Published by :
http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 08, August - 2017

Comparative Analysis of Composite Materials by Using Finite Element Method

Yogita U. Medhane[1]
Research Scholar,
Department of Mechanical Engineering,
Late G. N. Sapkal COE,
Nashik, India

Poonam S. Talmale[2]
Assistant Professor,
Department of Mechanical Engineering,
Late G. N. Sapkal COE,
Nashik, India

Abstract— **Composite is a material consisting of two or more constituents are combined at a ratio of matrix and fiber. Composites are using in most of areas especially in engineering field, in automobile, mechanical, aviation and marine industries. Last few years the scientist are mostly focus on study of composite material and design of components are made with the help of software and experimental analysis because this composite material has high strength to weight ratio. Need of today it to reduce weight and increases the strength of material. For this we are doing analysis of composite material by finding natural frequencies of material which will refer by few literatures. In this paper study and analyze the free vibration analysis of cantilever beam for the composite as well as steel material are carried out. Natural frequency and mode shape of the plates has been determined using ANSYS 17.1. Also comparative study of Steel, E glass epoxy and FRP is done for stress analysis. These materials are used for vibration analysis to observe the effect of a modal parameters of cantilever beam subjected to free vibration is analyzed with the help of analytical method and results obtained from numerical method i.e. finite element method (FEM) are compared. It is concluded that results obtained from FEM have a very good agreement with results obtained from analytically.**

Keywords— Composite, ANSYS, Natural Frequency, Stress, Finite element method.

I. INTRODUCTION

A Composite material, in mechanics sense, is a structure with the ingredients as element. Composites are materials consisting of two or more chemically distinct constituents on a macro-scale, having a distinct interface separating them and with properties which cannot be obtained by any constituent working individually. Composite contains matrix and reinforcement materials. The reinforcing fiber provides strength and stiffness to the composite, whereas the matrix gives rigidity and environmental resistance. The reinforcements can be fibers, particulates, and the matrix materials can be metals, plastics, or ceramics. The most common glass fibers are made of E-glass and S-glass. E-glass is the least expensive of all glass types and it has a wide application in fiber reinforced plastic industry. S-glass has higher tensile strength and higher modulus than E-glass. However, the higher cost of S-glass-fibers makes them less popular than E-glass. The E-glass fiber is a kind of glass fiber with low alkali, excellent strength, stiffness, ductility, insulation, heat resistance and moisture resistance. E glass is the primary reinforced material of wind turbine blades, having low cost and good applicability. It is a better match with many resins, and the molding process. However, as the density of the E-type fiber is large, it is generally used in smaller blades [6]. Composite materials are gaining popularity because of high strength, low weight, resistance to corrosion, impact resistance, and high fatigue strength. Other advantages include ease of fabrication, flexibility in design, and variable material properties to meet almost any application. Use of composite materials in various construction elements has increased substantially over past few decades. Composites are most promising materials for components of current and future engineering structures, with a significant demand at present in aircraft and aerospace industries. A variety of structural components made of composite materials such as turbine blades, vehicle axles, robot arms, aircraft wings, and helicopter blades can be approximated as laminated composite beams. Materials are particularly widely used, where a large strength-to-weight ratio is required. Likewise to isotropic materials, composite structures are subjected to various types of damage, mostly s and delamination.

Defects in a structural element affect its dynamical behavior and change its stiffness and vibration signatures. Subsequently, natural frequencies and mode shapes of structure contain information about location and dimensions of damage [1]. Fiber Reinforced Plastics (FRP) are commonly used in aircraft structure, high speed, military equipment's, civilian products, automotive and or engineering applications mainly because of high strength-to-weight ratio, high stiffness, good resistance to fatigue, and corrosion resistance advantages include ease of fabrication, flexibility in design, and variable material properties to meet almost any application. Thus vibration technique can be suitably used as a non-destructive test for detection of component to be tested Vibration analysis, which can be used to what types of changes occur in vibration characteristics, the combination of different materials has been used for many thousands of years to achieve better performance requirements. There are nowadays many examples in the aeronautical and automobile industries, and yet the application of composite materials is still growing, including now areas such as nautical industries, sporting goods, civil and aerospace construction. Composite structures have been widely used in many engineering examples in aeronautical, astronautically, and marine structures.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 08, August - 2017

II. LITURATURE REVIEW

Rajendar, kumar [1] in this paper the design and analysis of composite drive shaft is studied. The composite material has many advantages as high strength and stiffness. Here the author works on replacement of steel drive shafts with a composite material. Where parameter of design get minimize by achieving the objective of reducing the weight of shaft. FEM methods used to analyzing of Composite materials. Present work is conducted to analyze the composite drive shaft by the FEM software ANSYS 16.

Jauhari et al [2] studied three different formats of defects of hole as a discontinuity was with tensile testing machine. Certain changes observed between defects and properties. Results are in good agreement with general behavior of FRP composite material. Fibers are the mainly used constituents in the composite for increasing strength with considering orientation. From it shows it has good load carrying capacity with a less deflection and more strength.

Kalyana Chakravarthy et al [3] here author studied related to the composite specimen subjected to three point loading conditions. Material used for this specimen is GFRP and it was modeled in ANSYS. With loading conditions simulation are carried out and results were found.

Chopade et al [4] here is also author is mostly focus on to reduce the weight with increasing carrying capacity. For this solving the problem author replaces existing material into the composite materials. Where also developed stresses on the product are studied. Where FEA and Experimental analysis are carried out with the two different materials and comparison are discussed for conventional material and E-Glass/Epoxy material.

Esmael Adema et al [5] studied the experimental characterization properties for the E-glass/Epoxy & E-glass/Polyester material. In this paper is to present processing techniques of specimen preparation, conducting experiment are being carried out for observation porosity and fracture behavior of material. Strain rate effect of E-glass/epoxy and E-glass/polyester are studied. The graphs that are obtained from the tests were documented. Results show that the mechanical properties are dependent on rate of strain.

