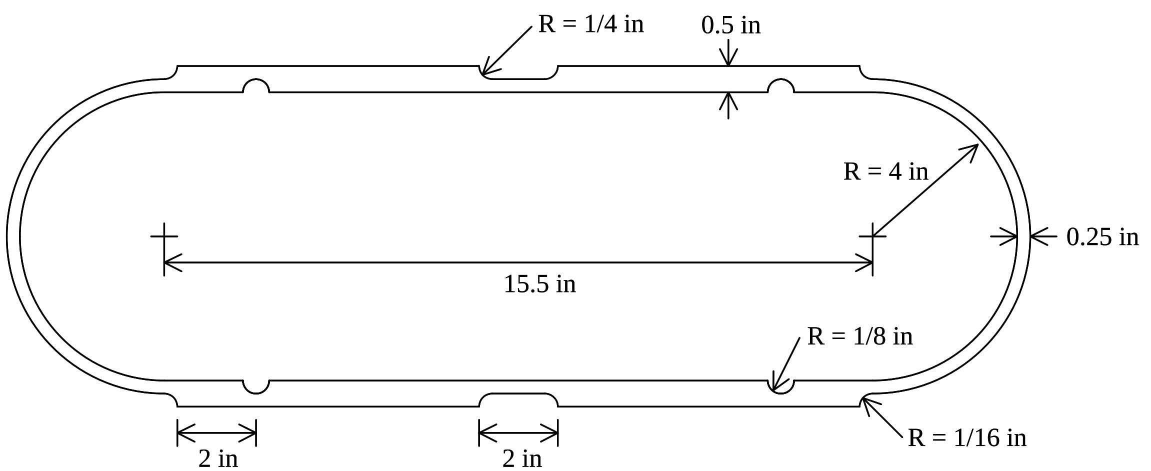
ANSYS Example: Axisymmetric Analysis of a Pressure Vessel

The pressure vessel shown below is made of cast iron (E = 14.5 Msi, ν = 0.21) and contains an internal pressure of p = 1700 psi. The cylindrical vessel has an inner diameter of 8 in with spherical end caps. The end caps have a wall thickness of 0.25 in, while the cylinder walls are 0.5 in thick. In addition, there are two small circumferential grooves of 1/8 in radius along the inner surface, and a 2 in wide by 0.25 in deep circumferential groove at the center of the cylinder along the outer surface.

In this example, ANSYS will be used to analyze the stresses and deflections in the vessel walls due to the internal pressure. Since the vessel is axially symmetric about its central axis, an axisymmetric analysis will be performed using two-dimensional, 8-node quadrilateral elements (Plane 82) with the axisymmetric option activated. In addition, the vessel is symmetric about a plane through the center of the cylinder. Thus, only a quarter section of the vessel needs to be modeled.

In ANSYS, an axisymmetric model must *always* be created such that the global Y-axis is the axis of symmetry, and the entire model should appear on the right side of the Y-axis (along the *positive* X-axis); i.e., no part of the model (elements, nodes, etc.) may be defined with negative X coordinates. Once the axisymmetric option is invoked, ANSYS will automatically apply axisymmetric boundary conditions along the Y-axis.



For model validation purposes, the stresses in the vessel walls away from any notches can be estimated using the thin-walled pressure vessel equations. Although the model does not specifically meet the criteria for the "thin-walled" assumption, these equations will still provide reasonably accurate values for model validation purposes. For a pressure vessel subjected to internal pressure only, the radial stress (σ_r) should vary from –p (–1.7 ksi) on the inner surface to zero on the outer surface. The hoop and longitudinal stresses are calculated as (p = 1700 psi, r = 4 in, t = 0.5 or 0.25 in):

$$\sigma_h \approx \frac{pr}{t} = 27.2\text{ ksi (thin section) or } 13.6\text{ ksi (thick section)}$$

$$\sigma_\ell \approx \frac{pr}{2t} = 13.6\text{ ksi (thin section) or } 6.8\text{ ksi (thick section)}$$

<u>ANSYS Analysis:</u>

Start ANSYS Product Launcher, set the Working Directory to C:\temp, define Job Name as 'Pressure Vessel', and click Run. Then define Title and Preferences.
Utility Menu → File → Change Jobname… → Enter 'Pressure_Vessel' → OK
Utility Menu → File → Change Title… → Enter 'Stress Analysis of an Axisymmetric Pressure Vessel' → OK
ANSYS Main Menu → Preferences → Preferences for GUI Filtering → Select 'Structural' and 'h-method' → OK

