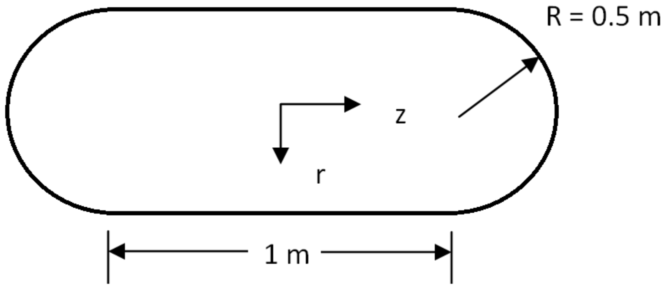
Tutorial 5. Axisymmetric Analysis

Consider a steel (E=200 GPa, ν=0.3) cylindrical pressure vessel with hemispherical end caps as shown below. The pressure vessel has an inner radius of R = 0.5 m and a wall thickness of t = 0.05 m. An internal pressure of 100 MPa is applied.



Theoretical Solution

For pressure vessels with R/t>20, thin walled theory gives side wall stresses:

$$\sigma_{rr} \approx 0$$
$$\sigma_{\theta\theta} \approx \frac{pR}{t} = 1000\, MPa$$
$$\sigma_{zz} \approx \frac{pR}{2t} = 500\, MPa$$
$$\sigma_{VM} \approx 866\, MPa$$

and end cap stresses

$$\sigma_{rr} \approx 0$$
$$\sigma_{\theta\theta} \approx \sigma_{\phi\phi} \approx \frac{pR}{t} = 500\, MPa$$
$$\sigma_{VM} \approx 500\, MPa$$

Finite Element solution

Start => All Programs => Dassault Systems SIMULIA Abaqus => Abaqus CAE => Create Model Database With Standard/Explicit Model
File => Set Working Directory => Browse to find desired directory => OK
File => Save As => save axisymmetric_tutorial.cae file in Work Directory

Module: Sketch
(Note: reorient geometry such that positive z-axis is vertical upward and positive r-axis is horizontal to the right)
Sketch => Create => Approx size - 5
Add=> Point => enter coordinates (.5,0), (.55,0), (.5,.5), (.55,.5), (0,1.0), (0,1.05) => select 'red X'
View => Auto-Fit
Add => Line => Connected Line => select point at (.5,5) with mouse, then (.5,0), (.55,0), (.55,.5) => right click => Cancel Procedure => Done
Add => Line => Connected Line => select point at (0,1.0) with mouse, then (0,1.05) => right click => Cancel Procedure => Done
Add => Arc => Center/Endpoint => select point at (0,.5), then (.5,.5), then (0,1.0) => Cancel Procedure => Done
Add => Arc => Center/Endpoint => select point at (0,.5), then (.55,0), then (0,1.05) => Cancel Procedure => Done

Module: Part
Part => Create => select Axisymmetric, Deformable, Shell, Approx size - 5=> Continue
Add => Sketch => select 'Sketch-1' => Done => Done

Module: Property
Material => Create => Name: Material-1, Mechanical, Elasticity, Elastic => set Young's modulus = 200e9, Poisson's ratio = 0.3 => OK
Section => Create => Name: Section-1, Solid, Homogeneous => Continue => Material - Material-1, plane stress/strain thickness – leave unselected => OK
Assign Section => select entire part by dragging mouse => Done => Section-1 => OK

Module: Assembly
Instance => Create => Create instances from: Parts => Part-1 => Dependent (mesh on part) => OK

Module: Step
Step => Create => Name: Step-1, Initial, Static, General => Continue => accept default settings => OK

Module: Load
Load => Create => Name: Load-1, Step: Step 1, Mechanical, Pressure => Continue => select interior edges (use shift key to select multiple edges) => Done => set Magnitude = 100e6 => OK

