MODELLING, SIMULATION AND ANALYSIS OF CANTILEVER BEAM OF DIFFERENT MATERIAL BY FINITE ELEMENT METHOD, ANSYS & MATLAB

Rishi Raj[*], Prabhat Kumar Sinha, Earnest Vinay Prakash
*Department of Mechanical Engineering, Shepherd School of Engineering and Technology,
Sam Higginbottom Institute of Agriculture Technology and Sciences,
Deemed University Allahabad (U.P.) INDIA.
Contact No.:+91-8005092200, Email: rishiraj.exe@gmail.com*

ABSTRACT - The dynamic analysis of a beam with multiple degree of freedom (MDOF) are studied in this paper. Due to the destructive effects of vibration in machines and structures due to resonance. In multiple degree of freedom system, there are n natural frequencies and the concept of resonance is complicated by the effect of mode shapes. In the present work cantilever beam of different materials and dimensions is considered for the dynamic analysis of free vibration at no load condition as well as comparison between materials. The modelling, simulation and analysis of cantilever beam is done by using ANSYS & MATLAB and theoretically by finite element method (FEM) for the evaluation of natural frequency and mode shape. By using Lagrange's equation, the formulation of equation motion for the beam is derived, through which stiffness and mass matrix is obtained. Eigen value approach is used for the calculation of natural frequency and mode shape. The frequency response function (FRF) are obtained by using MATLAB & ANSYS.
Keywords: Vibration analysis, MDOF, ANSYS, MATLAB, FEM, FRF, mode shape.

INTRODUCTION

Beams and beam like elements are main constituent of structures and widely used in aerospace, high speed machinery, light weight structure, etc and experience a wide variety of static and dynamic loads of certain frequency of vibration which leads to its failure due to resonance. Vibration testing has become a standard procedure in design and development of most engineering systems.[10] The system under free vibration will vibrate at one or more of its natural frequencies, which is the characteristic of the dynamical nature of system. The continuous system has multiple degree of freedom with multiple natural frequencies.[9][7][12] The natural frequency is independent of damping force because the effect of damping on natural frequency is very small. An n-degree of freedom system is governed by n coupled differential equation and has n natural frequency.[9][12] **Jacob Bernoulli** (1654-1705) revealed that the curvature of an elastic beam at any point is relational to the bending moment at that point. Later this theory was acknowledged by **Leonhard Euler** (1707-1783) in his investigation of the shape of elastic beams subjected to various loading conditions. The Euler-Bernoulli beam theory is the most commonly used because it is simple and provides realistic engineering approximations for many problems.[11] **Timoshenko** (1921-1922) suggested a beam theory which adds the effect of Shear as well as the effect of rotation to the Euler-Bernoulli beam. The Timoshenko model is a major enhancement for non-slender beams and for high-frequency responses and deduced the frequency equations and the mode shapes for various boundary conditions.[4] The improvement of finite element method is traced back by **Alexander Hrennikoff** (1941) and **Richard Courant** (1942) they stick to one essential characteristic: mesh discretization of a continuous domain into a set of discrete subdomains, usually called elements. **Olgierd Zienkiewicz** (1947) collected those methods together into what is called the Finite Element Method.[8] **Gurgoze, H.Erol** Frequency response function is the quantitative measure of the output spectrum of a system or device in response to a stimulus, and is used to characterize the dynamics of the system.[5] **D.Ravi Prasad** Modal analysis is a process of describing a structure in terms of its natural characteristics which are the frequency, damping and mode shapes –its dynamic properties. The change of modal characteristics directly provides an indication of structural condition based on changes in frequencies and mode shapes of vibration.[2]

FORMULATION

Lagrange's equation [6] to formulate equations of motion are given by

$$\frac{d}{dt}\left(\frac{dT}{d\dot{q}_i}\right) - \frac{\partial T}{\partial q_i} + \frac{\partial A}{\partial q_i} = Q_i \qquad (i = 1, 2, 3, \ldots\ldots\ldots\ldots n) \qquad \ldots\ldots\ldots(1)$$

Consider a single degree of freedom system that requires only one coordinate q to describe its behavior. To apply it in Eq.(1) to express the kinetic and potential energies of the system in terms of coordinate q and its derivative q˙. Let q=x then,

$$T = \frac{1}{2}m\dot{x}^2$$

$$A = \frac{1}{2}kx^2$$

Now, differentiating and substituting in Eq.(1).We get

$$\frac{d}{dt}\left(\frac{\partial T}{\partial q_i}\right) - \frac{\partial T}{\partial q_i} + \frac{\partial A}{\partial q_i} = Q_i$$

The governing differential equation of motion is given by

$mx^{..} + kx = 0$

Consider a system with two degree of freedom, consequently we need two coordinates x_1 and x_2 to formulate kinetic and potential energies,

$$T = \frac{1}{2}m_1 x_1^{.2} + m_2 x_2^{.2}$$

$$A = \frac{1}{2}k_1 x_1^2 + \frac{1}{2}k_2(x_2 - x_1)^2$$

Differentiating and substituting in Eq.(1).We get

$m_1 x_1^{..} + (k_1 + k_2)x_1 - k_2 x_2 = 0$

$m_2 x_2^{..} - k_2 x_1 + k_2 x_2 = 0$

We can express equation of motion in a matrix form by

$$\begin{bmatrix} m_1 & 0 \\ 0 & m_2 \end{bmatrix}\begin{Bmatrix} x_1^{..} \\ x_2^{..} \end{Bmatrix} + \begin{bmatrix} k_1 + k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix}\begin{Bmatrix} x_1 \\ x_2 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix}$$

Formulating mass matrix

The displacement of an axial member is expressed by using one-dimensional shape function S_i and S_j,

$$u = S_i U_i + S_j U_j \qquad\qquad \dots\dots\dots.(2)$$

For a dynamic problem the displacement function is a function of x and time t, i.e. u=u(x,t). The total kinetic energy of the member is the sum of the kinetic energies of its constituent particles

$$T = \int_0^L \frac{\gamma}{2}\dot{u}^2 dx \qquad\qquad \dots\dots\dots.(3)$$

The velocity of the member is expressed in terms of $U_i \ and \ U_j$ is given by

$$\dot{u} = S_i U_i + S_j U_j \qquad\qquad \dots\dots\dots.(4)$$

Substituting Eq. (4) into Eq. (3), we get

$$T = \frac{\gamma}{2}\int_0^L (S_i U_i + S_j U_j)^2 \, dx \qquad\qquad \dots\dots\dots.(5)$$

After taking derivatives as required by Lagrange's Equation (1). We get,

$$\frac{d}{dt}\left(\frac{\partial T}{\partial \dot{u}_j}\right) = \gamma\left[\int_0^L S_j^2 U_i^{..} dx + \int_0^L S_i S_j U_j^{..} \, dx\right] \qquad \dots\dots\dots.(6)$$

$$\frac{d}{dt}\left(\frac{\partial T}{\partial U_j}\right) = \gamma\left[\int_0^L S_i S_j U_i^{..} dx + \int_0^L S_j^2 U_j^{..} dx\right] \qquad \dots\dots\dots.(7)$$

Evaluating the integrals of Eq.(8) & Eq.(9). We get,

$$\gamma\int_0^L S_i^2 dx = \gamma\int_0^L \left(1 - \frac{x}{L}\right)^2 dx = \frac{\gamma L}{3} \qquad\qquad \dots\dots\dots.(8)$$

$$\gamma\int_0^L S_j^2 \, dx = \gamma\int_0^L \left(\frac{x}{L}\right)^2 dx = \frac{\gamma L}{3} \qquad\qquad \dots\dots\dots.(9)$$

Solving Eq. (8) through Eq.(9) into Eq.(6) and Eq.(7) gives the mass matrix,

$$[M] = \frac{\gamma L}{6}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix}$$

