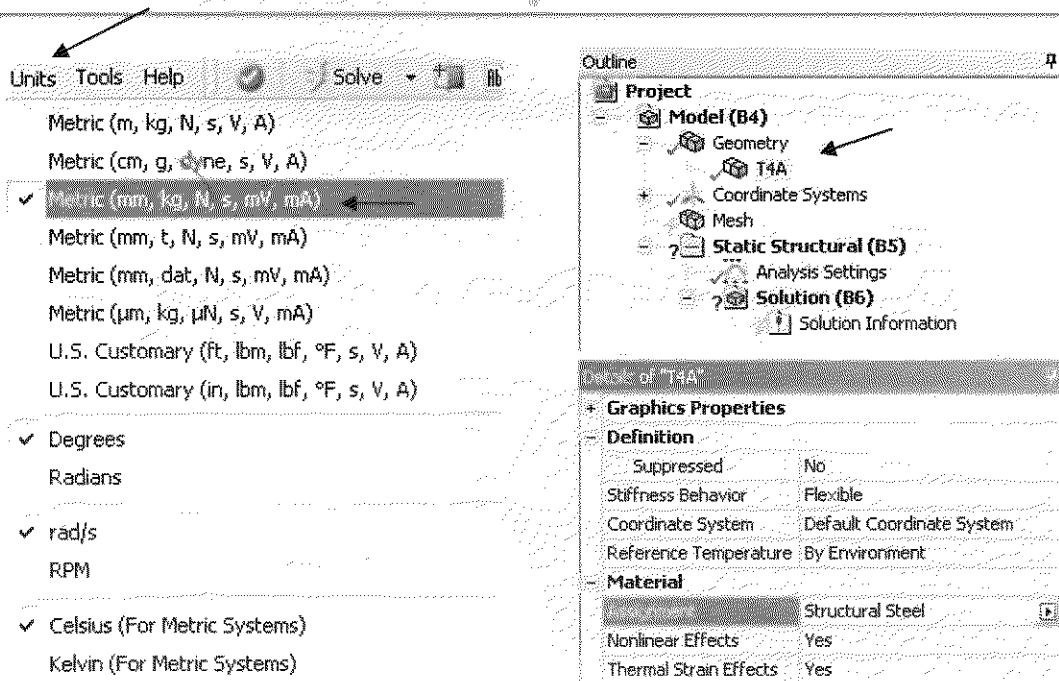**6.   Highlight Material** in the **Details of "T4A"** window.

Figure 4-13 Check Units and Material assignment.

To return to the **Project Schematic, Select** T4A – **Workbench** from the programs running shown in the taskbar at the bottom of the screen.

Figure 4-14 Workbench elements shown in the Taskbar.

7.   Double click **Engineering Data, cell B2** to view the **Material Properties Data for Structural Steel**, the **default material** that has been assigned to this part.
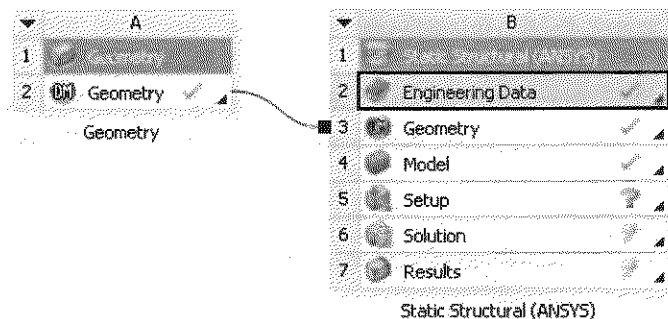

Figure 4-15 Project Schematic.

Be sure **View > Properties** and **Outline** are checked. Properties for the **default** material **Structural Steel** are shown.
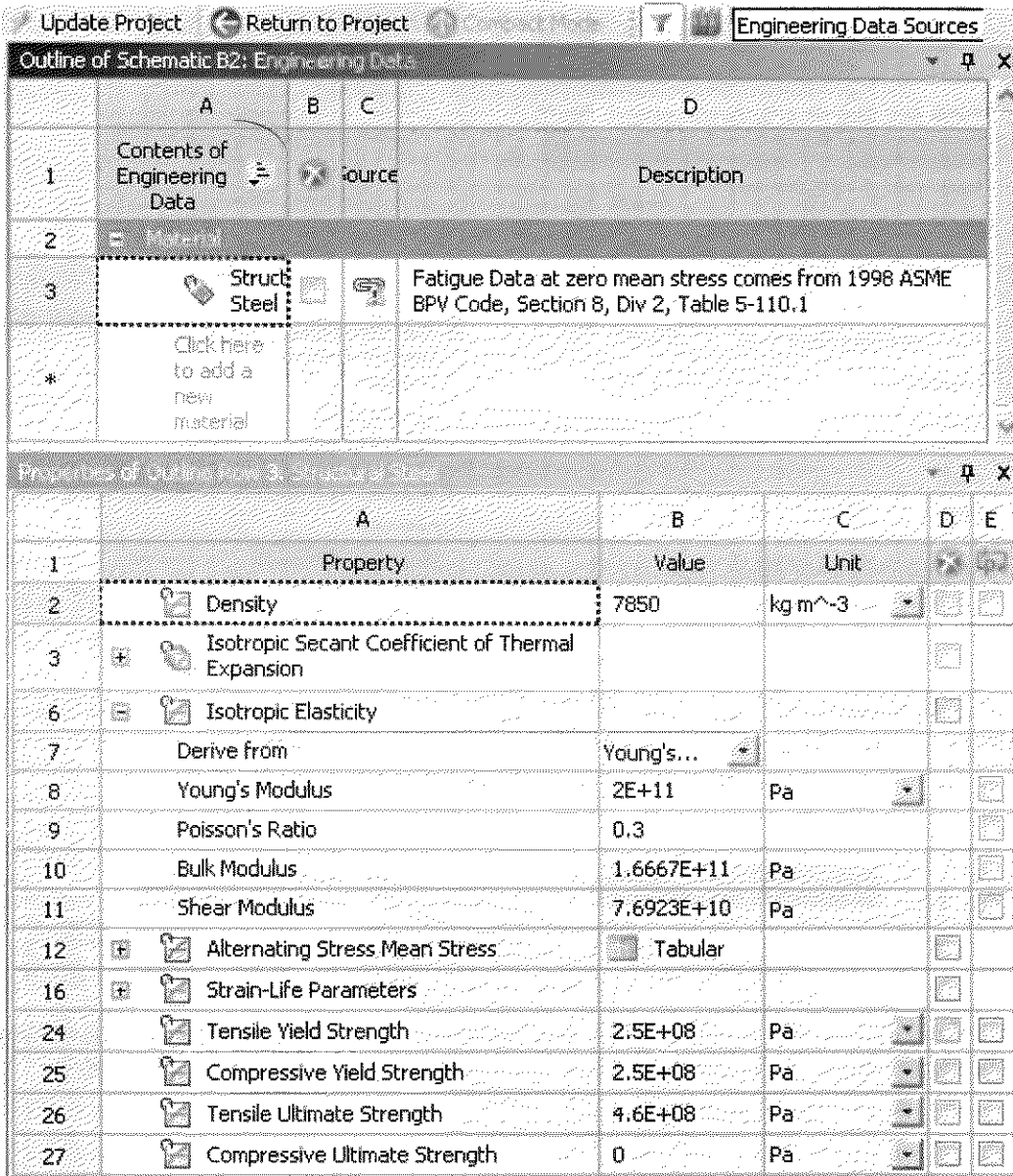
Figure 4-16 Material properties for structural steel.

