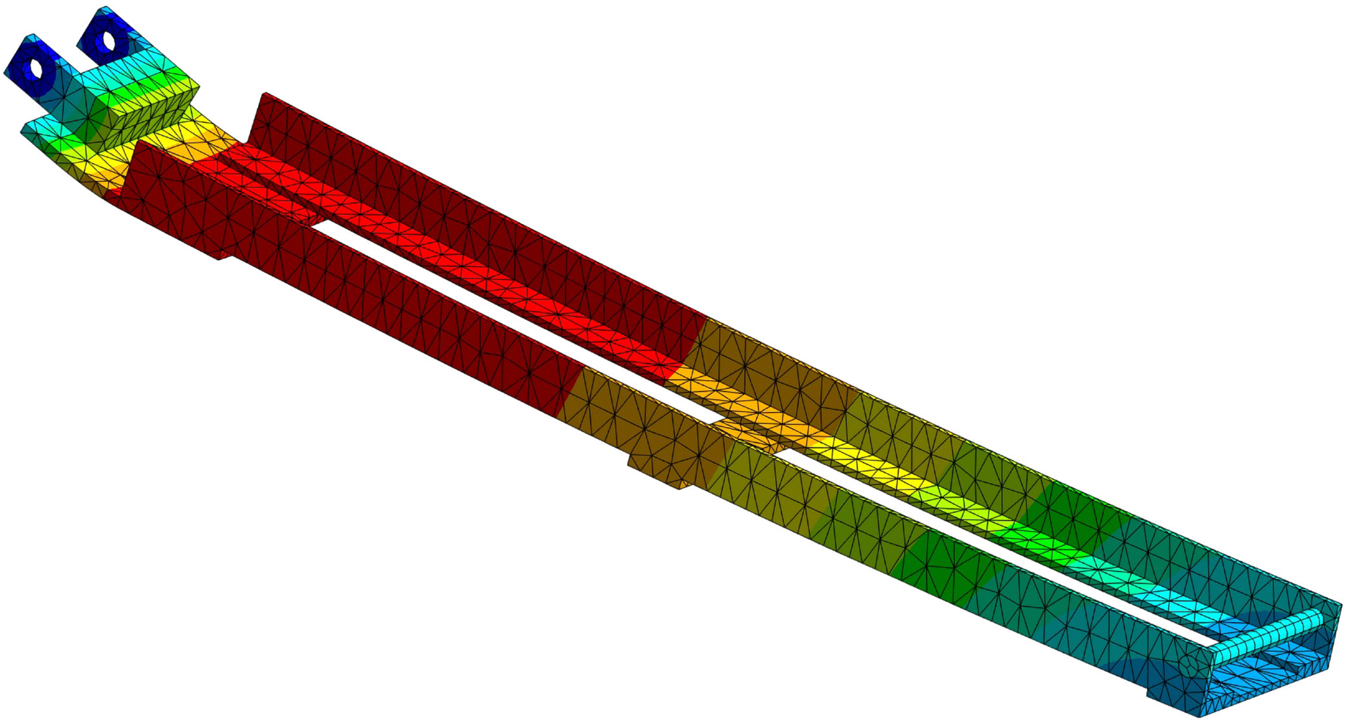
Ansys® Workbench Tutorial Release 2026

Structure & Thermal Analysis Using Ansys® Workbench Release 2026 Environment



Kent L. Lawrence

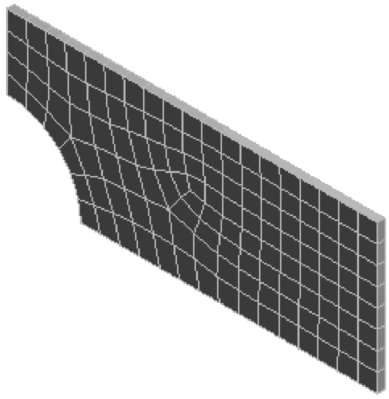
SDC PUBLICATIONS
Better Textbooks. Lower Prices.
www.SDCpublications.com

Visit the following websites to learn more about this book:

SDC PUBLICATIONS amazon.com Google books BARNES&NOBLE

Chapter 1

Ansys Mechanical I



1-1 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 an elastic plate with a central hole.
- ♦Stress response of an elastic plate using various FEM mesh densities.
- ♦The use of convergence criteria for controlling solution accuracy.

1-2 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 of 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 1A by solving a simple structural static mechanical response problem. In this problem we can check the computed maximum stress result separately by a hand calculation.

1-3 TUTORIAL 1A – PLATE WITH CENTRAL CIRCULAR HOLE

In this tutorial we will use the **Ansys Static Structural Analysis** module to compute the maximum deflection and stress in a thin elastic **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**.

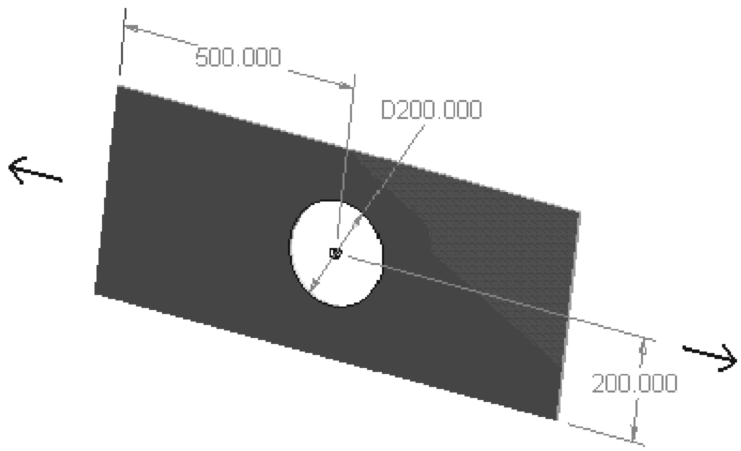


Figure 1-1 Thin plate (10 mm thick) with central hole; dimensions in mm.

First we need a **solid model of the plate**, and this can be created with **DesignModeler, Space Claim,** 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. We use the full thickness for convenience.

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

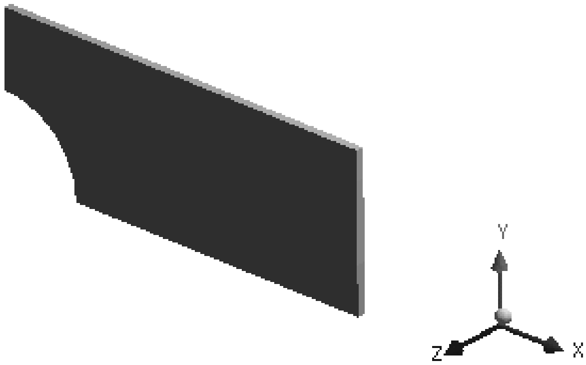


Figure 1-2 Quadrant of plate.

We follow the process 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 a slight difference in generated meshes.

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

Figure 1-3 Starting Ansys Workbench in Windows.

Double click  **Geometry** under **Toolbox** > **Component Systems** to initiate the geometry object in the **Project Schematic,** or just **drag Geometry** into the **Project Schematic** workspace.

See next figure.

Figure 1-4 Project Schematic details.

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

2. Select the **small blue triangle** for the additional information shown. 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 **T1A**. The project title, T1A, is displayed on the header as shown below and the **Workbench** project file **T1A.wbpj** and folder **T1A_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: Right click cell A2** to start **New Design Modeler Geometry.** Design Modeler loads up. Select problem **length units** and proceed to create geometry. Save your Design Modeler work when you are finished.

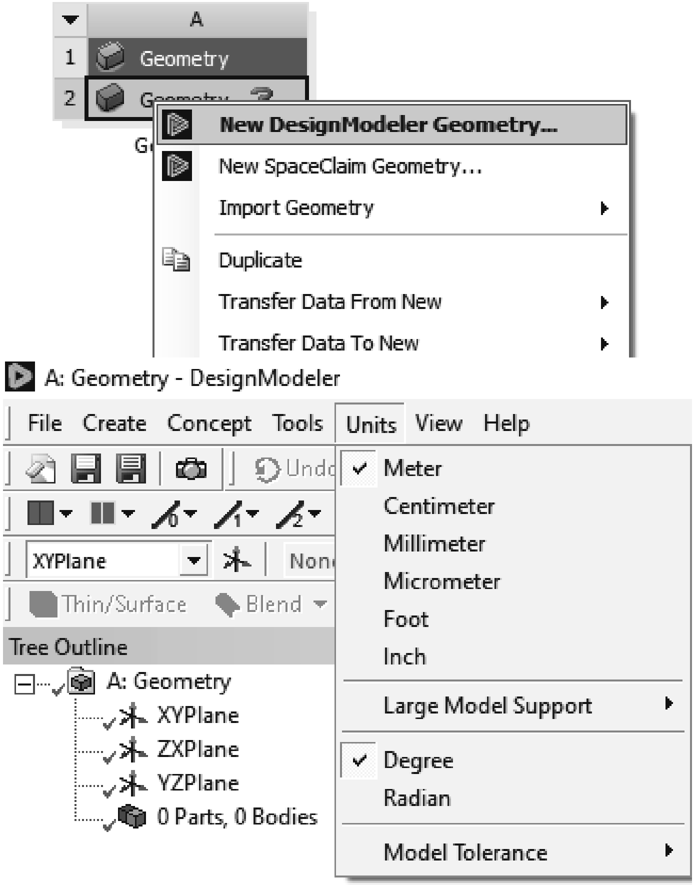


Figure 1-5 Start new geometry in DesignModeler and set units.

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

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

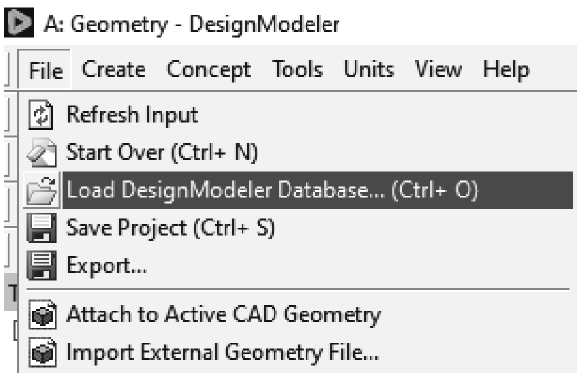


Figure 1-6 Loading existing DM file.

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.)

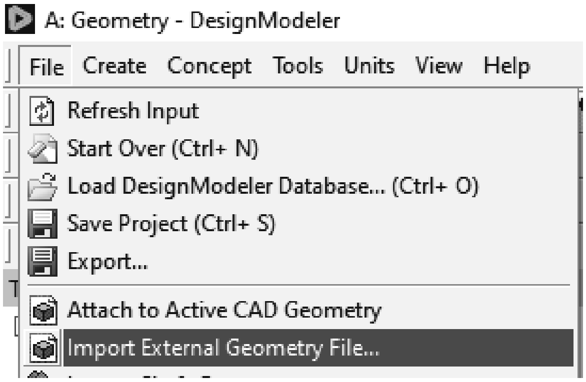


Figure 1-7 Importing from an alternate format.

Locate and load a file previously created in an alternate solid modeler.

D. >> Access Geometry in Active CAD system: Double click cell A2 to start **Design Modeler**. (You must **first** establish the link between Design Modeler and the CAD system.)

File > Attach to Active CAD Geometry

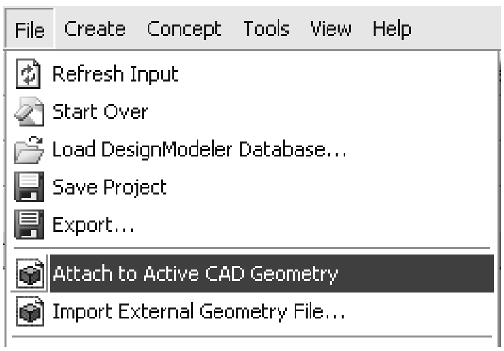


Figure 1-8 Attach to Active CAD Geometry.

E. >> Import from **alternate Solid Modeler. Right click** on **Geometry** and **select Import Geometry.**

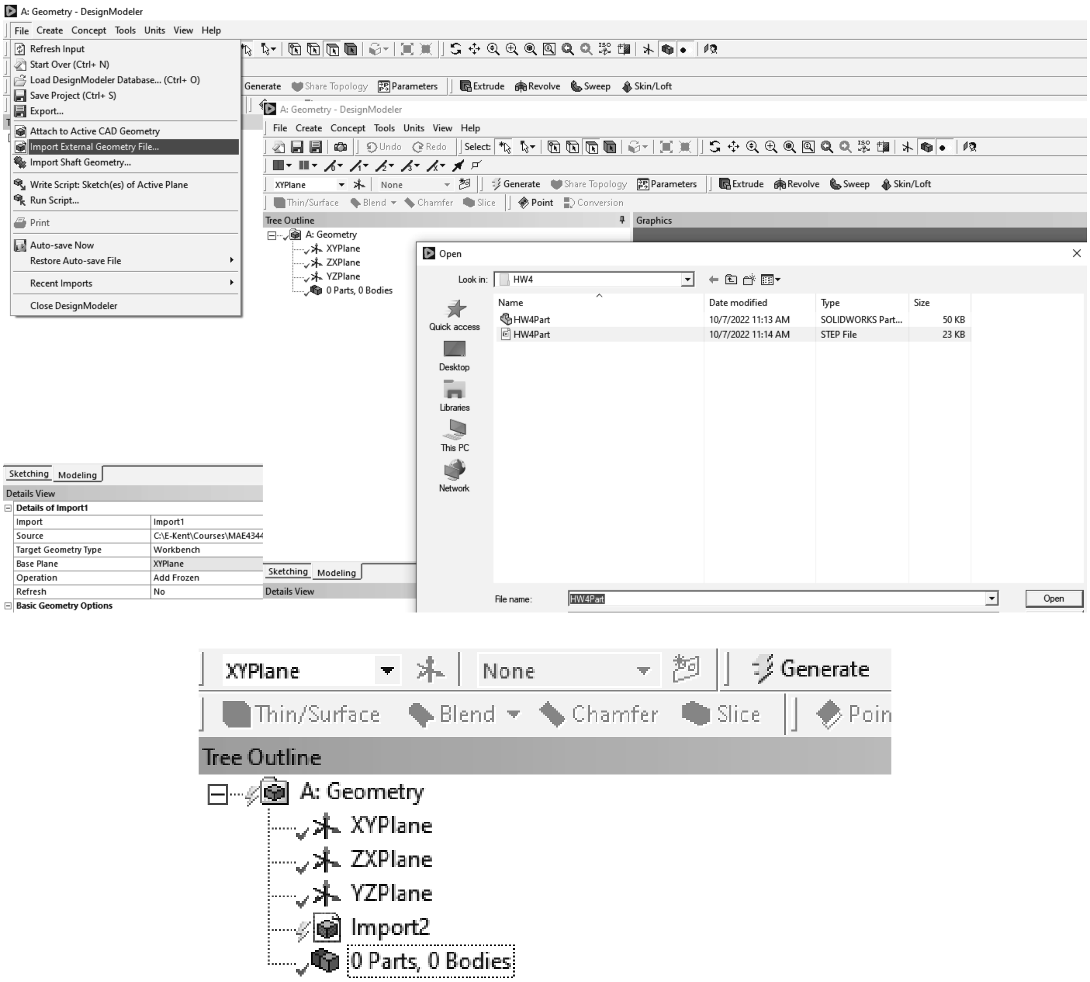


Figure 1-9 Import Geometry from Alternate Solid Modeler.

