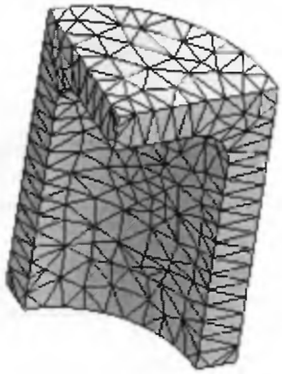
Chapter 5

ANSYS Mechanical II



5-1 OVERVIEW

This chapter covers stress and deflection simulation response of some three-dimensional solids representative of typical mechanical parts. We consider the simulation of the following objects

- ♦ Pressure Vessel
- ♦ Angle Bracket
- ♦ Clevis Yoke

5-2 INTRODUCTION

The close integration with DesignModeler and other solid modeling tools makes ANSYS Mechanical particularly well suited to the analysis of solids. We consider a few typical examples in the tutorials that follow.

5-3 TUTORIAL 5A - CYLINDRICAL PRESSURE VESSEL

A steel pressure vessel with planar ends is subjected to an **internal pressure of 35 MPa**. The vessel has an **outer diameter of 200 mm**, an over-all **length of 400 mm** and a **wall thickness of 25 mm**. There is a **25 mm fillet radius** where the interior wall surface joins the end cap as shown in the figure below.

The vessel has a longitudinal axis of **rotational symmetry** and is also symmetric with respect to a plane passed through it at mid-height. Thus the analyst need consider only the **top** or **bottom half** of the **vessel**.

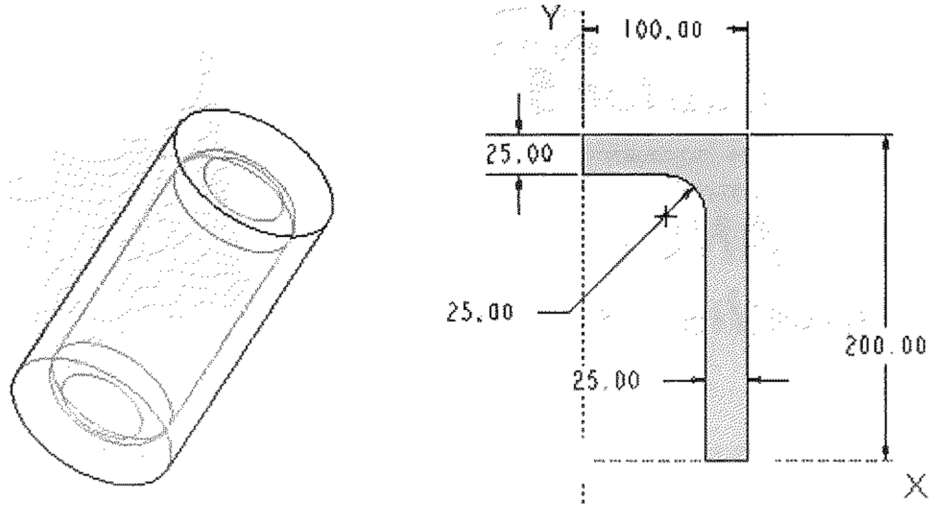


Figure 5-1 Cylindrical pressure vessel; dimensions in mm.

We will use a 90-degree segment of the solid model of the vessel for analysis. The symmetric nature of the geometry and loading means that displacements are zero in directions normal to the faces exposed by the vertical and horizontal cuts employed to create this one-eighth segment of the cylinder. Use ANSYS DesignModeler or other solid modeler to create a solid model of the upper portion of the vessel.

1. **Start ANSYS Workbench** and **DesignModeler** (or other Solid Modeler). **Create** or **attach the quadrant geometry**, and **Save** the **Project** using the name **T5A**.