Note that structural steel has **Tensile** and **Compressive Yield Strengths** of **250 MPa**. and that because it's unknown, zero has been assigned to the **Compressive Ultimate Strength**.

8.   Select Return to Project (Top of screen) ⊙ Return to Project .

The Mesh item in the project tree has a lighting bolt symbol next to it indicating that the finite element mesh for this simulation has not yet been created. Workbench simulation will automatically develop a finite element mesh appropriate to the problem.

9.   MESH: Right click **Mesh** and select **Generate Mesh**

The default mesh that is created consists of a little over one hundred **three-dimensional 8 or 20 node brick elements** as shown.
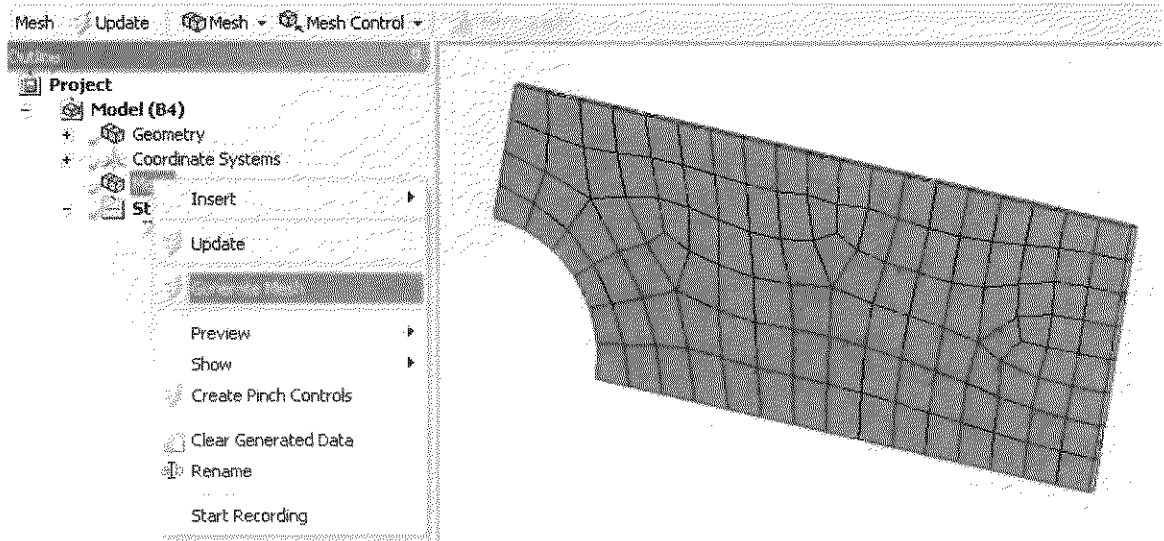
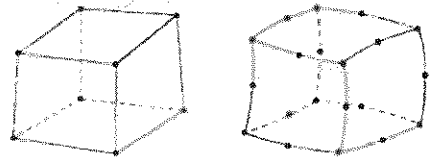Mesh   Update   Mesh ▾   Mesh Control ▾

Figure 4-17 Meshing the geometry.

10.   Select **Static Structural (B5)** from the Outline tree structure (next figure). The Environment displays loadings and boundary condition options available for this analysis.

Since only the upper half of the plate is being analyzed, apply **a Force of 50 kN** to the right end surface.

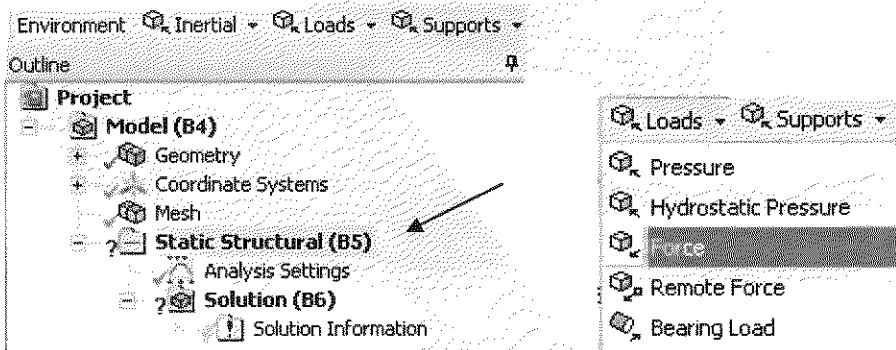11.   **APPLY LOADINGS:** Click **Environment > Loads > Force**.

Environment   Inertial ▾   Loads ▾   Supports ▾

Outline

Figure 4-18 Structural loads menu.

Be sure that the **Face** selection filter is highlighted 🔳 and **click on the area** on the right end of the solid model.

12.   **Geometry > Apply** (Note: It's easy to forget this step.)

