

Abaqus Tutorials

Prepared by:

David G. Taggart
Department of Mechanical, Industrial and Systems Engineering
University of Rhode Island
Kingston, RI 02881

Prepared for:

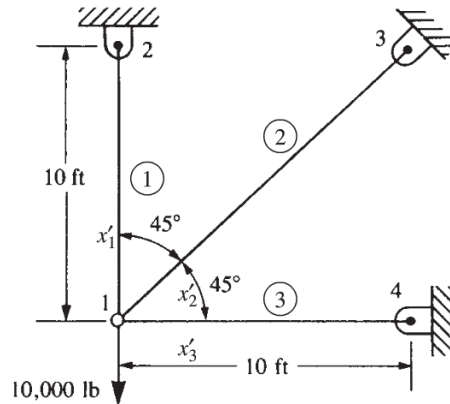
MCE 466 - Introduction to Finite Element Methods
Spring 2015

TABLE OF CONTENTS

1. 2D Truss Analysis.....	2
2. Beam Bending Analysis	5
3. Plane Stress Analysis.....	10
4. Axisymmetric Analysis	16
5. 3D Stress Analysis.....	20
6. Plate Bending Analysis.....	24
7. Column Buckling Analysis.....	26

Tutorial 1. 2D Truss Analysis

Problem: Determine the nodal displacements and element stresses for the truss shown below (ref. "A First Course in the Finite Element Method, 5th edition, Daryl L. Logan, 2012, example 3.5, pp. 92-95). Use $E=30 \times 10^6$ psi and $A=2$ in². Compare to text solution: $u_1 = 0.414 \times 10^{-2}$ in, $v_1 = -1.59 \times 10^{-2}$ in, element stresses = -1035, 1471, and 3965 psi.



Start => All Programs => Abaqus 6.X => Abaqus CAE => Create Model Database With Standard/Explicit Model

File => Set Working Directory => Browse to find desired directory => OK

File => Save As => save truss_tutorial.cae file in Work Directory

Module: Sketch

Sketch => Create => Name: truss-demo => Continue

Add=> Point => enter coordinates (0,0), (120,0), (120,120), (0,120) => select 'red X'

View => Auto-Fit

Add => Line => Connected Line => select (0,0) node with mouse, then (120,0) node, right click => Cancel Procedure

Add => Line => Connected Line => select (0,0) node with mouse, then (120,120) node, right click => Cancel Procedure

Add => Line => Connected Line => select (0,0) node with mouse, then (0,120) node, right click => Cancel Procedure=> Done

Module: Part

Part => Create => select 2D Planar, Deformable, Wire => Continue

Add => Sketch => select 'truss_demo' => Done => Done

Module: Property

Material => Create => Name: Material-1, Mechanical, Elasticity, Elastic => set Young's modulus = 30e6, Poisson's ratio = 0.3 => OK

Section => Create => Name: Section-1, Beam, Truss => Continue => set Material: Material-1, Cross-sectional area: 2

Assign Section => select all elements by dragging mouse => Done => Section-1 => OK => Done

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, Concentrated Force => Continue
=> select node at (0,0) => Done => set CF2: -10000 => OK

BC => Create => Name: BC-1, Step: Step-1, Mechanical, Displacement/Rotation => Continue
=> select nodes at (120,0), (120,120) and (0,120) using SHIFT key to select multiple nodes => Done => set U1: 0 and U2: 0

Module: Mesh

Set Model: Model-1, Object => Part: Part-1

Seed => Edges => select entire truss by dragging mouse => Done => Method: By number, Bias: None, Sizing Controls, Number of Elements: 1 => press Enter => Done

Mesh => Element Type => select entire truss by dragging mouse => Done => Element Library: Standard, Geometric Order: Linear: Family: Truss => OK => Done

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

Module: Job

Job => Create => Name: Job-1, Model: Model-1 => Continue => Job Type: Full analysis, Run Mode: Background, Submit Time: Immediately => OK

Job => Submit => Job-1

Job => Manager => Results (transfers to Visualization Module)

Module: Visualization

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)

Options => Common => Labels => select 'Show element labels: Black' and 'Show node labels: Red' => OK

Plot => Undeformed Shape

Plot => Deformed Shape

Plot => Contours => On Deformed Shape

Result => Options => unselect "Average element output at nodes"

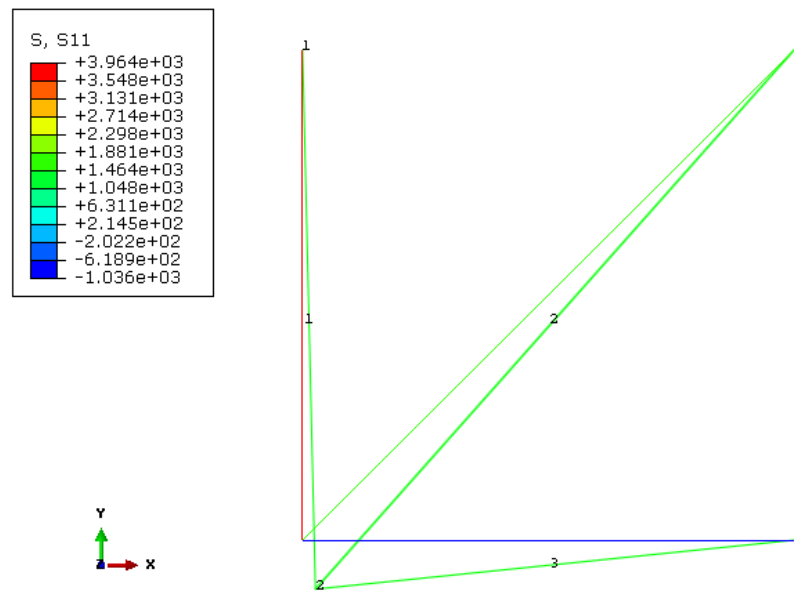
Result => Field Output => Component: S11 => OK

Ctrl-C => Copies graphics window to clipboard => Paste in MS Word, etc.

Report => Field Output => Variable => Position: Unique Nodal => select U: Spatial Displacements => Apply => Unselect U
 Report => Field Output => Variable => Position: Centroid => select S: Stress Components => Click on '>' and unselect all stresses except S11 => Apply => Cancel
 Open file 'Abaqus.rpt' and cut and paste desired results into MS Word
 File => Save => enter desired file name (Abaqus will append .cae)
 File => Exit

Results:

Deformed Mesh:



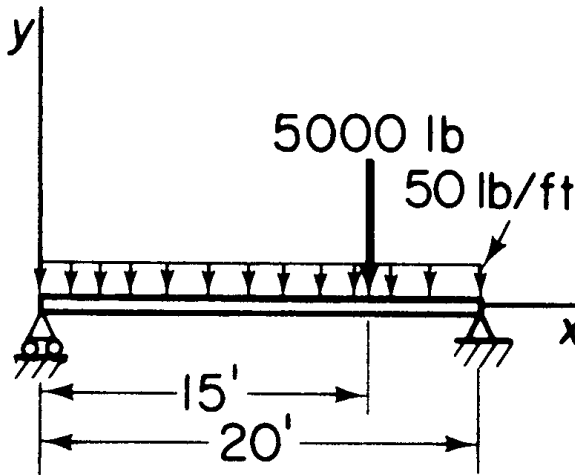
Tabulated Results (using cut and paste from Abaqus.rpt)

Node Label	U.Magnitude @Loc 1	U.U1 @Loc 1	U.U2 @Loc 1
1	0.	0.	7.92893E-33
2	16.3899E-03	4.14214E-03	-15.8579E-03
3	0.	-2.07107E-33	-2.07107E-33
4	0.	2.07107E-33	0.

