

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 7, July 2018

Finite Element Analysis of Multiscale Composite Beams

Akhil Dev V P¹, Abu Mani², Pankaj B Tambe³

M Tech Scholar, Department of Mechanical Engineering, Amal Jyothi College of Engineering, Kottayam,
Kerala, India¹

Research Scholar, School of Mechanical Engineering, VIT University, Vellore, Tamil Nadu, India²

Professor, VIT University, Vellore, Tamil Nadu, India³

ABSTRACT: This paper focuses on the computational natural frequency analysis (Modal analysis) of 12 layered multiscale composite beam elements. Composites due to their inherent capabilities that were regarded as one of the most advanced materials that drive today's technology. These practical attributes are very appropriate in the make of parts for the automobiles, aviation and other entertainment enterprises. Among that nanoparticles also play very important functions. A composite material has two phases; matrix phase and reinforcement phase. In this study properties of each phases are being enhanced with nanoparticles and enhancement in the desirable characteristics are focused. Polymer nano composite offers an extensive variety of promising application as a result of their much upgraded properties emerging from the fortification of nanoparticles. The synergistic effect of the constituents of multiphase composites provides new applications in various fields. The finite element analysis is a process in which the particular component is converted into small elements called discretization process, the elements are connected to one another by nodes and the complete set of elements is known as mesh. Thus FEM encrypts the various forces and deformations in each noded element.

KEYWORDS: Composite materials, Mechanical properties, Applications, Nanoparticles, Finite element analysis.

I. INTRODUCTION

The basic principle underlying the FEM is relatively simple. The body may be two, or three dimensional solid, is being modeled in soft wares like ANSYS, and it is subdivided into an assembly of small parts called elements – 'finite elements'. The elements are thought to be interconnected with each other, at interconnected joints, known as nodes. The total set or gathering of elements is known as mesh. The way of representing elements as a gathering of very small component system, was generally known as discretization, it was the first of several key steps in understanding the FEM analysis. The Finite Element Analysis (FEA) finds out the solutions for various parameters like temperature, stresses, or other needed unknown parameters etc. were done by limiting the special criterion called energy functional. An energy functional includes all the energies associated with the specific finite element model. Based on the law of conservation of energy, the finite element energy functional must equal to zero. The finite element method calculates the exact solution for any finite element model by minimizing the energy functional. So that, the basic equation for FEA is;

$$dF/dp = 0 \dots \dots \dots (I)$$

Where 'F' will be the energy functional and 'p' was unspecified grid point potential (In the case of mechanics, the potential was displacement.) to be determined. The total potential energy P can be expressed as,

$$\pi = \frac{1}{2} \int \sigma^T \epsilon dV - \int d^T b dV - \int d^T q dS \dots (II)$$

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 7, July 2018

Where σ and ε are the vectors of the stress and strain components at any point, respectively, d was the vector of displacement at any point, b the vector of body force components per unit volume, and q the vector of applied surface traction components at any surface point. In the finite element displacement method, only the displacement parameter was assumed to have unknown values at the nodal points, so that the dissimilarities within the element is described in terms of the nodal values by means of interpolation functions.

Thus, within any one element, $d = N u \dots \dots \dots$ (III)

Where N is the matrix of interpolation functions termed shape functions and u is the vector of unknown nodal displacements. (u equivalent to p in the basic equation for FEA).

The strains within the element can be expressed in terms of the element nodal displacements as,

$E = B u$, where B is the strain displacement matrix.

Finally, the stresses are related to the strains by using an elasticity matrix (eg, Young's modulus) as $\sigma = E \varepsilon \dots \dots$ (IV)

The total potential energy of the discretized structure will be the whole of the energy contributions of every individual component. The finite element method (FEM) is the supreme discretization procedure using in structural mechanics. The fundamental idea in the physical understanding of the FEM is the subdivision of the numerical model into disjoint (non over lapping) components of simple geometry called finite elements or elements for short. The response of each element was expressed in terms of a finite number of degrees of freedom described as the value of an unknown function, or functions, at an arrangement of nodal points. The reaction of the numerical model is then thought to be approximated by that of the discrete model obtained by connecting or gathering the accumulation of all components. The disengagement assembly concept occurs normally while examining many artificial and natural systems[1-3]. FEM Solution process procedures should be like, after the geometry of structure was over, then it will be divided into finite number of small elements, called discretization or meshing. Then the elements are connected at the nodes to form an approximate system of equations for the whole structure, generally it was known as the element matrix creation method. At last, the final step was solution, solving the system of equations involving unknown quantities at the nodes with boundary conditions. Thus get the desired results for analysis with values of unknown quantities.

