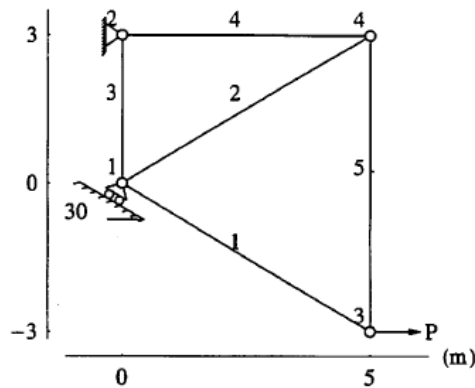


Due Tuesday, September 21st, 12:00 midnight

The first problem discusses a plane truss with inclined supports. You will need to modify the MatLab software from homework 1. The next 4 problems consider the analysis of beams as discussed in class. In problem 5, you will need to extend the provided beam element to account for linearly varying distributed load. In problem 6, you will need to modify the beam MatLab programs to analyze frames. Problems 4 and 7 refer to Ansys verification studies to be reviewed in the recitation.

Problem 1 - Analysis of trusses with inclined supports (MatLab)



The truss problems examined in earlier homework account for boundary displacement conditions posed directly in terms of the u, v, w displacements along the x, y, z axes. For a change, consider the five-bar truss shown above with an inclined roller support at node 1. All elements are made of the same material $E = 70 \text{ GPa}$ and $A = 10^{-3} \text{ m}^2$. The load is $P = 20 \text{ kN}$. Modify the finite element formulation examined in class (and programmed in the given MatLab files) to account for inclined supports. The modifications you can do can either be specific to this problem or can be general enough to accommodate many and different nodes with inclined support.

Hint: For those interested to program this for the general case, here is one approach using the penalty method. In some finite element modeling situations, it becomes necessary to introduce constraints between several different degrees of freedom. Such constraints are known as multipoint constraints and in general are expressed as follows:

$$c_{11}d_1 + c_{12}d_2 + \dots + c_{1n}d_n = q_1$$

$$c_{21}d_1 + c_{22}d_2 + \dots + c_{2n}d_n = q_2$$

$$\dots$$

where $c_{ij}, i, j = 1, 2, \dots$, and $q_i, i = 1, 2, \dots$, are specified constants and $d_i, i = 1, 2, \dots$, are the nodal degrees of freedom. In matrix form the constraints equations can be expressed as follows:

$$\mathbf{C}d = \mathbf{q}$$

where with m constraints \mathbf{C} is a $m \times n$ matrix and \mathbf{q} is a $m \times 1$ matrix.

We can then modify the FEM problem statement as follows:

Find \mathbf{d} such that

$$\text{Minimize } \phi = \frac{1}{2} \mathbf{d}^T \mathbf{K} \mathbf{d} - \mathbf{d}^T \mathbf{R} + \frac{1}{2} \mu (\mathbf{C} \mathbf{d} - \mathbf{q})^T (\mathbf{C} \mathbf{d} - \mathbf{q})$$

Subject to $\mathbf{C} \mathbf{d} - \mathbf{q} = 0$

With μ (the penalty parameter) being large, the minimization process forces the constraints to be satisfied. The necessary conditions for the minimum results in the following system of equations:

$$\frac{\partial \phi}{\partial \mathbf{d}} = 0 \Rightarrow \mathbf{K} \mathbf{d} - \mathbf{R} + \mu (\mathbf{C}^T \mathbf{C} \mathbf{d} - \mathbf{C}^T \mathbf{q}) = 0$$

Rearranging terms, the system of linear equations can be expressed as follows:

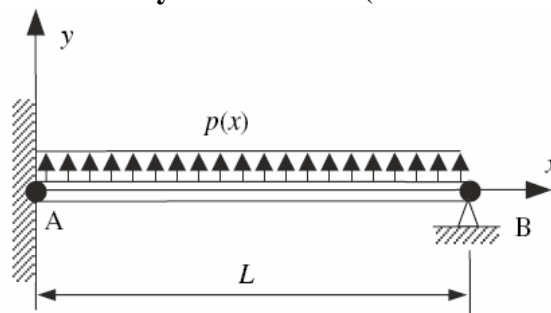
$$(\mathbf{K} + \mu \mathbf{C}^T \mathbf{C}) \mathbf{d} = \mathbf{R} + \mu \mathbf{C}^T \mathbf{q}$$

The performance of the method depends on the value chosen for the penalty parameter μ . Large values, say of the order of $\mu = 10^{10}$, give accurate solutions; however, the resulting system of equations may be ill-conditioned. If μ values are small as compared to other terms in the global equations, the solution will not satisfy the constraints very accurately. A general rule of thumb is to set μ equal to 10^5 times the largest number in the global K matrix.

So to program this you need to do two things: Introduce in your problem data the constraints in the matrix form $\mathbf{C} \mathbf{d} = \mathbf{q}$ and then modify the stiffness and load vectors as above!

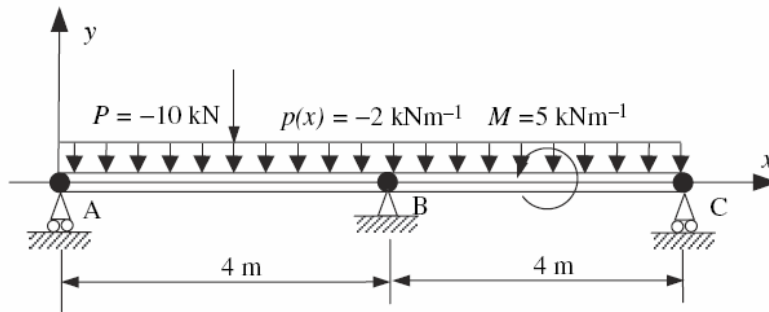
This problem looks difficult but the required solution is much shorter than the hint provided here!

Problem 2 - Analysis of a uniformly loaded beam (hand calculation)



Consider a beam AB subjected to uniform transverse loading as shown in the figure. Using a single finite element, calculate the maximum deflection by hand calculations.

Problem 3 – Analysis of a two-span beam (MatLab)



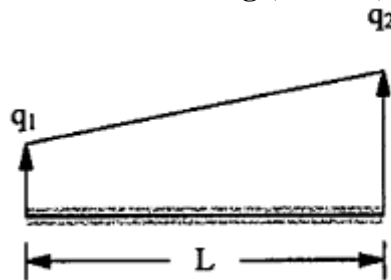
Consider a two-span beam shown in the Figure. The beam is subjected to uniformly distributed loading, point force at $x = 2 \text{ m}$ and point moment at $x = 6 \text{ m}$ as shown in above Figure. The beam bending stiffness is $EI = 2 \times 10^7 \text{ N m}^2$.

Using the finite element program provided, plot the deflection, bending moment, and shear force distribution of the beam. If you have four elements, what is the optimal mesh? Repeat the solution with the eight-element mesh, four for each span. Comment on the results. Is your solution right? How can you improve the finite element solution?

Problem 4 – Analysis of a two-span beam (Ansys)

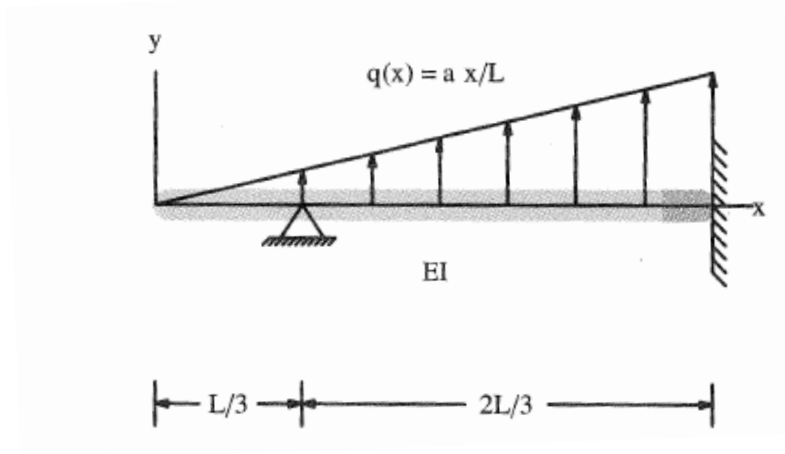
Repeat problem 3 using Ansys. Compare your answers with those computed via the MatLab programs. Provide a complete list of the command sequence used to solve this problem.

Problem 5 – Beams with non-uniform loading (MatLab)



Consider a beam finite element with trapezoidal loading as shown below. Derive the equivalent nodal forces for this element.

With this nodal force, modify the MATLAB codes to solve the following problem for a uniform beam subjected to a linearly increasing load. Plot the deflection, bending moment, and shear force distribution of the beam. The modifications should be general enough to accommodate many elements and put one element node at the support.



Assuming the following numerical data:

$$a = 5 \text{ kN/m}, \quad L = 5 \text{ m}, \quad E = 200 \text{ GPa}, \quad I = 10^6 \text{ mm}^4$$

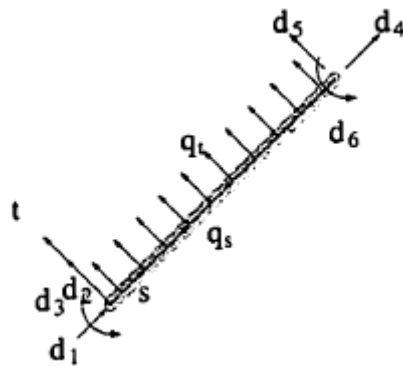
Problem 6 – Analysis of plane frames (MatLab)

This problem develops elements that combine (in an uncoupled way) the axial deformation of the truss 2-node elements from HW1 and the 2-node beam elements discussed in class. Members in a plane frame are designed to resist axial and bending deformations.

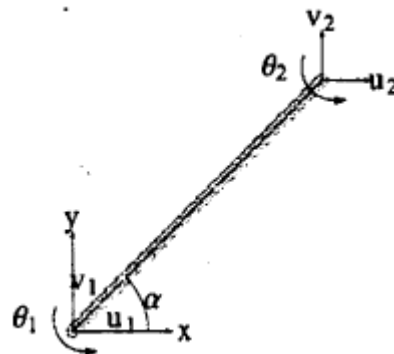
In the figures below, q_s refers to distributed axial load (load/length). Even though we did not discuss this type of loading in class, the equivalent nodal (FEM-consistent) forces (and also what you expect) are $q_s L/2$ (for constant q_s). The derivation is analogous to the one we did in class for uniform lateral loading in beam elements.

- a) Using the known results from 2-node truss (axial-deformation) and beam elements, derive the response of a plane frame element $[K^e]\{u^e\}=\{F^e\}$ in local and global coordinates (see Figures below). Modify the MatLab software provided for the analysis of beams to analyze plane frames.

Local coordinates



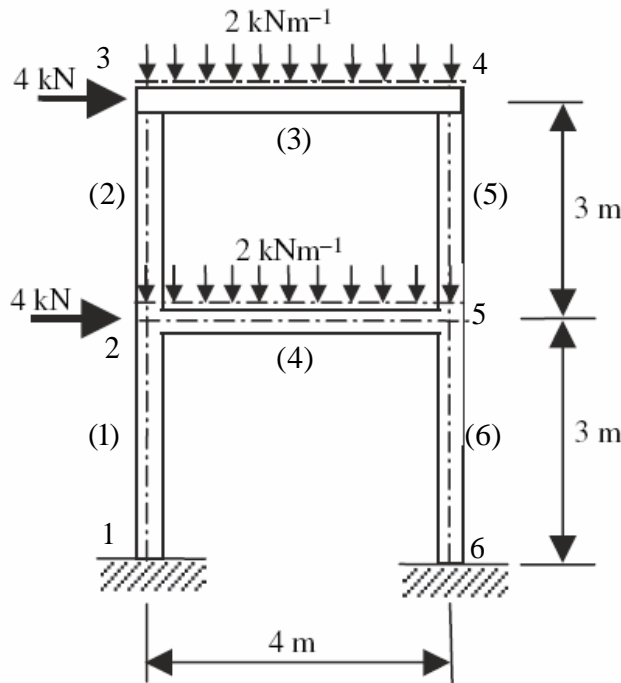
Global coordinates



Hint: There are three degrees of freedom per node, u , v and θ . The rotation matrix from global to local coordinate system is given by:

$$\begin{pmatrix} d_1 \\ d_2 \\ d_3 \\ d_4 \\ d_5 \\ d_6 \end{pmatrix} = \begin{pmatrix} \cos \alpha & \sin \alpha & 0 & 0 & 0 & 0 \\ -\sin \alpha & \cos \alpha & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos \alpha & \sin \alpha & 0 \\ 0 & 0 & 0 & -\sin \alpha & \cos \alpha & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{pmatrix} \begin{pmatrix} u_1 \\ v_1 \\ \theta_1 \\ u_2 \\ v_2 \\ \theta_2 \end{pmatrix}$$

- b) For the two-story frame shown below, use six 2-noded elements to compute the moments, shear forces, horizontal and vertical deflections. The bending stiffness for all beams and columns is $EI = 2.5 \times 10^7 \text{ N m}^2$ and $EA = 2.5 \times 10^9 \text{ N}$. For each element, report the computed axial forces, moments and shear forces at element ends. Choose the most left bottom node as the origin. The 4kN loads shown are concentrated forces. In this problem $q_s=0$.



Problem 7 – Analysis of plane frames (Ansys)

Repeat the solution of problem 6 using Ansys and provide a comparison with the MatLab results. Your solution write up should include all Ansys commands, element type, etc.