MTE 204 – Numerical Methods – S16

Project Ia (due Fri, May 20 @ 11:59 pm; 7.5% of total course mark)

Introduction

Project Ia has been designed to help students develop their mathematical modelling and algorithm design skills. This will allow students to familiarize themselves with basic concepts which will be needed for Project II. It is highly recommended that students start working on these problems as soon as possible. These projects must be completed in groups of five.

As part of this project, you will be required to mesh, build, assemble and numerically solve the global stiffness matrices. Each group will write a code, each of which should be generically capable of solving the types of problems shown below (i.e., no hard-coding). For Project Ia, you are permitted to use built-in libraries to solve for matrix inversion. However, this **MIGHT NOT** be the case for other projects.

Marking Criteria

The following marking metrics will be weighted as follows for Project Ia:

- Implementation of Mathematical Model into Code/Software (50%)
- Presentation and Discussion of Solution (50%)

Modeling Trusses[1]

Assume that all coordinates given are in metres and that cross-sectional area of all members is 0.0025 m^2. The Young's modulus for all blue elements is 200 GPa (steel) and for all red elements is 70 GPa (Aluminum). For all members, determine the final displacements of all nodes and reaction forces. Calculate the magnitude of stress in each member and indicate if the member is in compression vs tension.

For each problem, applied loads and displacements are prescribed. The applied forces are calculated using the student ID numbers of the group members. Student ID Numbers take the form of a 7 digit number in the form (abcdefgh). For example, if a student ID is 20212345, then $a = 2$, $e = 2$, $f = 3$.

Complete the following information:

Member #	a_i	b_i	c_i	d_i	e_i	f_i	g_i	h_i
1								
2								
3								
4								
5								
Σ								

*Note: For groups that do not consist of 5 members, update the table to reflect the number of members.

Using the following numbers, compute the magnitudes of the force;

$$F_a = 150 + 40\sum_{i=1}^{4}(c_i + d_i)$$

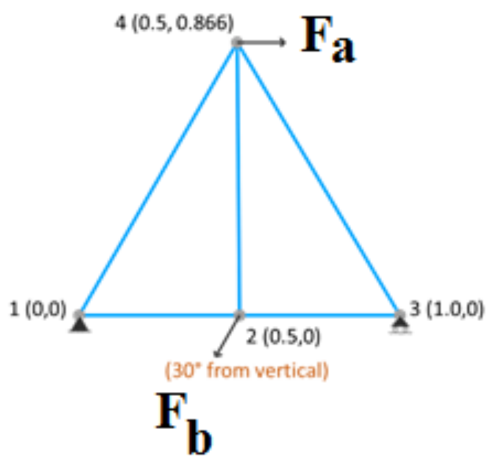
$$F_b = 30\sum_{i=1}^{4}(e_i)$$

$$F_c = 100 + 10\sum_{i=1}^{4} f_i$$

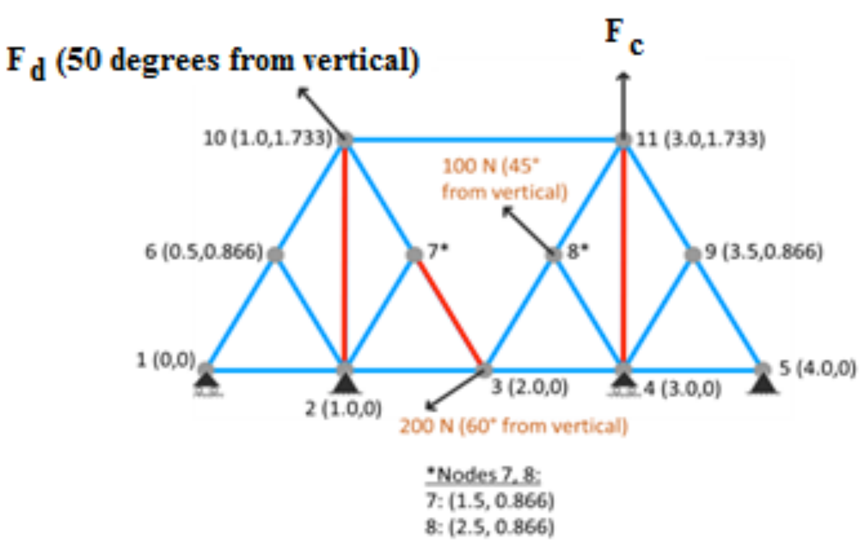
$$F_d = (-1)^{\left(\frac{F_b}{10}\right)} \times 50\sum_{i=1}^{4}(g_i + h_i)$$

Now, solve the following problems for the calculated forces;

Problem a)



Problem b)



[1]For a review on truss structures, please refer to: Beer et al., *Mechanics of Materials, 6th edition.* McGraw-Hill Education, 2011

Project 1a - Student Package

To be able to reduce the work load and the number of mistakes made by students, a student package that contains several MATLAB function skeletons, and input files has been provided.

The "main.m" is the main program that calls all the necessary functions, in the **correct** order, to perform a finite element analysis of a structure. Each function contains information about the data that is passed to the function, the operations to be performed, and the required output. **You cannot modify the "main.m" file,** ***except for changing names of reading and writing operations for different problems.*** This means that you **must** use essentially the same "main.m" program to solve both Problem a) and b).

When writing your code for this project, you must use all the functions provided to you as they are. This means you cannot modify the function call to add any additional information. These function calls contain all the information required to perform the operation as described at the beginning of each function. You cannot create any additional functions to perform the project.

The node list, element connectivity, element properties, nodal displacements/force boundary conditions have been provided for Problem b). A description of the structuring of these files can be found in "open_files.m". You must modify the applied magnitudes and directions in "nodeFORCES2.txt" according to your Student ID calculations. **You cannot modify the file structuring for Problem b).** The units are presented in millimeters for displacements, Newtons for force, and MPa for stress. Using this file structure in Problem b), you must create the required input files to perform Problem a).

Tips and Advice for Success

Before submitting your results, please verify that your results are correct. Problem a) is a simple truss that is a deterministic problem that can be solved analytically by hand. Thus, you can perform a hand-calculation to verify that your finite element program is correct.

When printing your results from your finite element analysis, you should check that nodes that are prescribed (i.e fixed boundaries) match the prescribed displacement (i.e. zero). Furthermore, please verify that global equilibrium is satisfied in all directions and that the applied forces are correct.

Submission

Prepare a document that presents your results for each question. For your submission documents, please include an appropriate title page, with the names and student IDs of all contributors, course number, course name, title, and date submitted. For each question, you are required to present the nodal displacements, reaction forces, magnitude of stress and the loading type in a tabular format. An example output is provided in the student package. You must provide your solution in millimeters for displacements, Newtons for force, and MPa for stress. The report should not exceed five pages.

All submissions will be submitted electronically using a designated dropbox posted on LEARN. All submission documents **must be** compiled as a .pdf file. All code used to generate your solutions **must be** submitted as a single file and compressed using a suitable format. **No paper copies will be accepted for grading.** Penalties will be applied for failing to satisfy these constraints on each submission (as presented in the Course Outline).