Formulating Stiffness Matrix

During the deformation the member under axial loading, the strain energy stored is given by

$$A^{(e)} = \int_v \frac{\sigma\varepsilon}{2}dV = \int_v \frac{E\varepsilon^2}{2}dV \qquad\qquad \dots\dots\dots.(10)$$

Now, the total potential energy Π for the body contains n elements and m modes is given by

$$\Pi = \sum_{e=1}^n A^{(e)} - \sum_{i=1}^m F_i u_i \qquad\qquad \dots\dots\dots\dots(11)$$

The deflection for an element with nodes I & j in terms of local shape function is given by,

$$u^{(e)} = S_i u_i + S_j u_j \qquad\qquad \dots\dots\dots\dots.(12)$$

The strain in each member is calculated by

$$\varepsilon = \frac{-u_i + u_j}{\varrho} \qquad \qquad \dots\dots\dots(13)$$

Substituting Eq.(13) into Eq.(10) yields the strain energy for an arbitrary element (e) and minimizing w.r.t. u_i and u_j leads in matrix form

$$\begin{Bmatrix} \dfrac{\partial A^{(e)}}{\partial u_i} \\ \dfrac{\partial A^{(e)}}{\partial u_j} \end{Bmatrix} = \frac{AE}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix} \qquad \qquad \dots\dots\dots\dots(14)$$

Eq. (14) is Stiffness matrix

$$[K] = \frac{AE}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

Solving mass and stiffness matrix for eigen values by using the formula
$$[M]^{-1}[K]\{U\} = \omega^2\{U\}$$

Material and Beam Specification

| Material	Specimen	Dimension
1. Steel Alloy	SM-1	(0.6m X 0.03m X 0.008m)
I. Modulus of Elasticity, E = 210 GPa	SM-2	(0.6m X 0.03m X 0.004m)
II. Density, ρ = 8030 Kg/m^3	SM-3	(0.42m X 0.03m X 0.008m)
III. Poisson's Ratio, ν = 0.30	SM-4	(0.42m X 0.03m X 0.004m)
2. Carbon Fiber Reinforced Plastic	SM-5	(0.6m X 0.03m X 0.008m)
I. Modulus of Elasticity, E = 220 GPa	SM-6	(0.6m X 0.03m X 0.004m)
II. Density, ρ = 1720 Kg/m^3	SM-7	(0.42m X 0.03m X 0.008m)
III. Poisson's Ratio, ν = 0.33	SM-8	(0.42m X 0.03m X 0.004m)

