

- In section 3 we covered:
 1. section 3.1: Representation of data and functions pertinent to various FEM objects (*e.g.*, elements, nodes, and dofs).
 2. section 3.2: Steps needed from reading elements and nodes to FEM solution for node and element dof values and forces.
 3. section 3.3: Simplifications that can be employed in Matlab and description of input and output file formats.
- The term project involves:
 - Development of FEM solver for four different element types: 1. bars, 2. beams, 3. trusses, and 4. frames. The process to implement element specific routines (*e.g.*, stiffness and load vectors and well as output function) is described in course notes (section 3.3).
 - Limitations of implementation (*e.g.*, no source term and natural boundary force, same dof for all nodes, and some other are described in 3.3).
 - Input file format is provided in 3.3.
 - Beside solving the FEM problem (all steps in 3.2), the project involves Input/Output operation in accordance with the format given in section 3.3
- Deliverables are:
 1. **All Matlab or C++ files** should be submitted in a **zip file** with name “LastNameFEM-Project.zip”.
 2. The **output file** corresponding to the truss problem shown in section 3.3. RunName = Truss. Input file is shown in that section and can also be downloaded from:
<http://rezaabedi.com/wp-content/uploads/2014/04/Truss.txt>.
 3. From Reference [1]: **Input File** and **output File** for Figure 22.4 (also shown in figure 1(a)). RunName = TrussExt.
 4. From Reference [1]: **Input File** and **output File** for Exercise 22.3 (also shown in figure 1(b)). All geometry and material parameters are provided in the reference [2] page 22-12 “The SI physical units to be used are: mm for lengths, N for forces, and MPa=N/mm² for elastic moduli. For the calculations use the following numerical data: $L = 10,000$ mm (10 m), $H = 6,000$ (6 m), $a = 500$ mm (0.5 m), $P = 4,800$ N, $E = 35,000$ MPa (high strength concrete). The member cross section area is $A = a^2$, and the flexural moment of inertia about the neutral axis is $I_{zz} = a^4/12$ ”
RunName = FrameExt.
 5. From Reference [1]: Exercise 22.4 (A concept question about Exercise 22.3).

References:

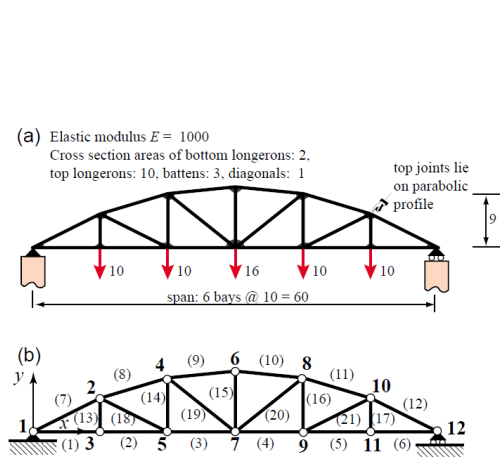
- [1]. Introduction to Finite Element Methods (ASEN 5007), Fall 2013, Department of Aerospace Engineering Sciences, University of Colorado at Boulder: Part III: Computer Implementation of Finite Elements: Chapter 22 Index. FEM Programs for Trusses and Frames.
Course URL: <http://www.colorado.edu/engineering/cas/courses.d/IFEM.d/>
Chapter 22 URL: <http://www.colorado.edu/engineering/cas/courses.d/IFEM.d/IFEM.Ch22.d/IFEM.Ch22.pdf>

Due Date:

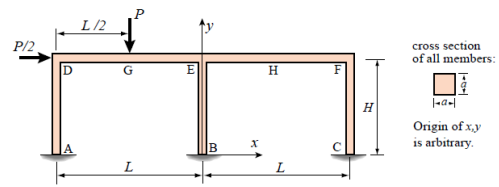
04/24/2015 for Input files needed for deliverables 3 and 4 (problems TrussExt and FrameExt)

05/04/2015 all other items.

(300 Points)



(a) Problem description for TrussExt



(b) Problem description for FrameExt

Figure 1: Brief description of two problems from reference [1]; for more information refer to the reference.