Deepanshu Bhatt et al [6] also study a laminate composite material having different type of layer and orientation. Experimental results are to be find out the value of shear stress, Maximum principle stress etc. The article reviews of various research literatures in this area are going to achieve the desired level. The specimen is modeled in ANSYS software based on the FRP composite laminate plate and FEA analysis is being done with the ANSYS software.

Kumar et al [7] replaces new material for wind turbine because failure i.e Al. FEA techniques used for the analysis of complex shape geometries. In this paper focus is on to find natural frequencies and vibration modes of specified wind turbine blade material. The results of the analysis are used to verify a structure's fitness for use. Analytical results verified with experimental results are shows good results to author.

Ratnaparkhi, Sarnobat [8] here studied the specimens of woven glass fiber and epoxy matrix plates and it manufactured with the help of hand-layup technique. It also analyze experimental results using modal analysis technique for the obtaining Natural frequencies. This experiment is used to validate the results from the FEA using ANSYS. The effects of different parameters including aspect ratio, and fiber orientation of woven fiber composite plates are studied in free-free boundary conditions.

Ghayour et al [9] said that the non-composite beams has the application of some useful technique are studied. Dimensionless parameters are identified from the equations of motion and the combined effects of the dimensionless parameters on the modal characteristics of the rotating composite beams are investigated through numerical studies.

Wang et al [10] describes the stress and vibration analysis procedure is developed to assess the static and dynamic characteristics of the Woven Composite Axial-Impeller under loading conditions particular to centrifugal force. This procedure is based on FEA results.

Mehmet Aktas et al [11] in this study, the impact response of unidirectional glass/epoxy laminates has been investigated by considering energy profile diagrams and associated load–deflection curves. Damage modes and the damage process of laminates under varied impact energies are discussed. An alternative method, based on variation of the excessive energy versus impact energy, is presented to determine penetration threshold. The primary damage mode was found to be fiber fracture for higher impact energies; whereas, it was indentation resulting in delamination and matrix cracks for smaller impact energies.

From literature review, we summarized work done by various researchers and engineers in field of vibration analysis mainly on composite materials. It includes different type of methods, software analysis and Experimental techniques are used for analysis. According to some researcher changes in dynamics characteristics can be used as an information source for detecting of vibrating beam.

III. THEORY AND FORMULATION

A. Geometry and Material Modeling of Specimen

Some components are complicated for inspection like Wind turbine blades which is made up of composite materials, they are complicated for inspection by non-destructive techniques they are multi-layered, have variable thickness and are made of anisotropic materials. By considering applicability and availability and depending upon engineering applications, in this paper following materials are selected for analysis.

Material Used: Steel, Fiber reinforced plastic and E-glass epoxy. According to testing of the steel cantilever beam its mechanical properties are taken from literature. A thin flat strip of specimen having a constant rectangular cross section was prepared in all cases. The dimension of the specimen was taken as given in Table No. I

TABLE I. DIMENSIONS OF SPECIMENS

Sr. No.	Length (mm)	Width (mm)	Thickness (mm)
1	845	55	5

B. *Equation of Free Vibration of Beam*

The beam made up from steel material with clamped at one (left) end free at right end and has same cross section. The cantilever beam dimensions are given in Table I

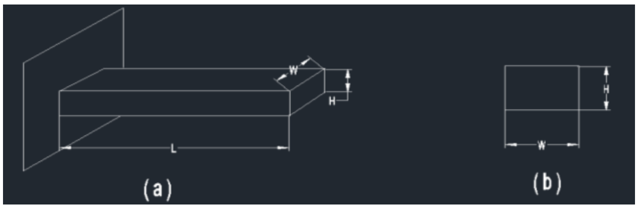


Fig. 1. Geometry of cantilever beam (a) Boundary condition of beam (b) cross-sectional view of beam

The free bending vibration of an Euler-Bernoulli beam of constant rectangular cross section is given by the following equation,

$$EI\,\frac{d^4 y}{dx^4} - m\omega^{2y} = 0 \qquad (1)$$

By defining,

$$\lambda^4 = \frac{m\omega_i^2}{\mathrm{EI}}$$

Equation is rearranged as a fourth order differential equation as follows,

$$\frac{d^4 y}{dx^4} - \lambda^4 y = 0 \qquad (2)$$

The general solution to the equation is,

$$y = A\ \cos\lambda_1 .x + B\ \sin\lambda_1.x + C\ \cos h\lambda_1.x + \sin h\lambda_1.x \qquad (3)$$

Where A, B, C, D are constants and λ_i is a frequency parameter. The element stiffness matrix of the beam is given as,

$$[\mathrm{K^e}] = \int [\,B\,(x)]^T\,EI\,[\,B\,(x)]dx$$

$$[\mathrm{B}\,(x)] = \{\mathrm{H}_1(x)\ \mathrm{H}_2(x)\ \mathrm{H}_3(x)\ \mathrm{H}_4(x)\} \qquad (4)$$

Where the Hermitian shape functions are defined as:

$$\mathrm{H}_1(\chi) = 1 - \frac{3x^2}{1^2} + \frac{2x^3}{1^3}$$

$$\mathrm{H}_2(\chi) = x - \frac{2x^2}{1} + \frac{x^3}{1^2}$$

$$\mathrm{H}_3(\chi) = \frac{3x^2}{1^2} - \frac{2x^3}{1^3}$$

$$\mathrm{H}_4(\chi) = -\frac{x^2}{1} + \frac{x^3}{1^2}$$

The natural frequency then can be calculated from the relation

$$[-\omega^2\,[\mathrm{M}] + [\mathrm{K}]]\{q\} = 0 \qquad (5)$$

From Euler's Beam theory theoretical formula for natural frequency given as follows

$$\omega_n = \beta^2 * \sqrt{\frac{\mathrm{EI}}{mL^4}} \qquad (6)$$

Where (β_i^2) is constant depend on the boundary conditions and i = 1, 2, 3…..n are the frequency corresponds to each mode. According to this theory calculated frequency values for each mode as follows