Modelling Analysis

ANSYS, Inc. is an engineering modelling and simulation software that offers engineering simulation solution sets in engineering simulation that a design process requires.[1] Here, we are using ANSYS WORKBENCH 14.0 in which modelling of beam is done in geometry component system, material is selected from engineering data library and simulation & analysis is performed in modal analysis system from where we obtained natural frequency and mode shapes for all specimens of both materials.
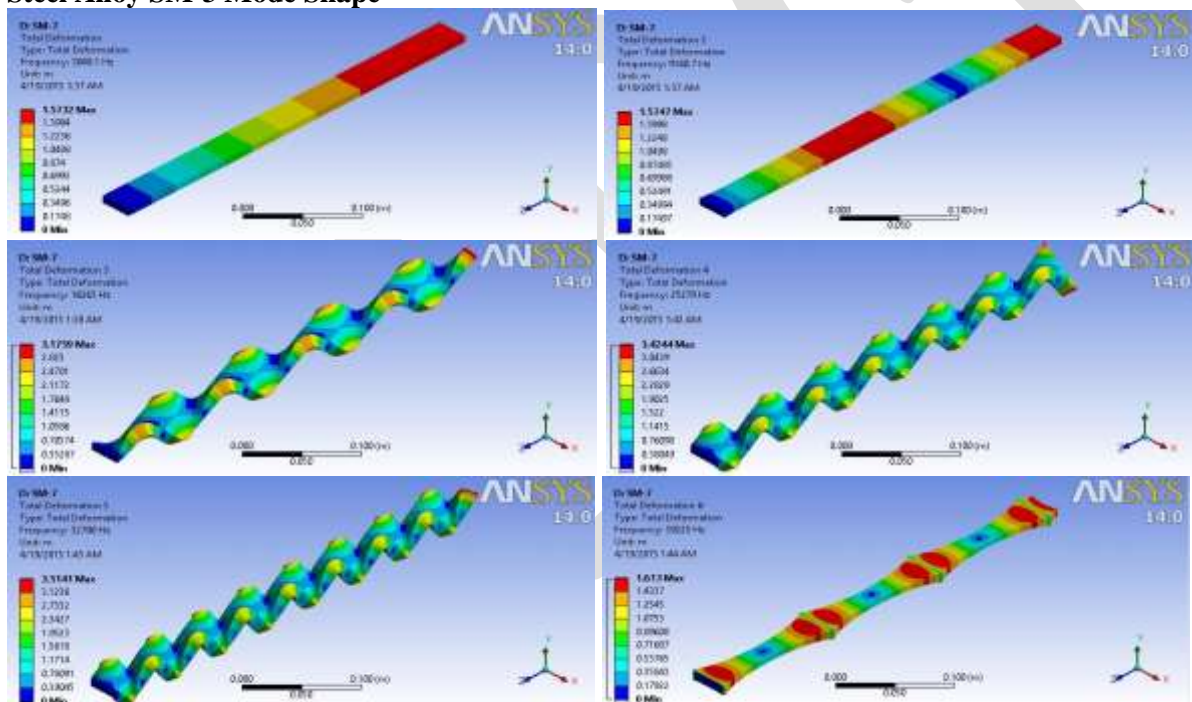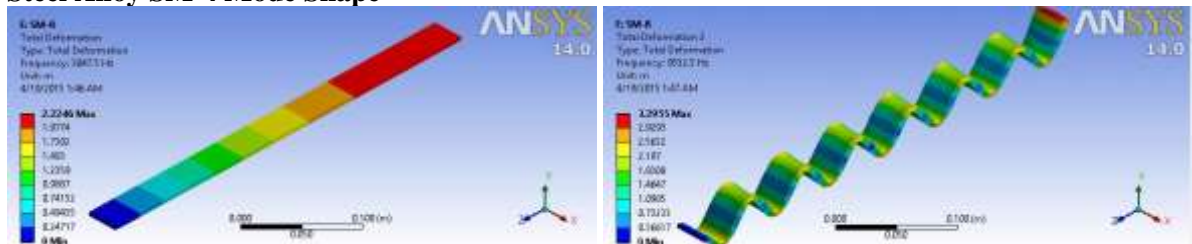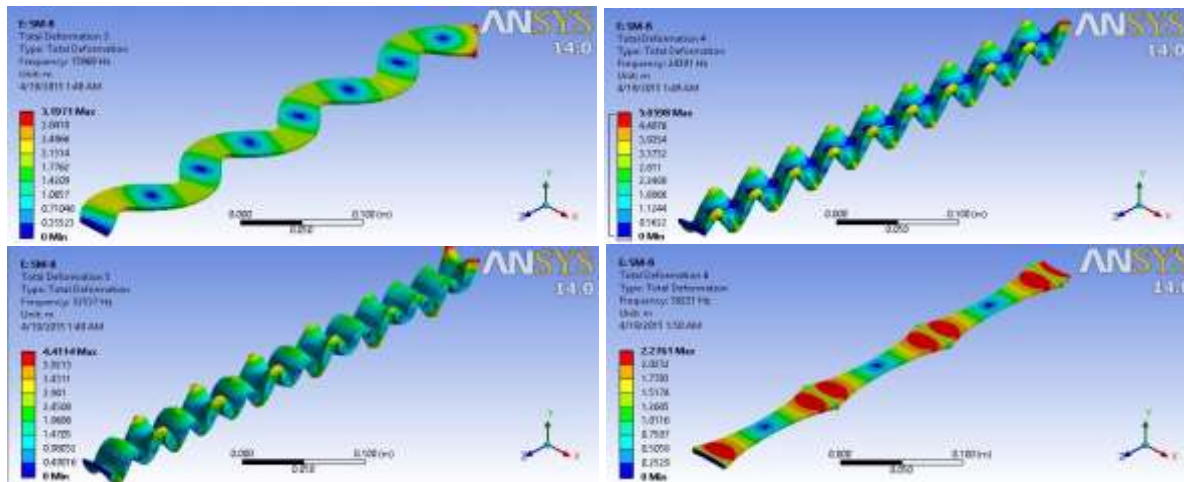
Steel Alloy SM-1 Mode Shape
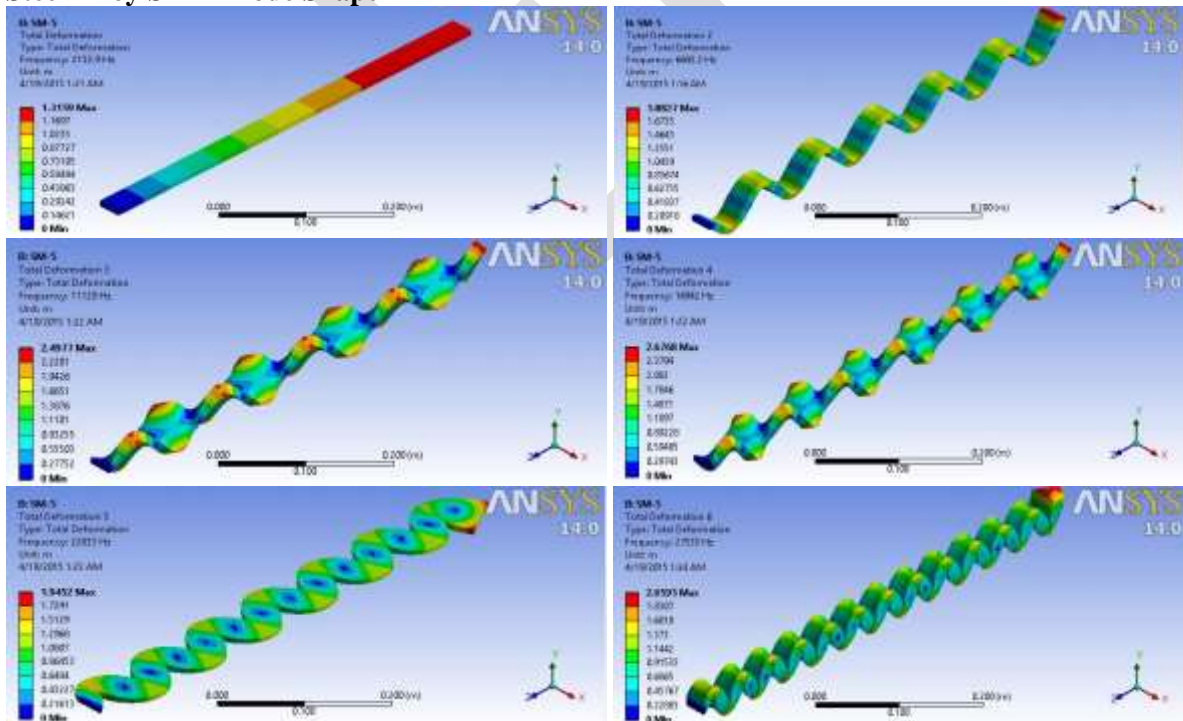
Steel Alloy SM-2 Mode Shape
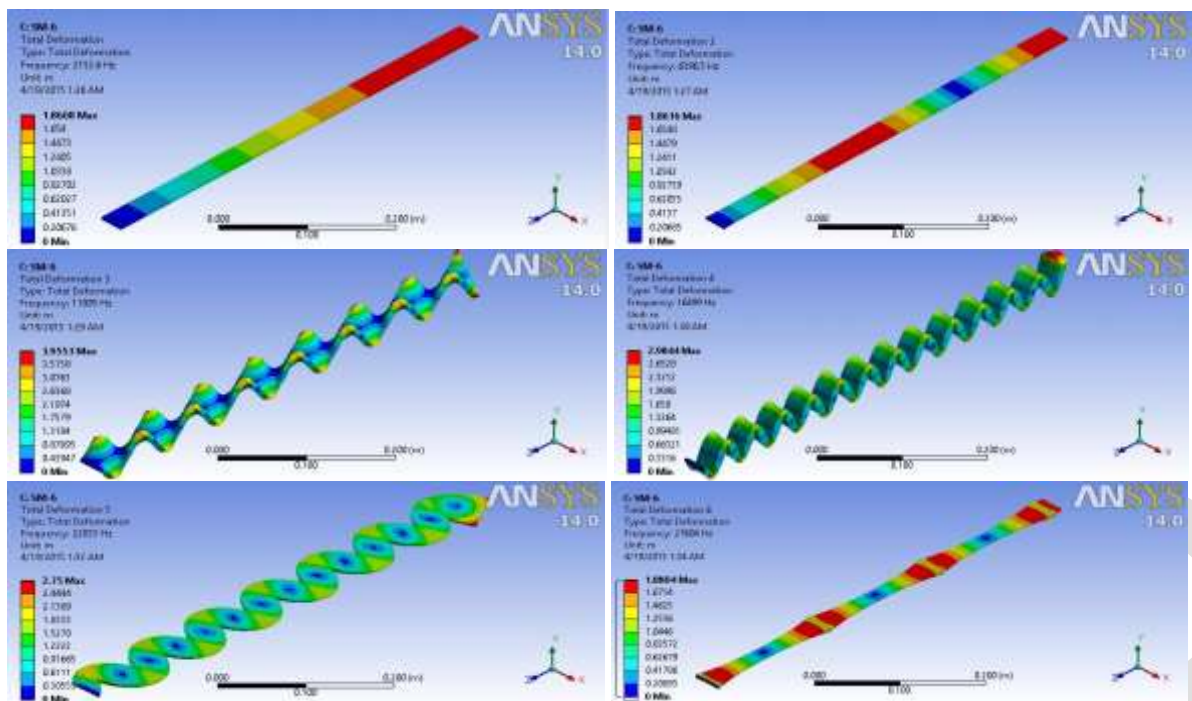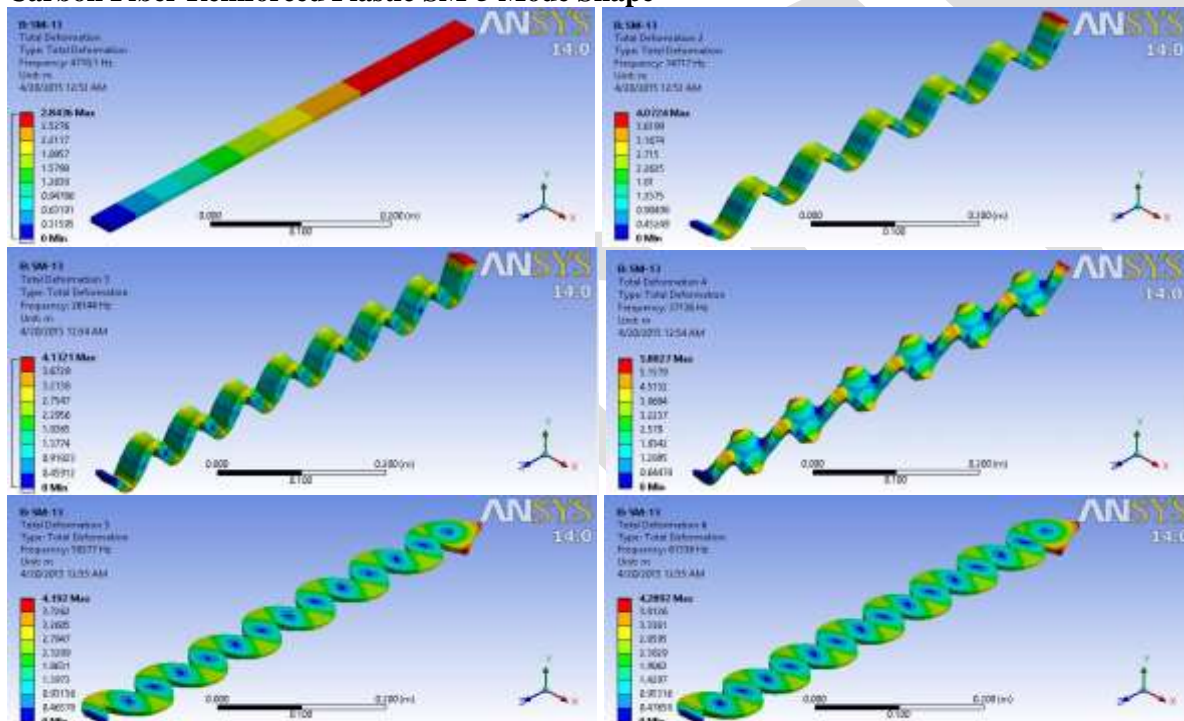
Steel Alloy SM-3 Mode Shape
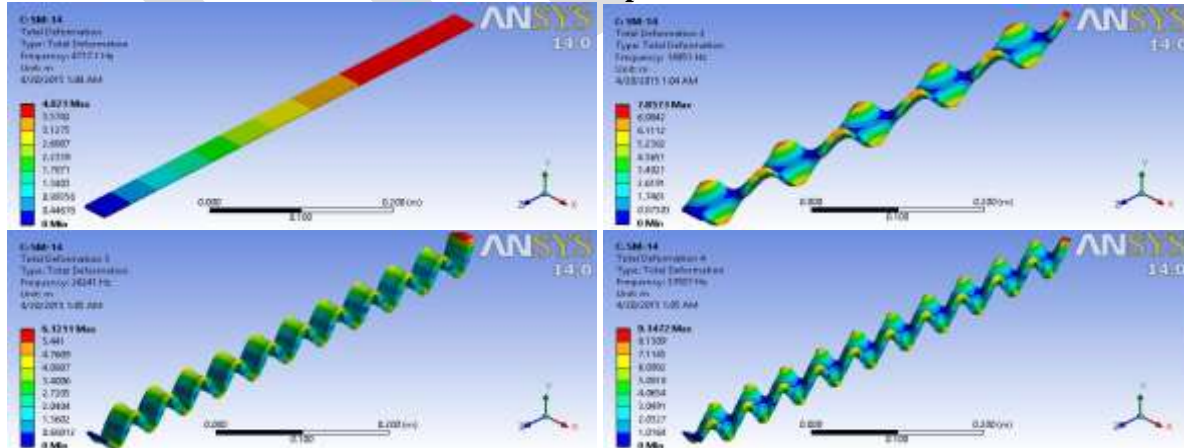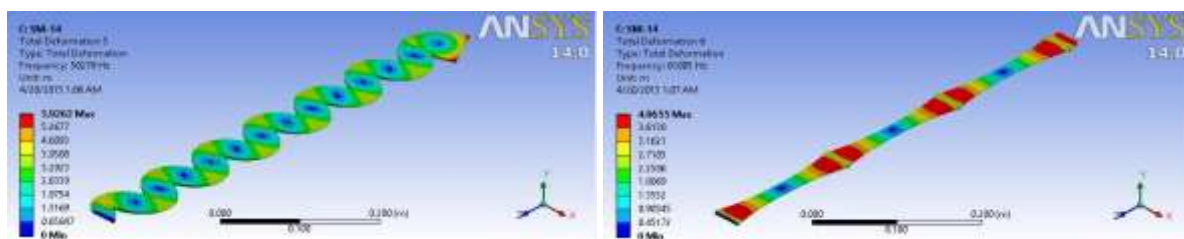