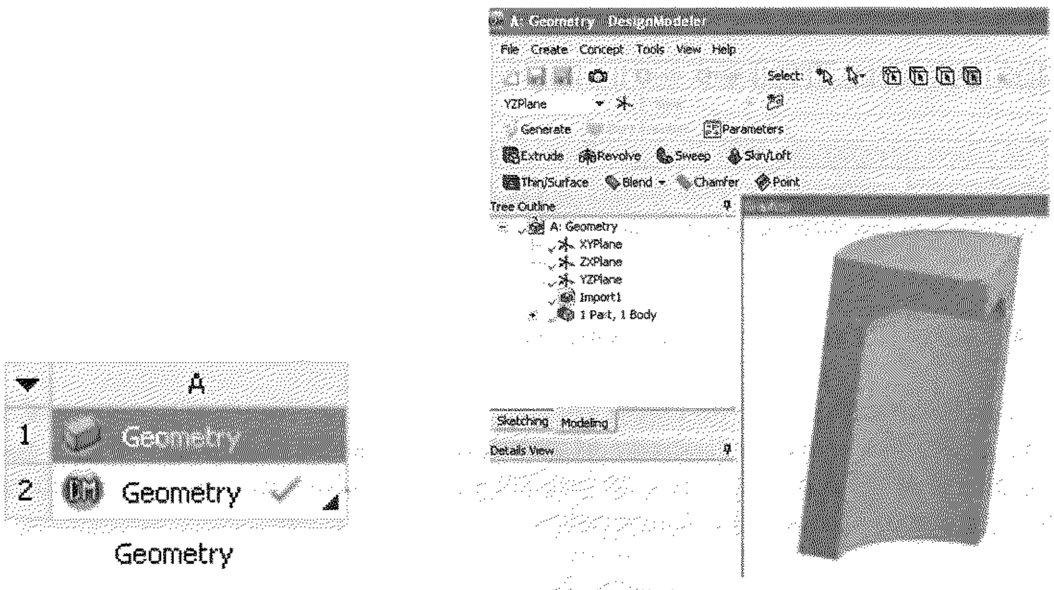


Figure 5-2 Project Schematic and quadrant of cylinder.

2. Add **Static Structural** from the **Toolbox > Analysis Systems** and **Share** the **Geometry.** Double Click **Model**.

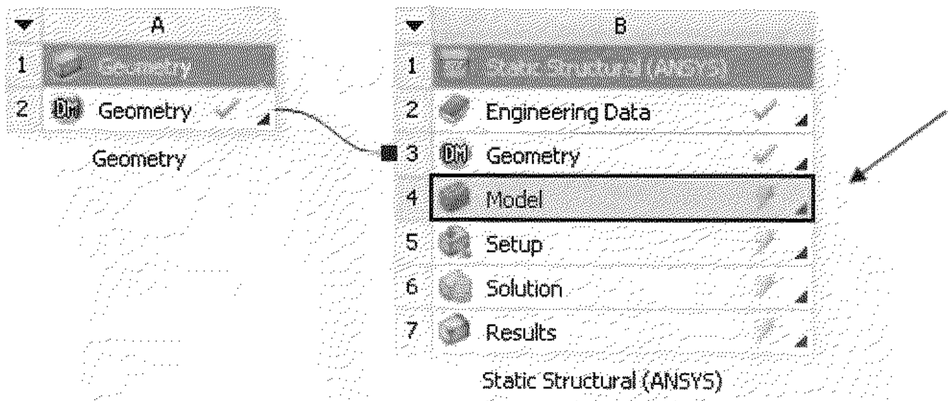


Figure 5-3 Project Schematic.

We will use the **default materials values for steel**. **Check the units settings**, then Right click **Model (B4)** and **rename as T5A-Vessel**.

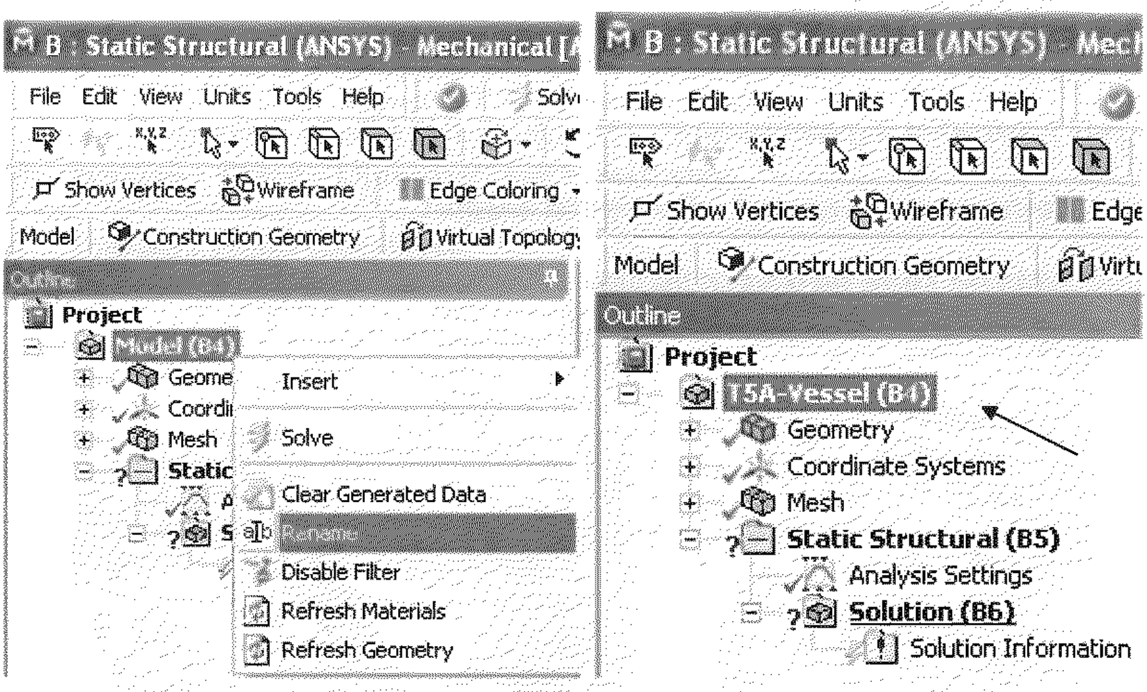


Figure 5-4 Rename Outline object.

