International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 09 Issue: 07 | July - 2025 SJIF Rating: 8.586 ISSN: 2582-3930

Determination of Natural Frequencies and Mode Shapes of a Cantilever Beam using Finite Element Method

Prasad J Waste[1], Bhagwan F Jogi[1], Debdatta Ratna[2], Hemant Warhatkar[1], Bikash Chakraborty[2], Nitin Ahire[2], Vishal Dalvi[2]

[1]Dr. Babasaheb Ambedkar Technological University, Lonere, Maharashtra, India
[2]Naval Materials Research Laboratory (DRDO), Ambernath, Maharashtra, India

---***---

Abstract - The study of vibration is critical in most engineering structures, with natural frequency and mode shape being two key parameters of a vibrating body. Identifying these parameters can be achieved through modal analysis, which is essential in designing mechanical systems. This paper investigates the vibration characteristics of a cantilever beam, aiming to determine its natural frequency and mode shape using finite element analysis (FEA) software. Determining the natural frequency of a beam involves calculating it based on physical properties and boundary conditions. Identifying the mode shapes of a beam involves recognizing the distinct deformation patterns that occur at specific frequencies. The natural frequencies and mode shapes are obtained using ANSYS software. The study focuses on a mild steel rectangular cantilever beam in a free-fixed condition, computing its modes and natural frequencies through FEA in ANSYS. The results are then utilized for further design processes.

Key Words: Modal Analysis, Natural Frequency, Mode Shapes, FEA.

1. INTRODUCTION

Vibration problems are often occurred in mechanical structure. Vibration is a time-dependent movement of objects relative to its equilibrium position. Vibrations can cause serious damages to the mechanical structures so they are undesirable. Every structure has its own natural frequencies. Body will vibrate at its maximum amplitude when the natural frequencies of structure match with external excitation force. This is called resonance. Due to resonance body may get damaged. Resonance can be avoided if natural frequencies of materials are known. Another important term used in the vibration of the structure is the mode shape. A mode shape is the deformation pattern acquired by a mechanical structure at the natural frequency. The shape obtained by structure at resonant frequency is described by mode shape. Modal analysis is a study of the dynamic characteristics of a structure or system under vibration and its response. The modal parameters of the structures can be extracted by modal analysis. The modal parameters are the fundamental elements that describe the response of a structure to free vibrations. The main purpose of modal analysis is to study the dynamic properties of structures like natural frequency, damping and mode shapes. So knowing these modal parameters helps to understand the structure's response to free vibration conditions [1-2]. The objective of this study is to find out the natural frequencies and mode shapes of the rectangular cantilever beam. Modal analysis has done in ANSYS software to find out natural frequency of the system and mode shapes.

In this work a modal analysis of mild steel single rectangular beam in free fixed condition is carried out. In the present paper, examination for modal parameters of simple rectangular cantilever beams is carried out. The FEA software ANSYS is used to predict the natural frequencies and mode shapes.

Supriya Sahu et al analysed the articulated robot having crack and without having crack for the different modes of frequencies on the finite element method (FEM). The model is designed in CATIA V5 with crack and without crack and a finite element model of the robot is established by utilizing the software of ANSYS Workbench [3]. Kumar et al. performed the free vibration study of heavy vehicle gearbox transmission casing using finite element simulation. The effect of mechanical properties of the four materials on natural frequency and mode shapes of heavy vehicle gearbox transmission casing was examined [4]. Ashwani Kumar et al. have considered the vibration problem of the femur bone to determine the natural frequency and mode shape using FEA method. The FEA based computer is used to identify the fracture location of the bone using free-free and fixed-fixed boundary condition [5]. The effect of boundary condition on the natural frequencies of simply-supported beams is studied by Radice J.J [6]. Migbar Assefa Zeleke analyzed machine tool columns for bending, torsional and modal analysis. Also carried out the study of effect of the orientation and aspect ratio of apertures on the static and dynamic rigidity of the machine tool structure [7]. Ramu I et al. propose a finite element approach to compute the natural frequencies and mode shapes of the FGM plate with different boundary conditions and with different power law index values of rectangular FGM plates based on the Kirchhoff plate theory [8]. Seon M. Han et al analysed four models viz. Euler Bernoulli, Rayleigh, shear and Timoshenko for the transversely vibrating uniform beam. The frequency equations for four sets of end conditions: free-free, clamped-clamped, hinged-hinged and clamped-free are obtained. Hamilton's variational principle is used to obtain the equation of motion for each model, and the expressions for boundary condition [9].

It is important to prevent problems occurred in mechanical structure due to vibrations because it can cause structural fatigue and damage. Resonance is a key aspect in dynamic analysis, which may lead to catastrophic failure. The modal parameters are required to avoid occurrence of resonance in structures due to external excitations. Modal analysis is a very helpful contribution in control of many difficulties encountered in practice due to vibration. Modal analysis received wide acceptance in various fields of science, engineering and technology, it has practical applications like investigations related to aeronautical engineering, automobile engineering and mechanical engineering. In the present scenario a cantilever beam with rectangular cross section is considered. The present investigation reports the dynamic characteristics of common structural material i. e. mild steel.

In this paper modal characteristics and harmonic response of cantilever beam is analysed using ANSYS workbench. The modal analysis is performed to understand the dynamic behaviour of the structure which results in natural frequency and mode shape of the structure. The harmonic analysis is done to find the maximum response of structure at different frequencies by using ANSYS.

2. MATERIAL SPECIFICATIONS AND BEAM DIMENSIONS

Material Specifications:- Material selected for modal analysis of rectangular cantilever beam is Mild Steel. The material properties of the mild steel are as shown in Table 1.

Table -1: Material Properties of Mild Steel

Young's modulus of Mild Steel	E = 210 x 109 (N/mm2)
Poission's ratio of Mild Steel	v = 0.24
Density of Mild Steel	ρ = 7870 (Kg/m3)

Dimensions of the beam are taken as shown in Table 2:-

Table -2: Dimensions of Cantilever Beam

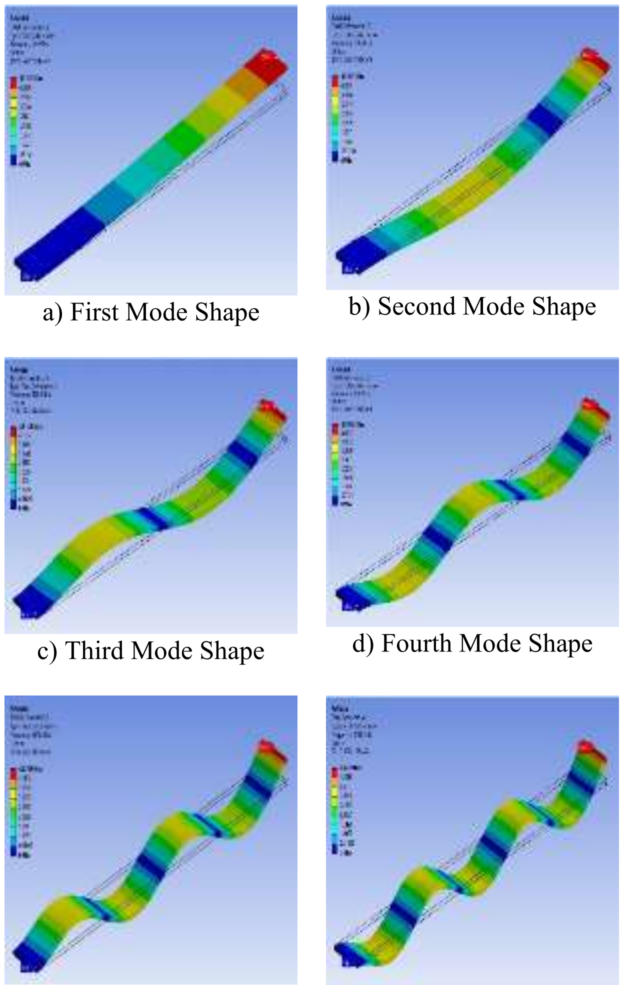
Length of beam	L = 288 mm
Width of beam	b = 29.07 mm
Thickness of beam	h = 2.86 mm

3. FINITE ELEMENT METHOD

Finite element techniques of analysis and simulation of mechanical systems is used to analyse the static and dynamic behaviour of the structural elements directly on the computer. The finite-element method is a powerful method of analysis that is used to provide approximations to solutions of static and dynamic problems for continuous systems. FEM divides a large problem into smaller and simpler parts, called finite elements. Using simple equations these finite elements are modelled and assembled into a larger system of equations that will simulate the entire problem. Efficient, general-purpose computer codes exist with suitable matrix assembler and equation solvers to calculate approximate solution for a specific problem.

3.1 Modal analysis by using ANSYS.

ANSYS workbench is used to find out natural frequencies and mode shape of cantilever beam in free fixed condition. Model of single rectangular cantilever beam is constructed in ANSYS and then computational modal analysis is performed to generate natural frequencies and mode shapes. Material properties as mentioned in table 1 are assigned to the model through material library. Boundary condition is applied by providing fixed support at one end of the beam, to calculate natural frequencies and mode shapes. Quadratic meshing type is used in order to split the beam into numerous elements. The number of elements and nodes generated are 4320 and 25037 respectively. Figure 1 shows the different mode shape obtained by modal analysis of a single rectangular cantilever beam in ANSYS.



a) First Mode Shape b) Second Mode Shape
c) Third Mode Shape d) Fourth Mode Shape
e) Fifth Mode Shape f) Sixth Mode Shape

Fig -1: Mode shapes of rectangular cross sectioned cantilever Beam obtained using ANSYS.

3.2 Harmonic response Analysis in ANSYS.

Harmonic response analysis is used to obtain the frequency response curves of a mechanical system which gives an idea of the peak response of the structure in frequency domain where the excitation is harmonic. The response of the structure is recorded at a given frequency range under a steady state sinusoidal (harmonic) loading. Figure 2 shows directional acceleration of X axis bode display of log Y type chart with viewing styles using average spatial resolution. Figure 3 represents directional acceleration of Y axis bode display of log Y type chart viewing styles using average spatial resolution. The maximum frequency considered for harmonic analysis in this work is 2505 Hz.

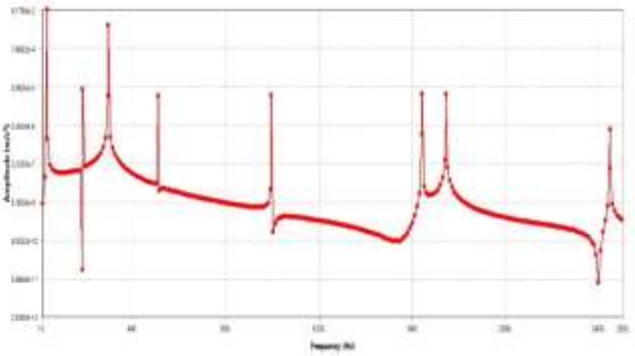


Fig -2: Directional acceleration of X axis orientation Bode display

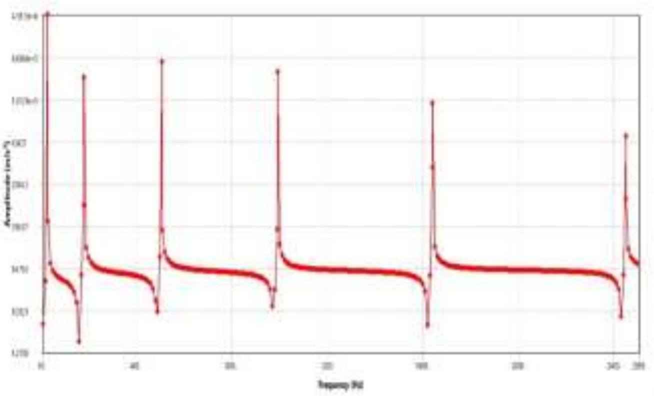


Fig -3: Directional acceleration of Y axis orientation Bode display

4. RESULT AND DISCUSSION

In this work, modal analysis of single rectangular cantilever beam has been carried out. Natural frequencies of single rectangular cantilever beam are determined by ANSYS software. Table 3 shows the natural frequencies and mode shapes obtained from FEA.

It can be observed that as the mode order increases, there is greater variation or error between natural frequency obtained by beam theory and natural frequency obtained by software. This may be due to the fact that as frequency is increased, range on which software have to operate rises. As FEA software works on approximate method which in turn increases the chances of error in the measured value over wide operating range.

Table -1: Natural frequencies obtained through modal analysis

Mode	ANSYS
1	28.89
2	180.94
3	506.51
4	992.4
5	1640.2
6	2449.3

Modal analysis is carried out with the application of the boundary conditions. Mode shapes at different output frequencies for cantilever beam are generated in ANSYS. The mode shapes generated in ANSYS for cantilever beam are presented in figure 1. Figure 1 show the different mode shapes with respect to the corresponding frequency of the beam. Minimum and maximum deflections at various frequencies are observed. The dialog box at the left side contains explicit values of deformations. The lowest value of the deformation was found at the bottom of the beam (Dark blue colour in figure1) while the maximum (Red colour in figure 1) is at the top of it. From these figures it can be observed that there is significant variation in mode shapes.

5. CONCLUSIONS

In this paper free vibration analysis of cantilever beam is carried out. Vibration characteristics of a mild steel cantilever beam are investigated. The modal analysis of a cantilever beam with a fixed-free boundary condition is performed. The natural frequencies are obtained using equation of motion and the boundary conditions. Mode shapes of structure at various natural frequencies are determined using finite element analysis technique. An attempt is made to understand the behaviour of the cantilever beam which is subjected to free vibration.

FEA is a suitable tool for predetermining natural frequencies for design of structure using active vibration control method which helps to modify design or apply some damping material to avoid resonance.

The creation and implementation of an FEA based model can accurately simulate the behaviour of a typical cantilever beam, as long as the necessary requirements (accurate model designs, material properties and boundary conditions) are provided. Boundary condition of the beam plays a very significant role in the vibration response of the beam.

REFERENCES

[1] ChakrabortyB C and Ratna Debdatta 2020 Polymers for Vibration Damping Applications (Elsevier)

[2] Inman D J 2014 Engineering Vibration 4th ed (Pearson)

[3] Sahu S, Choudhary B B, Biswal B B 2017 A vibration analysis of a 6 axis industrial robot using FEA*Materials Today: Proceedings 4 (2017) 2403–2410*

[4] Kumar A, Jaiswal H, Jain R Patil P 2014 Free vibration and material mechanical properties influence based frequency and mode shape analysis of transmission gearbox casing *Procedia Engineering 97 (2014) 1097 – 1106*

[5] Kumar A, Jaiswal H, Garg T, Patil P 2014 Free vibration modes analysis of femur bone fracture using varying boundary conditions based on FEA *Procedia Materials Science 6 (2014) 1593 – 1599*

[6] Radice J 2012 On the effect of local boundary condition details on the natural frequencies of simply-supported beams: Eccentric pin supports *Mechanics Research Communications 39 (2012) 1– 8*

[7] Zeleke M 2013 Modal analysis of machine tool column using finite element method*Int. J. of Mech., Ind. Sc. and Engg. Vol:7 No:4, 2013*

[8] Ramu I, Mohanty S2014 Modal analysis of functionally graded material plates using finite element method *Procedia Materials Science 6 (2014) 460 – 467*

[9] Han S, Benaroya H and Wei T 1999 Dynamics of transversely vibrating beams using four engineering theories *Journal of Sound and vibration (1999) 225(5), 935-988*