Steel Alloy SM-4 Mode Shape

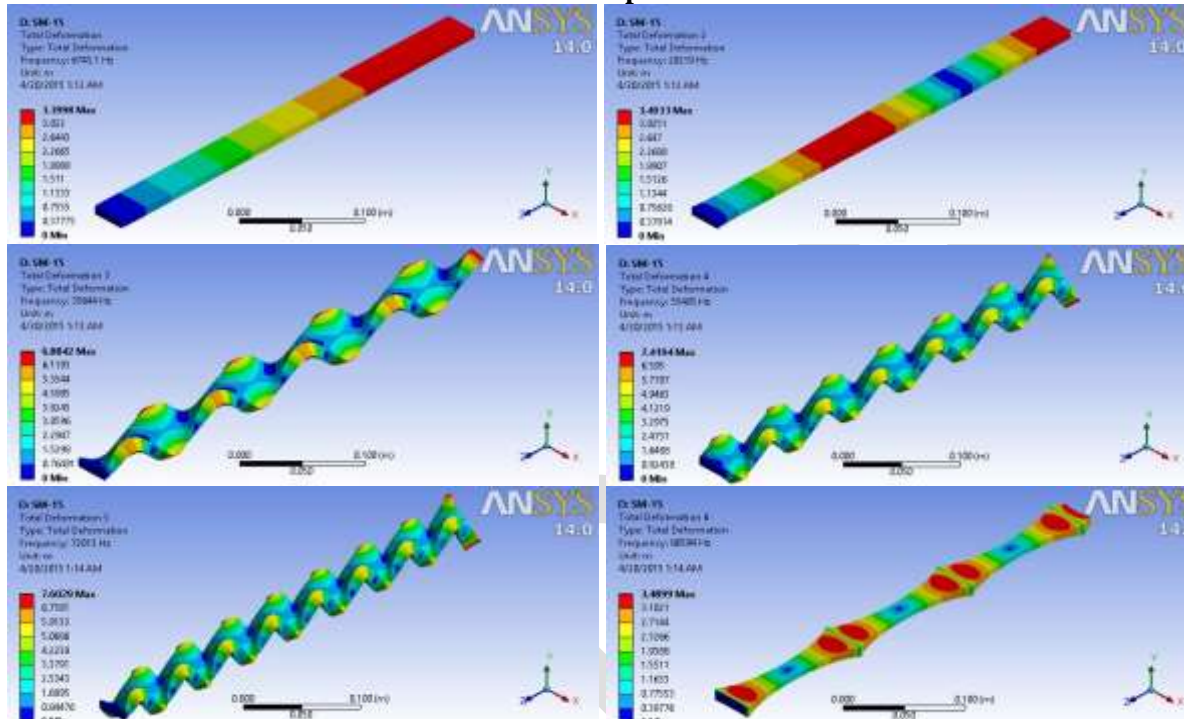**Carbon Fiber Reinforced Plastic SM-5 Mode Shape**

