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.
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: $\mathbf{u_i} = \mathbf{T_i v_i} = \begin{bmatrix} u1 \\ u2 \end{bmatrix} = \begin{bmatrix} c_{xi} & c_{yi} & c_{zi} & 0 & 0 & 0 \\ 0 & 0 & 0 & c_{xi} & c_{yi} & c_{zi} \end{bmatrix} \mathbf{v_i}$
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: $\mathbf{F_{axial,i}} = \mathbf{k_i u_i} = \frac{E_i A_i}{L_i} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \mathbf{u_i}$
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**
1	-0.000006	0.018524	0.000248
2	0.018524	0.000248	-0.000039
3	0.000248	-0.000039	0.018759
4	-0.000039	0.018759	0.000249
5	0.018759	0.000249	-0.000322
6	0.000249	-0.000322	0.001587
7	0.000000	0.000000	0.000000
8	0.000000	0.000000	0.000000
9	0.000000	0.000000	0.000000
10	0.000000	0.000000	0.000000

Table 1: Nodal Displacements (in inches)

Node	**x**	**y**	**z**
1	-0.000148	0.470499	0.006308
2	0.470499	0.006308	-0.001001
3	0.006308	-0.001001	0.476470
4	-0.001001	0.476470	0.006320
5	0.476470	0.006320	-0.008180
6	0.006320	-0.008180	0.040303
7	0.000000	0.000000	0.000000
8	0.000000	0.000000	0.000000
9	0.000000	0.000000	0.000000
10	0.000000	0.000000	0.000000

Table 2: Nodal Displacements (in mm)

1.8.2 Element stress:

Element	Stress (in psi)	Stress (in MPa)
Element 1	185293.860150	1277.556695
Element 2	275.266681	1.897898
Element 3	-22897.079154	-157.869865
Element 4	94664.749007	652.690725
Element 5	54137.997431	373.268499
Element 6	43746.819514	301.623821
Element 7	1857.373948	12.806148
Element 8	-167483.392912	-1154.757798
Element 9	37668.287170	259.713800
Element 10	2826.508549	19.488098
Element 11	185098.531523	1276.209951
Element 12	-2877.407415	-19.839034
Element 13	185098.531523	1276.209951
Element 14	43107.912878	297.218713
Element 15	4994.410000	34.435258
Element 16	59580.799060	410.795310
Element 17	-28316.181448	-195.233275
Element 18	-26353.194922	-181.698954
Element 19	42152.716385	290.632863
Element 20	58575.145210	403.861568
Element 21	2384.769377	16.442413
Element 22	6488.257501	44.734978
Element 23	79741.066917	549.795519
Element 24	-48213.605347	-332.421238
Element 25	-50066.125598	-345.193920

Table 3: Stress values

1.8.3 Scaled deformed structure plot :

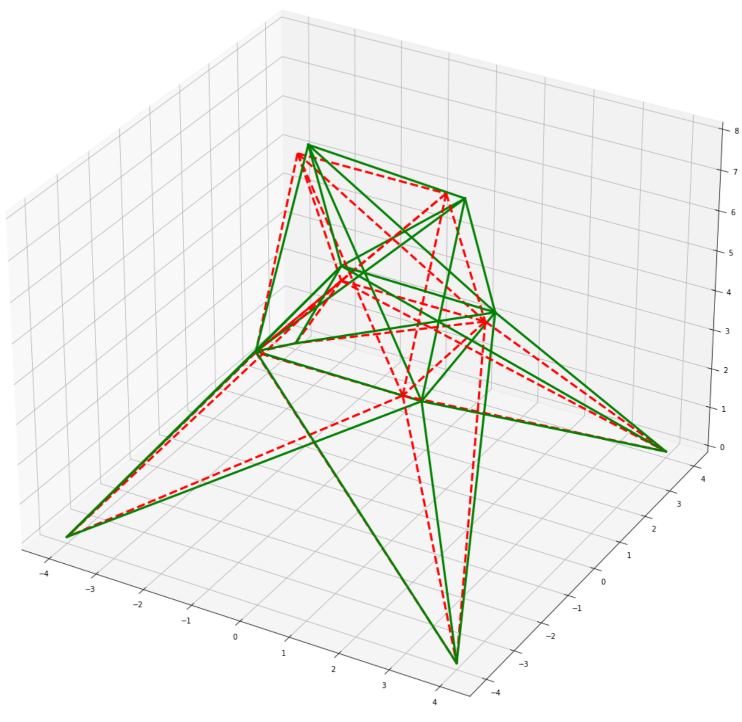

Figure 1: Scaled deformed structure plot (Scale = 20x)