ED5015 Mini Project 1

Sujay S ed20b065

21 Mar 2023

1 Report

The problem was solved using the direct stiffness method to compute the nodal displacements and element stresses. A generalised code that would solve for any 3D truss structure was developed using the principles of direct stiffness method in FEM. Following data was used for solving the given problem:

E	**A**	**F**
3×10^7 psi	$\frac{\pi D^2}{4} = \frac{\pi(2^2)}{4}$ $= 3.14159$ sq.inch	60000 lb

The code can be broken down into these segments:

1.1 Importing libraries:

The required libraries *(numpy, numpy, matplotlib.pyplot)* were imported

1.2 Getting inputs:

Number of elements, Number of nodes, Number of fixed nodes, Fixed nodes, Starting and ending node for each element, Area of each element, Modulus of Elasticity of each element, Coordinates of each node are accepted from user, and are stored accordingly for further computations. The coordinates are converted from feet to inches in the given problem.
In the given problem,
DOF: 3
No of elements: 25
Number of nodes: 10
Number of fixed nodes: 4

1.3 Transformation of stiffness matrix:

The transformed stiffness matrix for each element (in global coordinates) is computed using transformation matrix (Direction ratios) as follows:
For each element i,
$L_i = \sqrt{(x_{2i} - x_{1i})^2 + (y_{2i} - y_{1i})^2 + (z_{2i} - z_{1i})^2}$

$\cos\theta_x = \frac{x_{2i} - x_{1i}}{L_i} = c_{xi}$

$\cos\theta_y = \frac{y_{2i} - y_{1i}}{L_i} = c_{yi}$

$\cos\theta_z = \frac{z_{2i} - z_{1i}}{L_i} = c_{zi}$

$$K_{global i} = \frac{E_i A_i}{L_i} \begin{bmatrix} c_{xi}^2 & c_{xi}c_{yi} & c_{xi}c_{zi} & -c_{xi}^2 & -c_{xi}c_{yi} & -c_{xi}c_{zi} \\ c_{yi}c_{xi} & c_{yi}^2 & c_{yi}c_{zi} & -c_{yi}c_{xi} & -c_{yi}^2 & -c_{yi}c_{zi} \\ c_{zi}c_{xi} & c_{zi}c_{yi} & c_{zi}^2 & -c_{zi}c_{xi} & -c_{zi}c_{yi} & -c_{zi}^2 \\ -c_{xi}^2 & -c_{xi}c_{yi} & -c_{xi}c_{zi} & c_{xi}^2 & c_{xi}c_{yi} & c_{xi}c_{zi} \\ -c_{yi}c_{xi} & -c_{yi}^2 & -c_{yi}c_{zi} & c_{yi}c_{xi} & c_{yi}^2 & c_{yi}c_{zi} \\ -c_{zi}c_{xi} & -c_{zi}c_{yi} & -c_{zi}^2 & c_{zi}c_{xi} & c_{zi}c_{yi} & c_{zi}^2 \end{bmatrix}$$

1.4 Computing Global stiffness matrix (Assembling):

The global stiffness matrix is computed by assembling the individual stiffness matrices of each elements.
The size of the Global stiffness matrix = DOF*(No of nodes) = 3*10 = 30

1.5 Boundary Conditions:

The boundary conditions (fixed nodes, nodal loads) are applied using the inputs from user. The fixed nodes' entries are made zeros and stored in a new variable.
In the given problem,
$\mathbf{F_y}$: 60000lb for i= 1,2
Fixed nodes: 7,8,9,10

1.6 Solving for nodal displacements:

The fixed nodes entries (all zeros) are removed to solve for the other nodal displacements using *numpy's linalg.solve.* These are then stored in the displacement variable for later analysis.

1.7 Computing elemental quantities:

The elemental quantities (in local coordinates)- *axial displacements, elemental forces and stresses* are computed from nodal displacements.
For each element i,
Axial displacements: $\mathbf{u_i} = \mathbf{T_i}\mathbf{v_i} = \begin{bmatrix} u1 \\ u2 \end{bmatrix} = \begin{bmatrix} c_{xi} & c_{yi} & c_{zi} & 0 & 0 & 0 \\ 0 & 0 & 0 & c_{xi} & c_{yi} & c_{zi} \end{bmatrix} \mathbf{v_i}$
where $\mathbf{u_i}$ is the axial displacement vector, $\mathbf{v_i}$ is the displacement vector for element i in global coordinates, $\mathbf{T}$ is the transformation matrix. This can also be considered as the projection of displacement vector to the axial direction.
Axial forces: $\mathbf{F_{axial,i}} = \mathbf{k_i}\mathbf{u_i} = \frac{E_i A_i}{L_i} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \mathbf{u_i}$
Element stresses: $\sigma_{\mathbf{i}} = \frac{F_{axial,i}}{A_i}$