Carbon Fiber Reinforced Plastic SM-6 Mode Shape
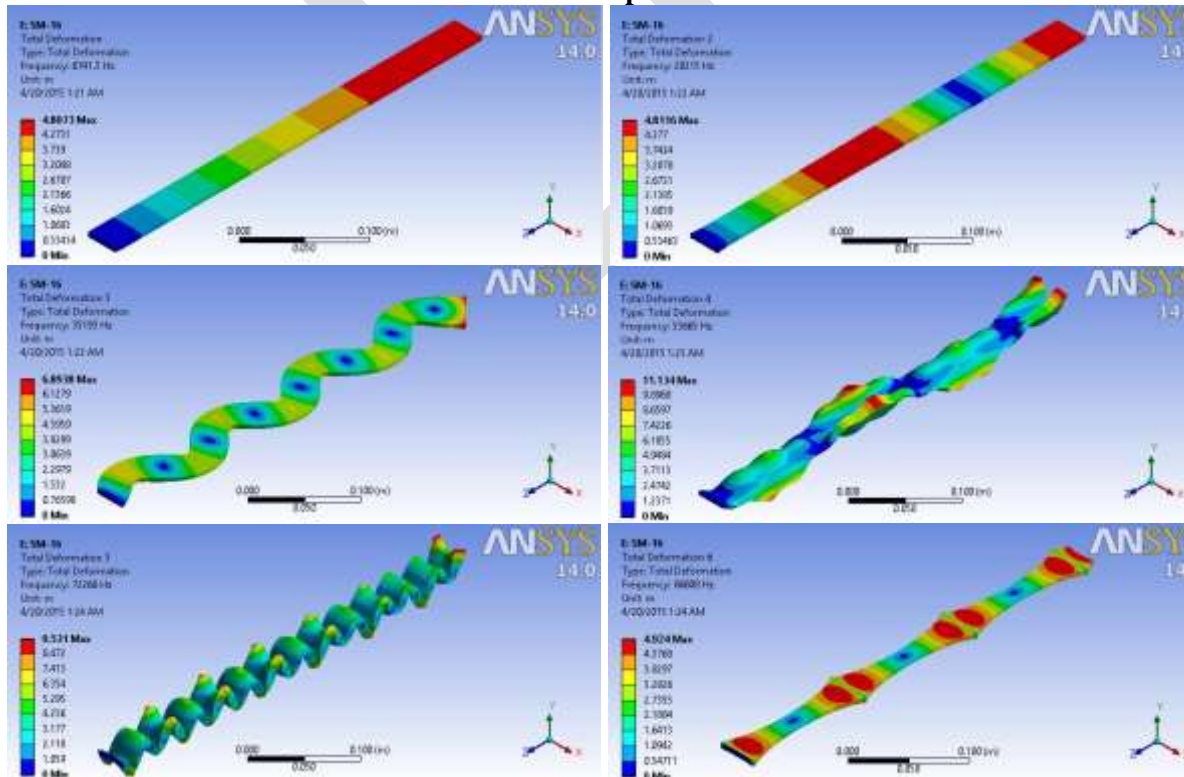
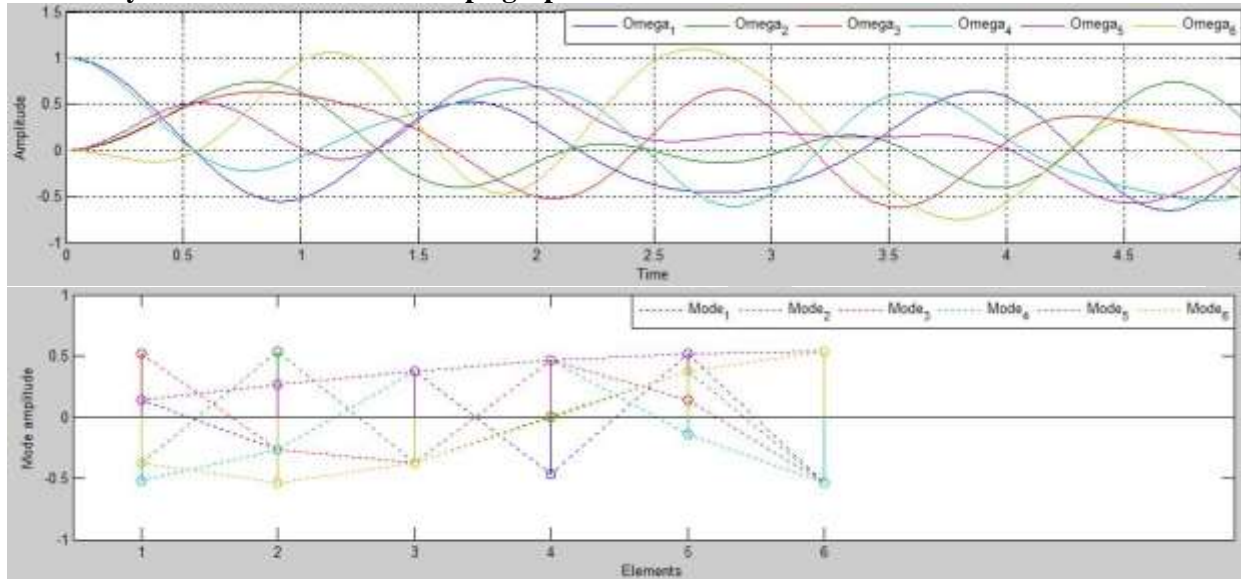**Carbon Fiber Reinforced Plastic SM-7 Mode Shape**

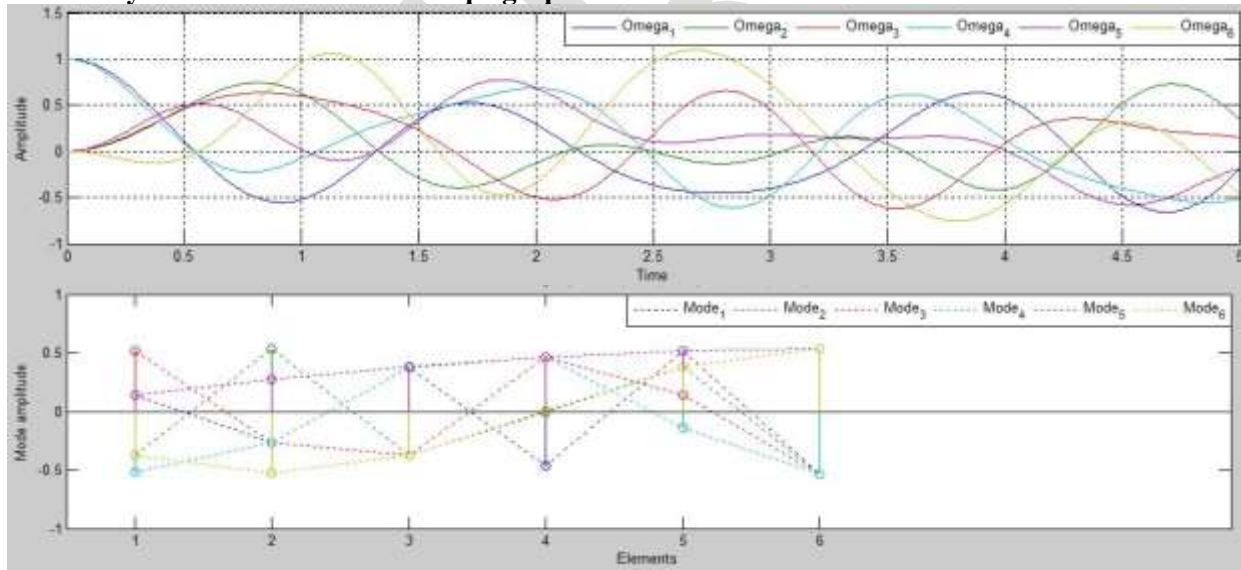**Carbon Fiber Reinforced Plastic SM-8 Mode Shape**

