

ANALYSIS OF LINEAR, ELASTIC- ISOTROPIC TRUSSES IN MATLAB USING FINITE ELEMENT METHOD

Lava Gurung
Department of Mechanical
Engineering
Kathmandu University
Dhulikhel, Nepal
lavagurung@gmail.com

Nishan Sapkota
Department of Mechanical
Engineering
Kathmandu University
Dhulikhel, Nepal
nishan205566@gmail.com

Vijan Bhandari
Department of Mechanical
Engineering
Kathmandu University
Dhulikhel, Nepal
mevijan35@gmail.com

Krishna Prashad Shrestha
Department of Mechanical
Engineering
Kathmandu University
Dhulikhel, Nepal
kp@ku.edu.np

Abstract— Engineering structures is an important aspect of human civilization and manual analysis of structure often deviates from actual value, so there is a need to use Finite Element Method technique for analyzing those structures. Thus, this paper presents FEM based codes developed in MATLAB programming which targets the analysis of 2D linear, elastic-isotropic structure. This program lists the results of displacement, stress-strain, reaction forces and plots the deformed shaped due to applied forces. Since the software is solitary only on truss analysis, it computes faster compare to other market software like ANSYS, COMSOL etc.

Keywords—linear structure, finite element method, MATLAB programming

I. INTRODUCTION

Considering various aspects of the complexities, the solutions which are baffling and sticking to the conventional method will eventually leads to trivial solution. Therefore, to make the calculation precise, numerical analysis is done. Among various forms of numerical analysis for solving problems of engineering and mathematical physics, Finite Element Method (FEM) is the most preferred one. Thus, it is very necessary to create a program-based on FEM, to meet today's digitalized world.

Thus, this developed software aims to able a common platform for all the linear elastic structure. Also due to of its simplicity, person having faint knowledge of structures and FEM can easily use this program.

Different Subject matter that required for completion of this project are.

A. Finite Element Method

Finite Element Method is a numerical method in which a complex domain defining a continuum is discretized into various simple geometric shape called finite elements. This process results in a set of simultaneous algebraic equations and the solution is obtained upon computing these equations. The material properties and governing equations considered over the finite elements are expressed in terms of unknown parameter at elemental nodes where different elements of the structure are connected [1]. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem [2]. Here, in this research, the principle of direct method of analysis was used. The basic principle used here is the sum of force acting on the element must be zero. If \mathbf{F}_1 and \mathbf{F}_2 are two force acting on the element, then;

$$\mathbf{F}_1 + \mathbf{F}_2 = \mathbf{0}$$

Also, the relation between force and displacement is given by: $\mathbf{F} = \mathbf{K} \cdot \mathbf{u}$, where, \mathbf{F} = Force, \mathbf{K} = stiffness, \mathbf{u} =displacement. So, if the two elements of similar material have displacement as \mathbf{u}_1 and \mathbf{u}_2 on applying force \mathbf{F}_1 and \mathbf{F}_2 then;

$\mathbf{F}_1 = k (\mathbf{u}_1 - \mathbf{u}_2)$ and $\mathbf{F}_2 = k (\mathbf{u}_2 - \mathbf{u}_1)$ where k is the stiffness of material.

Stiffness of material for truss is given by the product of modulus of elasticity with area divided by the elemental length [3]. That is

$$k = \frac{EA}{l} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

Hence, rearranging, the above equations becomes:

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \frac{EA}{l} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}$$

B. Truss

Trusses are defined as structures composed of slender bars connected to each other through pins at their end points. In practice, joints do not have to be pinned as long as bars passing through a joint intersect at a single point. A truss consists of three components, one is the top chord i.e. beam at the top, bottom chord, i.e. the beam at the bottom and other interior beams.

The deformation of a truss element can be found using the following equation:

$\delta = \frac{FL}{EA}$ or, $F = \left(\frac{EA}{L}\right) \delta$ where δ is the axial deformation, F is the axial force in the truss element, L is the length of the element, E is the Young's modulus, and A is the cross-sectional area of the element.