13.   **Define** (the force) **by** (X Y Z) **Components, X Component = 50 kN**
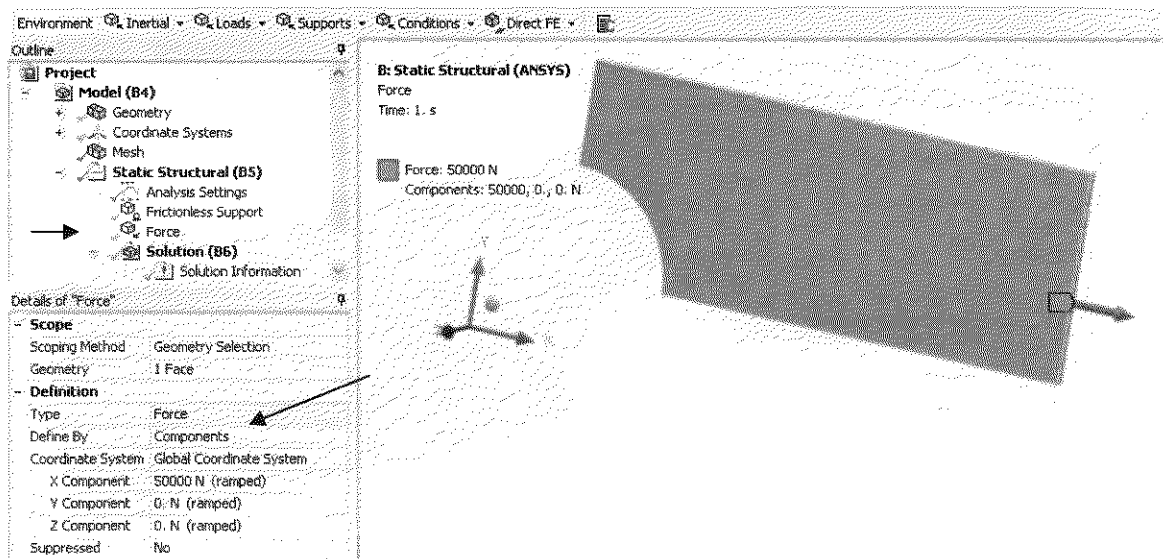

Figure 4-19 Tensile loading.

Next apply the **displacement constraints**; rotate the plate so that you can see the bottom, and small end. These are surfaces on planes of symmetry and no point on these surfaces can move across the plane of symmetry. Symmetry requires that we constrain the displacements perpendicular to these surfaces. We can use the **Frictionless Support** condition to do that. We also restrain the **back surface** also so as to prevent rigid body motion in a direction perpendicular to the plane of the plate. See the figure below.
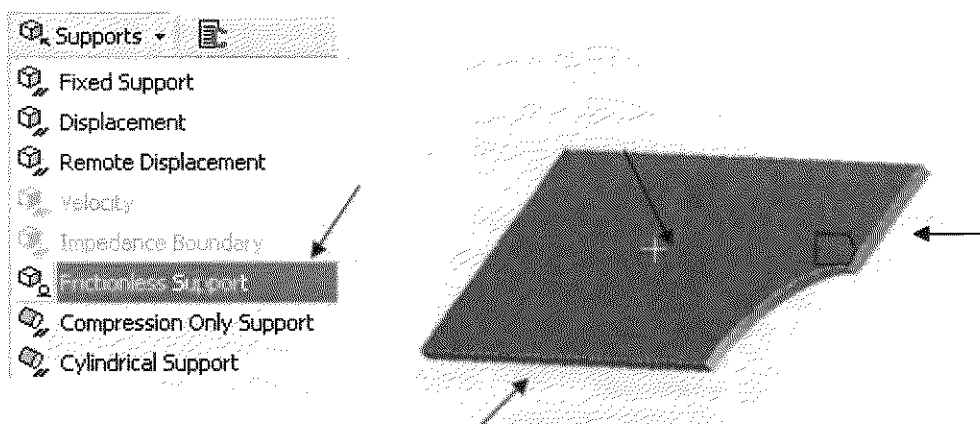

Figure 4-20 Displacement constraints.

14.   **Environment > Supports > Frictionless Support**

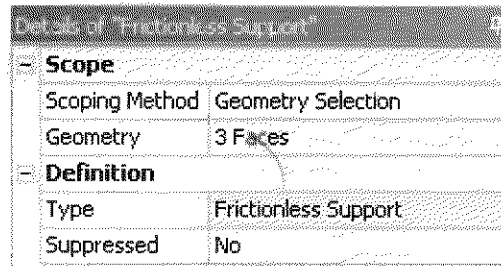15.   **Ctrl > Left Click** to select the **three** surfaces **> Apply.**

Figure 4-21 Frictionless Support constraints.

Check your work by clicking on each of the items under Environment in the model tree to be sure the loadings and constraints are applied as desired. Or click **Static Structural** to see all of the constraints you have applied.

If you find something wrong, just highlight the item in the model tree and edit it in the 'Details' box to correct the error, or Right Click, delete the item from the outline tree and apply the condition again.

(Note that the back face is not really a plane of symmetry. To be absolutely correct we should have sliced down through the 10 mm thickness and analyzed an octant instead of a quadrant. However since the 10 mm dimension is so small in comparison with the other dimensions, there is little error in the approach we used. Try it both ways to see.)
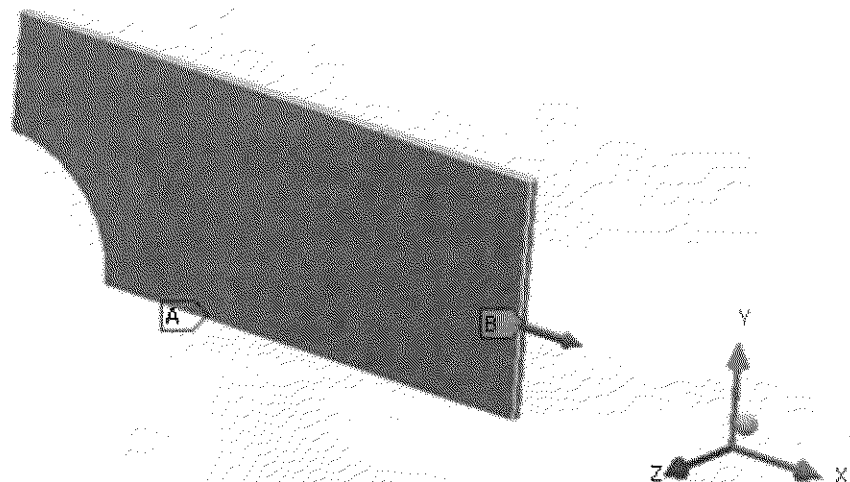

Figure 4-22 Structural Static 'Environment' settings.