Element Label	S.S11 @Loc 1
1	3.96447E+03
2	1.46447E+03
3	-1.03553E+03

Tutorial 2. Beam Bending Analysis

Consider the beam bending problem:



Assume that the beam is made of steel ($E=30 \times 10^6$ psi, $G=11.5 \times 10^6$ psi) and has a 2" deep x 5" high rectangular cross section ($I_z=(2)(5^3)/12=20.83$ in⁴, $I_y=(5)(2^3)/12=3.333$ in⁴). Determine the maximum deflection and stress in the bar and the using 8 beam elements. Compare the solution to the beam theory solution.

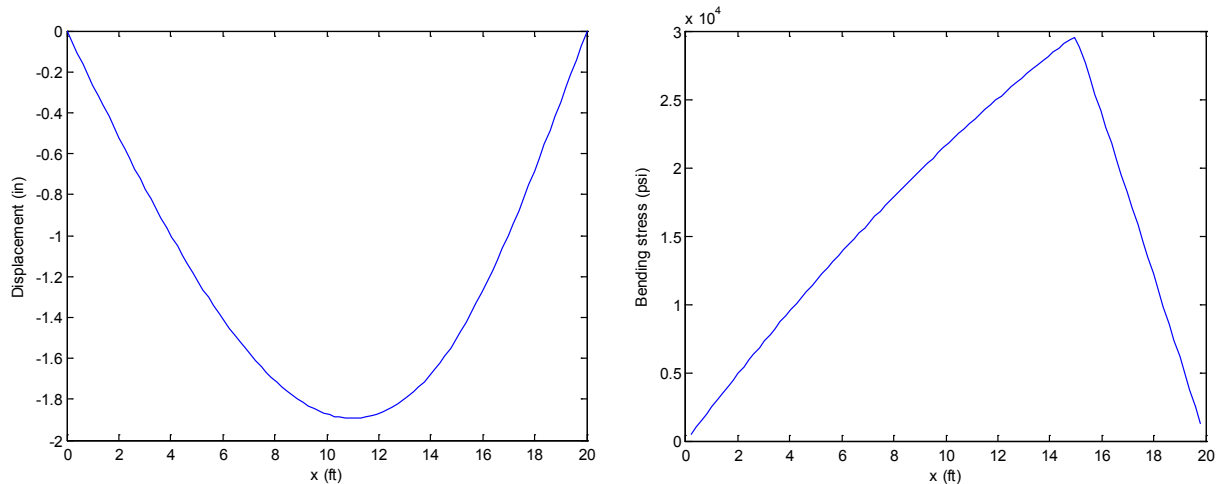
Beam theory solution

Beam theory gives the following displacement solution:

$$v(x) = \frac{Pbx}{6EI} (x^2 + b^2 - L^2) + \frac{wx}{24EI} (2Lx^2 - x^3 - L^3), \quad 0 \leq x \leq a$$

$$v(x) = \frac{Pa(L-x)}{6EI} (x^2 + a^2 - 2Lx) + \frac{wx}{24EI} (2Lx^2 - x^3 - L^3), \quad a \leq x \leq L$$

where $v(x)$ is the displacement, P is the concentrated force (-5000 lb), x is the distance from the left end of the beam, EI is the flexural stiffness of the beam, w_0 is the uniform distributed load (-50 lb/ft = -4.167 lb/in), $a=15$ ft and $b=5$ ft. The displacement field and bending stress distribution predicted by beam theory are shown below. Note that the maximum deflection, approximately -1.89 in, occurs between $x=11$ ft and $x=12$ ft and the maximum bending stress is approximately 29,700 psi at $x=15$ ft.



Finite Element solution

Start => All Programs => Abaqus 6.X => Abaqus CAE => Create Model Database With Standard/Explicit Model

File => Set Working Directory => Browse to find desired directory => OK

File => Save As => save beam_tutorial.cae file in Work Directory

Module: Sketch

Sketch => Create => continue

Add=> Point => => select 'red X'

Add => Line => Connected Line => enter coordinates (0,0), (180,0), (240,0), right click => Cancel Procedure => Done

Module: Part

Part => Create => select 2D Planar, Deformable, Wire, Approx size 200 => Continue

Add => Sketch => select 'Sketch-1' => Done => Done

Module: Property

Material => Create => Name: Material-1, Mechanical, Elasticity, Elastic => set Young's modulus = 30e6, Poisson's ratio = 0.3 => OK

Profile => Create => Generalized => A=10, $I_1 = 20.83$, $I_{12}=0$, $I_2=3.333$, J=0 => OK

Section => Create => Name: Section-1, Beam, Beam => Continue => Section Integration - Before Analysis => Profile Name: Profile-1 => Linear Properties => E=30e6, G=11.54e6 => Output Points => enter $(x_1, x_2) = (0, -2.5)$ and $(x_1, x_2) = (0, 2.5)$ => OK

Assign Section => select all elements by dragging mouse => Done => Section-1 => Done => OK

Assign Beam Section Orientation => select full model => Done => n_1 direction = 0.0,0.0,-1.0 => OK => Done

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, Line Load => Continue => select full model => Done => set Component 1 =0, Component 2 = -4.167 => OK

Load => Create => Name: Step-1, Step: Step 1, Mechanical, Concentrated Force => Continue => select point at (180,0) => Done => set CF2=-5000 => OK

BC => Create => Name: BC-1, Step: Step-1, Mechanical, Displacement/Rotation => Continue
=> select point at (0,0) => Done => U2=0 => OK

BC => Create => Name: BC-1, Step: Step-1, Mechanical, Displacement/Rotation => Continue
=> select point at (240,0) => Done => U1=U2=0 => OK

Module: Mesh

Set Model: Model-1, Object => Part: Part-1

Seed => Edges => select entire beam by dragging mouse => Done => Method: By size, Bias: None, Sizing Controls, Element Size=30 => OK => Done

Mesh => Element Type => select entire truss by dragging mouse => Done => Element Library: Standard, Geometric Order: Linear: Family: Beam, Cubic interpolation (B23)=> OK => Done

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

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

Job => Manager => Results (transfers to Visualization Module)

Module: Visualization

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)

Options => Common => Labels => select 'Show element labels: Black' and 'Show node labels: Red' => OK

Plot => Deformed Shape

Ctrl-C to copy viewport to clipboard => Open MS Word Document => Ctrl-V to paste image

Result => Field Output => select S, Component: S11 => Section Points => Top and Bottom => OK