After importing a **STEP** file, for example, **Click Generate.**

Use the **Static Structural Analysis System** for stress, strain, and deformation analysis.

3. Getting **back** to **Tutorial 1A**, with the geometry attached, **Double Click Toolbox > Analysis Systems > Static Structural** to add the analysis to the **Project Schematic.** (Or just **drag** it in from the Analysis Systems column to the Schematic.)

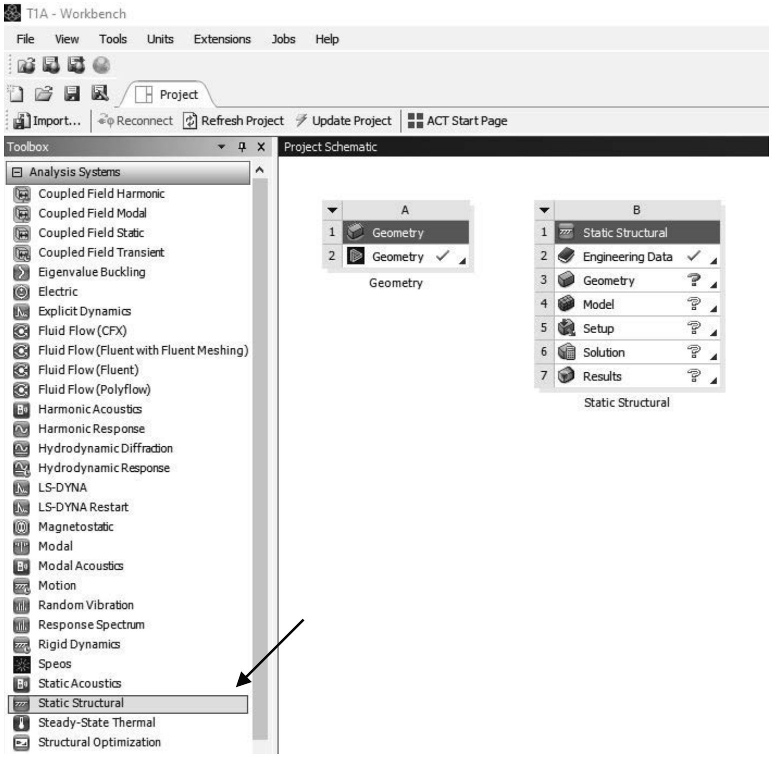


Figure 1-10 Adding Static Structural Analysis from the Toolbox.

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

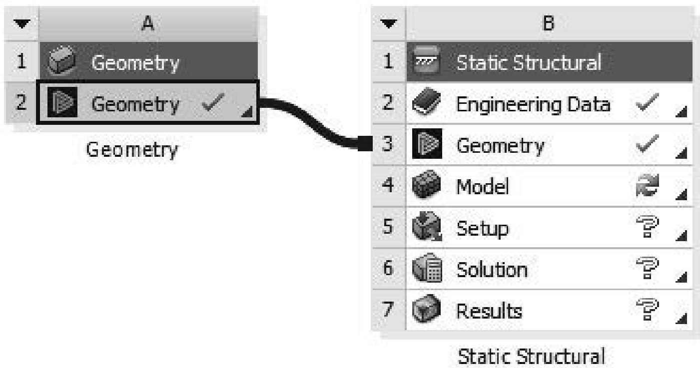


Figure 1-11 Sharing geometry.

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

In the lower left corner of the screen, you see "Starting Mechanical". Mechanical is a big file and takes a while to load. The Workbench display now shows the **Static Structural Analysis** and the tree structure on the left contains project items that include **Project, Model**, and **Static Structural**. See the figure below.

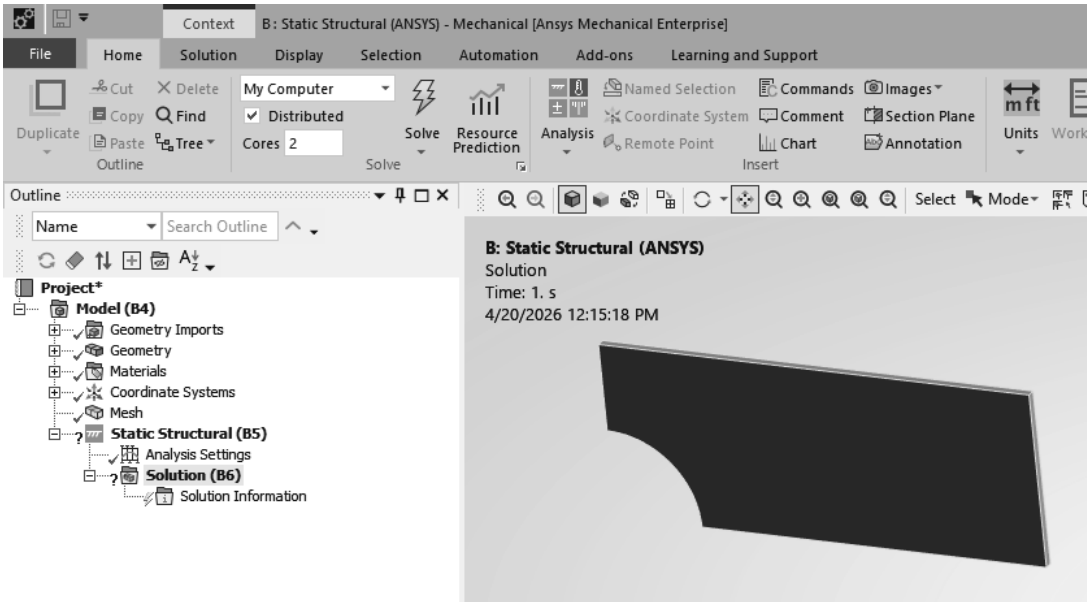


Figure 1-12 Plate quadrant in Static Structural Mechanical module.

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**.**

6. Home > Units > Metric (mm, kg, N, s, mV, mA) Default Material Structural Steel

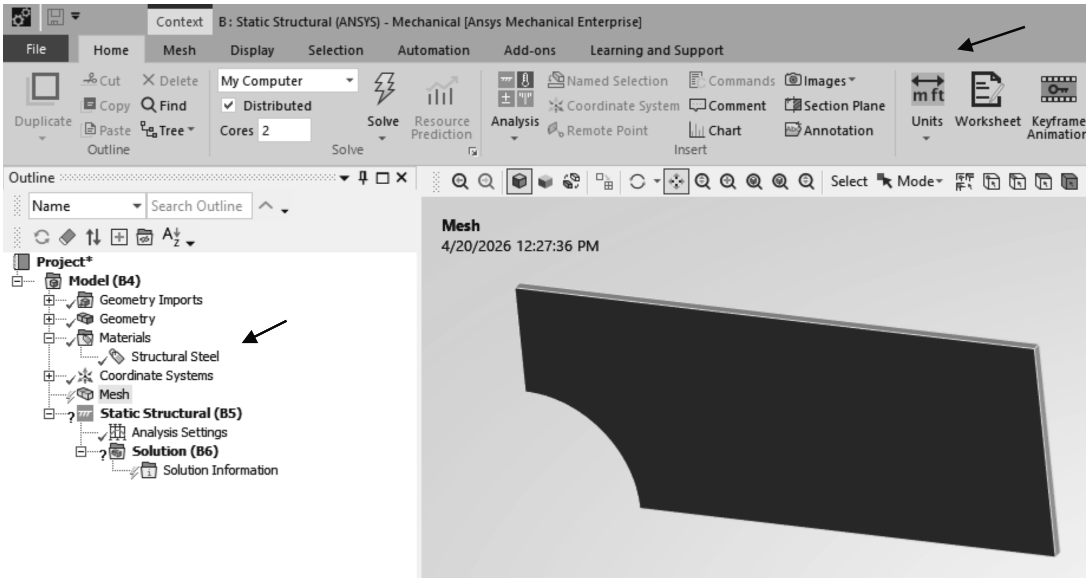


Figure 1-13 Check Units and Material assignment.

Structural Steel is the automatic **default material**. To return to the **Project Schematic, Select T1A – Workbench** from the programs running shown in the taskbar at the bottom of the screen.

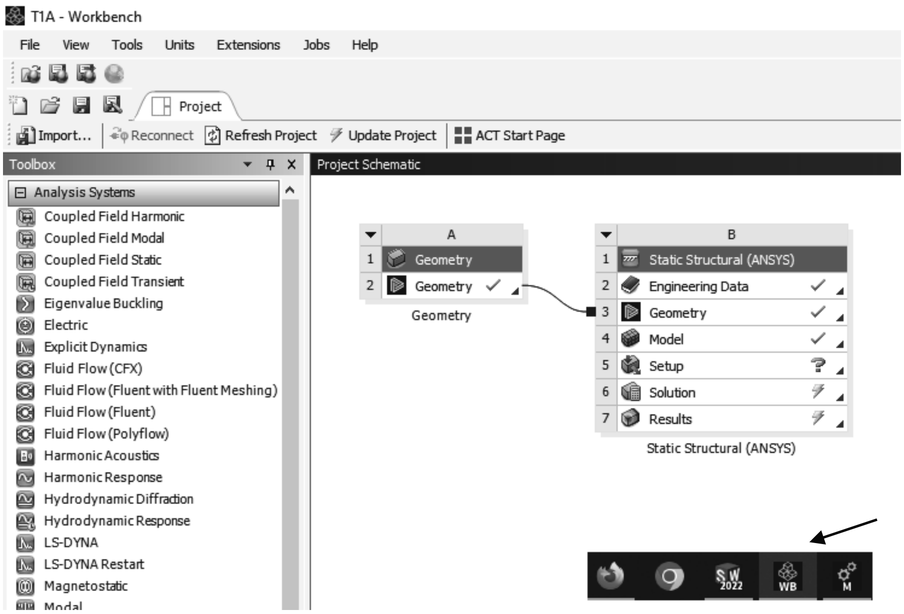


Figure 1-14 Workbench elements shown in the Taskbar.

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.

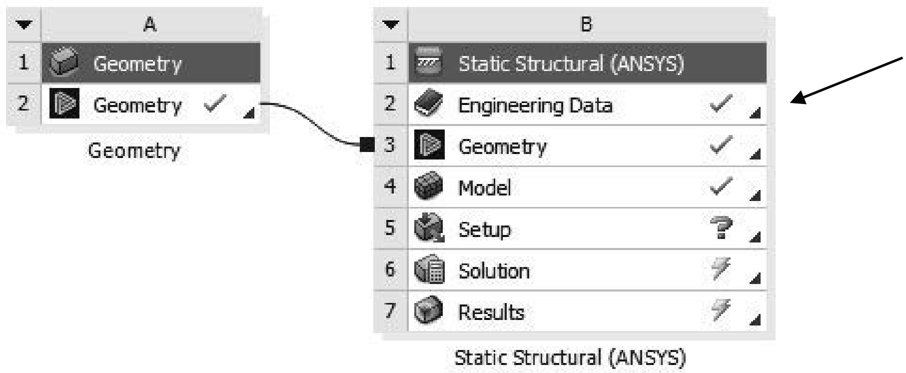
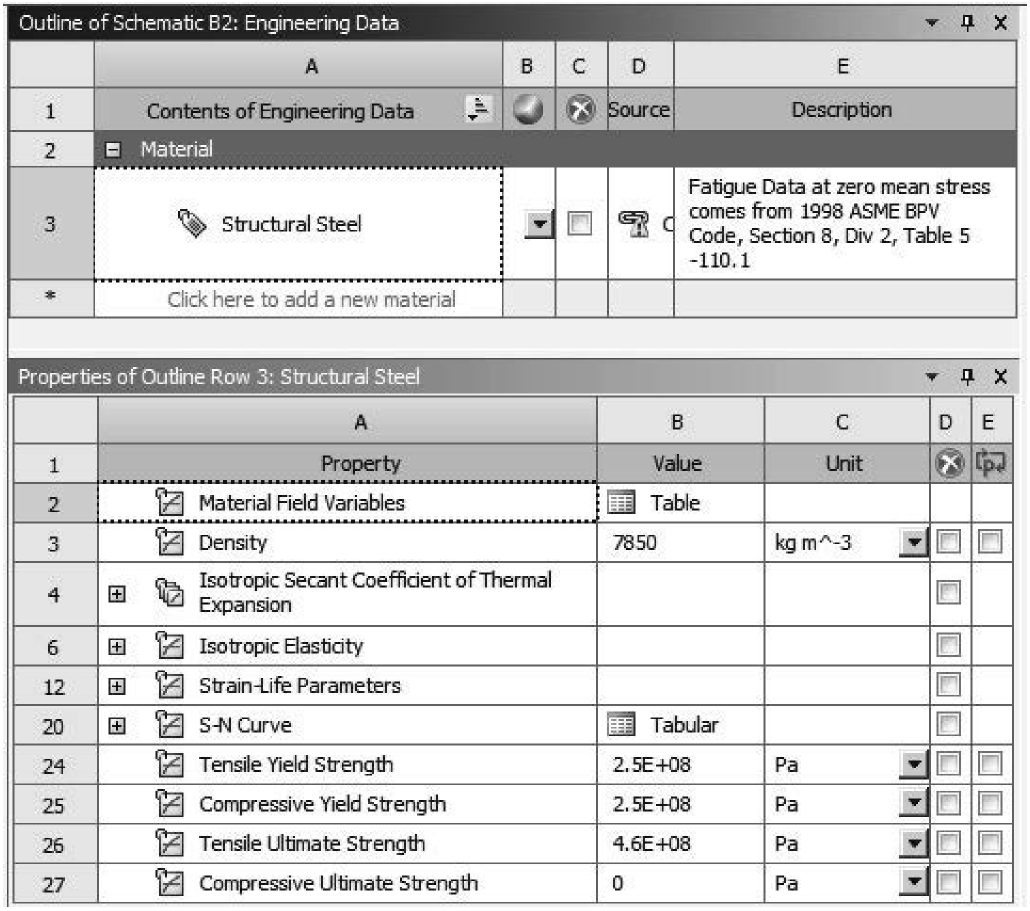


Figure 1-15 Project Schematic.

