Scientific Journal of Impact Factor (SJIF): 5.71

e-ISSN (O): 2348-4470
p-ISSN (P): 2348-6406

International Journal of Advance Engineering and Research Development

Volume 5, Issue 04, April -2018

DEVELOPING MATLAB CODE FOR ANALYSIS OF BEAM USING FEM METHOD

(Cantilever and Simply Supported Beam with, in between Support Conditions)

Arpitkumar G. Patel[1], R. G. Jivani[2] ,V. A. Pandya[3]

[1]*Research Scholar,* [2] *Associate Professor,* [3] *Assistant Professor*
[1,2,3]*Department of Mechanical Engineering, BVM Engineering College, V.V.nagar, Anand, India.*

Abstract--*Beam is an important member of any structure and bears heavy loads and hence before using beams one have to be very sure about its strength and deflection. Industrial beams have different types of support, shape and structural properties. Various types of loads are acting on it like point loads, distributed loads etc.so for each stage of design the calculations to be done.to overcome this situation In present work Developed MATLAB code for beam having different loads and different end conditions with different shape to calculate global stiffness matrix in MATLAB which gives result of deflection graph and also gives natural frequency without changing the full code, to just change the parameters.*

Keywords: - *Reaction at Different nodes, Deflection graph, mode frequency, FEM approach, MATLAB.*

I. INTRODUCTION:

A beam is a structure loaded by forces acting transversely (sideways) to it‘s length and this make the beam bend. Beams may be supported across a span in various ways e.g. simply supported beam, cantilever beam, overhanging beam & fixed beams.[1],[2],[3] as shown in Figure 1.These all types of support conditions are Incorporated with the MATLAB program. Also the different shape (Rectangular, circular and I beam) beam Incorporated.

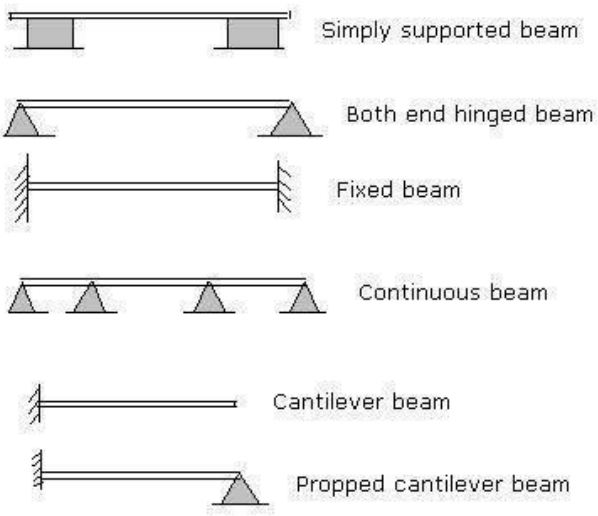

Figure 1 Different types of Support Conditions

This Beam has ability to sustain the different kind of loads and it varies with its support condition.[3],[4]
When load is applied on the beam according to its boundary conditions, shape of beam and types of material, it gives reaction parameters [5],[6] like natural frequency, deflection and stress are found out if the global stiffness matrix is prepared.

if the no. of element in beam is less then we can easily calculate with manually but when the no.of element is more then to carry stiffness matrix is difficult task to handle and solve so the main aim of present work is to make universal MATLAB program to calculate all the result parameter (deflection, stress at each nodes and natural frequency) with entering input parameter (material, shape and end conditions of the beam) this MATLAB program FEM approach is used.

II. FEM FOR BEAM

FEM is technique in which the beam is divided into different part for each part of it called as element & each element has two node according behavior of node element, we can find the resulting parameter of whole beam [7][8] here Consider a cantilever beam having length L and width b and the depth is d. at the free end the point load is acting due to its stiffness beam is deflect v. Fem method the beam is divided into several part so the part of it is called as element. And for that particular element the stiffness matrix is given by Hermitian shape functions.

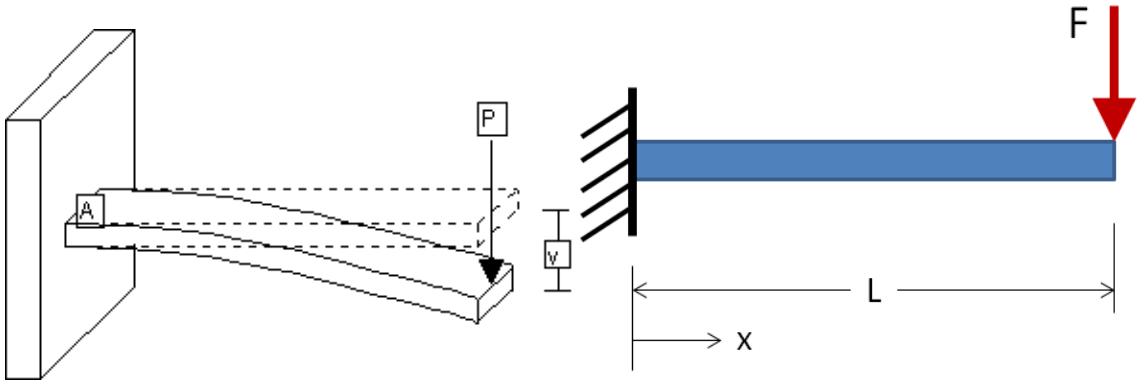

Figure 2 Cantilever Beam having point load on free end

The beam rigidity EI is assumed to be constant and is given by EI_e within the element. The element lol in equation is described as[8]