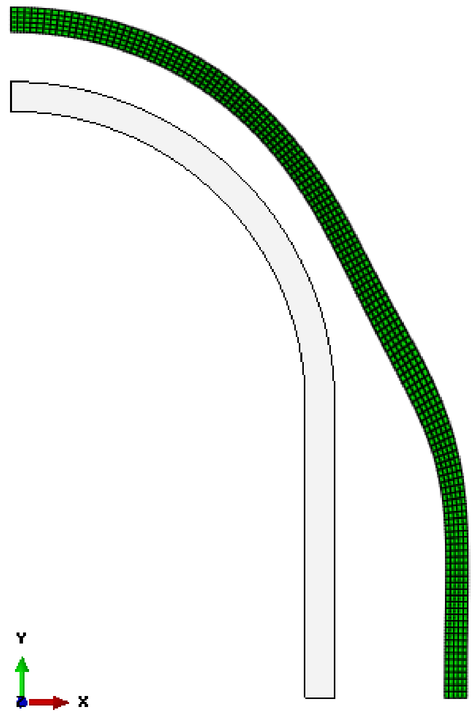
BC => Create => Name: BC-1, Step: Step-1, Mechanical, Symmetry/Antisymmetry/Encastre => Continue => select bottom edge (z=0) => Done => YSYM (U2=UR1=UR2=0)
BC => Create => Name: BC-2, Step: Step-1, Mechanical, Symmetry/Antisymmetry/Encastre => Continue => select left edge (r=0) => Done => XSYM (U1=UR2=UR3=0)

<u>Module: Mesh</u>
Set Model: Model-1, Object => Part: Part-1
To create partition separating side wall from end cap: Tools => Partition => Type: Face => Sketch => Add => Line => Connected Line => use mouse to draw line from top left to top right of side wall => right click => Cancel procedure
Seed => Edge by Size => select full model by dragging mouse => Done => Element Size=0.02 => press Enter => Done
Mesh => Controls => select full model => Element Shape => Quad => Structured => OK
Mesh => Element Type => Axisymmetric Stress => Quadratic/Quad (for 8-node quad) => OK => Done
Mesh => Instance => OK to mesh the part Instance: Yes => Done
Tools => Query => Mesh => Done (*displays number of nodes and elements at bottom of screen*)

<u>Module: Job</u>
Job => Create => Name: Job-1, Model: Model-1 => Continue => Job Type: Full analysis, Run Mode: Background, Submit Time: Immediately => OK
Job => Manager => Submit => Job-1
Job => Manager => Results (transfers to Visualization Module)

<u>Module: Visualization</u>
Viewport => Viewport Annotation Options => Legend => Text => Set Font => Size=14, Apply to: Legend, Title Block and State Block => OK => OK
View => Graphics Options => Viewport Background = Solid=> Color => White (click on black tile to change background color)
Plot => Select Undeformed Shape, Deformed Shape and Allow Multiple Plot States
Options => Common => Deformed Scale Factor => Uniform => Value: 100
Ctrl-C to copy viewport to clipboard => Open MS Word Document => Ctrl-V to paste image
Plot=> Contours => Result => Option => Set Nodal Averaging Threshold to 0% => Apply
Result => Field Output => Name , Invariant - Mises => OK
Ctrl-C to copy viewport to clipboard => Open MS Word Document => Ctrl-V to paste image
Tools => Query => Probe Values => Apply => select desired Field Output (S, Mises) => Probe Nodes => move cursor to desired location to view nodal results
Along bottom symmetry plane (z=0, .5<=r<=.55), Probe von Mises stress to show variation from 827 to 1,007 MPa (as compared to theoretical approximation of 1,000 MPa)
Along axis of symmetry edge (1<=z<=1.05, r=0), Probe von Mises stress to to show variation from 455 to 603 MPa (as compared to theoretical approximation of 500 MPa)

Undeformed and Deformed Shape:



Von Mises Stress contours