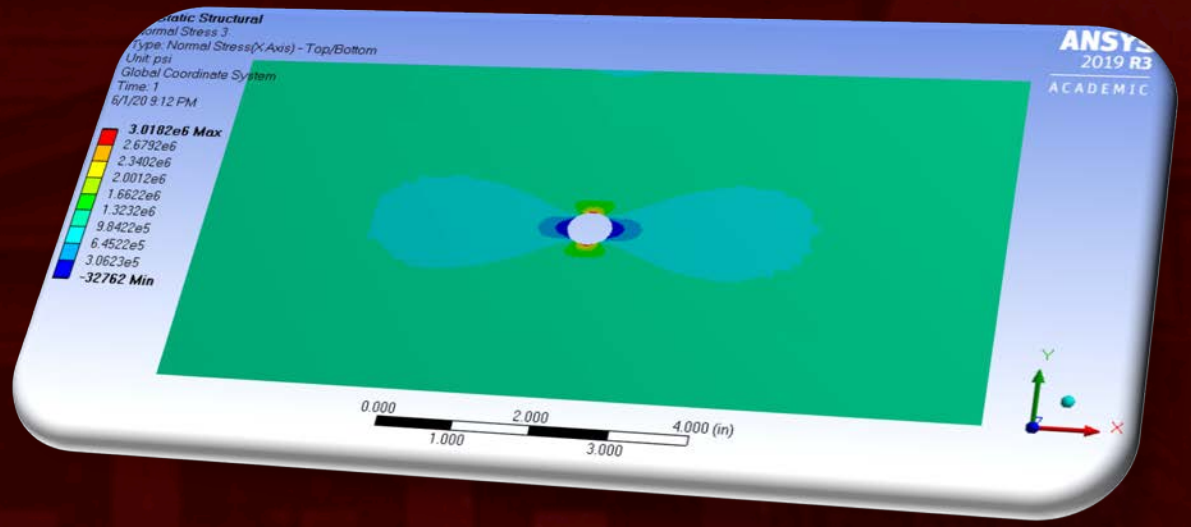
Introduction to FEA using Ansys-I

Umair Bin Asim
umair.asim@tamu.edu



Agenda

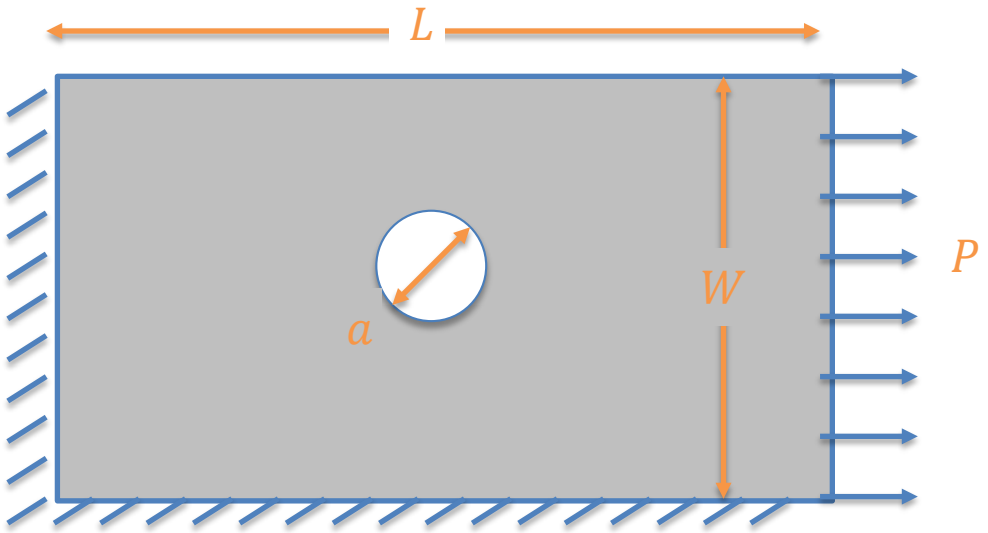
TEXAS A&M UNIVERSITY

Problem specification
Pre-analysis and start-up
Geometry
Mesh
Physics setup
Numerical solution and results
Verification and validation
Exercises

Problem specification

Plate with a circular hole:

- $P = 1 \times 10^6\ psi$
- $a = 0.5\ in$
- $W = 5.0\ in$
- $L = 10.0\ in$
- $t = 0.2\ in$
- $E = 29 \times 10^6\ psi$
- $\nu = 0.3$



