PROBLEM STATEMENT

This example is based on a thick-wall cylindrical pressure vessel that is closed on one end, but attached to a rigid pipe connection at the opposite end as illustrated in Fig. 1. This vessel is made of Alloy Steel and is subject to an internal pressure P_i=16.0 MPa and an external that approximates standard atmospheric pressure P_o=0.101 MPa. Cylinder dimensions necessary to apply classical stress equations are also included in Fig. 1. The goal is to determine tangential and radial stress variation through cylindrical walls of the pressure vessel.

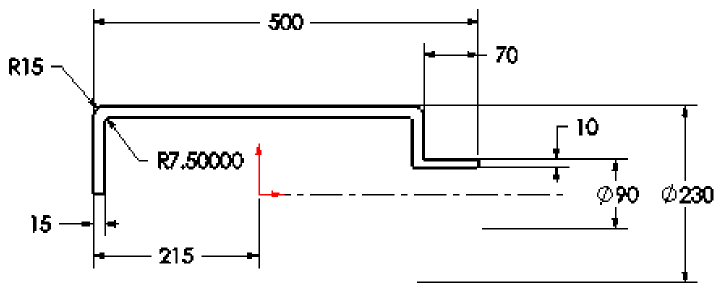
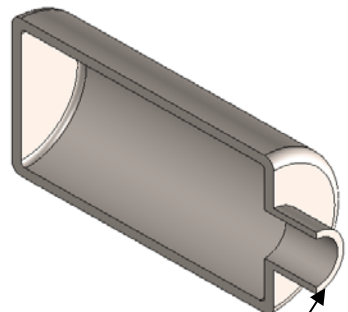


Figure 1 – Bascin dimensions and geometry of the thick-wall pressure vessel



Location of rigid pipe connection

CREATING THE MODEL

1. Create the model based on the dimensions above. NOTE: use the front plane to draw your sketch otherwise you will run into difficulties later in these instructions.

2. Revolve the sketch 180deg.

DEFINING THE STUDY

1. Click on Simulation and select Study from the pull-down menu.

    a. If Simulation is not an option in the menu then make sure it is check in Add-Ins…

2. Select Static and in the Name field, type “Thick Wall Pressure Vessel.”

ASSIGN MATERIAL PROPERTIES

1. Using the Simulation menu select Material and click on Apply Material To All…

2. Because users should be sufficiently familiar with this window, on your own select Alloy Steel (SS) and set the units to SI, then click [OK] to close the window.

DEFINE RESTRAINTS AND LOADS

1. Right click on Fixtures in the Feature Tree and select Fixed Geometry…

   The Fixture property manager opens.

2. In the Advanced options select the Symmetry button.

3. Select the front surface highlighted in Fig. 2.

4. Click on the check box to close the Fixture property manager.

Restraint symbols in the Z-direction appear on the model. Physically this means that existing symmetry restraints permit axial displacements (along the length of the cylinder) and radial displacements (change of cylinder diameter) when the cylinder is subject to internal or external pressure. These displacements are consistent with the half-model selected. At present, however, both X and Y displacements of the model are still possible because it is not fully restrained.

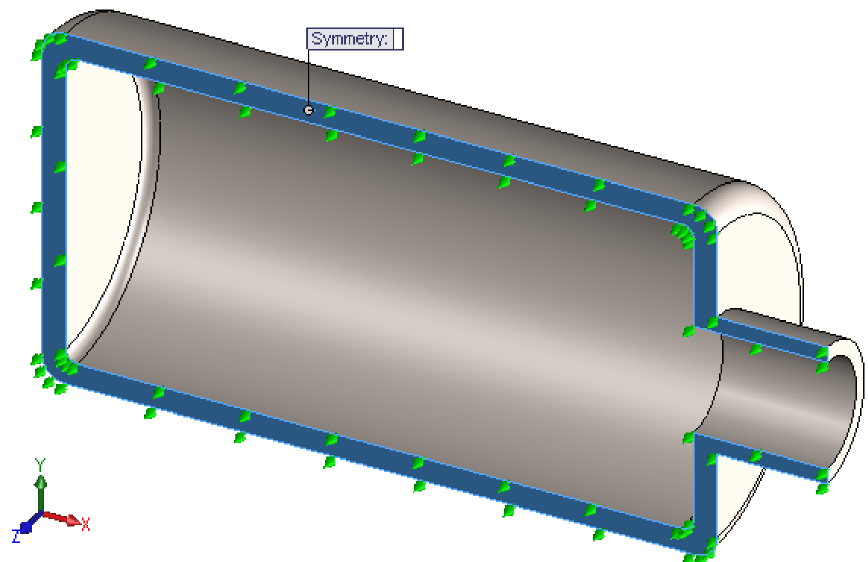


Figure 2 – Half-model of pressure vessel with symmetry restraints applied in the Z-direction.