<u>Enter the Preprocessor to define the model geometry:</u>

Define Element Type (Axisymmetric Option) and Material Properties.
ANSYS Main Menu → Preprocessor → Element Type → Add/Edit/Delete → Add… → Structural Solid Quad 8 node 82 (PLANE82) (define 'Element type reference number' as 1) → OK → Click Options… → Select 'Axisymmetric' for K3 (Element behavior) → OK → Close
ANSYS Main Menu → Preprocessor → Material Props → Material Models → Double Click Structural → Linear → Elastic → Isotropic → Enter 14.5e6 for EX and 0.21 for PRXY → Click OK → Click Exit (under 'Material')

Begin creating the geometry by defining two Circles for the spherical endcap, and Subtract Areas to create the vessel wall.
ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Circle → Solid Circle → Enter 0 for WP X, 0 for WP Y, and 4 for Radius → Apply → Enter 0 for WP X, 0 for WP Y, and 4.25 for Radius → OK
ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Subtract → Areas → Select (with the mouse) Area 2 (*bigger circle*) → OK → Select Area 1 (*smaller circle*) → OK

Create Lines through the center of the Circles and Divide the Areas along these Lines.
ANSYS Main Menu → Preprocessor → Modeling → Create → Lines → Lines → Straight line → Click on the Keypoints on the outer circle which are on the X-axis to create a Line parallel to the X-axis (*Circles are divided into four arcs by Ansys, with a Keypoint placed at the end of each arc*). Similarly, click on the Keypoints on the outer circle which are on the Y-axis to create a Line parallel to the Y-axis → OK
ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Divide → Area by Line → Select (with the mouse) the remaining Area (annulus)→ OK → Select the two Lines that we have created → OK
ANSYS Main Menu → Preprocessor → Modeling → Delete → Area and Below → Select the three Areas in the first, second, and third quadrants → OK

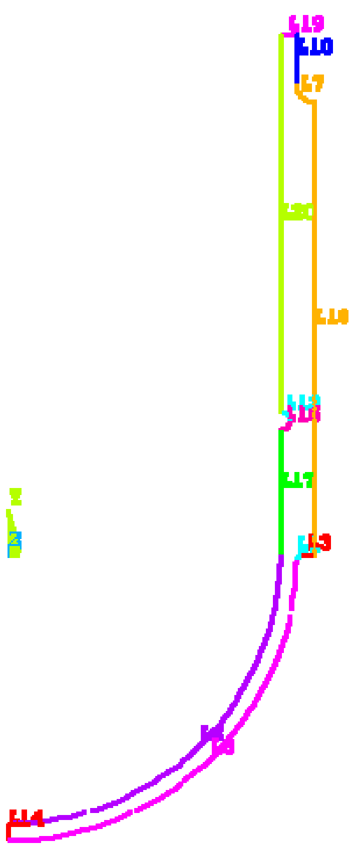
Define two Rectangles to create the walls of the cylindrical portion of the vessel (thick and thin sections). Define a Circle to create the circumferential groove on the inside of the vessel.
ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Rectangle → By Dimensions → Enter 4 and 4.5 for X-coordinates and 0 and 7.75 for Y-coordinates → Click Apply → Enter 4.25 and 4.5 for X-coordinates and 6.75 and 7.75 for Y-coordinates → OK
ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Circle → Solid Circle → Enter 4 for WP X, 2 for WP Y, and 1/8 for Radius → OK

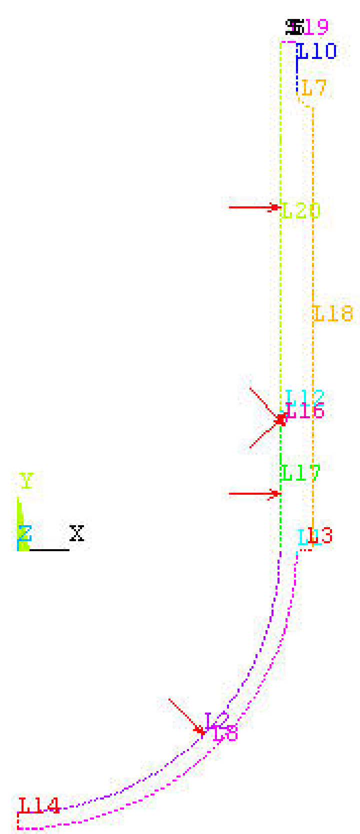
Subtract Areas to eliminate unused segments, and then Add all Areas to create a single Area for meshing.
ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Subtract → Areas → Select (with the mouse) the bigger rectangle → OK → Select the small rectangle and circle → OK
ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Add → Areas → Select 'Pick All' → OK

Create Line Fillets at the two transitions between the thick and thin sections.
Utility Menu → Plot → Lines
Utility Menu → Plot Ctrls → Numbering… → Click 'Line numbers' On → OK
ANSYS Main Menu → Preprocessor → Modeling → Create → Lines → Line Fillet → Select (with the mouse) the two Lines near the lower Fillet → OK → Enter 1/16 for Fillet radius → Apply → Select the two Lines near the upper Fillet → OK → Enter 1/4 for Fillet radius → OK

Create Areas within the two Fillets and add these Areas to the main Area. First zoom in on the area of interest using the plot controls.
ANSYS Main Menu → Preprocessor → Modeling → Create → Areas → Arbitrary → By Lines → Select (with the mouse) the Fillet and adjacent two Lines → OK
Repeat for the other Fillet.
ANSYS Main Menu → Preprocessor → Modeling → Operate → Booleans → Add → Areas → Select 'Pick All' → OK
Utility Menu → Plot → Lines

The geometry should appear as shown below in the figure on the left.





