

**ME304 Finite Element Analysis, Fall 2019**  
**HW 6: 3D Hooke's Law, ANSYS Script**

Hooke's law questions (see Section 1.2 in the textbook and posted notes on the web page)

1. Show that 3D Hooke's law becomes the familiar "1D" Hooke's law:  $\sigma = \epsilon E$  for uniaxial loading.  $\sigma = \epsilon E$  is referred to as "1D" since it is for uniaxial loading (1-directional loading).
2. Show that 3D Hooke's law results in  $\nu = -\epsilon_{\text{transverse}}/\epsilon_{\text{axial}}$  for uniaxial loading.
3. Show that for a circular shaft with a torsion load applied (no axial or bending loads), that **both** plane stress **and** plane strain exists on the free surface. Be sure to include Mohr's circle.

ANSYS Script – You have experienced using ANSYS Workbench, which is a Graphical User Interface (GUI) for creating and running FEA models in ANSYS. Clearly, ANSYS is **not** a CAD package! We will now go "behind the GUI" and use the command-line inputs that are created "behind the scenes" of the GUI. The command-line inputs may seem foreign and certainly non-intuitive, but they offer certain significant advantages when learning to use ANSYS, namely:

- a) When software is updated (including FEA s/w), it is common for GUI's to be modified, but the command-line inputs remain unchanged (the command lines you will be using existed in their exact same form since the creation of ANSYS in the 1970's. Although some new command lines have been created since then, existing commands are not modified).
- b) It should greatly reduce your frustration level for class assignments. The basic text files for most of the upcoming ANSYS assignments will be provided to you – all you need to do is cut and paste. You may be asked to modify certain command lines to see the effects, which will be easy to do. Creating similar assignments using GUI is nearly impossible.
- c) There is no "undo" button in ANSYS, even the GUI! Once you do something, it may be difficult or impossible to undo it. If you skip a step, that can also be very problematic. With script, you can plan exactly what you want done in advance and in the order you want to do it. You can take existing models and easily modify them for new problems.
- d) You can bring in your script for me to look at and discuss – you can't do that if everything is created using GUI.
- e) And remember, it is not the purpose of this class to help you develop competency with any particular FEA package, but rather to help you become familiar with FEA in general. Seeing what is done to create a model using script is applicable for all FEA packages (even if the command syntax may vary).

Here is how to run ANSYS script:

- 1) Open up APDL on your computer by clicking the windows key and searching for "launcher". "Mechanical APDL Product Launcher 19.1" should appear (the version may be different). Click that.
- 2) A window will appear asking you to name the file. Type whatever job name you want and store it wherever by clicking browse. After inputting Job name, click "run".

- 3) Open the script file you wish to use (for this assignment, the script is provided as a link on the course web page).
- 4) Select all the text and copy it by highlighting and right-click “copy” (I have had some problems with shortcut key ctrl+c – sometimes it works, sometimes it doesn’t)
- 5) In ANSYS APDL, you will see a long, horizontal, white, blank space (the Command Prompt line) towards the top that you can type into.



- 6) Right-click in that long-white boxed area and click paste (right-click, select Paste, or you can try ctrl+v). Hit enter once, and then hit enter again. It will state that “/CLEAR will destroy... Continue?” Select “YES”. It will take about 10 seconds to solve, and then it will tell you that the “solution is done”.

Postprocessing (viewing the results):

Creating the postprocessing steps in the original text file is ineffective as each thing you want to observe replaces the previous results. Therefore, we need to interactively view results by either using the GUI or entering commands on the Command Prompt line (the long white blank space at the top of the display screen). Once you learn the commands, typing in the Command line is easier, but you need to know the syntax. I will try to give you step by step instructions for both methods as needed.

Comments about script: In script, exclamation points (!) are used to indicate comments (text following exclamation points (!) are ignored by ANSYS).

Comment on units: ANSYS does not interpret units, therefore the user must be consistent. If you enter force as kN and lengths as mm, then stress will be expressed as kN/mm<sup>2</sup>. In which case, you need to input material properties with those same units (Young’s modulus should be entered as kN/mm<sup>2</sup> if those are the units you have chosen to use).

#### 4. For problem 4 in this assignment (which is based on Problem 2, HW4).

- a. Create a sketch of the beam showing dimensions, loading and constraints
- b. Create node table (similar to section 7.2 in the book)
- c. Create a load and constraints table (similar to section 7.2 in the book)
- d. Create a materials property table (not done in section 7.2) identifying required material properties (use the values given in the code below).
- e. Read through the script on the course web page titled “HW6-Prob4-CantBeam” to understand the basic input commands. Run the model in ANSYS by highlighting the text below and copy it into ANSYS.
- f. Determine the rotational and translational displacement at the ends of the beam (view and report the ANSYS results). Identify the node number corresponding to the end of the beam (in step “c”) below)
- g. Determine the maximum and minimum bending stresses (view and report the ANSYS results)

After running the model, ask it to display the results you are interested in seeing, such as:

- a) To see the y-displacement:  
GUI: From the menu tree on the left, select “General Postprocessing>Plot Results>Contour Plot>Nodal Solution” From the popup window, select DOF Solutions>Y-component of displacement. To see the undeformed shape, select from the popup window: Undisplaced shape key>deformed shape with undeformed edge  
Command line: to get the same results, type into the command line bar at the top:  
PLNSOL,U,Y,0,1
- b) To see the rotation (how much nodes “twist”; expressed in radians):  
GUI: follow the above, but in the popup window, select Z-component of rotation  
Command line: PLNSOL,ROT,Z
- c) To see specific nodal displacements:  
GUI: From the menu tree on the left, select “General Postprocessing>List Results>Nodal Solution” From the popup window, select DOF Solution>Y-component of displacement.
- d) To see plots of stress:  
GUI: From the menu tree on the left, select “General Postprocessing>Plot Results>Contour Plot>Element Solution” From the popup window, select Stress>X-component of stress
- e) To see specific element stress:  
GUI: From the menu tree on the left, select “General Postprocessing>List Results>Element Solution” From the popup window, select Stress>X-component of stress.
- f) To see specific nodes, at the command line type NPLOT. Then type NPLOT,1 What’s the difference?

**5. Repeat the above problem, but change element size to 1”. Are the results (of parts a through e) the same or different? Briefly discuss.** To make the appropriate modifications, you will need to cut and paste from the online-posted document into a text editor (MS Word should work), modify the script as desired, and cut and paste that into ANSYS.

**6. Using the help menu in ANSYS, describe what each of the following do. Online HELP can be accessed by selecting the question mark (?) to the left of the Command Prompt Line.** In the online help, search for the desired prompt (for example, search for “DK” for part a below).

- a) DK,1,ALL,0 (what is DK, what does the “1” mean, “ALL” mean, “0” mean?)
- b) FK,2,FY,-1000 (what is FK, what does the “2” mean, “FY” mean, “-1000” mean?)
- c) ALLSEL (what does ALLSEL mean?)
- d) PLNSOL,Item,Comp,KUND,Fact,FileID (what is PLNSOL, what are options for the beam problem for Item, Comp, and KUND?)