Because symmetry restraints restrain the model in the Z-direction only, it is necessary to apply additional restraints in the X and Y directions to prevent what is known as "rigid body motion." The type of restraints selected depends heavily upon the nature of actual operating conditions. Because the right end of the pipe connection is considered attached to a rigid (fixed) pipe extension, not shown, the remaining restraints are applied at that location. Fortunately, this fixed restraint does not interfere with finite element results in the thick-wall portion of the pressure vessel.

Continue to define remaining restraints and loads in the following steps.

5. Again, right click on Fixtures in the Feature Tree and select Fixed Geometry…

6. In the Standard options select Fixed Geometry.

7. Zoom in on the right end surface of the pipe extension, shown in Fig. 3, and click to select it. Fixed restraints symbols in the X, Y, and Z directions are added to the model.

8. Click on the check mark to close the Fixture property manager.

   The model is now fully restrained in all directions.

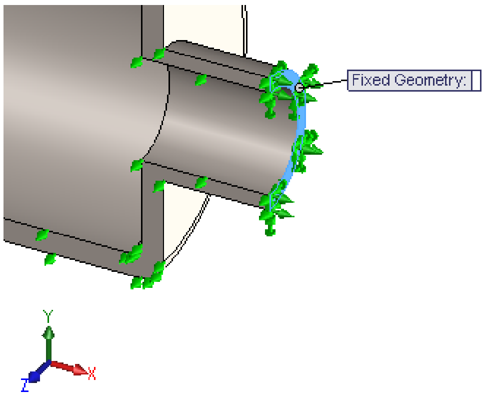


Figure 3 – The Fixed Geometry restraint added to the attachment surface associated with a rigid pipe connection.

The next step is to apply an internal pressure Pi=16.0 MPa to all internal surfaces of the pressure vessel. Proceed as follows.

9. Right-click External Loads and choose Pressure…

   The Pressure property manager opens.

10. For Pressure Type select ● Normal to selected face (if not already selected).

11. The Faces for Pressure field is highlighted to indicate it is active and awaiting user input.

12. Move cursor onto the model and click to select all interior surfaces (both cylinder ends, both internal fillets, and cylindrical surfaces of the cylinder and the pipe extension). A total of six faces (Face<1>, Face<2>, …Face<6>) should be listed in the Faces for Pressure field.

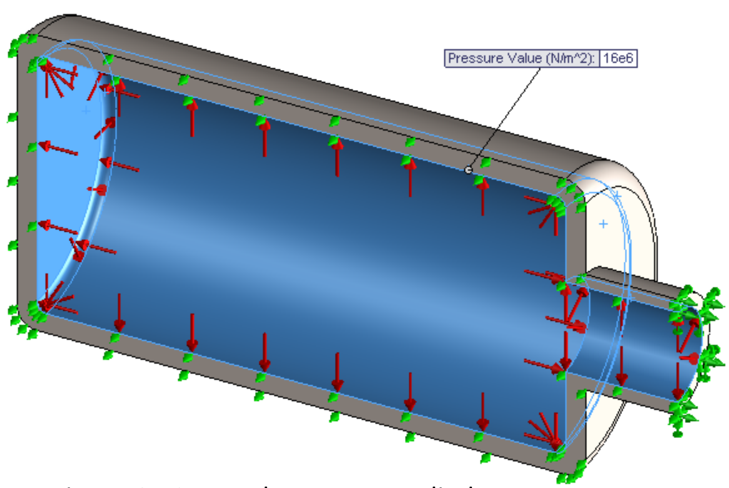


Figure 4 – Internal pressure applied to six faces within the thick-wall pressure vessel.

13. For Pressure Value, set the Units field to SI and in the Pressure Value field type 16.0e6 N/m^2.

14. Click on the check mark to close the Pressure property manager. The model should appear as shown in Fig. 4.

Next apply atmospheric pressure Po=0.101 MPa to all external surfaces of the model.

