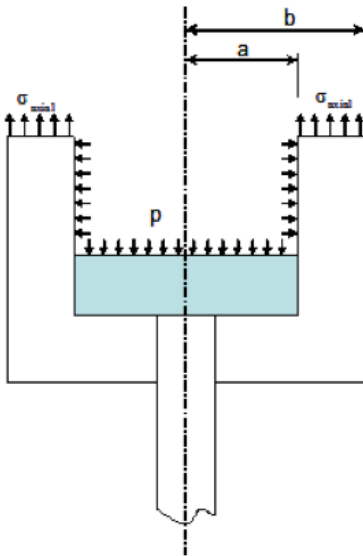
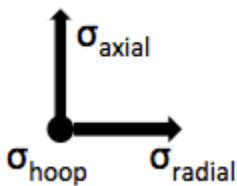

3D Finite Element Analysis of Thick-Walled Pressure Vessels

1. Problem Specification

Consider the following pressurized thick-walled hydraulic cylinder. The following figure shows a section through the mid-plane.



Stress directions in cylindrical coordinates:



σ_{hoop} is in the circumferential direction (out of the plane here)

a = inner radius = 1.5 in

b = outer radius = 2 in

Tutorial: FEA of Thick-Walled Pressure Vessels

Assume the cylinders are 18 inches long and the vessel is pressurized to 1000 psi. Here, we will be interested in finding the hoop, axial and radial stresses at the mid-length of the cylinders (@ 9 inches), to neglect the local effects of the end caps.

Compare the finite element results obtained from axisymmetric analysis to those calculated with the theoretical formulae for both thin-wall and thick-wall approximations.

Note: For this problem, the material choice will not affect the stresses; it will only affect the displacements and strains.

2. Pre-Analysis & Start-Up

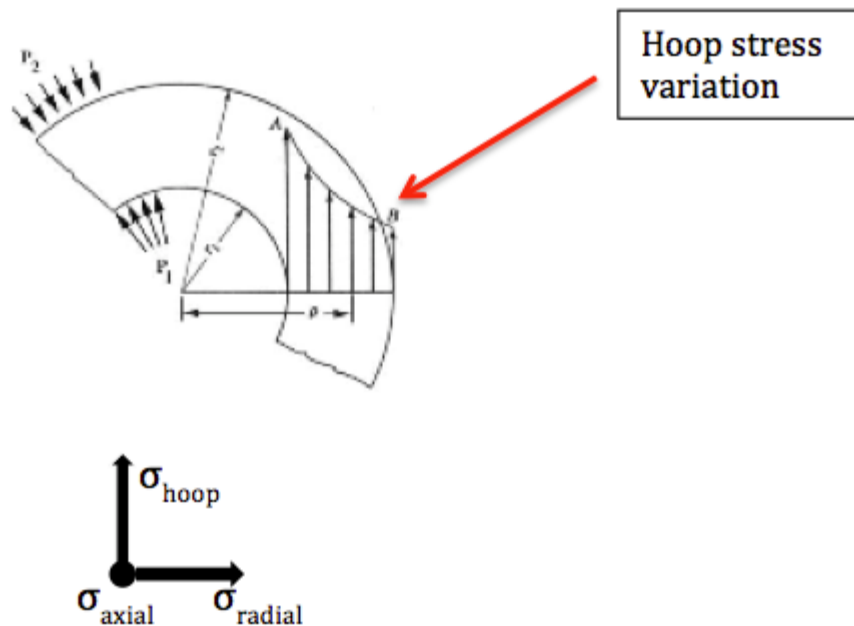
Pre-Analysis

The equations for stresses in thin- and thick-wall cylinders can be found in many mechanics of materials references, and are summarized here, with a = inner radius, b = outer radius, r = radial position where stress is to be found, and t = wall thickness.