Pre-Analysis

- Analytical solution exists only for an infinite plate with a hole
- Approximate analytical solutions
  - Displacement (without hole)

$$\Delta L = \frac{\sigma_{xx} \times L}{E} = \frac{1 \times 10^6 psi \times 5\ in}{29 \times 10^6 psi} = 0.34\ in$$

  - Radial stress (infinite plate with hole)

$$\sigma_r(r,\theta) = \frac{1}{2}\sigma_0\left[\left(1 - \frac{a^2}{r^2}\right) + \left(1 + 3\frac{a^4}{r^4} - 4\frac{a^2}{r^2}\right)\cos 2\theta\right]$$

  - Tangential stress (infinite plate with hole)

$$\sigma_\theta(r,\theta) = \frac{1}{2}\sigma_0\left[\left(1 + \frac{a^2}{r^2}\right) - \left(1 + 3\frac{a^4}{r^4}\right)\cos 2\theta\right]$$

  - Shear stress (infinite plate with hole)

$$\tau_{r\theta}(r,\theta) = -\frac{1}{2}\sigma_0\left(1 - 3\frac{a^4}{r^4} + 2\frac{a^2}{r^2}\right)\sin 2\theta$$

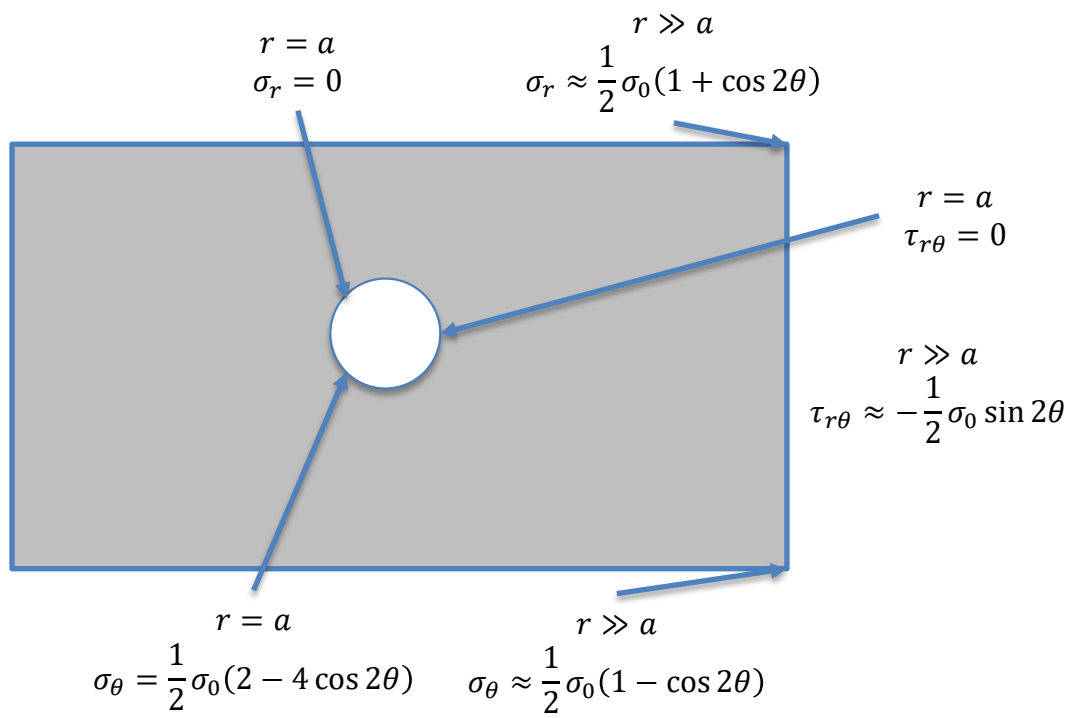
  - Normal stress (infinite plate with hole)

$\sigma_0 = 1 \times 10^6\ psi,$

$$\sigma_{nominal} = \frac{F}{A} = \frac{1 \times 10^6\ lb}{0.2 \times (5.0 - 0.5)\ in^2} = 1.11 \times 10^6\ psi$$

Concentration factor for an infinite plate with hole is $K = 3$:

$$\sigma_{xx_{max}} = K \times \sigma_0 = 3.0 \times 1 \times 10^6 = 3.0 \times 10^6$$



