Research Paper Engineering

Optimization of Engine Mounting Bracket Using FEA

* Mr. Pramod Walunje ** Prof. V.K. Kurkute

* Student of M-tech CAD/CAM, Second year, Bharti Vidyapeeth CollegenOf Engineering, Pune.

** Mechanical Department, Bharti Vidyapeeth CollegenOf Engineering, Pune

ABSTRACT

The Engine is one of the most important components of a road vehicle such as car. High performance sports car has their engine supported by bracket. It plays an important role in improving the comfort & work environment of a car. The improvement of engine bracket system has been the subject of intense interest for many years. It is necessary to design proper engine bracket for a car. As such, engine bracket has been designed as a framework to support engine. Fatigue of engine bracket has been continuously a concern which may lead to structural failure if the resulting stresses are severe and excessive. Prolonged exposure to whole-body stresses in the working environment may lead to fatigue and in some cases it damages the car. Objective of the project is to select the light weight material & reduce the weight of the bracket.

Keywords : Engine mounting bracket, fem, Ansys, optimization

1. Introduction

These The need for light weight structural materials in automotive applications is increasing as the pressure for improvement in emissions and fuel economy increases. The most effective way of increasing automobile mileage while decreasing emissions is to reduce vehicle weight. The incorporation of aluminium and magnesium alloys into automotive structures has steadily increased to meet all these requirements. At present the average use of aluminium per vehicle is of 87 kilograms. The strategy of increasing use of light alloy content in vehicle has proven to be a successful method of achieving fuel efficiency and environmental concerns. The magnitude of the production volumes has traditionally placed severe requirements on the robustness of the processes used in manufacturing. The strong emphasis on the cost has demanded the component manufacturers to improve the performance of their materials and to find the methods to deliver these materials at reduced cost.

2. Literature Review

A Powertrain mounting system includes a pair of engine and transmission mount assemblies. The engine mount assemblies includes an isolator assembly having an elastomeric bushing with an encapsulated mounting member, an integral snubber, and a bracket assembly having a one piece can arrange to house the bushing and having an interlock arranged to engage the bushing. The bushing is press-fit into the can and arranged to isolate the mounting member from the can thereby dampening powertrain vibration and controlling powertrain movement, and the snubber is arranged to limit movement of the bushing relative to the can. [3] The transmission mount assembly includes a support member, a pair of isolators positioned relative to each other, and a pair of mounting brackets positioned on the isolators; where in the isolators are arranged to maintain the mounting brackets in spaced relation to the support member thereby dampening powertrain vibration and controlling powertrain movement. The engine mount bracket assembly includes a one piece can arrange to house the isolator assembly, and a flange for attaching the engine mount bracket assembly to a vehicle frame component, the one piece can also include an interlock arranged to engage the elastomeric bushing. The engine mount isolator assembly is press-fit into the can and the interlock is arranged to retain the engine mount isolator assembly relative to the can, where in the integral snubber is arranged to limit movement of the mounting member and bushing to a predetermined amount relative to the can, and wherein the bushing is arranged to isolate the mounting member from the can thereby dampening powertrain vibration and controlling powertrain movement relative to the vehicle frame component. [6]

3. Finite Element Analysis

Finite Element Analysis is a 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 (post-processing). Simple shapes and simple problems can be, and often are, done by hand. Most real world parts and assemblies are far too complex to do accurately, let alone quickly, without use of a computer and appropriate analysis software. The numerical technique has advantages of experimental as well as analytical method. This analysis requires less resources compared to that of experimental methods.

4. Ansys Package

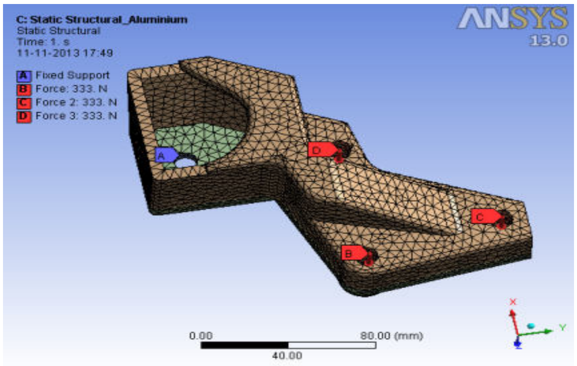
ANSYS is commercial finite-element analysis software with the capability to analyze a wide range of different problems. ANSYS runs under a variety of environments, like UNIX and Windows. ANSYS solves governing differential equations by breaking the problem into small elements. The governing equations of elasticity, fluid flow, heat transfer, and electromagnetism can all be solved by the finite-element method in ANSYS. The ANSYS has multiple windows incorporating Graphics user interface, pull down menus, dialog box and tool bars. There are three basic steps involved in ANSYS finite element analysis.

1. Preprocessing phase
2. Solution phase.
3. Post processing phase.

In the preprocessing module, ANSYS typically uses a CAD representation of the physical model and breaks it down into small pieces called finite "elements". This process is called "meshing." The type of elements can be selected as per the requirement. The area is then discretised i.e. meshed in elements by using meshing or mapped meshing option provided by the package. Once the mesh is generated, the boundary conditions such as external forces, pressures, and displacement constraints are applied. This makes the model ready for the finite element analysis.be followed by two 12-point blank lines.

5. Boundary Conditions Of Bracket

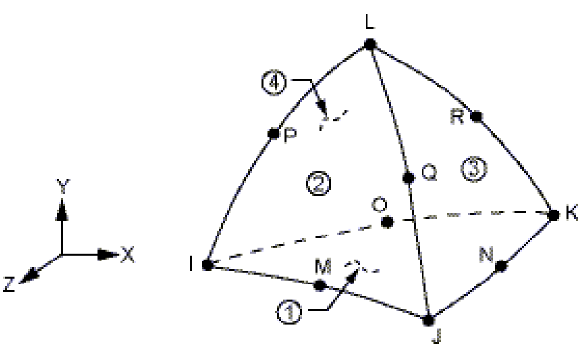
The existing design has 4 holes. One hole is fixed and remaining three has force of 333 N as shown in fig This force is produced by Thrust. There is also self weight (g).



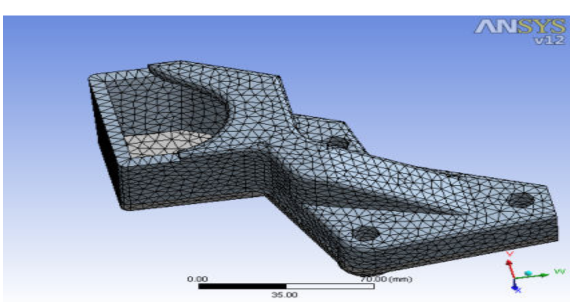
6. Meshing

SOLID 187 mesh element:

3-D 10-Node Tetrahedral Structural Solid. It has a quadratic displacement behavior and is well suited to modeling irregular meshes. The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyper elasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyper elastic materials.



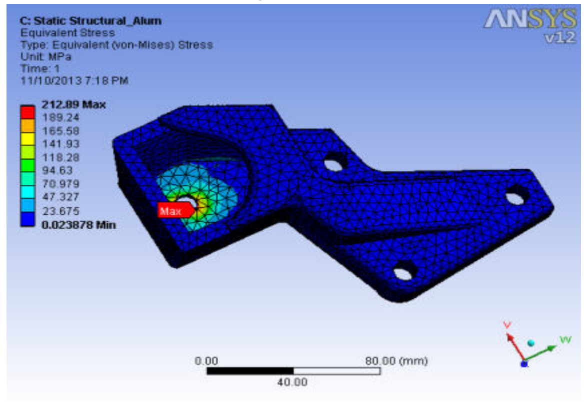
Solid 187 with 10 nodes Tetrahedral Structural Solidt



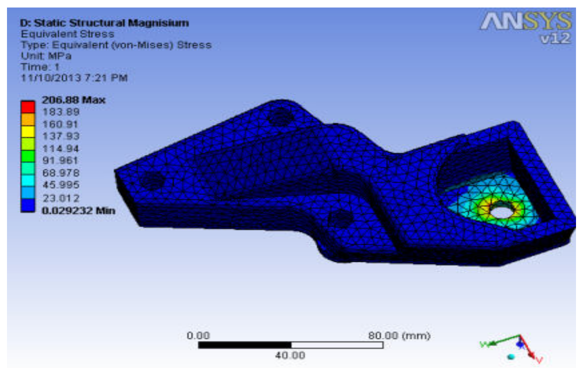
Meshed Model Of Bracket

6. Structural Static Analysis Of Bracket Using Materials Cast Itron, Aluminium & Magnesium.

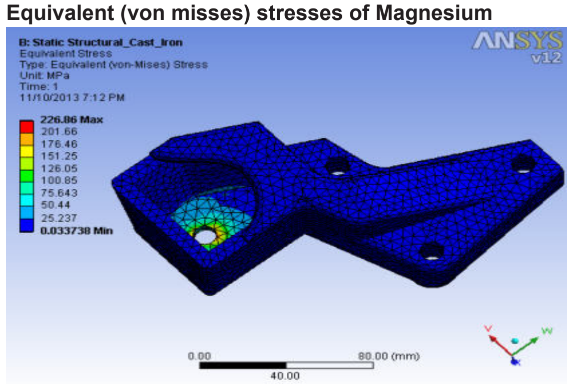
Von-Misses stresses can be derived from the static analysis of structure. As shown in figures below for all three materials.



Equivalent (von misses) stresses of Aluminium

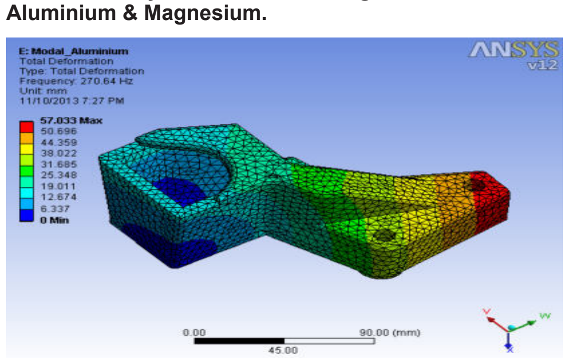


Equivalent (von misses) stresses of Magnesium

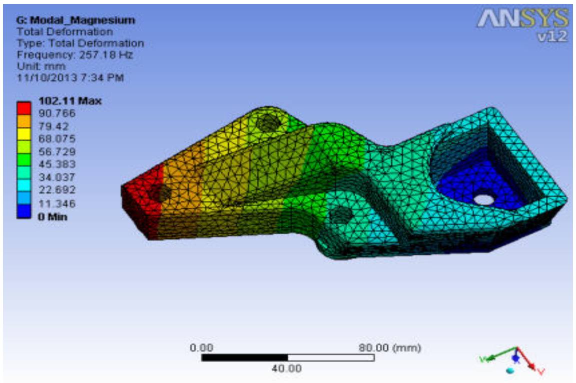


Equivalent (von misses) stresses of Cast Iron

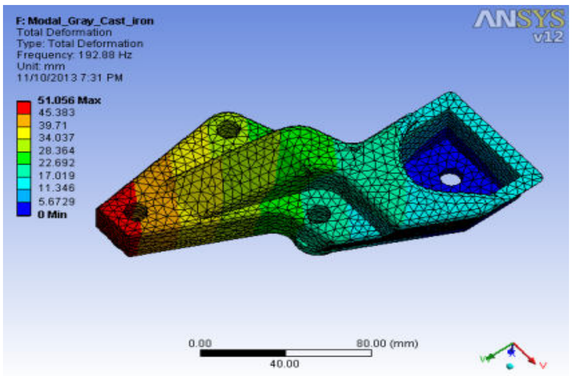
7. Modal Analysis of Bracket Using Materials Cast Iron, Aluminium & Magnesium.



Natural Frequency with 1st mode of Aluminium



Natural Frequency with 1st mode of Magnesium



Natural Frequency with 1st mode of Cast Iron

7. Experimental Analysis for Natural Frequency Analysis.
An FFT spectrum analyzer works in an entirely different way. The input signal is digitized at a high sampling rate, similar to a digitizing oscilloscope. Nyquist's theorem says that as long as the sampling rate is greater than twice the highest frequency component of the signal, the sampled data will accurately represent the input signal. In the SR7xx (SR760, SR770, SR780 or SR785), sampling occurs at 256 kHz. To make sure that Nyquist's theorem is satisfied; the input signal passes through an analog filter which attenuates all frequency components above 156 kHz by 90 dB. This is the anti-aliasing filter. The resulting digital time record is then mathematically transformed into a frequency spectrum using an algorithm known as the Fast Fourier Transform, or FFT.

8. Experimental Setup
The experimental set up for engine bracket is as shown in figure.

Bracket fixed to frame & connected to exciter

FFT connected to impact hammer, laptop & Accelerometer.

8. Results & Discussion
The need for low weight materials in the vehicle is driving Automobile manufacturers to look to new material for use in their products. Some studies suggest that 4.5 to 6% fuel efficiency can be gained for every 10% reduction in vehicle weight. The typical magnesium alloy cast part, is lighter compare to aluminium or Gray Cast Iron part. Comparison between Gray Cast Iron, Aluminium and Magnesium can be done for the material of the engine mounting bracket as follows

8.1. Stress Analysis Using Ansys

	Gray Cast Iron	Aluminium alloy	Magnesium alloy
Von-Misses stress (Max)	226 MPa	212 MPa	206 MPa
Mass (Kg)	1.3	0.55	0.36

8.2. Modal Analysis Using Ansys

	Gray Cast Iron (Hz)	Aluminium alloy (Hz)	Magnesium alloy (Hz)
Frequency	192	270	257

8.3 Experimental Analysis

NATURAL FREQUENCY	Gray Cast Iron	Aluminium Alloy	Magnesium Alloy
Ansys Software	192.77	270	257
Experimental Vibration Test (On FFT Analyser)	202.2	254.8	267.2
% Deviation	4.9	5.9	3.8

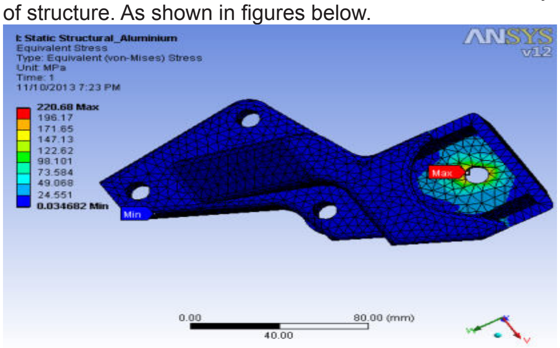
It can be seen from Stress analysis results that Von- misses stresses for Gray cast Iron & Aluminium are well within the yield stress of the material but von- misses stresses for Magnesium is greater than the yield stress(190 Mpa) so we can not select Magnesium for the bracket.

Also it can be seen from Modal Analysis results the values of frequencies are nearly same for Gray Cast Iron, Aluminium alloy bracket in Ansys software & Experimental Test results from FFT Analyser. The first excitation frequency value for Aluminium and Magnesium alloy is higher than that of the excitation frequency range of engine (230 Hz). The value of frequency for Gray Cast Iron is very less. So we can't choose Gray Cast Iron for bracket.

So, we can conclude from above results than Aluminium is the best suitable material for the bracket. So further we do the weight optimization for the aluminium Bracket by reducing its thickness by 2mm.

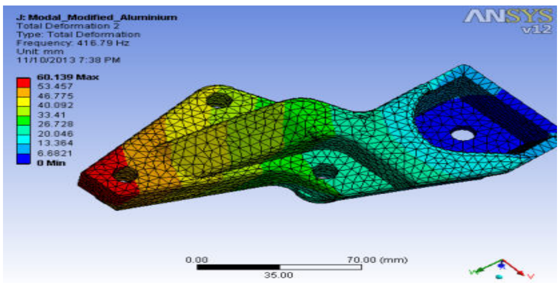
9. Static analysis of modified Bracket with Aluminium material.

Von-Misses stresses can be derived from the static analysis of structure. As shown in figures below.



Equivalent Von-misses stress for Modified bracket using Aluminium Material.

10. Modal Analysis for modified bracket using Aluminium.



Natural Frequency with first mode of modified bracket using Aluminium

11. Comparison Of Result between Original & Modified bracket with Aluminium Alloy.

	Aluminium alloy	Aluminium alloy (Modified Bracket)
Von-Misses stress (Max)	212 MPa	220 MPa
Mass (Kg)	0.55	0.43

12. Conclusion

It can be seen from above result that after redusing the thickness of the Bracket the von-misses stresses coming on the bracket are well within the yield strength (280 Mpa) of the Aluminium.

So, after selecting the light weight material Aluminium & reducing its thickness by 2 mm we have achieved reduction in the mass up to 0.43 Kg.

REFERENCES

[1] Zhang Junhong, Han Jun “CAE process to simulate and optimise engine noise and vibration” Mechanical Systems and Signal Processing 20 (2006) 1400–1409. | [2] Gabriel-Petru ANTON, Mihai PAVAL, Fabien SOREL, “APPLICATION ON AN UPDATED FINITE ELEMENT MODEL OF AN ENGINE IN THE AUTOMOTIVE INDUSTRY” SISOM 2011 and Session of the Commission of Acoustics, Bucharest 25-26 May. | [3] Senthilnathan Subbiah, O.P. Singh, “Effect of muffler mounting bracket designs on durability”, Engineering Failure Analysis 18 (2011) 1094–1107. | [4] Youngwoo Choia, Dohyun Jungb, Kyoungchun Hamc, “A study on the accelerated vibration endurance tests for battery fixing bracket in electrically driven vehicles”, Procedia Engineering 10 (2011) 851–856. | [5] S.K. Loh a, W.M. Chin a, Waleed F. Faris, “Fatigue analysis of Package Terminal Air Conditioner motor bracket under dynamic loading”, Materials and Design 30 (2009) 3206–3216. | [6] S. Irving *, F.Ferguson-Smith, X.Z. Hu, Y. Liu, “Comparative fatigue assessment of soft toe and nested bracket welded aluminium structures”, Engineering Failure Analysis 12 (2005) 679–690. |