The finite element analysis method have capabilities to solve a variety of problems ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis. The various types of analysis are:[4]

1. Structural analysis
2. Modal analysis
3. Thermal analysis
4. Fluid flow analysis.

Among this Modal analysis is utilized to calculate the vibrational characters (natural frequencies and mode shape) of a structure or a machine part while it is being designed. It can similarly be utilized for beginning stage for another, more detailed, dynamic examination, such as a transient dynamic investigation, a harmonic response analysis, or a spectrum analysis. Modal analysis is used to determine the natural frequencies and mode shapes of a structure. In the case of designing structures for dynamic loading conditions, finding natural frequencies and mode shapes are important parameters [5-7].

II. MULTISCALE COMPOSITE BEAM

Multiscale composite system is simply a composite material, which has several scales of constituents. The fiber epoxy composite system have macro and micro scale, to this when we adding nano fillers it has another scale called nano scale. That is the composite system so obtained has three different scales of geometrical constituents. It is not necessary that the final product will be in nano scale, it can be micro or macroscopic in size. So that it can called multiphase/multiscale composite system. Besides the matrix and reinforcement phase (micro or macro) scale, it has one nano scale too.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 7, July 2018

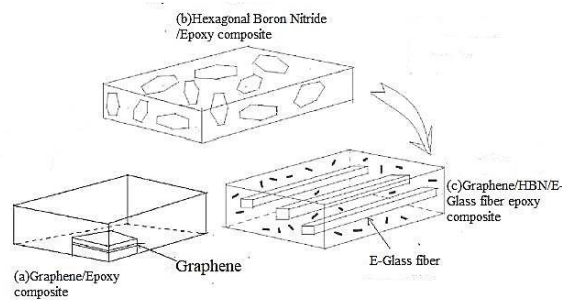


Fig1 Multiscale composite system

The above figure1 shows the sample representation of a multiphase graphene - hbn epoxy composite system. The finite element modeling method have capabilities to solve a variety of problems ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis.

III. MODELING OF COMPOSITE BEAMS

The modeling of the 12 layered composite beam elements was done using ANSYS 17.0. The basic dimension of the composite beam element that required is, Length = 400mm, breadth = 50mm and thickness = 4mm. The analysis was done with dimensions less than the original structure because, for the experimental analysis to done clamps are providing at the ends of the specimen so that much length should be avoided for modal analysis in ANSYS. As the fixed beam analysis are done by providing two clamps on two ends of the specimen at 50mm from each ends. Therefore the cantilever beam analysis was done by fixing the clamp at 100mm from the end of the specimen, so that the length for both analyses should be same. A typical finite element analysis on a software system needs the accompanying data:

1. Nodal point spatial locations (geometry)
2. Elements connecting the nodal points
3. Mass properties
4. Boundary conditions or restraints
5. Loading or forcing function details
6. Analysis options.

In spite of the fact that FEM is a discretization technique, the number of degrees of freedom (DOF) of a FEM model was necessarily finite. They are collected in a column vector called 'u'. This vector is typically called the DOF vector or state vector. The term nodal displacement vector for 'u' is held to mechanical applications. The first step in modeling or analysis was, to give the properties of the material for the analysis to done. Engineering data was the option in the software to give the material properties. The modeling of the 12 layered specimen was done in two ways that is providing fixed support in one end that is cantilever beam and another one is fixed beam.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 7, July 2018

III.i.E-GLASS FIBER- DGEBA EPOXY COMPOSITE SYSTEM

For the pure fiber - epoxy composite system the cantilever and fixed beam condition will be like this.