MATLAB Analysis

MATLAB (MATrix LABoratory) is a high-performance interacting data-intensive software environment for high-efficiency engineering and scientific numerical calculations. MATLAB enable the users to solve a wide spectrum of analytical and numerical problems using matrix-based methods, attain excellent interfacing and interactive capabilities, compile with high-level programming languages, ensure robustness in data-intensive analysis and heterogeneous simulations, provide easy access to straight forward implementation of state-of-the-art numerical algorithms, guarantee powerful graphical features. Here, we are using MATLAB 2014a in which simulation and analysis is performed to obtain frequency response function (FRF) and mode shape graph for all specimen of both Martials.[3]
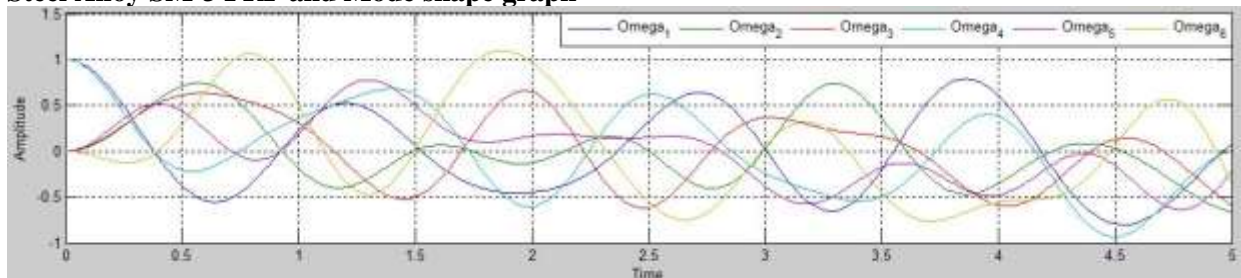
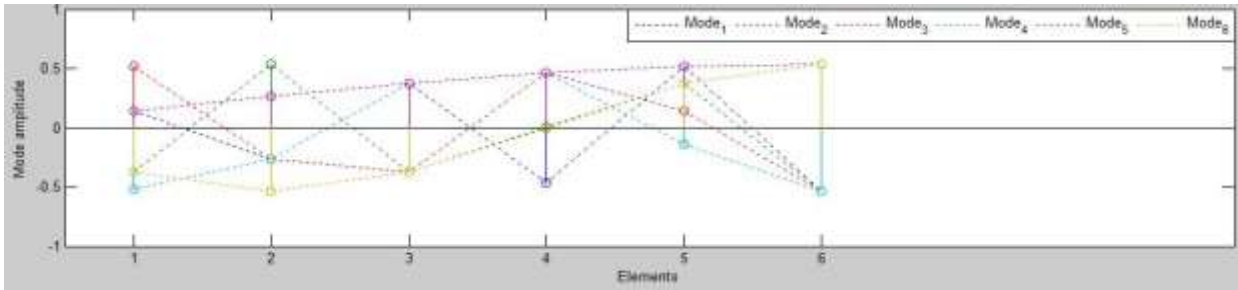**Steel Alloy SM-1 FRF and Mode shape graph**

Steel Alloy SM-2 FRF and Mode shape graph

Steel Alloy SM-3 FRF and Mode shape graph

Steel Alloy SM-4 FRF and Mode shape graph
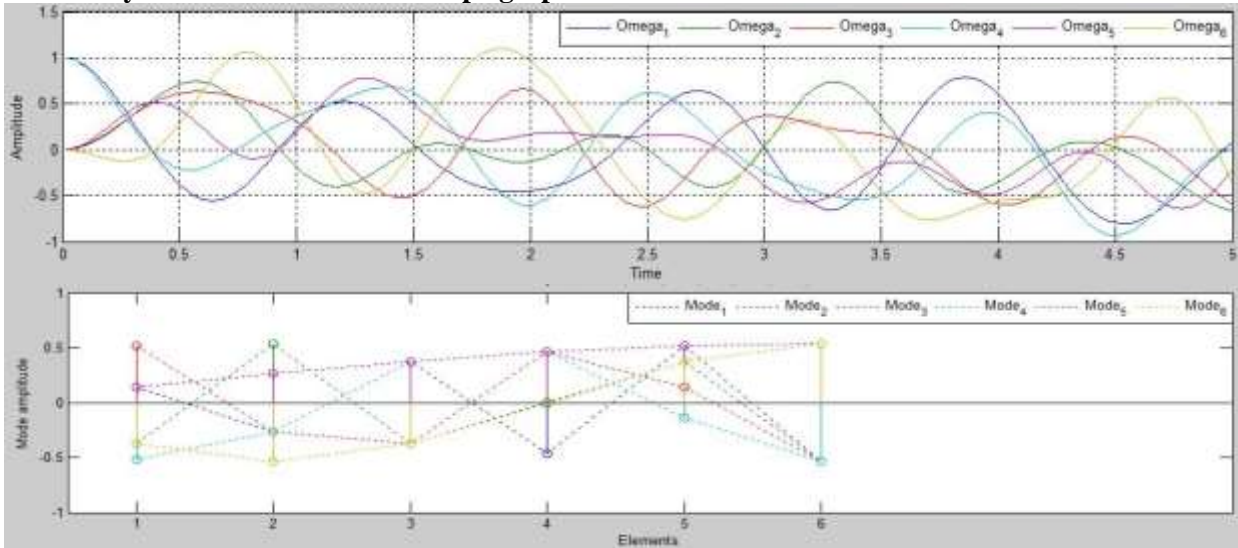
Carbon Fiber Reinforced Plastic SM-5 FRF and Mode shape graph

Carbon Fiber Reinforced Plastic SM-6 FRF and Mode shape graph

Carbon Fiber Reinforced Plastic SM-7 FRF and Mode shape graph

Carbon Fiber Reinforced Plastic SM-8 FRF and Mode shape graph

Finite Element Analysis

Consider a beam fixed at one end having length L, thickness T, Width W and using six element model.