The term that is multiplied by the deformation to get the force is the axial stiffness [4]. Hence, relation between force and deformation for the pin jointed truss element is written as [5].

$$F = K \cdot \delta, \text{ where,}$$

$$K = \frac{AE}{L} \begin{bmatrix} C^2 & CS & -C^2 & -CS \\ CS & S^2 & -CS & -S^2 \\ -C^2 & -CS & C^2 & CS \\ -CS & -S^2 & CS & S^2 \end{bmatrix}$$

Here, $C = \cos\theta$ and $S = \sin\theta$

C. MATLAB

MATLAB-MATrix LABoratory, is a very powerful software package that has many built-in tools for solving problems and for graphical illustrations. The simplest method for using the MATLAB product is interactively; an expression is entered by the user and MATLAB immediately responds with a result. It is also possible to write programs in MATLAB, which are essentially groups of commands that are executed sequentially [6]. MATLAB integrates computation, visualization, and programming environment. These factors make MATLAB an excellent tool for teaching and research [7].

Programming on MATLAB scripts satisfies the theoretical foundations given by finite element method. To compute the 2D linear structure, various finite element functions are introduced to mesh the input data as nodal points, elements and boundary conditions. Finally solving of these functions takes place and the problem is analyzed [8].

II. METHODOLOGIES

The program was completed through various steps. First, through the technique of mind mapping and brainstorming, different areas where this project could be useful were figured out. Then, all the essential subject matters were literature reviewed. Preparation of the FEM Model: Discretize/mesh the solid, define

material properties, apply boundary conditions and choosing approximate functions, formulate linear equations, and solving the equations were done. Through the testing of different types of truss and comparing with other method's result, the developed program is satisfactory.

A. MATLAB

First meshing of input data in matrix form is done to create the local and global stiffness matrix. To obtain a unique solution to given problem, it is necessary to specify at every point on the surface where boundary conditions are applied. Finally, numerical quadrature is used for the integration of finite element equations.

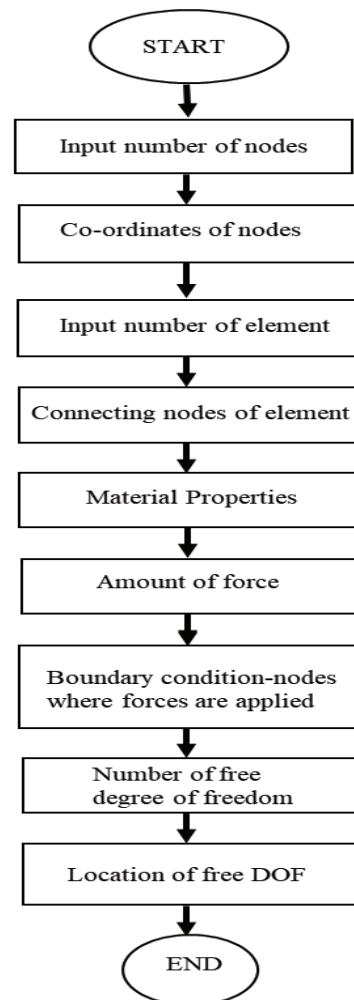


Fig 1: Flowchart of MATLAB program

The output shown by the program are in the form of graphical deformed plot and list results (nodal displacement, external forces and elemental stress-strain). The units of the outputs shown are in SI system. Below is an example of Kingpost analysis when 1000N load is applied, and units of result shown are, Nodal displacement in meters (m), Reaction forces in kilonewton (KN), Stress-strain in kilonewton per meter square (KN/m²).