Plot=> Contours => On Deformed Shape

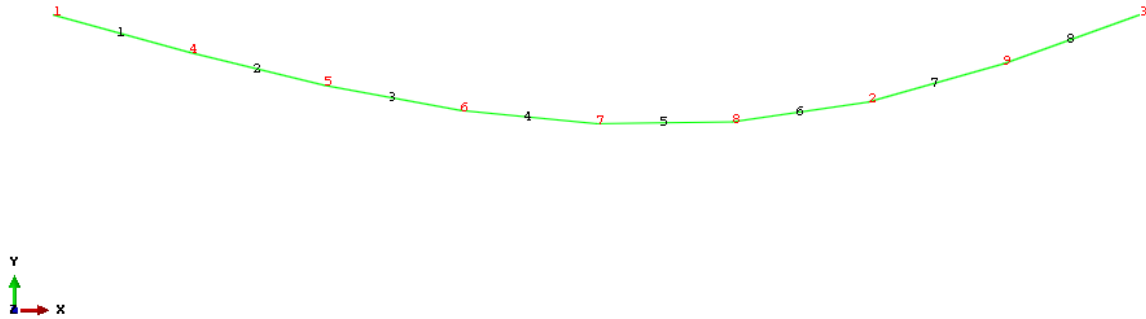
Report => Field Output => Variable => Position: Unique Nodal => select Spatial displacement: U2: Spatial Displacements, Rotational displacement: UR3 => OK

Report => Field Output => Variable => Position: Unique Nodal => select Stress components: S11, Section points - All => OK

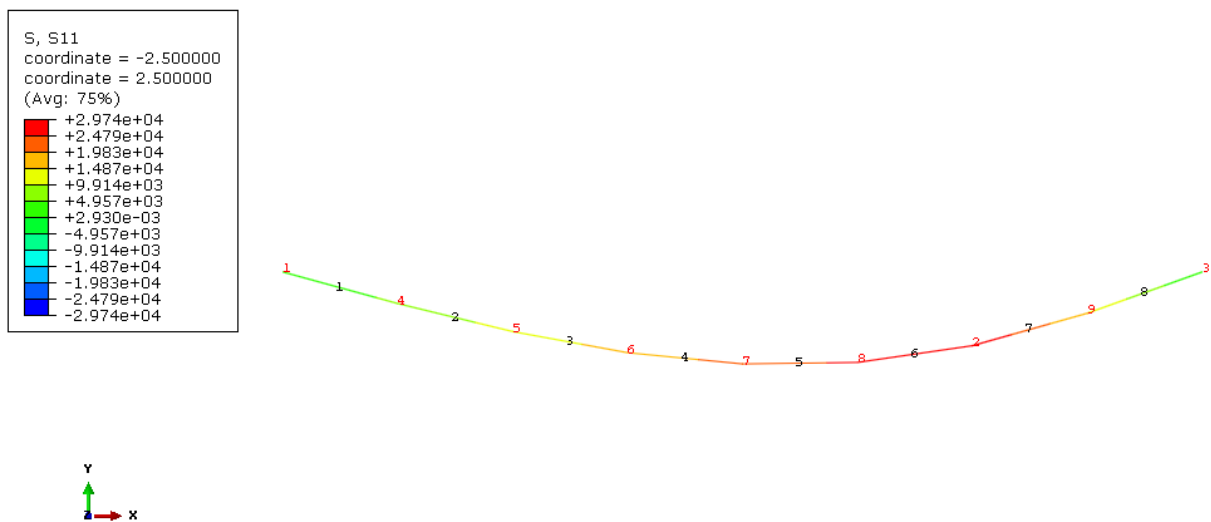
Cut and paste tabulated results from 'Abaqus.rpt' file to MS Word document.

Results:

Deformed Mesh



Bending Stress Contours



Tabulated Output:

Node Label	S.S11 @Loc 1
1	37.5090
2	29.7425E+03
3	37.5085
4	6.11361E+03
5	11.7396E+03
6	16.9155E+03
7	21.6413E+03
8	25.9169E+03
9	15.1150E+03

Minimum At Node 37.5085
3

Maximum At Node 29.7425E+03
2

Total 127.259E+03

Node Label	S.S11 @Loc 2
1	-37.5090
2	-29.7425E+03
3	-37.5085
4	-6.11361E+03
5	-11.7396E+03
6	-16.9155E+03
7	-21.6413E+03
8	-25.9169E+03
9	-15.1150E+03

Minimum At Node -29.7425E+03
2

Maximum At Node -37.5085
3

Total -127.259E+03

Node Label	U.U2 @Loc 1
1	-1.68753E-33
2	-1.50146
3	-4.18754E-33
4	-642.937E-03
5	-1.21341
6	-1.64391
7	-1.87232
8	-1.84194
9	-840.968E-03

Minimum At Node -1.87232
7

Maximum At Node -1.68753E-33
1

Total -9.55695

Node Label	UR3 @Loc 1
1	-21.8438E-03
2	17.0429E-03
3	29.0450E-03
4	-20.6136E-03
5	-17.0429E-03
6	-11.3119E-03
7	-3.60058E-03
8	5.91106E-03
9	26.0144E-03

