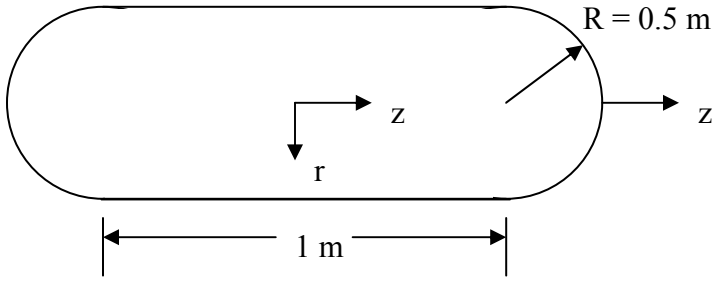
ABAQUS Tutorial – Axisymmetric Analysis

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:

$$\sigma_{rr} \approx 0$$

$$\sigma_{\theta\theta} \approx \frac{pR}{t} = 1000\ MPa$$

$$\sigma_{zz} \approx \frac{pR}{2t} = 500\ MPa$$

$$\sigma_{VM} \approx \frac{\sqrt{5}pR}{2t} = 1{,}118\ MPa$$

and end cap stresses

$$\sigma_{rr} \approx 0$$

$$\sigma_{VM} \approx \frac{pR}{\sqrt{2}t} = 707\ MPa$$

Finite Element solution (ABAQUS)

Start => Programs => ABAQUS 6.7-1 => ABAQUS CAE
Select 'Create Model Database'
File => Save As => create directory for files

Module: Sketch
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 - 1 => OK
Assign Section => select entire part by dragging mouse => Done => Section-1 => OK

Module: Assembly
Instance => Create => Part-1 => OK

Module: Step

Step => Create => Name: Step-1, Initial, Static, General => Continue => nlgeom off => OK

Module: Load
Load => Create => Name: Step-1, Step: Step 1, Mechanical, Pressure => Continue => select interior edges => Done => set Magnitude = 100e6 => OK
BC => Create => Name: BC-1, Step: Step-1, Mechanical, Symmetry/Antisymmetry/Encastre => Continue => select bottom edge => Done => YSYM (U2=UR1=UR2=0)
BC => Create => Name: BC-2, Step: Step-1, Mechanical, Symmetry/Antisymmetry/Encastre => Continue => select left edge => Done => XSYM (U1=UR2=UR3=0)

Module: Mesh

Seed => Edge by Size => select full model by dragging mouse => Done => Element Size=0.1 => press Enter => Done

Mesh => Controls => Element Shape => Quad

Mesh => Element Type => Axisymmetric => Quadratic/Quad (for 8-node quad) => OK => Done

Mesh => Instance => OK to mesh the part Instance: Yes => Done

Tools => Query => Region Mesh => Apply (*displays number of nodes and elements at bottom of screen*)

Module: Job

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

Results

Module: Visualization

Plot => Deformed Shape

Deformed Shape Options => Basic => Show superimposed undeformed plot => OK

View => Graphics Options => Background Color => White

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 (S11, S22, etc.) => Probe Nodes => move cursor to desired location to view nodal results

Report => Field Output => Setup => Number of Significant Digits => 6

Report => Field Output => Position - Centroid => Variable - S => Apply

Examine tabulated results in 'abaqus.rpt' file.