1.8 Results:

The required results were printed and are as follows:

1.8.1 Nodal Displacements:

Node	x	y	z
Node 1	6.585544×10^{-18}	0.237493	1.789648×10^{-17}
Node 2	6.280375×10^{-18}	0.237493	7.173915×10^{-18}
Node 3	-1.729046×10^{-3}	0.015628	-5.067904×10^{-2}
Node 4	1.729046×10^{-3}	0.015628	-5.067904×10^{-2}
Node 5	-1.729046×10^{-3}	0.015628	5.067904×10^{-2}
Node 6	1.729046×10^{-3}	0.015628	5.067904×10^{-2}
Node 7	0	0	0
Node 8	0	0	0
Node 9	0	0	0
Node 10	0	0	0

Table 1: Nodal Displacements (in inches)

Node	x	y	z
Node 1	1.672728×10^{-16}	6.032328	4.545705×10^{-16}
Node 2	1.595215×10^{-16}	6.032328	1.822174×10^{-16}
Node 3	-4.391776×10^{-2}	0.396951	-1.287248×10^{0}
Node 4	4.391776×10^{-2}	0.396951	-1.287248×10^{0}
Node 5	-4.391776×10^{-2}	0.396951	1.287248×10^{0}
Node 6	4.391776×10^{-2}	0.396951	1.287248×10^{0}
Node 7	0	0	0
Node 8	0	0	0
Node 9	0	0	0
Node 10	0	0	0

Table 2: Nodal Displacements (in mm)

1.8.2 Element stress:

Element	Stress (in psi)	Stress (in MPa)
Element 1	-2.543077e-13	-1.753391e-15
Element 2	-1.145821e+04	-7.900163e+01
Element 3	-1.145821e+04	-7.900163e+01
Element 4	1.145821e+04	7.900163e+01
Element 5	1.145821e+04	7.900163e+01
Element 6	-1.781941e+04	-1.228605e+02
Element 7	1.781941e+04	1.228605e+02
Element 8	-1.781941e+04	-1.228605e+02
Element 9	1.781941e+04	1.228605e+02
Element 10	2.316011e-12	1.596834e-14
Element 11	0.000000e+00	0.000000e+00
Element 12	2.881743e+03	1.986893e+01
Element 13	-2.881743e+03	-1.986893e+01
Element 14	-5.765940e+03	-3.975478e+01
Element 15	5.765940e+03	3.975478e+01
Element 16	-5.765940e+03	-3.975478e+01
Element 17	5.765940e+03	3.975478e+01
Element 18	-1.106078e+04	-7.626144e+01
Element 19	-1.106078e+04	-7.626144e+01
Element 20	1.106078e+04	7.626144e+01
Element 21	1.106078e+04	7.626144e+01
Element 22	2.158849e+04	1.488475e+02
Element 23	-2.158849e+04	-1.488475e+02
Element 24	-2.158849e+04	-1.488475e+02
Element 25	2.158849e+04	1.488475e+02

Table 3: Stress values

1.8.3 Scaled deformed structure plot :

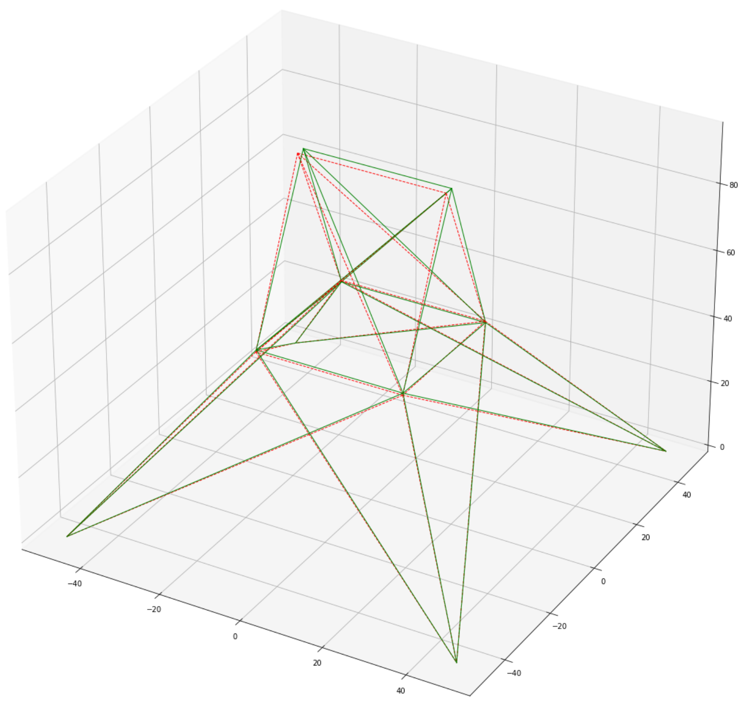

Figure 1: Scaled deformed structure plot (Scale = 10x)