15. Repeat steps 9 through 14 with the following two exceptions.

    a. In step 12, select all *exterior surfaces*. Once again, a total of six faces should be chosen.

    b. In step 13, type 0.101e6 N/m^2 in the Pressure Value field.

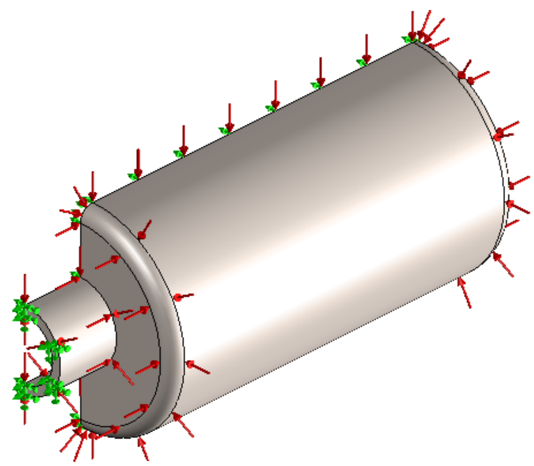


Figure 5 – Atmospheric pressure applied to exterior of the thick-wall pressure vessel.

Upon completion of step 15 (a) and (b), a view of the exterior of the pressure vessel should appear as shown in Fig. 5.

Mesh the Model

1. Right-click on the study and choose Properties… from the menu as shown in Fig. 6.

    a. Verify that the FFEPlus Solver is selected.

    b. Click [OK] to close the window.

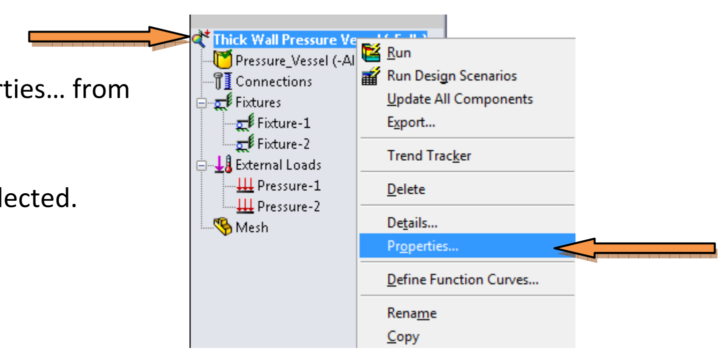


Figure 6

2. Right-click the Mesh folder and from the pull-down menu select Create Mesh… The Mesh property manager opens.

3. Under Mesh Density drag the slider toward Fine stopping in the approx. location shown in Fig. 7.

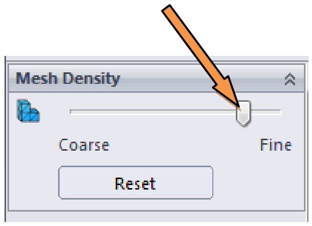


Figure 7 – Mesh Density Slider option

4. Under Mesh Parameters verify that other defaults settings appear as follows.

    a. Mesher Type set at Standard

    b. Verify that Units are expressed in mm

    c. Jacobian check: set to 4 points

5. Click on the check mark to complete the Mesh.

6. Verify that you have two elements span the thick-wall portion of the model, and only one element spans the thickness in the pipe connector neck as shown in Fig. 8. If not adjust the slider in step 7 until you get the correct elements.

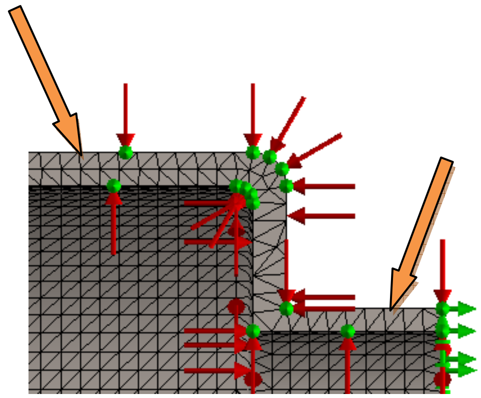


Figure 8 – Mesh elements

SOLUTION

Having defined restraints, loads, and mesh, the thick-wall model is subjected to analysis as outlined below.

1. Using the Simulation menu select Run.

    a. Delete any results that may appear under the Results folder.

