Instructor:
Prof. Dr. Dimitrios Lignos

Assignment #6: Nonlinear beam-column elements

Problem 1

The single-story steel moment-resisting frame (MRF) shown in Figure 1 has the same geometry with the one you analysed in Problem 2 of Assignment #5. The steel beam and connection characteristics are also the same. However, the steel columns can experience inelastic deformations. Particularly, columns comprise HEA450 shape steel cross sections, which are made of S355 steel (f_y=355MPa, strain hardening ratio, a=0.03). This can be approximated with a simple bilinear material model (Steel01 in OpenSees).

Use the OpenSees Navigator and calculate the following:

- The load at which strength degradation of the steel beam starts.
- Calculate the horizontal roof displacement at which the storey shear strength of the steel beam becomes zero.
- Calculate the horizontal roof displacement at which the steel columns yield.
- Qualitatively explain what are the main differences in this case compared to what you did in Problem 2 of HW#5?

Assumptions

- Model the steel beams (and springs) exactly the same way you did in HW#5.
- Model the steel columns as displacement-based beam-column elements with a fiber discretization of your choice. Justify your choice.
- Ignore the effects of gravity (P-Delta effects); therefore, use a Linear geometric transformation in your beam elements.
- Use the same displacement-controlled integrator up to **600mm** roof displacement, if possible, with an increment of 1mm (optional parameters: minimum 0.1mm and maximum 1mm). The reason is that you are dealing with softening in this problem.

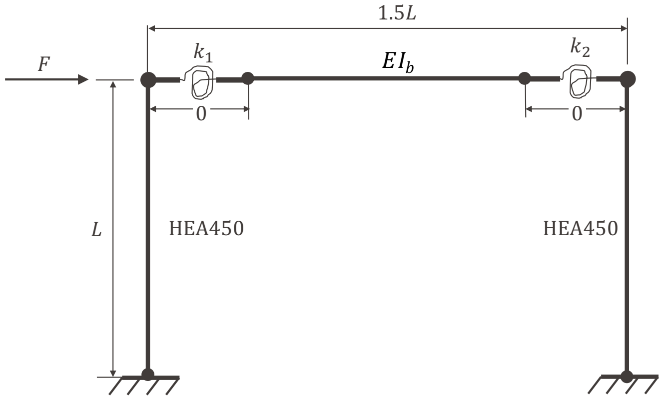
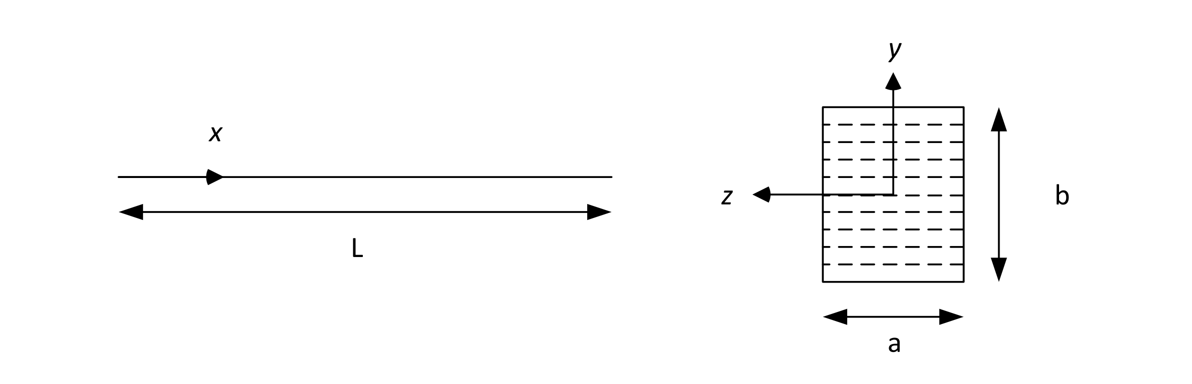


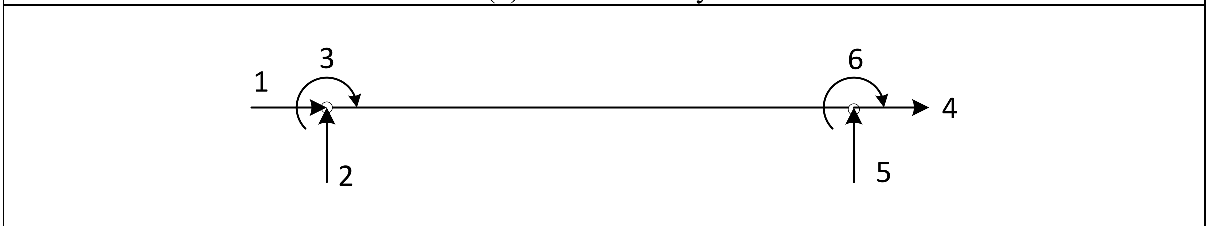
Figure 1. Single story steel moment-resting frame system

Problem 2

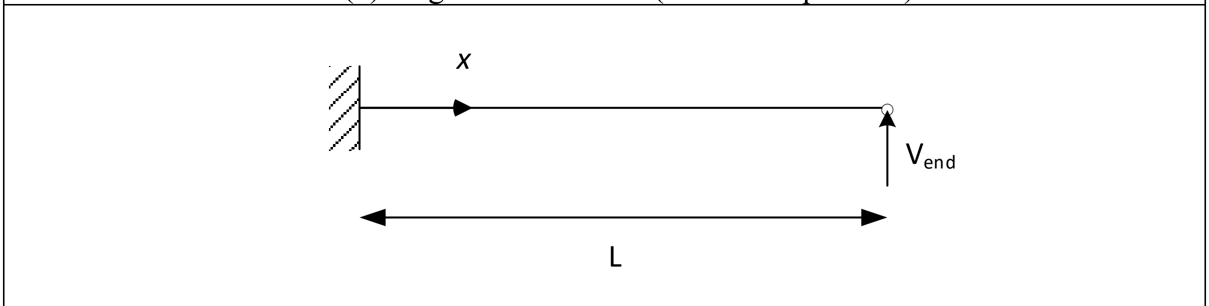
You are asked to analyze a cantilever beam-column subjected to a tip force. The problem is broken into steps to assist you with the formulation. Solve this problem by using MATLAB, Python or equivalent. Include your answers where indicated by "**Provide**" in your submission file, along with any supporting information that you think is necessary (e.g., code).



(a) Coordinate system



(b) Degrees of freedom (defined as positive)



(c) Cantilever model

Figure 2: Beam-column element definitions and cantilever model

Step 1 – Implement and validate a displacement-based fiber beam-column element

The coordinate system and degrees-of-freedom (DOF) of the beam-column element are shown in Figure 2. The formulation is based on the course material of Weeks 8 and 9. Assume the following:

- $a = 200mm, b = 200mm, L = 2000mm$ (uniform cross-section)
- $E = 200000MPa.$ (elastic material)
- Discretize the cross-section into n_f number of fibers similarly to the dashed lines in Figure 1a. The coordinates of the centroid for fiber $k = 1,2,\ldots,n_f$ are defined as follows:

$$y_k = -\frac{b}{2} + \frac{b}{2n_f} + \frac{b}{n_f} \cdot (k-1), \text{ and } z_k = 0, \tag{1}$$

- Use Gauss-Lobatto quadrature with 5 integration points along x.

- Use the following equations in your formulation (based on Week 8):

$$q = [q_1, q_2, q_3, q_4, q_5, q_6]^T; Q = [Q_1, Q_2, Q_3, Q_4, Q_5, Q_6]^T$$
$$l_k = [1, y_k]$$
$$d^s(x) = \begin{bmatrix} \varepsilon(x) \\ \varphi_z(x) \end{bmatrix} = B(x) \cdot q$$
$$f^s(x) = \begin{bmatrix} P(x) \\ M_z(x) \end{bmatrix} = \sum_{k=1}^{n_f} l_k^T * (\sigma_k(x) A_k(x))$$
$$k^s(x) = \sum_{k=1}^{n_f} l_k^T l_k * (E_k(x) A_k(x))$$
$$B(x) = \begin{bmatrix} \psi_1'(x) & 0 & 0 & \psi_2'(x) & 0 & 0 \\ 0 & \phi_1''(x) & \phi_2''(x) & 0 & \phi_3''(x) & \phi_4''(x) \end{bmatrix}$$
$$K^e = \int_0^L B(x)^T k^s(x) B(x)\ dx$$
$$Q = \int_0^L B(x)^T f^s(x)\, dx$$

Step 1 Objectives:

1) Write a function to compute the stiffness matrix, K^e, and element resisting force vector, Q. For now, assume that the stress is $1.0MPa$ in all fibers.
2) **Provide** the lowest value of n_f such that the error in your stiffness matrix with respect to the classic beam-column stiffness matrix (see Slide 19 of Week 6) is less than 2%. Compute the error using matrix norms, e.g., using the "norm(X)" function in MATLAB or "numpy.linalg.norm(X, 2)" in Python as follows:

$$Error(n_f) = \frac{\|K^e(n_f) - K_{classic}\|}{\|K_{classic}\|} \cdot 100 = \frac{norm(K^e(n_f) - K_{classic})}{norm(K_{classic})} \cdot 100 \qquad (2)$$

3) Verify that $Q_1 = -a * b$, $Q_4 = a * b$, and the remaining entries are zero when $\sigma_k = 1.0$ for all the fibers.

Step 2 – Conduct a load control analysis with your beam-column element

You will implement a load control method with Newton-Raphson iterations to solve the equilibrium equations (see Weeks 2 and 3 by Professor Beyer). The fixed and free DOFs are $[q_1, q_2, q_3]$, and $[q_4, q_5, q_6]$, respectively. Create a cantilever model using your displacement-based fiber beam-column element by fixing DOFs q_1, q_2, q_3. Assume the following:

- Use the n_f you calculated in Step 1 for the rest of this assignment,
- The initial displacement at step 0 is $q^0 = [0,0,0,0,0,0]^T$,
- The external load vector is $Q_{ext} = [0,0,0,0, V_{end}, 0]^T$ (tip load only),
- Assume that $V_{end} = 450\,000$. The load vector is to be applied using the force-control method with a force convergence criterion,
- Apply the load in 30 increments, i.e., $Q_{ext}^i = \frac{i}{30} * Q_{ext}$ per increment. The response will be computed using Newton-Raphson iterations at each increment.

Step 2 Objectives:

1) Set-up the cantilever model and apply the load Q_{ext} in 30 increments. Solve each increment using Newton-Raphson iterations. Record the nodal force and displacements at the end of each increment.
2) **Provide** a plot of Q_5 vs q_5 at each increment (there should be 30 points total).

3) **Provide** a check that your force-displacement stiffness agrees with the expected value based on mechanics for a cantilever ($K = 3EI/L^3$).

Use the following algorithm for the Newton-Raphson method to solve the applied load at increment *i*:

Input: q^{i-1}, the nodal displacements at the start of the increment, and Q^i_{ext}, the applied load for the increment.
Initialize:

$q = q^{i-1}$
$Compute\ the\ fiber\ stresses\ and\ tangent\ moduli\ using\ d^s(x) = B(x) \cdot q$
$Compute\ K^e and\ Q\ using\ your\ fiber\ element\ formulation$
$R = Q^i_{ext} - Q$
$tolerance = 1 \times 10^{-8}$
$maximum\ iterations = 16$
$j = 0$

Iterate until convergence is satisfied:

$j = j + 1$
$K_{ff} = K^e(free\ DOF, free\ DOF);\ R_f = R(free\ DOF)$
$Solve: K_{ff}\Delta q_f = R_f\ for\ \Delta q_f$
$q(free\ DOF) = q(free\ DOF) + \Delta q_f$
$Recompute\ fiber\ stresses\ and\ tangent\ moduli\ using\ updated\ q$
$Recompute\ K^e, Q\ with\ the\ updated\ stresses\ and\ moduli$
$R = Q^i_{ext} - Q$
$Exit\ if\ \frac{\|R\|}{\|Q^i_{ext}\|} < tolerance\ OR\ j > maximum\ iterations$
$Else\ continue\ to\ next\ iteration$

Return the final q and Q vectors.

The step "$Solve: K_{ff}\Delta q_f = R_f\ for\ \Delta q_f$" can be done using q_f = K_ff / R_f in MATLAB or numpy.linalg.solve(K_ff, R_f) in Python. The notation "$K_{ff} = K^e(free\ DOF, free\ DOF)$" indicates that you are selecting the free DOF only from K^e.

Step 3 – Plasticity: Add an elastic-perfectly-plastic material
Add material nonlinearity to your displacement-based beam-column element. The elastic-perfectly-plastic material law for monotonic loading is as follows:

$$\sigma = \begin{cases} E \cdot \varepsilon & abs(\varepsilon) < \varepsilon_y \\ sign(\varepsilon) \cdot \sigma_y & otherwise \end{cases} \quad (3)$$

Assume $\sigma_y = 355MPa$; therefore, $\varepsilon_y = \sigma_y/E$.
The strain in each fiber is computed using $\varepsilon_k = \varepsilon(x) + \varphi_z(x)y_k$ for the coordinate system provided. Then use the constitutive law to compute the stresses and tangent moduli.

Step 3 Objectives:

1) Write a function that returns the stress and tangent modulus for a given strain following the constitutive law provided, i.e., $[\sigma, E^t] = f(\varepsilon)$.

2) Use the above function to compute the stress and tangent modulus for each fiber in the cross-section at each integration point. Add this functionality to your Newton-Raphson method.
3) **Provide** the moment-rotation plot (q_5, Q_5) at each increment for the same loading as applied in Step 2, but now considering the elastic-perfectly-plastic material model.
4) Find approximately the tip load where the first fiber begins to yield, V_{end}^{y}.
5) **Provide** a comparison of the support moment at first yield, $Q_y = V_{end}^{y} * L$, with $M_y = \sigma_y * W_{el}$ (W_{el} is the elastic section modulus) for the cross-section. Comment on the results.

Bonus Step 1 – Consider varying cross-sectional area

This step is **not** necessary for full marks but may be interesting for you personally. Modify the area at each integration point by modifying the width in the z-axis: $a(x) = a\left(1 - 0.5 \cdot \frac{x}{L}\right)$. Note that the dimension b does not change.

1) Recompute the cantilever response in Step 3 using the varying cross-section.
2) **Provide** a plot comparing (q_5, Q_5) with a constant cross-section (from Step 3) with the response with a varying cross-section.

Bonus Step 2 – Use 3pt Gauss-Lobatto integration

Implement the 3pt Gauss-Lobatto integration (instead of the 5pt). Keep the cross-section constant in this step.

3) Recompute the cantilever response in Step 3 using the 3pt integration rule.
4) **Provide** a plot comparing (q_5, Q_5) with the 5pt integration rule (from Step 3) with the response with the 3pt integration rule. Comment on the results.