## Tutorial 6. 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, v=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 σ_{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.
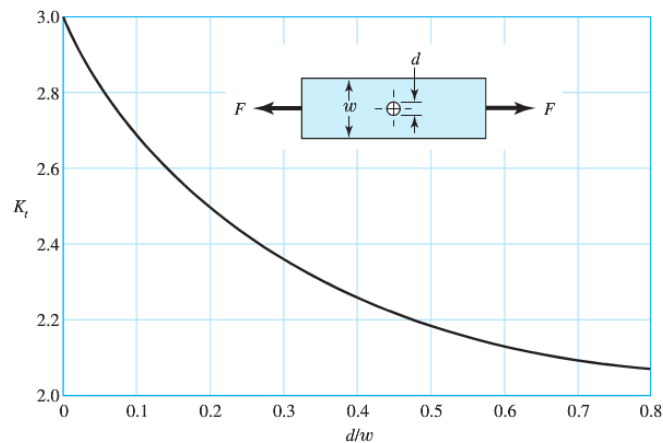
The geometry can be created using Abaqus drawing tools or by importing a part created in a CAD package. For this tutorial, we will demonstrate both creating the part in Abaqus and importing a part created in Solidworks. In Solidworks, saving the part in either ACIS (.sat) or Parasolid (.x_t) format works well.
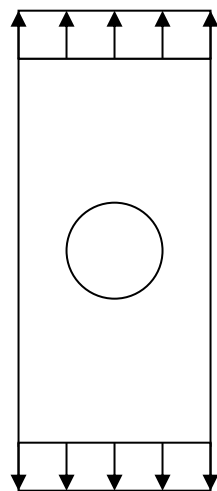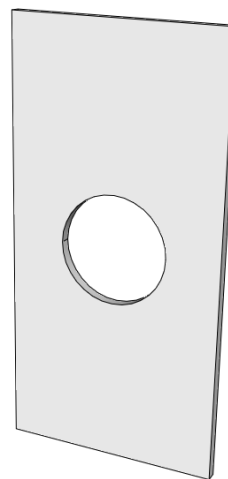
2-D Problem   3-D Model

Finite Element solution

Start => All Programs => 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 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
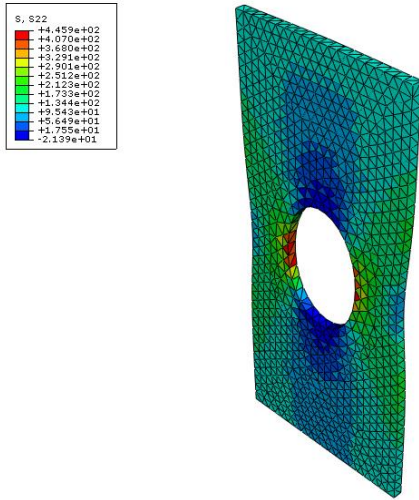tile to change background color)
Plot=> Contours => On Deformed Shape
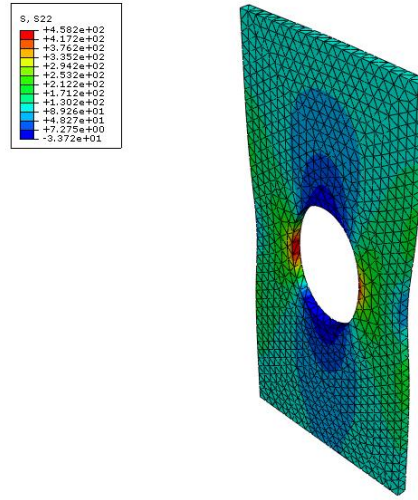Result => Option => Unselect "Average element output at nodes"
Result => Field Output => Name - S => Component = S22 => OK
Ctrl-C to copy viewport to clipboard => Open MS Word Document => Ctrl-V to paste image
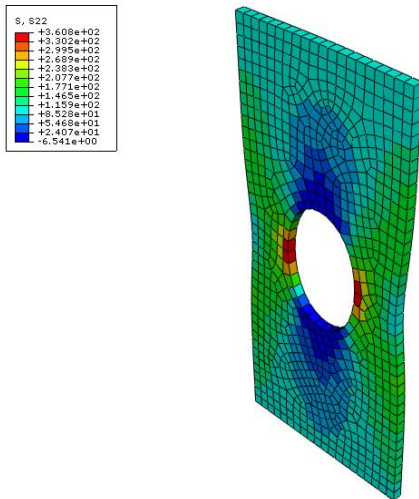
Tet elements – Linear
2,025 nodes
S22 (max) = 445.9 psi

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