To complete the model building process we need to specify what **result** quantity or quantities we would like to have calculated and displayed. In this problem we are most interested in the **stress** and **deformation** in the **X Axis direction**.
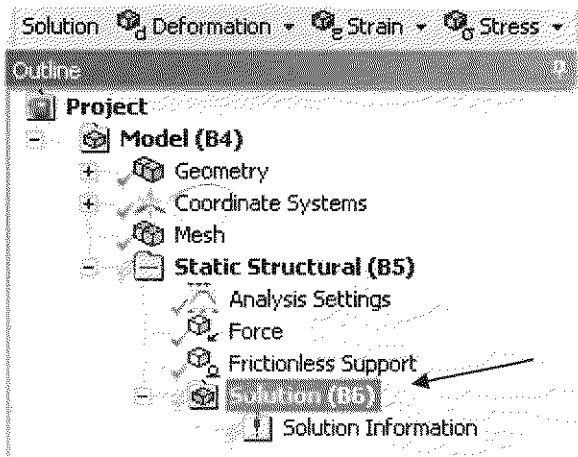
Figure 4-23 Solution result options menu.

16.   Solution > Stress > Normal > Orientation > X Axis

Also select **Deformation** and insert the **Directional Deformation** in the **X Axis**.
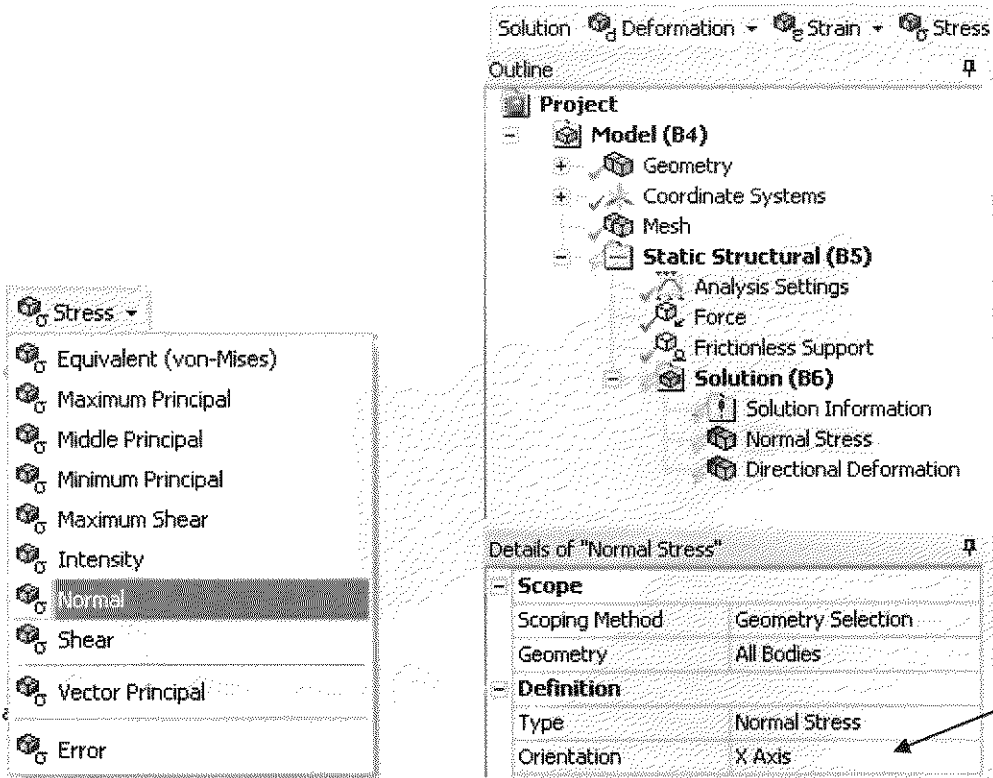

Figure 4-24 Select X-direction normal stress output.

Notice that Solution, Normal Stress, and Directional Displacement items have lightning bolt indicators meaning that we need to highlight one or the other and select ⚡Solve to complete the simulation solution.

17.   Solve ⚡Solve

The solution progress is shown in the **ANSYS Workbench Solution Status** window.
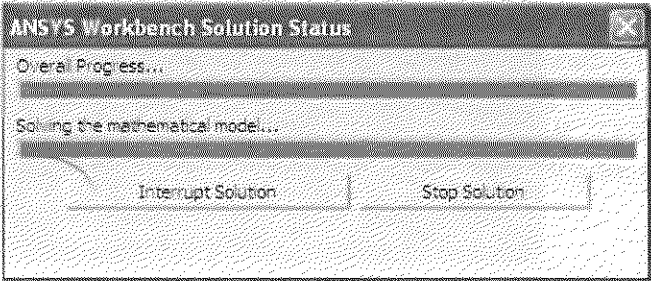
Figure 4-25 Solution status.

When the solution is found, click on the normal stress to view the computed stress results.

