Finite Element Analysis of Engine Mount Bracket

P.D. Jadhav 1, Ramakrishna 2
P.G. Student, Department of Mechanical Engineering, CMR IT, Andhra Pradesh, India 1
Associate Professor, Department of Mechanical Engineering, CMR IT, Andhra Pradesh, India 2

Abstract: The engine mounting plays an important role in reducing the noise, vibrations and harshness for improving vehicle ride comfort. The first and the foremost function of an engine mounting bracket is to properly balance (mount) the power pack (engine &transmission) on the vehicle chassis for good motion control as well as good isolation.

Present work deals with FEA analysis of engine mounting bracket. It includes the modeling of the engine mounting brackets by changing the material of component. Materials selected are Aluminum alloy and magnesium alloy. Analysis includes Static and Modal Analysis of engine mounting bracket using Square Cross section. The study shows that this bracket will have a dramatic weight reduction compare to standard aluminum alloy material and withstand high stress.

Keywords- Engine, Mounting, FEA, Modal Analysis.

Introduction

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 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. There are a number of noise and vibration sources that affect the vehicle body. The noise and vibration occur because the power that is delivered through bumpy roads, the engine, and suspension result in the resonance effect in a broad frequency band. The ride and noise characteristics of a vehicle are significantly affected by vibration transferred to the body through the chassis mounting points from the engine and suspension. It is known that body attachment stiffness is an important factor of idle noise and road noise for NVH performance improvement. The Automobile engine-chassis-body system may undergo undesirable vibrations due to disturbances from the road and the engine. The vibrations induced by the road or the engine at idle are typically at the frequencies below 30Hz. In order to control the idle shake and the road-induced vibrations, the engine mount bracket should be stiff and highly damped. On the other hand, for a small amplitude excitation over the higher frequency range (30-250Hz) from the engine, a compliant and lightly damped mount bracket is required for vibration isolation and acoustic comfort. So, the engine mount bracket must satisfy these two essential criteria.

In diesel engines the engine mounting is one of the major problems. Due to the Un-throttled condition, and higher compression ratio of the diesel engine, the speed irregularities particularly at low Speed and Low load conditions and are significantly higher than gasoline engines. Thus this results in vibration excitation. Due to vibration of engine the holes on the engine mounting Bracket get enlarged which results in the failure of bracket. By optimizing the thickness and shape of major mounting points made it possible to design a vehicle with optimized weight and performance at initial designing stages [1]. Studies shows that the brackets saved 38% mass (0.86 kg the expected and resultant benefit is different for each application. Range of savings are 20% to 38% in the authors’ experience to 0.53 kg)[4]. Structural optimization is an effective tool to obtain an optimum design. Comparison in terms of weight and component performance illustrates that structural optimization techniques are effective to produce higher quality products at a lower cost [5]. Dynamic analysis of vehicle components and systems is now an integral part of the engineering design process in the automotive industry [9]. The topology based design is used in the conceptual design phase and then, the shape optimization is subsequently employed using initial optimal topology [11].
The objective is to do a modal analysis of the engine bracket to determine whether the current design has a natural frequency which is lower than the excitation frequency of the engine bracket, in which case the design will be considered safe. Once safety of the design is established, and then optimization will be done material Optimization: The design will be tested for different materials, Aluminum, magnesium and Grey Cast Iron, and suitability of material will be tested.

**METHODOLOGY**

**BASIC STEPS IN THE FEA**

The basic steps involved in any finite element analysis consist of the following

1. **PREPROCESSING PHASE**
2. **SOLUTION PHASE**
3. **POSTPROCESSING PHASE**

Fig.1. FEA process

**STATIC STRUCTURAL ANALYSIS**

A static structural analysis is the analysis displacements, stresses, strains and forces on structure or a component due to load application. The structures response and loads are assumed to vary slowly with respect to time. There are various types of loading that can be applied in this analysis which are externally applied forces and pressures, and temperatures. A static structural analysis can either be linear or non linear.

**MODAL ANALYSIS**

Modal analysis determines the vibration characteristics of a structure or a particular component in the form of natural frequencies and mode shapes. From this analysis we can do more detailed dynamic analysis such as transient dynamic analysis, harmonic analysis or spectrum analysis. The natural frequencies and mode shapes are important in the design of a structure for dynamic loading conditions. In this analysis, only linear behavior is valid. Damping is not considered and applied loads are ignored in modal analysis. A static structural analysis is required first for performing pre stressed modal analysis.

The analysis of many engineering and automotive components is done with the help of finite element techniques. These analysis are very helpful for approving or making some design changes during the post processing stage. The design changes mainly depend on the product life cycle and helps design engineers or the analysts to finalise dimensions and material of the components. This analysis of an engine mounting bracket is done with the help of FEA software.

**FEA PRE PROCESSING FOR THE MOUNTING BRACKET:**

The pre-processing of the engine mounting bracket is down for the purpose of the dividing the problem into nodes and elements, developing equation for an element, applying boundary conditions, initial conditions and for applying loads. The information required for the pre-processing stage of the bracket is as follows,

- Material properties: The values of young’s modulus, poisons ratio, density, yield strength for Aluminium and magnesium are taken from material library of the FEA PACKAGE. At the point only one material is selected at a time, which is sufficient to show that a change of material is made depending upon the analysis results.
  Meshing: A solid element mesh is required to be generated. The meshing of the mounting bracket is done as hex mesh for steel, for Aluminium and for magnesium also.
• Loads: Specific values of load are implemented for a typical mounting bracket. The load is taken as 1000N which is considerable as the distributed weight of the engine is less than this value. Load is applied at the three holes of the engine mounting bracket, which are connected to the engine structure with the help of rigid elements such as nut and bolts.
• 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 and used as a rigid element.
• The FE mesh together with the loads and constraints is made in the meshing environment. The minimum and maximum are set, together with other mesh parameters such as element type and material. The selected object is ready for further analysis.

**POST PROCESSING OF THE ENGINE MOUNTING BRACKET:**

The acceptability of the design of the engine mounting bracket needs to be considered from the results of the analysis. The guidance for the modification of the bracket need to be available if the design is not considered to be acceptable for the engine mounting bracket are as follows.

1. Model acceptance criteria: the maximum von-Mises stress must be less than the material yield strength for the duration of the component. The deflection is considered and the maximum Von-Mises stress must be less than the yield strength for abuse load case.
2. Design modification criteria: the modifications are made if the design is not acceptable, such as increasing fillet, increasing material thickness and altering material specifications. The output in the form of the stress value is generated, when the analysis has been completed. This output data in the form of figures is then checked for determining the relevant information and compared with the acceptance criteria.
3. Result interpretation: from the results of the analysis the engineer required to understand the failure criteria for the particular component.
4. The Design Failure Criteria: maximum allowable deflection, maximum allowable stress is required to be known and documented. For this purpose the values of the Von-Mises stress is considered. There is also a requirement for recognizing the effect of the analysis on the engine mounting bracket due to load applied.

**ALUMINUM:**

Aluminum has only about one third the density of steel and the most commercial Aluminum alloys posses substantially higher specific strength than steel. A vehicle weight reduction would not only result in higher oil savings, but also gives a significant reduction in emission. For these reasons there is preference to use more Aluminum and replace steel in automotive applications.

But there are several obstacles in implementation of Aluminum in automotive industry. Some of these are,

• The forming limits of Aluminium are significantly lower than the steel. There are several chances of Aluminium to tearing at bends. This limits the shapes that can be fabricated and slows die design, die tryout and applications.
• Spring back problem is more in thicker sheets of Aluminium and it is hard to keep dimensional tolerances.
• Traditional vehicle body manufacturing technology implementing stamped sheet steel component cannot be sufficiently improved to meet future vehicle requirements because of the higher weight of the steel and the high cost and more time required for stamping tool development.

The analysis of the engine mounting bracket is done with the help of FEA PACKAGE software. In this process first the material selection done from the engineering data sheet of the FEA PACKAGE software. Then the meshing of the component is done from the MESH function of the software. The importance of the meshing in the whole process of analysis is very important from the point of result implementation. The element used in the meshing is solid 186 which are 20 node brick, hexagonal in shape and very good for the deformation problems. The
size and shape of the meshing is the factor which specifies the method of solution of FE problem in the any FEA software. By the process of refining, which is nothing but the changing the size of the mesh accurate result can carry out.

**Aluminum Alloy**

Aluminum alloy under consideration has following material properties:

- Young’s modulus – $7.1\times10^{10}$ N/m$^2$
- Poisson’s ratio – 0.33
- Density – 2770 Kg/m$^3$
- Yield strength in tension & compression – $2.8\times10^8$ N/m$^2$

**Stress analysis:**

Stress analysis of the engine mounting bracket is carried out by fixing the bracket at the large size end which is fixed to the vehicle body. Then the load is applied at the other end of the bracket for the purpose of static structural analysis.

Von-Mises stresses can be derived from the static analysis of structure. As shown in figure 3 the von mises stresses plots are taken. As seen from these plots of stress we can see that maximum stress of the value 26.758 MPa is found out at the bracket which is fixing the bracket and the vehicle structure. This is due to the load concentration at the fixed point.

Above figure 4, shows the deformation of the engine mounting bracket, when the load is applied at the side of bracket which is towards the engine connection. As seen from the deformation plot deformation is more at the end of the bracket due to the maximum load application is at this side of the bracket. The deformation of this part is 1.6258 mm, which can be seen from the above result. The deformation plot shows the variation in values from fixed side to the engine side of bracket. It is minimum at the fixed side and highest on the engine side.
MODAL ANALYSIS:

The modal analysis of the engine mounting bracket is carried out for determining natural frequencies and shapes of the natural vibration modes. These frequencies are the outputs given by the analysis of the engine mounting bracket.

From the above plot (figure 6) of the total deformation of the engine mounting bracket we can conclude that the maximum value deformation is taking place at the side of the bracket connected to the engine. This is due to the fact that vibrations are going to act on this part of the bracket. The deformation for the other side which is fixed to the body of the vehicle is almost zero for some part of it. The frequency of vibration for this particular mode of the bracket is 142.78 Hz.

The third mode of vibration of the engine mounting bracket is seen in the plot below (figure 7) for modal analysis. The part towards the engine of the vehicle is not deformed too much, but from this point to the side of the bracket which is fixed to the vehicle structure is varying with deformation. The higher value of deformation is at the upper corner of the bracket. The side towards the vehicle structure is largely deformed as seen from the figure. The frequency of vibration for this particular mode is 213.26 Hz.

From the above plot (figure 8) of the total deformation of the engine mounting bracket we can conclude that the maximum value deformation is taking place at the bottom side of the bracket connected to the engine. The frequency of vibration for this particular mode is 322.78 Hz.
As seen from the above figure 9 the values of deformation increasing minimum to maximum from the end which is connected to the vehicle structure to the end of engine. The maximum value of deformation is seen at the ends of engine. The frequency of vibration for this particular mode is 462.57 Hz. From the above plot (figure 10) of the total deformation of the engine mounting bracket we can conclude that the maximum value deformation is taking place at the bottom side of the bracket connected to the engine. The frequency of vibration for this particular mode of vibration is 630.5 Hz.

ALTERNATIVE MATERIAL “MAGNESIUM ALLOY”

Magnesium is the lightest of all metals used as the basis for constructional alloys. It is this property due to which automobile manufactures has to replace denser materials, not only steels, cast irons and copper base alloys but even Aluminum alloys by magnesium base alloys. The requirement to reduce the weight of car components as a result in part of the introduction of legislation limiting emission has triggered renewed interest in magnesium. A wider use of magnesium base alloys necessitates several parallel programs. These can be classified as alloy development, process development improvement and design considerations. The requirement to reduce the weight of car components as a result in part of the introductions of legislation limiting emission has triggered renewed interest in magnesium.

The advantages of magnesium alloys are listed as follows, lowest density of all metallic constructional materials. It posses high specific strength, good cast ability, which suitable for high pressure die casting good welding properties, higher corrosion resistance. Also compared with polymeric materials it posses better mechanical properties, better electrical and thermal conductivity and it is recyclable. The requirement to reduce the weight of car components as a result in part of the introduction of legislation limiting emission has triggered renewed interest in magnesium. The application of die cast magnesium for automobile components produced in large quantities is the one method for reducing vehicle weight. But these components must be cost competitive with material and processes in current manufacturing and gives high qualities to ensure adequate mechanical properties. One of the reasons for the limited use of magnesium has been some poor properties exacerbated by a lack of development work. It is not possible to use conventional alloying techniques to improve some of the properties, e.g. elastic constants. The solubility of alloying elements in magnesium is limited, restricting the possibility of improving the mechanical properties and chemical behavior. The crystal structure of magnesium is hexagonal which limited its inherent ductility.

Magnesium Alloy

Magnesium alloy under consideration has following material properties

- Young’s modulus – 4.5e+10 N/m2
- Poisson’s ratio – 0.35
- Density – 1800 Kg/m3
- Yield strength in tension and compression – 1.93e+8 N/m2

Stress Analysis

From the above (figure 11) lot of the total deformation of the engine mounting bracket we can conclude that the maximum value deformation is taking place at the bracket connected to the engine. This is the fact that maximum load and vibrations are going to act on this part of the bracket. The highest value deformation takes place on this side which is having value of 2.5696 mm. the deformation for the other side which is fixed to the body of the vehicle, is almost zero for some part of it. The deformation goes on increasing from the fixed end towards the side of the engine.
MODAL ANALYSIS:

It can be seen from the below plot of the deformation of the engine mounting bracket the deformation is maximum at the end part which is connected to the engine of the vehicle. The maximum value of the deformation is less at the end which is connected to the vehicle, which can be said at a fixed one. The deformation increases from the fixed end to the other side after some definite intervals. The frequency of vibration for this particular mode of the bracket is 75.261 Hz.

From the plot (figure 14) of the total deformation of the engine mounting bracket we can conclude that the maximum value deformation is taking place at the side of the bracket connected to the engine. This is due to the fact that vibration is going to act on this part of the bracket. The deformation for the other side which is fixed to the body of the vehicle is almost zero for some part of it. The deformation goes on increasing from the fixed end towards the side of the engine. The frequency of vibration for this particular mode of the bracket is 140.6 Hz.

The third mode of vibration of the engine mounting bracket is seen in the plot below (fig 15) for modal analysis. The part towards the engine of the vehicle is not deformed too much, but from this point to the side of the bracket which is fixed to the vehicle structure is varying with deformation. The higher value of deformation is at the upper corner of the bracket. The side towards the vehicle structure is largely deformed as seen from the figure above. The frequency of vibration for this particular mode is 210.62 Hz.

From the above plot (figure 16) of the total deformation of the engine mounting bracket we can conclude that the maximum value deformation is taking place at the bottom side of the bracket connected to the engine. The frequency of vibration for this particular mode is 318.52
Hz. As seen from the figure 17 the values of deformation increasing minimum to maximum from the end which is connected to the vehicle structure to the end of engine. The maximum value of deformation is seen at the of engine. The frequency of vibration for this particular mode is 455.91 Hz. 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 steel part. Comparison between steel, Aluminium’s and magnesium can be done for the material of the engine mounting bracket as follows,

RESULTS AND DISCUSSION

STRESS ANALYSIS:

<table>
<thead>
<tr>
<th></th>
<th>Aluminum alloy</th>
<th>Magnesium alloy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Von-Mises stress (max)</td>
<td>26.738 Mpa</td>
<td>26.483 Mpa</td>
</tr>
<tr>
<td>Total deformation (max)</td>
<td>1.6258 mm</td>
<td>2.5696 mm</td>
</tr>
</tbody>
</table>

Table 2: Stress Analysis for Engine Mounting Bracket

It can be seen from the above results that the value of the stress on the Aluminium bracket is not varying too much than the stress value of magnesium bracket for the same load. This is because of the fact that the area and force for both brackets are same. The values of the total deformation are varying in some large values. This is due to difference in the density of the materials.

MODAL ANALYSIS

The values of frequencies are nearly same for gray cast iron, Aluminum and magnesium bracket. The first excitation frequency value for magnesium is higher than that of excitation frequency range of engine (1-150 Hz).

<table>
<thead>
<tr>
<th>Frequency</th>
<th>Aluminum alloy in Hz</th>
<th>Magnesium alloy in Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>First Frequency</td>
<td>76.121</td>
<td>75.108</td>
</tr>
<tr>
<td>Second Frequency</td>
<td>142.78</td>
<td>140.6</td>
</tr>
<tr>
<td>Third Frequency</td>
<td>213.26</td>
<td>210.62</td>
</tr>
<tr>
<td>Fourth Frequency</td>
<td>322.78</td>
<td>318.52</td>
</tr>
<tr>
<td>Fifth Frequency</td>
<td>462.57</td>
<td>455.91</td>
</tr>
<tr>
<td>Sixth Frequency</td>
<td>630.5</td>
<td>622.94</td>
</tr>
</tbody>
</table>

Table-3: Frequency comparison for Aluminum and magnesium Alloy’s
From above graphs we can conclude that the convergence of A frequency for magnesium material is good. The first excitation frequency value for magnesium is higher than that of excitation frequency range of engine.

Conclusion

The finite element analysis tool has been used to analyze the engine mounting bracket using CAE Package. This work is a contribution to the development of new material for engine mounting bracket. The results obtained for the static structural and modal analysis have shown that the magnesium is better than aluminum. From the results it can be seen that the magnesium bracket is safe for the required application.

The main advantage of the magnesium engine mounting bracket is its light weight. It will help in decreasing the weight of the power train assembly, which can increase fuel efficiency. Magnesium is recyclable; therefore it is an eco friendly material. The magnesium bracket can be manufactured with less amount of time and it posses longer life compared to an aluminum bracket. The magnesium bracket is less susceptible to corrosion; therefore they are better for the application of bracket. The main problem of using magnesium instead of aluminum is its higher cost; but recent studies have shown that the difference between costs of aluminium and magnesium is decreasing. Also if the use of magnesium in industries increases, its manufacturing cost will be definitely reduced.

Thus it can be concluded that magnesium can be preferred over aluminum as a material for an engine mounting bracket. Research in the direction of implementing magnesium engine mounting bracket instead of aluminum bracket has also proved that magnesium brackets are better in various operating conditions.

References

5. Xiao-Yong Pan and Doni Zonni,Tuopu Group,Guo-Zhong Chai,Zhejiang University of Technology Yan-Qing Zhao and Cui-Cui Jiang,South China University of Technology, Structural Optimization for Engine Mount Bracket, SAE TECHNICAL PAPER SERIES, 2007-01-2419.
6. Guizhi Sun, Minxiang Wei, Jinju Shao and Man Pei,Nanjing University of Aeronautics and Astronautics, Automotive Powertrain Modeling and Simulation Based on AMESim, SAE TECHNICAL PAPER SERIES, 2007-01-3464.

9. R SINGH, Department of Mechanical Engineering, Ohio State University, Columbus, OH 43210, USA e-mail: singh.3@osu.edu, SaAdhana, Dynamic design of automotive systems: Engine mounts and structural joints, Vol. 25, Part 3, June 2000, pp. 319-330. # Printed in India, 01-06-2000.


11. Ali Rıza Yıldız, Necmettin Kaya and Ferruh Öztürk*, University of Uludağ, Mechanical Engineering Department, Bursa 16059, Turkey, Fax: (0224) 442 80 21, E-mail: ferruh@uludag.edu.tr, E-mail: ozturkf@asme.org, *Corresponding author, Orhan Alanku, TOFA-FIAT, Yalova Yolu, R&D Optimal design of vehicle components using topology design and optimisation.