- Deformed plot

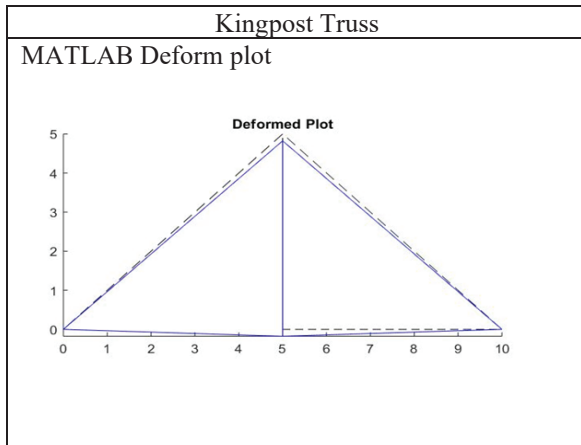


Fig 2: Deformed plot of Kingpost Truss

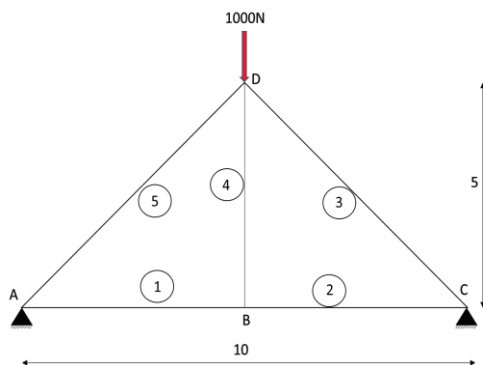
- Output (Kingpost truss)

```

NODAL DISPLACEMENT
NID    X-DISP    Y-DISP
1      0.000e+00  0.000e+00
2      0.000e+00  -7.071e-04
3      0.000e+00  0.000e+00
4      0.000e+00  -7.071e-04
EXTERNAL FORCES
NID    X-FORCE    Y-FORCE
1      5.000e+02  5.000e+02
2      0.000e+00  1.100e-13
3      -5.000e+02  5.000e+02
4      0.000e+00  -1.000e+03
ELEMENT STRESS
EID    STRAIN    STRESS
1      0.000e+00  0.000e+00
2      0.000e+00  0.000e+00
3      0.000e+00  0.000e+00
4      -7.071e-05  -7.071e+02
5      -7.071e-05  -7.071e+02
    
```

Thus, through this program, all resultant under the influence of load i.e. displacement, forces and stress strain are obtained, also deformed plot for better conceptualization is shown.

B. Mathematical method



Assuming arbitrary constrains for truss analysis whose values are near to that of aluminum,
 Young modulus of elasticity (E) = 10^6 KN/m²
 Area (A) = 1m²
 There are 4 nodes (A,B,C,D) and 5 elements making up the truss. Each element can be described as extending from one node to another. This also can be defined in a table below Elements with sines and cosines to be used in the stiffness matrix.

Element	Connecting nodes	Length (meter)	Cosine	Sine
1	A-B	5	1	0
2	B-C	5	1	0
3	C-D	7.07	-0.7	0.7
4	D-B	5	0	1
5	D-A	7.07	0.7	0.7

The equation for stiffness matrix is,

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

So, the matrix can be added easily.

The degree of freedom for each element stiffness matrix were added into the same degree of freedom in the structural matrix. The resulting structural stiffness matrix is shown below.

$$K = 10e^6 * \begin{bmatrix} 2.69 & 0.69 & -2 & 0 & 0 & 0 & -0.69 & -0.69 \\ 0.69 & 0.69 & 0 & 0 & 0 & 0 & -0.69 & -0.69 \\ -2 & 0 & 4 & 0 & -2 & 0 & 0 & 0 \\ 0 & 0 & 0 & 2 & 0 & 0 & 0 & -2 \\ 0 & 0 & -2 & 0 & 2.69 & -0.69 & -0.69 & 0.69 \\ 0 & 0 & 0 & 0 & -0.69 & 0.69 & 0.69 & -0.69 \\ -0.69 & -0.69 & 0 & 0 & -0.69 & 0.69 & 1.38 & 0 \\ -0.69 & -0.69 & 0 & -2 & 0.69 & -0.69 & 0 & 3.38 \end{bmatrix}$$

Equation for stiffness, displacement and force is,

$$KQ = F$$

Where K is the structural or global stiffness matrix, Q is the displacement of each node, and F is the external force matrix. This results in:

$$K * \begin{pmatrix} 1_x \\ 1_y \\ 2_x \\ 2_y \\ 3_x \\ 3_y \\ 4_x \\ 4_y \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ -1000 \end{pmatrix}$$

