# Optimization & Modal Analysis of Engine Mounting Bracket for Different Materials by Using Finite Element Analysis

**Tushar P. Kamble [1], Rajratna A. Bhalerao[2]**

[1] *M.E. Mechanical Design, Student, JSPM's JSCOE Hadapsar, Pune, Maharashtra, India*

[2] *Assistant Professor, Department of Mechanical Engineering, JSPM's JSCOE Hadapsar, Pune, Maharashtra, India*

---***---

**Abstract -** *The Engine in the vehicle is one of the most important components of on road vehicle such as car. High performance sports car has their engine component supported by the mounting bracket to its chassis frame. It plays a very much important role in improving the comfort & work environment of a car as well as the engine component. The improvement of the engine bracket system has been the subject of intense interest for many years. It is required to design the proper engine mounting bracket for a road vehicle. As such, the engine bracket has been designed as a frame work to support engine. Fatigue of engine mounting bracket has been continuously a concern which may lead to structural failure if the resulting stresses are high and excessive. Prolonged exposure to whole-body stresses in the working environment may lead to failure and in some cases it may damage the car. Therefore, the frequency of the bracket should be more than the engine frequency to avoid the resonance. The performance of selected materials with the help of modal analysis by using Finite element analysis*

***Key Words***: **Engine mounting bracket1, Finite element analysis2, Optimization3, Modal analysis4, Materials5 etc.**

## 1.INTRODUCTION

Engine Mounting brackets provide an important role to support the engine with the chassis body and at the same period they should provide the easy flexibility for repair, remove and other operations. The necessity for lower weight structural materials in automobile applications is increasing as the market competition for improvement in emissions norms and fuel economy. The easiest way to increase automobile mileage while decreasing emissions is to reduce the overall vehicle weight. The incorporation of aluminum and magnesium alloys into the automotive industry has increased to meet the all above requirements. The strategy of increasing use of the light alloys in vehicle has proven to be a successful method of achieving the best fuel efficiency and environmental concerns. The strong significance on the cost has demanded the component manufacturers to improve the performance of their product and to find the suitable methods to sell these materials at lower cost. These demands have created exciting challenges for designers, engineers and material researchers.

The engine mounting assembly includes a supporting plate to be attached to a vehicle frame component. A pair of elastomeric or hydraulic engine isolators placed relative to each other on the supporting plate, and a pair of engine mounting brackets arranged to be positioned on the isolators and attached to an engine assembly, where in the engine isolators e.g. Rubber are arranged to maintain the engine mounting bracket in relation to the support plate thereby dampening engine vibration and controlling engine vibration relative to the vehicle.

### 1.1 Problem Definition:

The current design of Engine mounting bracket is old & operational installation is providing difficult. Material of Bracket is costly & heavy. Hence there is need for light weight material selection required for bracket for on road cars, & light vehicles. Optimized design parameters found by analyzing current bracket in various load conditions of various types of Engines will consequently improve performance of new model with the help of Modal analysis.

## 2. LITERATURE SURVEY

Umesh s. ghorpade, et.al [2012] this paper deals on finite element analysis of engine bracket of car and natural frequency will be determined. Engine bracket has been 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.

P. Lakshmi Kala, V. Ratna Kiran [2015] this paper explains the process of optimization of natural frequency of engine bracket by finite element analysis by use of different lightweight materials. The strategy of increasing lightweight material in vehicle has proven to be successful method of achieving fuel economy and environmental concepts. Evaluation of engine mount bracket assembly was performed using FEA and modal analysis technique from the result it was found that bracket manufactured with Mg alloy gives optimized frequency.

Mr. Pramod Walunje, Prof. V.K. Kurkute [2013] in this Author says in an automotive vehicle the engine rest on bracket which is connected to the main frame or chassis of the body. The engine mount plays an important role in vehicle. Correct geometry and positioning of the mount bracket gives a good ride quality and performance. This paper discusses the modeling, finite element analysis, modal analysis and mass optimization of engine bracket for FSAE car. Since the FSAE car are high performance vehicle brackets tends to undergo continuous vibration so fatigue strength and durability calculations also have been done to ensure engine safety.

Jasvir Singh Dhillon, et.al [2014] this paper says that Engine mounts have an important function of containing firmly the power-train components of a vehicle. 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. Keeping this in mind the current paper discusses the modeling, Finite Element Analysis, Modal analysis and mass optimization of engine mount brackets for a FSAE car. As the brackets tend to undergo continuous vibrations and varying stresses, the fatigue strength and durability calculations also have been done to ensure engine safety.

## 3.LOCATION OF BRACKET

**Fig. 1:** Mounted Engine bracket

Above Figure 1 show the location of engine mounting bracket which is fitted with bolt to the vehicle engine holes. The vehicle's structure at the mounting location is important in regard to noise transmission, durability and crash. The above bracket attached to the transmission by use of two horizontal bolts. A limited vertical space is available for both the rubber isolators and bracket. With limited rubber volume it is not possible to allow the rubber to be as soft as it is required for maximum isolation of vibration. Therefore, the mounting bracket must be designed to be as stiff as possible. Through the use of Computer Aided Engineering, an optimized mounting bracket design for stiffness, strength and mass is obtained while still supporting a shortened cycle. Engine mount brackets must support the engine (along with stress, temperature, and corrosion as the necessary product, requirements), and are not to decrease the mount performance as well as meet first mode resonance and stiffness requirements.

## 4. FINITE ELEMENT MODELLING

### 4.1. SOLID187 Element Description

SOLID187 element is a higher order 3-D, 10-node element. SOLID187 has a quadratic displacement behavior and it is well suited to modeling irregular meshes & shapes (such as those produced from various CAD/CAM modules). The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions as shown in figure 2.

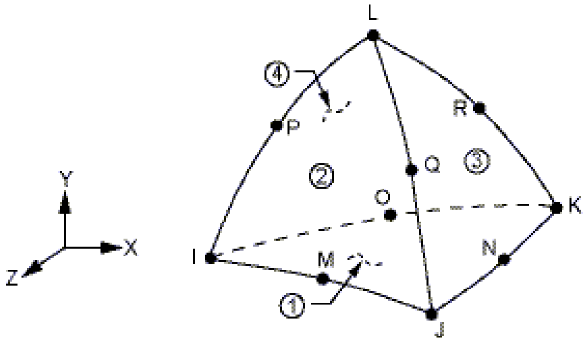


**Fig. 2:** 3-D 10-Node Tetrahedral Structural Solid

### 4.2. Boundary Conditions of the Engine Mounting Bracket:

The bracket has the three mounting holes. One is fixed to the vehicle chassis & on the other three holes' engine has been mounted having load of 60 N as shown in figure 3 below.

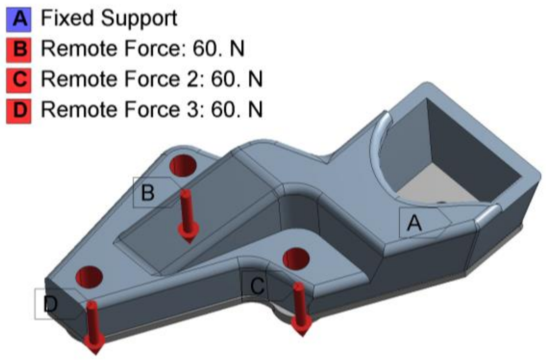


**Fig. 3:** Boundary conditions of Bracket

### 4.3. Meshing of Tetrahedral Structural Solid

**Fig. 4:** Meshed Model of bracket at 300000 nodes

### 4.4 Total Deformation of Bracket for various materials in loading condition

Total Deformation has been derived from the static analysis of structure using Ansys as shown in below figures

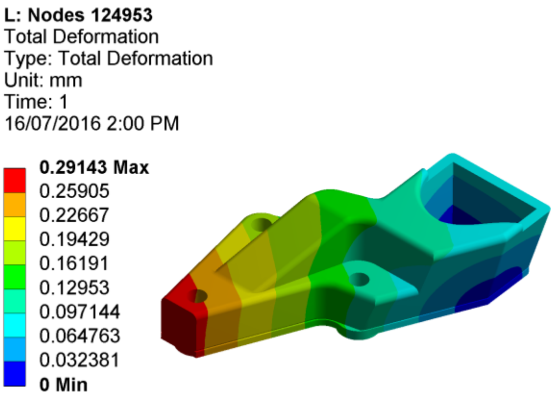


**Fig. 5:** Total deformation for Gray Cast Iron is 0.2914 mm (max) found at the loaded engine mounting hole

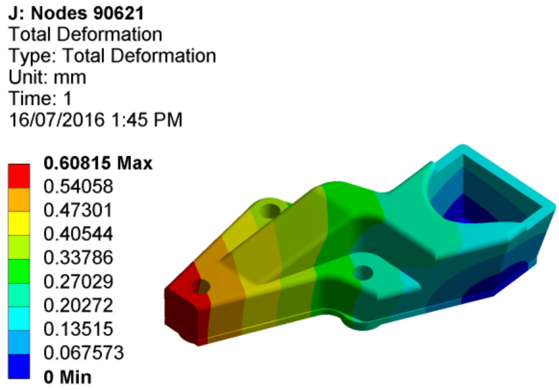


**Fig. 6:** Total deformation for Aluminum alloy is 0.6081 mm (max) found at the loaded engine mounting hole

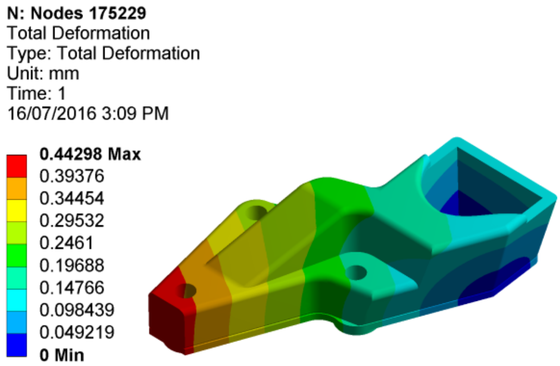


**Fig. 7:** Total deformation for Magnesium alloy is 10.91 mm (max) found at the loaded engine mounting hole

### 4.5 Structural Static Analysis of Bracket for various materials

Equivalent (von-Mises) stresses has been derived from the static analysis of structure using Ansys as shown in below figures

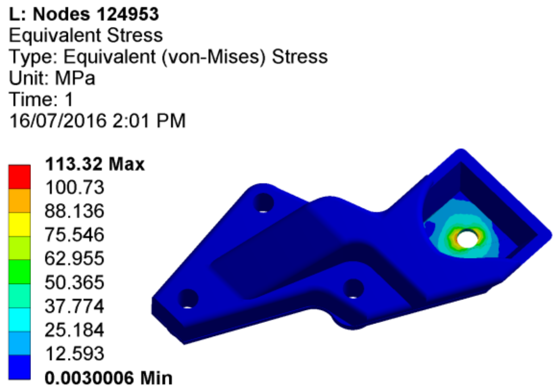


**Fig. 8:** Equivalent (Von-Mises) stresses for Gray Cast Iron is 113.32 Mpa max found at the fixed location of bracket

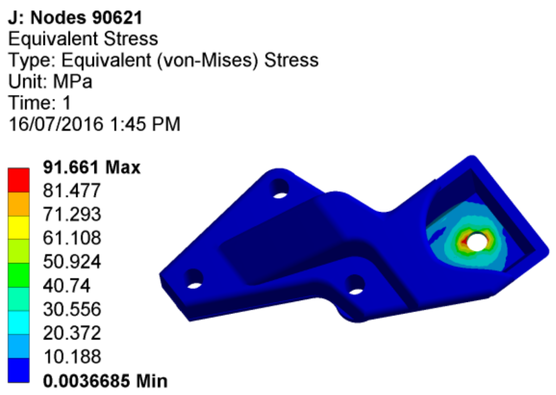


**Fig. 9:** Equivalent (Von-Mises) stresses for Aluminum Alloy is 91.66 Mpa max found at the fixed location of bracket

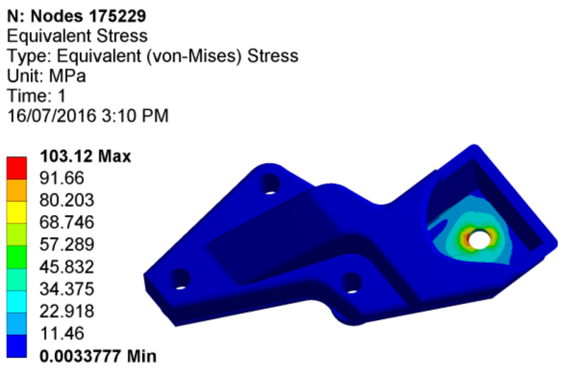


**Fig. 10:** Equivalent (Von-Mises) stresses for Magnesium Alloy is 103.12 Mpa max found at the fixed location of bracket

### 4.6 Modal Analysis of Bracket

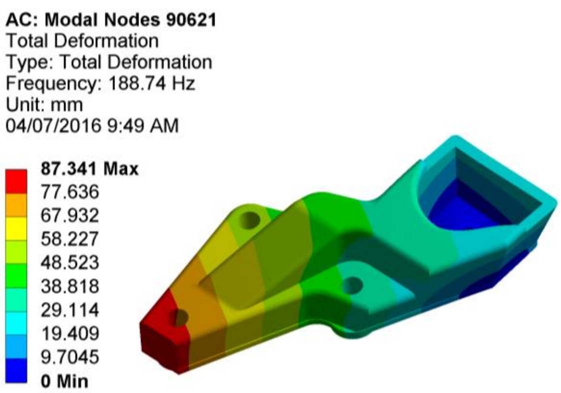


**Fig. 11:** Natural Frequency with 1st Node of Gray Cast Iron is 188.74 Hz

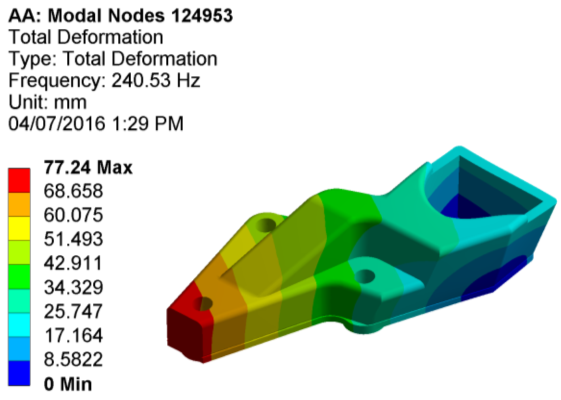


**Fig. 12:** Natural Frequency with 1st Node of Aluminum Alloy 240.53 Hz

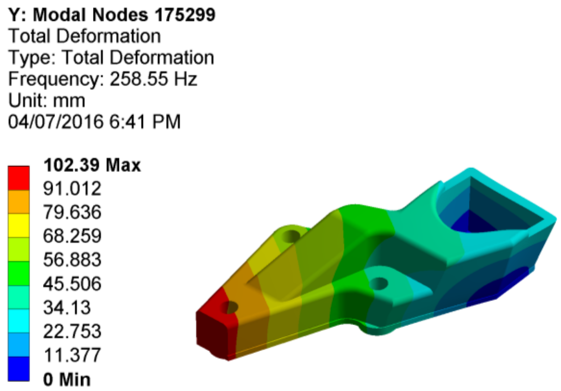


**Fig. 13:** Natural Frequency with 1st Node of Magnesium 258.55 Hz

## 5. EXPERIMENTAL VALIDATION:

The Bracket was tested on shaker table. The vibration signal is sampled by Eddy current sensor and DAQ (data acquisition system). Two excitation methods are used to measure natural frequency of the system. Traditional impulse method, that gives the system single impulse and periodic impulse method realized by electro-magnetically excited impulse.

### 5.1 Data Acquisition System (DAQ)

Data acquisition (DAQ) is the process of measuring an electrical or physical phenomenon such as voltage, current, temperature, pressure, or sound with a computer. A DAQ system consists of sensors, DAQ measurement hardware, and a computer with programmable software. Compared to traditional measurement systems, PC-based DAQ systems exploit the processing power, productivity, display, and connectivity capabilities of industry-standard computers providing a more powerful, flexible, and cost-effective measurement solution.

### 5.2 Shaker Table

**Fig.No.14** TIRA Shaker Table (5-2500Hz)

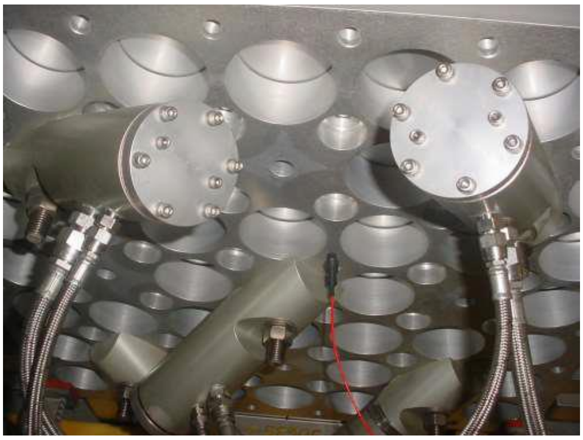

**Fig.No.15** Test Vibration Impactors

**Table 1-** Modal Analysis using Ansys

| Material / Type | Gray Cast Iron | Aluminum Alloy | Magnesium Alloy |
|---|---|---|---|
| From Experimental (Hz) | 190.25 | 239.3 | 260.7 |

## 6.RESULT & DISCUSSION:

All natural frequencies compared with experimental validation & select material which has higher than the engine frequency. But it is need to select the material which has highest frequency among three to avoid resonance

**Table 2-** Modal Analysis using Ansys

| Material / Type | Gray Cast Iron | Aluminum Alloy | Magnesium Alloy |
|---|---|---|---|
| From Ansys(Hz) | 188.74 | 240.53 | 258.55 |
| From Experimental (Hz) | 190.25 | 239.3 | 260.7 |

## 7. CONCLUSION:

It can be seen from Modal Analysis results the values of frequencies. The first excitation frequency value from table for material are Gray Cast Iron, Aluminum alloy & Magnesium Alloy in which frequency for Magnesium alloy is higher than that of the excitation frequency range of engine (230 Hz). The results are compared with experimental results and it was found that the bracket manufactured with Mg alloy gives optimized frequency.

So, we can conclude from above results that the Magnesium alloy bracket having experimental natural frequency 260.7 hz is the best suitable material for the engine mounting bracket assembly. So further we do the more material analysis & shape optimization for the Bracket.

## 8. ACKNOWLEDGEMENT:

This research work is sponsored by Vaftsy CAE Institute.
I would especially like to thank my guide Prof. Bhalerao R. A. and Mr. Vinaay Patil for guiding & providing necessary help as & when required.

## REFERENCES:

[1] Zhang Junhong, Han Jun "CAE process to simulate and optimize engine noise and vibration" Mechanical Systems and Signal Processing 20 (2006) 1400–1409

[2] Fengyan Yi, Meizhi Xie," Objective evaluation of engine mounting isolation" 2012 AASRI Conference on Modeling, Identification and Control, AASRI Procedia 3 (2012) 49 – 53

[3] K.S. Raghavan, T. Howard, J. Buttles, " Weight Reduction of Structural Vibration Isolation Hydro-Mount Bracket through Design Analysis and Use of Advanced High Strength Steels" 2009 SAE International, 09M-0168

[4] Jasvir Singh Dhillon, Priyanka Rao, V.P. Sawant, "Design of Engine Mount Bracket for a FSAE Car Using Finite Element

Analysis" ISSN: 2248-9622, Vol. 4, Issue 9 (Version 6), September 2014, pp.74-81

[5] Umesh S. Ghorpade, D. S. Chavan, Vinaay Patil, Mahendra Gaikwad, "finite element analysis and natural frequency optimization of engine bracket" IJMIE, Vol-2, Iss-3, Page no. 1-6, 2012

[6] Lakshmi Kala, V. Ratna Kiran, "Modeling and Analysis of V6 Engine Mount Bracket" IJIRSET, Vol. 4, Issue 7, Page No. 5907-5914, July 2015

[7] Mr. Pramod Walunje, Prof. V.K. Kurkute, "Optimization of Engine Mounting Bracket Using FEA" IJR, Volume: 2, Issue: 12, Page No. 72-75, Dec 2013

[8] A.S.Adkine, V.S.Kathavate, G.P.Overikar, S.N.Doijode," Static Behavior of Engine Mounting Bracket" IARJSET, Vol. 2, Issue 4,Page no.68-73, April 2015

[9] P.D. Jadhav, Ramakrishna," Finite Element Analysis of Engine Mount Bracket" IJAET, Vol. 1, Issue 4, Page1-10, September 2014

[10] Promit Choudhury, Chinmaya Hemanth Krishna,"Mathematical Modelling and Shape Optimisation of Front Damper Mount of Ashok Leyland 1612 Truck Using 3d Finite Element Method" The International Journal Of Engineering And Science (IJES), Volume 3, Issue 4, Pages 70-76, 2014