3. **Mesh >** (right click) **Generate Mesh**

The default mesh shown below is created. The default mesh uses tetrahedral elements and this has only one element through the thickness. Let's replace it with a finer mesh.

4. **Mesh > Right click > Clear Generated Data > Yes** (This removes the mesh.)

5. **Details of Mesh > Sizing > Relevance Center > Medium.** Generate new mesh.

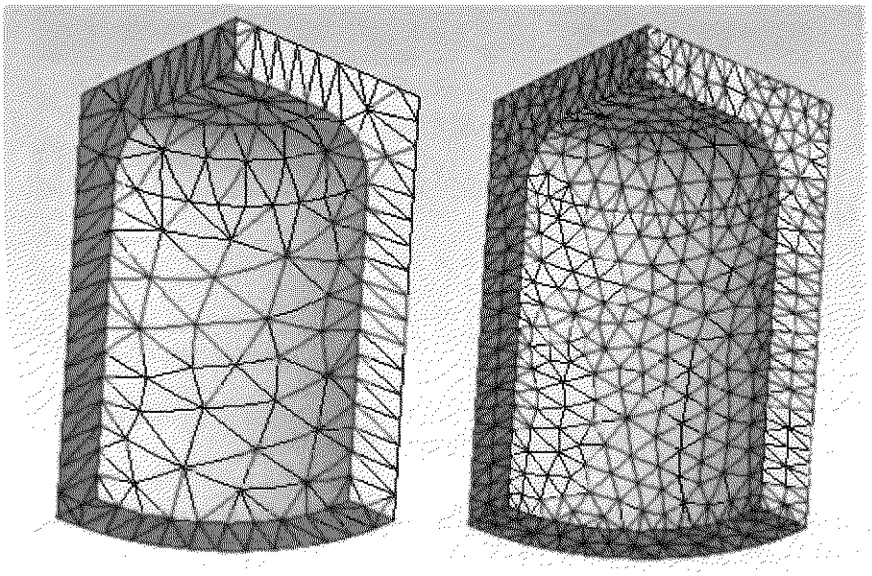


Figure 5-5 Initial and improved mesh.

Apply **Frictionless Support** boundary conditions on the planes of symmetry.

6. **Environment > Supports > Frictionless Support**

7. Select the **Three surfaces** on the planes of symmetry **> Apply.**

Details of "Frictionless Support"

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

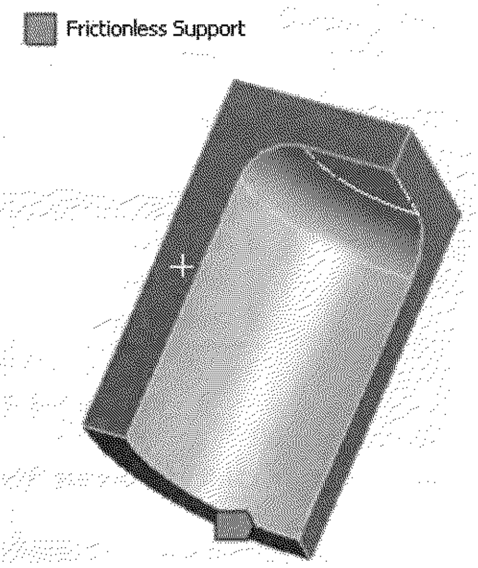


Figure 5-6 Displacement boundary conditions.

8. **Environment > Loads > Pressure > Ctrl Select** all interior surfaces **> Details of "Pressure" > Apply**

