ISSN : 2455-9679
Impact Factor : 3.340

IJRTSM

INTERNATIONAL JOURNAL OF RECENT TECHNOLOGY SCIENCE & MANAGEMENT

"A LITERATURE REVIEW ON FEA ANALYSIS OF ENGINE SUPPORTING BRACKET"

Rajendra Kumar Prajapati [1]***, Dr. S. L. Ahirwar*** [2]

[1] *M.Tech Scholar, Department of Mechanical Engineering, OIST , College Bhopal, MP, India.*

[2] *Associate Professor & HOD, Department of Mechanical Engineering, OIST , College Bhopal, MP, India.*

ABSTRACT

In a vehicle, the motor lays on supporting section which is associated with the principle outline or the skeleton of the vehicle. Engine supporting section assumes extremely noteworthy job in diminishing clamor, vibration and brutality caused because of motor and therefore has viable job in improving vehicle comfort. This current work represents the examination of motor supporting section by utilizing ANSYS with FEA. This current article represents survey of the work that has been recently done on motor supporting section. Further work incorporates Static and modular examination of motor supporting casing and to research whether the flow regular recurrence of motor mounting section is lower than that of self excitation recurrence of section. The continuous work, in this manner points in diminishing the undesired vibrations produced by the motor and street unpleasantness can get legitimately communicated to the casing through the supporting section.

Key Words: *Static Analysis, Engine supporting bracket, FEA*

I. INTRODUCTION

Engine is one of the most important components of a road vehicle such as cars, auto rickshaw, and heavy vehicles. Engine mounting bracket plays very significant role in reducing noise,vibration and harshness (NVH) characteristics thereof by increasing comfort ride. Engine bracket has been designed as a framework to support engine [1]. Vibration and fatigue of engine bracket can lead to structural failures when resulting frequencies and stresses exceeds permissible value.

In some studies structural characteristics of engine supporting frame along with its dynamic behaviour are also taken in consideration. Fatigue failure in engine due to prolonged exposure to whole-body vibration in the working environment in some cases it may damage the car. According to basic Non-linear vibration theory; if natural frequency of engine supporting frame manages to maintain well below the excitation frequency band then design is safe and can be taken along with the suitability of materials for required application.

Since engine supporting bracket is the connecting link between engine chassis and engine [2],it is always encountered by forces and vibration due to firing force and air or road resistances. An engine (vibration source) and several mounts connected to the vehicle structure are the two parts of engine supportive frame assembly. In the modern era of technology, engine systems have been successfully isolated the driver and passenger from noise and vibration generated by the engine. It was the interest of earlier researchers up to some extent to modify the design of supportive

ISSN : 2455-9679
Impact Factor : 3.340

assembly by changing the cross section of assembly from square to circular, passive to active and materials change like elastomeric to hydraulic. The main problem of increased fuel consumption is due to increased vehicle weight. The use of light weight optimized materials leads to a more favourable axle load distribution. Meeting the targets of the automotive industry in particular will require new drive concepts combined with the consistent implementation of lightweight construction. Noise, Vibration and Harshness (NVH) are an important vehicles characteristic motivating to achieve overall customer satisfaction [3]. Engine is mostly mounted to the front sub frame and once installed in a vehicle; this assembly has significant effect on driving conditions. A very few researchers have optimized the weight of engine assembly by varying the rib thickness. The topology optimization approach to create an innovative design of an engine mount bracket using Altair Optistruct software. The stiffness of the bracket must be designed in order to withstand the deformation caused due to variable loads.

The overall performance of the dynamic behaviour is strongly influenced by several factors and working standards. Besides road-tyre interaction power train is the major and important source of vibration [4]. For any operation to withstand and perform its function more precisely an engine supporting frame should posses enough toughness and strength, otherwise these vibrations would pass through mounting.

II. LITERATURE REVIEW

Erke Wang et. al [1995] presents some analytical results and some test results for different mechanical problems, tetrahedral and hexahedral shaped elements were used by simulated finite element analysis. The paper showed the comparison for linear static problems, modal analysis and nonlinear analysis.[1]

Umesh S. Ghorpade et. al [2012] have investigated in their research, carried out finite element analysis on engine mounting bracket. In their work materials used were aluminum alloy, magnesium alloy and gray cast iron. For analysing non linear problems, FEA is one of the best methods. It requires geometric inputs such as mesh generation. Various types of models can be applied for generating mesh such as solid works, three dimensional CAD systems. For this engine mounting bracket mesh were generated considering the mid surface and hex dominant quadrilateral and triangular elements. Mesh were generated using ANSYS (version 12) software, while at concluding remarks in the analysis Al alloy and Mg alloy have shown the same nature of natural frequency.

While judging the suitability between Al and Mg alloy, due good damping characteristics, Mg alloy can be preferred over Al and gray CI.[2]

Pramod Walunje et. al [2013] in their research on Optimization of Engine Mounting Bracket using FEA, finite element analysis was offered for analysis as it contains mathematical representation of a physical system comprising a part/assembly (model), material properties, and applicable boundary conditions (collectively referred to as pre-processing), the solution of that mathematical representation (solving), and the study of results of that solution (postprocessing). ANSYS is commercial analysis software strengthening its capabilities covering wide range of problems. Three basic steps involved in ANSYS are pre-processing, solution and post processing phase. The design was analysed for 4 holes. One hole is fixed and remaining three feed with the force of 333 N. This force was feed by Thrust. Self weight of the body was also taken into account. 3-D10-Node Tetrahedral Structural Solid was generated. It was having quadratic displacement behavior and is well suited to modeling irregular meshes. The element was defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions (Figure 8). While discussing their results, it can be anticipated that as the thickness of bracket was reduced the von-misses stresses were well within the permissible limits of the yield strength (280 MPa) for the aluminium. Reduction in the mass approximately upto 0.43Kg was observed when thickness of rib of aluminium material was reduced by 2 mm.[3]

Sameer U. Kolte et.al. [2013] have investigated the Structural Durability Analysis of Powertrain Mounting Bracket in the form of industrial research and done the major work on structural analysis to check the durability of specified part for suitable load and support conditions. The typical view and location of Powertrain mounting bracket is given in

 ISSN : 2455-9679
Impact Factor : 3.340

Figure 9a. The CAD model was imported from CATIA V5R21. Meshes are generated for tetrahedron (Figure 9b). The initial results obtained were used as a guide to do necessary design modifications in bracket to meet the strength criteria. The modified bracket was analyzed again. It was observed that the new bracket passes the strength criteria. Mean while. The purpose of an engine mounting bracket is to safely support the power-train system in all conditions. 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. The engine mounting connects the engine to the body. 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 engine or 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 a factor of safety, and applies this load to each engine mounting bracket.[4]

Yadavalli Basavaraj [2013] et. al reported in their work Design optimization of engine mounting bracket system different designs of the engine mount system have been evaluated for damageability starting from the base model. In the process of optimizing the damageability, study has been performed repetitively on engine mount with design configurations. While in methodology, authors have gone with the finite element model of engine mount is created using CAD & Hyper mesh and simulation of the engine mount is carried out using LS-Dyna. A non linear steady-state dynamic analysis is performed using dynamic properties for six different designs of commercially available rubbers and the results compared to the ones obtained using a hysteretic damping material model suggested by the automotive constructor. The finite element model of engine mount is created using CAD & Hypermesh and simulation of the engine mount is carried out using LS-Dyna. The analysis results which are obtained from two arm, three arm, four arm, filler arm and four arm symmetry engine mounts in which the design of four arm symmetry engine mount curve obtained from LS-Dyna approach follows exactly the experimental test curve and also this design has the highest natural frequency amongst all design iterations. The results indicated that the rubber used in the engine mount had increased the frequency from 1.2Hz (basic design) to 1.8Hz (four arm symmetry). As the design is changing in rubber, the mode of frequency increases and it has found that 1.8Hz is the frequency for the four arm symmetry mount design.[5]

M. Singh, D. Singh et. al. [2013] Analysed the automotive air-conditioning industry aiming at higher levels of quality, cost effectiveness and a short time to market, the need for simulation is at an all time high. In this work, the use of dynamics analysis is proposed in the simulation of the automobile air conditioning condenser assembly for the vibration loads. The condenser assembly has been analyzed using the standard testing conditions. The results revealed that the components of condenser assembly may fail due to resonance in dynamic analysis. Thereafter, the condenser assembly was optimized, resulting in a 2 % reduction in mass.[6]

Jasvir Singh Dhillon et.al. [2014] reported in Design of engine mount bracket for FSAE car using finite element analysis correct geometry and positioning of the mount brackets on the chassis ensures a good ride quality and performance. As an FSAE car intends to be a high performance vehicle, 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 vibration energy and keeping the stresses under a pre-determined level of safety should be achieved by careful designing and analysis of the mount brackets. While in experimental part, authors went for the basic model based on the engine positioning on the chassis. The entire modeling is done using CREO Parametric. Since the geometry suggested a long bracket, material selection became an important consideration due to its weight. To minimize the weight it was decided to make up the mount bracket of two components bolted to each other. One part would be welded on the chassis and the other would be bolted to the engine. HYPERMESH software was used for meshing and Analysis. Tetra-mesh is done because the dimensions of the bracket are comparable to each other and also in-order to obtain accurate results. The two components are bolted to each other and this is simulated using a bar element and are connected to the individual

ISSN : 2455-9679
Impact Factor : 3.340

bodies using rbe2 rigid elements. The two bolting positions on the component no. 2 have rbe2 elements for load application. The bolt size was selected to be M10 and this was to be analyzed for safety too. Mild steel and Aluminum with their suitable material properties were taken for experimental purpose. At the end authors have put their final concluding remarks; The design has been successfully optimized and modified from its preliminary stage. The addition of the rib helped in reducing the maximum deflection by 30.5% in the worst loading case. The maximum von-mises stress increased from 45.02MPa to 63.99MPa, but helped us achieved a more than satisfactory factor of safety of 3.3. Also, even after the addition of material through the rib, the bracket was mass optimized using HYPERMESH 11.0 and OPTISTRUCT Solver. The weight of the final design was 356 grams. compared to the previous 403 grams. The bracket successfully damps the engine vibrations according to physical testing data. Also, the performance of aluminium at high ambient temperatures of about 110 degrees Celsius was satisfactory, since there is a drop of only 5.5% in the yield strength which brings the factor of safety of 3.15. [7]

Monali Deshmukh et. al. [2015] have carried out their research in Analysis and Optimization of Engine Mounting Bracket reviewing that the brackets which supports the engine have to withstand against high static and dynamic deformations. Characteristics of bracket were determined by static structural and model analysis using ANSYS 15 package. Major influencing parameter taken was the acceleration. The existing model was changed for its rib geometry. Results were compared for both the designs. The existing model of an engine mounting bracket was a sheet metal part (Figure 4). Thickness of the part was kept uniform and the whole part was created by casting. So for analysis surface model was imported instead of solid model. Weight of the bracket and engine assembly was taken upto 170 kg (approximately). Input torque with 100 N/mm (x and z directions) and 200 N/mm (y-direction) stiffness was 1×105 N-mm, speed of rotation was taken into account for idling condition 5000 rpm and maximum 9000 rpm While concentrating towards harmonic responses, these are the cyclic responses produced by cycling loading. These are helpful in predicting dynamic behavior of the system. Harmonic response analysis is the technique used to determine the steady-state response of a linear structure to loads that vary harmonically with time. While in concluding remarks, area of rib was reduced and so is the overall weight of bracket. The modified design was observed 12.5% lighter than previous one. Chances of noisy operations through bracket were reduced significantly through harmonic response analysis.[8]

P. Laxmi Kala et. al. [2015] have studied Modelling and Analysis of V6 Engine Mount Bracket and put their views modelling of engine mounting bracket was done in CADAt initial stages theoretical study of engine mounting bracket was done. It was cross checked for vibration analysis too. While in the analysis CAD model of bracket was generated first, followed by mesh generation and application of boundary conditions. Finally, solution to the non linear system was given in the form of frequency analysis. From the model analysis it was found that natural frequency of aluminum material 1181 Hz. Use of CAE tool was turning out to be very significant lending towards easy visualization and time and cost saving from manufacturing point of view.[9]

Harshal Bankar et.al. [2015] In this study the researchers have said that simulation plays very important role in the Automotive industries for the higher levels of quality, better cost effectiveness and quick market response. In this paper, the use of dynamics analysis technique is used for the simulation of the compressor mounting bracket for various vibration loads. The standard testing conditions were used for the testing of the compressor mounting bracket. The results showed that resonance in the dynamic analysis is the major cause for the failure of the compressor mounting bracket, under static analysis, under the same magnitude of load resonance cannot be predicted. Thus, dynamic analysis gives best results for design validation of the compressor mounting bracket. William Nadir and Kim Yong presented a paper "Structural Shape Optimization Considering Both Performance and Manufacturing Cost" which says a structural shape optimization method that considers not only structural performance but also manufacturing cost. Most structural optimizations only take into account structural performance metrics such as stress, mass, deformation, or natural frequency. However, it is often observed that structural performance improves at the expense of manufacturing cost. This work explores the tradeoff between mass and manufacturing cost with the application of the abrasive water jet

ISSN : 2455-9679
Impact Factor : 3.340

(AWJ) manufacturing process. Structural performance, defined as maximum von Mises stress, is a constraint in this work. Work-in-progress results are presented for two structural design examples to demonstrate this tradeoff between mass and manufacturing cost while investigating shape optimization using non-uniform rational B-spines (NURBS). Additional work is still needed to complete this research project.[10]

Mr. Sagar N. 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.[11]

R. N. Yerrawar et. al. [2019] The heat exchangers used in cars are mounted on a bracket in the bonnet. Thepurpose of mountingbrackets is to safely support the vehicle air conditioning systemin all conditions. Since it is very difficult to changethe supporting locations and types of support afterthe air conditioning system is built, the mounting brackets must be verified in the design stage. This project intends to analyze the mounting brackets and developing the new designs of brackets. This paper includes study of design and analysis of heat exchanger (condenser) mounting brackets. In this paper, we will design the condenser mounting brackets to prevent the failure of condenser under static and dynamic loading conditions. The mounting brackets of Paccar Condenser are taken into study. After analyzing it, modified designs and different concepts of mounting brackets are suggested. The modelling of the bracket will be done in modelling software CATIA and will be analyzed using CATIA GPS.[12]

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 Ashok Leyland 220 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

Various authors find out results which are based on Engine supporting.

- Frame plays an important role in reducing noise, vibration and harshness (NVH characteristics) of the vehicle engine system in order to increase its ride comfort.
- However, there is still a need to improve the performance of engine mounting systems for the following two reasons: One reason is that the requirements of vibration and noise level isolation for passenger cars. The second reason is the modern car designs have a trend for lighter car bodies and more power-intensive engines.
- The proposed analysis procedures in the above survey can be used to study the effects of different materials and features like ribs in the bracket for improvements like increasing the
- natural frequency and lowering the stress levels.
- Mass optimization of the mounting bracket and cost reduction of the material.

ISSN : 2455-9679
Impact Factor : 3.340

- Shape optimization along with the suitability of materials can be anticipated through further research.

REFERENCES

1. Erke Wang, Thomas Nelson and Rainer Rauch, "A COMPARISON OF ALL-HEXAHEDRAL AND ALL TETRAHEDRAL FINITE ELEMENT MESHES FOR ELASTIC ANALYSIS", Proceedings 4th of international conference, 179-181. 1995

2. Umesh S. 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 pp 1-6., 2012

3. Mr. Pramod Walunje and Prof. V.K. Kurkute, "Optimization of Engine Mounting Bracket Using FEA", Indian Journal of Research, Volume 2, pp 72-75.,2013

4. Sameer U. Kolte, David Neihguk, and Abhinav Prasad, "Structural Durability Analysis of Powertrain Mounting Bracket" , Excerpt from the Proceedings of the COMSOL Conference in Bangalore,2013

5. Dr. Yadavalli Basavaraj, Manjunatha.T.H, "Design Optimization of Automotive Engine Mount System" International Journal of Engineering Science Invention, vol.2 Issue 3 pp 48-53, 2013

6. M. Singh, D. Singh, J. S. Saini, "Dynamic Analysis of Condenser Assembly of Automobile Air Conditioning System Using CAE Tools", Springer, J. Inst. Eng. Ser. 94(2):135–145, 2013

7. Jasvir Singh Dhillon, Priyanka Rao, V.P. Sawant, "Design of Engine Mount Bracket for a FSAE Car Using Finite Element Analysis" Int. Journal of Engineering Research and Applications, vol. 4, Issue 9, pp 74-81, 2014

8. Monali Deshmukh and Prof. K R Sontakke, "Analysis and Optimization of Engine Mounting Bracket" (2015), International Journal of Scientific Engineering and Research (IJSER), Volume 3, pp 131-136.

9. P.Lakshmi Kala and V.Ratna Kiran, "Modeling and Analysis of V6 Engine Mount Bracket" International Journal of Innovative Research in Science, Engineering and Technology, Vol. 4, pp 5907-5914, 2015.

10. Harshal Bankar, P. Baskar, "Dynamic Analysis of Air Conditioner Compressor Mounting Bracket",International Engineering Research Journal (IERJ) Special Issue 2 Page 4825-4829, ISSN 2395-1621, 2015.

11. Mr. Sagar N. Narute, 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.*

12. R. N. Yerrawar, M.A.Joshi "Design and Analysis of Heat Exchanger Mounting Brackets Used in Automobiles" IOSR Journal of Engineering (IOSR JEN) , 2278-8719 ,PP 44-51 , 2019.