ABAQUS Tutorial – 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, ν=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. Mechanical Engineering Design, 5th edition, Shigley and Mischke, 1989) value σ_{max}= 2.18 (200 psi) = 436 psi.

Charts of Theoretical Stress-Concentration Factors K_t*

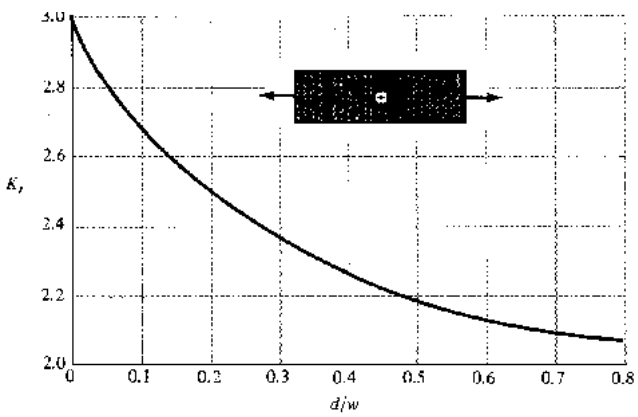
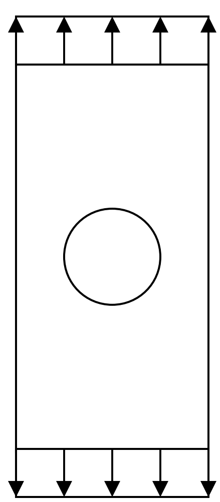


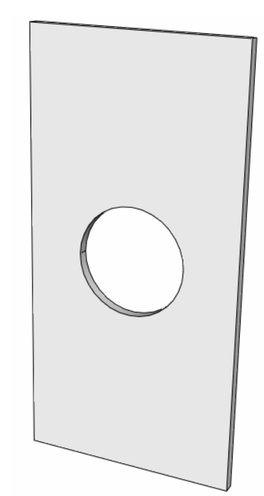
FIGURE A-15-1

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 (ABAQUS)

Start => Programs => ABAQUS 6.7-1 => ABAQUS CAE
File => Set Work Directory => select folder for Abaqus generated files
Select 'Create Model Database'
File => Save As => save .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 - 0.1 => OK
Assign Section => select entire part by dragging mouse => Done => Section-1 => OK

Module: Assembly
Instance => Create => Part-1 => Independent (mesh on instance) => 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 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

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 – note: teaching license limit is 10,000*)

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=> Contours => On Deformed Shape

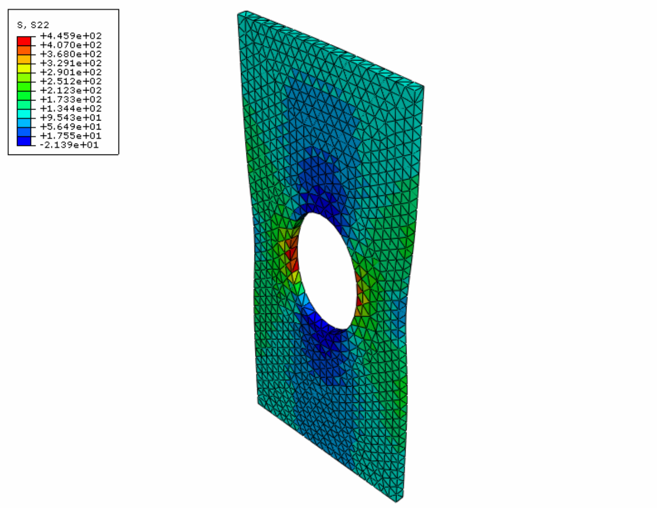
Result => Option => Unselect "Average element output at nodes"

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

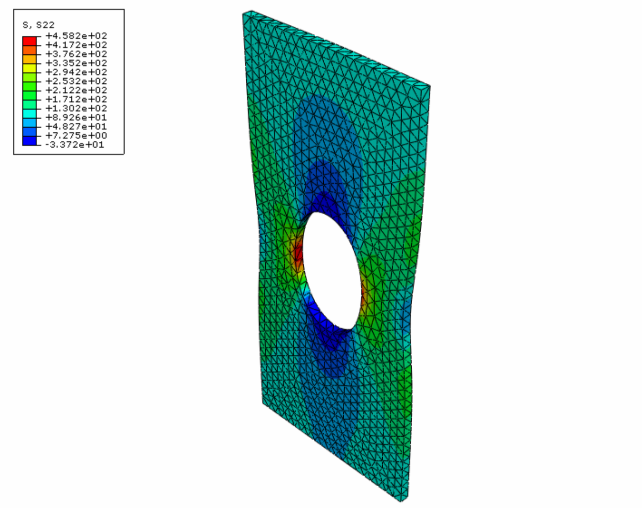
View => Graphics Options => Background Color => White

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

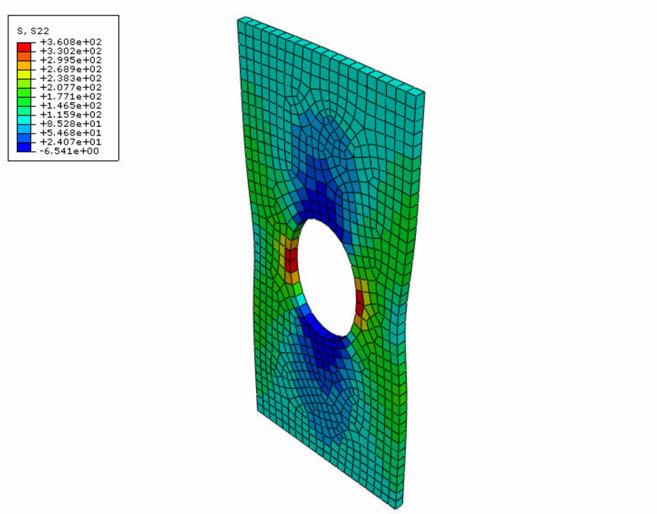
Tet elements – Linear
2,025 nodes
S22 (max) = 445.9 psi



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