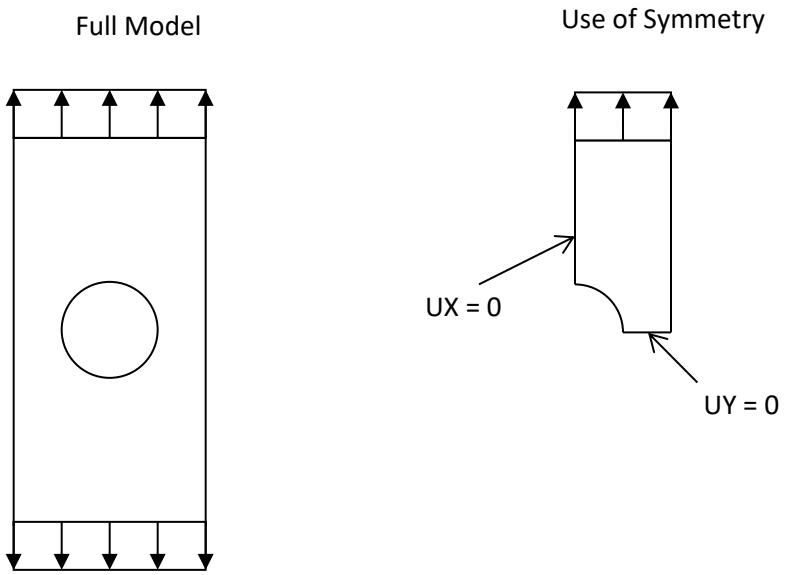
Tutorial 4. Plane Stress Analysis

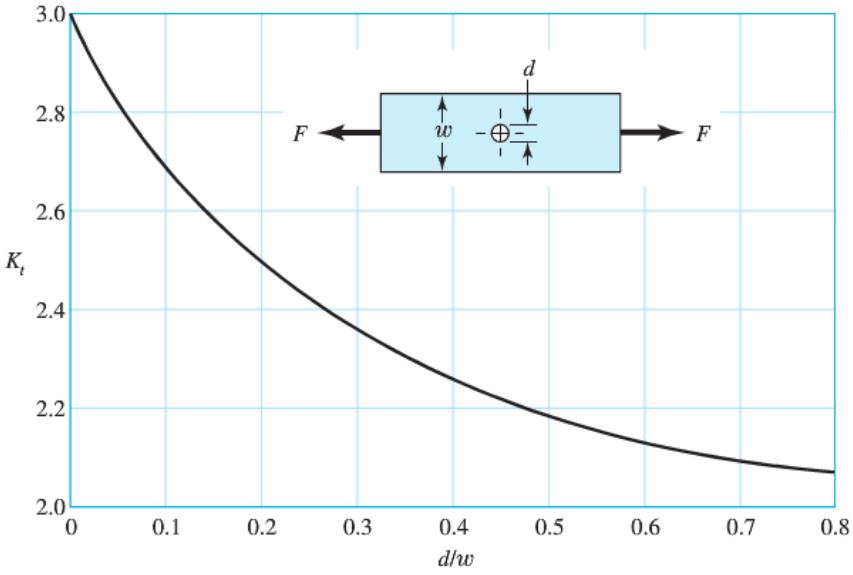
Consider the problem of a 4” x 2” x 0.1” aluminum plate (E=10e6 psi, ν=0.3) with a 1” diameter circular hole subjected to an axial stress of 100 psi.



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. “Shigley's Mechanical Engineering Design,” 10th Edition, Budynas and Nisbett, 2015) for the case d/w=0.5 which gives $\sigma_{max} \cong 2.18\ (200\ psi) = 436\ psi$.

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.



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 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*)

<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 => 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