$$[S^e] = \frac{EI_e}{l^3}\begin{bmatrix} 12 & 6l & -12 & 6l \\ 6l & 4l^2 & -6l & 2l^2 \\ -12 & -61 & 12 & -6l \\ 6l & 2l^2 & -6l & 4l^2 \end{bmatrix}$$

The uniform mass matrix for the beam element is described as:

$$[M^e] = \int_0^1 \rho A\,[H(x)^T][H(x)]dx$$

$$[M^e] = \frac{\rho Al}{420}\begin{bmatrix} 156 & 22l & 54 & -13l \\ 22l & 4l^2 & 13l & -3l^2 \\ 54 & 13l & 156 & -22l \\ -13l & -3l^2 & -22l & 4l^2 \end{bmatrix}$$

The deflection of the system then can be calculated from the relation (3.10)[9].

$$[S] * [Q] = [F]$$

Where the
[S] = Overall stiffness matrix
[Q] = Deflection matrix
[F] = Applied force
Using these matrices the global stiffness matrix and the mass matrix can be found out. But stiffness matrix and mass matrix, if the number of element is larger or the length of beam is larger than for accurate result and for time saving the MATLAB and other application are used. Here the main purpose of to make the MATLAB code is to utilize for finding the deflection graph and reaction for the different end condition and for each node can be calculated easily and also the natural frequency of the beam is find out up to 4th mode for a cantilever type of beam, for the complex problem it can be modified.

III. ABOUT MATLAB

MATLAB (MATrix-LABoratory) is an interactive system for matrix-based computation, designed for scientific and engineering use. MATLAB is a high performance language for technical computing. It integrates computation, visualization, and programming in an easy-to-use environment where problems and solutions are expressed in familiar mathematical notations. Typically uses include: Math and computation Algorithm development modeling, simulation, and prototyping Data analyzing, exploration and visualization Scientific and engineering graphics Application development including graphical user interface building MATLAB is a software package for high-performance numerical computation and visualization. It

provides an interactive environment with hundreds of built-in functions for technical computation, graphics, and animation. Best of all, it also provides easy extensibility with its own high-level programming language. MATLAB is an interactive system whose basic data element is an array that does not requires dimensioning. This allows solving many technical computing problems especially those with matrix and vectoring foundation.

IV. RESULT / DISCUSSION :

Here consider a cantilever beam having rectangular cross section 40x6 mm taking 10 element and at the last node (Free end), load is applied 9.81N in downward direction. The Input parameter in MATLAB is shown in figure 3.

