

A Review on Optimization of Engine Mounting Bracket

Sanket Vinchurkar^{#1}, Prof. P.M. Khanwalkar^{*2}

Pune University

Sinhagad College of Engineering, Vadgaon, Pune

Abstract)— *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. 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.*

Keywords — *Engine mounting bracket, Finite element analysis, Optimization.*

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. PROBLEM 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.

In this paper we are going to do the static analysis for the existing Engine mounting bracket,

with the same dimensions, later by knowing the results of the FEA we can than alter the design of the bracket as much as possible to reduce the stresses. Because in general, from the literature survey we found that the bracket fails at high engine speed.

Also, the weight of the engine is supported by the bracket so static analysis is must to check for stress concentration in the bracket.

III. LITERATURE REVIEW

Umesh S. Ghorpade, D. S. Chavan, Vinaay Patil, Mahendra Gaikwad[1] 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 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.

Lakshmi Kala, V.Ratna Kiran[2] With the above stated work done by the previous authors, these authors have taken non-linear vibrational theory into consideration. Improvement in the vibration control can be achieved by determining the natural frequency of the engine bracket. To achieve isolation, natural frequency must be away from the excitation energy to avoid resonating condition. Also, NVH is an important vehicles characteristic motivating to achieve overall customer satisfaction. The main role of engine mounting system as one of

the principle vehicle vibration isolating systems, besides suspension system, is to reduce the noise and vibration perceived by driver and to improve the ride comfort.

Abdolvahab Agharkakli, Digvijay Pradip Wagh[3] 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.

Dr.Yadavalli Basavaraj, Manjunatha.T.H[4] 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.

A.s.adkine, V.s.kathavate, G.p.overikar, S.n.doijode[5] have modeled and designed an engine mounting bracket and analyzed for carrying static loads. Also, modal analysis was carried out in order to find natural frequency of engine mount. If this natural frequency were to match with excitation frequency, resonance would occur. This work is carried by using ERW-1, ERW-2, aluminum and magnesium alloys for the engine mount bracket. The results are analyzed for stresses and deformations.

Sandeep Maski, Yadavalli Basavaraj[6] The authors have considered an engine mounting bracket made of three different materials i.e. cast iron, wrought iron and mild steel. The main objective is to select the best material from the obtained result under prescribed conditions. , modal analysis and static analysis carried out by which maximum von-misses stress and natural frequency are computed.

Monali Deshmukh, Prof. K R Sontakke[7] 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.

IV. CAD MODEL GENERATION

The Engine bracket is mainly consisting of two parts, mainly inner and outer part. CAD model then is made by the commands in CATIA of Pad, pocket, fillet, and geometrical selections in part design module. Parametric generation of drawings will help to get the dimensions useful in forces calculations in static loading conditions on a component.

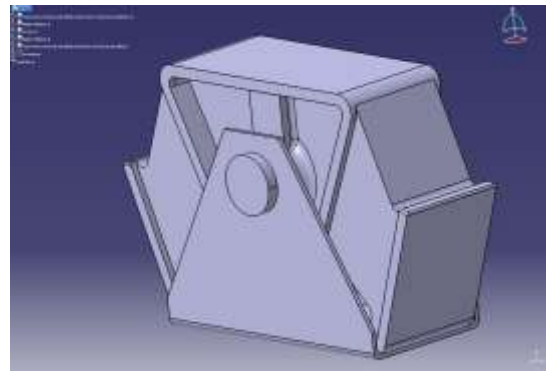


Fig. 1: Engine mounting bracket drawn in CATIAV5R19

V. STATIC STRUCTURAL ANALYSIS

The information required for the analysis of the bracket is as follows:

- Material properties: The values of young's modulus, poissons ratio, density, yield strength for mild steel are taken.
- Meshing: A solid element mesh is required to be generated. The meshing of the mounting bracket is done as tetra-hedral

mesh with 54829 Number of Nodes and 232277 numbers of elements.

- Loads: Specific values of load are implemented for a typical mounting bracket. The load is taken as 1500N which is considerable as the distributed weight of the engine is less than this value.
- Constraints: The nodes around the bracket mounting holes have a rigid element connecting them to the centre of the hole which has of its degree of freedom fixed. The element which is used to fix engine mounting bracket and body of the vehicle is fixed by constraining six degrees of freedom.
- The FE mesh together with the loads and constraints is made in the meshing environment. Solver deck is prepared for carrying out static analysis. The geometry is exported to .Cdb format to run into ANSYS.
- Post-processing is done in ANSYS. Following are the results for post processing.

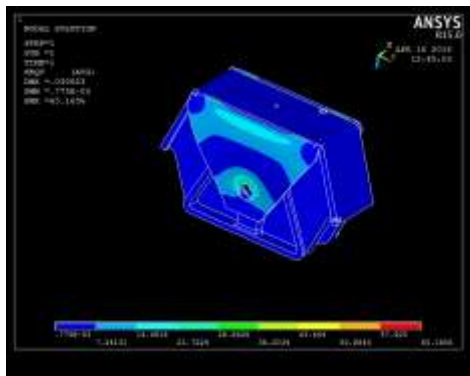


Fig. 2: Von-mises stress plot

Stress value for engine mounting bracket is 65.16 N/mm² which is well below the critical value. Hence, design is safe.

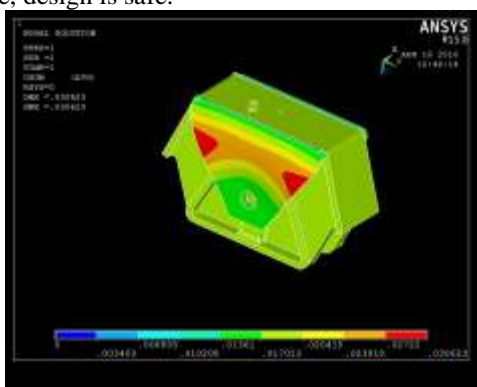


Fig. 3: Deformation plot

From above figure, deformation for engine mounting bracket is 0.03 mm.

VI. CONCLUSION

From results of finite element analysis it is observed that the maximum stress value is coming out to be 65.16 N/mm² which is within the safety limit. There is a great potential to optimize this safety limit which can be done by removing material from low stressed region thus optimizing its weight without affecting its structural behavior. The maximum displacement value is 0.03 mm which is also very less. Mass of bracket as calculated in CATIAV5 is 2.83Kg. So, if the material from low stressed region is removed this mass can be reduced to about 10-15% without affecting its strength.

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", Saadhana, 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 COMSSOI 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] 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.