

Finite Element Analysis on Cantilever Truss using Nodal Displacement Method

Yash Thakur¹, Yash Yogesh Bichu¹, Prathamesh Belgaonkar¹, Ved Gavade¹, Yogita Potdar².

¹KLS, Gogte Institute of Technology, Belagavi, 590008, India.

²Assistant Professor, Dept. of Mechanical Engineering, KLS, Gogte Institute of Technology, Belagavi, 590008, India.

Abstract - This paper focuses on the Finite Element Method (FEM) analysis of trusses using the Nodal Displacement Method. It contains a detailed methodology to solve a problem on a truss containing 6 elements and 5 nodes, which helps in determining the cross-sectional area of the elements and the same results have been validated by analysing the model using SolidWorks software. Here, the current example taken into consideration is from Abhishek Alloys PVT. LTD., Belgaum, Karnataka, India, where this truss will be used to support overhead rails.

An element will have material properties like Young's modulus, Poisson's ratio, cross-sectional area.

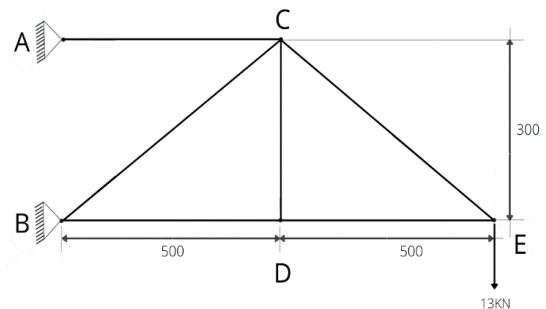


Fig-1: Line diagram of a truss

Key Words: Finite Element Method (FEM), Cantilever Truss, Nodal Displacement Method, Stiffness matrix, Load matrix, Displacement matrix.

1. INTRODUCTION

A truss is a combination of elements that are interconnected in such a way that it forms a rigid structure that has a high load-carrying capacity than a regular beam. Trusses have a huge advantage since they have high strength-to-weight ratio. A cantilever truss is similar to a cantilever beam in which one end is fixed and the other is free where the load can be applied and the load which is applied acts as a moment about the fixed end. Such structures are used in construction applications, industrial use, bridge constructions, etcetera. [1]

Finite Element Method (FEM) is a numerical analysis method used by engineers and mathematicians to obtain fairly accurate solutions to a partial differential equation that defines a wide range of physical and non-physical systems. [2]

Here, in this paper, the example taken for the study is from Abhishek Alloys PVT. LTD., Belgaum, Karnataka, India, where this truss will be used to support overhead rails. With the constraints and loading conditions provided by the industry, the truss is designed by the nodal displacement method which will give the cross-sectional area of the elements and the results obtained will therefore be validated using SolidWorks software.

2. Steps involved in Finite Element Method (FEM) analysis by Nodal Displacement method:

2.1 Discretisation of a continuum:

The physical structure (continuum) is divided into small elements. These elements are formed by joining the nodes.

In this case, 6 elements are connected between 5 nodes as shown in fig. 1. Let the cross-sectional area of the elements be 'A' mm², material used is plain Carbon-Steel of Young's Modulus 2*10⁵ N/mm² and the yield strength of the material is 220 N/mm². Here, the factor of safety is considered to be 2. [3]

The dimensions of the elements are as follows: AC= 500 mm, BD= 500 mm, DE=500 mm, CD= 300 mm, BC= 583.1 mm and CE= 583.1 mm.

Points A and B are fixed and the load of 13kN is applied at point E vertically downwards.

2.2 Selection of displacement model or approximate function:

The displacement model selected can be in the polynomial form or it can be a trigonometric function. Usually, for ease of calculations, polynomial form is preferred.