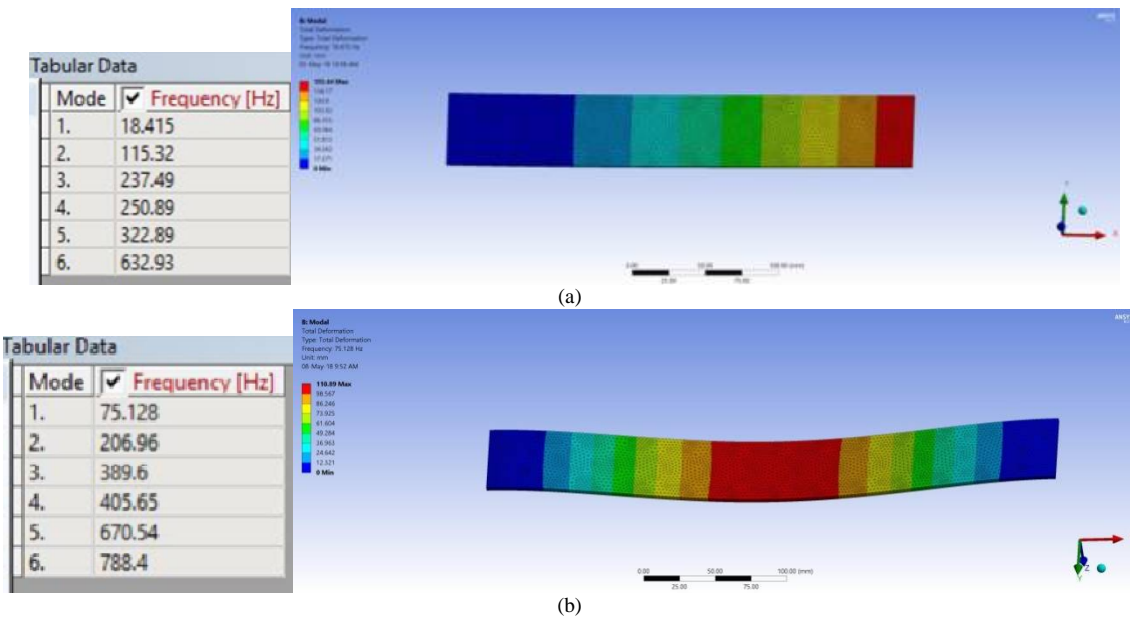


Figure 2 (a) E-glass fiber epoxy composite Cantilever beam: (b) E-glass fiber epoxy composite fixed beam.

III.ii E-GLASS FIBER- GRAPHENE EPOXY COMPOSITE SYSTEM

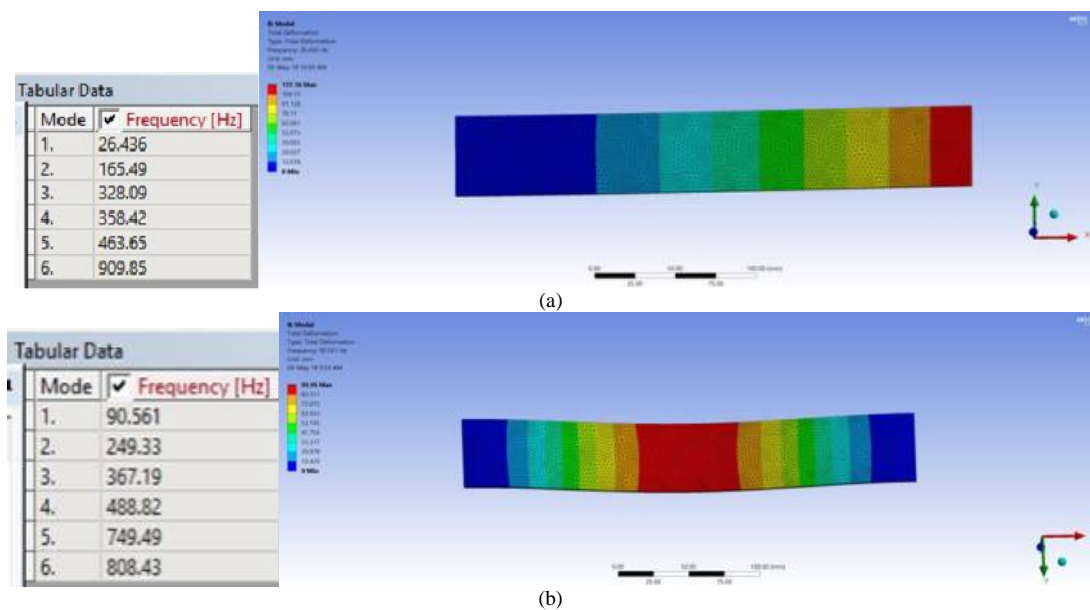


Fig 3 (a) E-glass fiber- Graphene epoxy composite cantilever beam : (b) E-glass fiber- Graphene epoxy composite fixed beam