	Thin-wall	Thick-wall
Hoop Stress	$\sigma_{\theta} = \frac{pa}{t}$	$\sigma_{\theta} = \frac{pa^2 (r^2 + b^2)}{r^2 (b^2 - a^2)}$
Axial Stress	$\sigma_{ax} = \frac{pa}{2t}$	$\sigma_{ax} = \frac{pa^2}{b^2 - a^2}$

Notice that in thick-wall theory, the hoop stress varies with the radial position, while the stress is assumed to be constant in thin-wall theory. Comparing the substitution of a and b for r in the hoop stress thick-wall equation will convince you that stress is greater on the inner surface. The hoop stress variation in thick-walled vessels can be depicted as follows (the view shown corresponds to looking from above the pressure vessel):

Tutorial: FEA of Thick-Walled Pressure Vessels



By using the parameters given in the problem statement and the above formulae for hoop stress, we find that the maximum hoop stresses using the thin-wall and thick-wall approximations yield **3000 psi** and **3571 psi**, respectively. This corresponds to a 16% difference which tells us that the thin wall theory might not be adequate for this geometry. Thin-wall theory actually gives good results when b/a ratio is less than 1.10, and that is not the case here.

Notice that the axial stresses are constant for both theories since they do not depend on radius. For this example, the thin-wall and thick-wall approximations yield **1500 psi** and **1285 psi**, respectively.

The radial stresses at the inner and outer surfaces can be deduced from the boundary conditions:

- The radial stress at the outer surface is **0 psi** since the traction is zero at a free surface.
- The radial stress at the inner surface is **-1000 psi** since it has to equal the applied normal traction (radial direction is also the normal direction here). The negative sign indicates that the applied traction is compressive.

The following tables display the results of these approximations:

Tutorial: FEA of Thick-Walled Pressure Vessels

		FEA Axisymmetric	Thin-Wall Equation	Thick-Wall Equation
HOOP STRESS (psi)	Inner Surface	?	3000	3571.4
	Outer Surface	?	3000	2571.4
AXIAL STRESS (psi)	(Constant for both theories)	?	1500	1285.7

		FEA Axisymmetric	Boundary Conditions
RADIAL STRESS (psi)	Inner Surface	?	-1000
	Outer Surface	?	0

We will fill in the missing information by performing an axisymmetric analysis using ANSYS. We will then compare the numerical results from ANSYS to the analytical results.

Start-Up

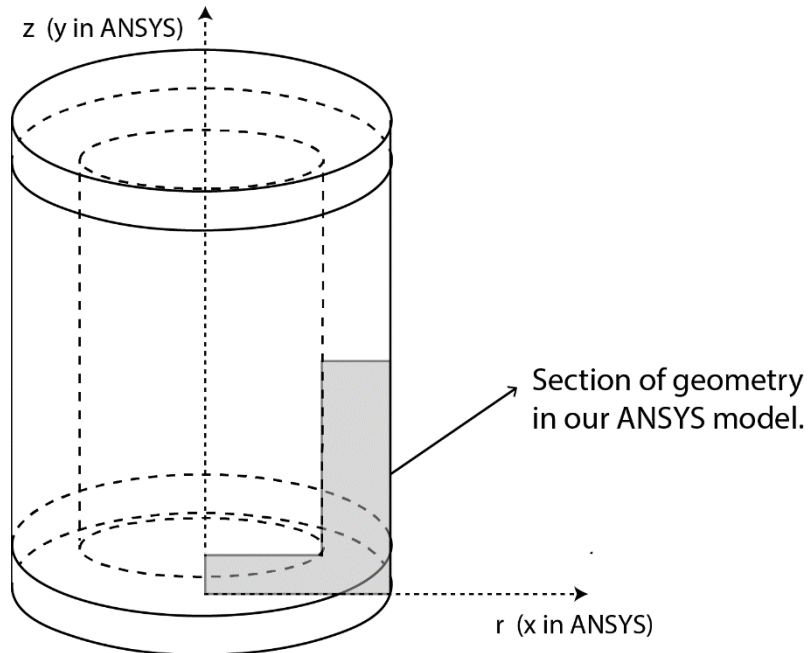
Launch ANSYS Workbench and start a "Static Structural" analysis in the project page as shown in the video below.