9. **Details of "Pressure" > Magnitude > 35 MPa**

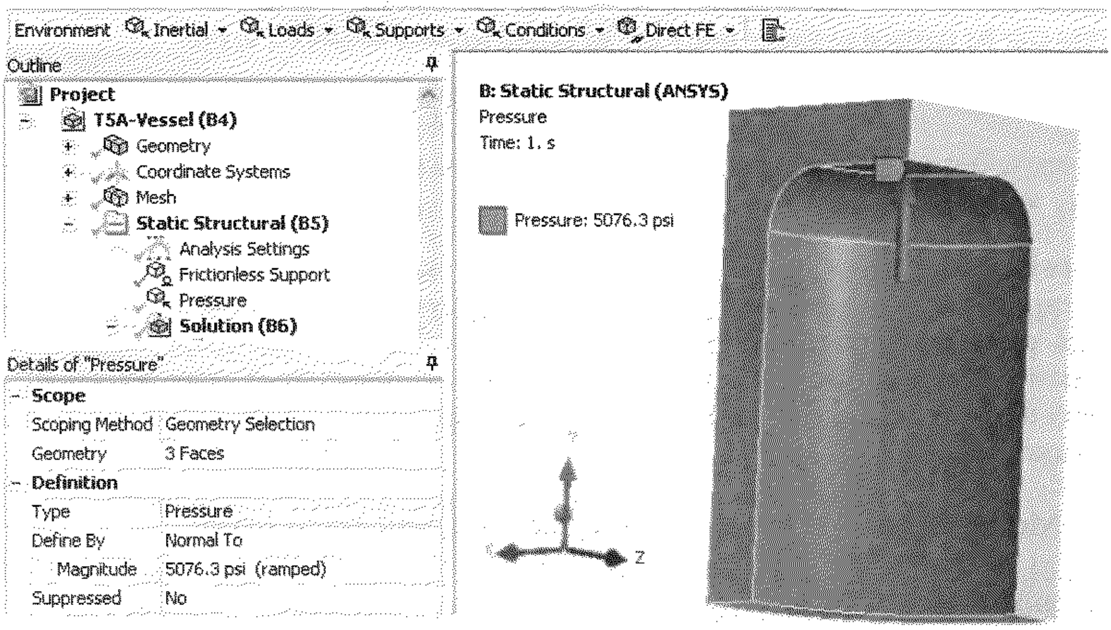


Figure 5-7 Apply pressure loading.

10. **Solution > Stress > Equivalent (von Mises) Stress**

11. **Solution >** (Right click) **Solve**

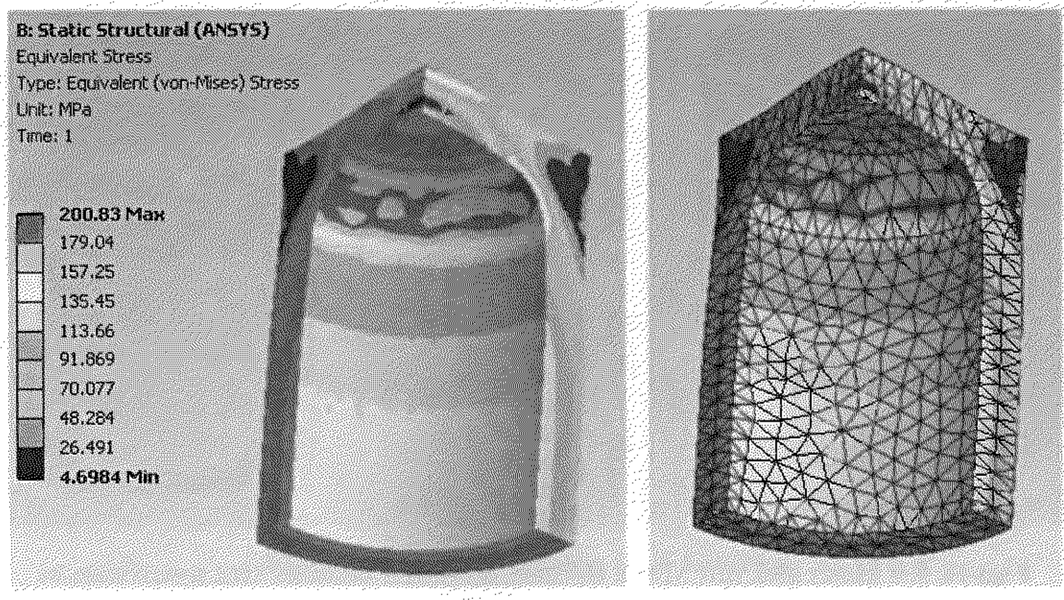


Figure 5-8 Computed von Mises stress using the default medium mesh.

We compute the von Mises or Equivalent Stress for this problem because the load is static and the steel material is ductile. The von Mises stress is the widely accepted predictor of yielding for cases such as this where a multi-axial state of stress is present. If the **von Mises** stress is greater than or equal to the **material yield stress** (as found in a uniaxial tensile test), yielding, however localized it may be, is predicted. The Equivalent Stress can be computed easily using the principal stresses as shown below.

$$\sigma_e = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2}}$$

We note in the figure above that the stress distribution is not smooth and uniform particularly on the inside corner fillet where the maximum values occur; the **unaveraged stress plot** shows discontinuous contours. The problem is symmetric in geometry, material properties, loading, and boundary conditions about the Y axis, so we would expect the solution to be also. We will manually adjust the mesh density to improve the solution.

12. Mesh > Details of "Mesh" > Advanced > Element Size > 4 mm

Now solve the problem again.

13. Solution > (Right click) **Solve**

The new mesh and von Mises stress distribution is shown below. Note that the mesh in most locations has several elements through the 25 mm wall thickness.