DISPLACEMENT ANALYSIS

1. Right-click on the Results folder and select Define Displacement Plot…

    a. Make sure URES: Resultant Displacement is listed under the Display options.

    b. Verify that Deformed Shape is checked as well as Show colors.

2. Complete the Plot using the check mark.

3. Hide all of the Fixtures and External Loads.

4. Move the cursor over the screen image to display an un-deformed outline of the model as illustrated in Fig. 9. Placing the cursor near a corner fillet or the mid-point of a wall works best.

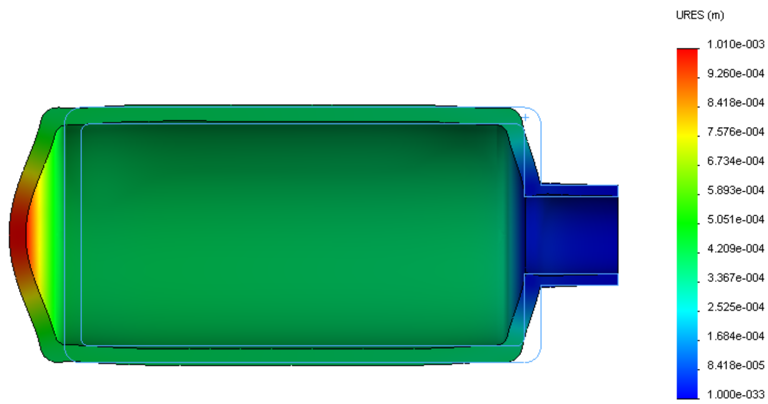


Figure 9

A comparison of the displacement plot with the un-deformed shape of the pressure vessel reveals the following, common sense, observations.

a. All displacements are away from the immovable pipe-end.

b. Bulging of both the left and right ends of the model contributes to longitudinal (axial) deformation away from the immovable end.

c. Longitudinal deformation (axial stretch) of the cylindrical section contributes to overall axial deformation.

d. Slight radial deformation (bulging) of cylindrical walls is observed.

e. Virtually zero displacement occurs in the pipe connection.

VONMISES STRESS ANALYSIS

The next topic for investigation is interpretation of stresses occurring in the model.

1. Right-click on the Results folder and select Define Stress Plot…

    a. Make sure VON: von Mises Stress is listed under the Display options.

    b. Verify that Deformed Shape is checked.

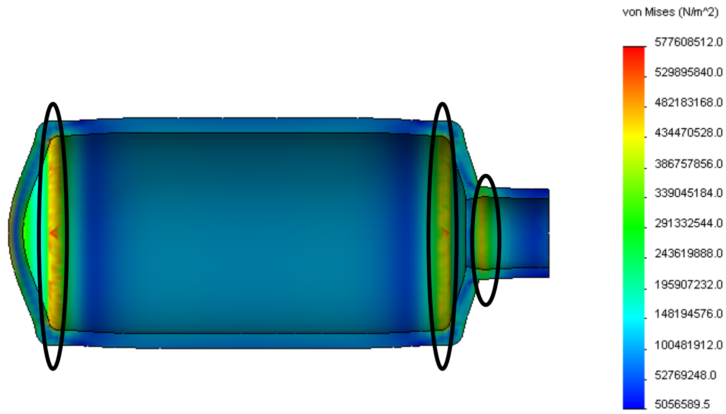


Figure 10

Notice that all plots and plot settings established for the Shell analysis are carried forward into the current analysis due to the Options settings defined at the beginning of this chapter. Regions of high von Mises stress at fillet radii on both ends of the model and in the vicinity of the pipe-to-cylinder connections are circled on Fig. 10. All von Mises stresses are below the material Yield Strength listed in the figure.

CALCULATING TANGENTIAL AND RADIAL STRESS

The next focus of this study is on tangential and radial stress distributions typically associated with analysis of pressurized thick-wall cylinders. However, before examining stress plots, it is helpful to have some expectation of magnitudes associated with these two stresses. Therefore, solutions based on Lame's equations for values of tangential and radial stresses at both the inside and outside surfaces of a thick-wall cylinder are included below.