TABLE II... ANALYTICAL RESULTS FOR NATURAL FREQUENCIES AT DIFFERENT MODE SHAPES

Sr. No.	Material	Mode	Natural Frequency (Hz)
1	Steel	1	2.618
		2	18.76
		3	32.87
		4	60.4
		5	79.54
		6	101.315

IV. SOFTWARE SIMULATION

In the present work, rectangular beam of composite materials are modeled. For modeling the beam Pro-e is used. The solid model prepared in Pro-e is imported in workbench ANSYS 17.1 to perform static analysis for impact loading. The dimensions of the beams are same for both that are 845mm length, 55mm width, and 5mm thickness. So beam specimen are shown following figure with dimensions,

A. *Modelling in Pro-e*

With the help of design software here, used Pro-e software to model a cantilever beam as shown in following figure, the modeling of beam as it is same in geometry for all materials which is used in this project.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 08, August - 2017

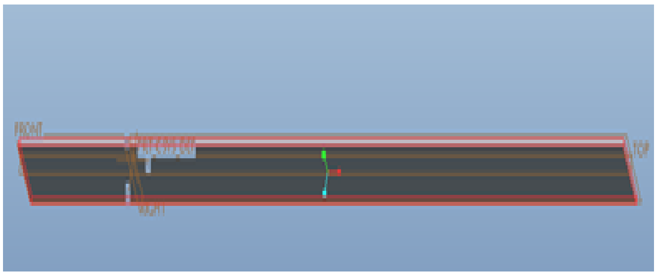


Fig. 2. A 3-D Design in Pro-e software

B. Meshing

The meshing of rectangular beam is carried out in ANSYS software. The finite element standard quality criteria considered in meshing of rectangular beam component. Total number of elements and nodes used to create the FE model of beam is 345 elements and 2788 nodes.

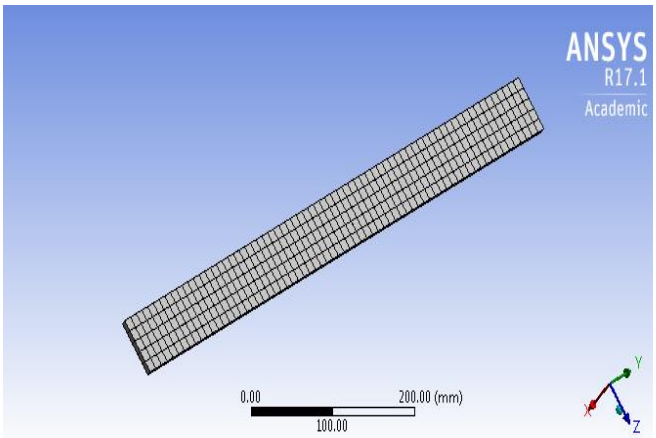


Fig. 3. Finite Element Model of the Cantilever Beam with Meshing.

C. Boundary Conditions

The finite element modeling of the beam with applying boundary conditions is as shown in Fig. No. 4.3, for the modal analysis of cantilever beam boundary conditions are one end of beam is fixed and other end is free.

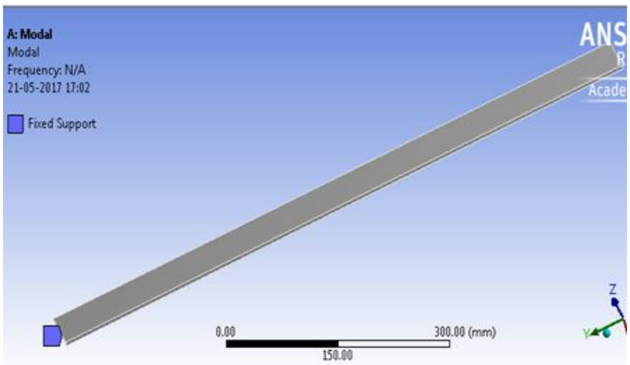


Fig. 4. Cantilever Beam with applying Boundary Conditions

D. Modal Analysis

Modes are inherent properties of a structure. Resonances are determined by the material properties (mass, stiffness, and damping properties), and boundary conditions of the structure. Each mode is defined by a natural frequency, modal damping, and a mode shape. If either the material properties or the boundary conditions of a structure change, its modes will change. For instance, if mass is added to a vertical pump, it will vibrate differently because its modes have changed. At or near the natural frequency of a mode, the overall vibration shape of a machine or structure will tend to be dominated by the mode shape of the resonance. So, basically, modal analysis is the study of the natural characteristics of structures. Understanding both the natural frequency and mode shape helps to design my structural system for noise and vibration applications. We use modal analysis to help design all types of structures including automotive structures, aircraft structures, spacecraft, computers, and tennis rackets, golf clubs the list just goes on and on.

Once we defined the boundary conditions after that, modal analysis is performed for three different materials which are used in this project for comparison. When an elastic system vibrates due to inherent forces, this state of system is known as free vibration. In state of free vibration, system vibrates at its natural frequency. The natural frequency and mode shapes are two important modal parameters in free vibration. The natural frequency is governed by geometric and material properties. FEA analysis was used for free vibration analysis. The natural frequency and mode shapes for steel and FRP, E glass epoxy which different composite materials were obtained. The finite element analysis using ANSYS software was used in modal analysis to obtain the natural frequencies for steel and FRP, E glass epoxy composite materials.