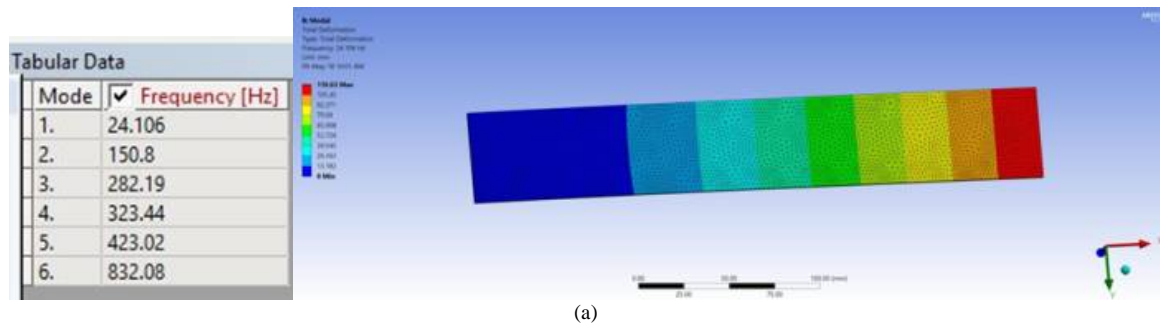
International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

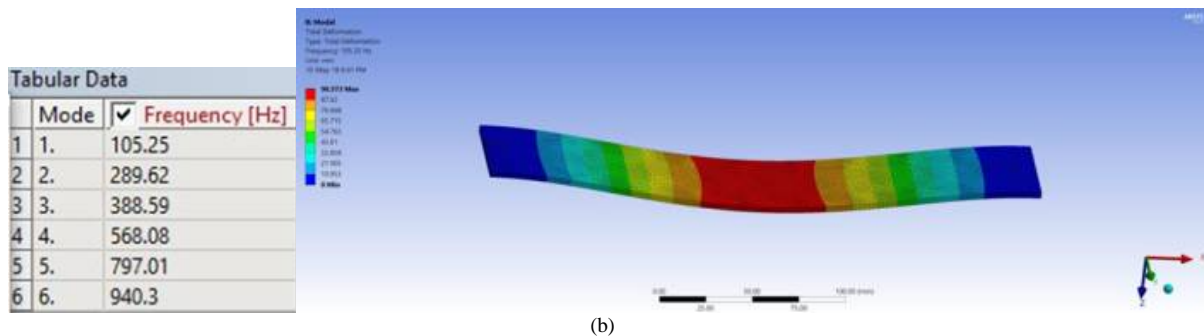
Visit: www.ijrset.com

Vol. 7, Issue 7, July 2018

III.iiiE-GLASS FIBER- HBN EPOXY COMPOSITE SYSTEM



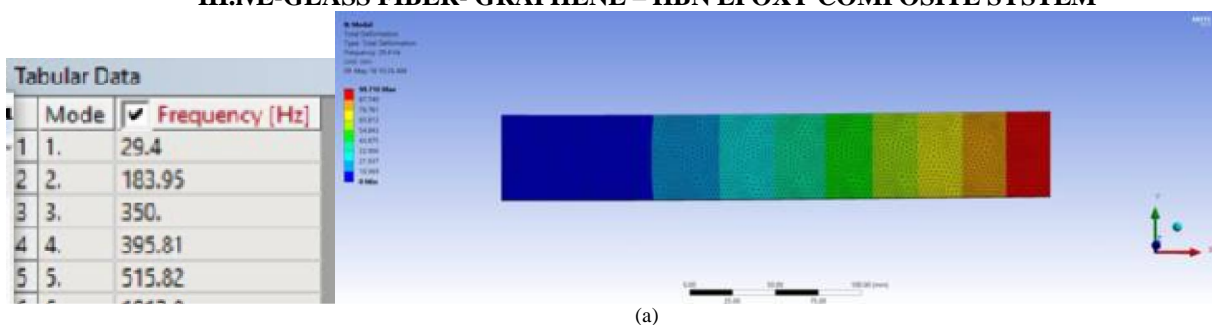
(a)