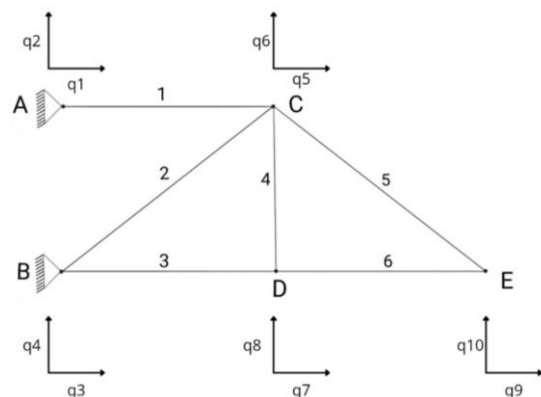


Fig-2: Displacement of each node.

Table -1: Element connectivity table:

Element	Nodal Displacements	
	1 st Node	2 nd Node
1	A (q ₁ , q ₂)	C (q ₅ , q ₆)
2	B (q ₃ , q ₄)	C (q ₅ , q ₆)
3	B (q ₃ , q ₄)	D (q ₇ , q ₈)
4	D (q ₇ , q ₈)	C (q ₅ , q ₆)
5	C (q ₅ , q ₆)	E (q ₉ , q ₁₀)
6	D (q ₇ , q ₈)	E (q ₉ , q ₁₀)

Here q₁, q₂ are displacements of node A (in X and Y direction respectively) due to the applied load. Similarly, q₃, q₄, q₅, q₆, q₇, q₈, q₉, and q₁₀ are the displacements of other nodes due to the applied load.

2.3 Formulation of Element Stiffness Matrix:

After a continuum is discretised with the desired element shapes, the individual stiffness matrix is formulated. The geometry of the element is defined by the global frame of reference. [4]

Table -2: Calculation of cosine and sine values.

Element	Coordinates		l _e	l = (x ₂ -x ₁) / l _e	m = (y ₂ -y ₁) / l _e
	(x ₁ , y ₁)	(x ₂ , y ₂)			
1	A (0,300)	C (500, 300)	500	1	0
2	B (0, 0)	C (500, 300)	583.1	0.86	0.51
3	B (0, 0)	D (500, 0)	500	1	0
4	D (500, 0)	C (500, 300)	300	0	1
5	C (500, 300)	E (1000, 0)	583.1	0.86	-0.51
6	D (500, 0)	E (1000, 0)	500	1	0

Here, the values for 'l' and 'm' for every element are calculated as shown in the table above and the element stiffness matrix is formulated by substituting these values.

$$K = A^*E/l_e \begin{pmatrix} l^2 & lm & -l^2 & -lm \\ lm & m^2 & -lm & -m^2 \\ -l^2 & -lm & l^2 & lm \\ -lm & -m^2 & lm & m^2 \end{pmatrix} \begin{matrix} q_1 & q_2 & q_5 & q_6 \end{matrix}$$

$$K_1 = A^*E \begin{pmatrix} 0.002 & 0 & -0.002 & 0 \\ 0 & 0 & 0 & 0 \\ -0.002 & 0 & 0.002 & 0 \\ 0 & 0 & 0 & 0 \end{pmatrix} \begin{matrix} q_1 \\ q_2 \\ q_5 \\ q_6 \end{matrix}$$

$$K_2 = A^*E \begin{pmatrix} 0.00126 & 0.00075 & -0.00126 & 0.00075 \\ 0.00075 & 0.00045 & -0.00075 & -0.00045 \\ -0.00126 & -0.00075 & 0.00126 & 0.00075 \\ -0.00075 & -0.00045 & 0.00075 & 0.00045 \end{pmatrix} \begin{matrix} q_3 \\ q_4 \\ q_5 \\ q_6 \end{matrix}$$

$$K_3 = A^*E \begin{pmatrix} 0.002 & 0 & -0.002 & 0 \\ 0 & 0 & 0 & 0 \\ -0.002 & 0 & 0.002 & 0 \\ 0 & 0 & 0 & 0 \end{pmatrix} \begin{matrix} q_3 \\ q_4 \\ q_7 \\ q_8 \end{matrix}$$

$$K_4 = A^*E \begin{pmatrix} 0 & 0 & 0 & 0 \\ 0 & 0.0033 & 0 & -0.0033 \\ 0 & 0 & 0 & 0 \\ 0 & -0.0033 & 0 & 0.0033 \end{pmatrix} \begin{matrix} q_7 \\ q_8 \\ q_5 \\ q_6 \end{matrix}$$

$$K_5 = A^*E \begin{pmatrix} 0.00126 & -0.00075 & -0.00126 & 0.00075 \\ -0.00075 & 0.00045 & 0.00075 & -0.00045 \\ -0.00126 & 0.00075 & 0.00126 & -0.00075 \\ 0.00075 & -0.00045 & -0.00075 & 0.00045 \end{pmatrix} \begin{matrix} q_5 \\ q_6 \\ q_9 \\ q_{10} \end{matrix}$$

$$K_6 = A^*E \begin{pmatrix} 0.002 & 0 & -0.002 & 0 \\ 0 & 0 & 0 & 0 \\ -0.002 & 0 & 0.002 & 0 \\ 0 & 0 & 0 & 0 \end{pmatrix} \begin{matrix} q_7 \\ q_8 \\ q_9 \\ q_{10} \end{matrix}$$

Once the element stiffness matrices are formulated in the global coordinates, the global stiffness matrix is to be formulated by adding the element stiffness matrices.

2.4 Formulation of Global Stiffness Matrix or Overall Stiffness Matrix:

$$\text{Global Stiffness Matrix} = A \times E \begin{pmatrix}
 q^1 & q^2 & q^3 & q^4 & q^5 & q^6 & q^7 & q^8 & q^9 & q^{10} \\
 0.002 & 0 & 0 & 0 & -0.002 & 0 & 0 & 0 & 0 & 0 \\
 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
 0 & 0 & 0.00327 & 0.00075 & -0.00126 & -0.00075 & -0.002 & 0 & 0 & 0 \\
 0 & 0 & 0.00075 & 0.00045 & -0.00075 & -0.00045 & 0 & 0 & 0 & 0 \\
 -0.002 & 0 & -0.00126 & -0.00075 & 0.00253 & 0 & 0 & 0 & -0.00126 & 0.00075 \\
 0 & 0 & -0.00075 & -0.00045 & 0 & 0.00422 & 0 & -0.0033 & 0.00075 & -0.00045 \\
 0 & 0 & -0.002 & 0 & 0 & 0 & 0.004 & 0 & -0.002 & 0 \\
 0 & 0 & 0 & 0 & 0 & -0.0033 & 0 & 0.0033 & 0 & 0 \\
 0 & 0 & 0 & 0 & -0.00126 & 0.00075 & -0.002 & 0 & 0.00327 & -0.00075 \\
 0 & 0 & 0 & 0 & 0.00075 & 0.00045 & 0 & 0 & -0.00075 & 0.00045
 \end{pmatrix} \begin{matrix} q_1 \\ q_2 \\ q_3 \\ q_4 \\ q_5 \\ q_6 \\ q_7 \\ q_8 \\ q_9 \\ q_{10} \end{matrix}$$

2.5 Solving Stress Matrix:

The allowable stress = Yield stress / Factor of Safety
 $\sigma_y = 220/2 = 110 \text{ N/mm}^2$ ($\sigma_1, \sigma_2, \sigma_3, \sigma_4, \sigma_5, \sigma_6 = 110 \text{ N/mm}^2$).

$$\sigma = E/L_e \begin{bmatrix} B \end{bmatrix} \begin{bmatrix} Q \end{bmatrix}$$

$$\sigma = E/L_e \begin{bmatrix} -1 & -m & 1 & m \end{bmatrix} \begin{bmatrix} Q \end{bmatrix}$$

$$\sigma_1 = 2 \times 10^5 / 500 \begin{bmatrix} -1 & 0 & 1 & 0 \end{bmatrix} \begin{bmatrix} q_1 \\ q_2 \\ q_5 \\ q_6 \end{bmatrix}$$

$$\sigma_2 = 2 \times 10^5 / 583.1 \begin{bmatrix} -0.86 & -0.51 & 0.86 & 0.51 \end{bmatrix} \begin{bmatrix} q_3 \\ q_4 \\ q_5 \\ q_6 \end{bmatrix}$$

$$\sigma_3 = 2 \times 10^5 / 500 \begin{bmatrix} -1 & 0 & 1 & 0 \end{bmatrix} \begin{bmatrix} q_3 \\ q_4 \\ q_7 \\ q_8 \end{bmatrix}$$

$$\sigma_4 = 2 \times 10^5 / 300 \begin{bmatrix} 0 & -1 & 0 & 1 \end{bmatrix} \begin{bmatrix} q_7 \\ q_8 \\ q_5 \\ q_6 \end{bmatrix}$$

$$\sigma_5 = 2 \times 10^5 / 583.1 \begin{bmatrix} -0.86 & 0.51 & 0.86 & -0.51 \end{bmatrix} \begin{bmatrix} q_5 \\ q_6 \\ q_9 \\ q_{10} \end{bmatrix}$$

$$\sigma_6 = 2 \times 10^5 / 500 \begin{bmatrix} -1 & 0 & 1 & 0 \end{bmatrix} \begin{bmatrix} q_7 \end{bmatrix}$$

Solving the above matrices, we get the below mentioned equations:

$$110 = 400 * (-q_1 + q_5) \quad (1)$$

$$110 = 343 * (-0.86 * q_3 - 0.51 * q_4 + 0.86 * q_5 + 0.51 * q_6) \quad (2)$$

$$110 = 400 * (-q_3 + q_7) \quad (3)$$

$$110 = 666.67 * (-q_8 + q_6) \quad (4)$$

$$110 = 343 * (-0.86 * q_5 + 0.51 * q_6 + 0.86 * q_9 - 0.51 * q_{10}) \quad (5)$$

$$110 = 400 * (-q_7 + q_9) \quad (6)$$

Solving these equations, we get,
 (Since nodes A and B are fixed, their displacements q_1, q_2, q_3 and q_4 are equal to zero)

- $q_5 = 0.275 \text{ mm}$,
- $q_6 = 0.165 \text{ mm}$,
- $q_7 = 0.275 \text{ mm}$,
- $q_8 = 8.25 * 10^{-7} \text{ mm}$,
- $q_9 = 0.55 \text{ mm}$,
- $q_{10} = -9.75 * 10^{-4} \text{ mm}$.

2.6 Formulation of Global Load Vector:

The load vector is formulated by defining the loads at individual nodes (both vertical and horizontal) in the direction of their displacements.

Note: Load vector is always a column matrix.

$$F = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

$$\begin{matrix} 0 \\ 0 \\ -13000 \end{matrix}$$

$$[F] = [K] * \{Q\}$$

where,

[F]= load vector and it is a column matrix.

[K]= stiffness matrix and it is a square matrix.

{Q}= displacement matrix and it is a column matrix.

2.7 Formation of Equilibrium Equation:

The equilibrium condition is given by the standard form which is,

$$\begin{pmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ -13000 \\ 0 \end{pmatrix} = A \times E \begin{pmatrix} 0.002 & 0 & 0 & 0 & -0.002 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0.00327 & 0.00075 & -0.00126 & -0.00075 & -0.002 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0.00075 & 0.00045 & -0.00075 & -0.00045 & 0 & 0 & 0 & 0 & 0 \\ -0.002 & 0 & -0.00126 & -0.00075 & 0.00253 & 0 & 0 & 0 & -0.00126 & 0.00075 & 0 \\ 0 & 0 & -0.00075 & -0.00045 & 0 & 0.00422 & 0 & -0.0033 & 0.00075 & -0.00045 & 0 \\ 0 & 0 & -0.002 & 0 & 0 & 0 & 0.004 & 0 & -0.002 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & -0.0033 & 0 & 0.0033 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -0.00126 & 0.00075 & -0.002 & 0 & 0.00327 & -0.00075 & 0 \\ 0 & 0 & 0 & 0 & 0.00075 & 0.00045 & 0 & 0 & 0 & -0.00075 & 0.00045 \end{pmatrix} \begin{pmatrix} q_1 \\ q_2 \\ q_3 \\ q_4 \\ * \\ q_5 \\ q_6 \\ q_7 \\ q_8 \\ q_9 \\ q_{10} \end{pmatrix}$$

Since q₁, q₂, q₃ and q₄ are equal to zero, cancelling the respective rows and columns and solving the matrix, we get,

$$-13000 = A * 2 * 10^5 [(0.00075 * q_5) + (-0.00045 * q_6) + (-0.00075 * q_9) + (0.00045 * q_{10})]$$

Substituting the values of q₅, q₆, q₉, q₁₀, we get,

$$-13000 = A * 2 * 10^5 [(0.00075 * 0.275) + (-0.00045 * 0.165) + (-0.00075 * 0.55) + (0.00045 * -0.000975)]$$

Therefore, A = 233.24 mm².

Hence, the cross-sectional area of each element should be 233.24 mm².

3. Validating results using software:

The dimensions of elements of the truss obtained from manual calculations applying Nodal Displacement Method are verified using SolidWorks software.

Considering a rectangular section of 50mm width, we get a thickness of 4.66 mm for the area to be 233.24mm². But the standard available plate thickness is 5 mm.

After constructing the 3-D model of the truss as per the above dimensions, we apply boundary conditions such as fixed points and loading conditions. Further, we mesh the model and analyse the stresses. Hence by analysing stresses, we get, the results as shown below:

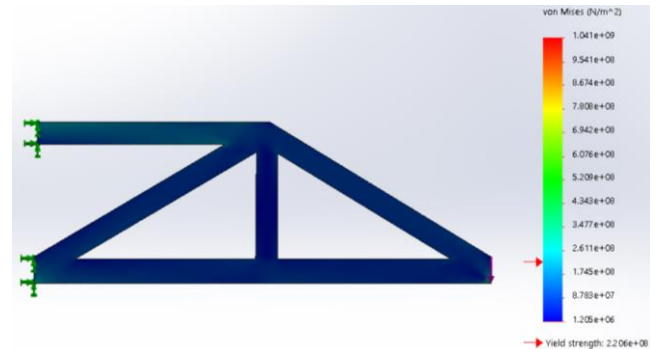


Fig -3: 3-dimensional model of truss and stress analysis

From the above figure, we can see that the stress lies in the “dark blue” region which is below 110 N/mm².

(Note : The value of stress obtained in software is less than the manually calculated value because we have considered the thickness of plate as 5mm instead of 4.66mm.)

4. CONCLUSION

Different design engineers use different techniques to analyse trusses. However, this paper deals with FEM analysis of truss using nodal displacement method and validating the results using software. Using this method, we were able to find the cross-sectional area of the truss for carrying the specified load and these results are in accordance with those obtained from the software. This method can also be applied to a wide range of trusses to find the cross-sectional areas of the element.

ACKNOWLEDGEMENT

We would like to extend our sincere gratitude to the company Abhishek Alloys PVT. LTD., Belgaum, India, for giving us the right platform and learning opportunity to increase our practical knowledge by providing us with the problem statement at hand.

REFERENCES

- [1] Mohamed Darwish, Khaled Nassar, "Design of steel truss cantilever formwork". DOI:10.13140/RG.2.2.35927.68006.
- [2] Zhangxin Chen "The Finite Element Method: Its Fundamentals and Applications in Engineering" <https://doi.org/10.1142/8155>.
- [3] "Machine Design Data Handbook" by S.C. Pilli, H.G. Patil. Published by I.K. International Publishing House Pvt. Ltd. (2014). ISBN 978-93-84588-03-8.
- [4] Zienkiewicz, O.C., Taylor R.L., Zhu J.Z. (2005), The Finite Element Method: Its Basis and Fundamentals (6th ed.), Oxford, UK: Elsevier Butterworth-Heinemann, ISBN 978-0750663205.