External video link: [https://youtu.be/ KJ97Gnwp8Y](https://youtu.be/KJ97Gnwp8Y)

3. Geometry

For axisymmetric models, we use cylindrical polar coordinates (r, θ, z) with no variation in the θ direction. So, we can just model a slice in the (r, z) plane as shown below.

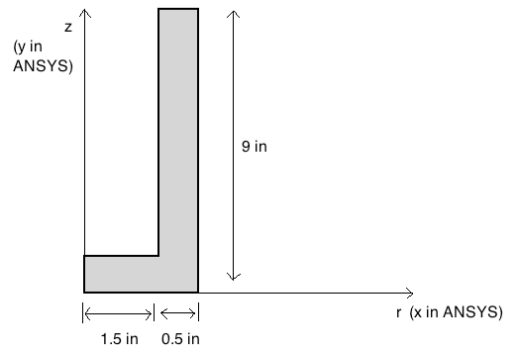
Tutorial: FEA of Thick-Walled Pressure Vessels



In ANSYS, the radial direction is x (rather than r) and the axial direction is y (rather than z). Confusing! We recommend that in an axisymmetric analysis, you think of the directions in ANSYS as radial & axial rather than “ x ” and “ y ”.

Note that for axisymmetric models in ANSYS, the y -axis is always the axis of symmetry. The corresponding 3D geometry can be generated by revolving the 2D section 360° about the y -axis (we'll do this later in the Numerical Results step).

Below is the 2D geometry we need to model:



We use symmetry to model only half the total length of the cylinder. We will first create a sketch and then a "surface body" from the sketch. The "surface body" is nothing but an area that we can

Tutorial: FEA of Thick-Walled Pressure Vessels

mesh and apply boundary conditions to. The video below shows the steps to be followed to create the sketch and surface body.

External video link: <https://youtu.be/nOn6uUC>IfM>

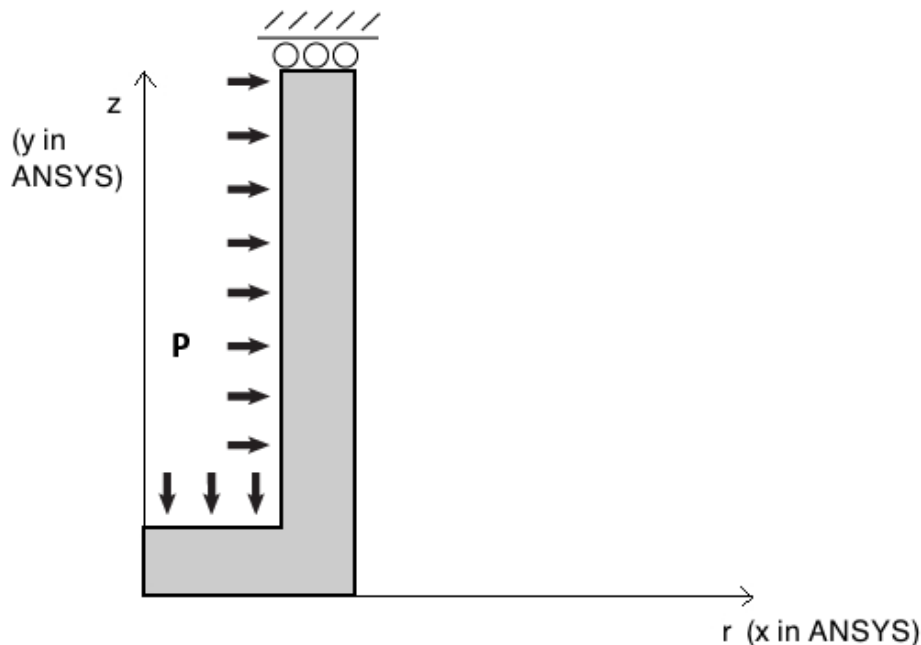
4. Mesh

The following video shows how to create a regular mesh for the rectangle using "Mapped face meshing".

