Finite Element Analysis and Natural Frequency Optimization of Engine Mounting Bracket

Mallikarjun B Patil[1], Prof. S.S Kelkar[2]
[1, 2] *Mechanical Engineering*
[1,3] *JSPM'S Jayawantrao Sawant College of Engineering, Hadapsar, Pune, India.*
Patilmallikarjun90@gmail.com[1]
satejkelkar@gmail.com[3]

Abstract— Present automotive market demands low cost and light weight component to meet the need of fuel efficient and cost effective vehicle. This in turn given the rise to more effective use of materials for automotive parts which can reduce the mass of vehicle at the same time enhance the performance. Reducing weight gives multi-disciplinary challenges such as stiffness, strength, fatigue life and NVH requirement. Noise, Vibration and Harshness are the main parameters that increase the discomfort to vehicle ride and these are originated from the engine. The engine mounting bracket plays the important role in reducing NVH and increases comfort. As the brackets supports to the engine and these are exposed to the high vibrations, forces and stresses (static as well as dynamic). Vibrations can be reduced or dissipated and keeping the stresses under the predetermined level of safety can be done by careful design, analysis and selecting material of brackets. In this paper the modal analysis of engine mounting bracket is done by using ANSYS 13.0. The design of the bracket is kept same but the material is changed to carbon fibers from steel. The results were generated by using ANSYS R15.0. With the proposed new material the weight of the bracket is reduced by approx. 75% and the stresses are in the permissible stress level.

Keywords: Engine Mounting Bracket, Modal Analysis, Optimization, FFT Analyzer

I. INTRODUCTION

The vehicle engine mounting system mainly consists of engine, transmission supplementary systems and several mount rubbers connected to the vehicle frame. The modern engine mounting systems have been successfully applied to isolate the driver and passenger from noise, vibration and harshness (NVH) generated from engine. Still researcher considers there is need to improve the performance of engine mounting system for the reasons of more vibration and noise isolation for the driver and passengers and modern trend of lighter car design and more power intensive engines. The comfort requirements, the weight reduction and the increased engine power sometimes have negative effects on vibratory behavior. These aspects are opposite to each other. Some improvement in the performance of engine mounting systems definitely plays an important role in resolving the comfort requirements. Redesigning, reanalyzing and performing the optimization of the current model play a vital role in improving the sustainability of the object and allow the engineer to give comfort to the customers.

Now the engine mounting brackets are mounted on the chassis and it gives seat to the engine. Due to the moving parts of engine at high speed and road roughness produces vibrations, it get transmitted to the chassis trough brackets. And these vibrations can damage the chassis as well as produces discomfort to the passengers and driver. Now if the brackets have their resonance frequency near to the engine working frequency then it may result in high amplitude vibrations and it may damage chassis and bracket. If the harmonic response values of the bracket are more than acceptable values then it results in high noise. Both the cases of vibration and noise, reduces the life of bracket, discomfort to the passengers and driver. Hence modal analysis and harmonic analysis of the bracket is necessary. The vibrations and noise can be reduced by shifting to the suitable dimensions and material of the bracket. But the static and dynamic forces on bracket can increase the deflection and stresses. Challenges are to keep the deflection and stresses in the permissible range, while changing the dimensions and material. In this paper an attempt is made to reduce the weight of the bracket by changing material and dimensions.

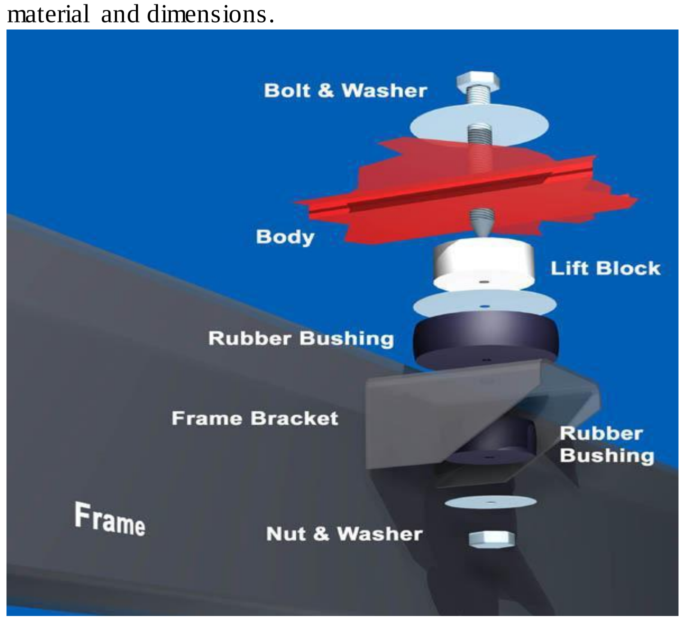

Fig.1 Typial Engine Mounting Bracket

II. OBJECTIVE

1. To study and perform vibration analysis of Engine Mounting Bracket.

2. To propose an optimized model with carbon glass fiber with better or same performance and reduced weight

III. PROBEM STATEMENT

Engine is one of the most important components of a road vehicle such as car. The improvement of engine bracket system has been the subject of intense interest for many years. We see that in the existing Engine mounting bracket there is issue of early brake down of the bracket which is mostly due to the vibration of engine and road condition. Vibration analysis is usually carried out to ensure that potentially catastrophic structural natural frequencies or resonance modes are not excited by the frequencies present in the applied load. Sometimes this is not possible and designers then have to estimate the maximum response at resonance caused by the loading.

IV. METHODOLOGY

The existing model of Engine Mounting Bracket is created using CATIA, stresses and modal analysis performed in ANSYS 10.0. Results obtained by analyzing existing model are stored for comparison purpose. Later topology optimization is performed in low stress zone; the regular shapes are removed to reduce the weight of the bracket. Three iterations were performed. The analysis is performed by changing the material from steel to carbon fiber.

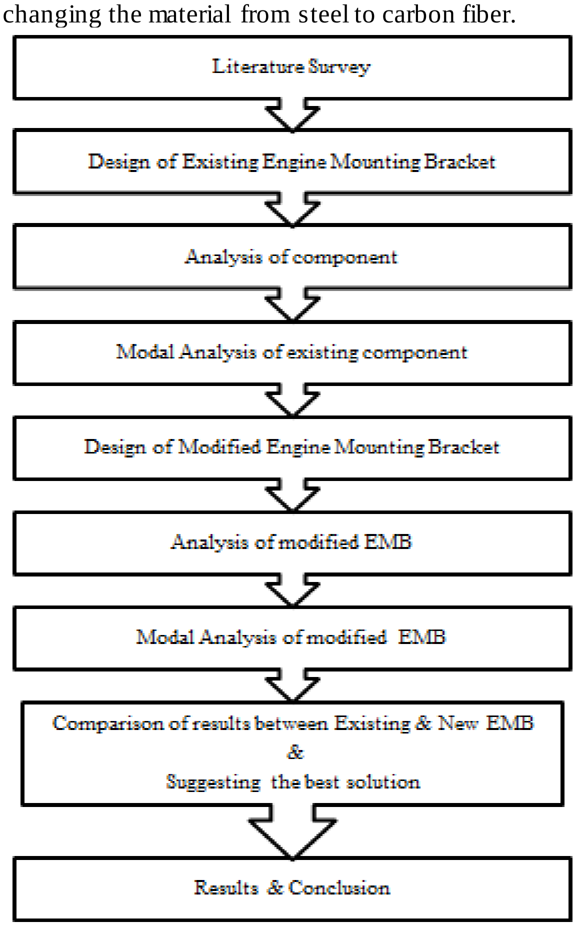

Fig. 2 Methodology.

V. ANALYSIS OF EXISTING ENGINE MOUNTING BRACKET

At present engine mounting bracket is made of mild steel material. So, first analysis is done using MS as material Steel rates well in terms of both yield strength and ultimate strength, particularly if it's carefully alloyed and processed. Steel also resists fatigue failure well which is extremely useful - even if the assembly flexes under load, such flexing need not lead to a critical failure. But the steel is denser than Carbon fiber. Hence the weight of steel is more than carbon fiber.

Table 1 Properties of MS Material

Property	**Value**
Young's Modulus, E	2.1×10^5 MPa
Poisson's Ratio ,ν	0.3
Density, ρ	7850 kg/m^3
Yield Stress, σ_{yield}	290 MPa
Ultimate Tensile Stress, σ_{uts}	390 MPa

A. *Existing Engine Mounting Bracket*

The existing model of Engine Mounting Bracket is prepared using CATIA. The boundary conditions and forces will be identified and applied, and stresses, deflection are estimated using ANSYS. Modal analysis is performed using ANSYS software to find the natural frequencies.

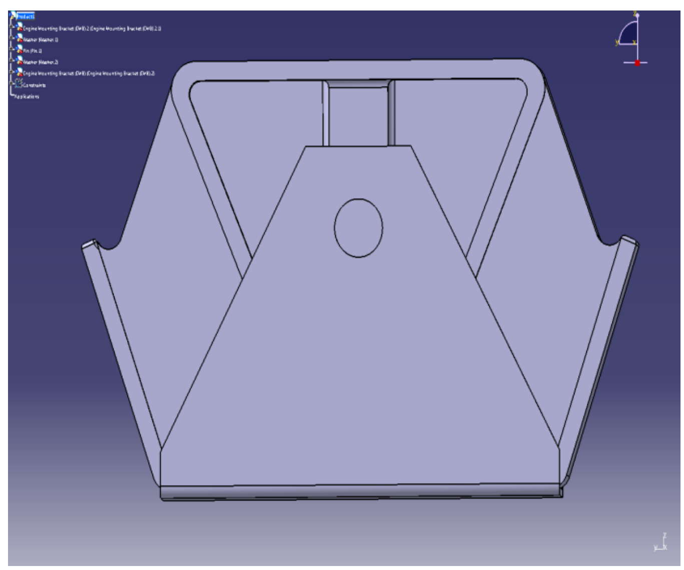

Fig. 3 CAD Model of Engine Mounting Bracket

B. *Tetrahedral Meshing of Existing Engine Mounting Bracket*

We are going to mesh the components using 3D elements. As all dimension of Engine Mounting Bracket are in proportion we use the tetrahedral elements for meshing. Less the mesh size more will be the number of elements and coarse the mesh size less will be the number of elements. As the number of elements increases the run time increases. After meshing elements are to be checked for Quality i.e. elements have some definite quality criteria which should be met by all elements.

C. Boundary Conditions and Forces on Existing Engine Mounting Bracket

After meshing is completed we apply boundary conditions. These boundary conditions are the reference points for calculating the results of analysis. In short we here go for the preparation of deck. Here we apply define and apply various loads. Different load steps are created which are to be applied

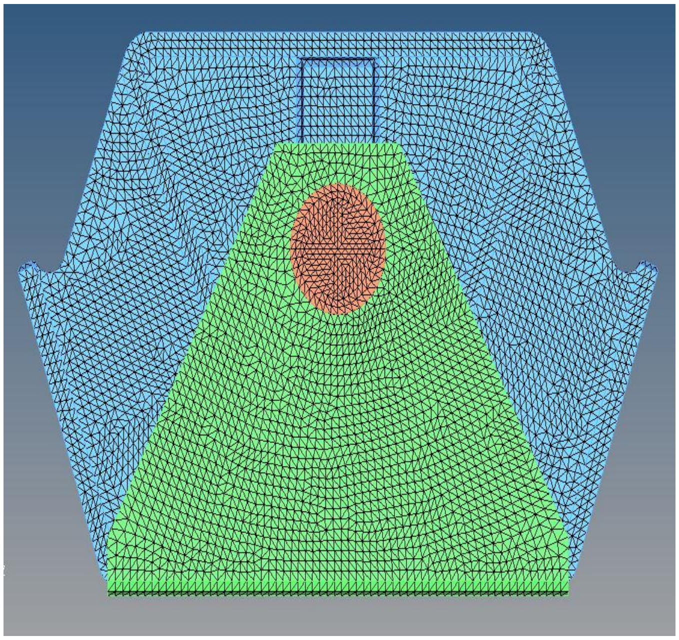

Fig.4 Mesh Model of Engine Mounting Bracket

during analysis. Here surrounding effect is been taken into consideration while applying loads. Elements are defined by their properties. Material properties such as density, modulus of elasticity, Poisson's ratio etc. is assigned to the elements. Here proper arrangements are made so that we can run the analysis in solver software. After the completion of process model is exported to the solver.

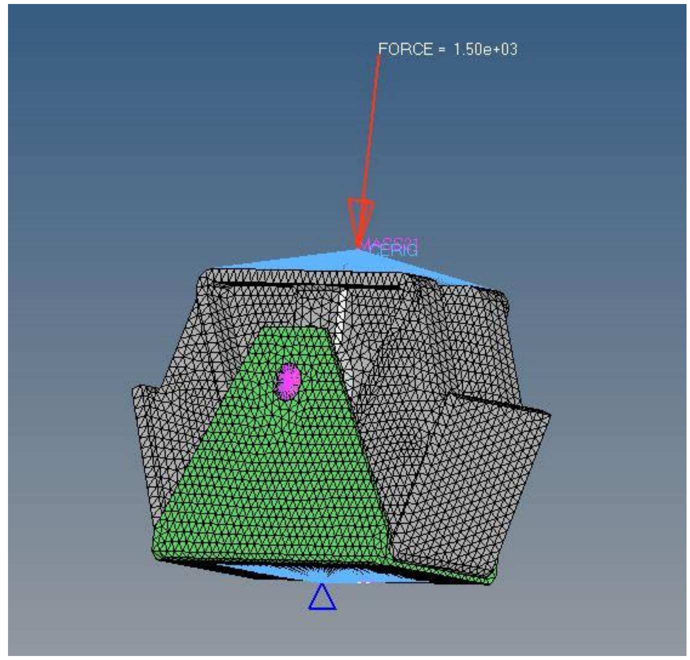

Fig.5 Boundary Conditions and Forces on EMB

D. Von Misses stresses in Engine Mounting Bracket

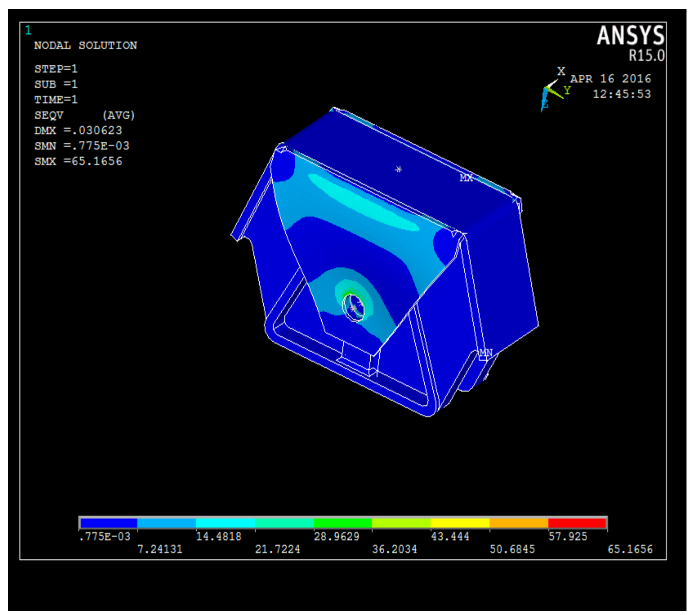

Fig. 6 Von Misses stresses in Engine Mounting Bracket

VI MODAL ANALYSIS

Modal analysis is the study of the dynamic properties of structures under vibrational excitation. Modal analysis uses the overall mass and stiffness of a structure to find the various periods at which it will naturally resonate. These periods of vibration are very important to note in vibration of any machine, as it is imperative that a components or nearby system's natural frequency does not match the frequency of machine. If a structure's natural frequency matches a component's frequency, the structure may continue to resonate and experience structural damage.

Physically "The mode shape" is the shape of the deformed structure if it is excited by a dynamic force which has the same frequency as the natural frequency of the structure. The mode shape has no unit.

A. 1ST Mode Shape

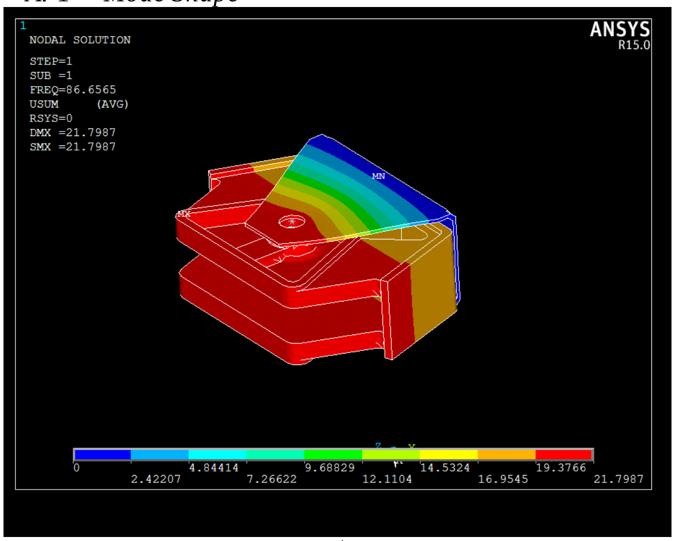

Fig.7 Frequency of 1st Mode is **86.65** hz.

B. 2nd Mode Shape

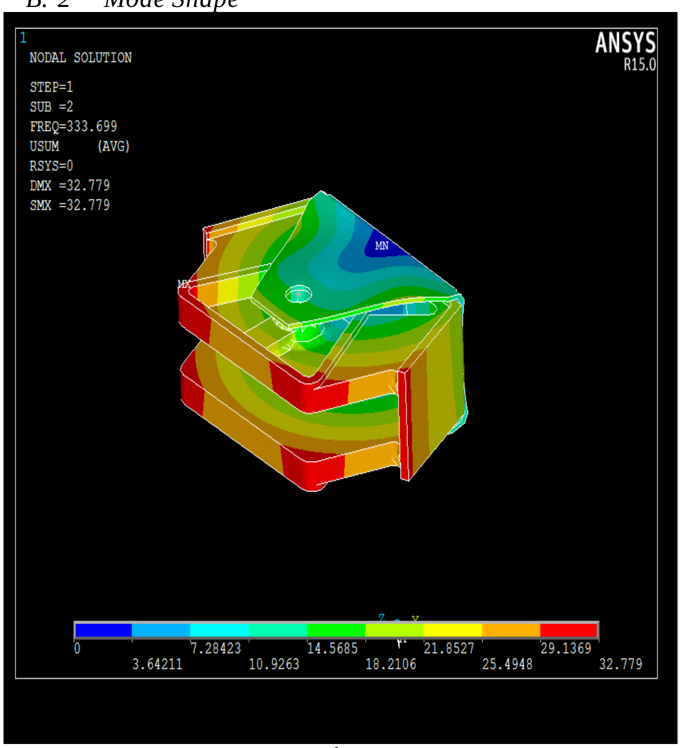

Fig.8 Frequency of 2nd Mode is **333.69** Hz.

C. 3nd Mode Shape

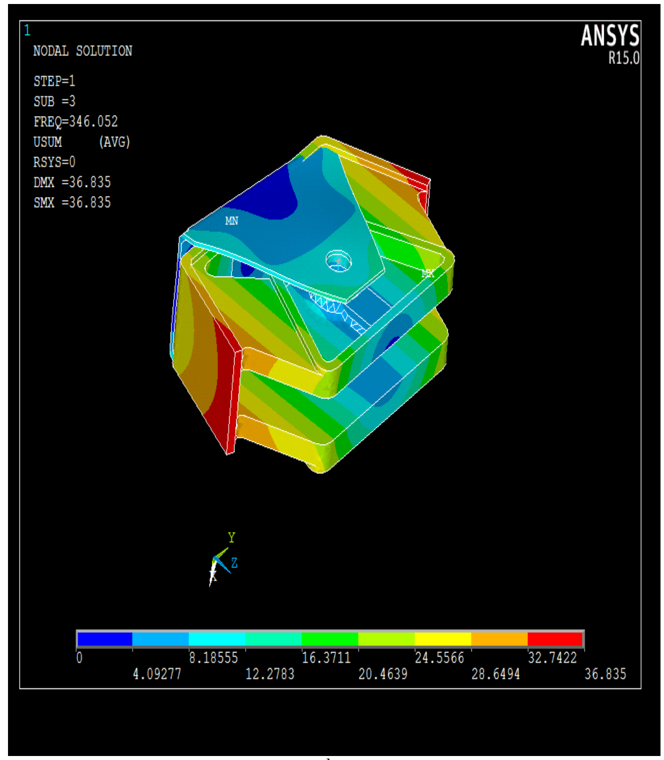

Fig.9 Frequency of 3rd mode is **346.05** Hz.

D. 4th Mode Shape

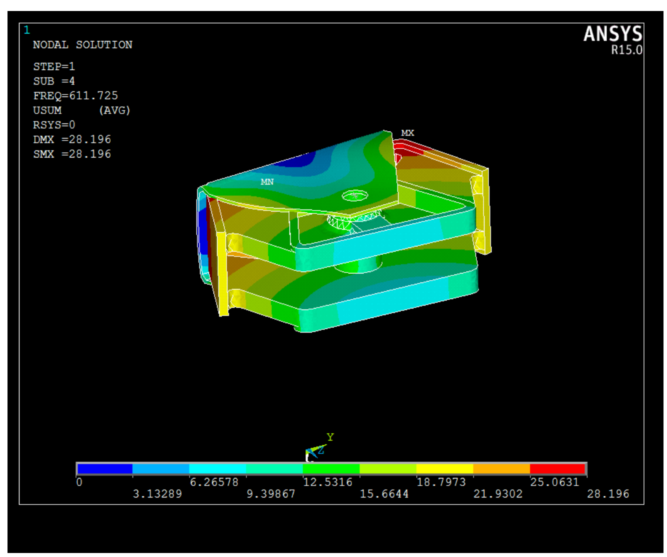

Fig.10 Frequency of 4th mode is **611.72** Hz.

E. 5th Mode Shape

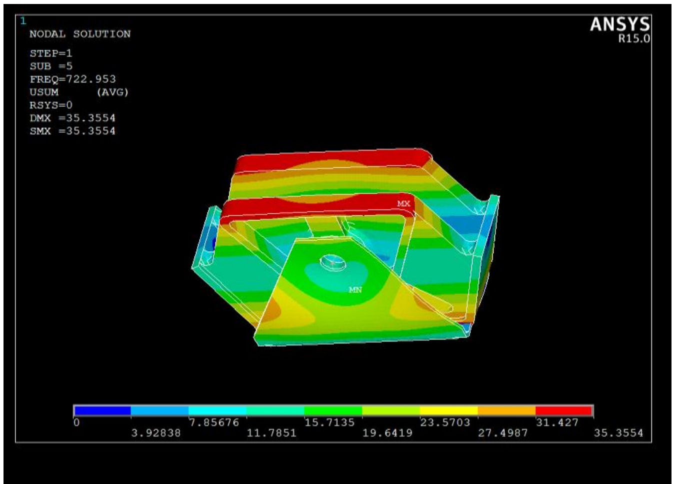

Fig.11 Frequency of 5th Mode shape is 322.95Hz

F. 6th Mode Shape

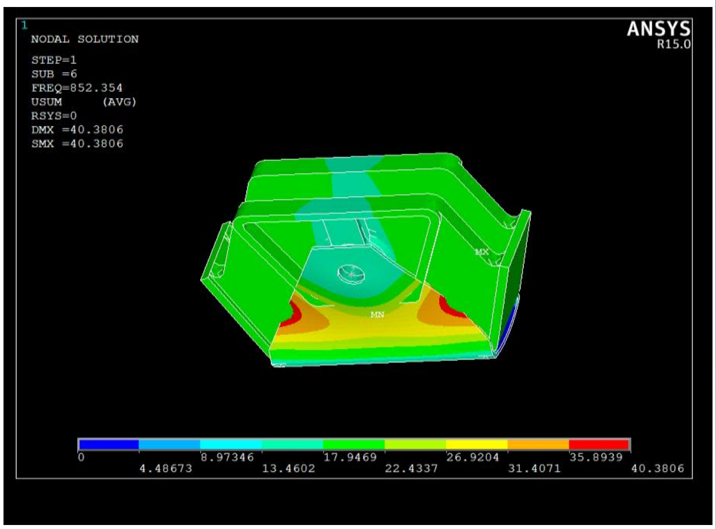

Fig.12 Frequency of 6th mode is **852.35** Hz.

VII. COMPOSITE MATERIAL

Composite materials consist of a combination of materials that are mixed together to achieve specific structural properties. The individual materials do not dissolve or merge completely in the composite, but they act together as one. Normally, the components can be physically identified as they interface with one another. The properties of the composite material are superior to the properties of the individual materials from which it is constructed. An advanced composite material is made of a fibrous material embedded in a resin matrix, generally laminated with fibers oriented in alternating directions to give the material strength and stiffness.

Properties which can be improved by using composite material are strength, fatigue life, stiffness temperature dependent behavior, corrosion resistance, thermal insulation; wear resistance, thermal conductivity, attractiveness, acoustical, insulation and weight.

Table 2 Material properties of Glass Fibers

Property	**Value**
E_1	40 GPa
E_2	6 GPa
E_3	40 GPa
Poisson’s Ratio ,ν	0.24
G_{xy}	15 GPa
G_{yz}	2.3 GPa
Gz_x	15 GPa
Density, ρ	2000 kg/m^3

A.Meshed Model for Composite Engine Mounting Bracket

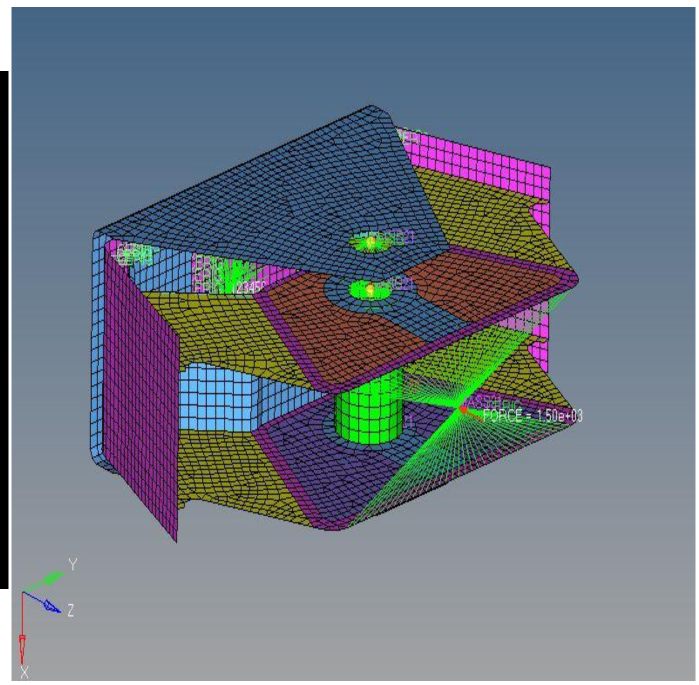

Fig.13 Meshed Model of Composite EMB

B. Displacement of Composite Engine Mounting Bracket

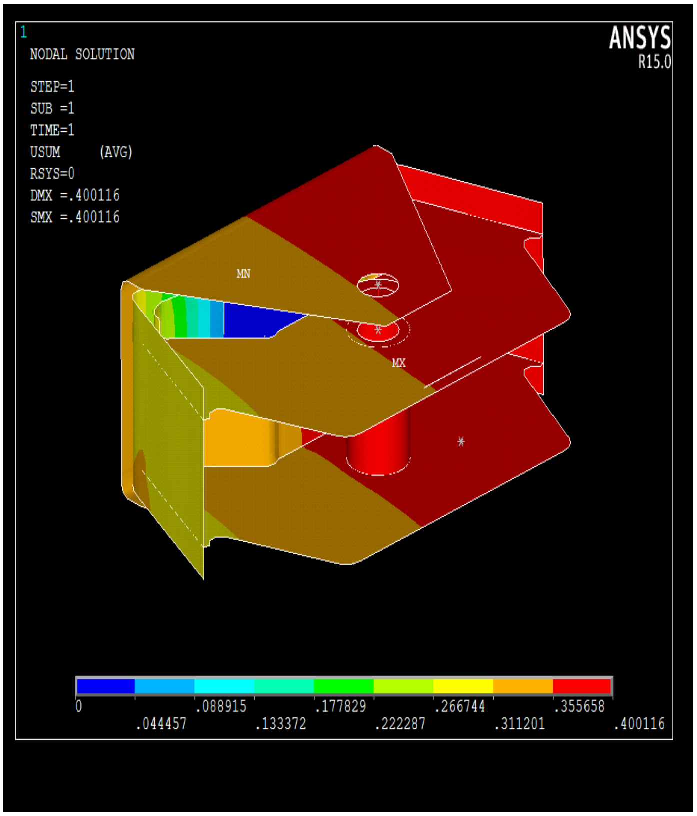

Fig.14 Displacement of Composite EMB

C. Von- Misses Stresses for Composite Engine Mounting Bracket

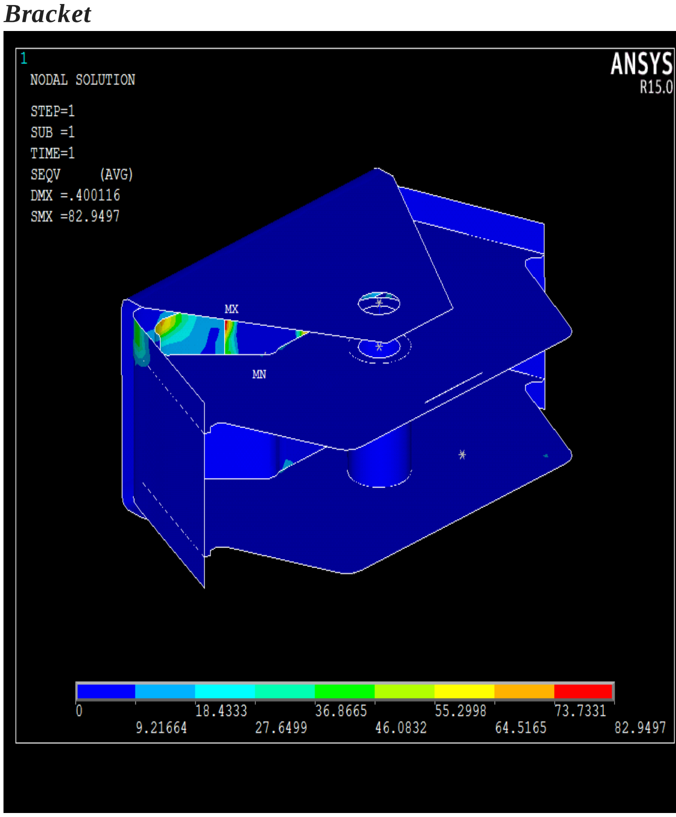

Fig. 15 Von Misses Stresses of Composite EMB

VIII MODE SHAPES OF GLASS FIBER ENGINE MOUNTING BRACKET

A. 1st Mode Shape

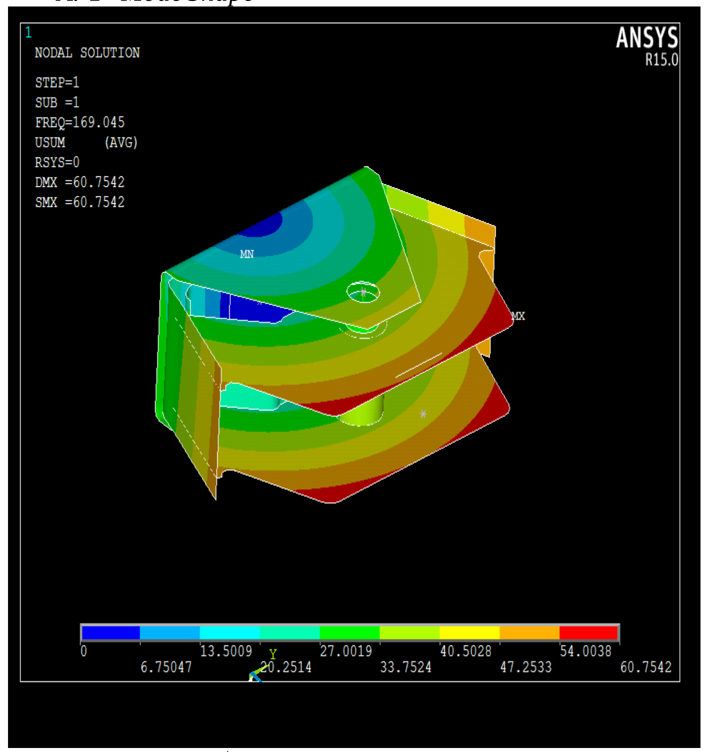

Fig. 16 1st mode shape frequency is 169.04Hz

B. 2nd Mode Shape

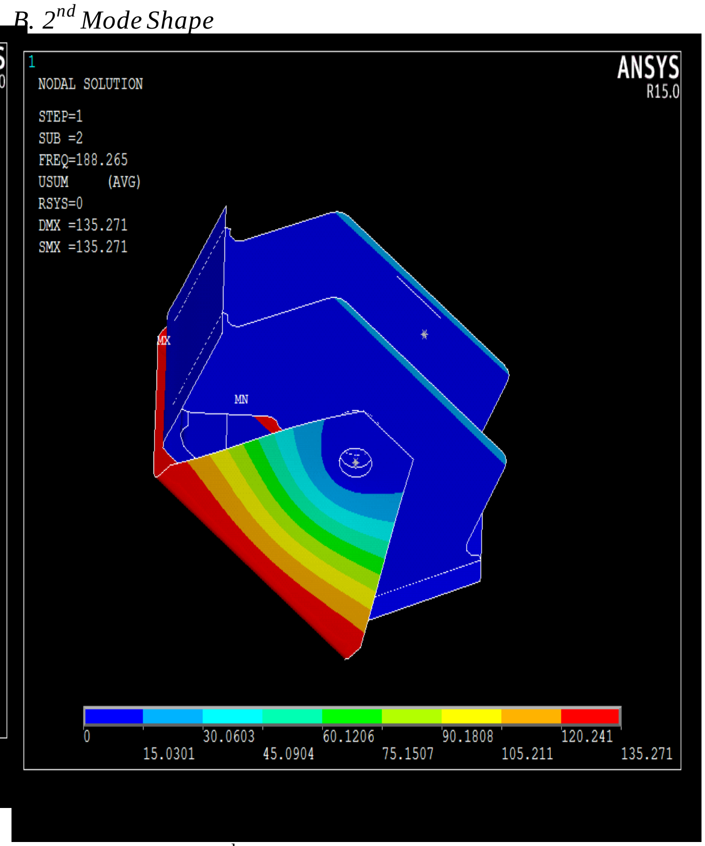

Fig. 17 2nd mode shape frequency is 169.04Hz

C. 3rd Mode Shape

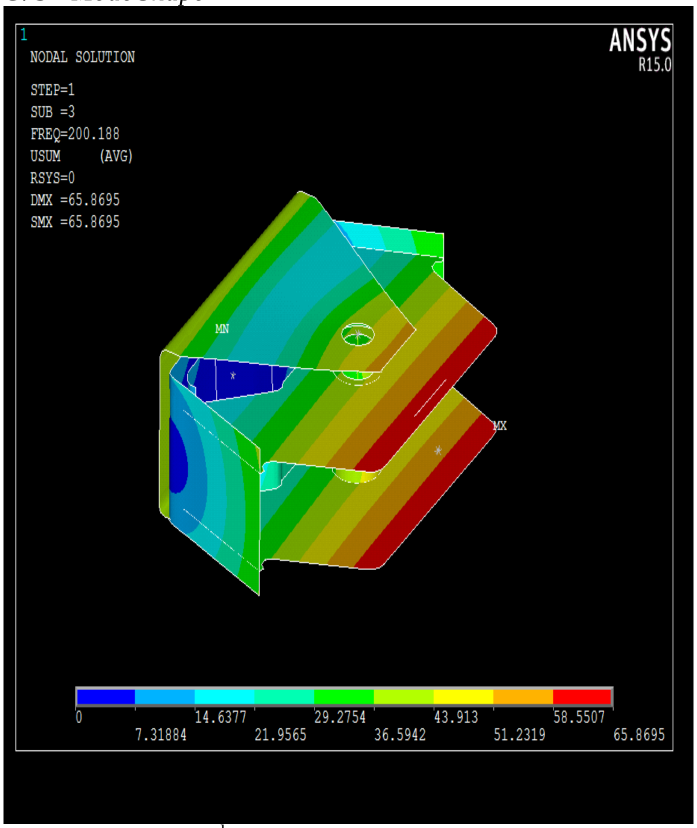

Fig. 18 3rd mode shape frequency is 200.18Hz

D. 4th Mode Shape

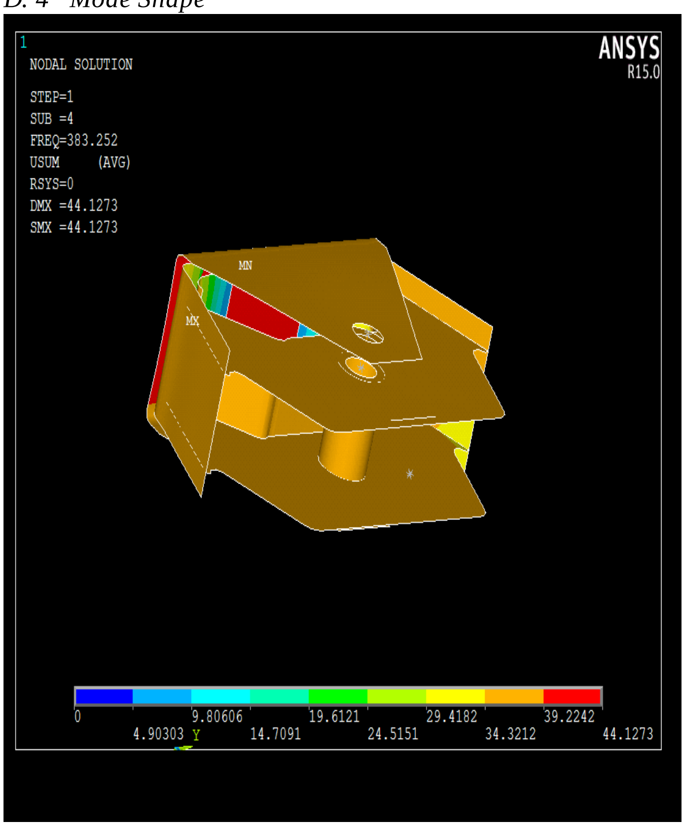

Fig. 19 4th mode shape frequency is 383.252Hz

E. 5th Mode Shape

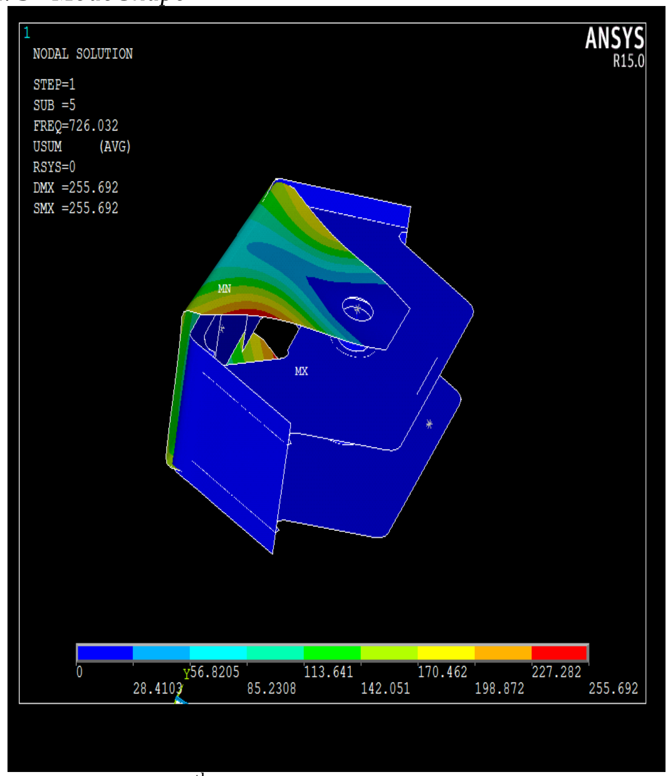

Fig. 20 5th mode shape frequency is 726.03Hz

F. 6th Mode Shape

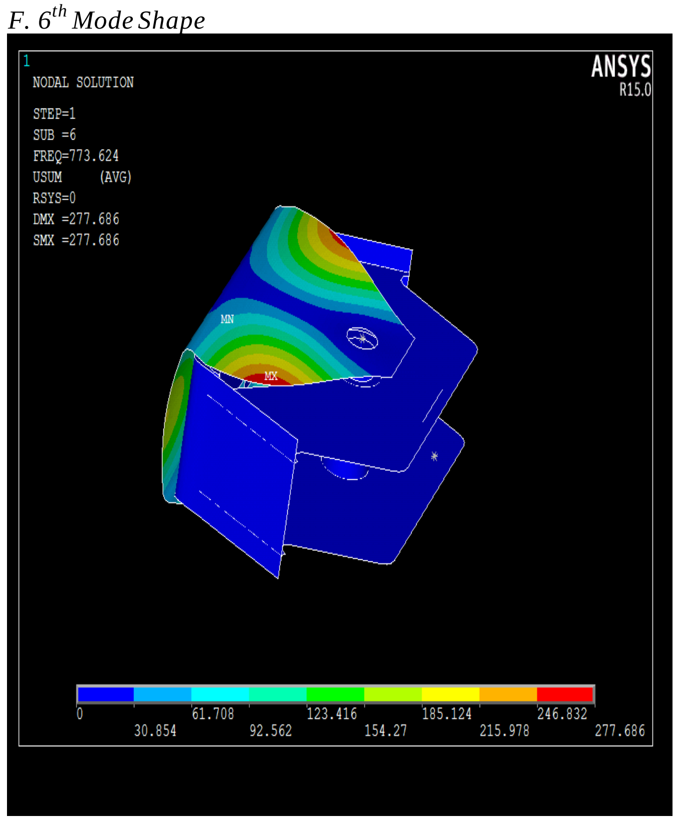

Fig. 21 6^{th} mode shape frequency is 773.62Hz

IX RESULTS AND DISCUSSION

A. Weight of the existing steel bracket is 2.83kg

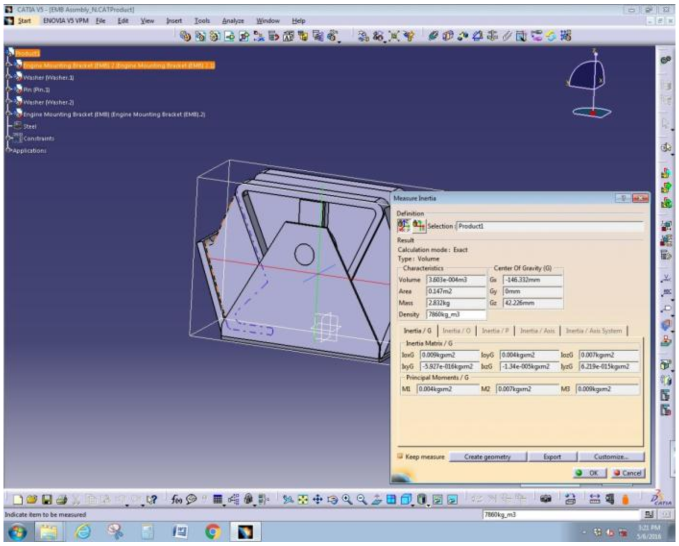

Fig. 22 Weight of steel Engine Mounting Bracket

B. Weight of the suggested composite model is 0.7kg

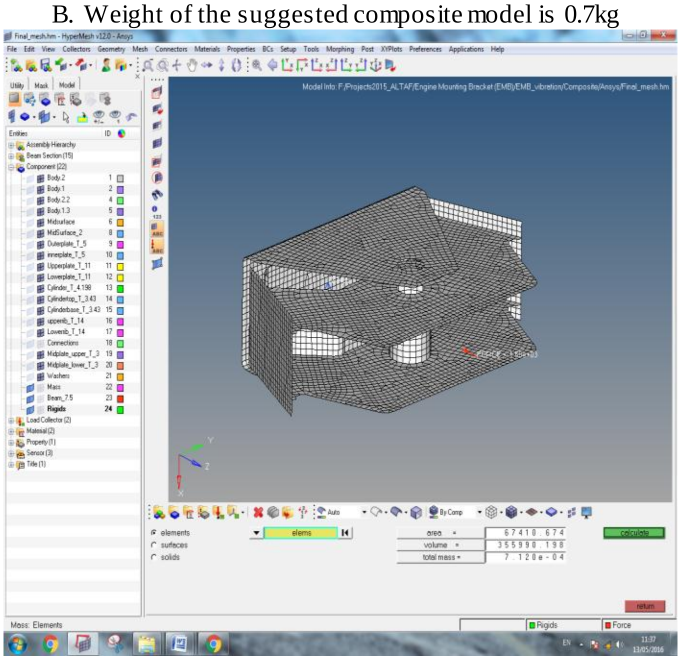

Fig. 23 Weight of composite Engine Mounting Bracket

As from above figures, it is clear that weight of the existing engine mounting bracket is 2.83 kg and weight of the suggested engine mounting bracket is 0.7kg. So it gives the weight reduction upto 75%.

Table 3 Comparison of Material Properties of Steel & Glass Fiber

S.No.	Material	Young's Modulus E	Poisson's Ratio υ	Density ρ	Yield Stress σ_{yield}	Ultimate Tensile Stress σ_{uts}
1	Steel	210 GPa	0.3	7850 kg/m^3	250 MPa	390 MPa
2	Glass Fiber	E1-40 GPa E2-6 Gpa	0.35	2000 kg/m^3	NA	2000 MPa

Table 4 Comparison of Analysis Results

Sr. No	Material	Max. Stress	Max. Displacement	Weight
1.	Steel	184.4 MPa	0.13 mm	2.78
2.	Glass Fiber	284.6 MPa	0.88 mm	1.98

X CONCLUSION

From results of finite element analysis it is observed that both steel and carbon glass fiber materials have stress values less than their respective permissible yield stress values. So the design is safe. From analysis results and comparison of properties of all the analysis, it is found that glass fiber is the material which is having the least density; also it is easily available. Also machining cost for glass fiber is less. Hence it is the best suited alternate material for engine mounting bracket and is expected to perform better with satisfying amount of weight reduction. The natural frequencies at each mode found after modal analysis of the suggested model are more than existing model; hence the bracket is safe under the vibrations.

XI FUTURE SCOPE

Fabricate the suggested composite glass fiber model of engine mounting bracket and natural frequency analysis is done by using FFT (Fast Fourier Transform) analyzer. Again other types of composite material like concrete, reinforced plastics, metal composites and ceramic composites can be used to reduce the weight of EMB. Also we can avoid the resonance condition and failure of EMB due to this.

References

[1] Umesh S. Ghorpade, D. S. Chavan, Vinaay Patil, Mahendra Gaikwad, "Finite Element Analysis And Natural Frequency Optimization Of Engine Bracket", International Journal of Mechanical and Industrial Engineering, ISSN No. 2231 –6477, Vol-2, Iss-3, 2012

[2] Abdolvahab Agharkakli, Digvijay Pradip Wagh, "Linear Characterization of Engine Mount and Body Mount for Crash Analysis", International Journal of Engineering and Advanced Technology ISSN: 2249 – 8958, Volume-3, Issue-2, December 2013

[3] Dr.Yadavalli Basavaraj, Manjunatha.T.H, "Design Optimization of Automotive Engine Mount System", International Journal of Engineering Science Invention ISSN (Online): 2319 – 6734, ISSN (Print): 2319 – 6726

[4] P.D. Jadhav , Ramakrishna, "Finite Element Analysis of Engine Mount Bracket", International journal of advancement in engineering technology, management and applied science ISSN no. 2349-3224

[5] Vikas Doundkar, Dhananjay Ghatage, Manish Madkaikar, Aniruddha Kulkarni, " Topology Optimization for Engine Mounting Arm with Fatigue", HTC2011

[6] Koushik . S"Static and Vibration Analysis of Engine Mounting Bracket of TMX 20-2 using OptiStruct", Altair Technology Conference 2013, India.

[7] R Singh "Dynamic design of automotive systems: Engine mounts and structural joints", Saadhanaa, Vol. 25, Part 3, June 2000, pp. 319±330.

[8] M.V. Aditya Nag "Material Selection and Computational Analysis on DOHC V16 Engine's Mounting Bracket Using COMSOL Multiphysics Software", 2012 COMSSOl conference

[9] Monali Deshmukh, Prof. K R Sontakke"Analysis and Optimization of Engine Mounting Bracket", International Journal of Scientific Engineering and Research, ISSN (Online): 2347-3878

[10] Chang Yong Song "Design Optimization and Development of Vibration Analysis Program for Engine Mount System", Technical Division, AhTTi Co., Ltd.,Korea

[11] K. Diwakar, Dr. B.E Kumar, "Design and Weight Optimization Mounting Bracket", International Journal of Advanced Engineering Research and Science, ISSN: 2349-6495

[12] V.S.Kathavate, A.S.Adkine, "Computational Fluid Dynamics: Research Tool for Analysis of Various Parts of Automotive Systems", International Conference on Global Trends in Engineering, Technology and Management, ISSN: 2231-5381

[13] Gaurav Bhusari, Prof. M Sohail Pervez, "Design and Analysis of fixture bracket engine assembly line" International Journal for Innovative Research in Science & Technology, Volume 2 | Issue 05 | October 2015 ISSN (online): 2349-6010

[14] K.S. Raghavan, T. Howard, J. Buttles, N. Law, "Weight Reduction of Structural Vibration Isolation Hydro- Mount Bracket through Design Analysis and Use of Advanced High Strength Steels", SAE International, 09M-0168

[15] Sebastian C S, Shibin Baby, Anthony Tony, "Design And Optimization Of Engine Mount Bracket", International Journal of Application or Innovation in Engineering & Management, ISSN 2319 - 4847

[16] T. Ramachandran, K. P. Padmanaban, "review on internal combustion engine vibrations and mountings", International Journal of Engineering Sciences & Emerging Technologies, ISSN: 2231 – 6604