Start-Up

- Make sure you have an HPRC account and VPN service installed
- Go to https://portal.hprc.tamu.edu/
- Select ‘Ada OnDemand Portal’
- Use your NetID and password to login
- In the toolbar at the top, select ‘Interactive Apps’ and select ‘Ansys Workbench’
- Select/enter these values from/in drop-down/textbox
  - Ansys version: ANSYS/2019R3, Number of hours: 4, Number of cores: 1, Memory per core (GB): 2, Node type: GPU, Email (optional): (Enter your email address to get the status of your session).
- Click the ‘Launch’ button
- A new page opens, wait if it says so. Once a session has been setup click on ‘Launch noVNC in New Tab’ button
- Ansys Workbench will open in a new tab.
- Make sure not to close the tab and keep on saving the progress in ‘/scratch/user/*netid*/ansys_first_tutorial’.
- And if you have then do not panic, go to the previous tab and click ‘Launch noVNC in New Tab’ button again

Start-Up

- Workbench will start in a new tab of your browser
  - make sure to click on ‘maximize’ button at the right top corner so that you see the complete window
- Familiarize yourself with the interface toolbar, toolbox, ribbon bar, status bar
- Select ‘Static Structural’ from the toolbox
- Select ‘U.S. Engineering…’ from toolbar>Units
- Create a new material and name it
- Linear isotropic elastic material having Young’s modulus of $29 \times 10^6 psi$ and Poisson’s ratio of 0.3

Geometry

- Double click on ‘Geometry’ option under ‘Static Structural’ system in ‘Project Schematic’ (those who are using their own computers, right-click on ‘Geometry’ and select ‘New DesignModeler Geometry’)
- Again make sure to ‘Maximize’ it and acquaint yourself with the interface.
- Update the units once again
- Follow in class instructions to create 2D geometry of a 10”x5” plate with 0.5” diameter hole in the center

Mesh

- Double click on ‘Model’ option under ‘Static Structural’
- ‘Maximize’ the new window
- Update the units to U.S. Customary (in, lbm,…)’ by clicking on ‘Metric…’ text on the lower right hand-side on the status bar.
- Follow in class instruction to discretize the geometry into 0.1” sized elements and refine it 3 times around hole

Physics Setup

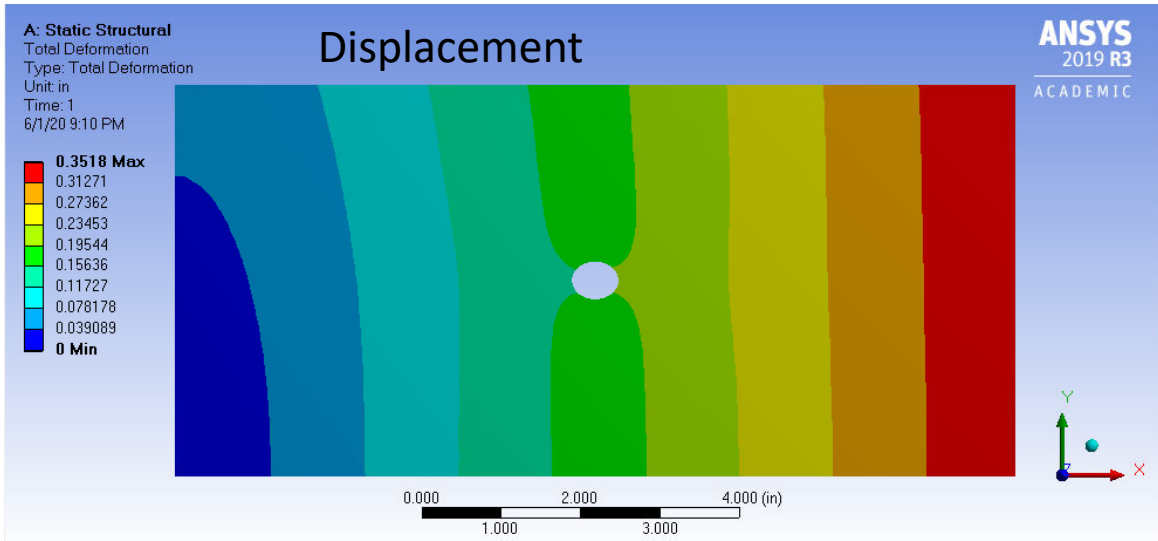
- Continue in the ‘Mechanical’ interface
- Assign the material properties created during start-up to the meshed geometry
- Select the behavior of the geometry as plane stress and assign a thickness of 0.2” to the geometry
- Apply ‘Frictionless Support’ (fixed displacement normal to edge/plane and no traction along the edge/plane)on the left and bottom edges and a ‘Pressure’ of $1 \times 10^6\ psi$ on the right edge.