E. Modal Analysis for Selected Materials

For Cantilever boundary condition of beam, Modal analysis has been carried out for Steel material,

- Steel Material

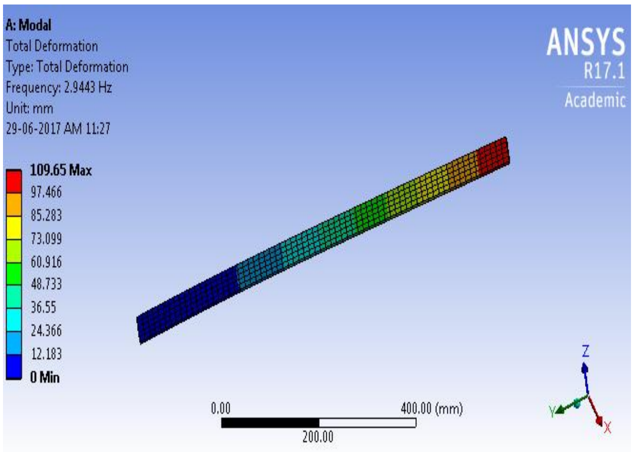


Fig. 5. Mode 1

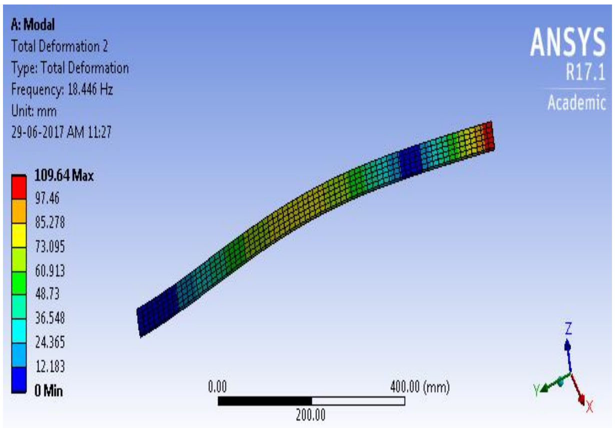


Fig. 6. Mode 2

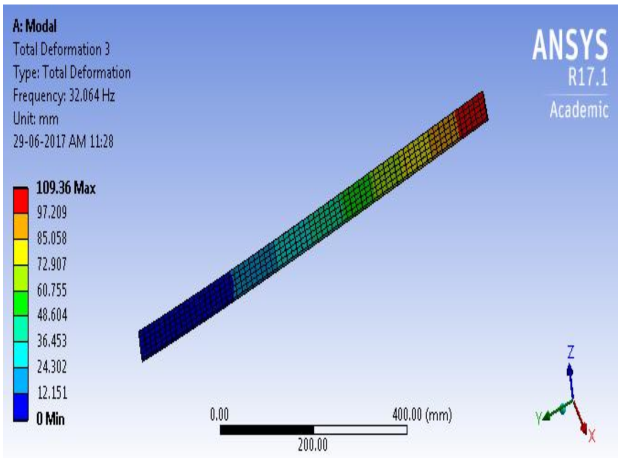


Fig. 7. Mode 3

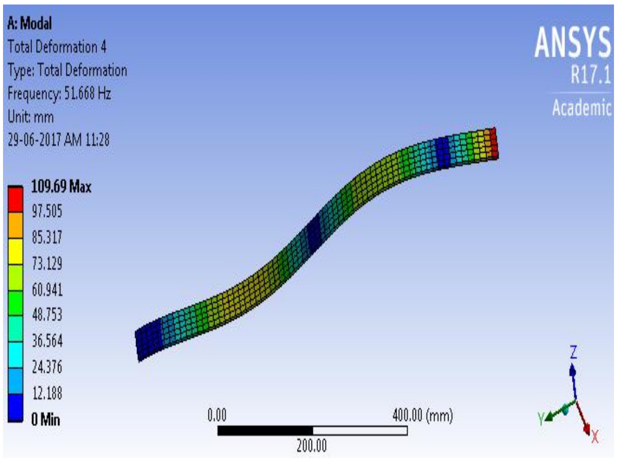


Fig. 8. Mode 4

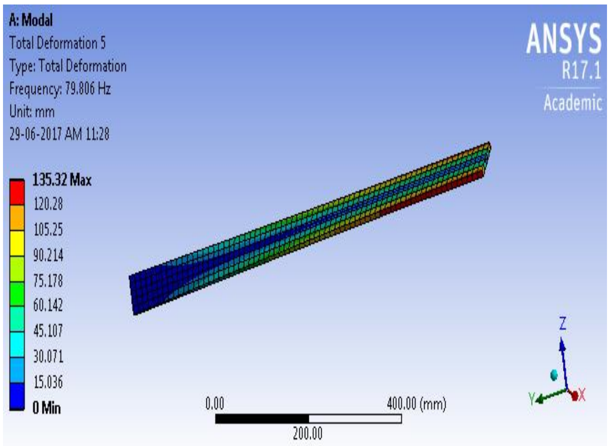


Fig. 9. Mode 5

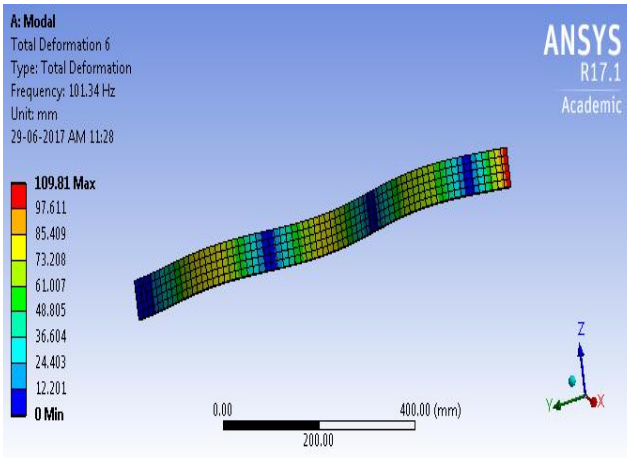


Fig. 10. Mode 6

➢ FRP Material

Similarly, modal analysis has been performed for FRP composite material as shown below,