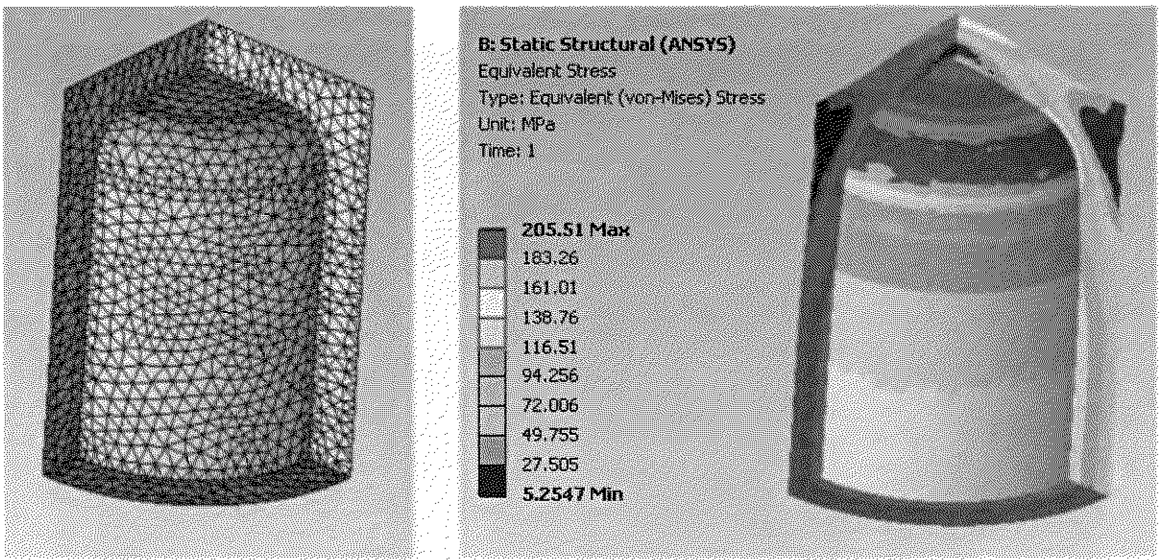


Figure 5-9 Mesh and von Mises stresses for 4 mm element size specification.

These results are not perfect but are much better; we'll accept them and move on. The **Mesh** object in the project tree displays information about the mesh parameters.

14. Mesh > Statistics

Statistics	
Nodes	15185
Elements	8502
Mesh Metric	None

Figure 5-10 Mesh statistics.

Let's find the stress components at the mid-plane of the complete cylinder (the bottom of our model). Add the normal stresses in the X, Y, and Z directions to the items in the solution object.

15. Solution > Stress > Normal > Details of "Normal Stress" > Type > Normal Stress > Direction > X Axis

16. Repeat for Y and Z axis normal stresses

17. Solution > (right click) **Evaluate All Results**

18. Solution > Normal Stress (X axis) The Sx distribution is shown below. Use the "**123 Probe**" tool to display the X stress at the inside of the cylinder on the face normal to the X axis. Probe