Applying Boundary Conditions:

i.e. node 1 and 3 are fixed and eliminating constrained terms then the equation becomes,

$$\begin{bmatrix} 4 & 0 & 0 & 0 \\ 0 & 2 & 0 & -2 \\ 0 & 0 & 1.38 & 0 \\ 0 & -2 & 0 & 3.38 \end{bmatrix} * \begin{Bmatrix} 2_x \\ 2_y \\ 4_x \\ 4_y \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ -1000 \end{Bmatrix}$$

Solving the above equation,

Displacement at node 2 is:

X-axis= 0, Y-axis= $-0.724*10^{-5}$ meter

Displacement at node 4 is:

X-axis= 0, Y-axis= $-0.724*10^{-5}$ meter

Computing Stresses

$$\sigma = \frac{E}{L} \{ -C \quad -S \quad C \quad S \} q$$

Thus, following values were obtained:

$$\sigma_1 = 0 \text{ KN/m}^2$$

$$\sigma_2 = 0 \text{ KN/m}^2$$

$$\sigma_3 = 0 \text{ KN/m}^2$$

$$\sigma_4 = -7.23*10^2 \text{ KN/m}^2$$

$$\sigma_5 = -7.23*10^2 \text{ KN/m}^2$$

C. VALIDATION OF RESULT

Numeric results of the program were compared with the solution from mathematical method.

- Numeric result comparison

TABLE 1: Deformation of Kingpost truss along X and Y axis

Deformation Solution (meter) King Post				
Node	MATLAB		Mathematical Method	
	X-Axis	Y-Axis	X-Axis	Y-Axis
1	0	0	0	0
2	0	-7.07E-04	0	-7.07E-04
3	0	0	0	0
4	0	-7.07E-04	0	-7.07E-04

TABLE 2: Stress in Kingpost truss across elements

Elemental Stress (KN/m ²) King Post		
Node	MATLAB	Mathematical Method
	Stress	Stress
1	0.000e+00	0
2	0.000e+00	0
3	0.000e+00	0
4	-7.071e+02	-7.23*10 ²
5	-7.071e+02	-7.23*10 ²

III. FINDINGS

The final result obtained from mathematical calculation and MATLAB program were identical and close to each other. And through the observation of the deformed plot, the deformation shape and direction can be visualized.

IV. CONCLUSION

The program is able to perform static analysis of 2D trusses that undergoes a linear elastic behavior. The program has features of material selection by giving the material property (Young Modulus of Elasticity), applying constant properties including area, length where they are necessary. Plotting, such as original and deformed geometry, stress and displacement behavior such as small and large displacement can be done. And various output as per the input data can be obtained as nodal displacement, reaction forces and elemental stress and strain. The developed MATLAB code can be a platform for initial familiarization in structures and can contribute in further research activities related to structures analysis.

V. REFERENCES

- [1] E. G. Thompson, Introduction to Finite Element Method, Singapur: John Wiley and Sons, 2005.
- [2] S. S. Rao, The Finite Element Method in Engineering, Burlington: Butterworth-Heinemann Publication, 2011.
- [3] D. L. Logan, A First Course in the Finite Element Method, india: Cengage Learning, 2015.
- [4] O. D. Weck and I. Y. Kim, "Finite Element Method," Massachusetts Institute of Technology, Cambridge, Massachusetts, 2014.
- [5] R. S., Finite Element Analysis in Engineering Design, New Delhi: S.Chand & Company Limited, 2008.
- [6] S. Attaway, A Pratical Introduction to Programming and Problem Solving, Boston: Butterworth-Heinemann Pbulication, 2013.
- [7] D. Houcque, Tntroduction to MATLAB for Engineering Students, Evanston: Northwestern University, 2005.
- [8] R. T. D. F. O.C Zienkiewicz, Introductory Lecture on the Finite Element Method, London: Elsevier Limited, 2014.