Be sure **View > Properties** and **Outline** are checked. Properties for the **default** material **Structural Steel** are shown.



Outline of Schematic B2: Engineering Data

	A	B	C	D	E
1	Contents of Engineering Data			Source	Description
2	Material				
3	Structural Steel				Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5 -110.1
*	Click here to add a new material				

Properties of Outline Row 3: Structural Steel

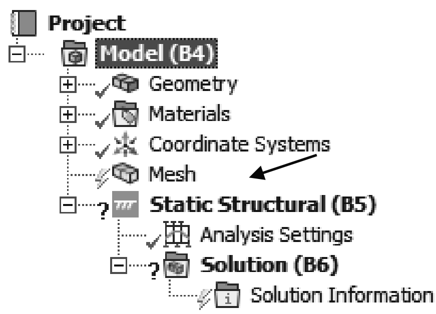
	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	7850	kg m^-3		
4	Isotropic Secant Coefficient of Thermal Expansion				
6	Isotropic Elasticity				
12	Strain-Life Parameters				
20	S-N Curve	Tabular			
24	Tensile Yield Strength	2.5E+08	Pa		
25	Compressive Yield Strength	2.5E+08	Pa		
26	Tensile Ultimate Strength	4.6E+08	Pa		
27	Compressive Ultimate Strength	0	Pa		

Figure 1-16 Material properties for structural steel.

Note that structural steel has **Tensile** and **Compressive Yield Strengths** of **250 MPa**., a **Tensile Ultimate Strength** of **460 MPa** and, apparently because it was not known when the table was assembled, **Zero** has been assigned to the **Compressive Ultimate Strength**. You can add the correct value as needed.

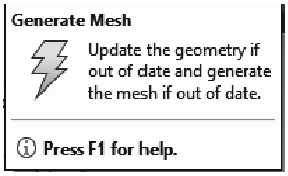
To **return** to the project

8. **Select Project** (Bottom of screen) .

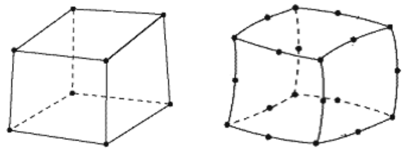


The **Mesh** item in the project tree has a lightning bolt symbol next to it indicating that the finite element mesh for this simulation has not yet been created. Workbench simulation will automatically develop a finite element mesh for the problem.

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



The default mesh that is created consists of several hundred **three-dimensional 8 or 20 node brick elements** as shown.



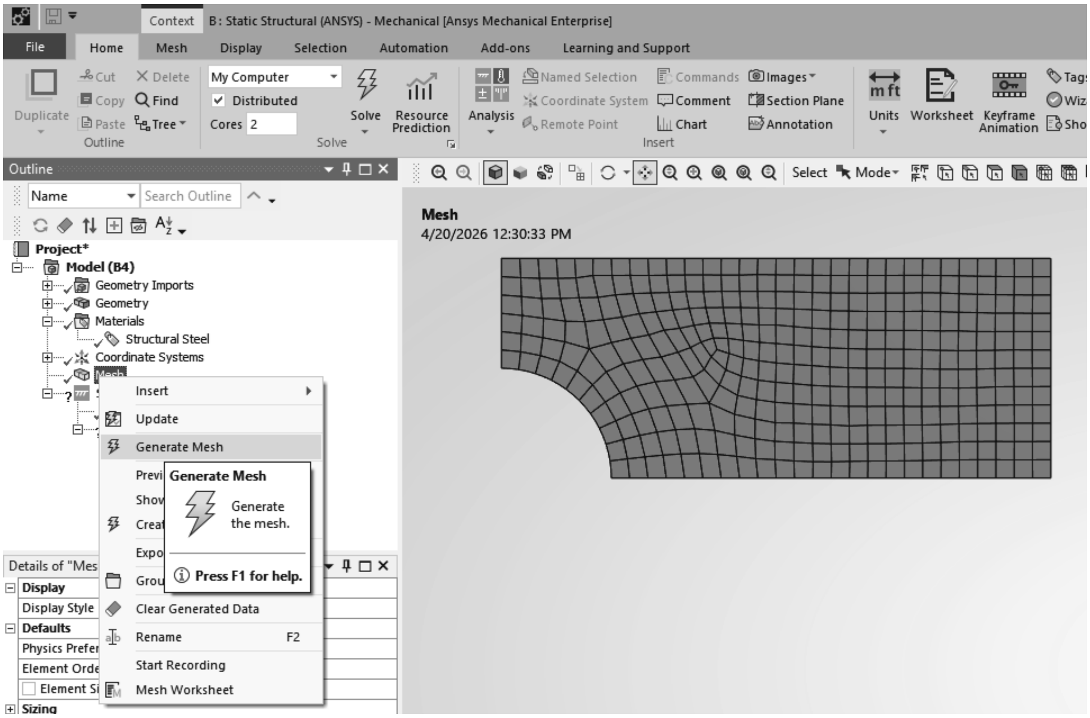
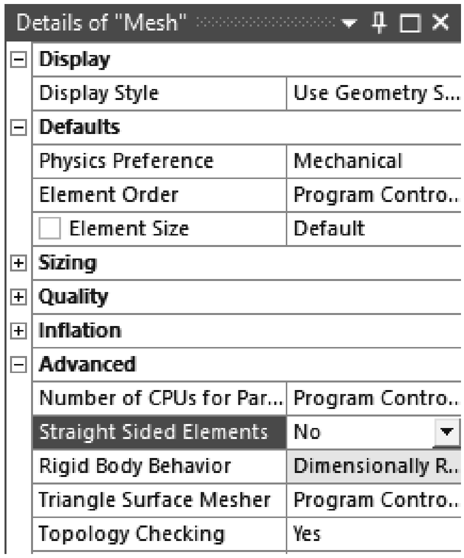


Figure 1-17 Meshing the geometry.

In **Details of Mesh, Advanced** we see that the **Straight Sided Elements option** is **No**, meaning that **20 node brick** elements have been used.



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**

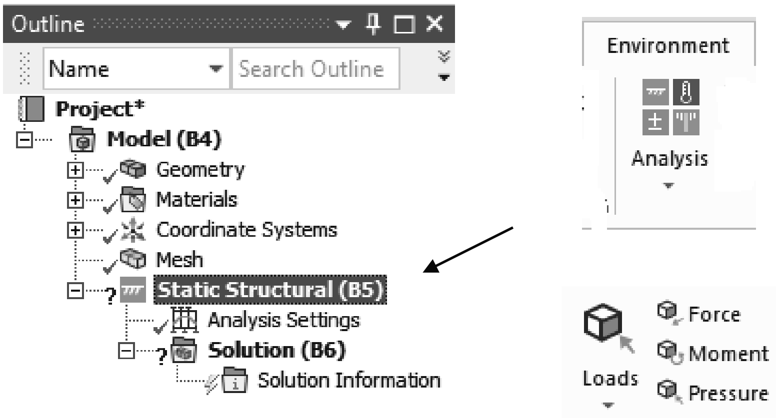
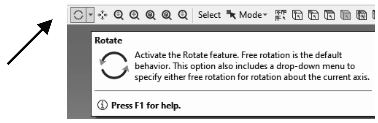


Figure 1-18 Structural loads menu.

Be sure that the **Face** selection filter is highlighted , rotate as needed 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 = 50000 N**

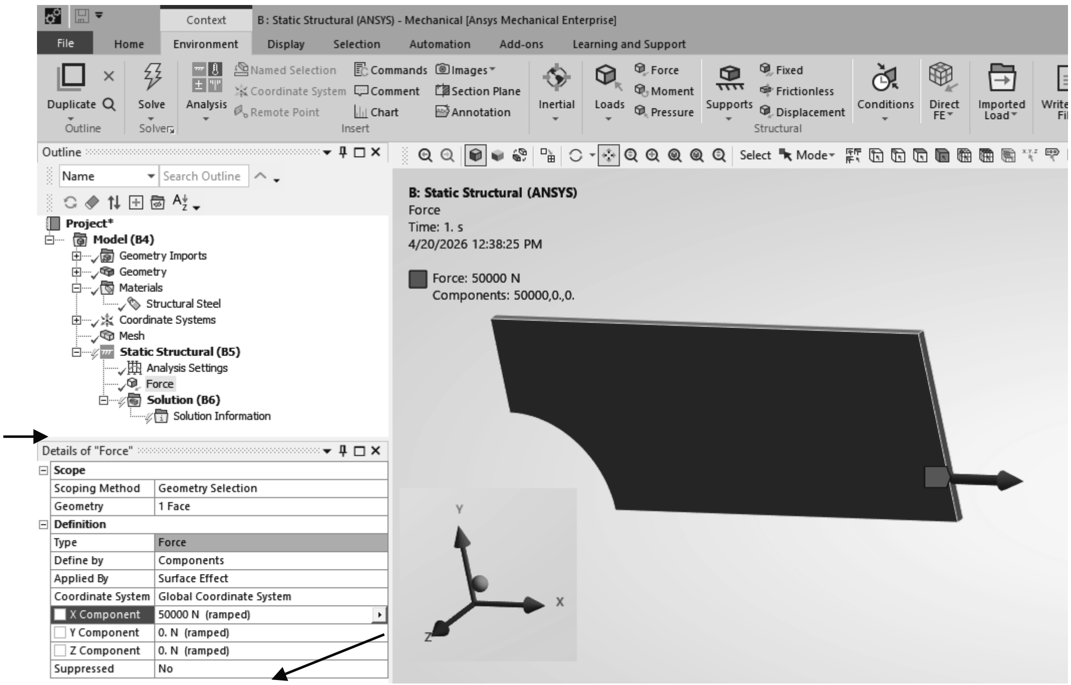


Figure 1-19 Tensile loading.

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** so as to prevent rigid body motion in a direction perpendicular to the plane of the plate. See the figure below. (The back is not a plane of symmetry, but the plate is thin with respect to its other dimensions. Compete rigor would require that we create a solid model with half the thickness of the full plate.)

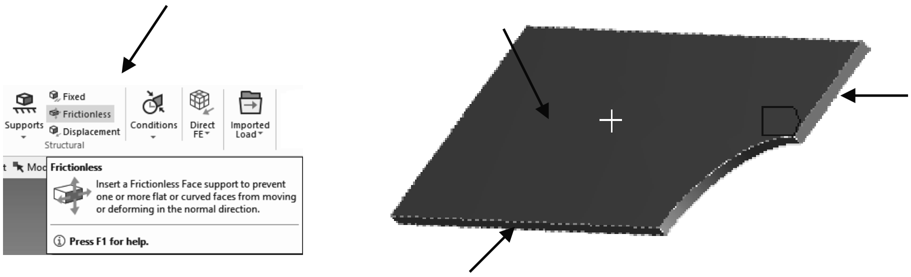


Figure 1-20 Displacement constraints.

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

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

Details of "Frictionless Support"

Scope	
Scoping Method	Geometry Selection
Geometry	3 Faces
Definition	
Type	Frictionless Support
Suppressed	No

Figure 1-21 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 very little error in the approach we used. Try it both ways to see. But be sure not to select both front and back surfaces; this creates a problem with zero strain in the z-direction.)

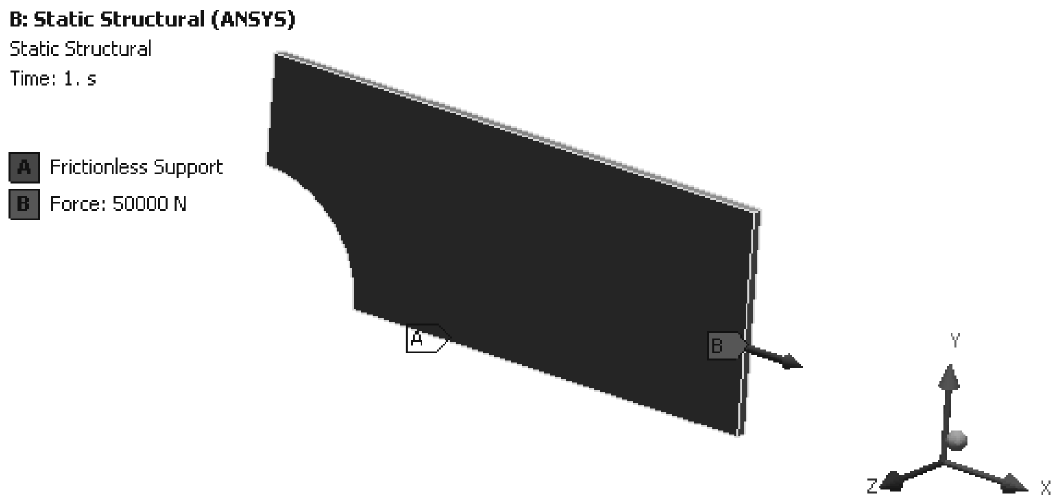


Figure 1-22 Structural Static 'Environment' settings.

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**.