Numerical Solution and Results

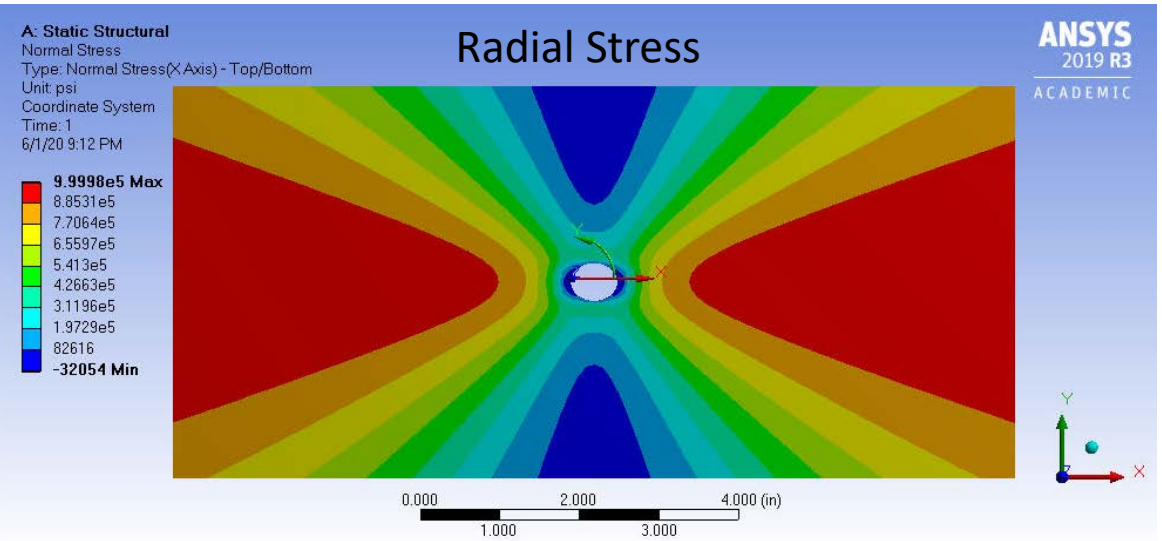
- Continue in the ‘Mechanical’ interface
- Call for the results of deformation and radial, tangential, normal (along x-axis) and shear stresses
- Solve for these results
- Look at the solutions for deformation and stress distribution in the plate