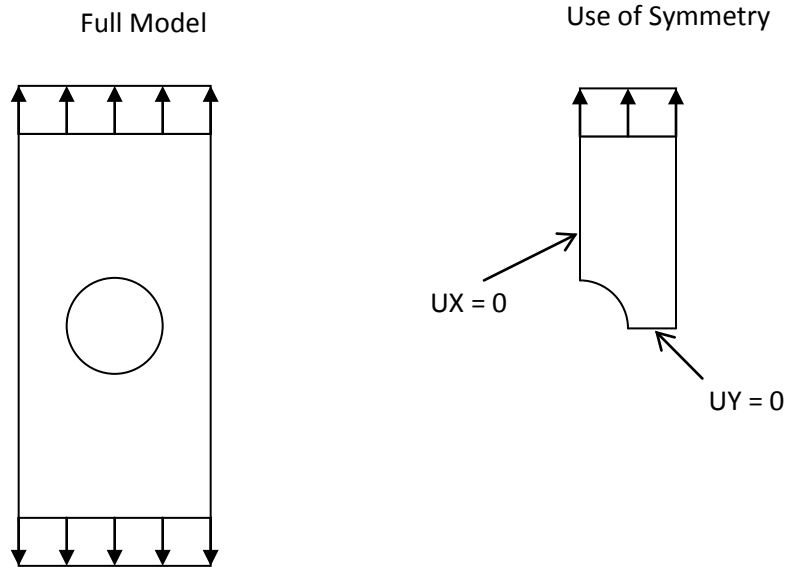
Minimum At Node -21.8438E-03
1

Maximum At Node 29.0450E-03
3

Total 3.60058E-03

Tutorial 3. Plane Stress Analysis

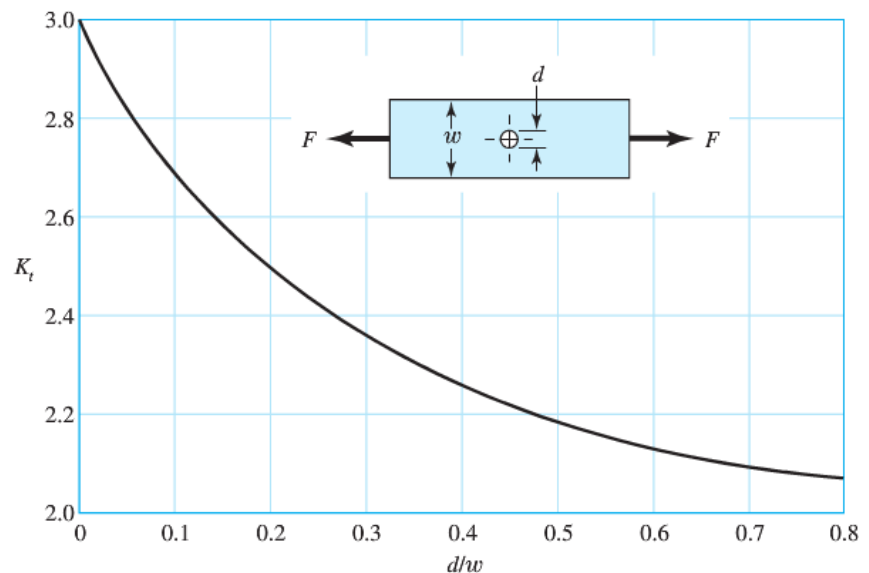
Consider the problem of a 4" x 2" x 0.1" aluminum plate ($E=10e6$ psi, $\nu=0.3$) with a 1" diameter circular hole subjected to an axial stress of 100 psi.



Determine the maximum axial stress associated with the stress concentration at the edge of the circular hole. Compare this solution with the design chart (ref. "Shigley's Mechanical Engineering Design," 10th Edition, Budynas and Nisbett, 2015) for the case $d/w=0.5$ which gives $\sigma_{\max} \cong 2.18 (200 \text{ psi}) = 436 \text{ psi}$.

Figure A-15-1

Bar in tension or simple compression with a transverse hole. $\sigma_0 = F/A$, where $A = (w - d)t$ and t is the thickness.



Finite Element solution

Start => All Programs => Abaqus 6.X => Abaqus CAE => Create Model Database With Standard/Explicit Model

File => Set Working Directory => Browse to find desired directory => OK

File => Save As => save plane_stress_tutorial.cae file in Work Directory

Module: Sketch

Sketch => Create => Approx size - 5

Add=> Point => enter coordinates (.5,0), (1,0), (1,2), (0,2), (0,.5) => select 'red X'

View => Auto-Fit

Add => Line => Connected Lines => select point at (.5,0) with mouse, then (1,0), (1,2), (0,2), (0,.5) => right click => Cancel Procedure => Done

Add => Arc => Center/Endpoint => select point at (0,0), then (.5,0), then (0,.5) => right click => Cancel Procedure => Done

Module: Part

Part => Create => select 2D Planar, 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 = 10e6, Poisson's ratio = 0.3 => OK

Section => Create => Name: Section-1, Solid, Homogeneous => Continue => Material - Material-1, plane stress/strain thickness - 0.1 => OK

Assign Section => select entire part by dragging mouse => Done => Section-1, Thickness: From section => 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 top edge => Done => set Magnitude = -100 => OK

BC => Create => Name: BC-1, Step: Step-1, Mechanical, Displacement/Rotation => Continue => select bottom edge => Done => U2=0

BC => Create => Name: BC-2, Step: Step-1, Mechanical, Displacement/Rotation => Continue => select left edge => Done => U1=0

Module: Mesh

Set Model: Model-1, Object => Part: Part-1

Seed => Edges => select full model by dragging mouse => Done => Method: By size, Bias: None, Sizing Controls, Element Size=0.1 => OK => Done

Mesh => Controls => Element Shape => Tri (for triangles), Quad (for quadrilaterals), or Quad dominated (for mixed triangles and quads - mostly quads), Technique: Free > OK

Mesh => Element Type => select full model by dragging mouse => Done => Element Library: Standard, Geometric Order: Linear, Family: Plane Stress => Linear/Tri (for CST), Quadratic/Tri (for LST), Linear/Quad (for 4 node quad), or Quadratic/Quad (for 8-node quad) => OK => Done (try varying element type, interpolation functions and mesh density)

To refine mesh locally:

Seed => Edges => Select bottom edge and arc (use Shift Key to select multiple edges) => Done => Method: By size, Bias: Single, Minimum size: .02, Maximum Size: 1, Flip Bias: Select and click on edges such that arrow points toward desired refined point => OK => Done

Mesh => Part => OK to mesh the Part: 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

Job => Manager => Results (transfers to Visualization Module)

Module: Visualization

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 => Deformed Shape

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

Ctrl-C to copy viewport to clipboard => Open MS Word Document => Ctrl-V to paste image

Result => Options => Unselect "Average element output at nodes" => OK

Result => Field Output => Name - S => Component = S22 => OK

Plot => Contours => On Deformed Shape

Ctrl-C to copy viewport to clipboard => Open MS Word Document => Ctrl-V to paste image

Tools => Query => Probe Values => select desired Field Output (click on Field output variable icon) and select desired component (S11, S22, etc.) => OK => Probe Nodes => move cursor to desired location to view nodal results

Tools => Path => Create => Node List => Continue => Add Before => select nodes along bottom edge => Done => OK

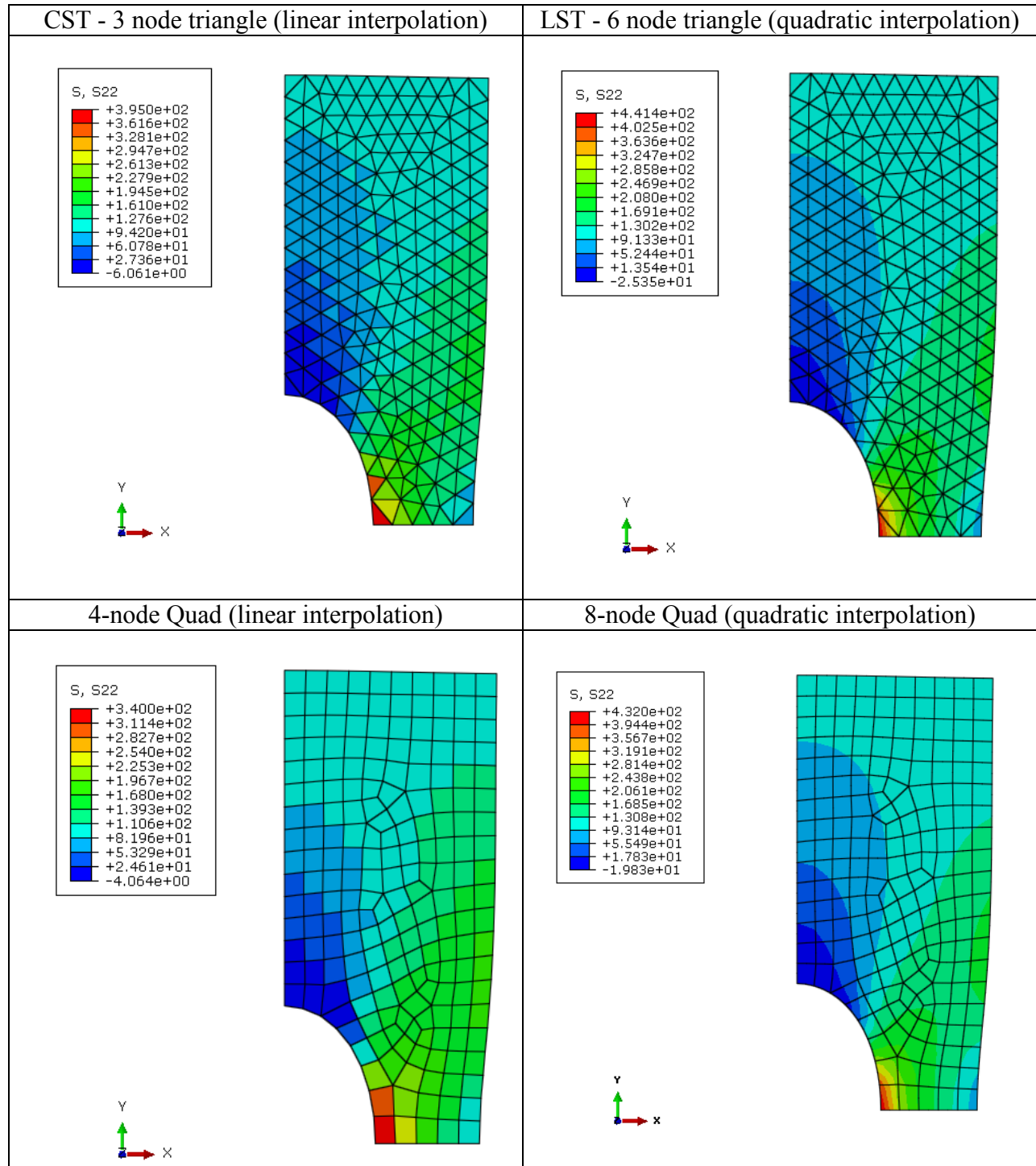
Tools => XY Data => Create => Source: Path => Continue => X Distance => Plot