In this example, the irregular geometry will be Free Meshed with Quad Elements. Better control of Element sizing and distribution can be obtained with Mapped Meshing, but this would require that additional sub-Areas be defined within the main Area that have a regular (four-sided) geometry. Using Free Meshing, all Elements in the model will be approximately the same size. In the first run, we will choose a Global Size (approximate Element edge length) of 0.1 in.
ANSYS Main Menu → Preprocessor → Meshing → MeshTool → Under 'Size Controls: Global' click Set → Enter 0.1 for 'Element edge length' → OK → Under 'Mesh:' select Areas, Quad and Free → Click Mesh → Select (with the mouse) the Area → OK

<u>Enter the Solution Menu to define boundary conditions and loads and run the analysis:</u>

ANSYS Main Menu → Solution → Analysis Type → New Analysis → Select Static → OK

The Boundary Conditions and Loads can now be applied. ANSYS will automatically apply the Axisymmetric Boundary Conditions along the Y-axis. However, we must apply the Symmetry Boundary Conditions along the upper edge of the model. Finally, the Pressure can be applied on all lines that make up the inner surface of the vessel. The magnitude should be input as the actual value – no reduction is needed to account for axisymmetry (ANSYS automatically makes the necessary adjustment of Loads in an Axisymmetric model).
ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → Symmetry B.C. → On Lines → Select the Line on top of the model (19) → OK
ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Pressure → On Lines → Select (with the mouse) all the Lines on the inside of the vessel (20,12,16,17 and 2) → OK → Enter 1700 for 'Load PRES value' → OK
The pressure will be indicated by arrows, as shown above in the figure on the right.

Save the Database and initiate the Solution using the current Load Step (LS).
ANSYS Toolbar → SAVE_DB
ANSYS Main Menu → Solution → Solve → Current LS → OK → Close the information window when solution is done → Close the /STATUS Command window

<u>Enter the General Postprocessor to examine the results:</u>

First, plot the Deformed Shape.
ANSYS Main Menu → General Postproc → Plot Results → Deformed Shape → Select Def + undeformed → OK

A Contour Plot of any stress component can be created. The radial, hoop (tangential), and longitudinal stresses should be checked to verify the model. Also, stress values at any particular node can be checked by using the "Query Results" command, selecting the desired component, and then picking the appropriate node. For this model, along the cylindrical portion of the vessel, x represents the radial direction, y represents the longitudinal direction, and z represents the hoop (tangential) direction. Powergraphics must be disabled to query results at nodes.
ANSYS Toolbar → POWRGRPH → Select OFF → OK
ANSYS Main Menu → General Postproc → Plot Results → Contour Plot → Nodal Solu → Select 'Stress' and 'X-Component of stress' (or Y or Z) → OK
ANSYS Main Menu → General Postproc → Query Results → Nodal Solution → Select 'Stress' and 'X-direction SX' (or SY or SZ) → OK → Select Nodes in the region of interest (may be helpful to zoom in on region)

Compare the finite element stresses to the values calculated using the thin-wall equations. If the values are within reason (away from notches, etc.), proceed. For the purposes of failure analysis, we must select an appropriate failure theory. A plot of the von Mises stress is useful for identifying critical locations in the vessel. However, since the vessel is made of cast iron (brittle material), the "Maximum-Normal-Stress" failure criterion may be more appropriate (or Coulomb-Mohr or other similar failure theories). Create Contour Plots of the von Mises and 1st Principal stresses.

ANSYS Main Menu → General Postproc → Plot Results → Contour Plot → Nodal Solu → Select 'Stress' and 'von Mises stress' → OK

ANSYS Main Menu → General Postproc → Plot Results → Contour Plot → Nodal Solu → Select 'Stress' and '1st Principal stress' → OK

The plot of the model can be expanded around the axisymmetric axis to get a better view of the full model. For this plot, Powergraphics must be enabled.

ANSYS Toolbar → POWRGRPH → Select ON → OK

Utility Menu → PlotCtrls → Style → Symmetry Expansion → 2-D Axi-Symmetric… → Select 'Full expansion' → OK

Note the locations of the maximum stresses in the vessel. Are the critical locations where you would expect them to be? If not, why? Do you think the current model is accurate, or might there be some discretization error? Record the magnitudes and locations of the maximum stresses, and then refine the mesh and re-run the analysis to check for possible discretization error.