Tangential stress at outside surface:

$$(\sigma_t)_o = \frac{p_i r_i^2 - p_o r_o^2 - \dfrac{r_i^2 r_o^2 (p_o - p_i)}{r_0^2}}{r_o^2 - r_i^2}$$

Tangential stress at inside surface:

$$(\sigma_t)_i = \frac{p_i r_i^2 - p_o r_o^2 - \dfrac{r_i^2 r_o^2 (p_o - p_i)}{r_i^2}}{r_o^2 - r_i^2}$$

Radial stress at inside surface:

$$(\sigma_r)_i = \frac{p_i r_i^2 - p_o r_o^2 + \dfrac{r_i^2 r_o^2 (p_o - p_i)}{r_i^2}}{r_o^2 - r_i^2} = \frac{p_i r_i^2 - p_o r_o^2 + p_o r_o^2 - p_i r_o^2}{r_o^2 - r_i^2} = \frac{-p_i (r_o^2 - r_i^2)}{(r_o^2 - r_i^2)} = -p_i$$

Radial stress at outside surface:

$$(\sigma_r)_o = \frac{p_i r_i^2 - p_o r_o^2 + \dfrac{r_i^2 r_o^2 (p_o - p_i)}{r_o^2}}{r_o^2 - r_i^2} = \frac{p_i r_i^2 - p_o r_o^2 + p_o r_i^2 - p_i r_i^2}{r_o^2 - r_i^2} = \frac{-p_o (r_o^2 - r_i^2)}{(r_o^2 - r_i^2)} = -p_o$$

Viewing these results is investigated below:

TANGENTIAL AND RADIAL STRESS ANALYSIS

The next topic for investigation is interpretation of Tangential stresses occurring in the model.

1. Right-click on the Results folder and select Define Stress Plot…
    a. Make sure Z Normal Stress is listed under the Display options.
    b. Verify that Deformed Shape is unchecked.
    c. Click on the check mark to complete the plot.
2. Next we will determine the actual values of the Tangential Stress.
    a. Right mouse click on the Z Normal Stress results and choose probe.
    b. Zoom into the top center of the vessel. Pick the bottom line and 3 points above it and conclude with the top line as shown in Fig. 11.

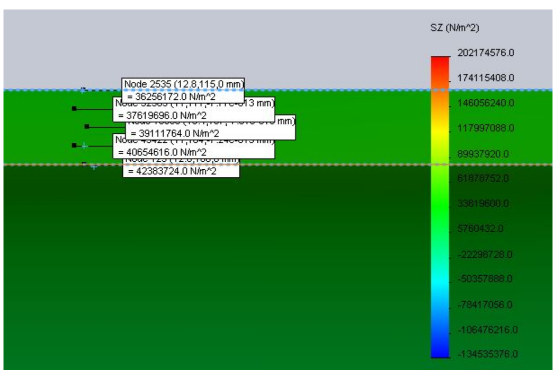


Figure 11

    c. While still in the Probe Results window and with the 5 points listed in the results click on the plot icon under the Report Options to obtain your results plot as shown in Fig. 12.

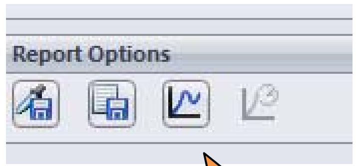


Figure 12

3. Repeat steps 1 and 2 to determine your Radial Stress, but choose Y Normal Stress instead of Z Normal Stress.

A quick comparison of classical results found using Lame's equations and finite element results for tangential stress magnitudes at the inside and outside wall surfaces yields the following:

At the inside wall surface:

$$\%\ difference = \left[\frac{FEA\ results - classical\ results}{FEA\ results}\right]100$$

At the outside wall surface:

$$\%\ difference = \left[\frac{FEA\ results - classical\ results}{FEA\ results}\right]100$$

This concludes examination of tangential stress variation through the cylinder wall by conventional methods.

Screen Capture the following to provide in your report:

Note: Windows 7 has a nice screen capture tool called Snipping Tool which can be found by searching for it in the start menu.

1. Loads and Restraints shown with arrows
    a. Change the arrow colors and provide a legend
2. Mesh image showing two elements span the thick-wall portion of the model, and only one element spans the thickness in the pipe connector neck
3. von Mises Stress
    a. Also include a second image showing the max and min areas
4. Radial Stress image
    a. Also include a second image showing the 5 points that you probed.
    b. The plot graph of the 5 points probed.
5. Tangential Stress image
    a. Also include a second image showing the 5 points that you probed.
    b. The plot graph of the 5 points probed.