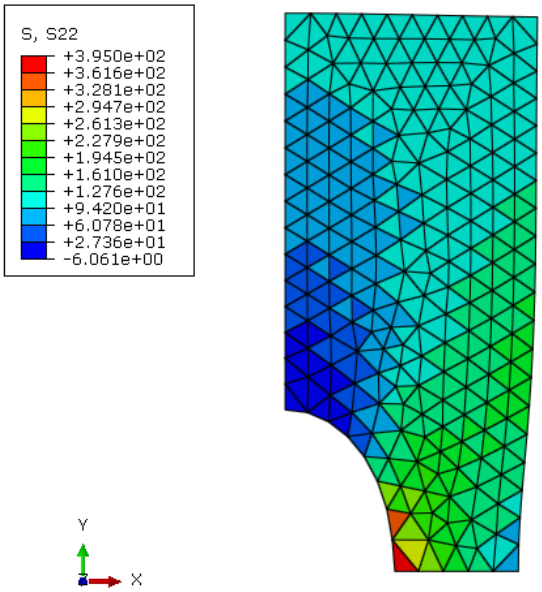
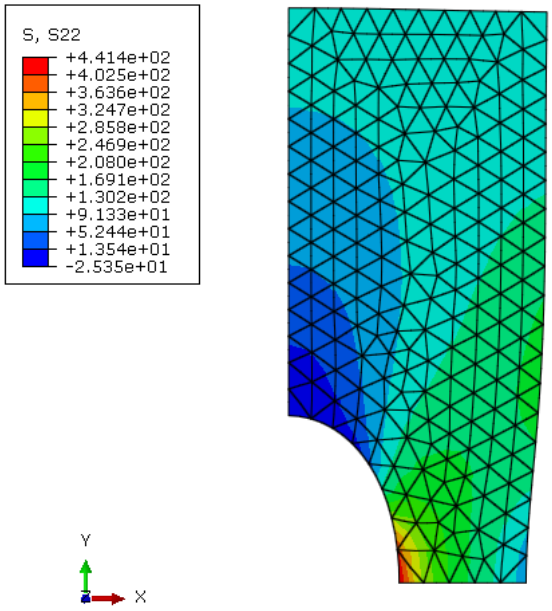
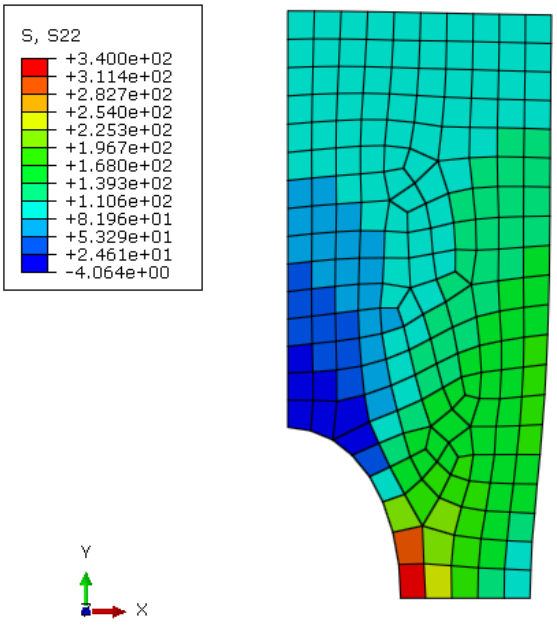
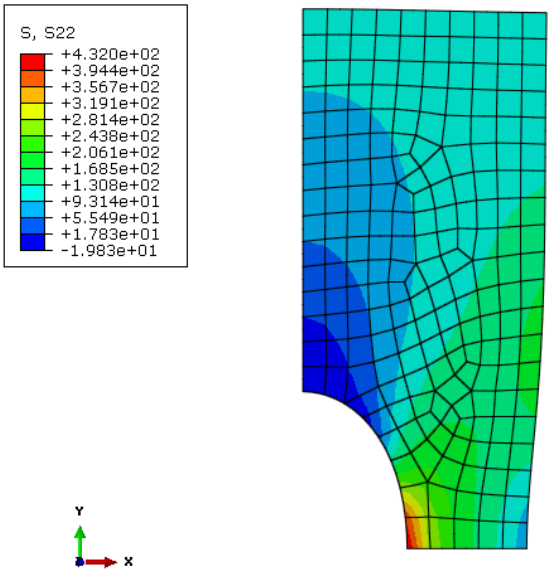
Tools => XY Data => Create => Source: Path => Continue => X Distance => Plot

