

- Solve the following truss example:

From Reference [1]: Figure 22.4 (also shown in figure 1). RunName = TrussExt.

- You can refer to the following link from the Rice University

[https://www.clear.rice.edu/mech403/DemoFiles/ANSYS\\_demo/ANSYS\\_BeginnerTruss.pdf](https://www.clear.rice.edu/mech403/DemoFiles/ANSYS_demo/ANSYS_BeginnerTruss.pdf) for a demo on solving a truss in Ansys if you plan to use Ansys for the solution of this problem.

- Deliverables are:

1. Commercial code input file to be executed (*i.e.*, TrussExt.db file with for Ansys). I also strongly recommend to submit text files generated by the software, listing the nodal positions, element properties and numbering, and element section areas from the preprocessor stage; then if the input file cannot be opened and your solutions are not correct, I can give you partial credit for this assignment.
2. The image file TrussExtDeformed.\* (*e.g.*, TrussExtDeformed.png) of the truss deformed geometry superposed on its undeformed geometry. In Ansys use
  - Main Menu  $\Rightarrow$  General Postproc  $\Rightarrow$  Plot Results  $\Rightarrow$  Deformed Shape
3. The text file TrussExtReactions.txt listing all 3 reaction forces (x and y directions for node 1 and direction y for node 12). In Ansys use:
  - Main Menu  $\Rightarrow$  General Postproc  $\Rightarrow$  List Results  $\Rightarrow$  Reaction Solution.
  - In List Reaction Solution pick All items for Item to be listed, OK.
4. The text file TrussExtElementNodalSolutions.txt including nodal solutions (x, y, and total displacement values). In Ansys follow the steps:
  - Preferences  $\Rightarrow$  general Postproc  $\Rightarrow$  list Results  $\Rightarrow$  Nodal Solution.
  - In List Nodal Solution  $\Rightarrow$  nodal Solutions  $\Rightarrow$  DOF Solution  $\Rightarrow$  Displacement vector sum, OK.
5. The text file TrussExtElementForces.txt listing all 20 elements axial forces. This needs listing element solutions in their local coordinate system and the output is generated differently in FEM codes. In Ansys use,
  - Main Menu  $\Rightarrow$  General Postproc  $\Rightarrow$  List Results  $\Rightarrow$  List Element Solution  $\Rightarrow$  element Solution  $\Rightarrow$  Line Element Results

### References:

[1]. Introduction to Finite Element Methods (ASEN 5007), Fall 2013, Department of Aerospace Engineering Sciences, University of Colorado at Boulder: Part III: Computer Implementation of Finite Elements: Chapter 22 Index. FEM Programs for Trusses and Frames.

Course URL: <http://www.colorado.edu/engineering/cas/courses.d/IFEM.d/>

Chapter 22 URL: <http://www.colorado.edu/engineering/cas/courses.d/IFEM.d/IFEM.Ch22.d/IFEM.Ch22.pdf>

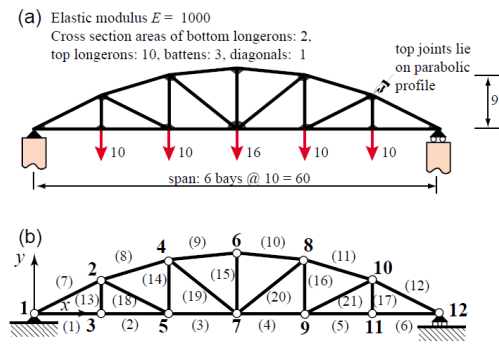


Figure 1: Truss problem description from reference [1]; for more information refer to the reference.

Due Date: 03/01/2015 (200 Points)