[1.] Damir HODŽIĆ

BENDING ANALYSIS OF CANTILEVER BEAM IN FINITE ELEMENT METHOD

[1.]University of Bihać, Faculty of Technical Engineering, Bihać, BOSNIA & HERZEGOVINA

Abstract: Paper describes bending of cantilever beam and its analysis using finite element method. Cantilever beam is a structural member and in this paper a two dimensional Finite Element model for steel material beam has been developed to study. This work aims to analyze bending analysis of cantilever beam with boundary conditions. The outputs of finite element model are used to investigate effect of point load on integrity of beam and mechanical properties of material.
Keywords: cantilever beam, bending analysis, finite element method

1. INTRODUCTION

The finite element method is a numerical procedure for solving engineering problems and mathematical physics problems. In most cases when the analyzed structure has a complex geometry, when the load is complex and when the structures are made of different materials, it is not possible to find a solution in analytical form. Analytical solution means obtaining analytical formulas for calculating the required characteristics (deflection, temperature, stress etc.) at different points in the structure. To obtain such data, differential or partial differential equations should be solved. This can only be done for very simple problems. For complex geometry and complex load it is not possible to find solutions in analytical form. Therefore, numerical methods are used and the most commonly used method is the finite element method. Solving problems by the finite element method comes down to solving a system of algebraic equations. The obtained solutions are approximate and refer to certain points of the structure. The modeling process consists of discretizing the continuum. Such a model consists of finite elements, which are connected in nodes along boundary common lines or common surfaces. Equations are set for each finite element and their combination gives the equations of the whole structure. Depending on the type of problem being solved, appropriate data are obtained as solutions. Thus, in the case of calculating the stress-strain state of a structure, the results are the displacements of each node of the structure and the stresses within each element. Displacements and stresses are due to the action of an external load. In problems not related to structural analysis unknown in the nodes may be some other physical quantities e.g. temperature [1].

The finite element method has the greatest advantage that it can be expanded in all areas and is therefore available to all technical and scientific disciplines. Today, in addition to the known linear calculations, the more complicated ones are also carried out.

The structure to be calculated is broken down into a multitude of small building blocks, the finitely large elements, the physical properties of which are known exactly or approximately. The finite number of links corresponds to the nodes that connect the individual elements. These links correspond to a linear system of equations that can be solved numerically particularly well [2, 3].

2. BENDING OF CANTILEVER BEAM

To show how to solve a cantilever beam using Finite Element Method it will be analysed beam in Figure 1, which is fixed at one end and supported by a roller support at the other end. It is loaded by concentrated load of 20 [kN] at the center of the span. Modulus of elasticity of the beam material is $E=2\cdot10^5$ [MPa] and moment of inertia is $I=2500$ [cm^4]. It is necessary to calculate deflection under the load [4,5].

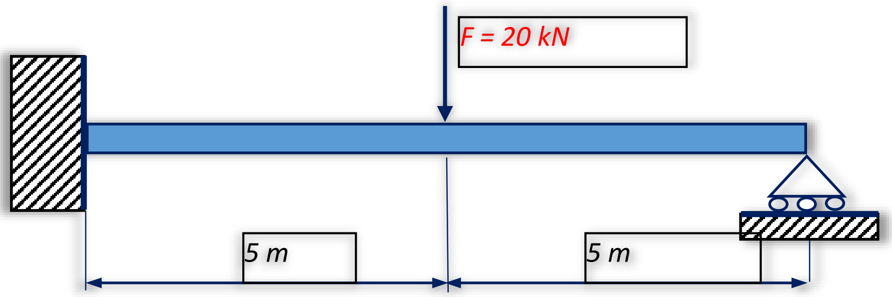

Figure 1. Cantilever beam to be analyzed

First, let the beam divide into two parts and make a free body diagram, Figure 2.

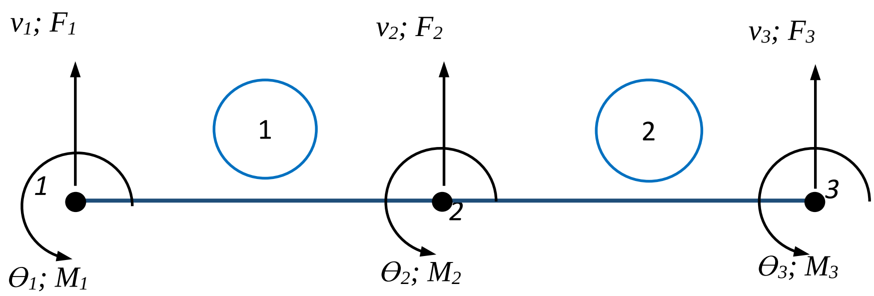

Figure 2. Free body diagram of loaded cantilever beam

In figure 2 factors are:

- ≡ Vertical deflection – v
- ≡ Shear force – F
- ≡ Slope deflection - θ
- ≡ Bending moment – M