Results

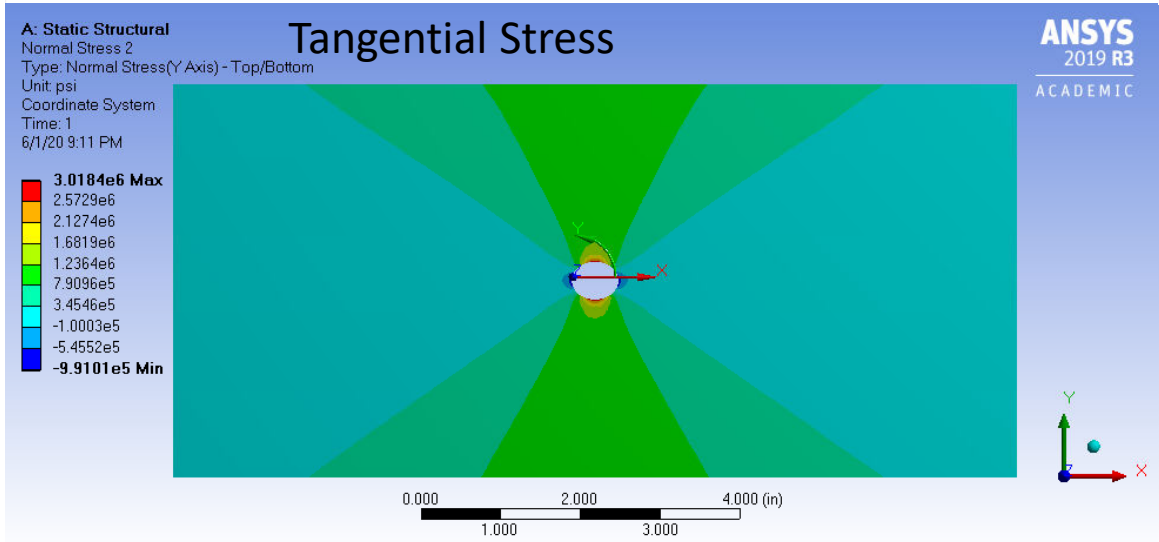
Displacement



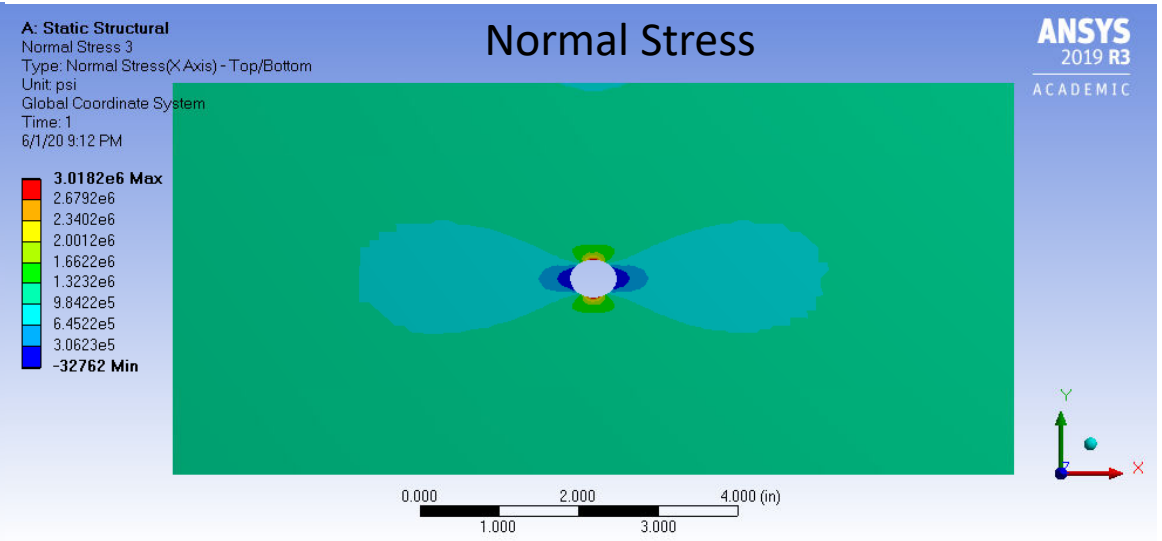
Radial Stress



Tangential Stress



Normal Stress



Verification and validation

TEXAS A&M UNIVERSITY

Comparison of the numerical results with analytical approximate solutions:

Quantity	Analytical	Numerical
Max displacement	$0.34\ in$	$0.35\ in$
Max σ_{xx}	$3.0 \times 10^6\ psi$	$3.02 \times 10^6\ psi$
σ_r Max, Min	$1.0 \times 10^6, 0.0\ psi$	$0.99 \times 10^6, -0.03 \times 10^4\ psi$
σ_θ Max, Min	$3.0 \times 10^6, -1.0 \times 10^6\ psi$	$3.02 \times 10^6, -0.99 \times 10^5\ psi$
$\tau_{r\theta}$ Max, Min	$0.5 \times 10^6, -0.5 \times 10^5\ psi$	$0.67 \times 10^5, -0.67 \times 10^5\ psi$

Do not forget to save the project.

Exercises

- Files of the Workbench project we worked on in today's session are available on ada through the portal (https://portal-ada.hprc.tamu.edu)
- In Ansys Workbench, open the following file
  /scratch/training/intro_ansys/intro_fea_ansys_day1/fea_first_session.wbpj
- Use the File/save as menu to save the project in your scratch space:
  /scratch/user/*netid*/*filename* (replace *netid* with your netid)
- Next, try refining the mesh and comparing the results
- Model quarter of the model and apply 'Symmetry' on the new edges using the procedure given at https://confluence.cornell.edu/display/SIMULATION/Plate+With+a+Hole+-+Physics+Setup and compare the results
- Attempt the exercises given at https://confluence.cornell.edu/display/SIMULATION/Plate+With+a+Hole+-+Exercises

Thank you