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

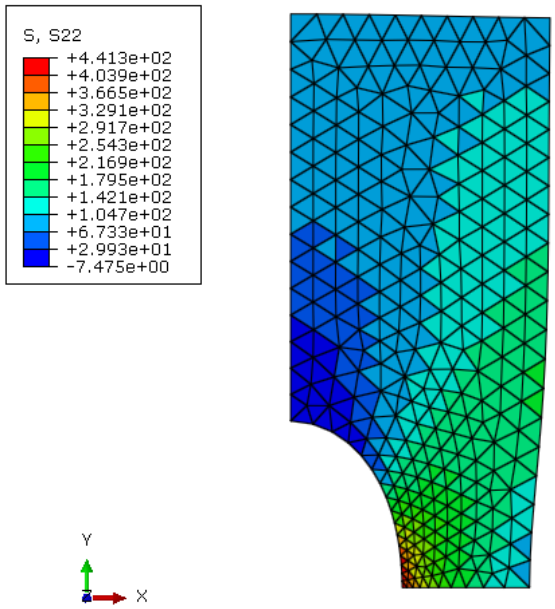
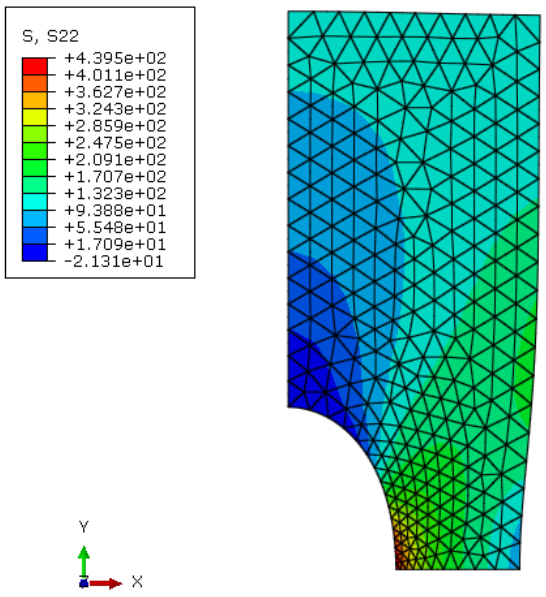
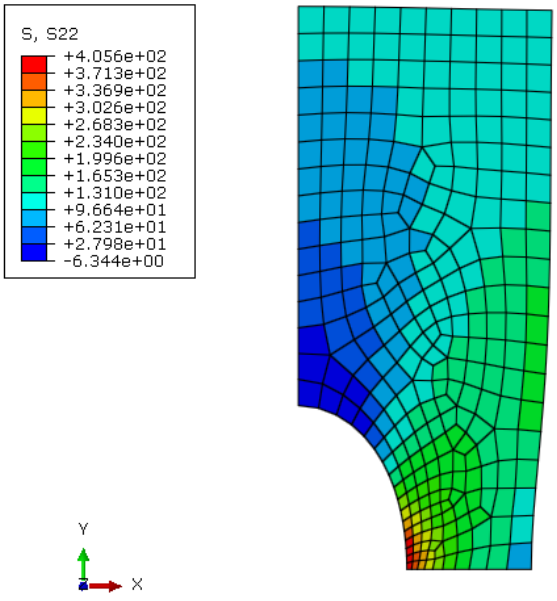
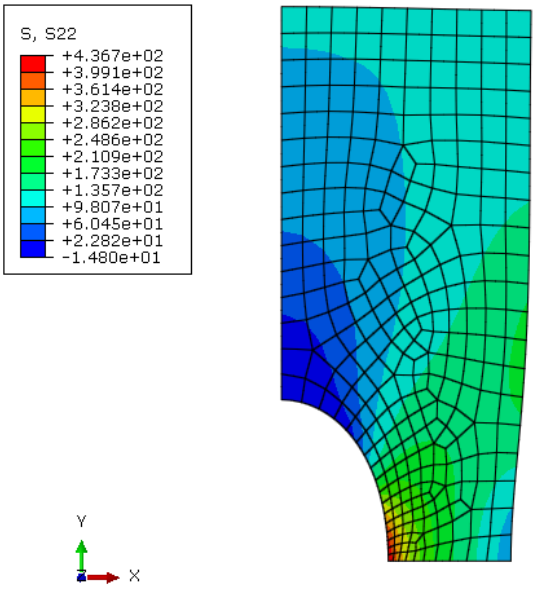
Report => Field Output => Position - Centroid => Variable – Mises, S11, S22, S12 => Apply

Examine tabulated results in 'Abaqus.rpt' file.

Typical Results (no edge bias)

CST - 3 node triangle (linear interpolation)	LST - 6 node triangle (quadratic interpolation)
	
4-node Quad (linear interpolation)	8-node Quad (quadratic interpolation)
	

Typical Results (with edge bias)

CST - 3 node triangle (linear interpolation)	LST - 6 node triangle (quadratic interpolation)
	
4-node Quad (linear interpolation	8-node Quad (Quadratic interpolation)
	

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