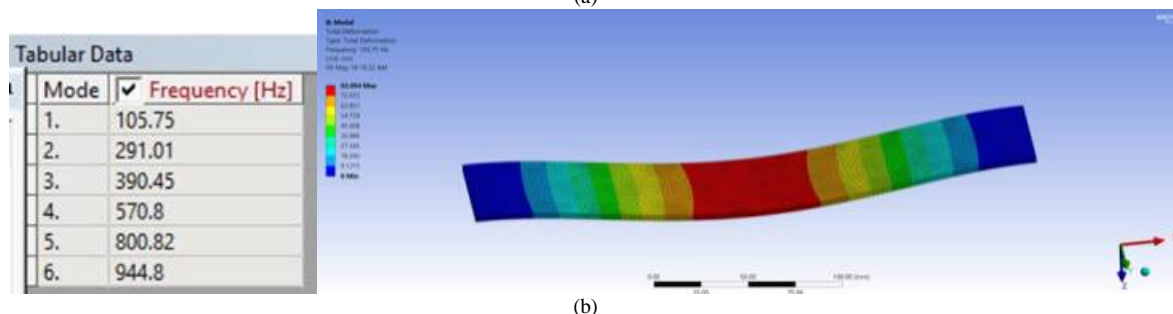
(b)

Fig 4 (a) E-glass fiber- HBN epoxy composite cantilever beam : (b) E-glass fiber- HBN epoxy composite fixed beam.

III.ivE-GLASS FIBER- GRAPHENE – HBN EPOXY COMPOSITE SYSTEM



(a)



(b)

Fig 5(a) E-glass fiber- Graphene – HBN epoxy composite cantilever beam : (b) E-glass fiber- Graphene – HBN epoxy composite fixed beam.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 7, July 2018

IV. RESULT AND DISCUSSION

The figures 2(a), 2(b), 3(a), 3(b), 4(a), 4(b), 5(a) and 5(b) are the modal solution images of the composite beam element. From the results, it was clear that with the combination of the nanoparticles properties that the natural frequency of each beam element was increasing [table 1](#). Also the layer by layer addition of fibers favors the distribution of stresses to a great extent, so that stresses will transferred to the entire 12 layer of the element. Hence the stress concentration will be less, also from the results it was clear that with increased stiffness of the specimens with addition of nano fillers the deformation of the beam also decreases [table 2](#). Addition of the fillers increased natural frequency of the system as well as decreases the deformation. Therefore the enhancement of the composite material by the particular addition of fillers favors a new way to the applications of composite materials.

No	Modes	Cantilever beam ANSYS results (Hz)				Fixed beam ANSYS results (Hz)			
		Glass fiber-Epoxy(GE)	GE-HBN	GE-Graphene	GE-HBN-Graphene	Glass fiber-Epoxy(GE)	GE-Graphene	GE-HBN	GE-HBN-Graphene
1	1	18.41	24.106	26.45	29.426	75.128	90.56	105.25	105.75
2	2	115.32	150.8	165.49	183.95	206.96	249.33	289.62	291.01
3	3	237.49	282.19	328.09	350	389.6	367.19	388.59	390.45
4	4	250.89	323.44	358.42	395.81	405.65	488.82	568.08	570.8
5	5	322.89	423.02	463.65	515.82	670.54	749.49	797.01	800.82
6	6	632.93	832.08	909.85	1013.8	788.4	808.43	940.3	944.8

Table 1: Natural frequencies of beam elements.

Deformation in (mm)	Cantilever beam ANSYS results (mm)				Fixed beam ANSYS results (mm)			
	Glass fiber-Epoxy(GE)	GE-HBN	GE-Graphene	GE-HBN-Graphene	Glass fiber-Epoxy(GE)	GE-HBN	GE-Graphene	GE-HBN-Graphene
	155.44	118.63	117.6	98.718	110.89	98.573	93.95	82.094

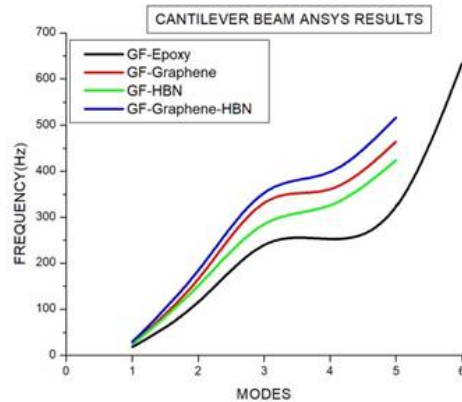
Table 2: Deformations of beam elements.