Ctrl-C to copy viewport to clipboard => Open MS Word Document => Ctrl-V to paste image

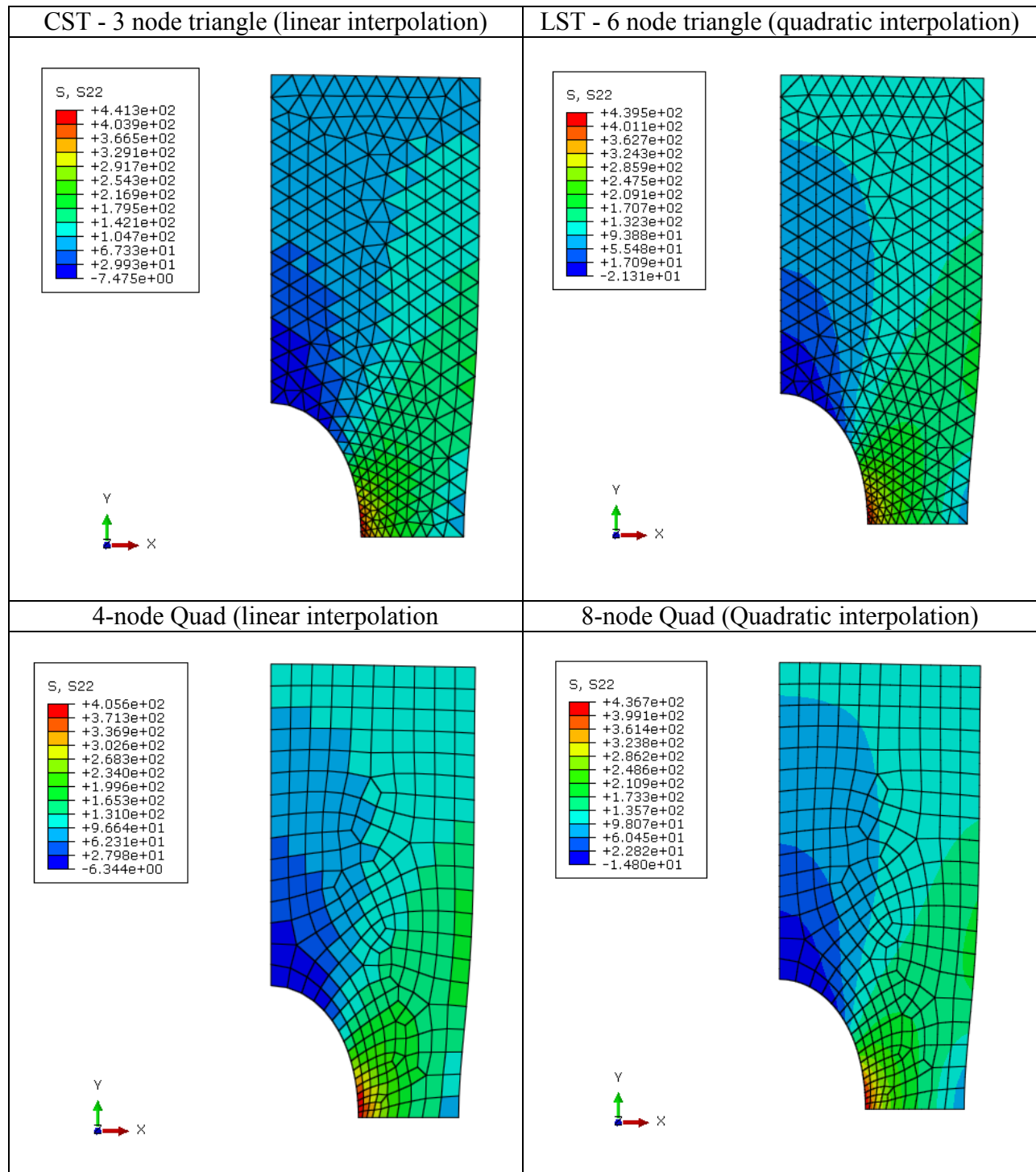
Report => Field Output => Position - Centroid => Variable - Mises, S11, S22, S12 => Apply

Examine tabulated results in 'Abaqus.rpt' file.

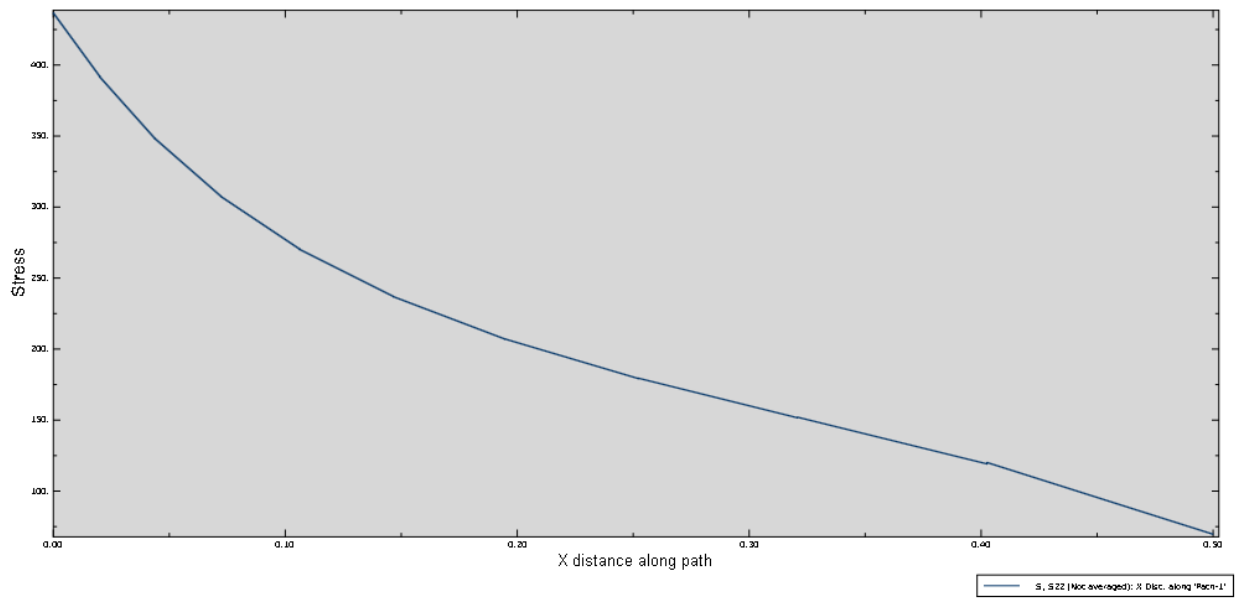
Typical Results (no edge bias)



