ES 128: Computer Assignment #1

Due in class on Monday, 22 Feb 2010

The ABAQUS input file of the truss structure shown in Fig. 1 is handed out – truss1.inp. You can download an electronic version of the file from the course website: http://www.seas.harvard.edu/courses/es128/.

The detailed explanation of the sample file will be given during the computer lab.

After the computer lab, you are asked to practice this sample problem by yourself, and then develop suitable input files for the remaining problems (#1-3) in this computer assignment. It will be simplest to approach that by editing *truss1.inp* into *.inp* files that describe your structures. The *ABAQUS STANDARD USER'S MANUAL* and *ABAQUS KEYWORDS MANUAL* will help with understanding the commands in *truss1.inp* (*KEYWORDS* are statements beginning with * in an *.inp* file, such as *NODE, *MATERIAL, *STEP, etc.).

If you have any questions please contact Prof. Katia Bertoldi (bertoldi@seas.harvard.edu) or teaching fellow Jian Zhu (jzhu@seas.harvard.edu).
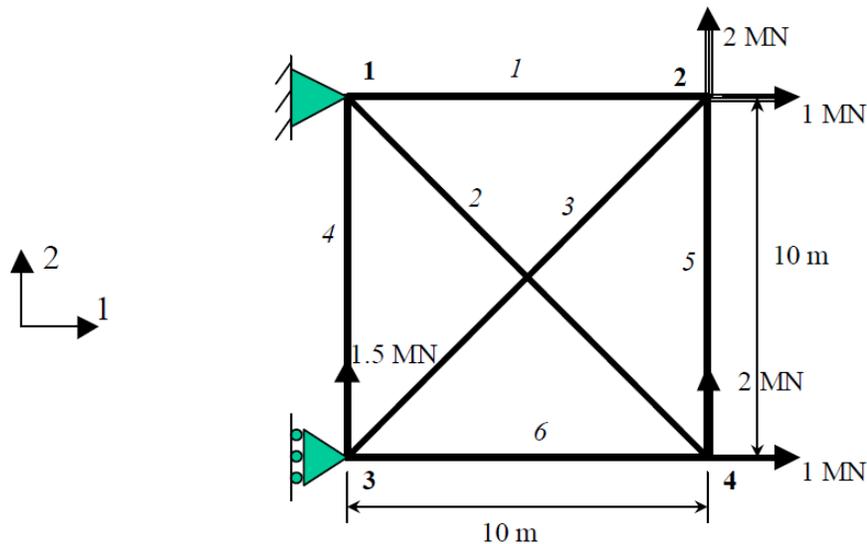

Figure 1. Truss structure

Tasks:

Task 1. Practice the use of ABAQUS by using the sample input file truss1.inp. From truss1.dat, print the stress (S11) and strain (E11) for all truss members, and nodal displacements (U1 and U2) of all nodes. Then, as a warm-up exercise, restrain both u1 and u2 for node #3. In other words, by fixing all the displacement components of both nodes 1 and 3, member #4 is now isolated from the structure. (Hint: To do so, simply replace "n3, 1, 1, 0.0" by "n3, 1, 2, 0.0" in the *Boundary keyword.) Compare the new results (stress, strain, and nodal displacements) with the old results.

Task 2. Now develop an input file the following problem.

A small railroad bridge is constructed of steel members, all of which have a cross-sectional area of 3250 mm². A train stops on the bridge, and the loads applied to the truss on one side of the bridge are as shown in Fig. P4.9. Estimate how much the point R moves horizontally because of this loading. Also determine the nodal displacements and element stresses.
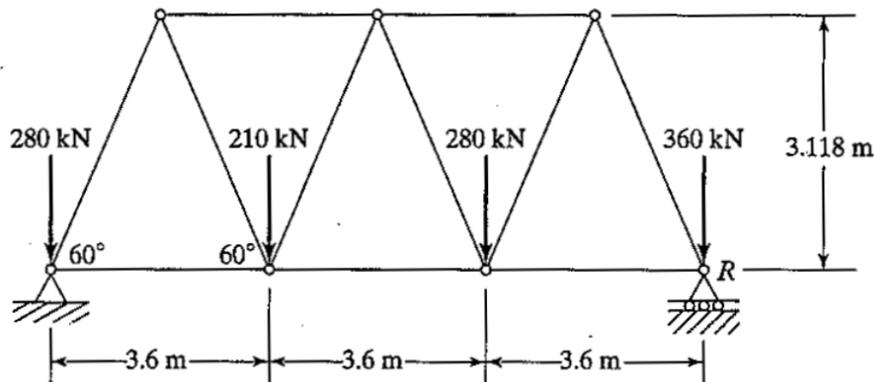

Analyze this railroad bridge problem by using ABAQUS. Find out the nodal displacements and element stresses for every node and element, including the horizontal displacement of point R. The railroad bridge is made from steel and has a Young's modulus of 210 GPa.

