ME 304 – Finite Element Analysis, Fall 2019
Assignment: 1D Linear Springs in ANSYS

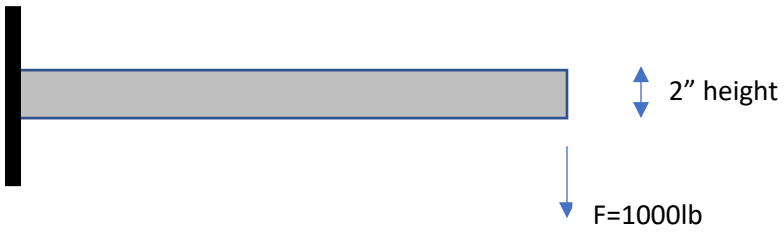
Model the following problem using ANSYS Workbench (search in Kiosk for "Workbench"). Use problem/example 7.2 in the textbook as a guide. You should also printout the "Book Augmentation" notes on the course web page (it provides additional details that the book omits).

For all FEA work please always include a sketch of the structure you are modeling, hand calculations (per strength of materials methods), tables defining nodes, elements and boundary conditions (as is done in Section 7.2). Only after completing the hand-work should you then proceed to work on the computer. Include printout of important results and include numerical results if comparing directly with hand-calculations.

Also: This is just a reminder to remember to advise your new students on saving their files to their P drives or OneDrive through https://myapps.up.edu when using the virtual machine kiosks before logging off. This will ensure they do not lose any of their saved work upon the reset of the virtual machines.

For all 3 problems in this assignment, assume the bar is structural steel with 2" X ¼" cross section. Use Young's modulus of 29Mpsi for hand-calculations (that is what ANSYS is using for Structural Steel). For each problem, determine by hand first, then using ANSYS: axial stress, bending stress, displacement at each node, reaction forces at the supports (the "wall").

1. Model a steel bar (2" X ¼"), 40 inches long, with an axial load of 1000 pounds applied at one end (fixed at the other).
2. Model a steel cantilever beam (2" height X ¼" thickness), 40 inches long, with a bending load of 1000 pounds applied at one end (fixed at the other).



3. Reminder, you've solved this problem by hand in hw 3 – so include that work with this assignment. You may assume this is a 100" long steel bar, but create the model with nodes at 40" from each end.