ISSN : 2455-9679
Impact Factor : 3.805

IJRTSM

INTERNATIONAL JOURNAL OF RECENT TECHNOLOGY SCIENCE & MANAGEMENT

"A STUDY ON OPTIMIZATION OF ENGINE MOUNTING BRACKET"

Shiva Goud [1], Mr. Dheeraj Mandliya [2]

[1] M.Tech Scholar, Dept. of Mechanical Engineering, Malwa Institute of Technology, Indore ,MP, India

[2] Assistant Professor, Dept. of Mechanical Engineering, Malwa Institute of Technology, Indore ,MP, India

ABSTRACT

The purpose of an engine mounting bracket is to safely support the power-train system in all conditions. Many researcher were studied the engine mounting bracket. Vibration and fatigue analysis has been carried out to know the structural failure. Structural failure will occur if vibrations and stresses are excessive and severe. Performed topology and gage optimization on three automotive components, that is, transfer case assembly, step bar module and shock reinforcement assembly for a truck with the objective to reduce the mass without compromising on the structural requirements of the components Since it is very difficult to change the supporting locations and types of support after the engine is built, the mounting brackets must be verified in the design stage. This paper includes study of design and analysis of engine mounting bracket. CAD model has been generated through reverse engineering. Engine mounting bracket of Mahindra Scorpio has been taken into study. After analyzing it, Scope for the optimization of engine bracket is suggested.

Key Words: *CAD, Engine, bracket, optimization, Vibration.*

I. INTRODUCTION

An engine mount is the part that holds the engine to the body or to the engine cradle (sub-frame) of the car. In a typical car, the engine and transmission are bolted together and held in place by three or four mounts. If the mounting bracket does not have appropriate stiffness, it can cause noise and vibration. This vibration is passed from the mounting to the body causing body vibration. A weak bracket can also lead to rolling vibration of the engineer shock from deceleration and acceleration. Due to these factors, it is very important that the engine mounting bracket have enough stiffness and strength. Strength analysis needs to be performed to verify the bracket properties early in the design stage. The strength analysis computes the magnitude of a load from the mass of the engine, including factor of safety, and applies this load to each engine mounting bracket. The stress analysis is performed with these boundary conditions and the analyst verifies that results are within an acceptable range.

II. LITERATURE REVIEW

Umesh S. Ghorpad et. al [2012] has studied the engine mounting bracket. Vibration and fatigue analysis has been carried out to know the structural failure. Structural failure will occur if vibrations and stresses are excessive and severe. Prolonged exposure to whole-body vibration in the working environment may lead to fatigue and in some cases it damages the car. Generally, the most important vibration relevant excitations in a car engine can be identified as follows:- combustion force; main bearing reaction forces including mass forces damper function and flywheel whirling, modified by the front-end damper; piston side forces including secondary motion; camshaft bearing reaction forces

ISSN : 2455-9679
Impact Factor : 3.805

including mass forces, opening and closing impacts and bearing impacts; valve opening and closing impacts; valve train forces caused by chain/belt movement or gear drive; gear train forces inside the transmission; drive train reaction forces and moments. Automotive engine mounting system must satisfy the primary tasks such as engine movement, engine rigid body dynamic behavior, and vibration isolation. The design and development of mounting bracket through use of Ansys software to achieve the requirements for mounting system and optimize the mount.[1]

Abdolvahab et al [2013] The engine mount and body mount is a complex joint assembly comprising of rubber bushings on the top and bottom of the frame bracket, a bolt, and retainer. The engine/body mounts are designed to carry the horizontal impact load in an impact and to isolate the noise, vibration and harshness (NVH), occurring during driving from entering the passenger compartment. The authors in this paper have studied mathematical model and compared its results with MATLAB simulations. The mounts are treated at the component level, and mathematical models for the same are evaluated to get the required characteristics. The mounts are modelled as spring and damper system subjected to impact loading that occurs during crash events. The approximation of input pulse has been described mathematically, which then serves to find the characteristics of the mounts. The change in the characteristics of mounts with the change in the velocity of impact has also been studied. [2]

Dr.Yadavalli et al [2014] These authors have focused more on avoiding resonance and damping of frequency. If there exist unbalanced loads in engine body, resonant vibration occurs. This resonant vibration increases if chassis has unitary or frameless construction. This has forced designers to direct their attention to the development of high quality engine mounting devices in order to ensure that improved comfort in riding and silencing shall not be offset by fatiguing vibration effects. In this paper an engine mounting bracket is designed to reduce the transmission of engine vibration to the chassis. [3]

Monali Deshmukh et al [2014] In this paper, authors have studied harmonic response and vibration damping of an engine mounting bracket. 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. 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. Existing bracket design is optimized to meet the above requirements.[4]

Mrs. Monali Gund et al [2020] Parameter like fee of car and fuel efficiency within the primary introduced about thru the use of weights of the automobile inside the car industries as consistent with the safety desired that is very critical to layout moderate weight hassle. The air conditioners applied in cars are set up on a bracket in the bonnet. This task intends to investigate the bracket and optimize the burden with the aid of retaining the same fabric of the bracket. Weight good deal will now not absolutely restrict the raw fabric price, but additionally increase the overall performance, even despite the truth that very minute. The discover approximately of the topology optimization is finished as in line with the requirement of the bracket layout. This contemporary paper highlights the factors for the failure of the mounting bracket and the impact of the optimization through manner of various assessment. In this task, we've got were given designed an ac mounting bracket. Through the use of modelling software software the modelling of the bracket is finished and analyzed the usage of ansys. The glass fiber bracket is designed using format of experiments and analyzed in ansys.[5]

S. Kirthana et al [2018] The usage of engine mounts is the best solution for dampering the effect of vibrations and transmitting forces between the engine and the automotive body structure. This paper deals with the topology optimization of engine mounting bracket of 'Chevrolet beat' using the tools CATIA V5R20 for modeling and Hyper works for finite element analysis. The main objective of the work is to minimize the weight of the engine mounting bracket by considering the design and material layout. For different material layout and different designs the stresses and weights are computed and compared to arrive ate the best model under prescribed conditions.[6]

P. H. Bha et al [2018] The Compressor mounting bracket in automobiles is used to safely support the AC compressor

of a car in every condition. Since it is very difficult to modify the mounting positions and support types after the compressor and the engine are built, the mounting brackets must be verified in the design stage. The energy absorption characteristics of the compressor mount are mainly affected by two variables: material and design. Hence, the design of the compressor mount becomes a critical aspect in terms of vehicle crashworthiness. The present work deals with FEA analysis of compressor mounting brackets. It includes the modeling of the compressor mounting bracket in CATIA V5. The structural analysis of the compressor mounting bracket is carried out using ANSYS. The compressor mounting bracket is then optimized to reduce weight by 64% and also to reduce material use. The optimized model is analyzed to withstand compressor load and design safety. Testing is performed to validate the numerical and experimental result.[7]

Mr. Sagar N. Narute et al [2017] Parameter like cost of vehicle and fuel efficiency mostly influenced by weights of the vehicle in the automotive industries as per the safety standard this is very important to design light weight component. The air conditioners used in cars are mounted on a bracket in the bonnet. This project intends to analyze the bracket and optimize the weight by keeping the same material of the bracket. Weight reduction will not only reduce the raw material cost, but also increase the efficiency, though very minute. The study of the topology optimization is done as per the requirement of the bracket design. This study also highlights the factors for the failure of the mounting bracket and the effect of the optimization by various analysis. In this project, we have designed an AC mounting bracket. The modelling of the bracket is done in modelling software and analyzed using ANSYS. The glass fiber bracket is designed using design of experiments and analyzed in ANSYS.[8]

III. PROBLEM SPECIFICATION

Weight optimization of the components mounted on the automobile is one of the measure area of study in today's engineering studies. We need to design the compressor support bracket for Bus, using different materials including conventional as well as composite materials. We need to find out optimum solution for this application's support bracket even considering material and manufacturing costs

IV. CONCLUSION

Finite element analysis of engine bracket of car and natural frequency will be determined. Engine bracket will be designed as a framework to support engine. The main concern is for vibration and fatigue of engine bracket which may lead to structural failure if resulting vibration and stress are excessive.

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 ,2014.

4. Monali Deshmukh, Prof. K R Sontakke―Analysis and Optimization of Engine Mounting Bracket‖, International Journal of Scientific Engineering and Research, ISSN (Online): 2347-3878,2014.
5. Mrs. Monali Gund, Mr. Gitay M.J., Mr. Pawar Y.N., Mr. Kasabe P.V.International "Design and Analysis of AC mounting bracket using Composite Material " Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 Volume: 07 Issue: 05 | May 2020.

ISSN : 2455-9679
Impact Factor : 3.805

6. S. Kirthana , Mohammed Khaja Nizamuddin "Finite Element Analysis and Topology Optimization of Engine Mounting Bracket "S.Kirthana et al./ Materials Today: Proceedings 5 (2018) 19277–19283.

7. P. H. Bhat, S. S. Sarawade, S. H. Gawande "Numerical and Experimental Analysis of Compressor Mounting Bracket in Automobiles" International Review of Mechanical Engineering (I.RE.M.E.), Vol. 12, N. 1 ISSN 1970 - 8734 January 2018.

8. Mr. Sagar N. Narute1, Prof. Dr. Santosh G.Taji, Prof. Prabhuling G. Sarasambi3 "Design and Analysis of AC Mounting Bracket Using Composite Material"IJSART - *Volume 3 Issue 9 –SEPTEMBER 2017.*

9. 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 ,2013.

10. Vikas Doundkar, Dhananjay Ghatage, Manish Madkaikar, Aniruddha Kulkarni, ― Topology Optimization for Engine Mounting Arm with Fatigue‖, HTC2011.

11. Koushik . S―Static and Vibration Analysis of Engine Mounting Bracket of TMX 20-2 using OptiStruct‖, Altair Technology Conference 2013, India.

12. R Singh ―Dynamic design of automotive systems: Engine mounts and structural joints‖, Saadhanaa, Vol. 25, Part 3, June 2000, pp. 319±330.

13. M.V. Aditya Nag ―Material Selection and Computational Analysis on DOHC V16 Engine's Mounting Bracket Using COMSOL Multiphysics Software‖, 2012 COMSSOl conference.

14. Chang Yong Song ―Design Optimization and Development of Vibration Analysis Program for Engine Mount System‖, Technical Division, AhTTi Co., Ltd.,Korea.

15. A.S.Adkine, V.S.Kathavate, G.P.Overikar, S.N.Doijode ―Static Behaviour of Engine Mounting Bracket‖, International Advanced Research Journal in Science, Engineering and Technology, ISSN (Online) 2393-8021, ISSN (Print) 2394-1588, Vol. 2, Issue 4, April 2015.