Now remove the middle bar in the bottom.
(a). What will happen to this structure?
(b) Use ABAQUS to verify your thought.
(c) Constrain the horizontal displacement at R. Find the reaction force.

Task 3. Now analyze using ABAQUS the 3D truss structure shown below

(a) Analyze the three-dimensional truss shown in Fig. P4.18. Identify the tetrahedral patterns in the truss.

(b) Generate the coordinates and connectivity if the two-stage truss shown is extended to 10 stages.

All members
cross-sectional area 900 mm^2
E = 200 GPa

5 kN
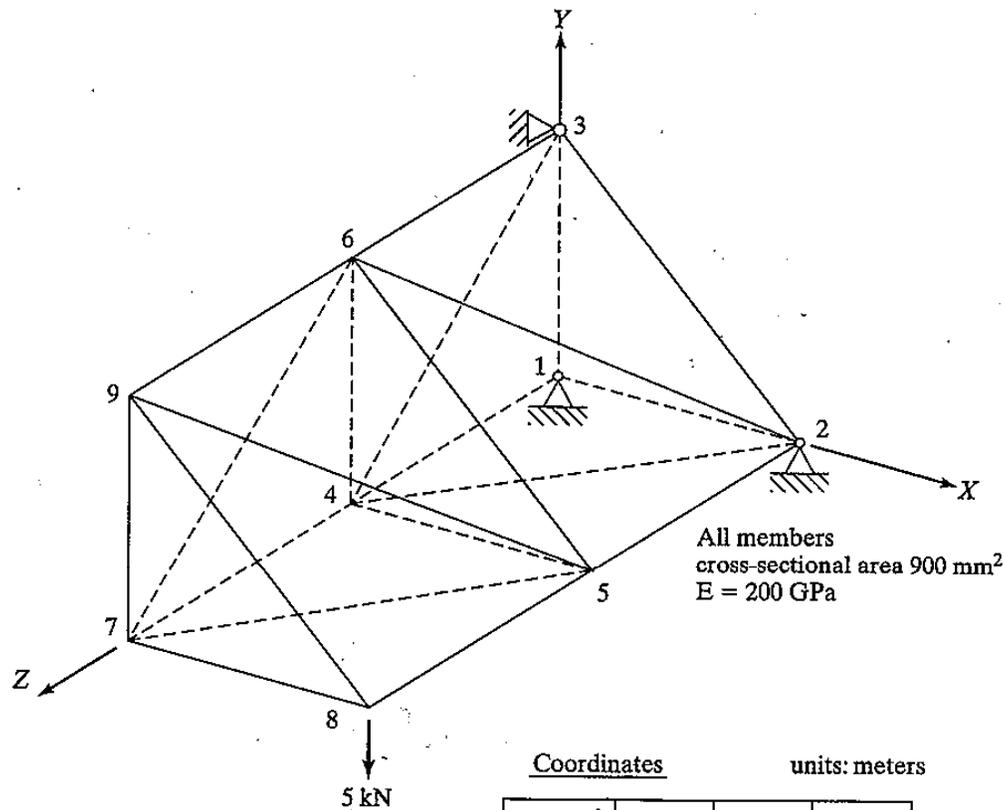
Connectivity

1 - 2	4 - 5	7 - 8
1 - 3	4 - 6	7 - 9
2 - 3	5 - 6	8 - 9
1 - 4	4 - 7	
2 - 5	5 - 8	
3 - 6	6 - 9	
2 - 4	5 - 7	
3 - 4	6 - 7	
2 - 6	5 - 9	

Coordinates units: meters

Node	X	Y	Z
1	0	0	0
2	0.25	0	0
3	0	0.25	0
4	0	0	0.5
5	0.25	0	0.5
6	0	0.25	0.5
7	0	0	1.0
8	0.25	0	1.0
9	0	0.25	1.0

Find out the nodal displacements and element stresses for every node and element. (Hint: 3D truss element is T3D2).