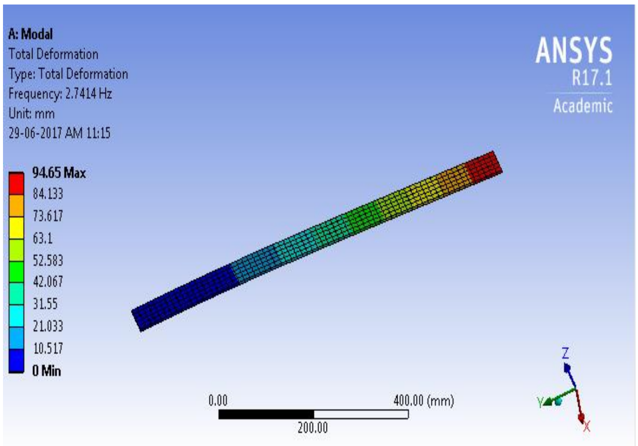


Fig. 11. Mode 1

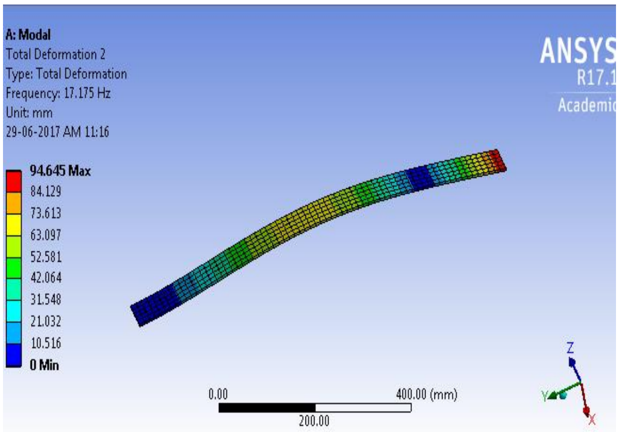


Fig. 12. Mode 2

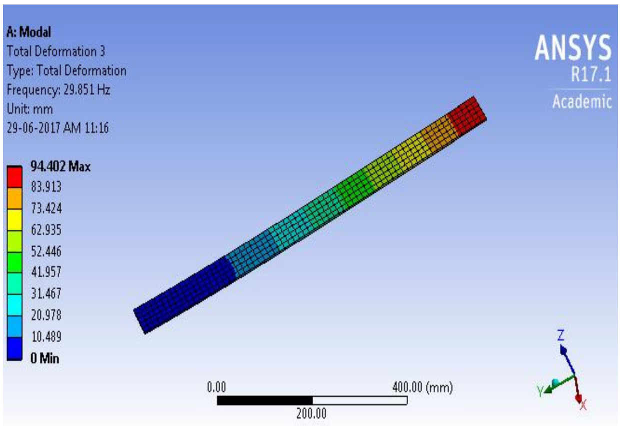


Fig. 13. Mode 3

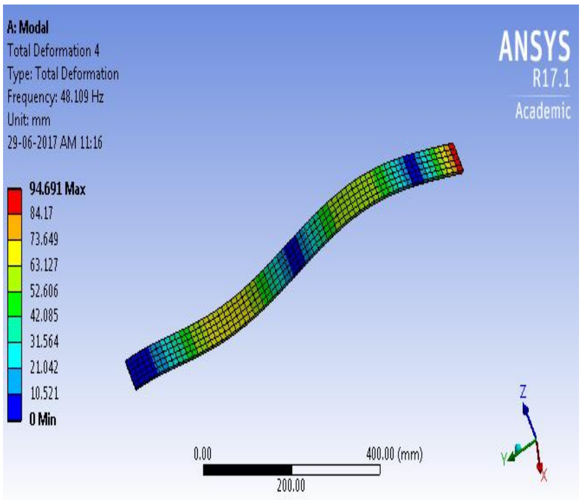


Fig. 14. Mode 4

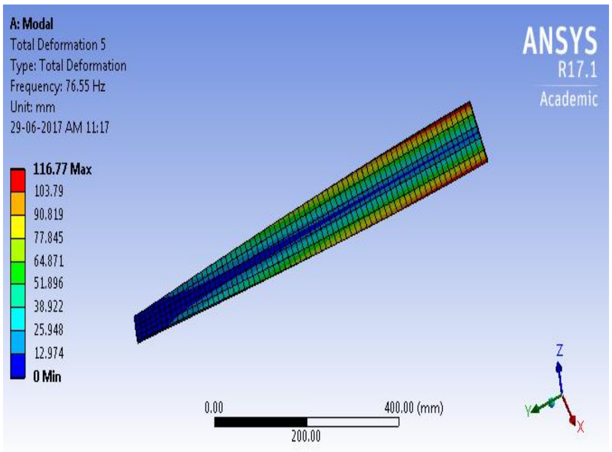


Fig. 15. Mode 5

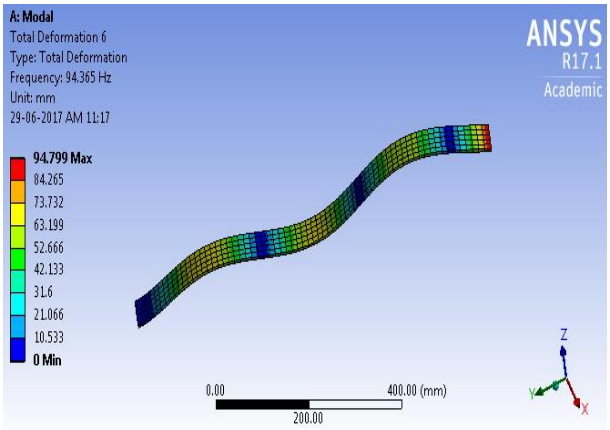


Fig. 16. Mode 6

- E-glass epoxy Material

Here, Modal analysis has been carried out for E-glass epoxy material,

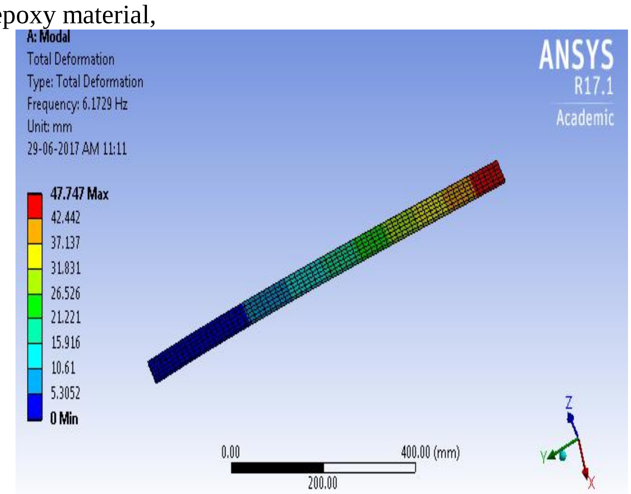


Fig. 17. Mode 1

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 08, August - 2017

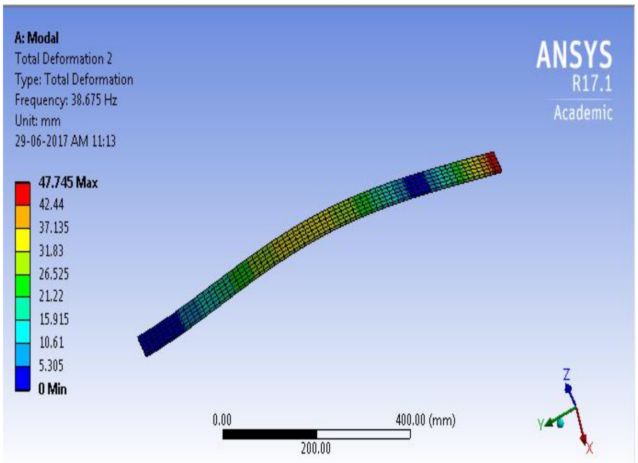


Fig. 18. Mode 2