Typical Results (with edge bias)

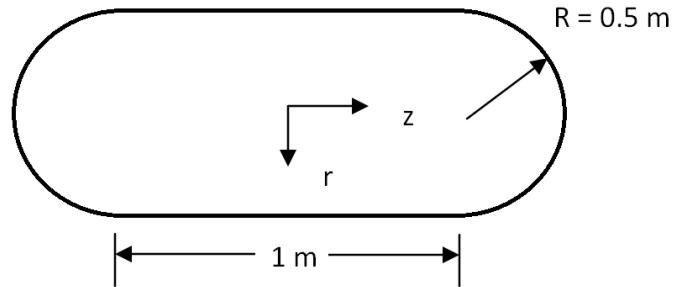


Stress Distribution along $y=0$ (bottom edge of mesh)



Tutorial 4. Axisymmetric Analysis

Consider a steel ($E=200$ GPa, $\nu=0.3$) 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:

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

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

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

$$\sigma_{VM} \approx 866 \text{ MPa}$$

and end cap stresses

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

$$\sigma_{\theta\theta} \approx \sigma_{\phi\phi} \approx \frac{pR}{t} = 500 \text{ MPa}$$

$$\sigma_{VM} \approx 500 \text{ MPa}$$

Finite Element solution

Start => All Programs => Abaqus 6.X => 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 ($U_2=U_{R1}=U_{R2}=0$)
BC => Create => Name: BC-2, Step: Step-1, Mechanical, Symmetry/Antisymmetry/Encastre =>
Continue => select left edge ($r=0$) => Done => XSYM ($U_1=U_{R2}=U_{R3}=0$)

Module: Mesh

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*)

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

Job => Manager => Results (transfers to Visualization Module)

Module: Visualization

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

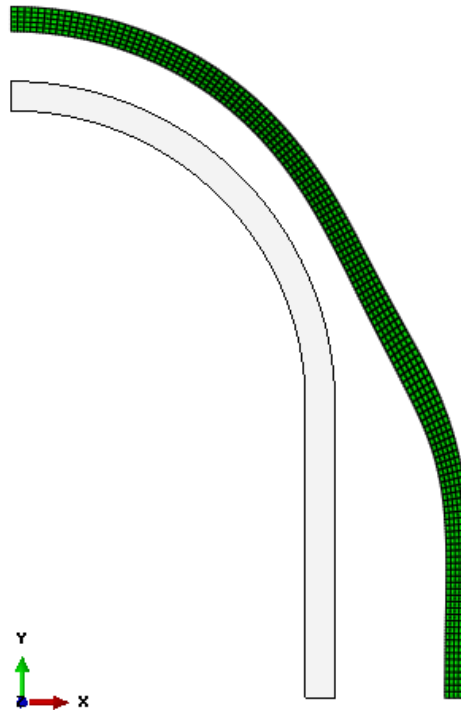
Ctrl-C to copy viewport to clipboard => Open MS Word Document => Ctrl-V to paste image

Tools => Query => Probe Values => Apply => select desired Field Output (S, Mises) => Probe
Nodes => move cursor to desired location to view nodal results

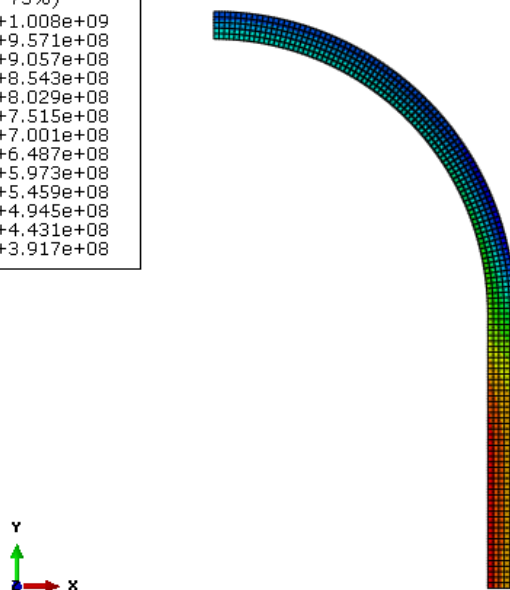
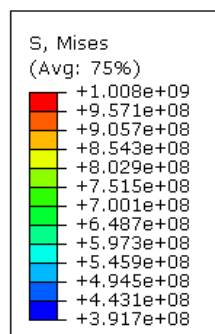
Along bottom symmetry plane ($z=0$, $.5 \leq r \leq .55$), Probe von Mises stress to show variation from
827 to 1,007 MPa (as compared to theoretical approximation of 1,000 MPa)

Along axis of symmetry edge ($1 \leq z \leq 1.05$, $r=0$), Probe von Mises stress to show variation
from 455 to 603 MPa (as compared to theoretical approximation of 500 MPa)

Undeformed and Deformed Shape:



Von Mises Stress contours



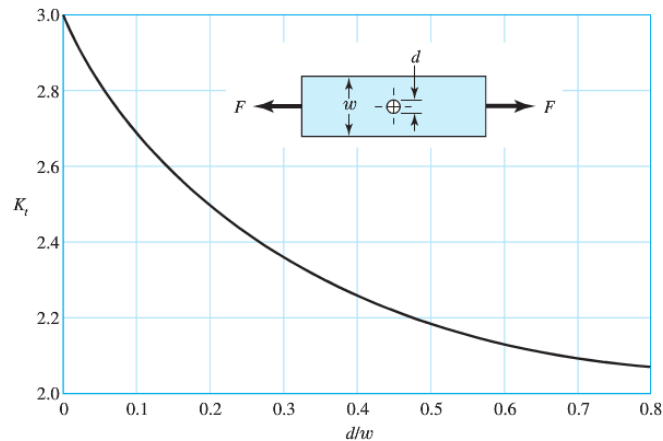
Tutorial 5. 3D Stress Analysis

Consider the problem studied previously using plane stress analysis. While nothing is gained by using a 3D finite element analysis for this problem, it does provide a simple demonstration case. For this demonstration, we will not impose symmetry as we did for the plane stress analysis. Again, this is not ideal modeling practice.