There are 3 nodes and 2 elements at the free body diagram and in each node it is necessary to consider all 4 factors. Modulus of elasticity and moment of inertia are same for both beam elements. Now it has to be calculated nodal deflection for both beam elements using stiffness matrix. Common stiffness matrix for beam element is:

$$[K] = \frac{E \cdot I}{l^3}\begin{bmatrix} 12 & 6l & -12 & 6l \\ 6l & 4l^2 & -6l & 2l^2 \\ -12 & -6l & 12 & -6l \\ 6l & 2l^2 & -6l & 4l^2 \end{bmatrix} \tag{1}$$

Due to the same datas for both elements (element length, modulus of elastixity and moment of inertia) stiffness matrices fort both elements will be identical:

$$[K_1] = [K_2] = 400 \cdot \begin{bmatrix} 12 & 6l & -12 & 6l \\ 6l & 4l^2 & -6l & 2l^2 \\ -12 & -6l & 12 & -6l \\ 6l & 2l^2 & -6l & 4l^2 \end{bmatrix} \tag{2}$$

The first term of equation (2) is calculated:

$$\frac{E_1 \cdot I_1}{{l_1}^3} = \frac{2 \cdot 10^7 \cdot 2500}{500^3} = 400 \, \frac{N}{cm} \tag{3}$$

Now, input the values for length l = 5 [m] in the matrices and mark rows and columns of the matrices with vertical and slope deflection. Element 1 – vertical and slope deflection of nodes 1 and 2 and element 2 - vertical and slope deflection of nodes 2 and 3.

$$[K_1] = 400 \cdot \begin{array}{c} \begin{array}{cccc} v_1 & \Theta_1 & v_2 & \Theta_2 \end{array} \\ \begin{bmatrix} 12 & 3000 & -12 & 3000 \\ 3000 & 10^6 & -3000 & 5 \cdot 10^5 \\ -12 & -3000 & 12 & -3000 \\ 3000 & 5 \cdot 10^5 & -3000 & 10^6 \end{bmatrix} \end{array} \begin{array}{c} \\ v_1 \\ \Theta_1 \\ v_2 \\ \Theta_2 \end{array}$$

$$[K_2] = 400 \cdot \begin{array}{c} \begin{array}{cccc} v_2 & \Theta_2 & v_3 & \Theta_3 \end{array} \\ \begin{bmatrix} 12 & 3000 & -12 & 3000 \\ 3000 & 10^6 & -3000 & 5 \cdot 10^5 \\ -12 & -3000 & 12 & -3000 \\ 3000 & 5 \cdot 10^5 & -3000 & 10^6 \end{bmatrix} \end{array} \begin{array}{c} \\ v_2 \\ \Theta_2 \\ v_3 \\ \Theta_3 \end{array} \tag{4, 5}$$

Global stiffness matrix for beam can be calculated as:

$$[K] = 400 \; \begin{array}{cccccc|c} v_1 & \Theta_1 & v_2 & \Theta_2 & v_3 & \Theta_3 & \\ 12 & 3000 & -12 & 3000 & 0 & 0 & v_1 \\ 3000 & 10^6 & -3000 & 5 \cdot 10^5 & 0 & 0 & \Theta_1 \\ -12 & -3000 & 24 & 0 & -12 & 3000 & v_2 \\ 3000 & 5 \cdot 10^5 & 0 & 2 \cdot 10^6 & -3000 & 5 \cdot 10^5 & \Theta_2 \\ 0 & 0 & -12 & -3000 & 12 & -3000 & v_3 \\ 0 & 0 & 3000 & 5 \cdot 10^5 & -3000 & 10^6 & \Theta_3 \end{array} \tag{6}$$

Finite element equation is formatted:

$$[K] = [u] \cdot [F] \tag{7}$$

where: [u] – displacement vector; [F] – deformation vector

Now, global stiffness matrix can be written as:

$$[K] = 400 \cdot \begin{bmatrix} 12 & 3000 & -12 & 3000 & 0 & 0 \\ 3000 & 10^6 & -3000 & 5\cdot 10^5 & 0 & 0 \\ -12 & -3000 & 24 & 0 & -12 & 3000 \\ 3000 & 5\cdot 10^5 & 0 & 2\cdot 10^6 & -3000 & 5\cdot 10^5 \\ 0 & 0 & -12 & -3000 & 12 & -3000 \\ 0 & 0 & 3000 & 5\cdot 10^5 & -3000 & 10^6 \end{bmatrix} \cdot \begin{bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \end{bmatrix} = \begin{bmatrix} F_1 \\ M_1 \\ F_2 \\ M_2 \\ F_3 \\ M_3 \end{bmatrix} \tag{8}$$

Boundary conditions for the beam are:

- Fixed support (node 1) vertical and slope deflection is null $v_1 = \theta_1 = 0$
- Roller support (node 3) vertical deflection and moment is null $v_3 = M_3 = 0$
- Node 2 – $M_2 = 0$; $F_2 = -20\,000$ N

And now it is possible to calculate vertical and slope deflections at nodes 2 and 3 and reactions at the fixed support, node 1:

$$v_2 = -3{,}646 \text{ [cm]}$$
$$\theta_2 = -0{,}003125 \text{ [rad]}$$
$$\theta_3 = -0{,}0125 \text{ [rad]}$$
$$F_1 = 13{,}75 \text{ [kN]}$$
$$M_1 = 3{,}75 \text{ [kNm]}$$

3. SOFTWARE ANALYSIS

Of course, it is possible to analyse example of cantilever beam in software and to compare results wit the results calculated with finite element method. For this example sofware Ansys was used and the results for vertical and slope deflections at nodes 2 and 3 and reactions at the fixed support are shown in Figure 3 [4].

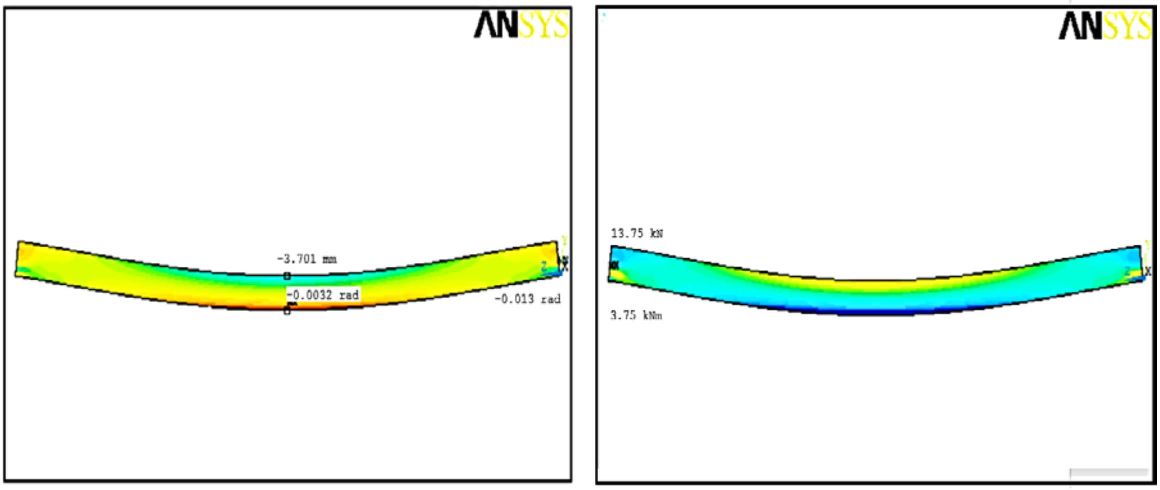

Figure 3. Results for vertical and slope deflections at nodes 2 and 3 and reactions at the fixed support using software Ansys

Now, compare calculated results and results obtained by software, Table 1.

Table 1. Comparison of results

Factor	FEM	Software
v_2	-3.646	-3.701
θ_2	-0.003125	-0.0032
θ_3	-0.0125	-0.013
F_1	13.75	13.75
M_1	3.75	3.75

4. CONCLUSION

The paper presents a simple example of a cantilever beam for which the basic parameters were calculated using the finite element method and using software. A comparison of the obtained results shows that the results are quite equal and the best match is in fixed support of cantilever beam. The finite element method is an efficient method by which the parameters on the structure can be determined. The results obtained by this method are with high accuracy and are reliable in the analysis and calculation of structures.

Note:

This paper is based on the paper presented at International Conference on Applied Sciences – ICAS 2021, organized by University Politehnica Timisoara – Faculty of Engineering Hunedoara (ROMANIA) and University of Banja Luka, Faculty of Mechanical Engineering Banja Luka (BOSNIA & HERZEGOVINA), in Hunedoara, ROMANIA, 12–14 May, 2021.

References

[1] Zaimović Uzunović N, Lemeš S 2002 Metod konačnih elemenata, Dom štampe Zenica
[2] Kunow A 2000 Finite-Elemente-Methode – Anwendungen und Lösungen, Huthig Verlag Heidelberg
[3] Kasat A S, Varghese V 2012 Finite element analysis of prestressed concrete beams, International Journal of Advanced Technology in Civil Engineering, Volume 1, Issue 4.
[4] https://www.youtube.com/watch?v=yfyElneBW98
[5] Shelke S 2017, Static/Modal Analysis of Cantilever Beam, International Journal of Advance Research and Innovative Ideas in Education, Volume 3, Issue 4.
[6] ANSYS Tutorial

ISSN 1584 – 2665 (printed version); ISSN 2601 – 2332 (online); ISSN-L 1584 – 2665