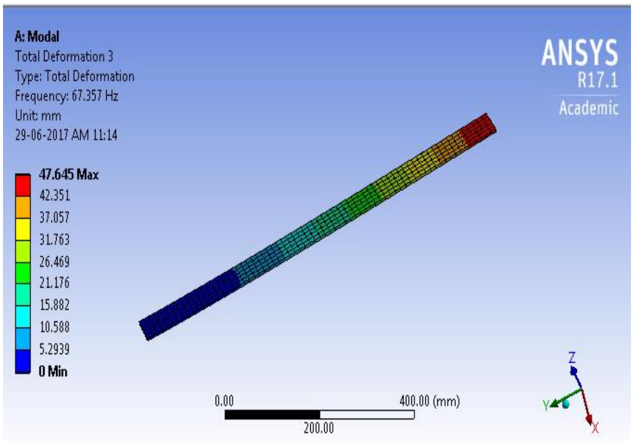


Fig. 19. Mode 3

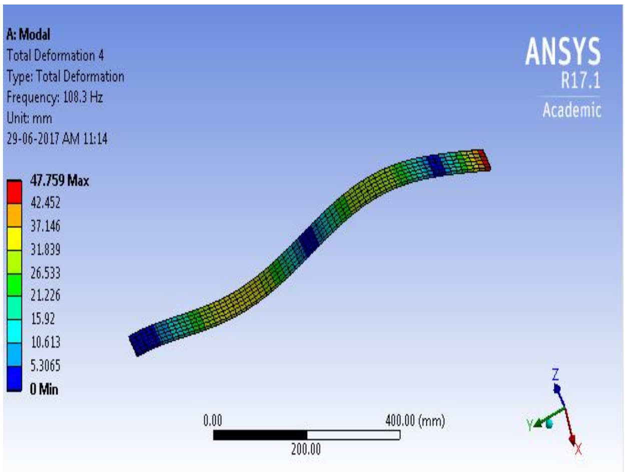


Fig. 20. Mode 4

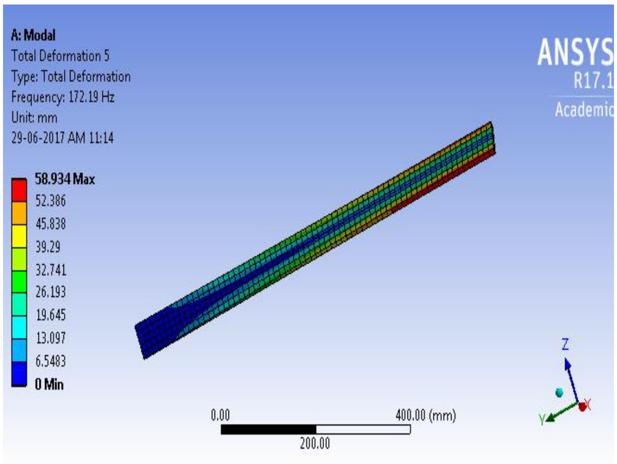


Fig. 21. Mode 5

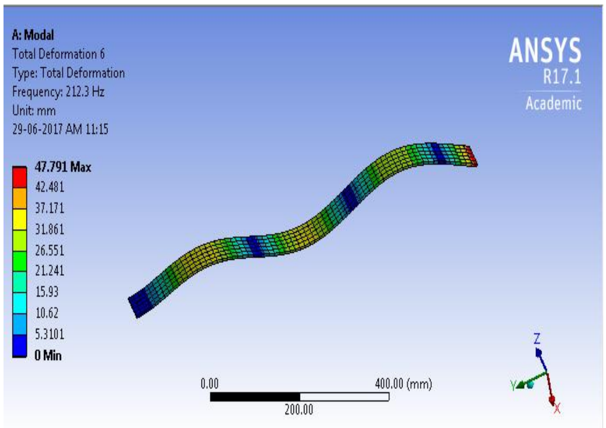


Fig. 22. Mode 6

F. *Structural Analysis*

Structural analysis was performed for stresses for all materials. So only these two parameters were considered for structural and modal analysis. The result of this analysis evaluates the static failure condition of cantilever beam. In this part of structural analysis the beam undergo in 3 point flexural bending test, so the boundary conditions are different as compared to modal analysis of cantilever composite beam.