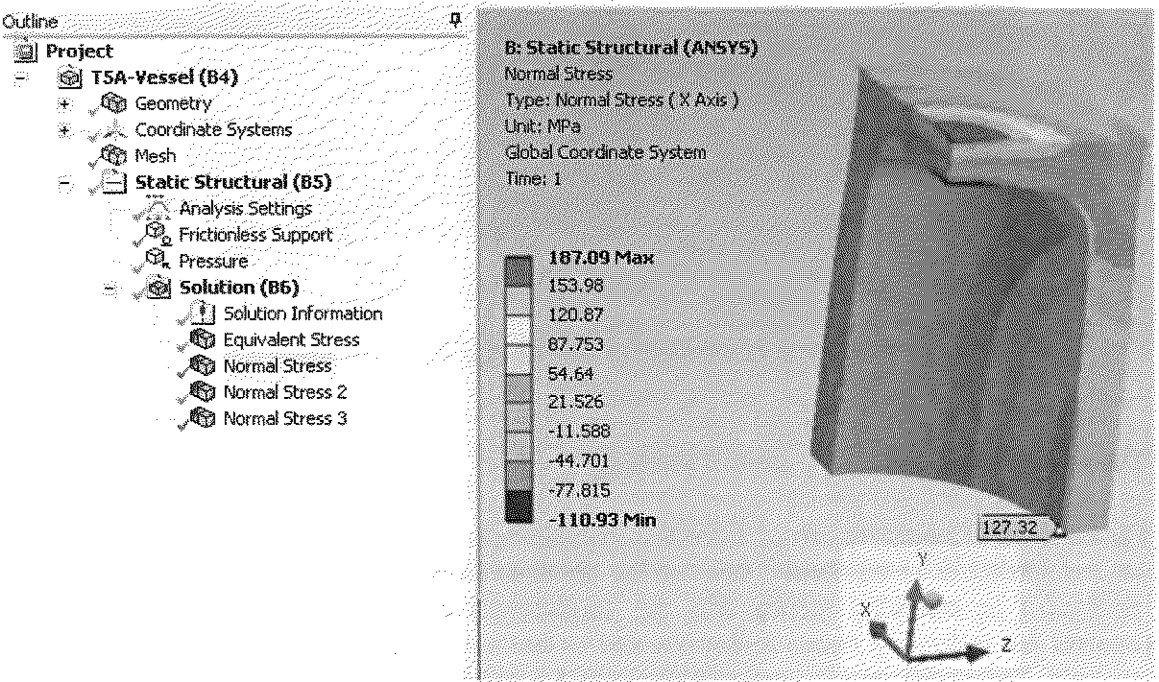


Figure 5-11 Normal stress in X direction.

For the orientation shown, the normal stress in the X direction is what we normally call the cylinder **hoop stress**. The value shown is 127 MPa. Similarly we find the normal stresses in the Y and Z directions at this point to be 46.5 MPa and -34.9 MPa. These correspond to the **axial stress** and the **radial stress** in the cylinder.

Compute the radial deflection also.

19. Solution > Deformation > Directional > Details of "Directional Deformation" > Type > Directional Deformation > Orientation > Z Axis > Evaluate All Results

The result as shown in the next figure is about 0.0465 mm.

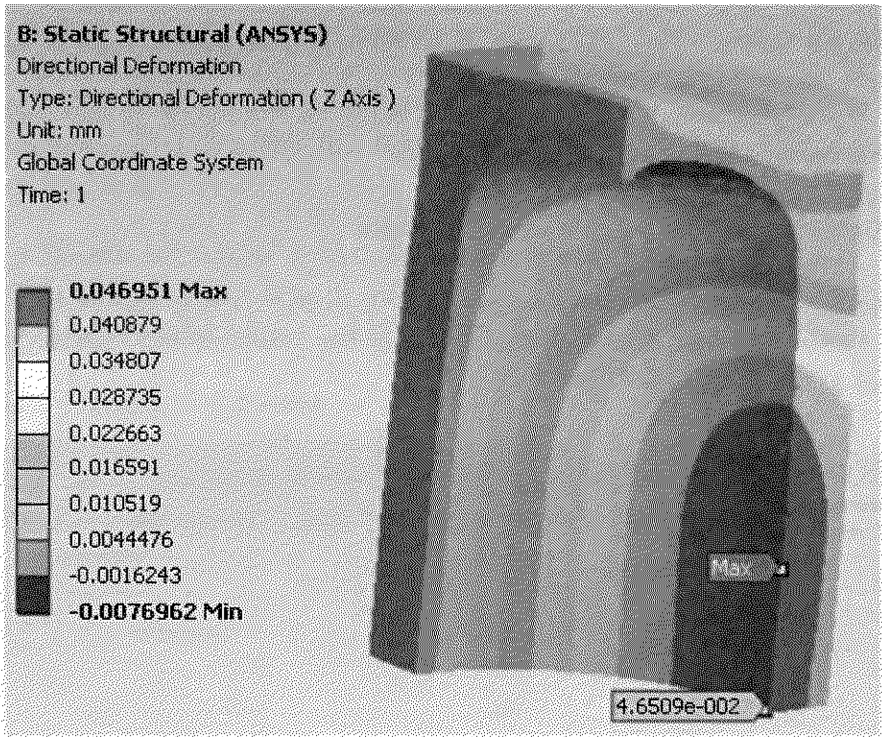


Figure 5-12 Deflection in the Z (radial) direction.

Theoretical Solution

The stresses at points removed from the stress concentration associated with the end caps can be predicted reasonably well using **thick-walled cylinder** equations from solid mechanics or elasticity theory. For the geometry and loading of this example, theory predicts the following radial deflection and stress components at the **inner surface** of the cylinder. The results are summarized in the table below. (Ref: solid mechanics text.)

Table 5A.1 - Pressure Vessel Results

	Theoretical	Workbench	Error, per cent
Hoop Stress (Sx), MPa	125	127	1.9
Axial Stress (Sy), MPa	45	46.5	3.3
Radial Stress (Sz), MPa	-35	-34.9	0.3
Radial Deflection, mm	0.0458	0.0469	1.5

Thus the Workbench model arrives at a solution at vessel midplane within a few per cent of the theoretical values, and, since the theoretical solution assumes a uniform distribution for the axial stress, the Workbench solution may give a better picture of the actual stress state in that instance. Before we quit, let's examine an **alternative meshing**.

In ANSYS Workbench we can manage the mesh in a number of different ways in addition to those already discussed. We illustrate by continuing with an alternate analysis of the pressure vessel.

First clear the results already calculated and return to the basic mesh.

