ME304 Finite Element Analysis, Fall 2019

By completing this assignment, students shall demonstrate the ability to:

- Apply linearity and unit loads in design using ANSYS
- Validate the workbook Echidna Beam using ANSYS
- Validate Superposition using ANSYS
- Use path plotting with ANSYS APDL to determine stresses along a "path"
- Provide assessment of element shapes

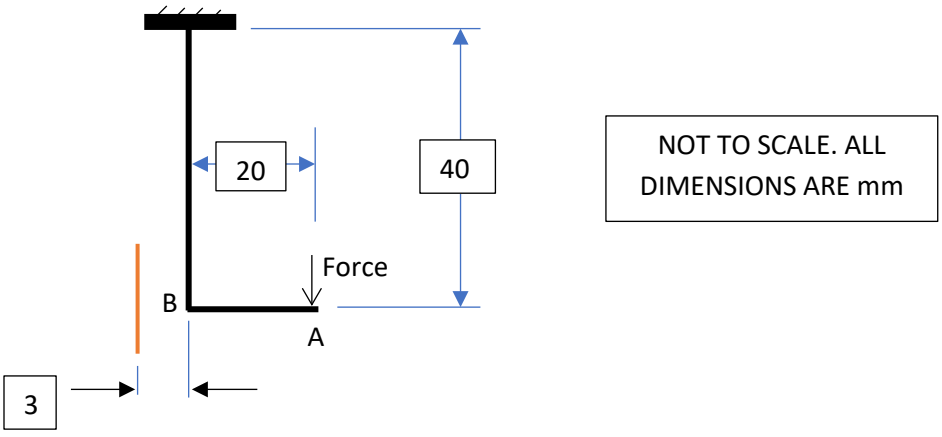
1. The company you are working for is in the process of designing a bracket. They want you to evaluate 2 design criteria. Both criteria involve determining how much force can be applied at point A (see sketch) to:

   i) cause point B to move 3mm to the left. The concern is that when a certain load is applied at the tip of the bracket, the bracket will deflect and interfere with another component.

   ii) cause yielding somewhere in the bracket (you do not need to worry about stress concentrations in the corner or attachment point).

   The bracket is made from a square cross section steel wire (210GPa) 3mmX3mm, and Sys = 200MPa. To solve this problem:

   a) Create an FEA model using ANSYS APDL script (you may use any of the 4 elements that was used in previous assignments).
   b) Apply a unit load (1 newton) downward at point A and determine the displacement in the x-direction of point B and the maximum von Mises stress in the bracket.
   c) Since this is a linear model, use the information from part (b) to determine how much force is required to move point B 3mm to the left and how much force is required to cause it to yield.
   d) Validate your answer in part (c) by applying the force in part (c) and see if point B moves 3mm, and a different force to determine if it will yield.
   e) Briefly discuss – what is the maximum allowable force; which criterion would fail with the lowest load: deflection or yielding?
   f) What to submit: Appropriate hand work (neat and following the standard problem-solving format) with appropriate ANSYS screen shots to answer the questions asked above.



2. Script for the Echidna beam is provided on the course page – run it with at least two different SMRTSIZE sizes. Analyze the stress in the x-direction. Note that the beam will look much longer than the workbook beam – the workbook beam is not to scale.
   a) Check element shape warnings (directions provided below). Include screenshots (zoom in and crop) to show elements of concern.
   b) Discuss stress results near the holes. Zoom in near the holes – save a screenshot of stress near the holes and submit with this assignment. Take a photo of your hand-colored Echidna beam and paste that into this assignment. If you didn't complete that part of the assignment, you may use someone else's work – but give them credit in your assignment. (You may submit gray-scale rather than color).
   c) Compare/discuss the results. Do you think the FEA model produced reasonable results even if there were element warnings?

3. Create an FEA model of the workbook "Superposition beam" using PLANE182 elements (do BEAM elements calculate in-plane shear stress? – that's a rhetorical question). Model with:
   a) only the aardvark
   b) only the bunny
   c) both.

   Use "path" commands (see directions below) to determine bending stress (σ_x) and in-plane shear stress (τ_{xy}) at the three points of interest (workbook). By hand, add the results from the aardvark only to the bunny only and compare with the results when both are on the beam. Briefly discuss how the FEA compared with your workbook results.

How to check for element shape warnings:

- After running the model, select from the GUI left side menu:
  - >General Postproc > Check Elem Shape > Plot Warning/Error Elements
- You may need to zoom in to see elements flagged as problematic

How to create a "path" and determine stresses along it:

- When creating the geometry, consider placing key points at the ends of the paths you wish to analyze. To create a path, you need to identify specific nodes, and ANSYS usually places node #5 at key point #5, etc. – *usually* – you should always confirm it when you create the path. Type NPLOT,1 into the command line and zoom in to see specific node numbers. To create a path:
  - >General Postproc > Path Operations > Define Paths > By nodes…enter the node numbers for both ends of the path (hit enter after each), select OK, give the path a name
  - >Map onto path…select desired stress and **enter a *Desired Label*** in the dialog box
  - >Plot path item > plot onto graph>*Desired Label* name you gave it.
  - To cut and paste actual values: >Plot path item > list path items>*Desired Label*