➢ Boundary conditions

For structural analysis 3 point flexural bending test is used for that to do FEM analysis boundary conditions are applied in such a way in that two ends of plate are fixed and load is applied at center portion of plate.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 08, August - 2017

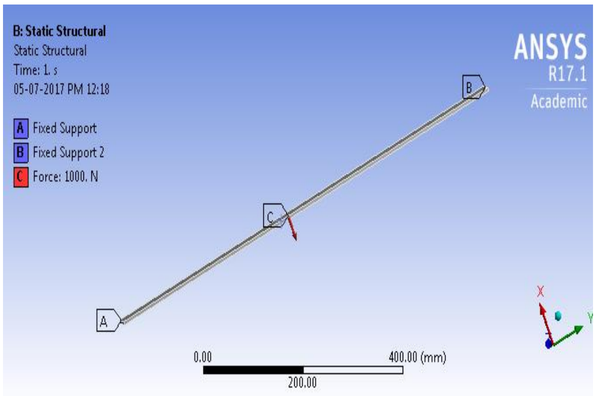


Fig. 23. Boundary Conditions Apply for Static Structural Analysis

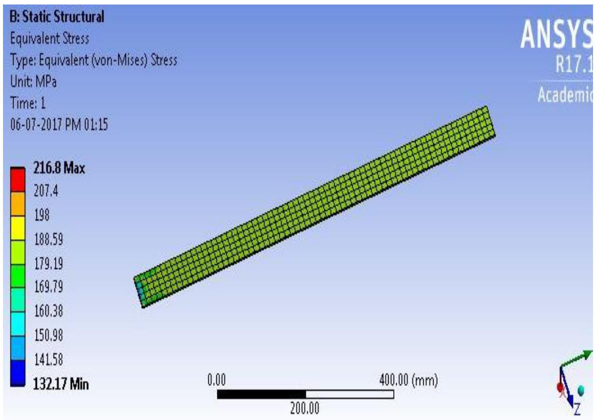


Fig. 24. Static Structural Analysis for Steel Material

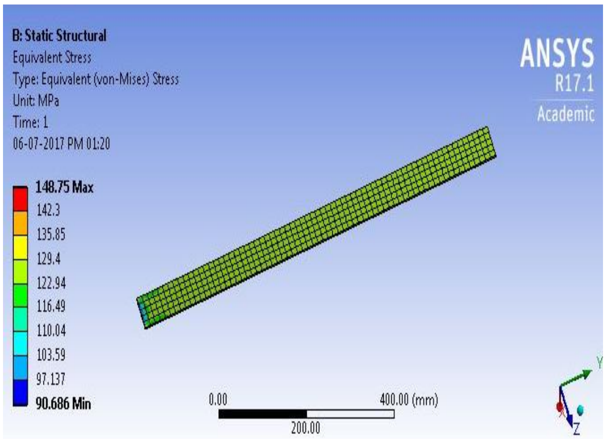


Fig. 25. Static Structural Analysis for FRP Material

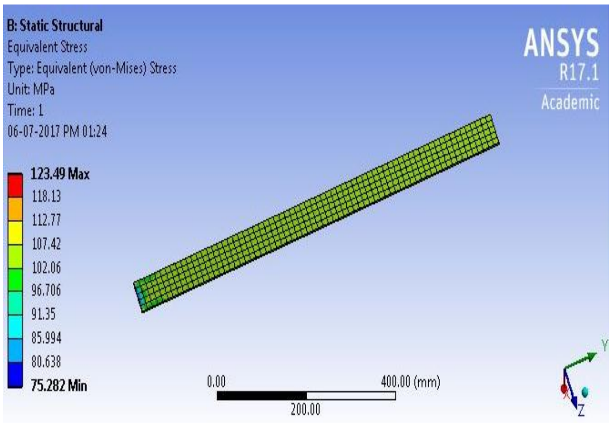


Fig. 26. Static Structural Analysis for E glass epoxy Material

V. RESULTS AND DISCUSSION

The comparison of results got from the FEM was done. The basic concept behind this is the software that is theoretical and the experimental results were compared and found out the better results. The results and discussion were based on the observation so for drawing the conclusion the results are very important. Natural frequencies were extracted and compared for Steel, FRP, and E-glass epoxy. Since the results are in good agreement, the finite element analysis can be used to generate many sets of input-output data for comparison with experimental data from which we can make the conclusions are in last part of this paper, from the finite element analysis by using ANSYS 17.1 following results are obtained for frequencies at different mode shapes.

