ME304 Finite Element Analysis
Fall 2019

Notes to augment the textbook problem 7.2 as applied to homework 4, problem 1 (and similar for other HW 4 FEA problems).

Follow the textbook closely! The following notes are details that the textbook does not include in 7.2. If no notes are made here, then assume the textbook is sufficiently clear (do not omit textbook steps). The brackets below correspond to bracket-steps in the textbook ([2] is step 2 in the section).

7.2.2
Select the Units tab at the top, select inches
[2] double click
[3] right click, select New Design Modeler

7.2.3
[1] icon should appear near top/center
[2] icon should appear near mid/center
[3] [4] for the first node in the hw, enter coordinates of 0, 0, 0 (x, y, z) – default is zero, so no need to enter values.
[5] icon should appear near mid/left
Repeat [2] and [3]
[4] enter X = 40 inch; Y = 0; Z = 0
[5]

7.2.4
Follow [1] through [6] once (we have only 1 element in the first hw problem)

7.2.5
[1] Concept/Cross Section/Rectangle (the book is using a different cross-section)
[2] type dimensions B=0.25 inch; H=2 inch

7.2.6
[2] click on Cross-Section, select Rect1 (book uses different cross-section)
Can skip [3] through [7]

7.2.7
Return to Workbench (tab at bottom of screen)
[1] double click Model (it will take several seconds before anything appears to happen)

7.2.8
[2] this looks different than what's shown in the textbook. Under "Defaults" click on Element Size and enter 40 inches (or larger)
Mesh/Generate Mesh (icon at left/bottom of toolbar)

7.2.9

To activate being able to use the mouse to select a node (btw, nodes are referred to as “vertices”), right click in the drawing area, select Curser Mode>Vertex

[1]Supports/Fixed Support (bottom of toolbar), click on node 1 (located at 0, 0, 0). “Fixed support” means this will prevent the node from moving in all 6-DOF.

A blue box should appear at node 1 indicated is “fixed” (constrained in all 6 DOF).

[3] [4] click on Geometry, select Apply

[5] loads/force click on second node to apply a force there. You will want to define the force vector – click on Define By Vector and then select Components. If you want 1000 pounds in the x-direction, enter 1000 pounds in the x-direction. A red arrow should appear in the drawing going in the x-direction (aligned with the bar).

Click Geometry, 1 Vertex: Apply

7.2.11

[2] appears bottom of toolbar

[3] NOTE: select the Tools icon at <u>bottom</u> of toolbar, not the Tools tab at the top.

[4] select “stress” from bottom/left toolbar

After “Solving”, view the results:
Min combined stress
Max combined stress
Min bending stress
Max bending stress
Also view displacements – you can select “probe” from the tool bar, that allows you to select nodes to see what their displacements (and internal forces?) are.
Are the results in agreement with the hand calculations?

NOTE: for hw 4, once you have problem 1 solved (axial load), problem 2 is trivial. Save your work once you have completed problem 1. Then modify the applied load from axial force to bending force. Here’s how:
The forces and constraints are defined in “Model > Mechanical” – see 7.2.7
You may need to select “Geometry” tab at the very bottom of the ANSYS screen to get to the appropriate screen.
Then as per 7.2.9 step [5] enter the desired force component. Problem 1 had Fx=1000 pounds. Problem 2 has Fx=0, but either Fy or Fz = 1000 (it depends upon how you have created the rectangular cross-section. See HW 4, problem 2 for the desired loading direction.)....then click on Geometry, 1 Vertex Apply. Then select “Solve”…etc.