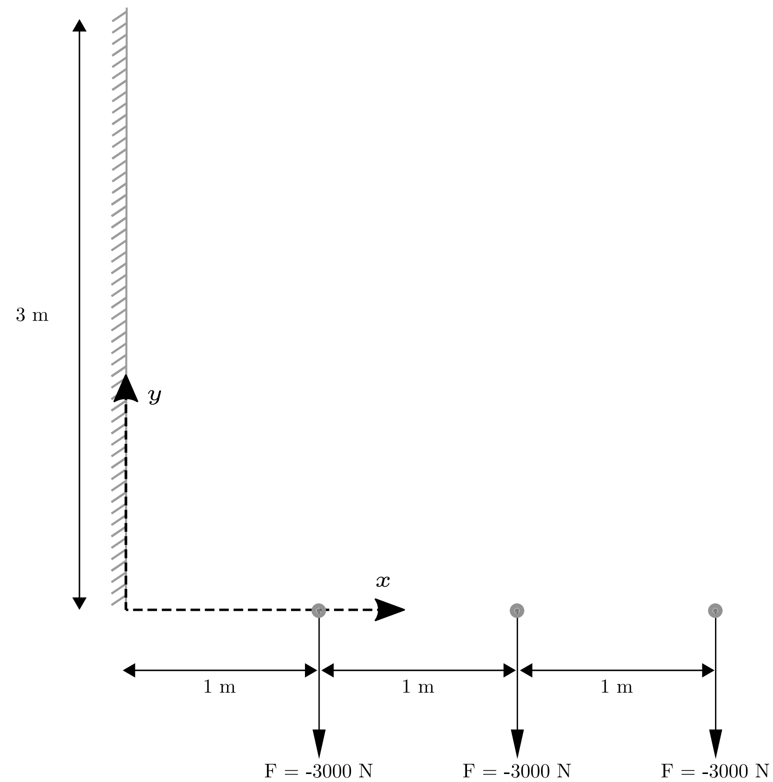
ESSE 4370 – Course Project

Due: April 1, 2021

Design a structure to support the loads depicted in the figure below where the loads must be suspended from above (all members must have undeformed nodal coordinates $0 \leq y_i \leq 3$ m)



A choice of material is possible, the structure can be constructed out of:

i) Steel with E = 200 GPa, yield strength σ_y = 250 MPa, and density ρ = 7850 kg/m^3

ii) Aluminum with E = 69 GPa, yield strength σ_y = 95 MPa, and density ρ = 2700 kg/m^3

The same modulus of elasticity $E^{(e)}$ and cross-sectional area $A^{(e)}$ must be used for all elements. Marks will be given for the most mass-efficient solution capable of carrying the load where mass efficiency will be defined as the sum of the forces divided by the total mass of the structure used to support them.

<u>Part 1</u>

Use Abaqus to design your structure. In your report:

1) Provide a figure of your designed structure with nodes and elements clearly labeled.
2) State the material and cross-sectional area and geometry.
3) Compile a table giving:
    a. Node number and coordinates (hint: check the .inp file)
    b. Whether a boundary condition is present there (where a 0 indicates none and a 1 indicates the presence) in *x* and *y*

    c. Force at node in *x* and *y*
4) Compile a table by element that provides:
    a. Connected nodes.
    b. $L^{(e)}$, and $\theta^{(e)}$ for each element.
5) Present the resulting nodal displacements for your structure in tabular form.
6) Give the reaction forces at the relevant nodes.
7) Give the stress for each element in tabular form.
8) Plot the undeformed and deformed (with a scale factor) shapes of the structure with stress contour shown.
9) Describe your design process and the iterations involved (give the mass efficiency and plots of the structure as it evolved with stress values and commentary) along with an explanation of material choice and cross-sectional area/geometry chosen.

Part 2

Check your solution by writing a Matlab program that automates the solution. Focus on reusability, the code should solve any truss problem in 2D, not just this one. Provide the code snippet with comments on how it works and the output.

1) Input Section
    a. Create a matrix that stores the nodal coordinates (row is the node, column 1 is *x*, column 2 is *y*).
    b. Create individual vectors for each of the parameters $E^{(e)}$, $A^{(e)}$, $L^{(e)}$, and $\theta^{(e)}$ for all of the elements from Assignment 1.
    c. Create the B.C. vector.
    d. Create the global force vector.
    e. Create the element interconnectivity info matrix. This matrix should have as many rows as elements, with each row containing the nodes connected by that element.
2) Solution Section
    a. Write a *function*, that generates the element stiffness matrix in global coordinates given the element parameters.
    b. In the main program, write an assembly routine that assembles the global stiffness matrix using the information from the element interconnectivity information matrix. (Remember to consider how many degrees of freedom there are per node).
    c. Reduce the equations.
    d. Solve for the nodal displacements.
3) Post-Processing Section
    a. Solve for the reaction forces.
    b. Calculate the stress in each element.
    c. Plot the undeformed and deformed (with a scale factor) shapes of the structure.

Part 3

1) Comment on any differences between the Abaqus solution and your developed code.
2) Are there any parts of the final design that concern you?