18.   Solution > Stress > Normal Stress (In the graphics window - Right click > View > Front; use the pull-down menu to **Show Elements**.)
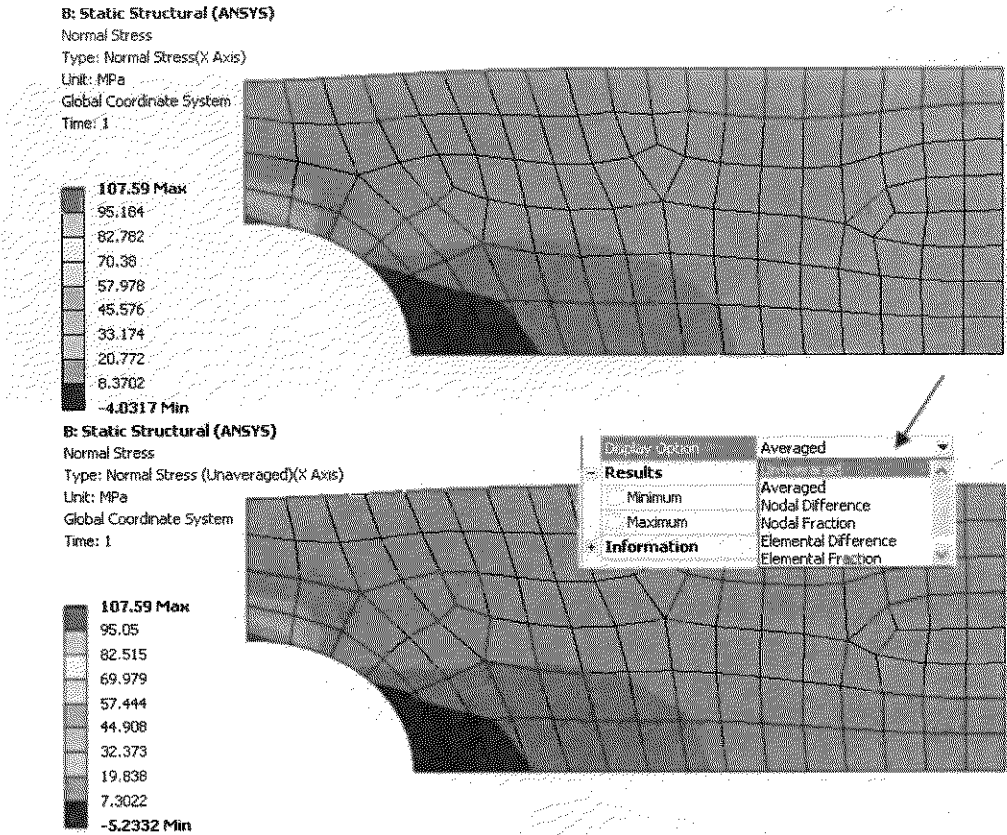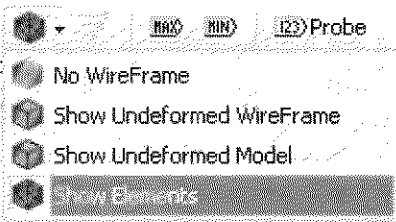
Figure 4-26 X direction normal stress; **Averaged** Display, **Unaveraged** Display.

The solution for the X direction normal stress shows a **maximum value** of **107 MPa**. This value is well below the material yield stress of 250 MPa, so the elastic solution that we have performed is valid.

To check this result, find the theoretical stress concentration factor for this problem in a text or reference book or from a web site. For the geometry of this example we find $K_t = 2.17$. We can compute the maximum stress using (K_t)(load)/(net cross sectional area). Using the net section of the whole bar and total loading of 100 kN we obtain:

$$\sigma_{x\,MAX} = 2.17 * F * /[(0.4 - 0.2) * 0.01] = 108.5 MPa$$

The computed maximum value is **107.6 MPa** which is less than **one percent error** (assuming that the published value of K_t is correct, that is).

If we list the solution information we find that 113 **SOLID186** elements were used in the model. A search of the ANSYS Mechanical Help system shows these to be **20 node brick elements**.

⬛ **Solution (B6)**   113 SOLID186
  ⬛ Solution Information

The **maximum deformation** in the X direction is about **0.083 mm**.

B: Static Structural (ANSYS)
Directional Deformation
Type: Directional Deformation(X Axis)
Unit: mm
Global Coordinate System
Time: 1

0.083109 Max
0.073875
0.064641
0.055406
0.046172
0.036937
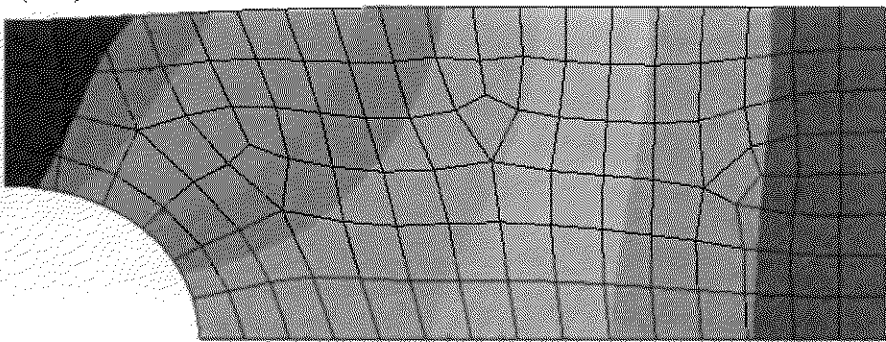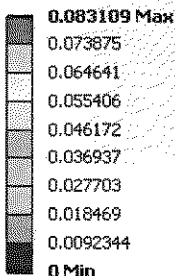0.027703
0.018469
0.0092344
0 Min

Figure 4-27 X direction deformation.

Before we move on, let's compute and display the stress **error estimate** for this problem. Insert a stress error in the solution item in the project tree.

19. **Solution > Stress > Error > Solve**   ⌁ Solve

Computed results are shown in the next figure.

B: Static Structural (ANSYS)
Structural Error
Type: Structural Error
Unit: mJ
Time: 1

0.042548 Max
0.03782
0.033093
0.028365
0.023638
0.01891
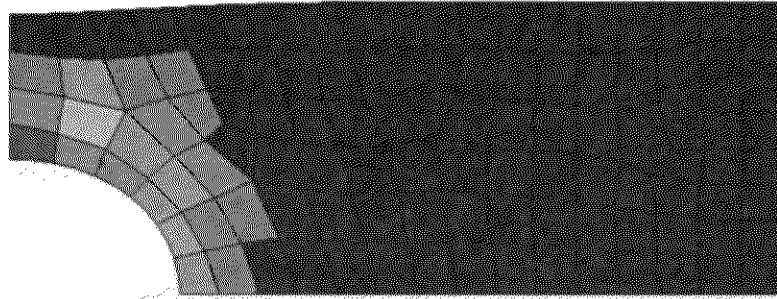0.014183
0.0094555
0.0047281
5.8971e-7 Min

Figure 4-28 Computed structural error estimates.