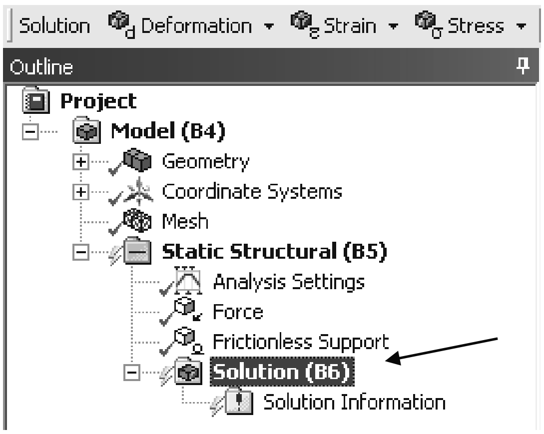


Figure 1-23 Solution result options menu.

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

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

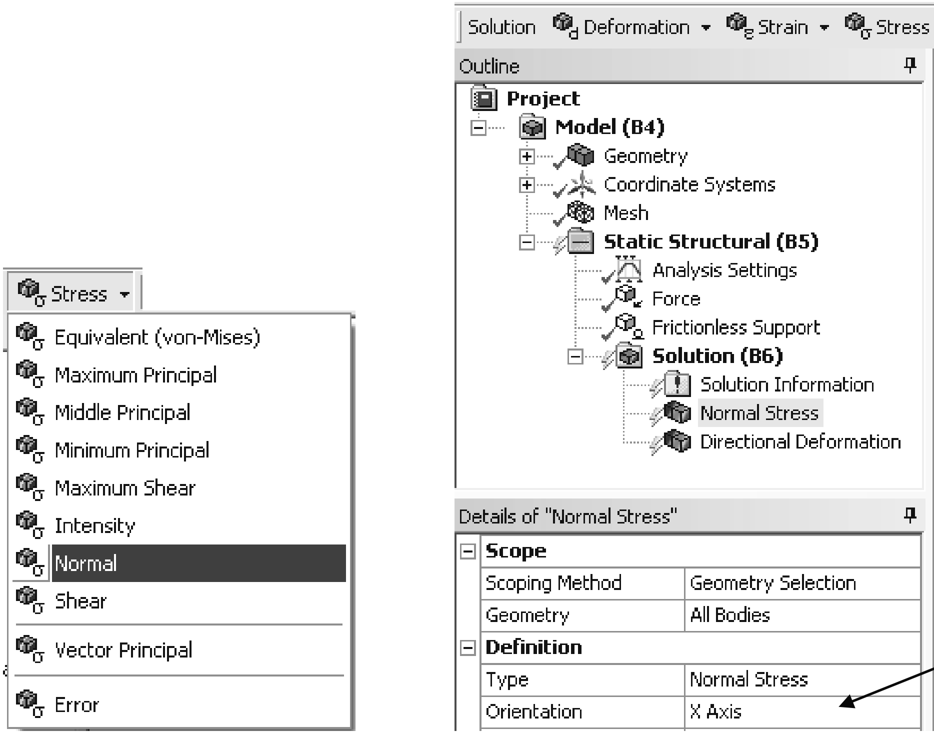


Figure 1-24 Select X-direction normal stress output.

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 

to complete the simulation solution.

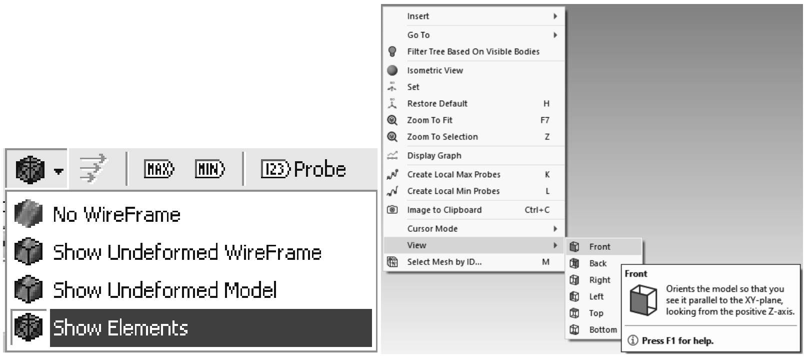
17. Solve 

The solution progress is shown in the lower left corner of your screen.

Figure 1-25 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 **Edges** to **Show Elements**)



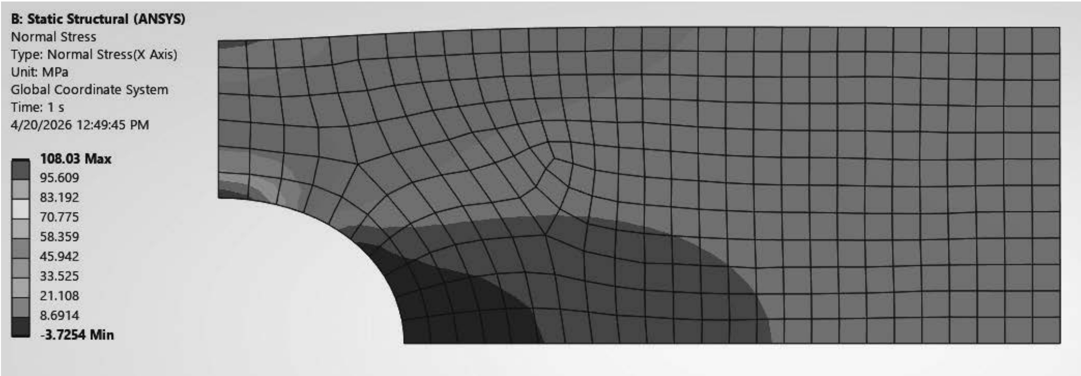


Figure 1-26 X direction normal stress; **Averaged** Display.

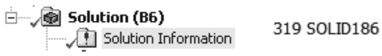
The solution for the X direction normal stress shows a **maximum value** of **108 MPa**. This value is well below the material yield stress of 250 MPa, so the elastic solution that we have performed is valid. The averaged-at-node stress values for a mesh of just over 300 elements are shown above.

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 Ansys computed maximum value is **108.03 MPa** which is much less than **one per cent error** (assuming that the published value of K_t is correct, that is).

If we list the solution information, we find that 319 **SOLID186** elements were used in the model. A search of the Ansys Mechanical Help system shows these to be **20 node brick elements**.



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

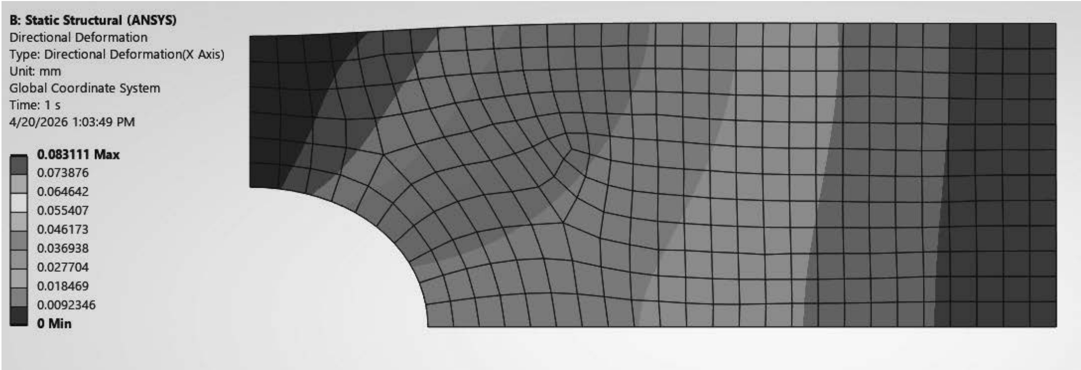


Figure 1-27 X direction deformation.

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 

Computed results are shown in the next figure.

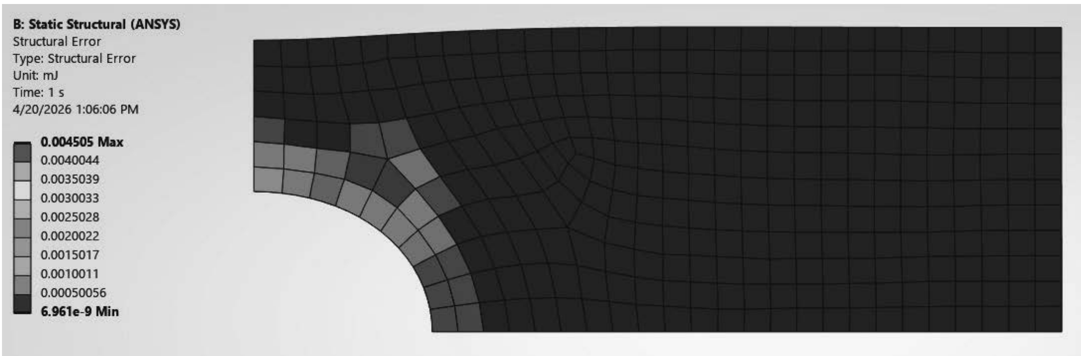


Figure 1-28 Computed structural error estimates.

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 only 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, thus 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.

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 = 108 MPa = **15,668 psi**
Max deflection = 0.083 mm = **0.0033 inches**

To **increase the accuracy**, you can 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. Change the element size using Details of Mesh.

20. **Mesh > Details of Mesh > Element Size > 10 mm > Solve** (922 elements)

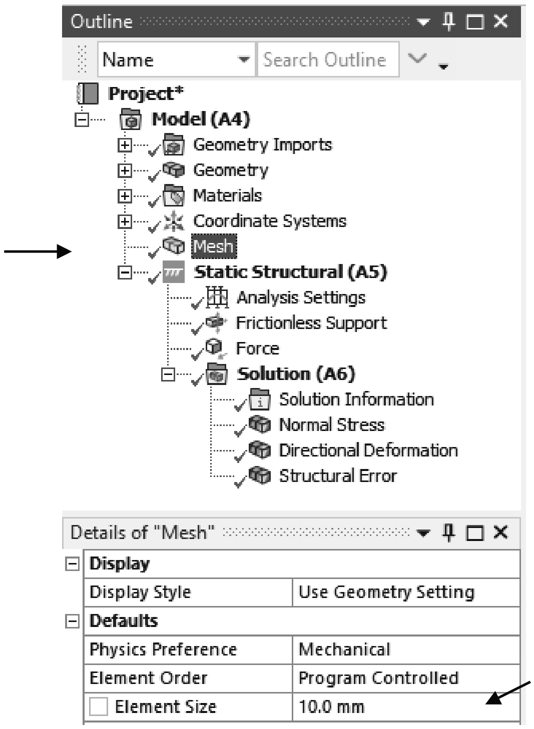


Figure 1-29 Mesh settings.

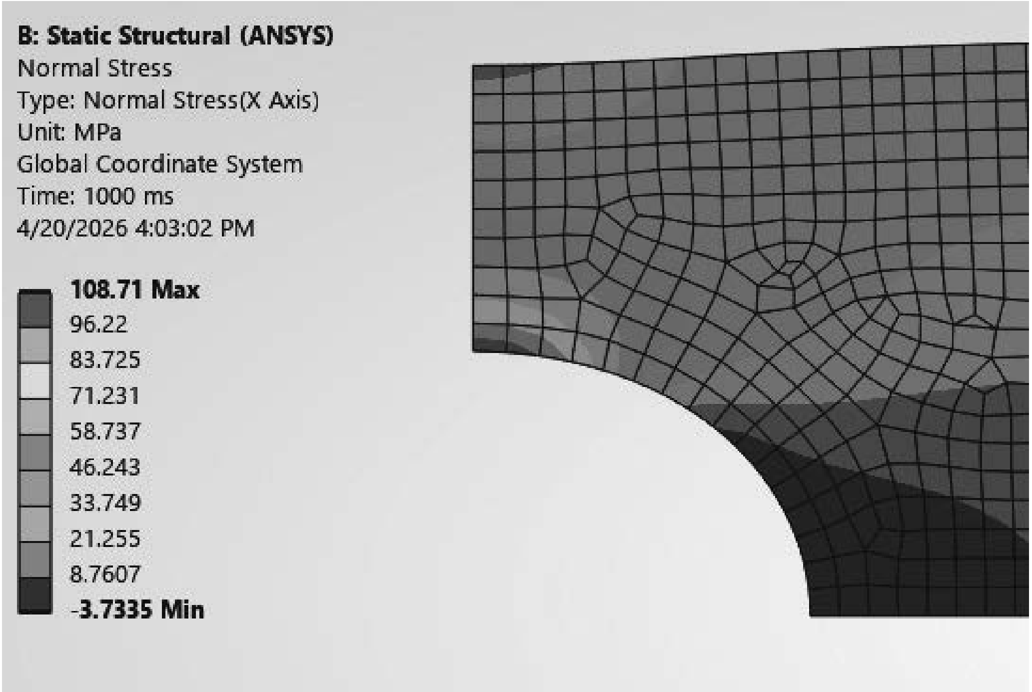


Figure 1-30 Stress based on a 10 mm mesh.

21. **Repeat** this process using a 5 mm mesh (7302 elements).

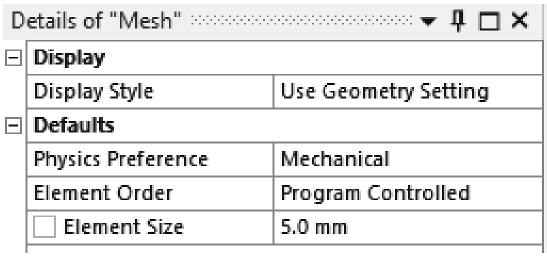


Figure 1-31 Mesh settings for 5 mm.

