

## Lab 1, Part III (FEM) Hints and Tips

Recall the basic steps needed to create a finite element model!

- Create nodes
- Define element type
  - May need to include section property data too
- Create elements based on the nodes (this is known as meshing)
- Define material properties
- Define boundary conditions (supports – pins, rollers, etc.).
- Apply external loads

**It is good practice to type up the commands into a text file and save this to the remote desktop (as well as your own computer, shared drive, etc.). Once you have the commands typed up, you can then go to File→Read Input From (and select the text file). Please review the various tutorial files (ansys\_demo\_truss and ansys\_beam\_elements) on Canvas.**

The script file below can rapidly generate the mesh of the truss using rod/truss (link180) elements.

!This demo file helps build the truss mesh (nodes, elements)

!The loads, boundary conditions, material properties and element section properties must still be defined

/prep7 ! enter pre-processor

n,1,0,0,0 ! Create 4 nodes

n,18,0,0.25,0

n,35,0,0.25,0.25

n,52,0,0,0.25

ngen,17,1,1,52,17,0.25,0,0,1! Copy nodes 17 times

et,1,link180 !defines rod, or truss element

E,1,2 ! Create 2 cell structure without the bars at the end

E,2,3

E,18,19

E,19,20

E,35,36

E,36,37

E,52,53

E,53,54

E,1,18

E,2,19

E,18,35

E,19,36

E,35,52

E,36,53

E,52,1

E,53,2

E,1,35

E,53,19

E,18,36

E,36,20

E,52,36

E,36,54

E,52,2

E,2,54

E,18,2

E,2,20

Egen,8,2,all ! Copy 8 times, increment nodes by 2 with each copy, copy all elements that currently exist

E,17,34 ! Add elements at the end of the truss

E,34,51

E,51,68

E,68,17

E,51,17

### **Task 1: Plot Results from FEM**

ANSYS can generate the deflected shape of the truss.

- General post-processor → Plot Results → Deformed Shape

ANSYS can also provide a contour plot of the forces in the truss.

- General post-processor → Plot Results → Contour Plot
- See the ansys\_demo\_truss text file on Canvas
  - One can extract the forces in the element using the etable command.
  - These results can be plotted: General post-processor → Plot Results → Contour Plot → Elem Table

Remember to consult a website that lists the ANSYS commands! This helps you better understand the various options, settings, restrictions on elements, etc.

**[https://www.mm.bme.hu/~gyebro/files/ans\\_help\\_v182/ans\\_cmd/Hlp\\_C\\_CmdTOC.html](https://www.mm.bme.hu/~gyebro/files/ans_help_v182/ans_cmd/Hlp_C_CmdTOC.html)**

## Task 2: Compare with Analytical and FEM Results

You are asked to find the reaction forces (forces at the supports), displacements, and internal forces.

Reactions → General Post-Processor → List Results → Reaction Solution

Displacements → General Post-Processor → List Results → Nodal Solution → DOF Solution

One can obtain the internal forces using the ETABLE command. See the ansys\_demo\_truss text file on Canvas. One needs to first create a table, and then tell ANSYS what to extract. The internal forces for link180 element can be extracted via the ETABLE command.

**ETABLE**, *Lab*, *Item*, *Comp*, *Option*

- The label is the user-defined name of the table.
- The item shall be 'smisc' – see link below. We want to extract the force in the element, and to do this the 'comp' needs be set as '1.'
  - [https://www.mm.bme.hu/~gyebro/files/ans\\_help\\_v182/ans\\_elem/Hlp\\_E\\_LINK180.html](https://www.mm.bme.hu/~gyebro/files/ans_help_v182/ans_elem/Hlp_E_LINK180.html)

### Task 3: Uncertainty Analysis

You are asked to discuss the following:

- Imperfect joints, such as free-play, may reduce the effective stiffness of the bars.
  - How to change the stiffness? What material property is akin to stiffness? Can this be changed/lowered in your FE model?
- Free-play in the in-line load cell may also reduce the effective stiffness of the bar containing the in-line load cell.
  - See the above comment, but here we may wish to only reduce the stiffness in a particular element (as opposed to changing the stiffness in all of the elements).
  - One may need to incorporate several material models, or types, within ANSYS (one for the element to be modified, and another for the rest of the elements).
  - See the ansys\_demo\_truss text file and look at the 'mp' commands (this is where material properties are defined).
    - See the ansys\_beam\_elements text file on Canvas as this tutorial made use of multiple material types (each having different properties).
- Friction at supported joints may require considering different boundary conditions.
  - You are likely modeling the truss with pinned conditions on one side and a roller at the other side.
  - Could the boundary conditions be changed to simulate friction?
    - Consider adding a small load at the roller that opposes the motion of the truss when it is loaded.
- Manufacturing imperfections may change the relative position of the joints and the cross-section of the struts.
  - A mistake in manufacturing could perhaps alter the cross-section of the elements (maybe making them thinner, say).
  - How can this be changed? What element property could be adjusted, or modified?
- The bar model assumes a uniform axial stress distribution in the struts; thus, the struts have no internal bending moments and the joints are modeled as perfect pin-joints. Consider that the joints can transmit bending moments and the struts provide bending stiffness, i.e., act as beams.
  - How to analyze? Consider using a different element, one that can incorporate bending! See the beam 188 element and review the ANSYS tutorial (text file on Canvas) – ansys\_beam\_elements
  - More information on the beam188 element is here:  
[https://www.mm.bme.hu/~gyebro/files/ans\\_help\\_v182/ans\\_elem/Hlp\\_E\\_BEAM188.html](https://www.mm.bme.hu/~gyebro/files/ans_help_v182/ans_elem/Hlp_E_BEAM188.html)