

ME304 Finite Element Analysis, Fall 2019
HW 8: FEA, Aspect Ratio, Symmetry, Boundary Conditions

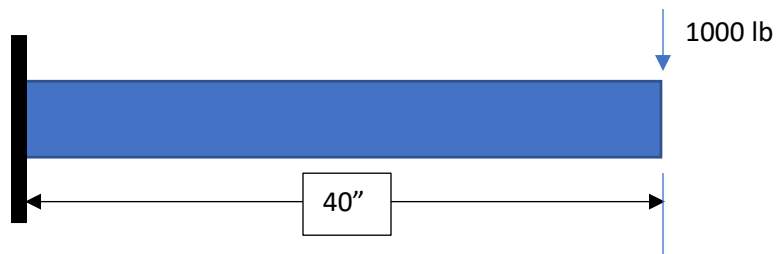
Student Outcomes. After completing this assignment, students shall be able to:

- 1) Apply boundary conditions (loads and constraints) using ANSYS script
- 2) Explain the effect of element aspect ratio on 2D elements (PLANE182)
- 3) Utilize symmetry in 2D problems to reduce the size of the FEA model
- 4) Create a line body model (BEAM189 elements) and analyze the results.

There are only three problems in this assignment. Student outcome #1 above is embedded in all. In the various scripts provided, anywhere there is an “xxx” – that’s for you to do some work.

1. Aspect ratio

The aspect ratio of an element is defined as the longest length divided by the shortest length. For best results, elements should **not** have a high aspect ratio (there is no specific definition of how “high” is “high”). For this exercise, you will study 2 differently oriented elements, each with two different aspect ratios (4 total models). All models will have 80 elements. You will be modeling a cantilever beam relatively similar to the Camel Beam in the workbook (same cross-section: 4”Height X 3”).



As always, before running the FEA model, analyze this by hand. Determine the bending stress (yes, turn this in as part of the assignment, include FBD, follow problem-solving format...)

To run ANSYS, you will need to modify the script provided (link on course web page) as follows: Apply boundary conditions (constraints and applied load) – fill in the xxx’s in the script provided. If needed, feel free to look up each command in ANSYS help, or from previously used scripts. Here is where the code needs revised:

```
!Constrain in ALL directions the nodes at left edge:  
NSEL, XXXX  
D, XXXX
```

```
!Apply force at top, 30” from attachment:  
NSEL, xxxxx  
NSEL, xxxxx
```

F,xxxx

Once you have created the boundary conditions, run the model. This model has an aspect ratio of 8 (4/0.5=8) for all elements – being longer vertically than horizontally.



The geometry of the beam was created using key points and connecting them with lines. Specific dimensions of the elements are achieved by dividing the top and bottom lines into lengths of “HORIZ” and dividing the left and right lines into lengths of “VERT”:

```
HORIZ=0.5 !Element size across top and bottom  
VERT=4 !Element size across edges
```

```
LESIZE,1,VERT  
LESIZE,3,VERT  
LESIZE,2,HORIZ  
LESIZE,4,HORIZ
```

And then meshing the area within the 4 lines:

```
/PNUM,AREA,1  
AL,ALL  
AMESH,ALL
```

After creating the constraints and loads, run the model as above (HORIZ=0.5, VERT=4) and record the maximum and minimum bending stress (stress in the x-direction). Then change the orientation of the elements (HORIZ=4, VERT=0.5). Record the max/min bending stress.

Then run it again with an aspect ratio of 2 (HORIZ=2, VERT=1)...record max/min.

Then run it again with an aspect ratio of 2 but reoriented (HORIZ=1, VERT=2)...record max/min.

Compare all FEA answers with the hand calculations (for bending stress) and briefly discuss the effect of aspect ratio and element orientation in this particular problem. No screen prints required.

H	V	x-dir max	x-dir min (max compression)
0.5	4		
4	0.5		
1	2		
2	1		

2. Symmetry

You will model the center plate with a hole in the center shown on page 58 of the workbook (100mmX30mmX10mm with a 20mm dia hole), and an axial load of 10kN. The complete script for this model is provided on the course web page. Run it, and determine max stress in the x-direction and the maximum and minimum in the y-direction. Record those values and **do a screen capture of the x-direction stress (include in hw submission)**.

An incomplete script file is provided (web page) for quarter symmetry (1/4 of the entire plate). You will need to determine appropriate loads and constraints, then run the model. Analyze the x and y direction stress as in the full-plate model. Record values and **do a screen capture of the x-direction stress (include in hw submission)**. Discuss/compare with the full-plate model above.

Remember – all engineering work should communicate effectively. For this problem, that would suggest a good sketch of the model and describing constraints you applied to achieve symmetry.

	x-dir max	y-dir max	y-dir min
Full-model			
¼ - model			

3. Frames and Machines

From a “statics” perspective, what is the difference between “trusses” and “frames” or “machines”? ...yes, that’s a question you need to answer for this.

Caterpillar (manufacturer of large equipment) is needing an FEA model of a backhoe boom assembly (the mechanical elements from the tractor to the bucket – but does not include the tractor itself). Details will be provided later.



Minimal script is provided for this problem. You may cut-paste from previous problems to complete the script provided. Assume all structural components are steel and have a 20 cm by 20 cm rectangular cross section. Here is the geometry and loading to model. It is held in place by 2 pinned connection (remember from statics: pins prevent translation in all directions, and allow rotation). There is a 1kN force applied at 45 degrees from horizontal as shown. Determine the maximum von Mises stress in the structure, and identify its location. Regarding constraints: each node has 6 DOF. Be sure to constrain the two nodes at the pinned connections in 5 of the 6DOF – allow Z-rotation (assuming you are creating the geometry in x-y plane).

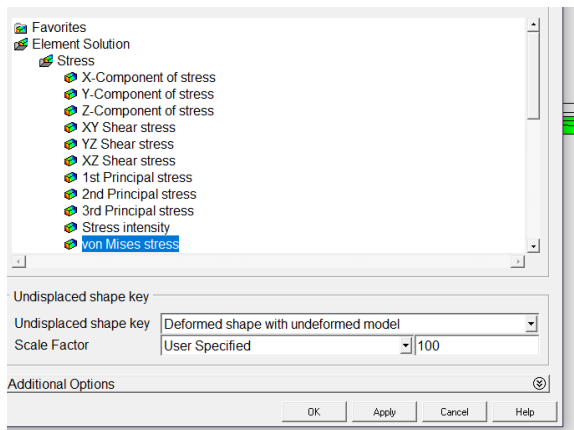
Not required, but here are a few simple things an engineer might want to do after running the model.

Determine the forces acting on the supports:

type NPLOT,1 in the Command Line to show the location and numbers of all nodes. Identify node numbers of the constrained nodes. From the left menu tree, select List Results>Nodal Loads>All OR... from the left menu tree, select List Results>Reaction Solu

See how the structure deforms (changes shape)

Plot von Mises stress, but in the window for selecting stress, show this:



To do other things ... spend a few minutes exploring options from the menu tree. Play.