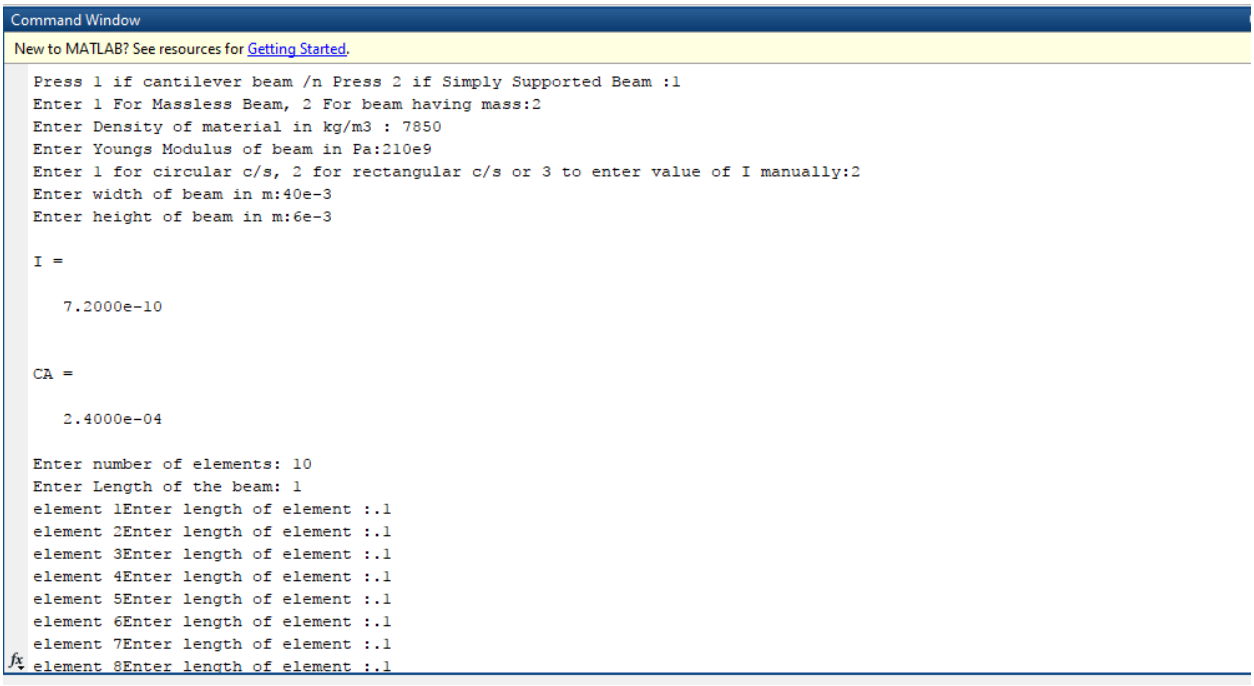

Figure 3 Input Parameter of Problem in MATLAB window

When the same beam is analyses with MATLAB code and the ansys software, the deflection we are getting is almost same with an error 3.2% shown in table 1.The Deflection Graph which is generated by the MATLAB program is Shown in Figure 4.

Table 1 Analysis of Maximum Deflection of Cantilever Beam

	Matlab Result	Ansys Result	Error (%)
Max Deflection	23.5	22.599	3.2

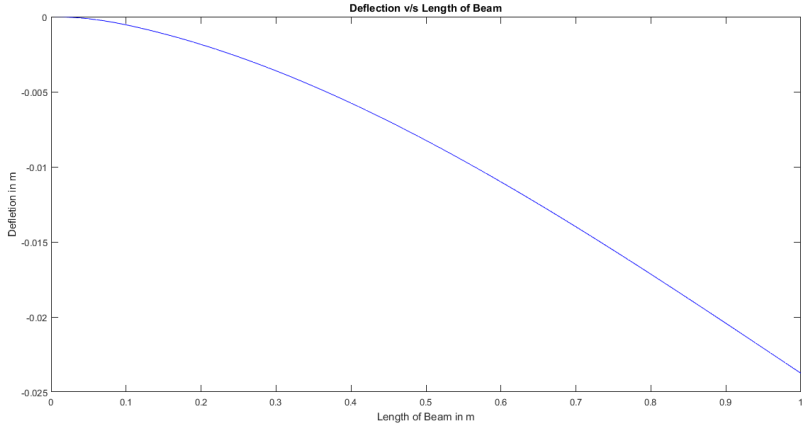

Figure 4 Deflection v/s Length of Beam

When the same modal is used for finding the natural frequency then we find that in both the way, using MATLAB and ansys the result are shown in table.

Table 2 Analysis of Natural Frequency

Natural Frequency	Matlab Result	Ansys Result	Error(%)
1st Mode	5.013	4.907	2.0985
2nd Mode	31.416	30.752	2.1158
3rd Mode	87.968	86.101	2.1226
4th Mode	172.379	168.720	2.1229

From the table the error is given on an average (2.11) %.
The same code is applicable with the support, in between the end condition.so that for a constant cross section and for deflection parameter the MATLAB code is versatile in use.
The above Result can be produce with different end condition and for that, only input parameter is changed therefore the Result will Change so, the MATLAB Code is User defined.

V. CONCLUSION:

A quick and efficient way of analyzing the beam design is developed through MATLAB code which can be used to solve beam design problems faster and errorless.

- Using this MATLAB code, We Conclude that for a constant cross section with different end condition having different point load and moment, we can find the Reaction force and deflection graph without taking much time.
- When the number of node in the beam is larger, then also Global Stiffness Matrix is produced and this Global Matrix is used to find the other parameters of the beam i.e. Natural Frequency, Stress produced in a element.
- In the Academic, for more detail study of Beam Problem, or for Experimental purpose, it can be used for better interpretation about the FEM approach as well as the co-relation of the MATLAB Coding.

REFERENCES

[1] Rajput, .K. Strength of Materials (2007), S Chand & Co Ltd, Page No. 310-312
[2] Ryder, G.H. Strength of material (1989), Macmillan Press Ltd., Page No. 153
[3] Timoshenko, S.P. and Young, D.H. Elements of strength of Materials (2003), East-West private ltd. Page No. 111-135
[4] Black, Paul H. &Adams, O. Eugene, Machine design (2001), Tata McGraw-Hill. Page No. 120.
[5] Singh, Dr. Sadhu, Strength of material (2013), Tata McGraw-Hill. Page No. 429-434
[6] Bhandari, V.B., Machine design (2006), Tata McGraw-Hill. Page No. 204-206
[7] J. N. Ready, 1993, Introduction to the finite element method, McGraw-Hill, 2nd Edition, p.(143-155),New York
[8] Henri P. Gavin, 2012, Structural element stiffness matrices and mass matrices, Structural Dynamic.
[9] G. Falsone, D. Settineri, 2011, An Euler–Bernoulli-like finite element method for Timoshenko beams, Mechanics Research Communications 38 (2011) 12–16.