External video link: <https://youtu.be/AYiZiekBxeY>

5. Physics Setup

Here are the boundary conditions we will impose on the model:



Symmetry boundary condition at the top edge implies zero displacement in the axial (y) direction. The top edge represents a plane of symmetry since it is located at the mid-length of the cylinder and we are modeling only half the length. In ANSYS, we impose this symmetry boundary condition using a "Frictionless support".

In an axisymmetric model, no displacement constraints are necessary in the radial direction to prevent rigid body motion in that direction. This is because radial displacement represents expansion/contraction of the structure which is resisted structurally.

Tutorial: FEA of Thick-Walled Pressure Vessels

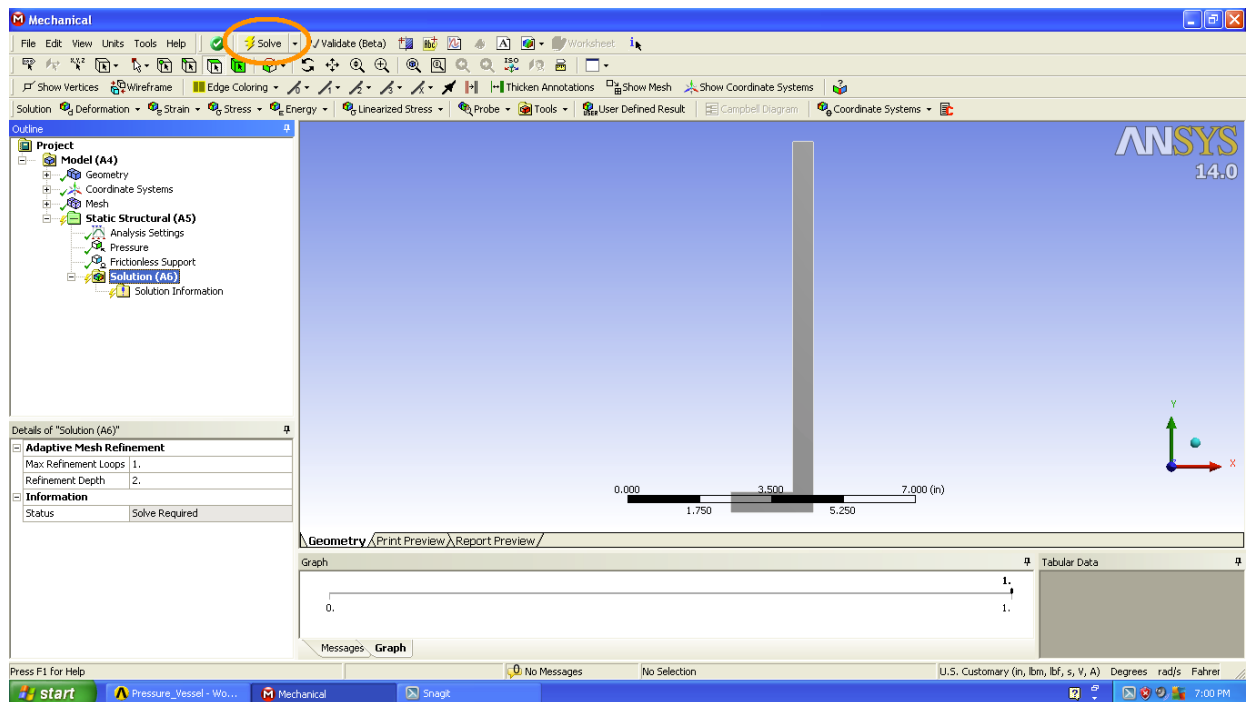
The following video shows how to specify the physics of the problem: axisymmetric analysis, material properties (Young's modulus and Poisson ratio) and boundary conditions. These settings get fed into the element formulation when obtaining the numerical solution later.