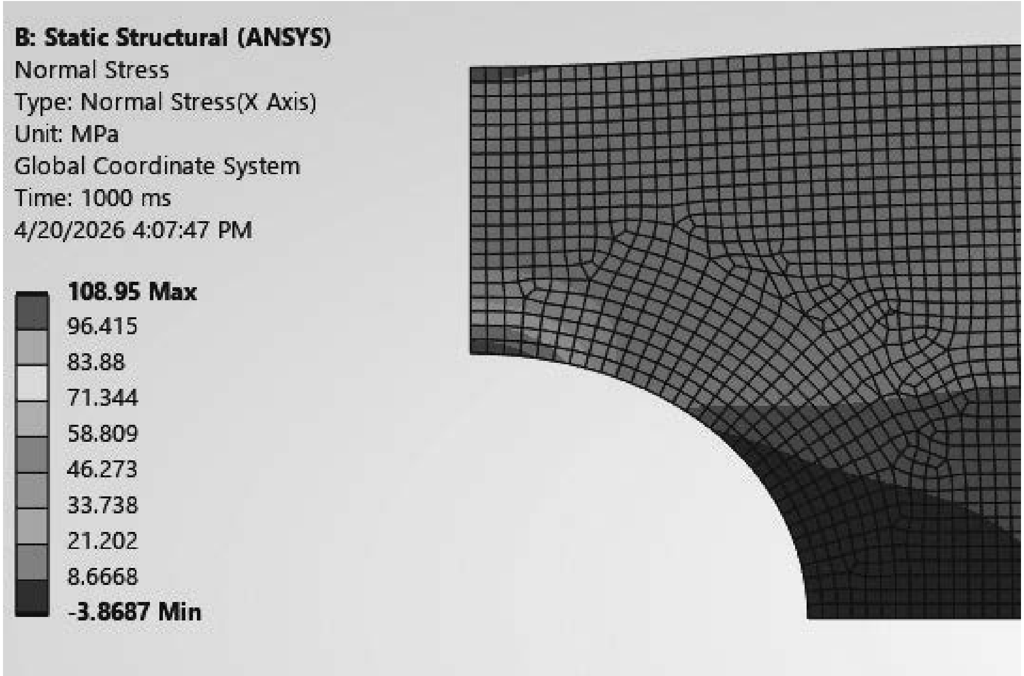
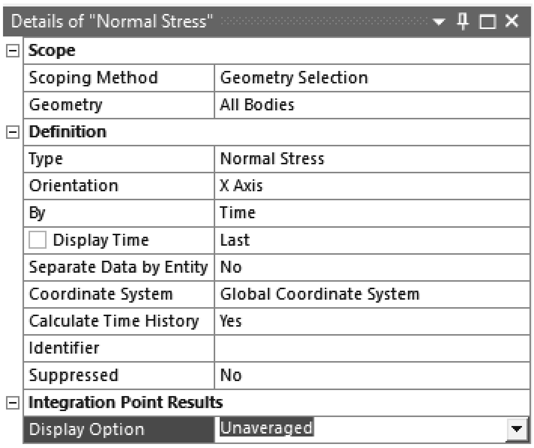


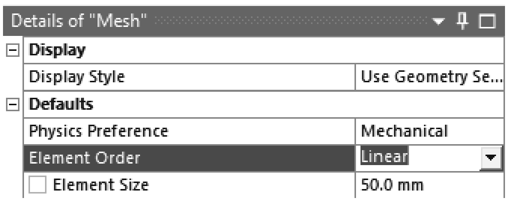
Figure 1-32 Stress based on the 5 mm fine mesh.

The original mesh gave very accurate results, so the subsequent refinements were not necessary in this case.

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

Next use the mesh details to change from **curved sided elements** (quadratic) to **straight sided elements** (linear). Those computed results are shown on the **right** side in the figure below.





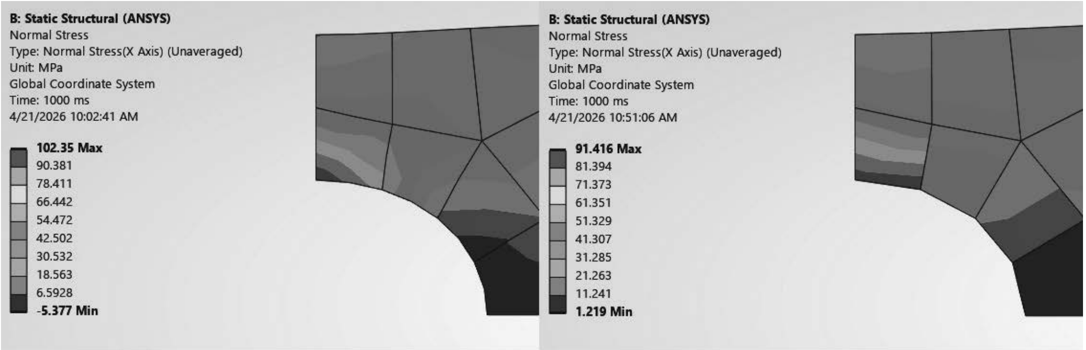


Figure 1-33 Stress estimates based on a very coarse mesh, quadratic vs. linear elements.

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 102 MPa., around 6 per cent in error. With eight node elements (linear, no curved sides) it's even worse; **91 MPa,** an **error** of **17**%! This shows how easy it is to get the wrong answer. Experiment with the mesh. **Always explore** the solution space for your problem.

Table 1A.1 - Results Summary, Curved Sided Elements

Number of Elements	Maximum Deformation in X direction mm	Maximum Stress in X direction MPa
36	0.0831	102.4
319	0.0831	108.3
922	0.0831	108.7
7302	0.0831	109.0

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**, but you don't know it. 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 derivatives of the displacement distributions within the object, i.e., $\varepsilon_x = \partial u / \partial x$, where ***u*** is the displacement in the x-direction. And the stresses are proportional to strains (Hooke's law).

Note in the coarse mesh figure above that the calculated maximum stress is only **91 MPa**, 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?

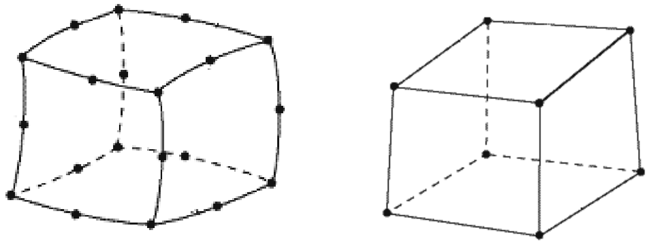


Figure 1-34 20 node and 8 node solid elements.

Most of the previous results were calculated using the default 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) 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 over 10 per cent**. The smoothness is **forced** in the software by averaging the results of all elements joining a node before plotting. So beware of always using **averaged stress results**!

At the completion of the solution, the project schematic shows all items checked off in the Static Structural matrix as shown below.

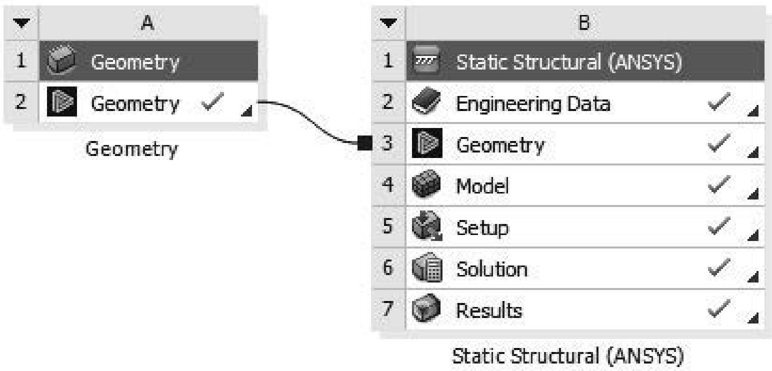


Figure 1-35 Project Schematic after solution.

25. Save your work and **close the T1A Workbench Project.**

The files associated with this project are shown in figure below. **T1A.wbpj** is the Workbench project file, and the **T1A_files directory** contains the supporting files as shown.

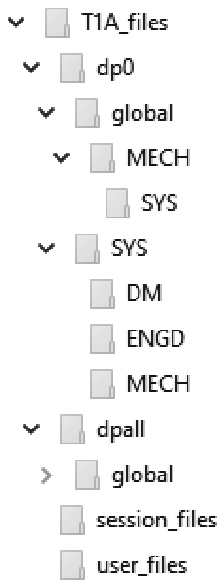


Figure 1-36 Files stored after Workbench Project T1A has been closed.

Take a minute to verify the file structure and note that a good deal of storage is required for the initial mesh model. More space may be required if intermediate files are created during solution.

1-4 TUTORIAL 1B – PLATE WITH CENTRAL SQUARE HOLE

Suppose we take the plate of Tutorial 1A and replace the circular hole with a 200 mm **square hole**. The upper right quadrant of the model is shown below.

1. **Open a new Workbench Project; create the model shown below** and **save as T1B.**

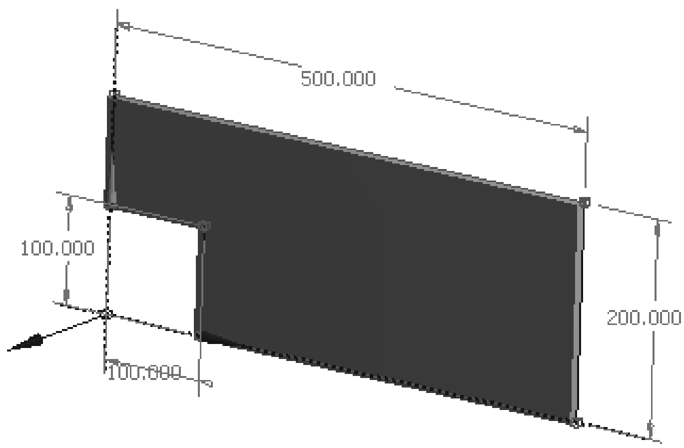


Figure 1-37 Plate with square hole; dimensions in mm.

2. Start a new **Static Structural Analysis** using the quadrant of a plate with a square central cut out. **Share Geometry** and **Double Click Model** to start **Mechanical.**

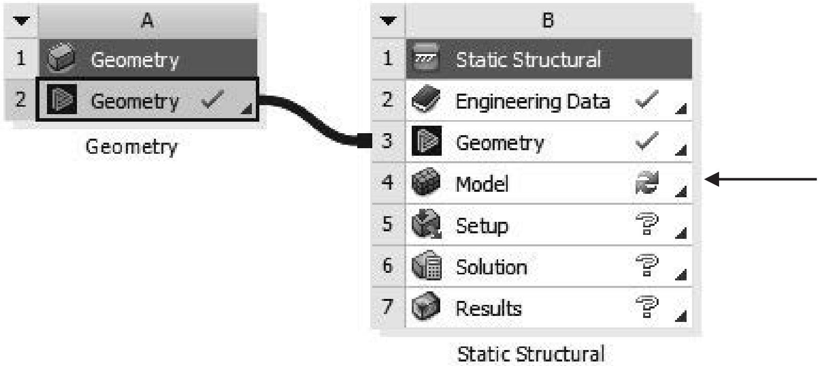


Figure 1-38 Project schematic.

3. **Mesh > Generate Mesh** The next figure shows the default mesh for this problem.

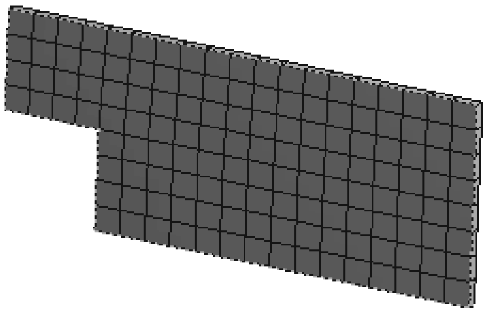


Figure 1-39 Default mesh for the plate with square hole.

This default mesh consists of 136 brick elements as shown above. Default material is steel.

4. **Apply the displacement boundary conditions** and **loading** as in **Tutorial T1A.**

Include the **X direction normal stress** in the computed quantities. (Your results may vary slightly depending on software version.)

5. **Solution > Stress > Normal > Details of "Normal Stress" > Orientation > X Axis**

6. **Solve** 

and view the normal stress solution. Also click the **Max** icon.

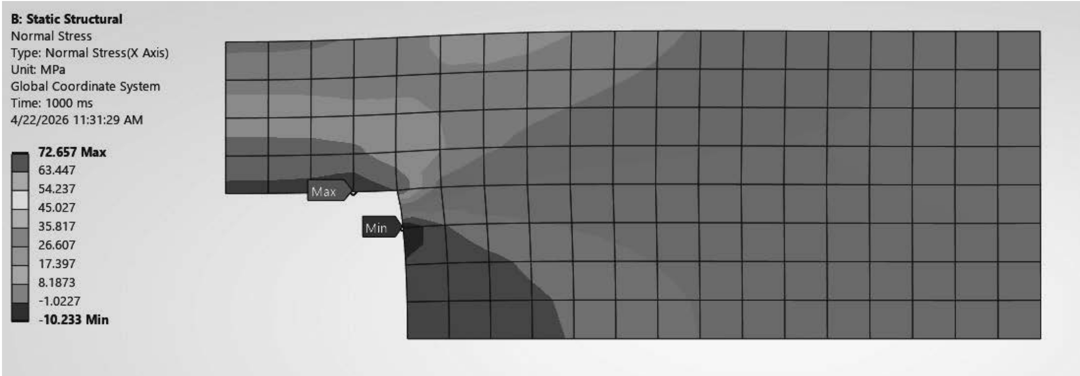


Figure 1-40 Normal stress in. the X direction.

7. **Right click** and **select** the **front view**. The averaged smooth stress contours show a maximum stress of around **73 MPa**.

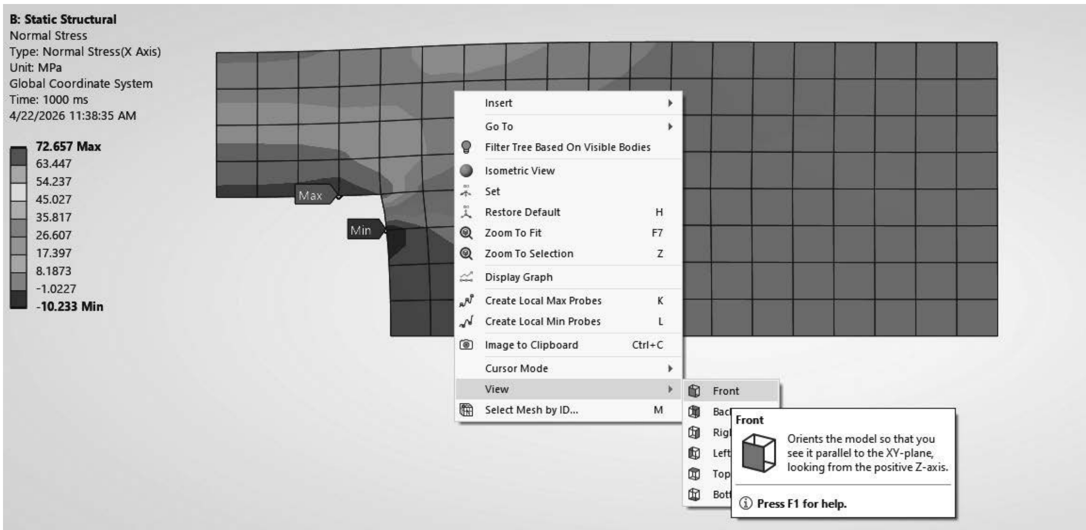


Figure 1-41 Normal stress using the default mesh.

The default mesh consists of elements that are 25 mm by 25 mm and 10 mm thick. Suppose we reduce the element size and examine the effect on the computed stresses. In the outline tree click on **Mesh**, then in **Details of "Mesh" change** the **element size** from **25 mm** to **10 mm**.

8. **Mesh > Details of "Mesh"**

9. **Defaults > Element Size > 10 mm**

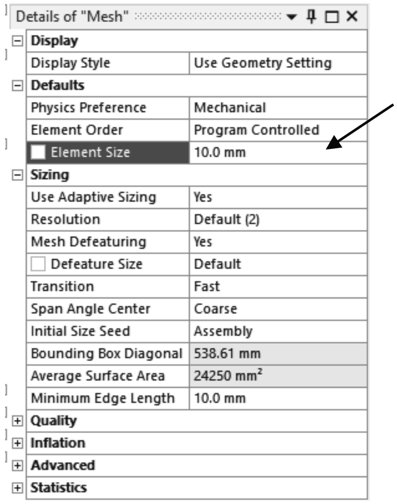


Figure 1-42 Set the element size.

10. Right click Mesh > Generate Mesh

The new mesh with 900 elements is shown below.

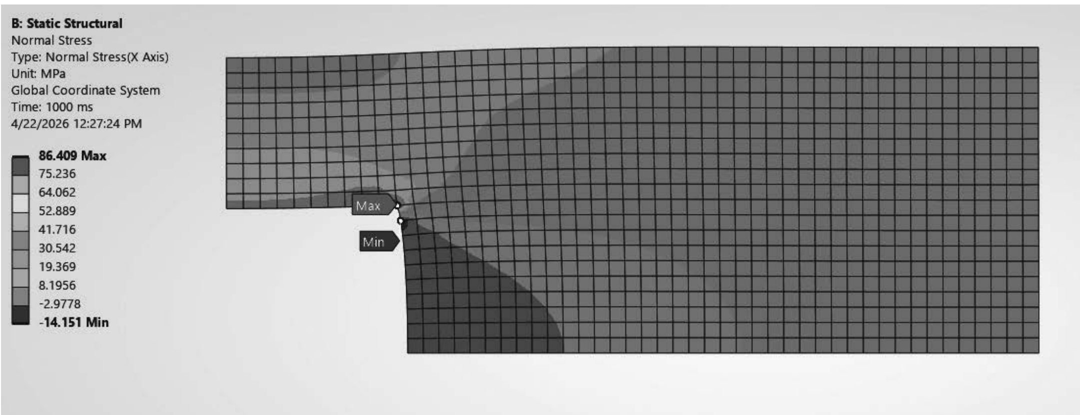


Figure 1-43 Normal stress for 10 mm elements

11. Solve 

and view the normal stress solution.

The computed **maximum** normal stress is **86 MPa**, an increase of about **18 per cent**.

Now reduce the mesh size to **5 mm**. This creates a 7200 element mesh.

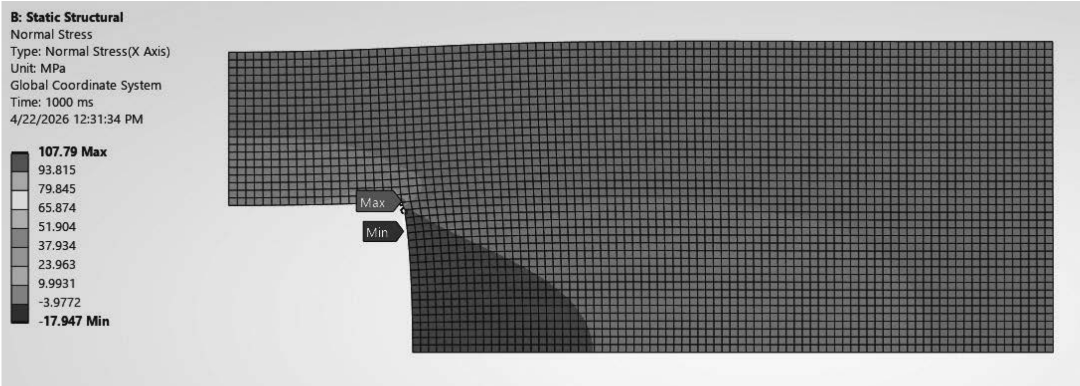


Figure 1-44 Normal stress for 5 mm elements.

The computed **maximum** normal stress is **108 MPa**, a further increase of around **26 per cent**.

Finally, reduce the mesh size to **3 mm**, producing a model with over 40,000 elements.

Note, this large number of elements may exceed the allowable for the software version of Ansys Workbench Mechanical that you are using.

12. **Details of "Mesh" > Element Size > 3 mm**

13. Again **Solve** 

and view the normal stress solution.

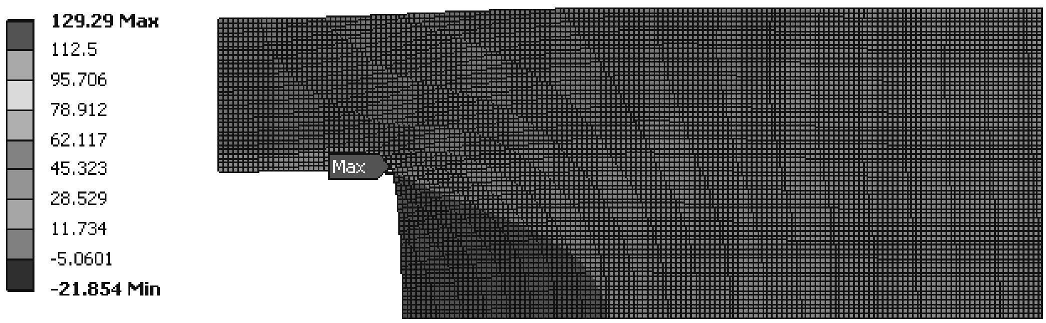


Figure 1-45 Normal stress for 3 mm elements.

The newly computed maximum normal stress in the X-direction is **129 MPa**, an increase over the previous value of **19 per cent**. So, **what is happening?**

The behavior of the maximum stress results we just computed is due to the **stress singularity** at the corner of the square hole. The corner has a zero radius of curvature where the two edges meet, and as the radius of an interior corner or other such interior notch approaches zero, the computed stress approaches an **infinite value**. (See a representative stress concentration factor chart.)

If the material is ductile and the loads are static, the high stresses at this location may not be of concern in the actual use of the part. If the material is brittle **or** the loading is repetitive (fatigue situation), then it is **very important** to model the **actual geometry**, **including any small fillet radius** that may exist, and thus determine the **correct stress** value in the vicinity of the **notch**.

During manufacture of a real plate, it would be difficult to make the edges of the hole meet with an exactly zero radius at the corner. Or it may be that a specific corner radius is required for the proper function of the part. Suppose for our plate the actual **corner radius** is **15 mm.** Make this geometry change in the model of the plate with the solid modeler you are using.

14. Return to the **Project Schematic** and **double click Geometry**. Working in **DesignModeler** we will add the fillet radius to the plate with square hole interactively. (If you are using another modeler, modify the geometry in that solid modeler and reload the STEP or IGES part file.)

Figure 1-46 Project schematic view.

15. Select the **Edge Filter** , Click the interior **Corner Edge** and then select **Fixed Radius Blend** 

Enter 15 mm > Generate 

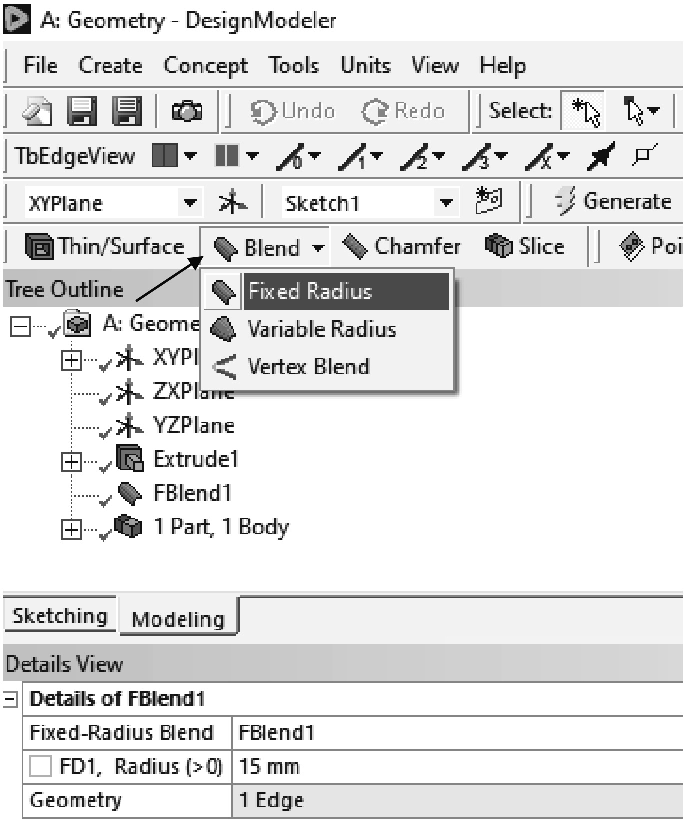


Figure 1-47 Part modification in DesignModeler.

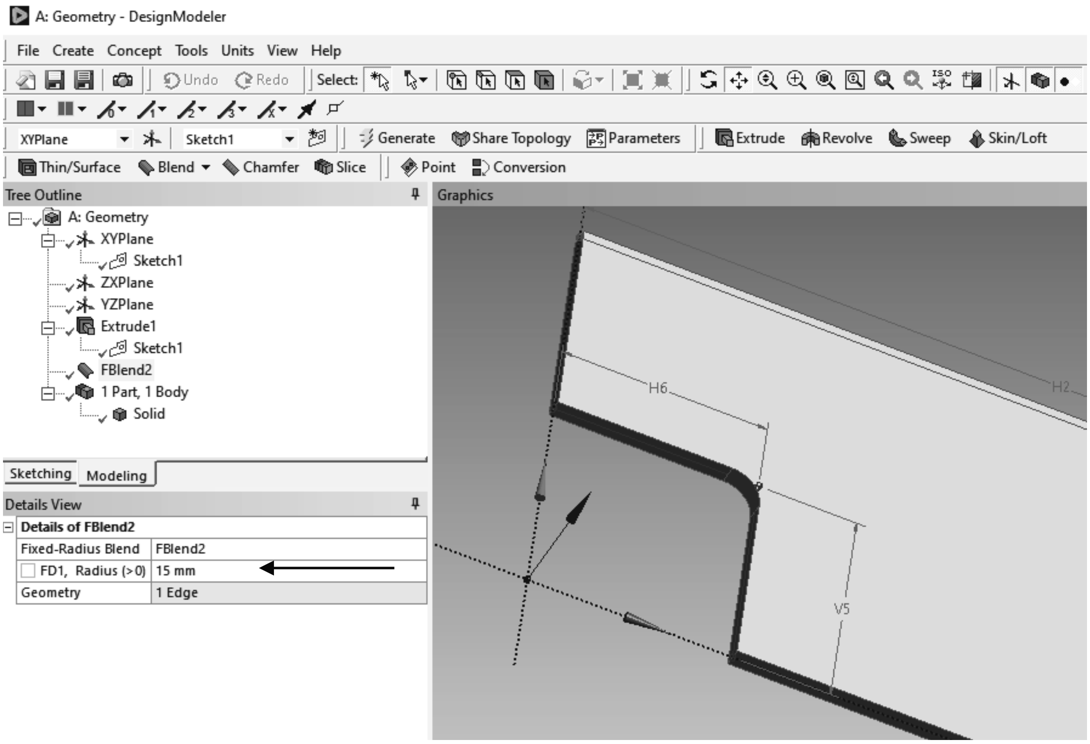


Figure 1-48 Updated DesignModeler geometry.

Now **update** the Static Structural Analysis model to the new geometry.

16. Return to the **Project Schematic** view**.** Left click on the **small blue triangle** in **cell B4, Model**.

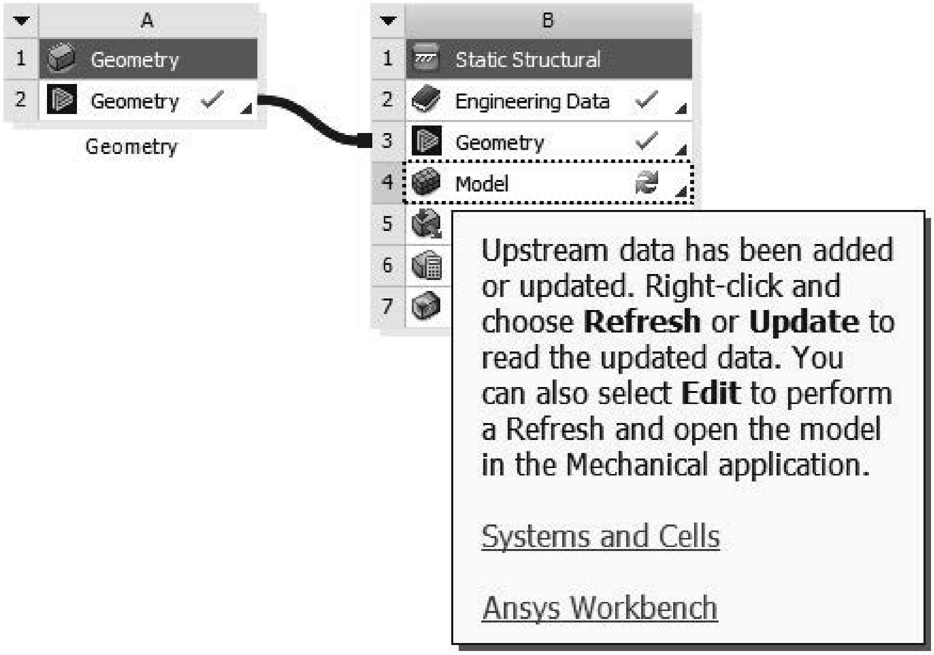


Figure 1-49 Model information screen.

