MTE 204 Project 2

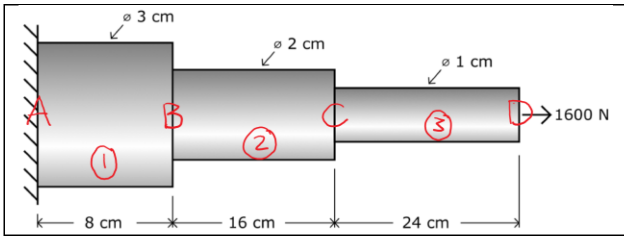
Isaac Zhang

Stepped Shaft

The stiffness matrix for an element in the global coordinate system for a 2D FEM problem is as follows:

$$[K] = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

where E is Young's elastic modulus, A is the cross-sectional area of the element, and L its length. Then, consider the setup and constants obtained from the project outline:



Element	Length L [m]	Area A [m^2]
1	0.08	0.0016π
2	0.16	0.0064π
3	0.24	0.0144π

Young's elastic modulus $E = 69$ GPa

Then, the stiffness matrices for individual elements in the global coordinate system are as follows, in ascending order from the 1st element to the 3rd element, with its corresponding displacement vector:

$$[K]_1\begin{Bmatrix} u_A \\ u_B \end{Bmatrix} = \frac{EA_1}{L_1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_A \\ u_B \end{Bmatrix} = 6.0967 \times 10^8 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_A \\ u_B \end{Bmatrix}$$

$$[K]_2\begin{Bmatrix} u_B \\ u_C \end{Bmatrix} = \frac{EA_2}{L_2}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_B \\ u_C \end{Bmatrix} = 1.3548 \times 10^8 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_B \\ u_C \end{Bmatrix}$$

$$[K]_3\begin{Bmatrix} u_C \\ u_D \end{Bmatrix} = \frac{EA_3}{L_3}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_C \\ u_D \end{Bmatrix} = 2.2580 \times 10^7 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_C \\ u_D \end{Bmatrix}$$

Combining the above elemental stiffness matrices based on u and v components of each node's displacement vector yield the following expression for the global stiffness matrix:

$$\begin{Bmatrix} P_A \\ P_B \\ P_C \\ P_D \end{Bmatrix} = 10^6 \cdot \begin{bmatrix} 60.97 & -60.97 & 0 & 0 \\ -60.97 & 74.51 & -13.55 & 0 \\ 0 & -13.55 & 15.81 & -22.58 \\ 0 & 0 & -22.58 & 22.58 \end{bmatrix}\begin{Bmatrix} u_A \\ u_B \\ u_C \\ u_D \end{Bmatrix}$$

Recall that node A is the fixed end incapable of having displacements in the x or y direction, the boundary conditions for this stepped shaft. This means that the rows and columns associated with u_A, and v_A can be omitted from the prior expression, as these displacements are equal to zero. Substituting the value of the external applied force in for P_D, a new expression is obtained:

$$\begin{Bmatrix} P_B = 0 \\ P_C = 0 \\ P_D = 1600 \end{Bmatrix} = 10^6 \cdot \begin{bmatrix} 74.51 & -13.55 & 0 \\ -13.55 & 15.81 & -22.58 \\ 0 & -22.58 & 22.58 \end{bmatrix} \begin{Bmatrix} u_B \\ u_C \\ u_D \end{Bmatrix}$$

Using Matlab's *linsolve*, the following displacements are obtained, with node A being stationary:

Node	Displacement (mm)
2	-0.03151
3	-0.17327
4	-0.10241

Then, the stress and engineering strain for each element is as follows:

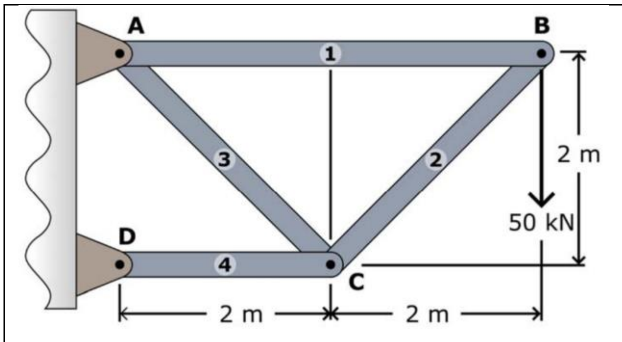
Element	1	2	3
Stress (MPa)	2.2635	5.093	20.3718
Strain	3.2805e-05	7.3811e-05	0.00029524

2D Truss

The stiffness matrix for an element in the global coordinate system for a 2D FEM problem is as follows:

$$[K] = \frac{EA}{L}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix}$$

where $c = \cos\theta$ and $s = \sin\theta$ and E is Young's elastic modulus, A is the cross-sectional area of the element, and L its length. Then, consider the setup and constants obtained from the project outline:



Element	Length L [m]	Angle θ [°]
1	4	0
2	$2\sqrt{2}$	45
3	$2\sqrt{2}$	135
4	2	0

Young's elastic modulus $E = 200$ GPa
Cross-sectional area $A = 4 \times 10^{-4}\ \mathrm{m}^2$
Yield Strength $\sigma_y = 250$ MPa

Then, the stiffness matrices for individual elements in the global coordinate system are as follows, in ascending order from the 1st element to the 4th element, with its corresponding displacement vector:

$$[K]_1\begin{Bmatrix} u_A \\ v_A \\ u_B \\ v_B \end{Bmatrix} = \frac{EA}{4}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix}\begin{Bmatrix} u_A \\ v_A \\ u_B \\ v_B \end{Bmatrix} = 2 \times 10^7\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}\begin{Bmatrix} u_A \\ v_A \\ u_B \\ v_B \end{Bmatrix}$$

$$[K]_2\begin{Bmatrix} u_B \\ v_B \\ u_C \\ v_C \end{Bmatrix} = \frac{EA}{2\sqrt{2}}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix}\begin{Bmatrix} u_B \\ v_B \\ u_C \\ v_C \end{Bmatrix} = 2.83 \times 10^7\begin{bmatrix} 0.5 & 0.5 & -0.5 & -0.5 \\ 0.5 & 0.5 & -0.5 & -0.5 \\ -0.5 & -0.5 & 0.5 & 0.5 \\ -0.5 & -0.5 & 0.5 & 0.5 \end{bmatrix}\begin{Bmatrix} u_B \\ v_B \\ u_C \\ v_C \end{Bmatrix}$$

$$[K]_3\begin{Bmatrix} u_A \\ v_A \\ u_C \\ v_C \end{Bmatrix} = \frac{EA}{2\sqrt{2}}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix}\begin{Bmatrix} u_A \\ v_A \\ u_C \\ v_C \end{Bmatrix} = 2.83 \times 10^7\begin{bmatrix} 0.5 & -0.5 & -0.5 & 0.5 \\ -0.5 & 0.5 & 0.5 & -0.5 \\ -0.5 & 0.5 & 0.5 & -0.5 \\ 0.5 & -0.5 & -0.5 & 0.5 \end{bmatrix}\begin{Bmatrix} u_A \\ v_A \\ u_C \\ v_C \end{Bmatrix}$$

$$[K]_4\begin{Bmatrix} u_C \\ v_C \\ u_D \\ v_D \end{Bmatrix} = \frac{EA}{2}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix}\begin{Bmatrix} u_C \\ v_C \\ u_D \\ v_D \end{Bmatrix} = 4 \times 10^7\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}\begin{Bmatrix} u_C \\ v_C \\ u_D \\ v_D \end{Bmatrix}$$

Combining the above elemental stiffness matrices based on u and v components of each node's displacement vector yield the following expression for the global stiffness matrix, with external:

$$\begin{Bmatrix} P_{A,x} \\ P_{A,y} \\ P_{B,x} \\ P_{B,y} \\ P_{C,x} \\ P_{C,y} \\ P_{D,x} \\ P_{D,y} \end{Bmatrix} = 10^6 \cdot \begin{bmatrix} 34.14 & -14.14 & -20.00 & 0 & -14.14 & 14.14 & 0 & 0 \\ -14.14 & 14.14 & 0 & 0 & 14.14 & -14.14 & 0 & 0 \\ -20.00 & 0 & 34.14 & 14.14 & -14.14 & -14.14 & 0 & 0 \\ 0 & 0 & 14.14 & 14.14 & -14.14 & -14.14 & 0 & 0 \\ -14.14 & 14.14 & -14.14 & -14.14 & 68.28 & 0 & -40.00 & 0 \\ 14.14 & -14.14 & -14.14 & -14.14 & 0 & 28.28 & 0 & 0 \\ 0 & 0 & 0 & 0 & -40.00 & 0 & 40.00 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_A \\ v_A \\ u_B \\ v_B \\ u_C \\ v_C \\ u_D \\ v_D \end{Bmatrix}$$

Recall that nodes A and D are pinned joints incapable of having displacements in the x or y direction, the boundary conditions for this truss. This means that the rows and columns associated with u_A, v_A, u_D, and v_D can be omitted from the prior expression, as these displacements are equal to zero. Substituting the value of the external applied force in for $P_{B,y}$, a new expression is obtained:

$$\begin{Bmatrix} P_{B,x} = 0 \\ P_{B,y} = -50000 \\ P_{C,x} = 0 \\ P_{C,y} = 0 \end{Bmatrix} = 10^6 \cdot \begin{bmatrix} 34.14 & 14.14 & -14.14 & -14.14 \\ 14.14 & 14.14 & -14.14 & -14.14 \\ -14.14 & -14.14 & 68.28 & 0 \\ -14.14 & -14.14 & 0 & 28.28 \end{bmatrix} \begin{Bmatrix} u_B \\ v_B \\ u_C \\ v_C \end{Bmatrix}$$

Using Matlab's *linsolve*, the following displacements are obtained, with nodes A, D being stationary:

Node	Direction	Displacement (mm)
B	Horizontal	2.5
	Vertical	-14.5721
C	Horizontal	-2.5
	Vertical	-6.0361

Then, the stress, internal force, change in length, and engineering strain for each element is as follows:

Element	1	2	3	4
Stress (MPa)	125	-176.8034	176.8034	-250
Internal Force (kN)	50	-70.7214	70.7214	-100
Change in Length (mm)	2.5	-2.5004	2.5004	-2.5
Engineering Strain	0.000625	-0.00088402	0.00088402	-0.00125
Factor of Safety	0.5	0.70721	0.70721	1

The above factor of safety is calculated from the elements stress and the material's yield strength of $\sigma_y = 250$ MPa. Evidently, the 4th element in the original setup is at the threshold for yielding. One method to address this is with a second 4th element, effectively doubling its cross-sectional area A.