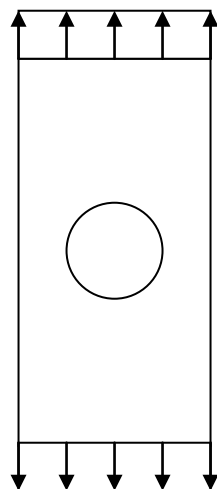
The problem to be considered is a 4" x 2" x 0.1" aluminum plate ($E=10e6$ psi, $\nu=0.3$) with a 1" diameter circular hole subjected to an axial stress of 100 psi. Determine the maximum axial stress associated with the stress concentration at the edge of the circular hole. Compare this solution with the design chart (ref. Shigley's Mechanical Engineering Design, 10th Edition, Budynas and Nisbett, 2015) for the case $d/w=0.5$ which gives $\sigma_{\max}=2.18$ (200 psi) = 436 psi.

Figure A-15-1

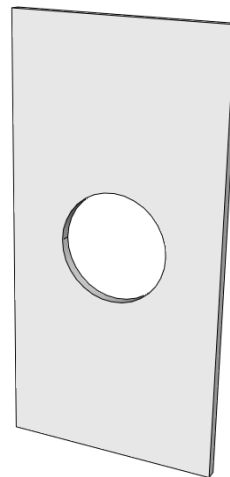
Bar in tension or simple compression with a transverse hole. $\sigma_0 = F/A$, where $A = (w - d)t$ and t is the thickness.



The geometry can be created using Abaqus drawing tools or by importing a part created in a CAD package. For this tutorial, we will demonstrate both creating the part in Abaqus and importing a part created in Solidworks. In Solidworks, saving the part in either ACIS (.sat) or Parasolid (.x_t) format works well.



2-D Problem



3-D Model

Finite Element solution

Start => All Programs => Abaqus 6.X => Abaqus CAE => Create Model Database With Standard/Explicit Model

File => Set Working Directory => Browse to find desired directory => OK

File => Save As => save three_D_tutorial.cae file in Work Directory

Creating the geometry in Abaqus:

Module: Sketch

Sketch => Create => Approx size - 50

Add=> Line => Rectangle => (-1,-2), (1,2) => right click => Cancel Procedure

View => AutoFit

Add=> Line => Circle => (0,0), (0,.5) => right click => Cancel Procedure

Done

Module: Part

Part => Create => select 3D, Deformable, Solid, Extrusion => Continue

Add => Sketch => select 'Sketch-1' => Done => Done => Extrude depth = 0.1

Importing the part (created by Solidworks, saved as ACIS .sat):

File => Import => Part => select file "plate_w_hole.sat" => OK => OK

Module: Property

Material => Create => Name: Material-1, Mechanical, Elasticity, Elastic => set Young's modulus = 10e6, 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 top face => Done => set Magnitude = -100 => OK

View => Rotate => rotate model to expose bottom face => red X

BC => Create => Name: BC-1, Step: Step-1, Mechanical, Displacement / Rotation => Continue => select bottom face => Done => U2 =0

BC => Create => Name: BC-2, Step: Step-1, Mechanical, Displacement / Rotation => Continue
=> select lower left corner of front face (where $x=-1$, $y=-1$, $z=.1$) => Done => $U1=U3=0$
(this prevents rigid body motion)

BC => Create => Name: BC-3, Step: Step-1, Mechanical, Displacement / Rotation => Continue
=> select corner of back face (where $x=-1$, $y=-1$, $z=0$) => Done => $U1=0$ (this prevents
rigid body rotation about the y-axis)

Module: Mesh

Seed => Edge by Size => select entire model => Done => Element Size=0.1 => press Enter =>
Done

Mesh => Controls => Element Shape => Hex /Sweep or Tet/Free

Mesh => Element Type => 3D Stress => Hex/Linear/Reduced Integration unselected, Hex/
Quadratic/Reduced Integration unselected, Tet/Linear or Tet/Quadratic => OK

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

Job => Manager => Results (transfers to Visualization Module)

Module: Visualization

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)

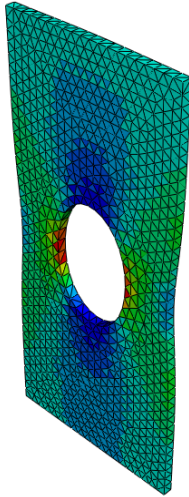
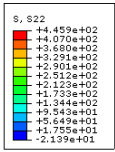
Plot=> Contours => On Deformed Shape

Result => Option => Unselect "Average element output at nodes"

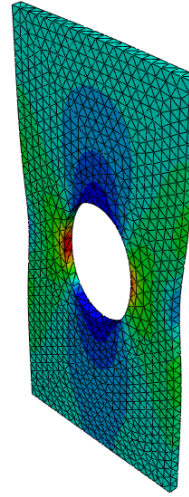
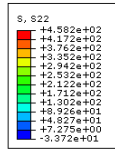
Result => Field Output => Name - S => Component = S22 => OK

Ctrl-C to copy viewport to clipboard => Open MS Word Document => Ctrl-V to paste image

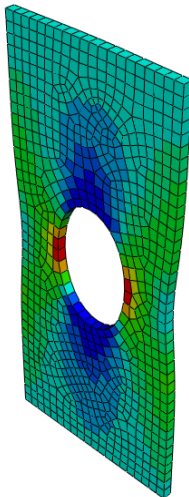
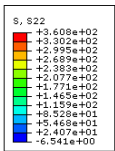
Tet elements – Linear
2,025 nodes
S22 (max) = 445.9 psi



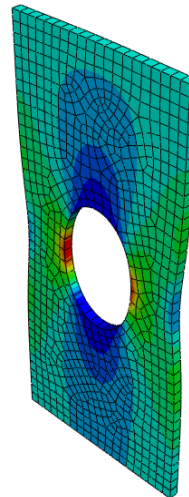
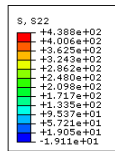
Tet elements – Quadratic
12,234 nodes
S22 (max) = 458.2 psi



Quad elements – Linear
1,798 nodes
S22 (max) = 360.8 psi



Quad elements – Quadratic
6,141 nodes
S22 (max) = 438.8 psi



Tutorial 6. Plate Bending Analysis

Consider a circular aluminum plate ($E=10\text{e}6$ psi, $\nu=0.3$) of radius 10" and thickness 0.2". The plate is simply supported around its outer perimeter and is subjected to a transverse pressure of 10 psi. Using plate (shell) elements, determine the deflection at the center of the plate. Plate theory gives the plate deflection as

$$w = \frac{PR^4}{64D} \left(\frac{5+\nu}{1+\nu} \right)$$

where

$$D = \frac{Et^3}{12(1-\nu^2)}$$

For our case, the predicted deflection is 0.290".