20. Right click Mesh > Clear Generated Data > Yes

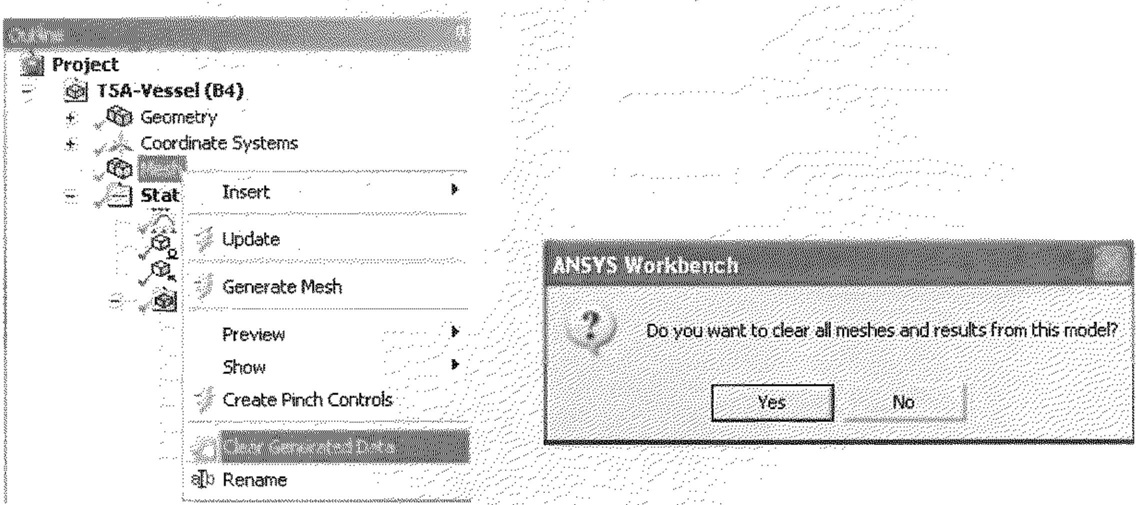


Figure 5-13 Clear results dialog.

21. Mesh > Details of "Mesh" > Advanced > Element Size > (enter 0, return) Default (resets the element size)

22. Mesh > Relevance Center > Fine > Generate Mesh

Since the interior fillet is the source of stress concentration we will use this geometry to refine the mesh.

23. **Mesh** > (right click) **Insert** > **Refinement** (Select the interior fillet; see next figure.)

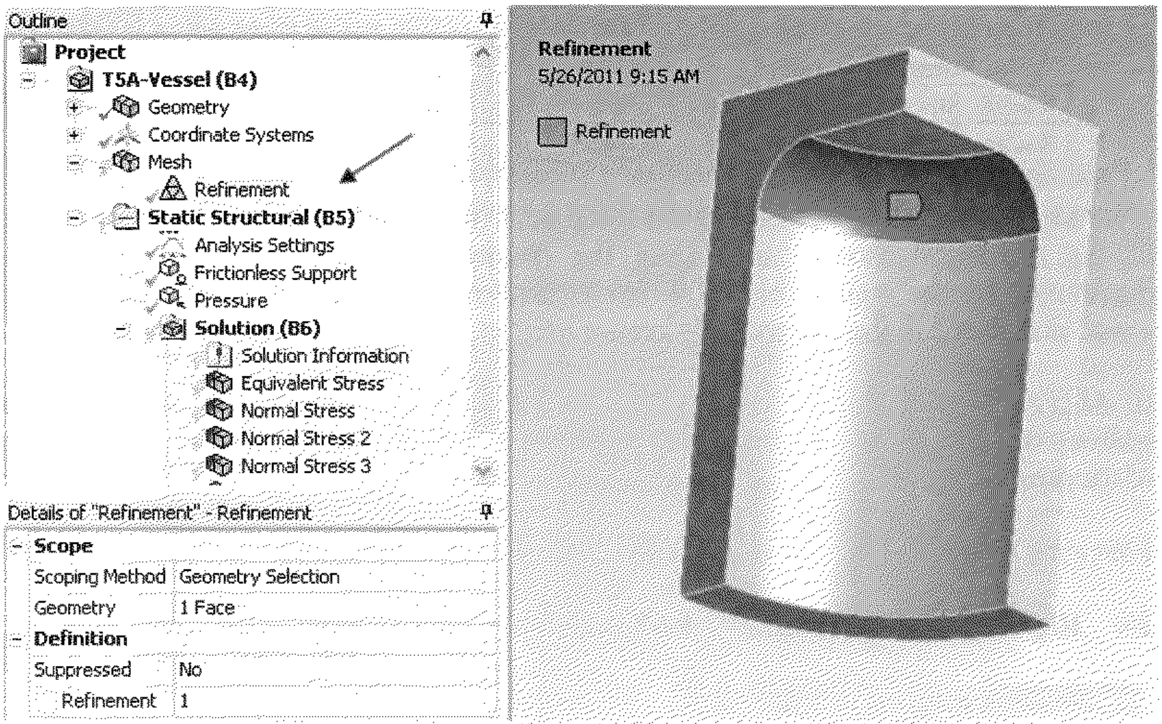


Figure 5-14 Refinement defined by a critical surface.

24. **Details of Refinement > Geometry > Apply; Refinement 1**

25. **Mesh > Generate Mesh** (Gives us the following mesh, dense in the high stress area.)

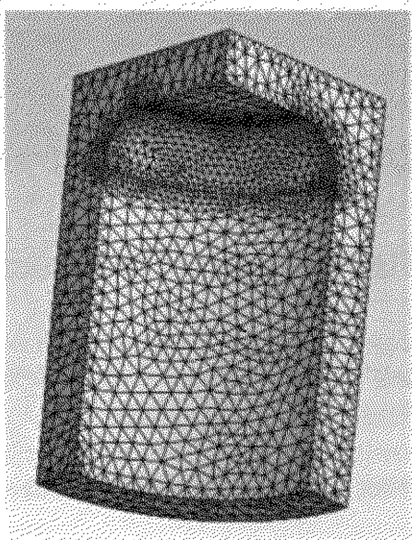


Figure 5-15 Locally refined mesh.

Let's solve this problem and examine the computed results.

26. **Solution > Solve**

The von Mises stress distribution is shown next, and we see reasonably smooth contours and computed magnitudes only slightly different from values for the most dense mesh obtained by manual manipulation of the element size specification in the meshing parameters details dialog box.

The **Solution Information** object or **Mesh > Statistics** shows this model to have about **35,000 nodes** and **22,000 elements.**

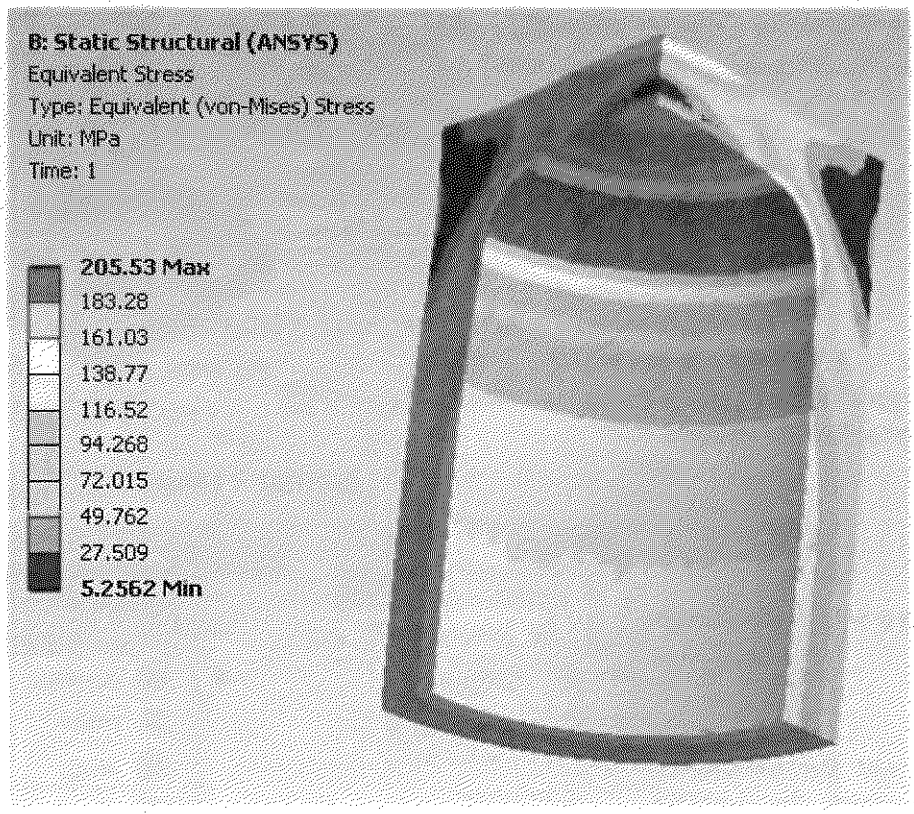


Figure 5-16 Equivalent (von-Mises) Stress.

Another meshing option we have is to insert a convergence object and perform adaptive mesh solutions as discussed in the previous chapter. We leave this as a problem at the end of the chapter.

Because every plane passing through the vertical axis, (Y Fig 5-10) is a plane of symmetry, we can also make this a much smaller problem by creating a model that is just a **small 10 degree wedge** with frictionless supports, instead of the 90 degree wedge used above. Try it. An even smaller, faster solving model is based on a planar, **axisymmetric analysis**. We will solve the problem that way in Chapter 8.