International Journal of Scientific Engineering and Research (IJSER)
www.ijser.in
ISSN (Online): 2347-3878, Impact Factor (2014): 3.05

Analysis and Optimization of Engine Mounting Bracket

Monali Deshmukh[1], Prof. K R Sontakke[2]

[1]Student of ME CAD/CAM, Pankaj Laddhad Institute of Technology and Management Studies, Buldana, Sant Gadge Baba Amravati University

[2]Department of Mechanical Engineering, Pankaj Laddhad Institute of Technology and Management Studies, Buldana, Sant Gadge Baba Amravati University

Abstract: ***The engine mounting plays an important role in reducing the noise, vibrations and harshness for improving vehicle ride comfort. The brackets on the frame that support the engine undergo high static and dynamic stresses as well as huge amount of vibrations. Hence, dissipating the vibrations and keeping the stresses under a pre-determined level of safety should be achieved by careful designing and analysis of the mount brackets. Keeping this in mind in this paper, Static Analysis, model analysis of engine mounting is done as well as harmonic response analysis of bracket as a part of dynamic analysis is performed with the FEA software package ANSYS 15.0.The existing model is optimized and a novel model was proposed to reduce the weight of the rib of the engine mounting bracket as well as the harmonic response in term of acceleration is checked to ensure that the proposed model will not result in to noisy operation. The results of the stresses, deformations and harmonic response for the both models of the engine mounting bracket were compared to each other. With the proposed model of the engine mounting bracket 12.5%weight reduction is achieved maintaining an acceptable level of yield stress and harmonic response***.

Keywords: Mounting Bracket, Static Analysis, Dynamic Analysis, Optimization

1. Introduction

Design engineers always aims at improvement in each and every part of automobile system. Automobile industry continued improving since from many years with the efforts conducted for the purpose of modification of the mechanical parts of vehicles in order to improve their performance and vibration response. These characteristics have a vital impact on the mechanical performance of the overall system balance. In addition, redesigning the mechanical models play an important role in improving the sustainability of the system against the resultant stresses and strains, therefore, significant consideration should be taken for this when designed by engineers.

In an automotive vehicle, the engine rests on brackets which are connected to the main-frame or the skeleton of the car. Hence, during its operation, the undesired vibrations generated by the engine and road roughness can get directly transmitted to the frame through the brackets. This may cause discomfort to the passenger(s) or vibrations might even damage the chassis. Also, at high operating frequencies noise becomes a serious concern. Hence, damping of these engine vibrations and checking the harmonic response of bracket while designing is essential. The Noise & Vibration Harness analysis (or NVH) is one the most important considerations in automotive designing today. If the brackets have their resonance frequencies close to the operating engine frequencies, then the large amplitude of vibration get generated which may cause its fatigue failure or breakage, thus reducing its estimated or desired life. And if the harmonic response values of bracket is more than acceptable range it results in to generation of noise. Hence it is required to check the harmonic response of designed bracket.

Vibration damping can be either provided by using separate dampers (anti-vibration mounts) or by suitably deciding the material and dimensions of the brackets. Moreover, the brackets also undergo deflection under static and dynamic loads. This deflection should be under permissible limits.

Some vehicle required to be highly maneuverable and quick with high rates of acceleration and deceleration, Hence the mounting of the engine should be well constrained and the mount brackets need to be light-weight and designed to safely bear the inertial loads and maximize vibration transmission. The response of vibrating system can be in the form of displacement, velocity, acceleration or sound. These have to be kept under safe or acceptable limits while designing. Here the response in term of acceleration is taken into focus. Static structural and model analysis of bracket is done and results are used for analyzing harmonic response of bracket. Existing bracket design is optimized. The modified designed has been reanalyzed using FEA before finalization.

2. Methodology

In this paper, Static structural and Model analysis were used to determine the characteristics of the engine mounting bracket. And Harmonic response analysis is done to check the response of structure in terms of acceleration. To obtain a comprehensive design, the existing model is modified by changing its rib geometry. Again the modified bracket had undergone same analysis procedure and the results were compared for both the designs.

The purpose was to determine the stresses, deformations and harmonic response of bracket that affect's on the engine mounting brackets and to optimize the design that make a reduction in the weight of the rib of the engine mounting models. Work has been carried out with ANSYS 15.0 software.

3. Analysis of Engine Mounting Bracket

3.1 Existing Model of Engine Mounting Bracket:

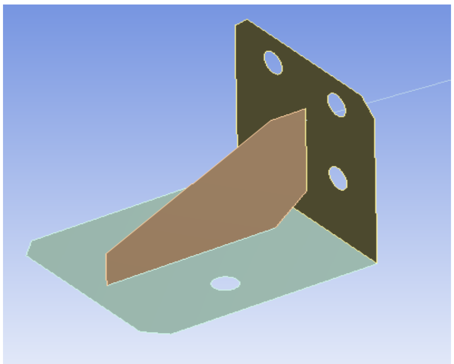

Figure 1: Engine mounting bracket

The model shown above is taken for analysis. The existing model of an engine mounting bracket is a sheet metal part. And it has less curvatures and the thickness of the part is uniform unlike model created by casting procedure. Hence for analysis purpose surface model of bracket was imported as shown in figure 1 instead of solid model.

3.1.1Boundary conditions And Properties

The engine mounting bracket to be analyzed is a part of engine weighing 170kg. Torque used as an input is 1×10^5 N-mm. Isolators stiffness is 200 N/mm in Y direction and 100N/mm in X and Z directions. Idling: 5000 rpm and maximum: 9000 rpm. Existing model is made up of structural steel with following properties:

Young's modulus =2e5 Mpa
Poisons ratio = 0.33
Density =7850 kg/m^3
Yield strength = 270 Mpa

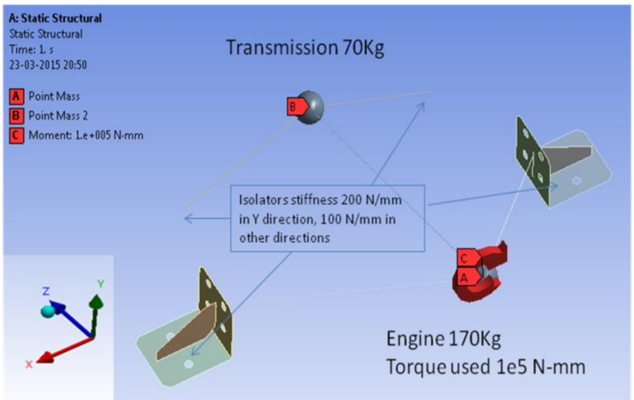

Figure 2: Boundary conditions

3.2 Static Structural Analysis

A static structural analysis is the analysis displacements, stresses, strains and forces on structure or a component due to load application. The structures response and loads are assumed to vary slowly with respect to time. There are various types of loading that can be applied in this analysis which are externally applied forces and pressures, and temperatures.

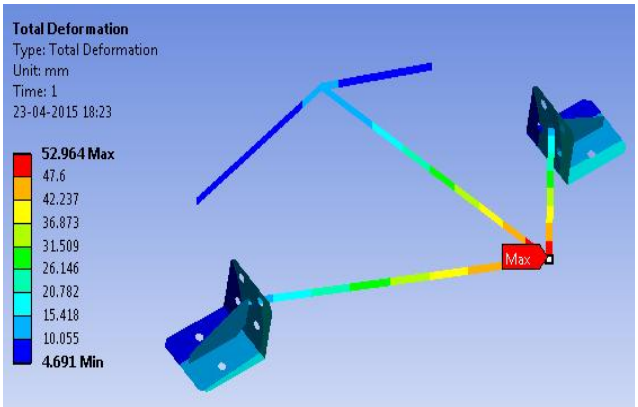

Figure 3: Total deformation

The total deformation is 52.964mm for existing bracket as shown in figure 3

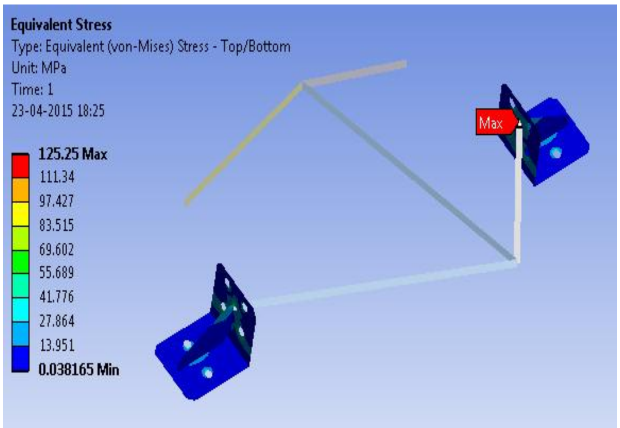

Figure 4: Von-Mises stress

Figure 4 shows equivalent Von-Mises stresses. Von- Mises stresses are resultant stresses or yield strength of material, after which deformation starts. Maximum value of stress is 125.25Mpa and the color plot shows the portion where the maximum amount of stress acts.

3.3. Modal Analysis

Modal analysis determines the vibration characteristics of a structure or a particular component in the form of natural frequencies and mode shapes. From this analysis we can do more detailed dynamic analysis such as transient dynamic analysis, harmonic analysis or spectrum analysis. The natural frequencies and mode shapes are important in the design of a structure for dynamic loading conditions. In this analysis, only linear behavior is valid. Damping is not considered and applied loads are ignored in modal analysis. A static structural analysis is required first for performing pre stressed modal analysis. This analysis was done to find out the natural frequencies and mode shapes of the bracket.

The operating frequency range is 250Hz-450Hz.

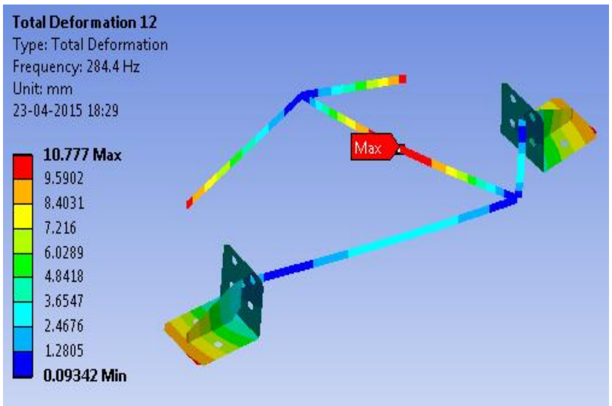

Figure 5: Mode 12

Figure 5 shows twelfth mode of vibration. The frequency of vibration for this particular mode is 284.4 Hz. Figure 6 shows thirteenth mode of vibration and the frequency of vibration for this mode is 377.36Hz.

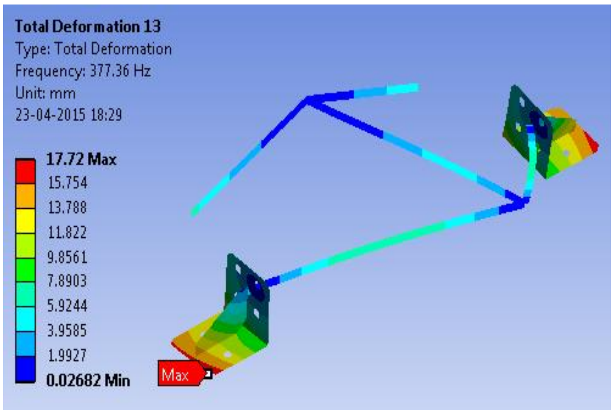

Figure 6: Mode 13

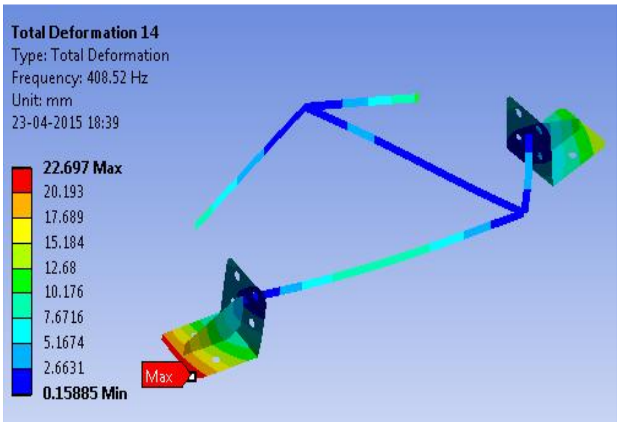

Figure 7: Mode 14

Figure 7 shows fourteenth mode of vibration and here bracket is subjected to maximum deformation at the end section of its base. The frequency of vibration for this particular mode is 408.52 Hz.

Figure 8 shows fifteenth mode of vibration. The frequency of vibration for this particular mode is 408.52 Hz.

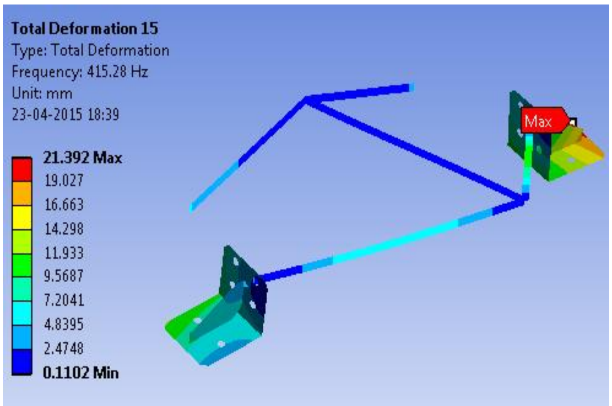

Figure 8: Mode 15

3.4 Harmonic Response Analysis

3.4.1Definition of Harmonic Response Analysis

Any sustained cyclic load will produce a sustained cyclic response (a harmonicresponse) in a structural system. Harmonic response analysis gives you the ability to predict the sustained dynamic behavior of your structures, thus enabling you to verify whether or not your designs will successfully overcome resonance, fatigue, and other harmful effects of forced vibrations.

3.4.2 Uses of Harmonic Response Analysis

Harmonic response analysis is a technique used to determine the steady-state response of a linear structure to loads that vary harmonically with time. The idea is to calculate the structure's response at several frequencies and obtain a graph of some response quantity (usually displacements) versus frequency. "Peak" responses are then identified on the graph and stresses reviewed at those peak frequencies.

Result from model analysis is used as input for calculating harmonic response. Acceleration of 3×G = 3×9806.6 = 29420 mm/s^2 is used. The engine operating frequency range is 250 Hz to 450Hz, modes 12 to 15 are within engine operating frequency range. Hence mode shape 13 is chosen and its maximum deformation location is used for checking response output.

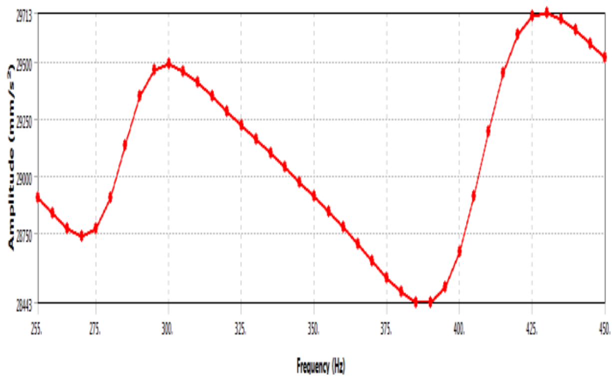

Figure 9: Harmonic response graph

Results were plotted for harmonic response of bracket with 40 solution intervals. Figure 9 shows graph between amplitude (mm/s^2) and frequency (Hz). Graph shows 40 points for which response is calculated. For engine operating frequency range 250 to 450 Hz the amplitude varies from 28443 mm/s^2 to 29713 mm/s^2.

4. Optimization of Engine Mounting Bracket

4.1 Suggested Model:

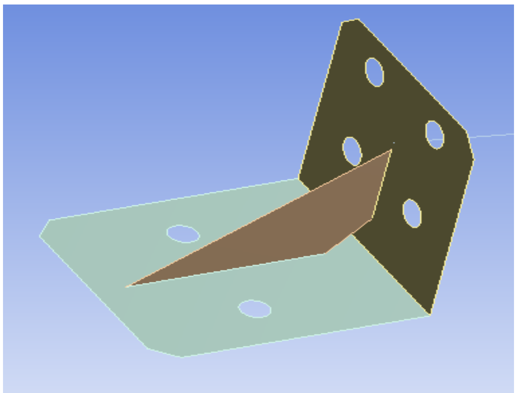

Figure 10: Suggested model

The model shown in figure 10 is optimized model. After studying the result of analysis of engine mounting bracket, certain design changes have been made here. Here in this optimized model, geometry of rib which connect two faces of bracket is changed. Suggested optimized model has a rib with reduced cross sectional area than previous model. Hence it results in to weight reduction of overall structure.

But to finalize the optimized design, it is needed to analyze the bracket with same boundary conditions applied to the previous model. And if the results obtain fall within the limits obtained from analysis of initial (original) design, then only the optimized design can be finalized.

4.2 Static Structural Analysis

Static structural analysis is performed for the optimized design with same boundary conditions and input values.

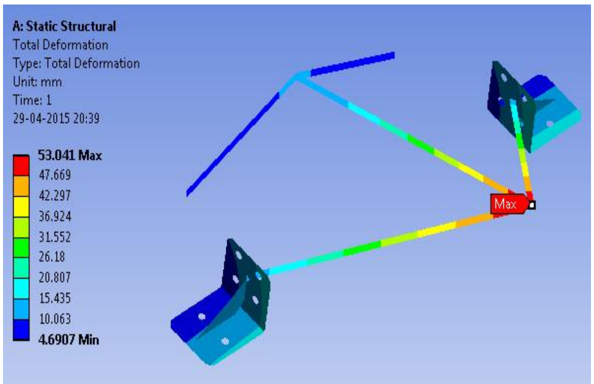

Figure 11: Total deformation

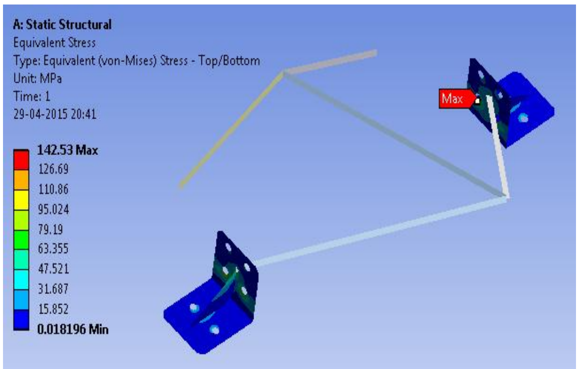

Figure 12: Von-Mises stress

4.3 Model Analysis

This analysis was done to find out the natural frequencies and mode shapes of the bracket.

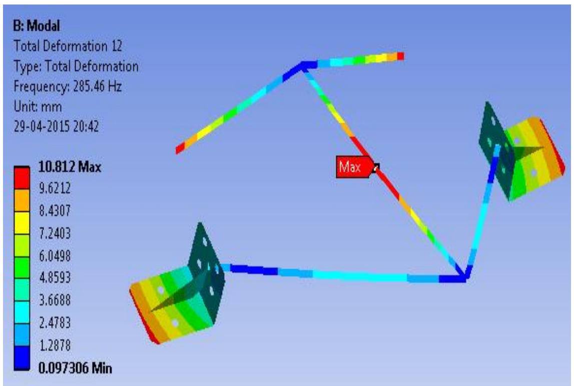

Figure13: Mode 12

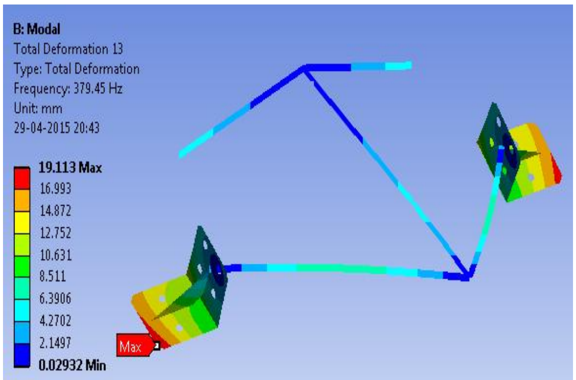

Figure 14: Mode 13

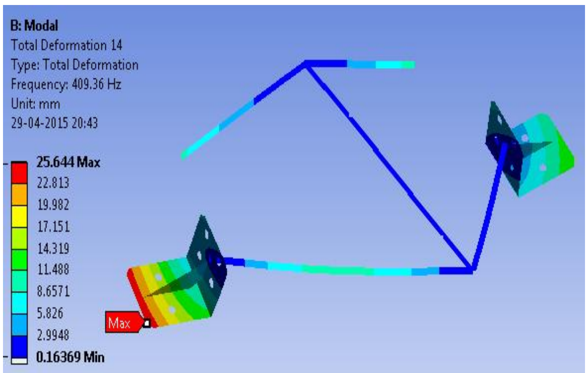

Figure 15: Mode 14

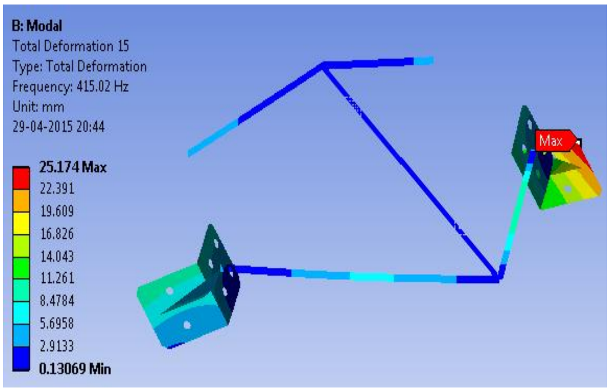

Figure 16: Mode 15

4.4 Harmonic Response Analysis

Result from model analysis is used as input for calculating harmonic response. Acceleration of 3×G = 3×9806.6 = 29420 mm/s^2 is used. As the engine operating frequency range is 250 Hz to 450Hz, mode shape 13 is chosen and its maximum deformation location is used for checking response output.

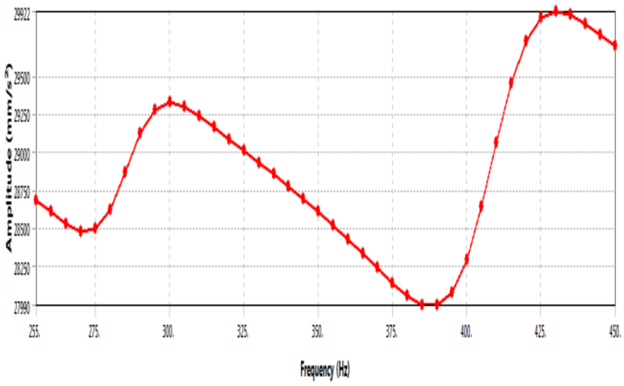

Figure 5.13: Harmonic response graph.

Results were plotted for harmonic response of bracket with 40 solution intervals. Figure 5.13 shows graph between amplitude (mm/s^2) and frequency (Hz). Graph shows 40 points for which response is calculated. For engine operating frequency range 250 to 450 Hz the amplitude varies from 27990 mm/s^{2}to 29922 mm/s^2.

5. Result and Discussion

Result obtained from static structural analysis, model analysis and harmonic analysis are listed and compared with each other in this section.

Table 6.1: Comparison analysis result

	Initial Bracket	Optimized Bracket
Total deformation (max)	52.964 mm	53.041 mm
Von-Mises stresses (max)	125.25 Mpa	142.53 Mpa
Harmonic response (mm/s^2)	28443 to 29713	27990 to 29922
Weight of bracket	2 kg	1.75 kg

Engine operating frequency range lies between 250 to 450 Hz and the amplitude varies from 28443 to 29713 mm/s^2 for initial un-optimized bracket. For optimized bracket amplitude varies from 27990 to 29922 mm/s^2.

Response of optimized structure in term of vertical acceleration shows slightly greater value of maximum acceleration than initial bracket design, but it shows very little difference. The resultant harmonic response of optimized structure is well within safe limit. Hence it is acceptable. But if the amplitude shows noticeable deviation it results into noisy operation.

With comparison of all the resultant values it can be seen that deformation values of optimized bracket are nearly similar as that of initial bracket. Stress value for optimized bracket is slightly greater than that of initial bracket, but it lies within maximum permissible limit range of bracket hence is acceptable. And the values of harmonic response in term of vertical acceleration for optimized bracket also nearly match with initial bracket and hence the value is acceptable.

6. Conclusion

The optimization of engine mounting bracket is attempted by applying certain changes in its design and shape. After comparison of results obtained from analysis performed, it is concluded that the optimization attempted is found successful. And with the optimization done in the bracket it is found that as the cross sectional area of rib is reduced the overall weight of the bracket is also reduced. The modified design of bracket is obtained which is 12.5% lighter by weight than initial on-optimized bracket. This results in material saving, and overall cost reduction.

Also it is confirmed by harmonic analysis that the harmonic response of modified bracket is within safe limit. So the chances of noisy operation of structure due to its design are minimized.

References

[1] SahilNaghate and Sandeep Patil (2012), "Modal Analysis of Engine Mounting Bracket Using FEA", International Journal of Engineering Research and Applications (IJERA), Vol. 2, 1973-1979.
[2] K A Zakaria, S Abdullah, M J Ghazali, M Z Nuawi, S M Beden and Z M Nopiah (2013) "Fatigue Strain Signal Behavior and Damage Assessment of an Engine Mount Bracket", Journal of scientific and Industrial Research Vol. 73, February 2014, pp 112-116.
[3] Umesh Ghorpade, D.S.Chavan, Vinaay Patil and Mahendra Gaikwad,"Finite Element Analysis and Natural Frequency Optimization of Engine Bracket", International Journal of Mechanical and Industrial Engineering (IJMIE) ISSN No.2231-6477, Vol-2, Iss-3,2012
[4] Pramod Walunje and prof. V. K. Kurkute (Dec 2013) "Optimization of Engine Mounting Bracket Using FEA", PARIPEX – International journal of research.
[5] Pavan B. Chaudhari and Dr. D. R. Panchagade (2012), "Comparison of Magnesium, Aluminium and Cast Iron to obtain Optimum Frequency for Engine Bracket using Finite Element Analysis", International Journal of

Engineering Research andApplications (IJERA) , Vol. 2, 1016-1020.

[6] Sameer U. Kolte, Dvid Neihguk, and Abhinav Prasad, “Structural Durability Analysis of Powertrain Mounting Bracket”, excerpt from the Poceedings of the 1012 COMSOL conference in Bangalore.

[7] M V Aditya Nag, “Material Selection and Computational Analysis on DOHC V16 Engine’s Mounting Bracket Using COMSOL Multiphysics Software” excerpt from the Poceedings of the 1012 COMSOL conference in Bangalore.

[8] Haval Kamal Asker, “Comparison of The Mechanical Properties Of Different Models of Automotive Engine Mounting” ARPN Journal of Engineering and Applied Sciences, Vol. 8, no. 6, June 2013

[9] P.D. Jadhav, Ramakrishna, “Finite Element Analysis of Engine Mount Bracket” International Journal of Advancement in Engineering Technology, Management and Applied Science, Vol. 1, September 1014.

Author Profile

Monali Deshmukh received the B.E. degree in Mechanical Engineering from Anuradha Engineering College, Chikhli affiliated to Sant Gadge Baba Amravati University, Amravati in 2013. She is a research scholar, presently pursuing ME 2nd year from Pankaj Laddhad Institute of Technology and Management Studies, Buldana affiliated to Sant Gadge Baba Amravati University, Amravati.