The error estimates shown above can be used to help identify regions of potentially high error in the solution and thus show where the model might benefit from a more refined mesh. These error estimates are used in Workbench automatic adaptive meshing and convergence procedures we discuss later.

For now we note that the Structural Errors shown are error **estimates** based upon the difference between the computed **smoothed (averaged) stress distribution** for the object and the stresses calculated by the finite element method for **each element in the mesh (unaveraged)**. The data are expressed in an energy format (energy is nonnegative) so that the sign of the difference between the estimated stress and the computed stress does not influence the results. The estimated error is displayed for each element.

For accurate solutions the difference between the smoothed stress and the element stress is small or zero, so small values of Structural Error are good. We know the above solution is accurate because we compared our result with tabulated results. The small error estimates shown above reflect this, as do the Averaged and Unaveraged results plotted above.

If you want the computed results in a **different system of units**, just change the units in Mechanical after you've computed the results. In the current problem, for example:

Max stress Sx = 107.5 MPa = **16,504 psi**
Max deflection = 0.083 mm = **0.0033 inches**

Units

✔ U. S. Customary (in, Lm, lbf, °F, s, V, A)

To **improve the accuracy** we'll generate a model with more elements. There are many ways to control the mesh in Workbench. In what follows we use one of the sizing options.

20.   **Mesh > Details of Mesh > Relevance Center > Medium** (Coarse is the default.) **Solve** again.
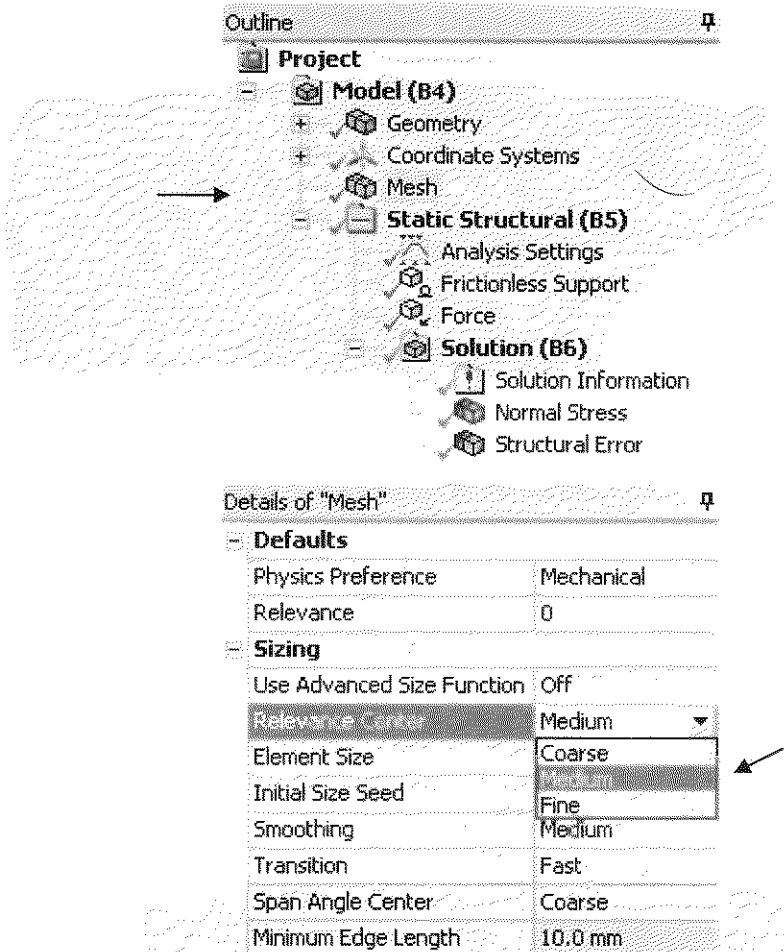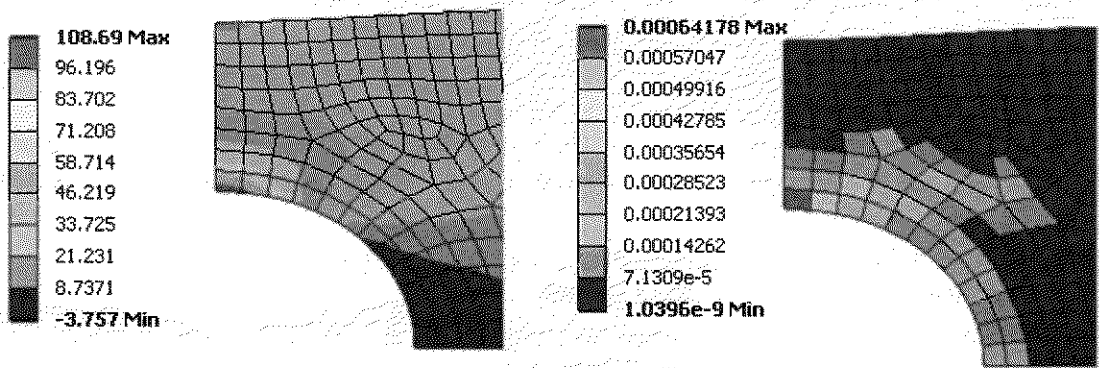
Figure 4-29 Relevance Center mesh settings.

Figure 4-30 Stress and error estimates based on a medium mesh.

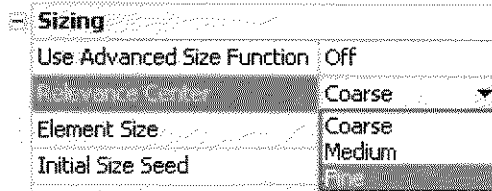21.   **Repeat** this process using the **Fine Mesh** setting.
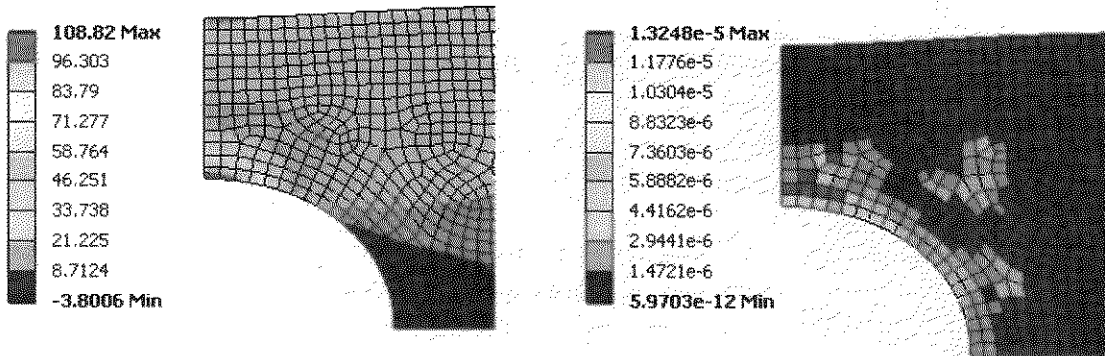
Figure 4-31 Relevance Center Fine mesh settings.

Figure 4-32 Stress and error estimates based on the fine mesh.

22. **Let's compute another solution** for a **mesh** that is **coarser** than the first one. Set the **Element Size = 50 mm** and remesh. We get the results shown below.
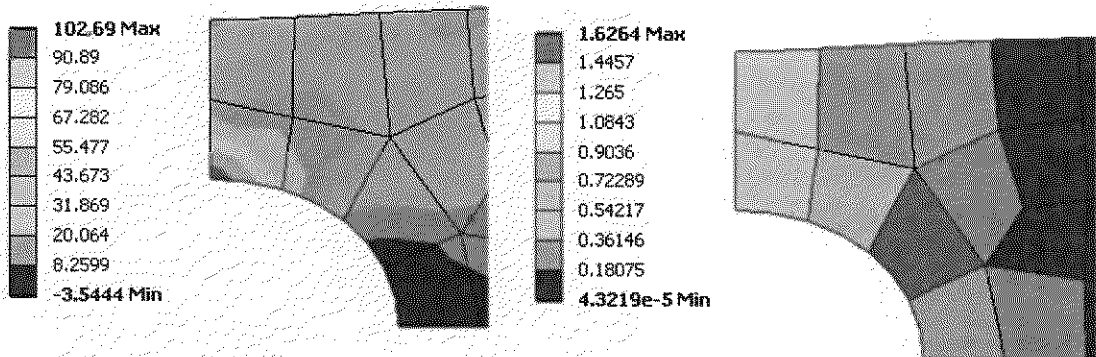

Figure 4-33 Stress and error estimates based a very coarse mesh.

These results are summarized in the table below. Note that with the finer meshes we obtain a maximum stress of about 109 MPa. With the very coarse mesh we get about 103 MPa., about 6 percent in error.

Table 4A.1 - Results Summary

Number of Elements	Maximum Deformation in X direction mm	Maximum Stress in X direction MPa
36	0.0831	102.7
137	0.0831	107.8
511	0.0831	108.8
3182	0.0831	108.8

It is important to note the **approximate nature** of the **Finite Element Method** and the **convergence** of the **stress results with mesh refinement** as illustrated in the above table. Note that with a coarse mesh you can get results that are **in error,** only about 6 percent in this case, but don't accept results from an initial mesh without question. (Set the element size to 75 mm, and see what you get.)

Also note that the tabulated **displacement results** show no change to three significant figures indicating that the displacements converge more rapidly than the stresses. This is usually the case. The strains (hence stresses) are the spatial directives of the displacement distributions within the object, i.e., $\varepsilon_x = \partial u / \partial x$, etc.

We will do two more experiments before moving on. Return to the **50 mm mesh** and change the **stress plotting option**.

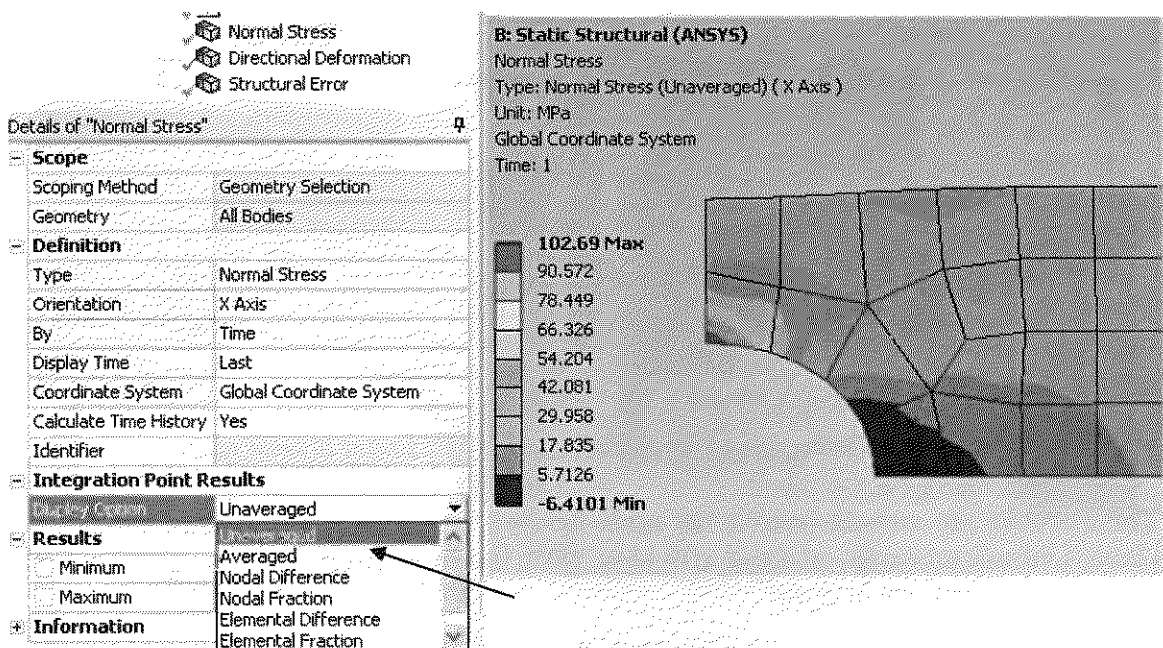23. **Select Normal Stress > Display Option > Unaveraged. Recalculate the solution.**

Figure 4-34 Unaveraged stress plot.

The figure above is not much different from the previous stress plot except there are some small contour discontinuities between elements in the low stress region.

Let's now use a different finite element. In the mesh details box, **drop** the **element midside nodes**.

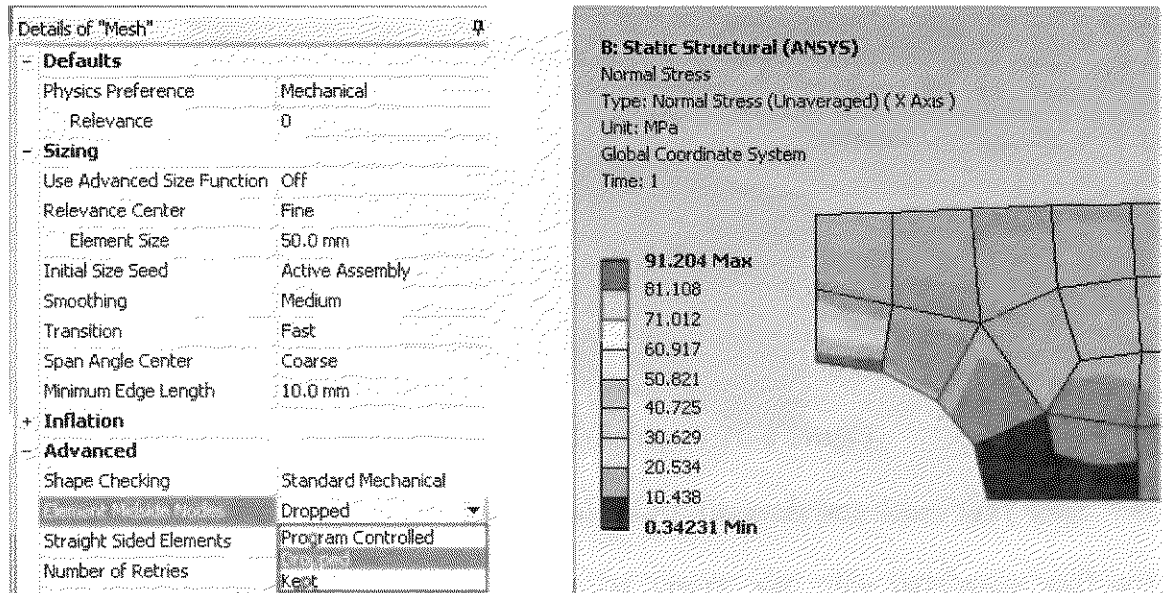24.   **Select Mesh > Advanced > Element Midside Nodes > Dropped**

Figure 4-35 Stress results found using 8-node brick elements.

In the figure above note the maximum stress is only **91 MPa** (a 13 percent error), and the stress contours are discontinuous at many element boundaries. We **should not have a discontinuous stress distribution** in a uniform solid. What's up?
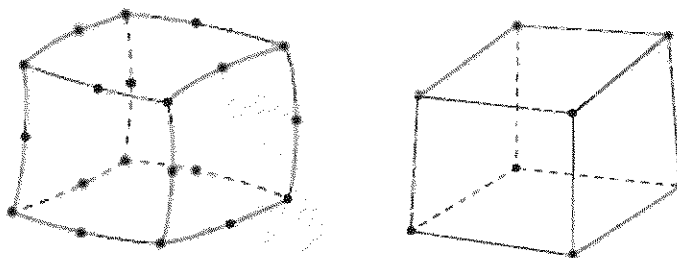

Figure 4-36 20 node and 8 node solid elements.

The previous results were all calculated using brick elements **with midside nodes,** giving **20 nodes per element.** These last results were found using bricks with **nodes only at the corners, 8-node elements.** Higher order elements (those with midside nodes) generally give better results because of the better representation of curved geometry and of the

displacements, strains and stresses within the element. That's what we see here. Note also that if you plot the averaged stress display for the 50 mm coarse mesh you get a misleadingly smooth display, but the results are still in **error by 13 percent**.

At the completion of the solution, the project schematic shows all items checked off in the Static Structural matrix as shown below.
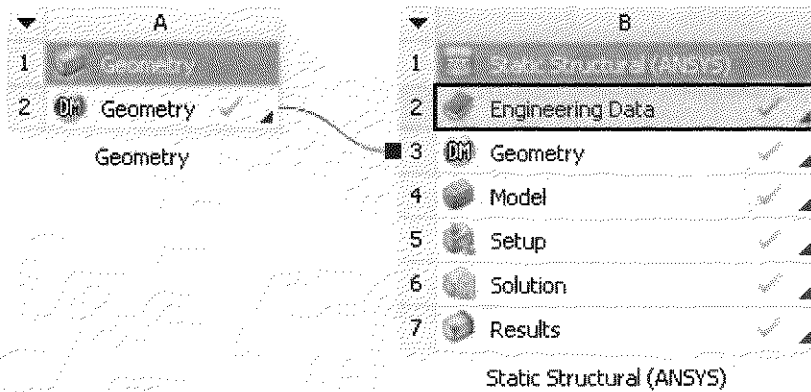
Figure 4-37 Project Schematic after solution.

25.   Save your work and close the T4A Workbench Project.

The files associated with this project are shown in figure below. **T4A.wbpj** is the Workbench project file, and the **T4A_files directory** contains the supporting files as shown.
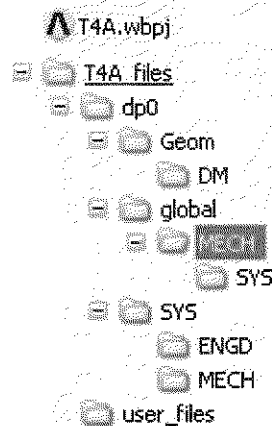
Figure 4-38 Files stored after Workbench Project T4A has been closed.

Take a minute to verify the file structure and note that about 3 MB storage is required for the medium mesh model. More space may be required if intermediate files are created during solution. It may also be costly to move these files around a network as the problem is being solved.