External video link: <https://youtu.be/OtoeNYOTphc>

Note: We perform an axisymmetric analysis by clicking on **Geometry**, expanding **Definition**, and selecting **Axisymmetric** for the 2D behavior.

6. Numerical Solution

To obtain the numerical solution, click solve. ANSYS formed the stiffness matrix for each element, assembled the global stiffness matrix and inverted it to get the nodal displacements. This is the bulk of the computation that ANSYS performs. All the results that we will look at next such as the deformed shape and the stresses are derived from these nodal displacements.



7. Numerical Results

The following video shows how to plot the deformed shape to check if our boundary conditions have been applied correctly.

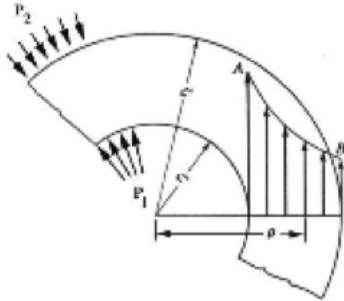
External video link: <https://youtu.be/7GLabKnRr2I>

The hoop, axial and radial stresses can be found in a similar fashion.

Tutorial: FEA of Thick-Walled Pressure Vessels

External video link: <https://youtu.be/jaoQMnjTos4>

A really neat way to represent hoop stress graphically is by using the “path” tool in ANSYS. The following video shows how to display stress as a function of radius much like the figure below.



External video link: <https://youtu.be/e6V7pTat0cM>

The “path” tool can also be used to show the axial and radial stresses along the top edge. Try it!

The next video shows how to display the full 3d model by revolving the 2D section 360° about the y-axis. Note that it comes from a different project, but the steps are the same.

External video link: <https://youtu.be/cvqjodZH7A>

8. Verification and Validation

"Verification and validation" can be thought of as a formal process for checking results. We previously performed some sanity checks on the deformed shape. A further basic check is how the results change on refining the mesh. The following video shows how to recalculate the results on a refined mesh. Note that it comes from a different project, but the procedure is the same.

External video link: https://youtu.be/oqKk62f_e9o

Mesh refinement results:

Tutorial: FEA of Thick-Walled Pressure Vessels

		FEA (Element size: 0.1in)	FEA (Element size: 0.05in)	FEA (Element size: 0.01in)
HOOP STRESS (psi)	Inner Surface	3566.9	3570	3570
	Outer Surface	2569.7	2570.9	2570.9
AXIAL STRESS (psi)	(Constant for both theories)	1285.8	1285.8	1285.8
RADIAL STRESS (psi)	Inner Surface	-995.4	-998.5	-998.5
	Outer Surface	1.7	0.5	0.5

Our results don't change significantly on refining the mesh. The medium and finest meshes yield virtually identical results. So we'll use the results from the medium mesh (element size of 0.05 in) to compare with the thin-walled and thick-walled hand calculations.

Comparison with thin and thick-walled theory from the pre-analysis:

Tutorial: FEA of Thick-Walled Pressure Vessels

		FEA Axisymmetric	Thin-Wall Equation	Thick-Wall Equation
HOOP STRESS (psi)	Inner Surface	3570	3000	3571.4
	Outer Surface	2570.9	3000	2571.4
AXIAL STRESS (psi)	(Constant for both theories)	1285.8	1500	1285.7

		FEA Axisymmetric	Boundary Conditions
RADIAL STRESS (psi)	Inner Surface	-998.5	-1000
	Outer Surface	0.5	0

Our FEA results compare very well with thick-walled theory where agreement is within 0.01-0.03% with theory. The thin-walled theory is beginning to lose validity. For this geometry, the FEA predictions differ from thin-walled theory by as much as 20%.