International Journal of Innovative Research in Science, Engineering and Technology

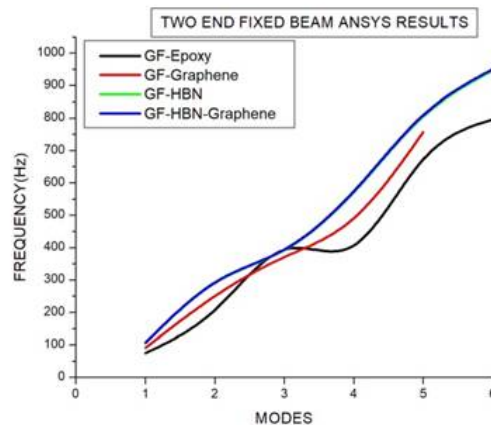
(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 7, July 2018



(a)



(b)

Figure 3.5 (a) Computational Modal results of cantilever composite beams : (b) Modal results of fixed beams

The figure 3.5 (a) shows the graphs of comparison of results obtained by computational modal analysis cantilever beam condition of composite beams. From the graph it was clear that with the addition of nanoparticles the natural frequency of the structure increasing. Result was same for fixed beam condition also fig 3.5(b) . In most of the traditional composites, the modulus can predict. The main reason why the modulus was changed because of change in load transfer. That is if a high modulus filler is added to a lower modulus polymer, then load transfers from the matrix to the filler, leading to an increase in modulus. Here, due to the crystalline structure of graphene and HBN the adhesion increases. Due to the higher d-spacing the surface area of filler increases and hence the adhesion rate also increases. Also because of the increased bonding between added particles and the composite system which resists elongation, the stiffness of the specimen increases and hence the natural frequency also increases. Dynamic response study was an important area, which includes many applications. Mainly in structure related and automotive related fields. Various auto motives are making their products by analyzing the materials dynamic responses with many conditions. One of the good example was that the famous Land Rover automobile manufacturing company uses dynamic response analysis and studies to their vehicles, for better traction control and stability in roadways. It was one small example, like these more industrialists uses these technique for the better performance of their structures or products. Hence materials with good stiffness and dynamic behavior capability can find many applications in various fields. Also by the inclusion of nano fillers to the composition, composite material finds new way of their application. Thus materials with novel properties can find their applications in advanced field

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 7, July 2018

V. CONCLUSION

The modal analysis on different composite beams with or without nano fillers were done successfully, and the following conclusions were drawn.

- The addition of nano fillers to the polymer matrix composite favors the enhancement of certain properties. The resulting thermal, mechanical, optical, magnetic or conducting properties of the nanocomposites are influenced by the filler properties as well as from the fillers surface properties. This was due to the exact binding of fillers with the matrix as well as the reinforcement. The interesting nanoparticles in the present cases are, Hexagonal Boron Nitride (HBN) and Graphene.
- The fillers are added to 0.2 wt% of the epoxy matrix in order to control the agglomeration in composite system.
- With the successive addition of these fillers to the composite system, it was clear and concluded that the natural frequency of the structure was increasing. From the linear natural frequency equation of vibration $f_n = \frac{1}{2\pi} \sqrt{\frac{k}{m}}$, it was clear that stiffness is proportional to frequency, as stiffness increases natural frequency also increases.
- As well as for cantilever beam condition and fixed beam condition the natural frequency of the composite system was increasing. Apart from increase in natural frequency there also decrease in deformation of the beams as well.

REFERENCES

- [1] FEM notes, <http://femnotes.pdf.com>
- [2] Charles H. Roche Accorsi L.M (1998) A new finite element for global modeling of delaminations in laminated beams. Finite Elements in Analysis and Design, 31, 16-177.
- [3] Bambole A.N., Desai Y.M (2007) Hybrid-interface finite element for laminated composite and sandwich beams. Finite Elements in Analysis and Design,43, 1023- 36.
- [4] Chennakesava R Alavala (2010) Finite Element Methods, Second Edition, PHI Learning.
- [5] Brian J. Schwarz & Mark H. Richardson (1999) Experimental Modal Analysis. Vibrant Technology, Inc. Jamestown, California.
- [6] Crystal Instruments, Santa Clara, USA. <http://crystalinstruments.com>.
- [7] Inman, Daniel J. (2001) Engineering Vibrations, Second Edition, Prentice Hall., New Jersey.