17. Right Click in cell B4, Model and Select **Update**.

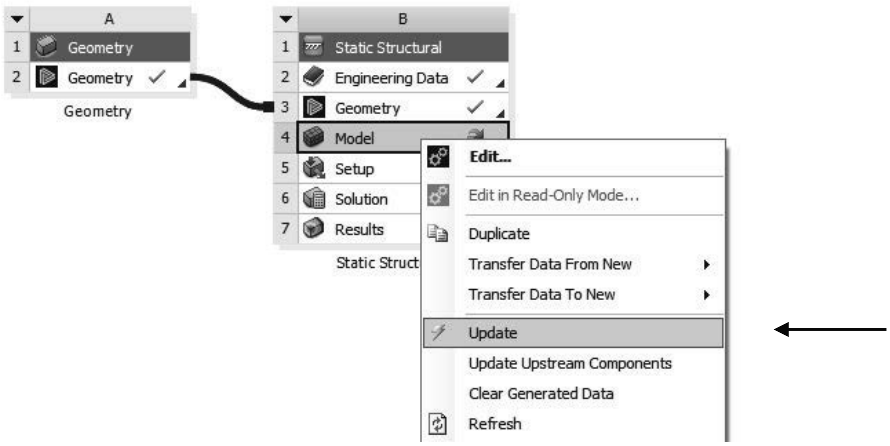


Figure 1-50 Update the model.

After the update is complete, return to the Static Structural Mechanical screen.

18. Enter 0 for **Element Siz**e to reset it to the **Default**.

Details of "Mesh"

Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
Element Size	Default
Sizing	
Use Adaptive Sizing	Yes
Resolution	Default (2)
Mesh Defeaturing	Yes
Defeature Size	Default
Transition	Fast
Span Angle Center	Coarse
Initial Size Seed	Assembly
Bounding Box Diagonal	538.61 mm
Average Surface Area	24250 mm^2
Minimum Edge Length	10.0 mm

Figure 1-51 Reset element size.

The project now is updated; the material, load and frictionless supports remain the same; the analysis proceeds as before. 

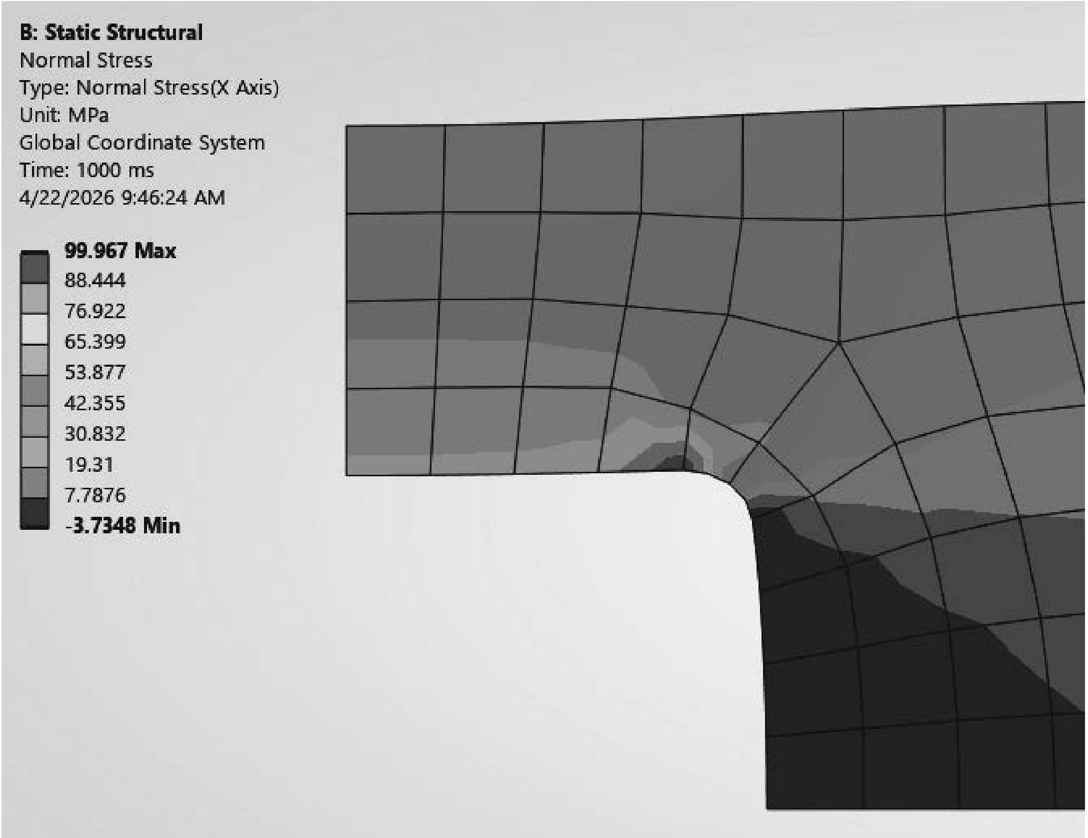


Figure 1-52 Default mesh for modified geometry.

The computed maximum stress associated with the 15 mm corner radius is about **100 MPa**.

We can control the accuracy of the solution manually as we did in previous examples or we can request an **automatic iterative solution** process to be employed that will use models with successively smaller element sizes while monitoring the change in the requested solution quantity. This is essentially an automation of what we did manually earlier.

Select maximum Normal Stress as the quantity to monitor during iterative solution.

19. Right click **Normal Stress > Insert >** Left click **Convergence**

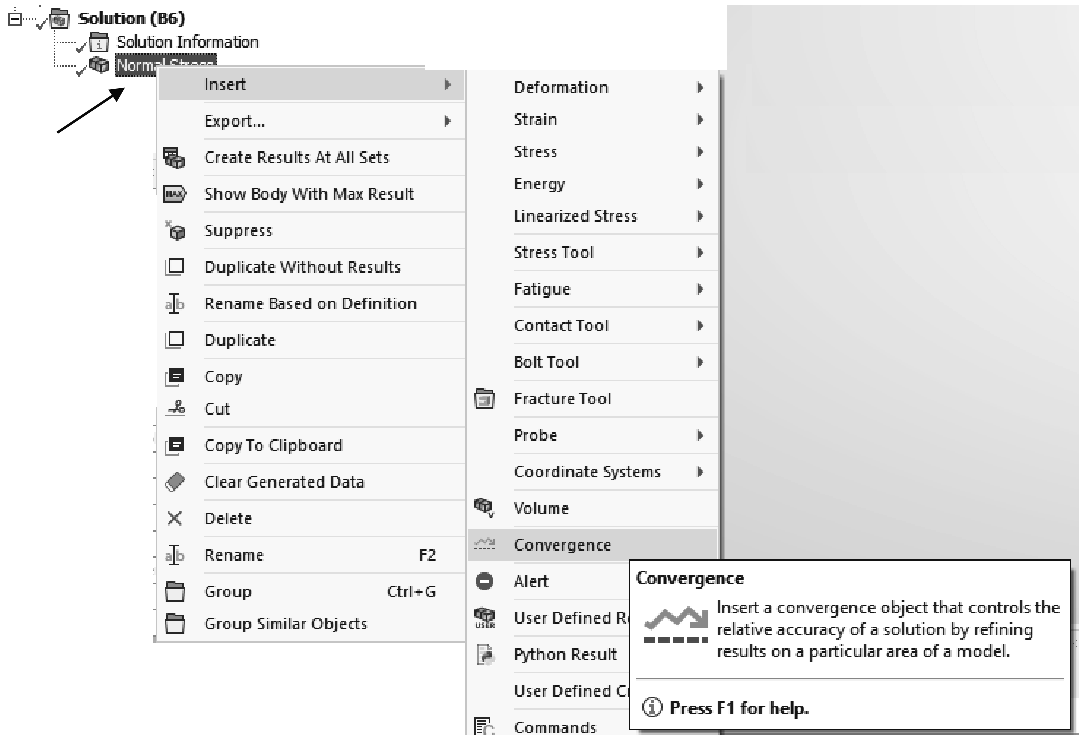


Figure 1-53 Insert convergence criterion.

Define the allowable change in the maximum Normal Stress between solutions.

20. Details of "Convergence" > Allowable Change > 0.1 per cent

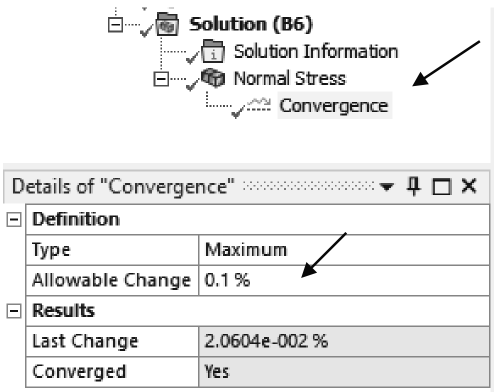


Figure 1-54 Allowable change in the normal stress.

Define the default number of mesh refinements and depth (refinement gradations).

21. Solution > Max Refinement Loops > 1; Refinement Depth > 2

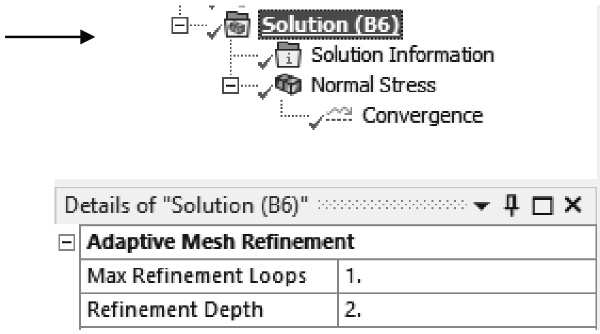


Figure 1-55 Adaptive refinement parameters.

22. Right click **Solution** or **Normal Stress > Solve** 

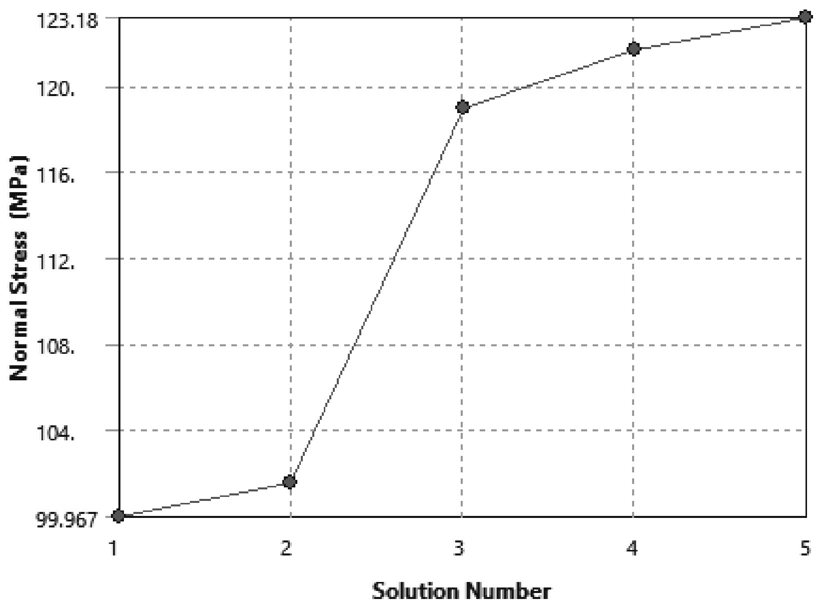
Figure 1-56 Solution status.

23. Once the solution process has finished, click on **Convergence**.

The convergence results shown in the next figure are displayed. The model is refined from 1081 nodes and 134 elements to one with 4797 nodes and 2602 elements. The maximum normal stress in the X-direction changes from **99.967 MPa** to **101.57 MPa**, a total change of about **1.6 per cent.** Continue to press the solve icon until the repeated solution **change** in successive computed maximum normal stress values is less than 0.1 per cent as requested. See the next figure.

You may **not** want to use this convergence process for problem types you are very familiar with, but in certain cases it can be a useful tool.

Convergence History



	Normal Stress (MPa)	Change (%)	Nodes	Elements
1	99.967		1081	134
2	101.57	1.5942	4797	2602
3	118.95	15.761	9336	5580
4	121.73	2.3048	17366	11136
5	123.18	1.1859	60756	40955

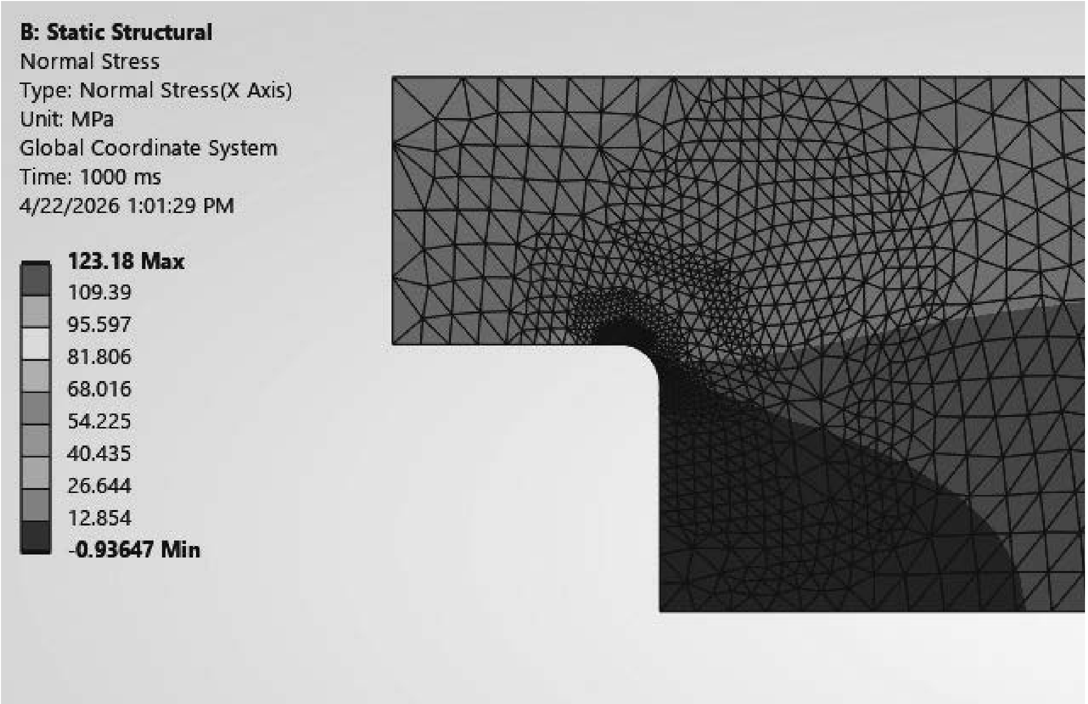


Figure 1-57 Final mesh and stress convergence history.