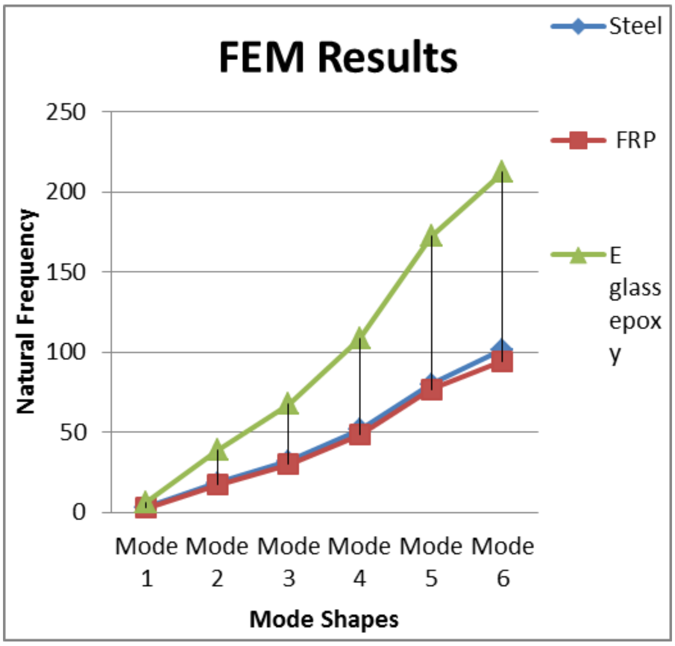
The results of simulation by using ANSYS 17.1 software results obtained for Steel, FRP and E-glass epoxy composite materials are compared as in Table III

TABLE III. COMPARISON OF RESULTS FOR STEEL, FRP, AND E-GLASS EPOXY MATERIAL

Mode Shape	FEA Natural Frequency (Steel)	FEM Natural Frequency (FRP)	FEA Natural Frequency (E- glass Epoxy)
1	2.944	2.7414	6.1729
2	18.446	17.175	38.675
3	32.064	29.851	67.357
4	51.668	48.109	108.3
5	79.806	76.55	172.19
6	101.34	94.365	212.3

From the above result table we can find different natural frequencies at different mode shape, so we can draw graph on the basis of results to show comparison between FEM work for Steel, FRP and E glass epoxy materials. Following Graph-5.1 shows the graphical representation for experimental results for Steel, FRP and E glass epoxy materials.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 08, August - 2017



Graph. 1. Comparison of FEM results at different Mode Shapes and Natural Frequencies

From the above graph it is noticed that as mode number is increasing natural frequencies are increasing. It is due to non-dimensional number increases corresponding mode number increases.

VI. CONCLUSIONS

From results and discussion we can made some conclusions as E Glass Epoxy material gives higher natural frequencies by comparing it with Steel and FRP. We can see that theoretical and FEM results are in good agreement. We depict sets of data natural frequencies corresponding to analytical and FEM analysis. Natural frequencies for E-glass epoxy are higher as compared to Steel and FRP material at different modes; from FEM analysis we can see deformed shape corresponding to each mode of vibration shows significant changes at localized position, also we can predict the behavior of that particular material at a particular natural frequency.

REFRENCES

[1] Mr. N. Rajendar Mr. k. Niranjan Kumar ―Design and Analysis of Composite Drive Shaft‖, International Journal of Science, Engineering and Technology Research (IJSETR) Volume 6, Issue 1, January 2017, ISSN: 2278 – 7798

[2] Nitin Jauharia,, Raghvendra Mishrab, Harischandra Thakur, ―Stress analysis in FRP composites‖, Published by Elsevier GmbH, Perspectives in Science 2016 - 8,50—52.

[3] Kalyana Chakravarthy P, Dr. Raghunandana.K, ―Stress Analysis of Glass Fiber Reinforced composites used in Wind turbines‖, ISSN (PRINT): 2393-8374, (ONLINE): 2394-0697,volume-2, issue-2,2015.

[4] SushilB.Chopade, Prof.K.M.Narkar, Pratik K Satav, Design and Analysis of E-Glass/Epoxy Composite Mon leaf Spring for Light Vehicle‖, International Journal of Innovative Research in Science, Engineering and Technology, Vol. 4, Issue 1, January 2015, DOI: 10.15680/IJIRSET.2015.0401026

[5] Esmael Adema, Er. Mukesh Didwaniab, Gurala Muralidhar Reddy, Ermias G. Koricho, ― Experimental Analysis of E-Glass /Epoxy & E-Glass /polyester Composites for Auto Body Panel‖, American International Journal of Research in Science, Technology, Engineering & Mathematics, 10(4), March-May, 2015, pp. 377-383.

[6] Deepanshu Bhatt , Yogesh Mishra , Dr. P.K. Sharma , A Review of Static and Free Vibration Analysis of Laminated Composite Plate by using Finite Element Method‖, International Journal of Advanced Engineering and Global Technology Vol-2, Issue-10, October 2014, ISSN No: 2309-4893

[7] Ashwani Kumar, Arpit Dwivedi, Vipul Paliwal, Pravin P Patil, Free Vibration Analysis of Al 2024 Wind Turbine Blade Designed for Uttarakhand Region Based on FEA‖, 2nd International Conference on Innovations in Automation and Mechatronics Engineering, ICIAME 2014, 2212-0173 © 2014 The Authors. Published by Elsevier Ltd., Procedia Technology 14, 2014, 336 – 347.

[8] Sharayu U. Ratnaparkhi, S. S. Sarnobat P.,Vibration Analysis of Composite Plate‖, International Journal of Modern Engineering Research (IJMER) Vol.3, Issue.1, Jan-Feb. 2013, pp-377-380 ISSN: 2249-6645.

[9] R. Ghayour, M. Ghayour, S. Ziaei-Rad, Vibration analysis of tapered rotating composite beams using the hierarchical finite element‖, Applied and Computationa lMechanics 4 (2010) 157–170, Received 21 June 2010; received in revised form 19 November 2010.

[10] Jifeng Wang, Jorge Olortegui-Yume, Norbert Müller, Stress and Vibraiton Analysis for Woven Composite Axial Impeller‖, ASME 2010 Power Conference, July 13-15, 2010.

[11] Mehmet Aktas, Cesim Atas, Bulent Murat Icten, Ramazan Karakuzu An experimental investigation of the impact response of composite laminates‖, Composite Structures 87, 0263-8223, 2008 Elsevier 307–313.