For calculating natural frequency, first of all calculate global mass [M] and stiffness [K] matrix which is given by equation of motion after applying boundary condition are
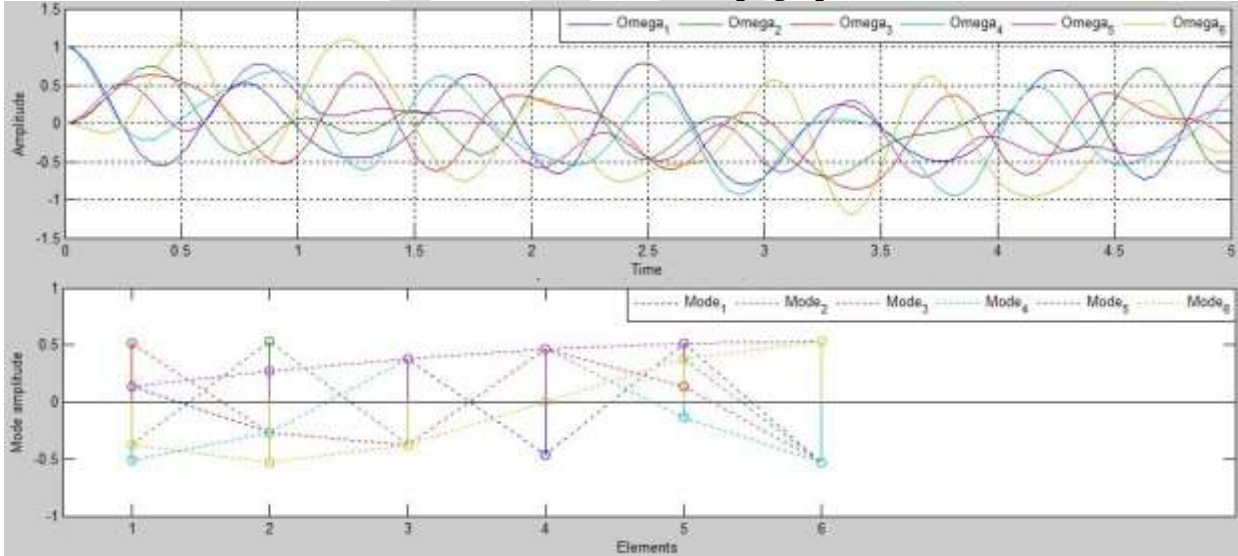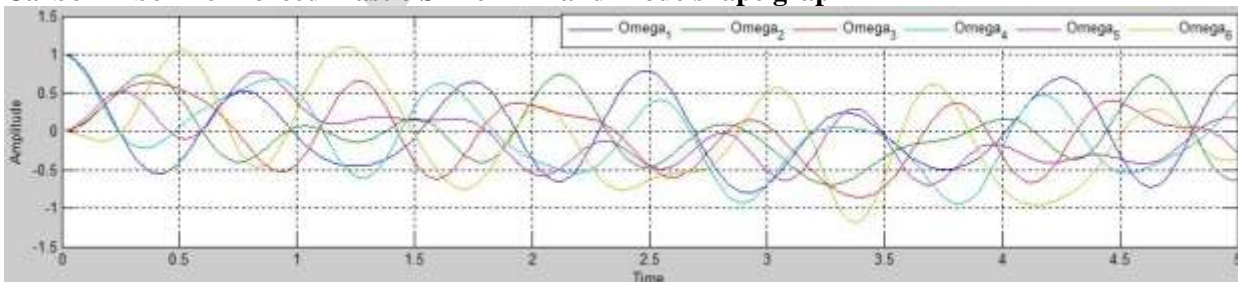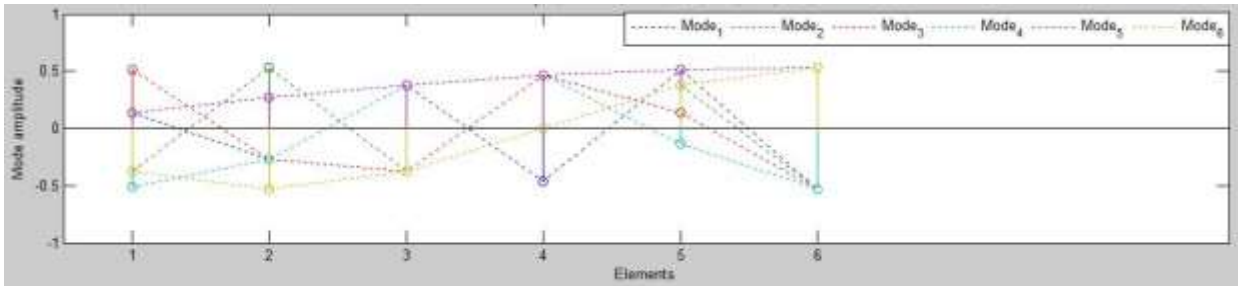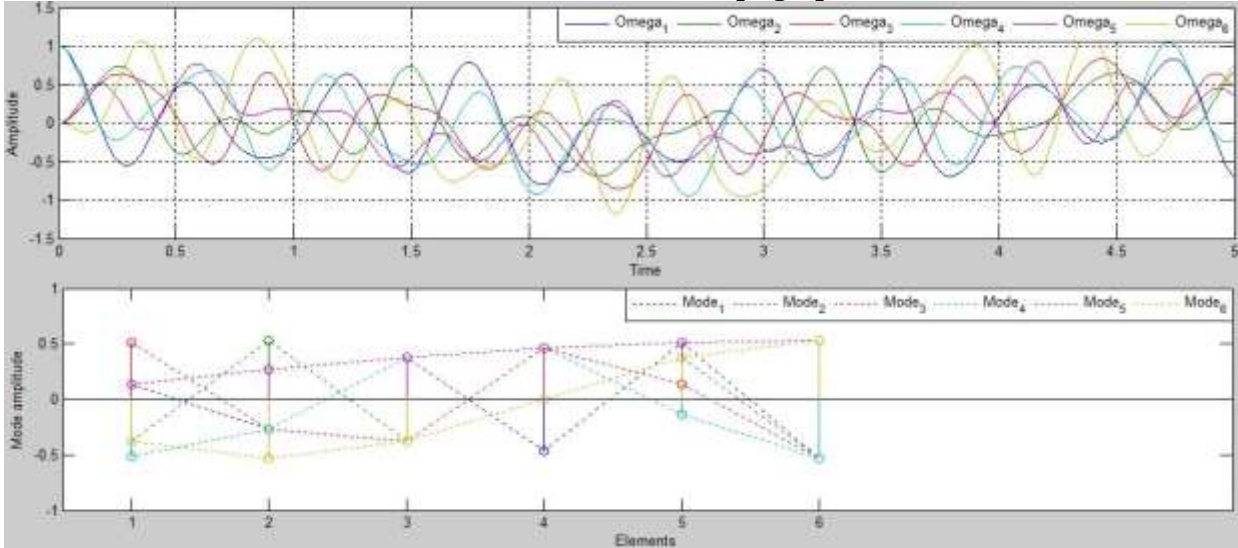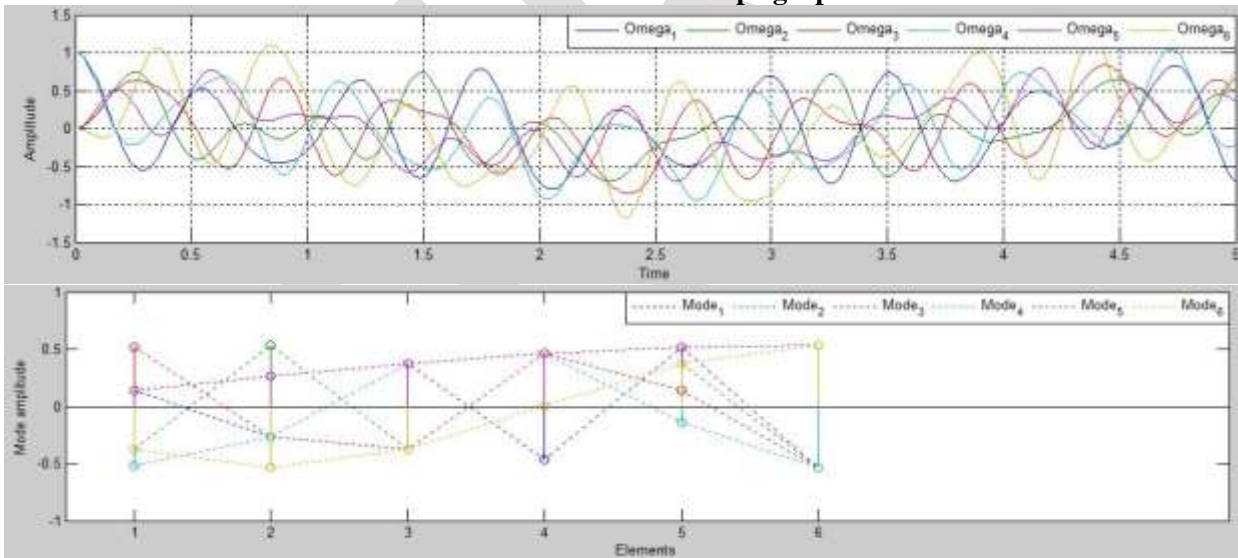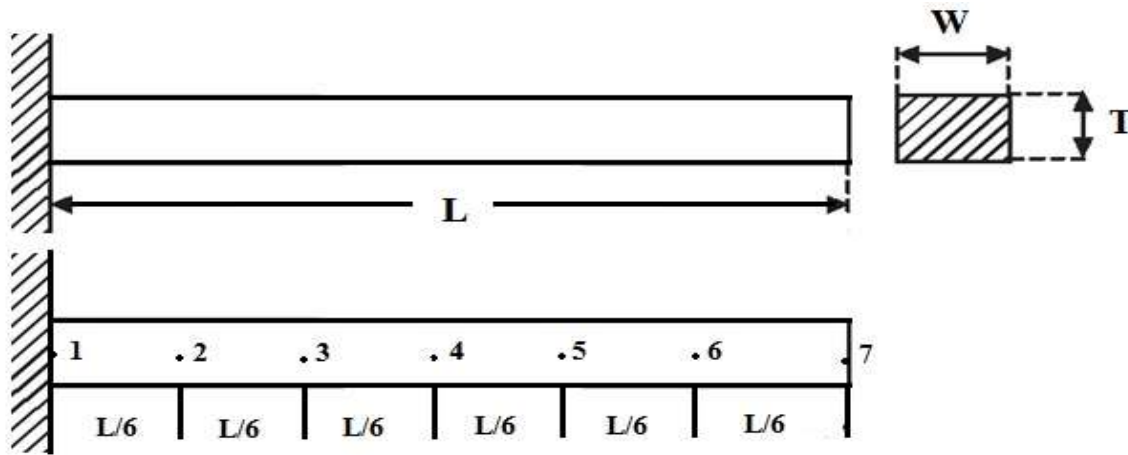
$$[M]^G = \begin{bmatrix} M_1 + M_2 & M_2 & 0 & 0 & 0 & 0 \\ M_2 & M_2 + M_3 & M_3 & 0 & 0 & 0 \\ 0 & M_3 & M_3 + M_4 & M_4 & 0 & 0 \\ 0 & 0 & M_4 & M_4 + M_5 & M_5 & 0 \\ 0 & 0 & 0 & M_5 & M_5 + M_6 & M_6 \\ 0 & 0 & 0 & 0 & M_6 & M_6 \end{bmatrix}$$