Another refinement method that can be used to ensure accuracy of results is the manual insertion of a mesh refinement using the fillet radius in the high stress area as the target geometry. The results produced thereby are shown below. We once again converge to a solution of 123 MPa for the maximum normal stress in the X-direction.

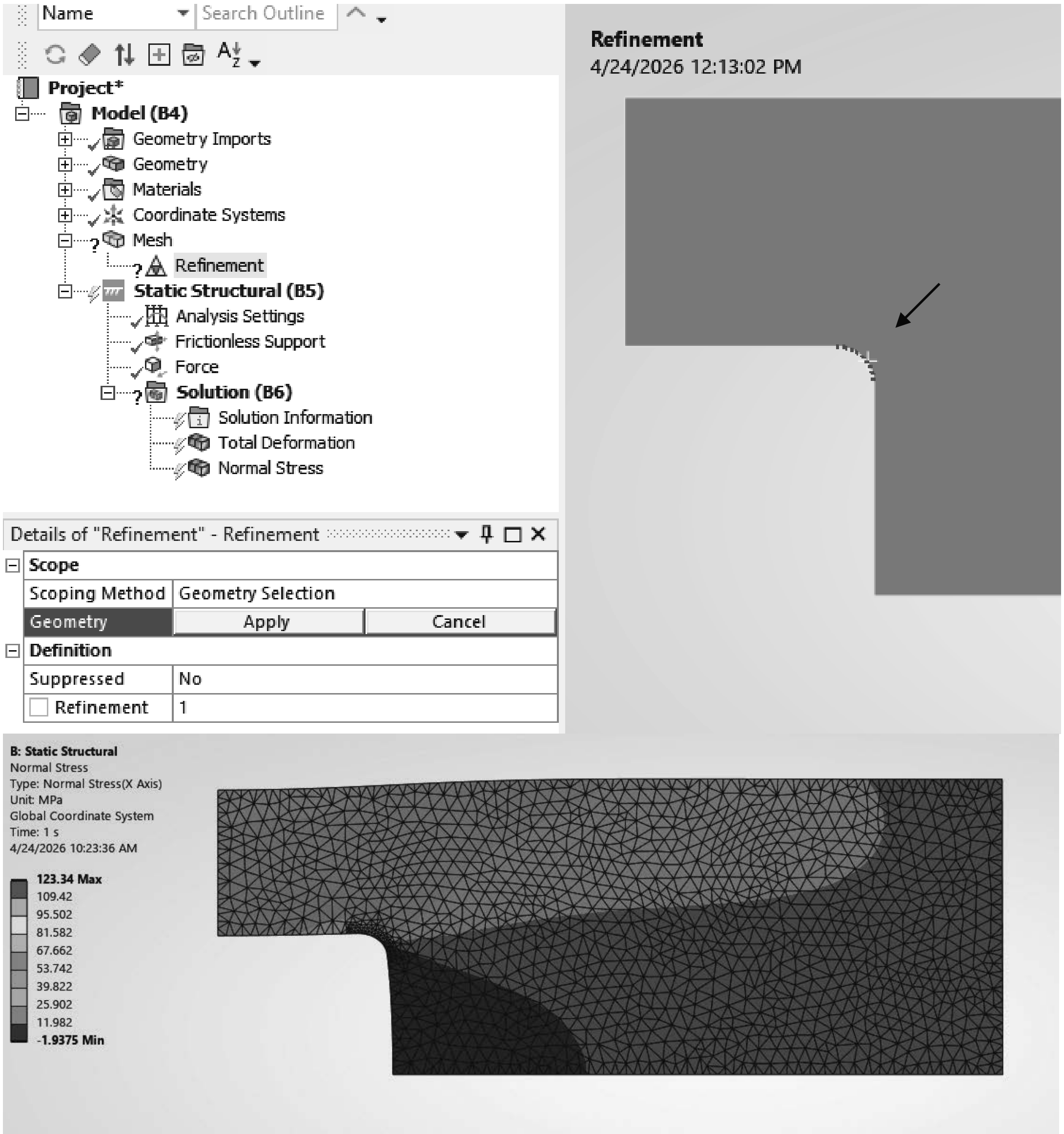
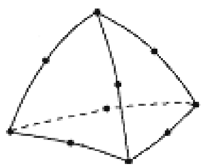


Figure 1-58 Mesh refinement using fillet radius as the target geometry.

The convergence path could be altered by the choice of the mesh refinements. A uniform mesh refinement followed by a local refinement using this convergence object will likely produce different intermediate results. Experimentation and experience are important. **Note** also that to facilitate the automatic meshing process the element was changed by the software from the 20 node brick elements to **ten-node tetrahedron elements**.



24. Save your work.

1-5 SUMMARY

The tutorials in Chapter 1 illustrate basic concepts of stress computation with Structural Static Mechanical simulation coupled with Ansys DesignModeler or another solid modeler. We note that computed values must be **examined carefully** to evaluate their accuracy and that manual or automatic **convergence studies** can and **should** be used to ensure quality of results. **Don't necessarily accept** the results of the **first mesh** you try.

Ansys Elements used in this chapter: **SOLID186,** Homogenous Structural Solid 8-node and 20-node bricks as well as 10-node tetrahedral, pyramid and prism variations as required by automatic meshing of volumes.

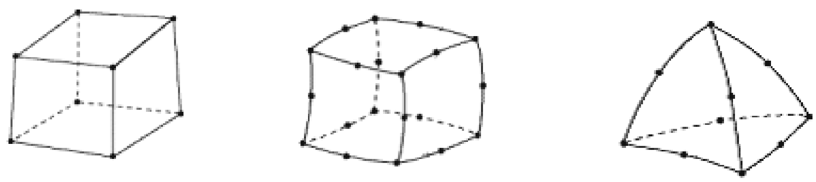


Figure 1-59 Elements used in this chapter.

In the next chapter we will extend these ideas to more complex parts and introduce additional simulation modeling options.

1-6 PROBLEMS

Throughout the remainder of the book express your computed results using **3** to **4 significant digits accuracy** unless otherwise indicated and convert computed results so as to be able to present results in both systems of generally accepted units for engineering work. That is, work the problem **using the units in which it is defined**, show the computed results in those units, and convert the results so that you can present answers in both **MKS** (**mm KS**) and **IPS** (**FPS**) systems. See Tutorial 1A.

Always **show the meshes** you use in your solution, and for problems in stress analysis, it's also a good idea to first plot **UNAVERAGED results** as a quick visual check on accuracy; again see T1A.

1-1 Consider a thin **structural steel** plate as shown below. Compute the **maximum normal stress** in the horizontal, X direction if **100 kN tensile load** is applied to the left and right end surfaces. Compare your result with the value you compute using tabulated stress concentration factors for this geometry.

Use the three planes of symmetry this problem has and analyze only **one eighth** of the solid. Put frictionless supports on the planes of symmetry and apply **one-fourth** of the load, that is (100/4) kN. See Tutorial 1A, step 15. Present both **Averaged** and **Unaveraged** Stress results plots.

If you have time, use only two planes of symmetry, apply one half of the load as we did in Tutorial 1A1. Compare your results to those obtained using three planes of symmetry.

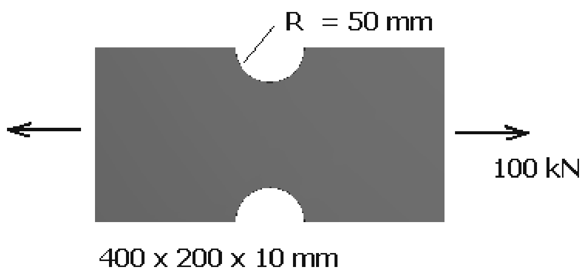


Figure P1-1

Figure P1-2

1-2 Place a 25 mm diameter hole 100 mm to left and to the right of the vertical centerline but at the quarter points on the horizontal centerline of the part in Problem 1-1. Find the magnitude and location of the **maximum normal stress**. Compare this result to the theoretical maximum stress in the plate **without the added holes** using tabulated stress concentration factors to determine the theoretical result.

1-3 Consider the long, thin structural steel beam shown below, fixed at one end, free at the other. Apply a pressure of **200 psi** on the upper edge. Compute the vertical

deflection at the free end and the **maximum bending stress** at the **top edge** at the **mid length** of the beam. Look up the theoretical solution in your solid mechanics book and compare your FEM results with the results you can compute from beam theory.

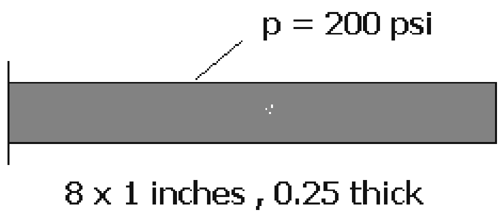


Figure P1-3

1-4 Repeat the problem described in Problem 1-3 but make the beam **1 x 1 x 0.25 inches**. Slender beam theory no longer applies to this geometry. Determine the theoretical solution for the end deflection of this cantilever beam from solid mechanics theory and compare with your FEM result. Comment on the differences.

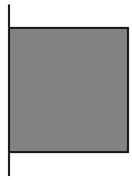


Figure P1-4

1-5 Repeat the problem described in Problem 1-3 but drill a 0.25 inch diameter hole at the center of the beam.

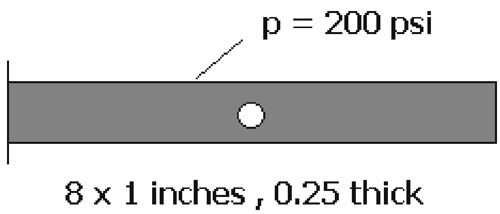
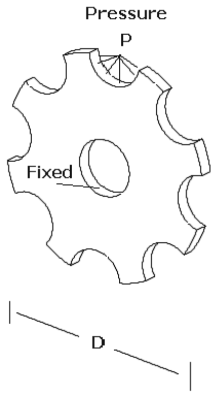


Figure P1-5

1-6 The central bore of the indexing wheel and the 8 slots are all 1.0 inch in diameter. The thickness is 0.5 inch and the diameter D = 4 inches. Find the **maximum principal strain** in **pressurized indexing slot** if the applied pressure is 5,000 psi. Ignore the strains in the **completely fixed central hole**. Report the calculated decimal value and also report the result in terms of **micro-strain**, that is, the computed value multiplied by 10^6. See figure below.

1-7 Repeat problem 1-6 but use a vertical plane to slice off half of the geometry and put a frictionless support on the exposed plane of symmetry.



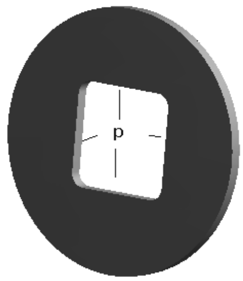


Figure P1-6 and **Figure P1-7**

1-8 A 20 inch diameter cylinder has an 8 in square hole with 1.0 inch radius corner fillets. The thickness is 0.75 inch. Find the maximum values for **von Mises stress**, **principal stress**, **shear stress**, and **displacement** if the front and back surfaces have **frictionless supports** (condition of plane strain) and the interior has a pressure of 7500 psi.

1-9 The **structural steel frame shown** is **fixed** at its base and **loaded** with a **pressure** of **P = 2 MPa** as shown. Cut a section plane through the vertical segment at point ***a*** halfway up. Draw a **free body diagram** showing the forces and moment acting on the cross section of the upper portion. Use a **hand calculation** to compute the **Bending Stress +/- Axial Stress** at the left edge point ***a***. **Also** use **Ansys Workbench** to compute the maximum vertical stress and compare your results.

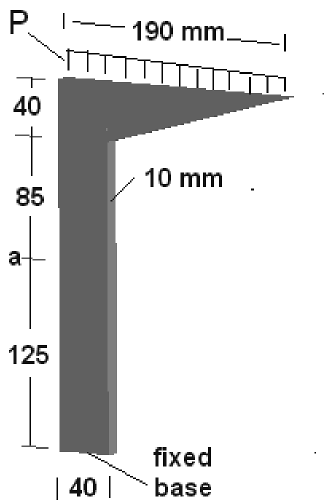


Figure P1-8

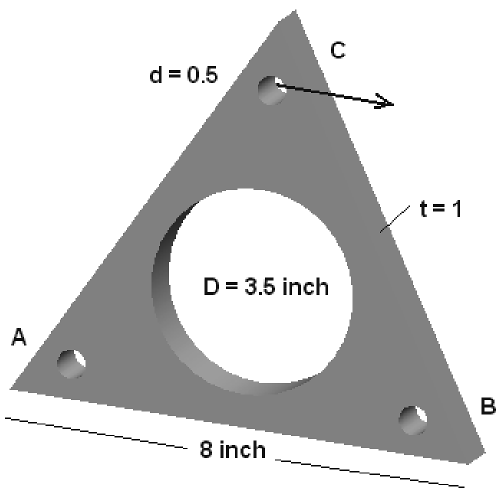


Figure P1-9

1-10 The equilateral triangle is **8 inches** on each side, and the **thickness** is **1 inch**. The **0.5 inch** diameter holes are symmetrically placed radially **3.5 inches** from the **center**.

The part is supported at holes A and **B with cylindrical supports,** fixed radially and axially but free tangentially to allow rotation. A **horizontal load** in the mid plane of the plate of **15,000 lbf** is applied to hole **C.** Find the magnitude and location of:

a. max von Mises Stress

b. max Principal Stress

c. max Principal Strain

d. max Horizontal Displacement

NOTES: