International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347-6435(Online) Volume 4, Issue 1, January 2015)

Modal Analysis of Cantilever Beam with T-Section using Finite Element Method

Sampath S S[1], Sawan Shetty[2], Chithirai Pon Selvan M[3]

School of Engineering & Information Technology, Manipal University - Dubai, United Arab Emirates

Abstract— **This paper deals with the vibration characteristics of a beam. A modal analysis is carried out on a cantilever beam with T section. The beam is modeled, designed and analyzed in in finite element analysis software ANSYS. The cantilever beam which is fixed at one end is vibrated to obtain the nature of vibration. All the degrees of freedom on the free end are taken into account. Modes and the natural frequencies pertaining to it are computed in finite element analysis software.**

Keywords— **Cantilever beam, ANSYS, modal analysis, degrees of freedom.**

I. INTRODUCTION

Vibration can be easily defined as an oscillation which is the analogous to the motion of the particles of a mass (of air or the like) whose state of equilibrium has been disturbed [2]. In engineering field vibration behavior of an element plays a key role without which it is incomplete. Resonance is a key aspect in dynamic analysis, which is the frequency of any system matches with the natural frequency of the system which may lead to catastrophes or system failure. Modal analysis has become a major alternative to provide a helpful contribution in understanding control of many vibration phenomena which encountered in practice [2]. Determining the nature and extent of vibration response levels and verifying theoretical models and prediction are both major objectives. Vibration in the system is some time desired and most of the time it's undesired. If the resultant forces in the system are equal to zero it represents equilibrium condition and when there is unbalanced force exists it leads to vibration. If the motion is described fully by only one time-dependent coordinate, such a system is termed a 1-degree-of-freedom (DOF) vibration model. When more than one coordinate becomes necessary, the discrete system is said to have multiple degrees-of-freedom. In a single degree of freedom system there exist a single mode and the one natural frequency. But, in multi degree of freedom system there exist more than one natural frequency which will make the system to vibrate in various modes.

Problems are often occurred in mechanical structure due to vibrations. It is important to prevent such problems because it can cause structural fatigue and damage [5]. The structure itself has a certain internal properties and it is important to understand its characteristics. In order to do it, the first important thing to do is modal analysis for data acquisition [9]. The main purpose of modal analysis is to obtain the data that will be used for other vibrational analysis. Practical applications the modal parameters are required to avoid resonance in structures affected by external periodic dynamic loads. There are various practical applications of modal analysis over various fields of science, engineering and technology like investigations related to aeronautical engineering, automobile engineering and mechanical engineering. The experimental modal analysis received wide acceptance in structural engineering application, particularly for identification of modal properties of bridges, damage detection of structures using modal data and structural health monitoring, dynamic finite element method (FEM) updating of structures etc. The present investigation reports the dynamic characteristics of common structural materials [6].

In the present scenario a cantilever beam with T section is considered. T-shaped cross section serves as a flange or compression member in resisting compressive stresses. The T-beam has a disadvantage compared to an I-beam because it has no bottom flange with which to deal with tensile forces. One way to make a T-beam more efficient structurally is to use an inverted T-beam with a floor slab or bridge deck joining the tops of the beams.

The vibration taking place in the beam is determined and also its nature of vibration is found. Modal analysis of the beam is carried out using Finite element software. Both modelling and analysis is carried out using ANSYS [1]. Meshing of the element is done in order to split into numerous elements. After meshing the boundary conditions are defined to determine various modes [2]. Different frequency of vibration is determined. Conventional equations which govern the element are used.

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347-6435(Online) Volume 4, Issue 1, January 2015)

In the present case a T section length of 10 m long and having the cross-sectional area of 50 m2 is considered. One end of the beam is fixed and the system is subjected to vibration. The material which is assumed is titanium with the density 4500 Kg/m^3.

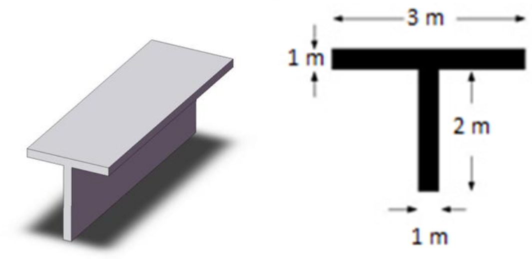

Fig 1: T Section with dimensions

The following element is modelled, meshed and analysed in software ANSYS CFX and the results and compared. Finite element approach provides the solution at every node of consideration [4].

The present problem was structured as a sequence of fundamental problems built on simple models that determine structural property of the element under study. The models proceed from the simple toward the complex. The objective is to uncover the most fundamental optimization principles (or design trade-offs) that can be put to practical use in real applications. The method of analysis and optimization is the combination of vibration analysis & structures which are used subsequently in many engineering applications [5].

II. Related Work

Pavol Lengvarský et.al [1] considered cantilever beam and various degrees of freedom on this end were taken into account, beam cannot move and rotate. Mode shapes and natural frequencies are computed in programs ANSYS and SolidWorks. Chandradeep Kumar et.al [2] determined the nature and extent of vibration response levels and verified the theoretical models with the analysis software. Vijaykumar et.al [3] carried out modal analysis on the tungsten cantilever beam. Mode shapes and natural frequencies are computed in programs ANSYS and Solid Works with numerical formulation of the direct solver including the block Lanczos method. [5] Chao Ming Ching and Slamet Widodo [5] performed modal analysis for steel cantilever beam has been done in this experiment by using ME'scopeVES modal analysis software.

The analysis results obtained from experiment are then compared with the theoretical results and also the results obtained by using other approach, in this case using finite element method (FEM) that is provided by ANSYS software. The error between theoretical and experimental results is about 2.91%. It is also in agreement with the results obtained from finite element method analysis.

III. Methodology

When the system is subjected to vibration it need not vibrate in a single mode, as the increase in the frequency the amplitude and modes of vibration will vary. The general equation of vibration shown in equation (1) is given in the matrix form, in which all the forces are equated. If the overall sum of the forces is equal to zero then the system is said to be in equilibrium condition. A beam is fixed at one end and the other end is simply supported the fundamental frequency is given by an equation (2). Table 1 shows the variation of radius of gyration to determine the fundamental frequency for various modes.

General equation of vibration

$$[M]\{\ddot{x}\} + [C]\{\dot{x}\} + [K]\{x\} = \{f\} \qquad \text{---- (1)}$$

Fundamental frequency of vibration for a cantilever beam

$$f = \frac{K}{2\,\pi\,L^2}\sqrt{E.I / m} \qquad \text{---- (2)}$$

To determine the fundamental frequency for different modes with the variation of radius of gyration.

Table 1:
Frequency calculations with the variation of radiation of gyration

Mode	1	2	3	4
K	15.4	50	104	178

IV. Modelling And Analysis

Current study involves geometric modelling of the beam using ANSYS CFX and its simulation is performed using the same [1]. Beam is subjected to vibrations and the behavior is studied in the analysis platform. As the intensity of vibration increases, system will start vibrating in more than one degree of freedom. Modeling of the beam is done according to the defined dimensions in ANSYS.

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347-6435(Online) Volume 4, Issue 1, January 2015)

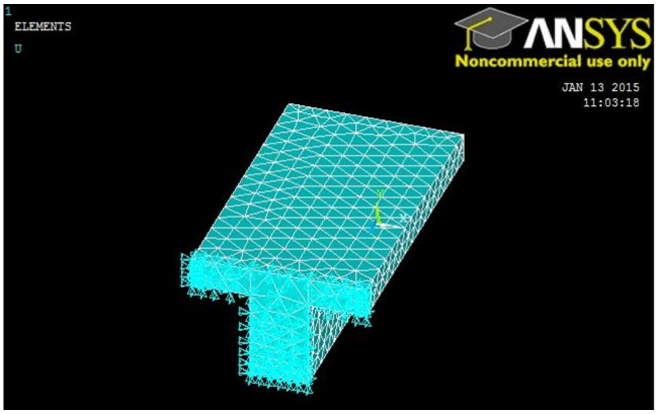

Fig 2: Meshing of the T cross sectional beam

Meshing is discretizing of an element into finite number of parts and each element is considered and solved separately [2]. Mesh generation is the practice of generating a polygonal or polyhedral mesh that approximates a geometric domain. The term "grid generation" is oftenly used interchangeably. Typical uses are for rendering to a computer screen or for physical simulation such as finite element analysis or computational fluid dynamics. After this step a one end is made to fix before the application of vibration.

Since Titanium is considered as the material for the beam, density of the material is 4500 kg/m3. Modeling and Meshing is done using FEA and the simulation is performed. Behavior of the beam is determined by selecting different modes and the frequency corresponding to the mode is noted down [1, 2].

By means of the numerical solution, vibration analysis of the entire element is achieved [1].Validation of the results obtained in the FEA is compared with the classical equation. It can also be compared with the modal analysis obtained for cantilever beam [1, 2 and 5].

V. RESULTS AND DISCUSSION

The simulation is carried out with the application of the boundary conditions. Analysis is carried out and the mode change with respect to the corresponding frequency is noted down. Figure 3 to figure 7 show the deformations, minimum and maximum deflections at various frequencies. Table 2 represents the various frequencies at different modes.

Table 2:
Fundamental frequencies at different modes in ANSYS

***** INDEX OF DATA SETS ON RESULTS FILE *****

SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	38.374	1	1	1
2	40.794	1	2	2
3	68.234	1	3	3
4	149.52	1	4	4
5	158.91	1	5	5

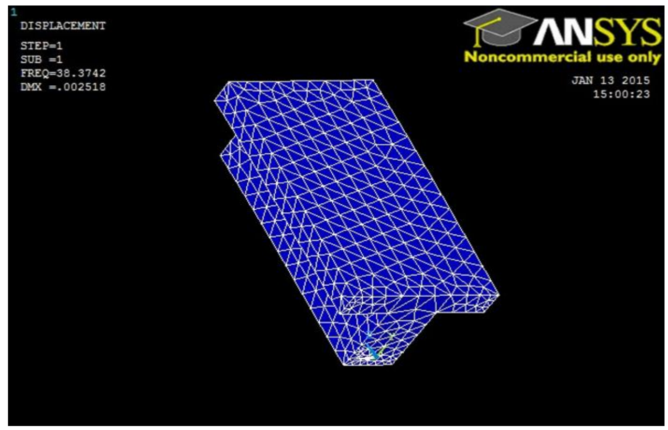

Fig 3 (a): Maximum deflection in mode 1(Frequency= 38.374 Hz)

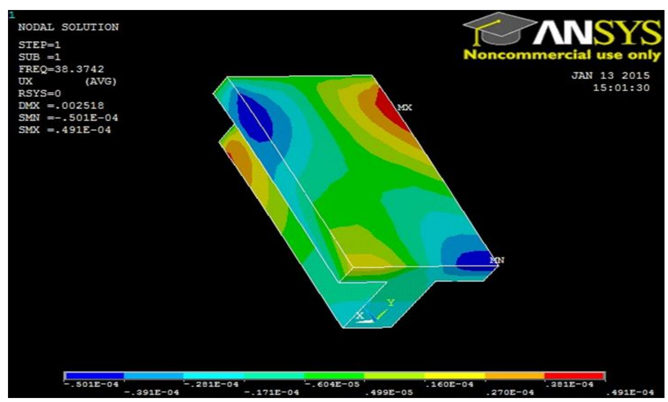

Fig 3 (b): Minimum and maximum result value in in mode (Frequency= 38.374 Hz)

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347-6435(Online) Volume 4, Issue 1, January 2015)

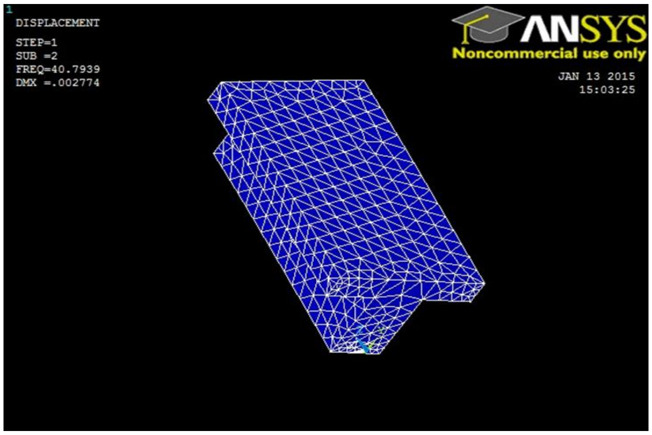

Fig 4 (a): maximum deflection in mode 2 (Frequency = 40.794 Hz)

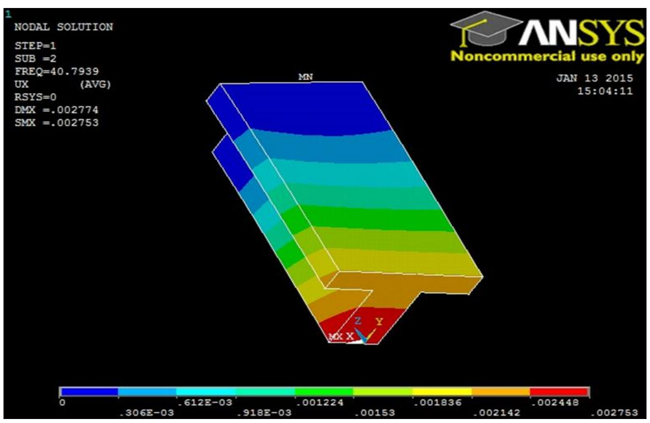

Fig 4 (b): Minimum and maximum result value in in mode 2 (Frequency = 40.794 Hz)

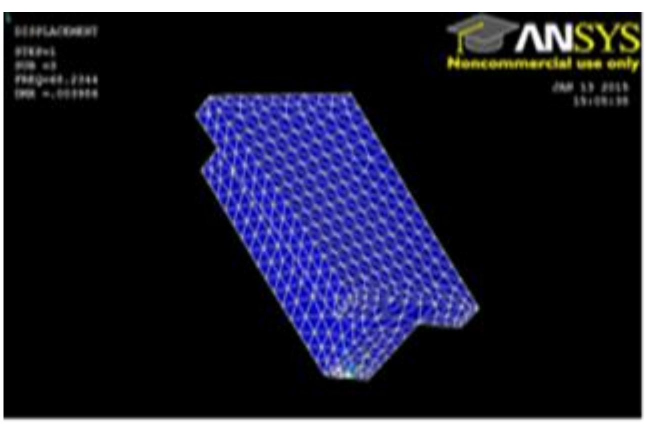

Fig 5(a): maximum deflection in mode 3(Frequency= 68.234 Hz)

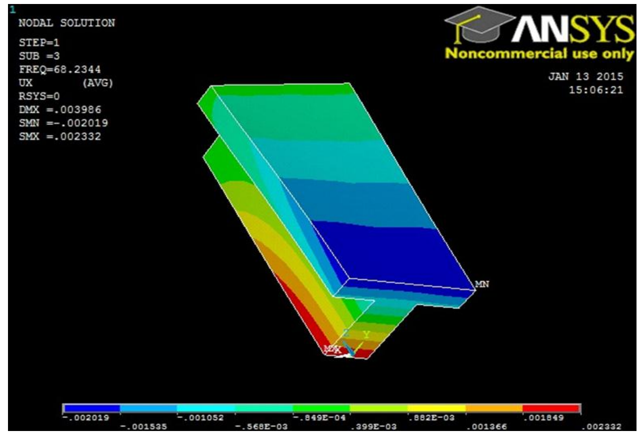

Fig 5 (b): Minimum and maximum result value in in mode 3 (Frequency= 68.234 Hz)

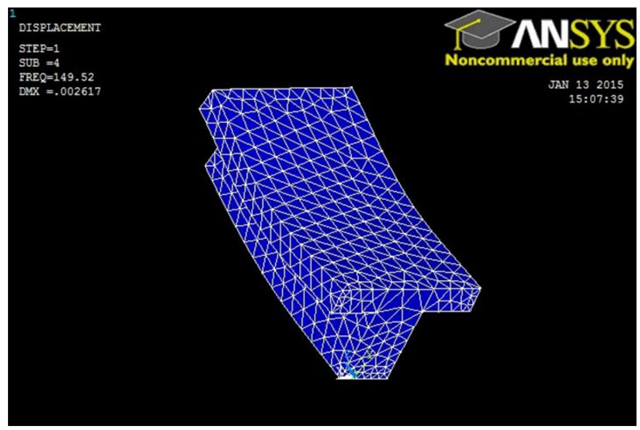

Fig 6 (a) : maximum deflection in mode 4(Frequency= 149.52 Hz)

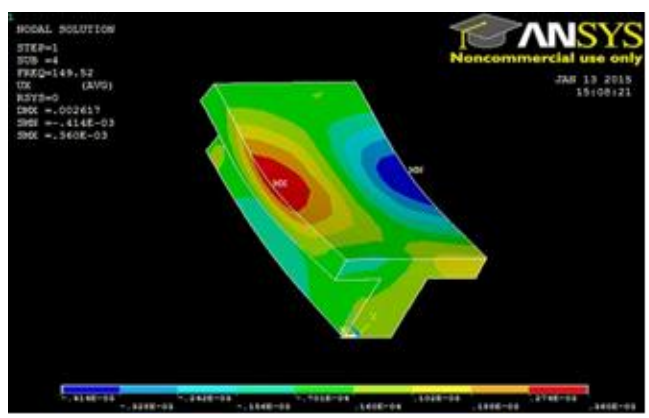

Fig 6(b): Minimum and maximum result value in in mode 4 (Frequency= 149.52 Hz)

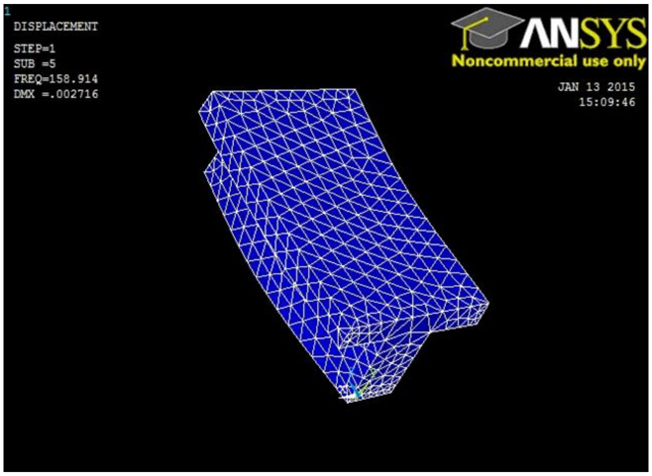

Fig 7(a): maximum deflection in mode 5(Frequency= 149.91 Hz)

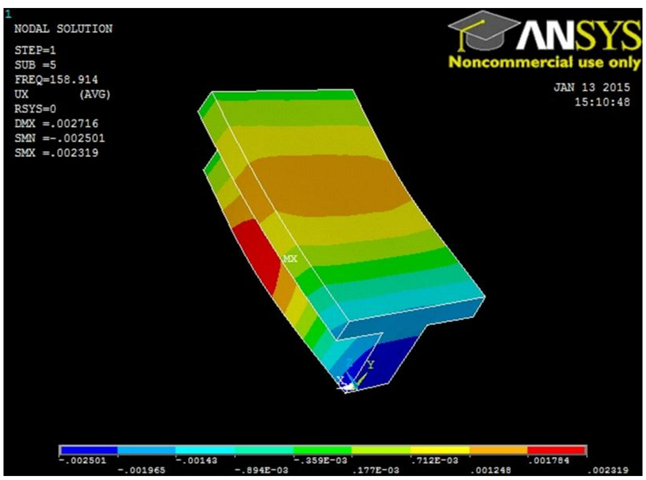

Fig 7 (b): Minimum and maximum result value in in mode 4 (Frequency= 149.91 Hz)

Figures 3 (a), 4 (a), 5 (a), 6 (a) and 7 (a) represent the maximum deformations in various modes. Figures 3 (b), 4 (b), 5 (b), 6 (b) and 7 (b), represent the minimum and maximum result values in different modes. Maximum deformation region determines the tension and the minimum deformation region identifies the compression regions in different phase of vibration [1, 2].

Figure 8 explains how the frequency of vibration rises non linearly which leads to the increase in the number of modes.

Table 3:
Fundamental frequency variation with the number of modes

Modes of vibrations	Frequency (Hz)
1	38.374
2	40.794
3	68.234
4	149.52
5	158.91

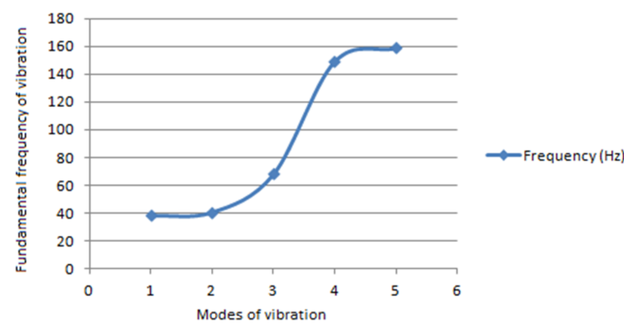

Fig 8: Fundamental frequency variation with the number of modes

Table 4:
Maximum deformation due to the rise in Fundamental frequency

Fundamental Frequency (Hz)	Maximum deformation (m)
38.374	0.002518
40.794	0.002774
68.234	0.003986
149.52	0.002617
158.91	0.002716

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347-6435(Online) Volume 4, Issue 1, January 2015)

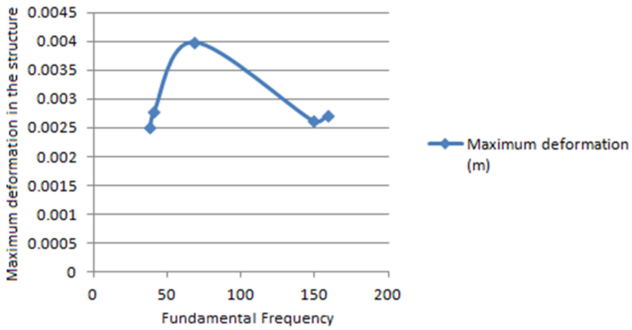

Fig 9: Maximum deformation variation with the fundamental frequency (No of Modes)

From the figure 9, it can be concluded that as there is rise in the vibration level the degrees of freedom increases in which it leads to the deformation of the structure [1]. In the present case, for the T section, the maximum deformation rose till the 68 Hz and there was a sudden drop in the curve where the element experiences reasonably less deformation. At a frequency of 158 Hz, there is again rise in the maximum deformation. Overall nature of the curve is a sinusoidal which may again rise to a particular point of frequency and again drop.

VI. CONCLUSIONS

In the present analysis, vibration characteristics of a T sectional cantilever beam are investigated. Modes of vibration at various frequencies are determined using finite element analysis technique. An attempt is made to understanding the behaviour of the beam which is subjected to different degree of vibration. Study shows that, there is a rise and fall in the maximum deflection values with the increase in the modes which means material will be continuously under tension and compression. If the deformation is very minimum and in the elastic limit, it can return to its original position, but if it exceeds the limit of elasticity there will be a permanent deformation. On repetition of this nature of vibration leads to a failure. Therefore material selection is significant to keep the elongations within the limit. It is possible to obtain an optimum solution by selecting a material which has better structural performances. Further analysis can be carried out by changing the dimensions of T section or selecting beams of different sections. Analysis of the curved beams can also be carried out in the similar manner and the results can be compared with the current scenario. Analysis can be further carried for more than 5 modes and the behaviour of the beam can be determined

Nomenclature

M= Mass of the element, kg

C= Damping coefficient, Ns/m

k= Stiffness, N/m

x= Displacement or deflection, m

$\dot{x}$= Velocity, m/s

$\ddot{x}$ =Acceleration, m/s^2

f= Fundamental frequency, Hz

K= Radius of gyration, m

L= Length of the element, m

E= Young's Modulus of the material, N/m^2

I= Moment of inertia, m^4

REFERENCES

[1] Pavol Lengvarský, Jozef Bocko, Martin Hagara, "Modal Analysis of Titan Cantilever Beam Using ANSYS and SolidWorks", American Journal of Mechanical Engineering, 2013, Vol. 1, No. 7, PP: 271-275.

[2] Chandradeep Kumar, Anjani Kumar Singh, Nitesh Kumar, Ajit Kumar, "Model Analysis and Harmonic Analysis of Cantilever Beam by ANSYS" Global journal for research analysis, 2014, Volume-3, Issue-9, PP:51-55.

[3] K.vijaykumar , G.shivaraju , U.sreekanth , Mr.aluri david ," Tungsten Cantilever Beam Using Ansys (Modal Analysis), The International Journal Of Engineering And Science. 2014, Volume 3, issue 4, PP: 53-59.

[4] Mohammad Vaziri, Ali Vaziri, Prof. S.S. Kadam, "Vibration analysis of a cantilever beam using F.F.T analyzer", International Journal of Advanced Engineering Technology, 2013, volume 4, Issue 2, PP: 112-115.

[5] Chao Ming Ching and Slamet Widodo, "Modal Testing and Analysis of Cantilever Beam"PP: 1-14.

[6] Pragnesh K. Chaudhari, Dipal Patel, Vipul Patel, "Theoretical and Software Based Comparison of Cantilever Beam: MODAL ANALYSIS" International Journal of Innovative Research in Advanced Engineering, 2014, volume 1, issue 5, PP: 75-79.

[7] Nitin N More, "Finite Element Analysis of Piezoelectric Cantilever", International Journal of Innovations in Engineering and Technology, 2013, volume 2, issue 3, PP: 100-105.

[8] Adik R. Yadao, Irshad A. Khan, Dayal R.Parhi, "Modal Analysis of a Multiple Cracked Cantilever Bar", IOSR Journal of Mechanical and Civil Engineering, 2014, PP:12-15.

[9] Migbar Assefa, "Modal Analysis of Machine Tool Column Using Finite Element Method", International Journal of Mechanical, Aerospace, Industrial and Mechatronics Engineering, 2013, Volume 7, No 4, PP: 278-287.

[10] R. S. Mohan, A. Sarkar, A. S. Sekhar, "Vibration analysis of a steam turbine blade", Inter-noise, 2014, PP: 1-10.