$$[K]^G = \begin{bmatrix} K_1 + K_2 & K_2 & 0 & 0 & 0 & 0 \\ K_2 & K_2 + K_3 & K_3 & 0 & 0 & 0 \\ 0 & K_3 & K_3 + K_4 & K_4 & 0 & 0 \\ 0 & 0 & K_4 & K_4 + K_5 & K_5 & 0 \\ 0 & 0 & 0 & K_5 & K_5 + K_6 & K_6 \\ 0 & 0 & 0 & 0 & K_6 & K_6 \end{bmatrix}$$

And solving $[M]^{-1}[K]\{U\} = \omega^2\{U\}$ for the eigenvalues, we get $\omega_1, \omega_2, \omega_3, \omega_4, \omega_5, \omega_6$.
The natural frequency calculated by finite element method and simulation analysis are as follows:-

Frequency and Mode shape Table

S.No.	Specimen	Mode Shape (ANSYS)	Natural Frequency(KHz)		
			ANSYS	MATLAB	FEM
1.		25	6.645	6.569	6.569
2.		12	2.133	2.135	2.135
3.	SM-1	36	11.129	11.441	11.441
4.		47	16.942	16.982	16.982
5.		57	22.833	22.950	22950
6.		65	27.535	27.539	27.539
7.		33	6.396	6.550	6.550
8.		16	2.132	2.158	2.158
9.	SM-2	48	11.095	11.458	11.458
10.		61	16.449	16.978	16.978
11.		74	22.833	22.958	22.958
12.		95	27.604	27.547	27.547
13.		22	9.140	9.384	9.384
14.		11	3.048	3.057	3.057
15.	SM-3	33	16.263	16.344	16.344
16.		43	25.279	24.261	24.261
17.		52	32.780	32.787	32.787
18.		60	39.225	39.343	39.343
19.		30	9.532	9.358	9.358
20.		15	3.047	3.019	3.019
21.	SM-4	43	15.969	16.347	16.347

22.		59	24.391	24.249	24.249
23.		78	32.537	32.804	32.804
24.		91	39.231	39.326	39.326
25.		25	14.717	14.587	14.587
26.	SM-5	12	4.718	4.753	4.753
27.		37	26.144	25.403	25.403
28.		47	37.136	37.708	37.708
29.		57	50.277	50.960	50.960
30.		67	61.336	61.148	61.148
31.		34	14.951	14.587	14.587
32.	SM-6	16	4.717	4.753	4.753
33.		49	26.241	25.403	25.403
34.		62	37.837	37.708	37.708
35.		74	50.278	50.960	50.960
36.		95	61.005	61.148	61.148
37.		22	20.219	20.751	20.751
38.	SM-7	11	6.743	6.761	6.761
39.		33	35.644	36.137	36.137
40.		43	55.485	53.641	53.641
41.		52	72.013	72.492	72.492
42.		60	86.594	86.986	86.986
43.		29	20.215	20.751	20.751
44.	SM-8	15	6.741	6.761	6.761
45.		43	35.193	36.137	36.137
46.		58	53.665	53.641	53.641
47.		78	72.266	72.492	72.492
48.		91	86.608	86.986	86.986

CONCLUSION

Equation of motion derived from Lagrange's equation is best finite element method for calculating natural frequency at any number of nodes. Because in earlier researches based on natural frequency the result obtained by theoretical calculation is maximum of four natural frequencies. But equation of motion derived by Lagrange's equation is capable for calculating any number of natural frequency.

In case of multiple degree of freedom, ANSYS provides the maximum number of natural frequency and mode shapes for cantilever beam under several boundary conditions which shows that cantilever beam has infinite degree of freedom.

In frequency response function graph obtained by MATLAB, shows the dynamical nature of cantilever beam is vibrating at more than one natural frequency at the same time under no load condition.

In present study the natural frequency of cantilever beam of two material (Steel Alloy and Carbon Fiber Reinforced Plastic) is calculated by varying the length and the thickness of beam at constant width. Analysis shows that the performance of carbon fiber reinforced plastic is higher than the steel alloy because all specimens of metal carbon fiber reinforced plastic has higher value of frequency than the steel alloy.

In all specimen one natural frequency have difference greater than or equal to one in ANSYS and finite element method results. Because ANSYS provides the numbers of frequencies and still there are frequencies which exist, shows that cantilever beam has multiple degree of freedom.

In the analysis the result obtained by ANSYS and MATLAB is verified by Finite element method at six degree of freedom. Therefore doing practical and simulation on software under same boundary condition gives the same result. Because in earlier researches based on natural frequency result obtained by practical is verified by the result obtained by simulation software. Then, practical is equal to simulation with lot of possibility and minimum error.

REFERENCES:

[1] ANSYS WORKBENCH, Reference guide, Chapter-1, Page-3, 2014.

[2] D. Ravi Prasad and D.R. Seshu, "A study on dynamic characteristics of structural materials using modal analysis", *Asian Journal of Civil Engineering*, Volume 9, Number 2, Pages 141-152, 2008.

[3] Ferreira A.J.M, Matlab codes for finite element analysis. Portugal, Springer publication.

[4] Majkut, Leszek, Free and forced vibrations of Timoshenko beams described by Single Difference Equation, AGH University of Science and Technology, Faculty of Mechanical Engineering and Robotics, Cracow, *Poland Journal of Theoretical and Applied Mechanics* 47, 1, pp. 193-210, Warsaw 2009.

[5] M. Gurgoze, H.Erol, "Determination of the frequency response function of cantilever beam simply supported –In span ", *Journal of Sound & Vibration*, Pages 372-378. Jan. 2001.

[6] Moaveni, Saeed, Finite element analysis: theory and application with ANSYS, third edition, Pearson, 596-600, 1999.

[7] R.Crawford, Steve Crawford, The simplified handbook of vibration analysis: Applied Vibration Analysis, Volume-2, 1992.

[8] R. Davis. R. D. Henshell and G. B. Warburton a Timoshenko beam element, Department of Mechanical Engineering, University of Nottingham, Nottingham NG7 2RD, England (Received 20 March 1972) *Journal of Sound and Vibration* (1972) 22 (4), 475-487.

[9] Steidel.R, An introduction to mechanical vibration, third edition, New York, John Wiley and Sons, 1971.

[10] S.S.Rao, Mechanical Vibrations, Fifth Edition, Page-17, ISBN 978-0-13-212819-3 (978-0-13-212819-3 : alk. paper) 1. Vibration. I. Title. TA355.R37, 2010.

[11] Seon M. Han, Haym Benaroya and Timothy Wei, Dynamics of transversely vibrating beams using four engineering theories: Mechanical and Aerospace Engineering, Rutgers, the State University of New Jersey, Piscataway, NJ 08854; S.A. *Journal of Sound and vibration* (1999) 225(5), 935-988.

[12] iitg.vlab.co.in/?sub=62&brch=175&sim=1077&cnt=2048