MTE 204 Numerical Methods

Project #2: Finite Element Analysis Applied to Uniaxial and Truss Structures

Fall 2022

Due: December 2, 2022, at 11:59 pm

Project Introduction

Finite element analysis (FEA) is commonly used in engineering to solve differential equations and systems of differential equations. FEA provides the ability to handle non-uniform geometries, model complex boundary conditions, interpolate values at any point in the solution domain, and collapse a complex problem into an easy-to-solve system of linear equations. To that end, FEA is widely encountered in many engineering disciplines.

One of the most common applications of FEA in mechanical engineering is the calculation of stresses and strains in structures. In this project, you will analyze two structures using the simplest finite element formulation, the truss element. The truss element is a one-dimensional element that is used to analyze axially loaded members or segments.

Project Requirements

In this project, you will perform FEA upon two different structures: a stepped shaft experiencing an axial load and the deformation of a simple truss structure. For both structures, you will:

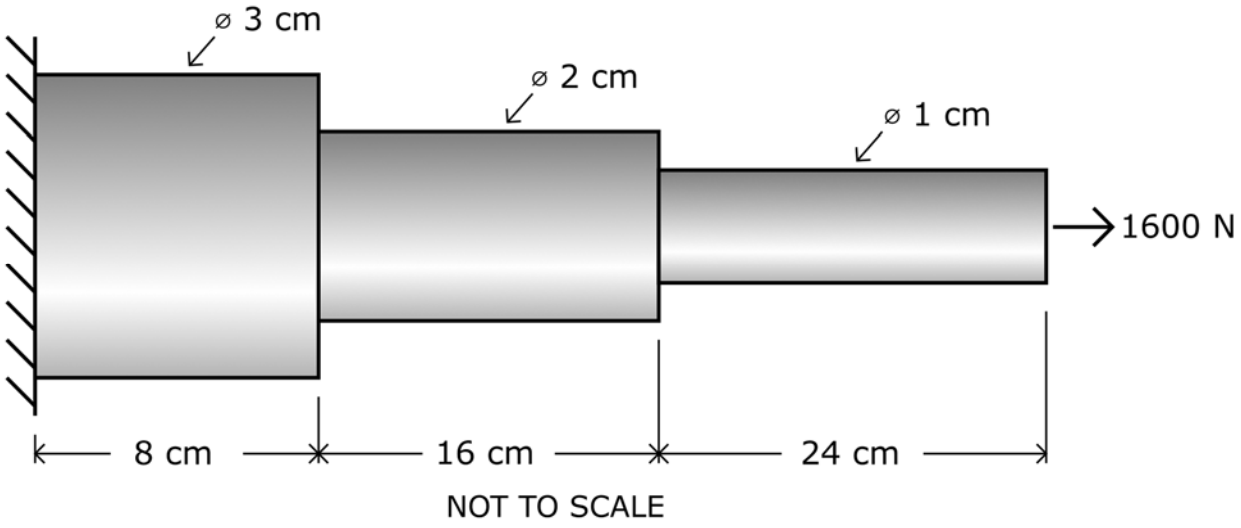
1) Write out the element formulation for each element using the notation and techniques described in the lectures.
2) Assemble the global system of equations manually, taking care to follow the numbering scheme dictated in each problem.
3) Use MATLAB to solve for the global system of equations. You may use any method available in MATLAB in this step, but this method must be reported.
4) Report the force in each element and the deflection of each node.

The Due date for this project is **December 2nd at 11:59 pm**. The project will be completed on an individual basis. Completed projects will be submitted to Waterloo LEARN, where a DropBox will be made available. Your project submission will consist of the following two items:

1) A short write-up (maximum five pages) that includes the element formulation, assembly of the global system of equations, and a summary and discussion of the results that addresses what is asked in the problem statement. Handwritten submissions are acceptable but must be clear and legible. A formal title page and other preamble (table of contents, list of figures, etc.) are not required.
2) The MATLAB script(s) used to solve the global system of equations. Upload one script for each problem. Do not upload more (or less) than two scripts. Include comments in your MATLAB script(s) to clarify your approach, variable selection, and methodology for the graders.

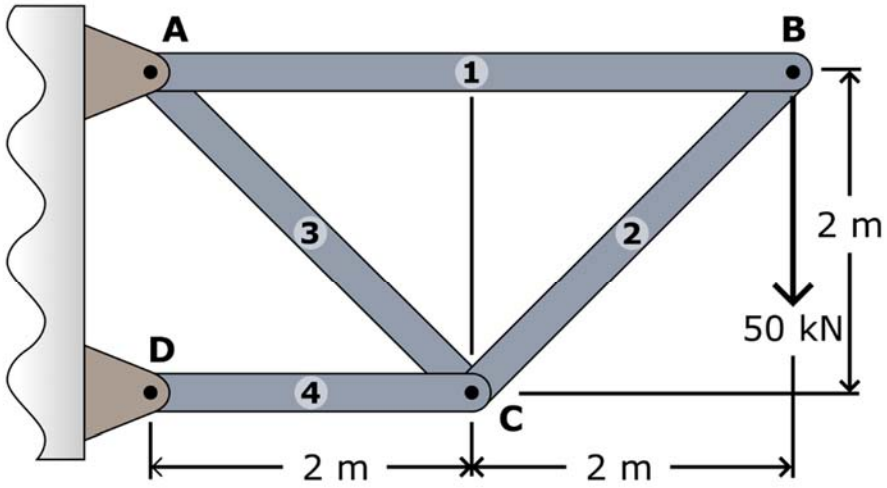
Problem #1: Stepped Shaft under Axial Loading

The following stepped shaft is loaded axially at its tip. The shaft is turned from 6061-T6 aluminum ($E = 69$ GPa, $\rho = 2700$ kg/m^3). Using FEA, determine the displacement at the tip and each step of the shaft measured from the fixed boundary condition. Additionally, report the engineering stress and strain in each stepped section of the shaft. Summarize the results in a table. Number the nodes and elements such that they begin at 1 and are increasing from the fixed boundary condition toward the free end, when assembling the global system of equations. This problem requires three elements. Confirm your FEA result with a hand calculation.



Problem #2: Deformation of a Truss Structure

The following truss structure is loaded vertically at joint $\boldsymbol{B}$. All members are fabricated from A36 steel ($E = 200$ GPa, $\rho = 7850$ kg/m^3, $\sigma_y = 250$ MPa) and have a cross-sectional area of 400 mm^2. Using FEA, report the following (in a table): the interior force in each element (with appropriate sign to indicate tension or compression), the change in length of each element and the engineering stress and engineering strain in each element. Additionally, calculate the horizontal and vertical displacements at each pin joint. Determine if any component of the structure will exceed the yield strength of the material. Calculate the factor of safety for each element. Propose a design change for any elements that are unsafe, or predicted to fail. Note that the mass of the truss elements can be considered negligible and should not be included in the calculations. Use the numbers provided in the schematic when numbering elements and assembling the global stiffness matrix.



NOT TO SCALE