Finite Element solution

Start => All Programs => Abaqus 6.X => Abaqus CAE => Create Model Database With Standard/Explicit Model

File => Set Working Directory => Browse to find desired directory => OK

File => Save As => save plate_tutorial.cae file in Work Directory

Module: Sketch

Sketch => Create => Approx size - 50

Add => Circle => center point (0,0), perimeter point (10,0) => right click => Cancel Procedure
=> Done

Module: Part

Part => Create => select 3D, Deformable, Shell, Planar => Continue

Add => Sketch => select 'Sketch-1' => Done => Done

Module: Property

Material => Create => Name: Material-1, Mechanical, Elasticity, Elastic => set Young's modulus = 10e6, Poisson's ratio = 0.3 => OK

Section => Create => Name: Section-1, Shell, Homogeneous => Continue => Shell thickness = 0.2 => Material - Material-1 => 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 top face => Done => set Magnitude = 10 => OK

BC => Create => Name: BC-1, Step: Step-1, Mechanical, Displacement / Rotation => Continue => select perimeter => Done => U1=U2=U3 =0

Module: Mesh

Model Tree => Parts => Part-2 => double click on Mesh

Seed => Edge by Size => select entire model => Done => Element Size=0.5 => press Enter => Done

Mesh => Controls => Element Shape => Quad

Mesh => Element Type => Shell => Quadratic => 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

Job => Manager => Results (transfers to Visualization Module)

Module: Visualization

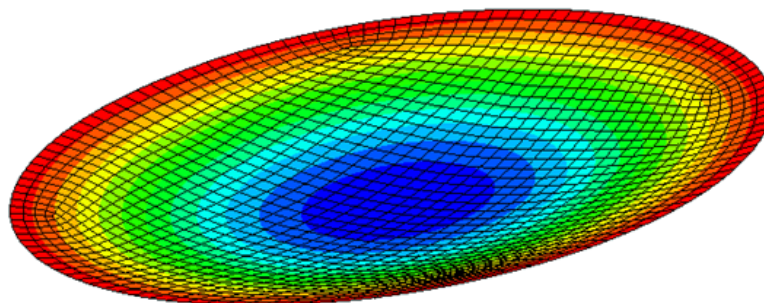
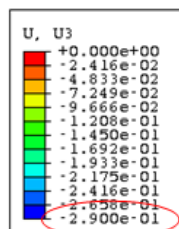
Viewport => Viewport Annotation Options => Legend => Text => Set Font => Size=14, Apply to: Legend, Title Block and State Block => OK => OK

View => Graphics Options => Viewport Background = Solid=> Color => White (click on black tile to change background color)

Plot=> Contours => Result => On Deformed Shape

Result => Field Output => Name - U => Component = U3 => OK

Ctrl-C to copy viewport to clipboard => Open MS Word Document => Ctrl-V to paste image



Tutorial 7. Column Buckling Analysis

Consider a 5 m column with a 10 cm circular cross-section ($R=0.05\text{m}$) loaded in axial compression. The column is pinned at its ends. Determine the critical buckling modes and corresponding mode shapes

Theoretical Solution

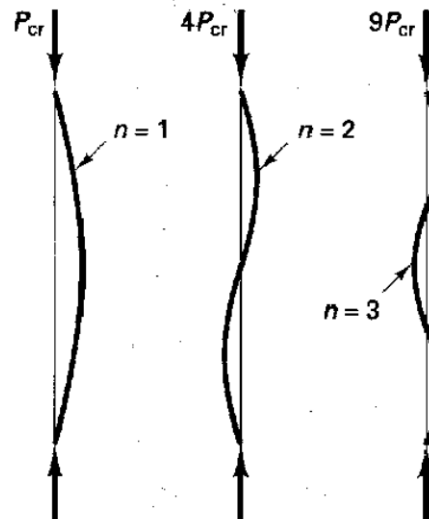
The theoretical Euler buckling loads are given by

$$P_{cr} = \frac{n^2 \pi^2 E I}{L^2}$$

For a steel column ($E = 200 \text{ GPa}$) with $I = 4.909\text{e-}6 \text{ m}^4$, the critical buckling loads and mode shapes are given by

Table 1. Theoretical Buckling Loads

n	P_{cr}
1	3.876e5
2	1.550e6
3	3.488e6
4	6.202e6
5	9.690e6
6	1.395e7



First three mode shapes

Finite Element solution

Start => All Programs => Abaqus 6.X => Abaqus CAE => Create Model Database With Standard/Explicit Model

File => Set Working Directory => Browse to find desired directory => OK

File => Save As => save buckling_tutorial.cae file in Work Directory

Module: Sketch

Sketch => Create

Add=> Point => enter coordinates (0,0), (0,5) => select 'red X'

Add => Line => Connected Line => select point at (0,0) with mouse, then (0,5) , right click =>

Cancel Procedure => Done

Module: Part

Part => Create => select 2D Planar, Deformable, Wire, Approx size 10 => 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

Profile => Create => Circular => r=.05 => OK

Section => Create => Name: Section-1, Beam, Beam => Continue => Section Integration – Before Analysis => Profile Name: Profile-1 => Basic => E=200e9, G=77e9 => OK => OK

Assign Section => select all elements by dragging mouse => Done => Section-1 => Done

Assign Beam Section Orientation => select full model => Done => n₁ direction = 0.0,0.0,-1.0 (enter) => OK => Done

Module: Assembly

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

Module: Step

Step => Create => Name: Step-1, Procedure Type: Linear Perturbation, Buckle => Continue => Number of Eigenvalues requested: 6 => OK

Module: Load

Load => Create => Name: Load-1, Step: Step 1, Mechanical, Concentrated Force => Continue => select point at (0,5) => Done => set CF 1 =0, CF 2 = -1 => OK

BC => Create => Name: BC-1, Step: Step-1, Mechanical, Displacement/Rotation => Continue => select point at (0,0) => Done => U1=U2=0

BC => Create => Name: BC-1, Step: Step-1, Mechanical, Displacement/Rotation => Continue => select point at (0,5) => Done => U1=0

Module: Mesh

Model Tree => Parts => Part-2 => double click on Mesh

Seed => Edge by Size => select full model by dragging mouse => Done => Element Size=.25 => press Enter => Done

Mesh => Element Type => select full model by dragging mouse => Done => Element Library: Standard, Geometric Order: Linear, Family: Beam, Cubic interpolation (B23) => OK => Done

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

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

Job => Manager => Results (transfers to Visualization Module)

Module: Visualization

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)

Result => Step/Frame => view Eigenvalues (Buckling Loads) - see Table 2 below

Plot => Select Undeformed Shape, Deformed Shape and Allow Multiple Plot States

Plot => Deformed Shape

Ctrl-C to copy viewport to clipboard => Open MS Word Document => Ctrl-V to paste image

Plot=> Contours => Result => Field Output => select S, Max. Principal => Section Points => Category: 'beam general' => select section points at +/- 2.5 to view stress contours.

Ctrl-C to copy viewport to clipboard => Open MS Word Document => Ctrl-V to paste image

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

Report => Field Output => Variable => Position: Unique Nodal => select U: Spatial Displacements, UR3: Rotational Displacements, S: Max. Principal => Apply

Cut and paste tabulated results from 'Abaqus.rpt' file to MS Word document.

Table 2. Buckling Loads (FEA)

1	Mode	1: EigenValue = 3.87579E+05
2	Mode	2: EigenValue = 1.55033E+06
3	Mode	3: EigenValue = 3.48844E+06
4	Mode	4: EigenValue = 6.20257E+06
5	Mode	5: EigenValue = 9.69442E+06
6	Mode	6: EigenValue = 1.39674E+07

Buckled Mode Shapes:

