RESEARCH ARTICLE OPEN ACCESS

Design of Engine Mount Bracket for a FSAE Car Using Finite Element Analysis

Jasvir Singh Dhillon[1], Priyanka Rao[2], V.P. Sawant[3]
[1&2]Research Scholar, Department of Mechanical Engineering, Smt. Kashibai Navale college of Engineering, Pune, India.
[3]Assistant Professor, Department of Mechanical Engineering, Smt. Kashibai Navale college of Engineering, Pune, India.

ABSTRACT
Engine mounts have an important function of containing firmly the power-train components of a vehicle. 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 vibrational energy and keeping the stresses under a pre-determined level of safety should be achieved by careful designing and analysis of the mount brackets. 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.
Keywords: FEA; Modal Analysis; Static Analysis; Optimization; Mounting Bracket

I. INTRODUCTION

In an automotive vehicle, the engine rests on brackets which are connected to the main-frame or the skeleton of the car. Hence, 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 might even damage the chassis. When the operating frequency or disturbance approaches the natural frequency of a body, the amplitude of Vibrations gets magnified. This phenomenon is called as resonance. This magnification is most severe in low frequency ranges up to 50Hz. Also, at high operating frequencies noise becomes a serious concern. Hence, damping of these engine vibrations becomes an important function of the mount brackets. The Noise & Vibration Harness analysis (or NVH) is one the most important considerations in automotive designing today. If the brackets have their resonance frequencies close to the operating engine frequencies, then the large amplitude of vibration may cause its fatigue failure or breakage, thus reducing its estimated or desired life. Vibration damping can be either provided by using separate dampers (anti-vibration mounts) or by suitably deciding the material and dimensions of the brackets. Moreover, the brackets also undergo deflection under static and dynamic loads. This deflection should be under permissible limits.

A Formula Student car is required to be highly maneuverable and quick with high rates of acceleration and deceleration. Hence the mounting of the engine should be well constrained and the mount brackets need to be light-weight and designed to safely bear the inertial loads and maximize vibration-transmission.FEA has been done to check the frequency and loading response of the brackets before finalizing the design. Mass optimization has been carried out to save material and reduce the weight. The modified designed has been re-analyzed using FEA before finalization. The design of the brackets has been experimentally validated after the actual implementation and testing of the vehicle. [1]Karl D. Hammond have investigated that the amplitude of vibration at constant frequency changes after prolonged exposure to same vibration at the same frequency. [2] The natural frequency of a material decreases as additional load is applied to the system or increase in mass or thickness of the material.[3]The response of vibrating system can be in the form of displacement, velocity, acceleration or sound. These have to be kept under safe or acceptable limits while designing. [4]The bracket should be designed to keep the resonance frequency(or frequencies) out of operating range.

II. DESIGN STAGES

a) PRE-PROCESSING

i) CAD Modelling

This stage involves making 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.

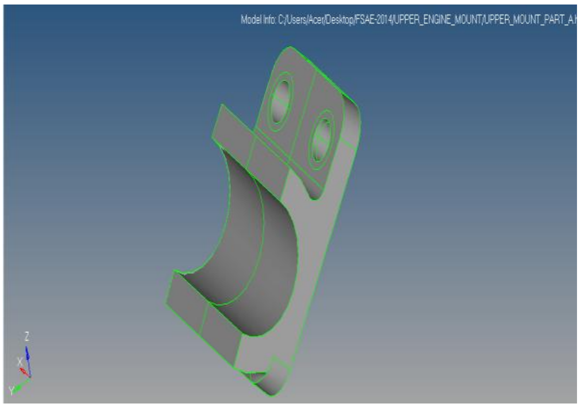

Fig.1. Component no. 1

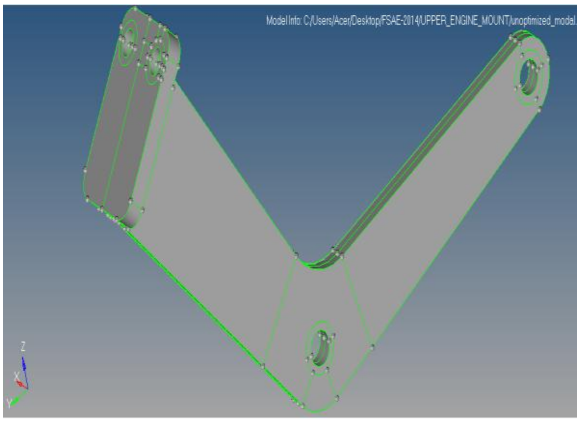

Fig.2. Component no. 2

ii) MESHING

It was decided to 3-D mesh the two component CAD models. 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.

Fig.3. Assembly

The two components are bolted to each other and this is simulated using a bar element and are connected to the individual 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.

iii) MATERIAL SELECTION

Mild Steel: Component 1 was made of mild steel as it was to be welded on the tubular steel chassis.

Young's modulus, E= 2.1 e+05 MPa
Poisson's ratio, u = 0.3
Density, rho = 7.9e-09 tonnes/mm^3

Aluminium 6063: Component 2 was decided to be of this material due to its light weight.

Young's modulus, E= 7.8 e+04 MPa
Poisson's ratio, u = 0.33
Density, rho= 2.1e-09 tonnes/mm^3
Yield Strength = 214 MPa

Changes in material-selection could be made once the analysis was done.

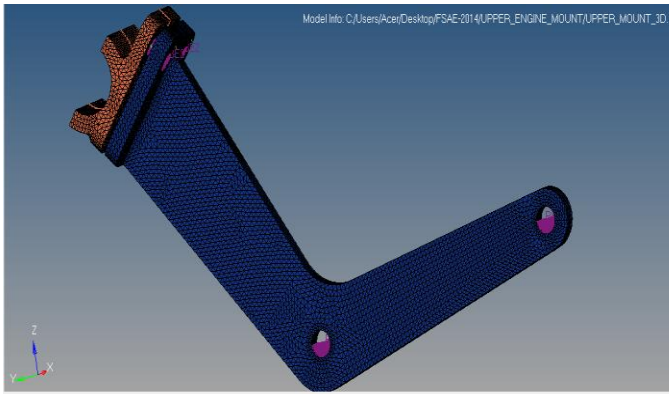

Fig.4. Meshed Assembly

iv) STATIC & DYNAMIC FORCES

The forces that act upon externally when the system is not under motion are applied in static analysis. This involves weight of the engine distributed on each of the mounting brackets, manually applied loads and thermal stresses. The deflection and strains in the bracket components are then checked for safety.

A *Suzuki gsxr 600cc*, 4-cylinder engine has been used for our car. The weight of the engine is 67 kgs.

There are a total of four mounting positions for the engine on the chassis. According to the position of the centre of gravity of the engine from the bracket positions the static weight transfer on the current bracket was theoretically calculated to be 190 N. This weight is further distributed on the two bolting positions available on the engine. Performance of aluminium at high temperatures up to 110 degree Celsius was also checked.

Dynamic forces involves those experienced by the vehicle in motion and which vary with time. For this, the maximum G-forces during acceleration, braking and cornering are considered.

Acceleration loading= 1 G

Cornering loading = 1.2 G's

Vibrational acceleration loading was checked using an accelerometer.

v) CONSTRAINTS

Deciding the boundary conditions is very important in Finite Element Analysis. Here, the welded area is directly constrained or connected to rbe2 elements. Bolts are connected to the brackets using rbe2 rigid elements.

b) MODAL ANALYSIS

This analysis determines the response to externally applied transient vibrations. Modal analysis gives us the form or mode shapes corresponding to the natural frequencies without considering any applied forces. The mode shapes obtained are important because they show us the direction of deflection and free amplitude of vibration for each natural frequency.

Since the brackets are pre-stressed the modal analysis needs to be done under this condition. Maximum amplitude of deflection under free vibration was checked for each design at each of the resonant frequencies lying in the operating frequency-range.

Idling: 2100 RPM

Maximum: 12500 RPM

Now,

$f = N/60*(c/n)$, where

N- Engine RPM

f- Frequency of vibration in Hz

c- Number of engine cylinders

n- Number of crank rotations per cycle

Hence, the operating range was calculated to be 70Hz-417Hz.

c) FATIGUE ANALYSIS

Since the brackets are subject to continuously varying loads, their fatigue life needs to be calculated. First the approximate average mean and amplitude stresses are calculated. Then from the calculated endurance limit, the no. of life cycles is determined.

But due to the presence of weld joint on the chassis, the fatigue life reduces. Hence, the type and dimensions of the weld is suitably decided. Also to increase the weld area the holding bracket is given a curve according to the profile of the chassis member.

d) ACCEPTANCE CRITERIA

The maximum von-mises stress under worse loading condition obtained from the FEA results should be safely below the yield strength of the material. Because yielding is considered to be failure of the design, even if there might be no breakage. The maximum deflection is also noted down to compare the preliminary and final design.

III. PRELIMINARY DESIGN

a) STATIC ANALYSIS RESULTS

Only the engine weight and external manually applied loads are considered here

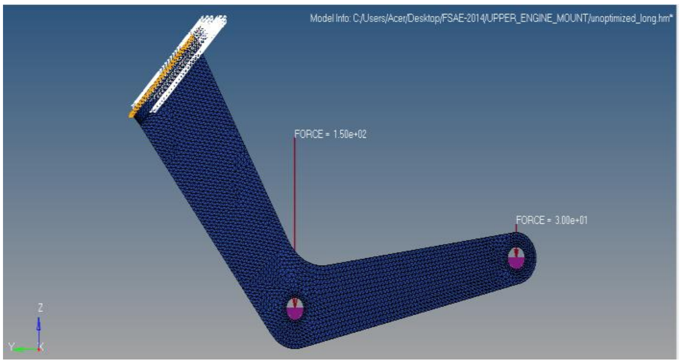

Fig.5. Boundary conditions

The analysis was done in OPTISTRUCT solver. The diagram shows the boundary conditions for component 2. Plot showing von-mises stress is as below:

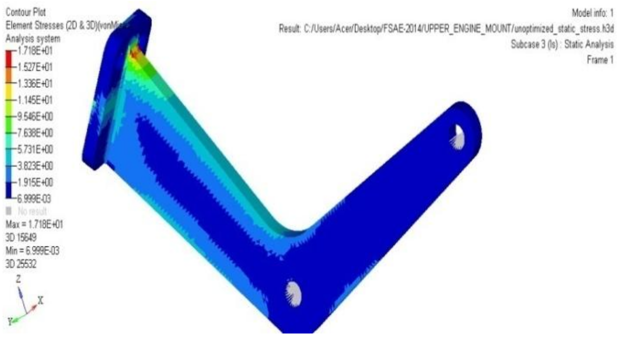

Fig.6. Stress-plot

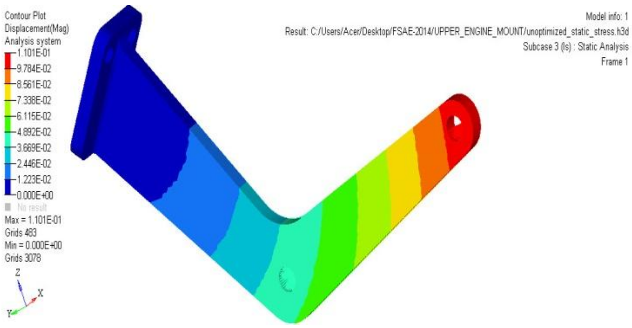

Fig.7. Deflection plot

Maximum von-mises stress in static loading is 17.2 MPa according to the plot. This is because bending moment is maximum at the constrained edge of the bracket. Stress reduces along the upper surface and is negligible in the centre portion.

The maximum deflection is 0.11 mm at the farther most point from the constrained position.

b) DYNAMIC FORCE ANALYSIS RESULTS

In this static, cornering and braking forces are considered together. The stress and displacement plots are shown below.

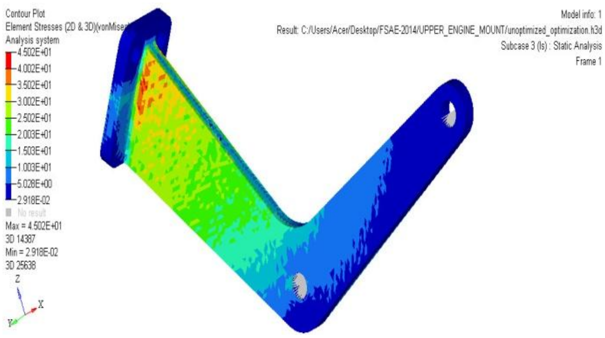

Fig.8. Stress

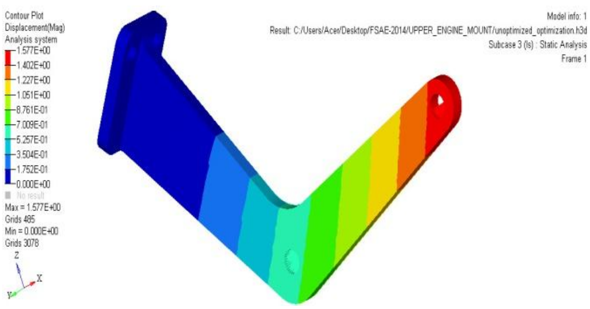

Fig.9. Displacement

Maximum von-mises stress is 45.02MPa. This stress occurs again at the fixed edge or base of the bracket due to the sharp contour transition. The maximum displacement shown is 1.577 mm. This magnitude will be lesser in actual conditions since the other three mount brackets constrain the engine’s movement too. Weight of the bracket- 403 grams

c) MODAL ANALYSIS RESULTS

Normal mode frequency analysis was done without considering any externally applied forces and without considering any amount of damping.

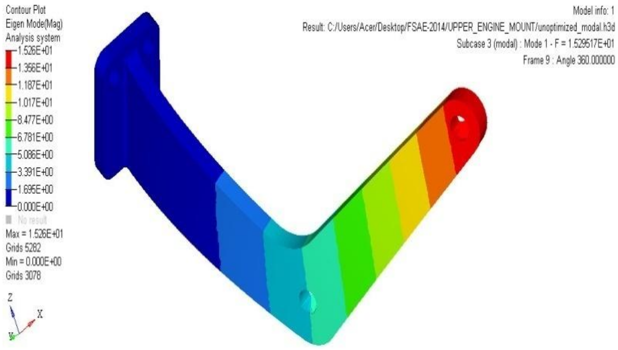

Fig.10. 1st Mode shape

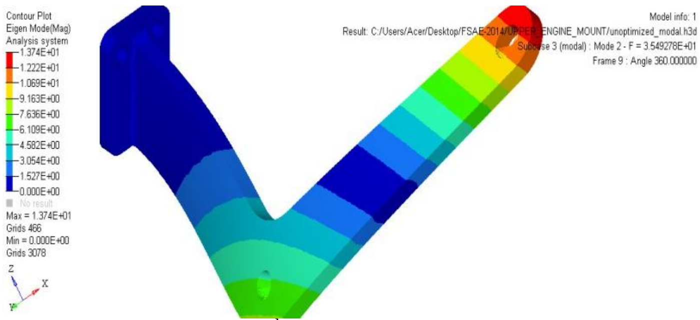

Fig.11. 2nd Mode shape

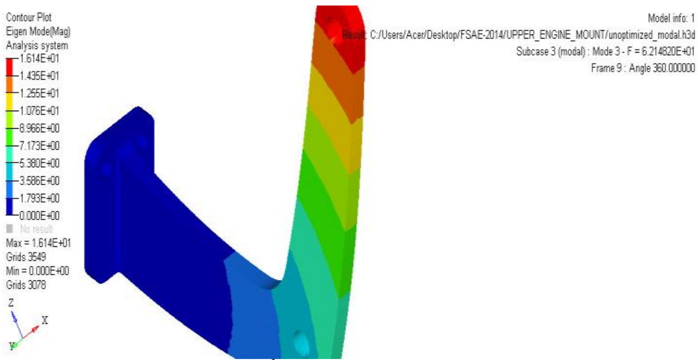

Fig.12. 3rd Mode Shape

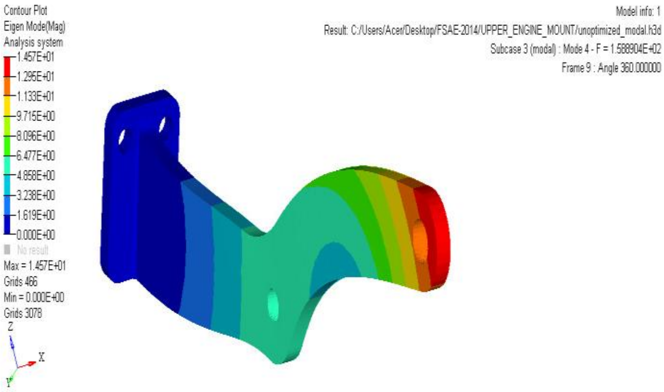

Fig.13. 4th mode Shape

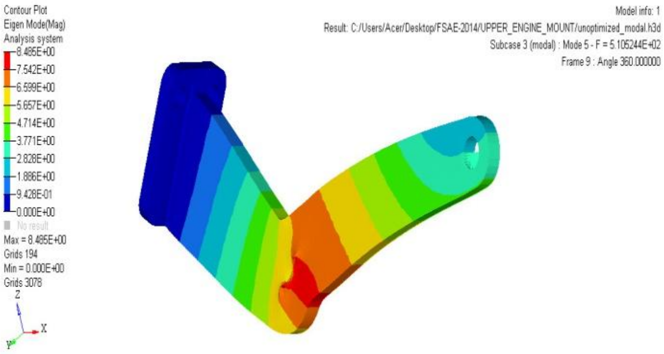

Fig.14. 5th Mode shape

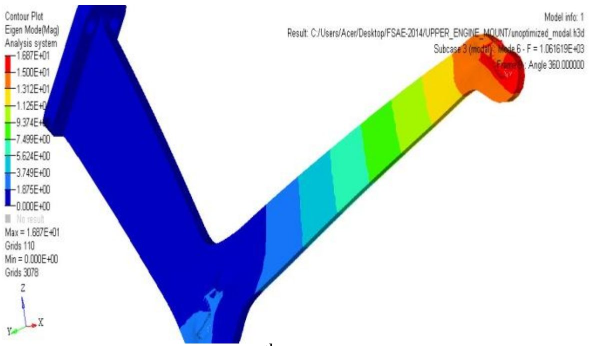

Fig.15. 6th mode shape

IV. COMPONENT No.1 ANALYSIS

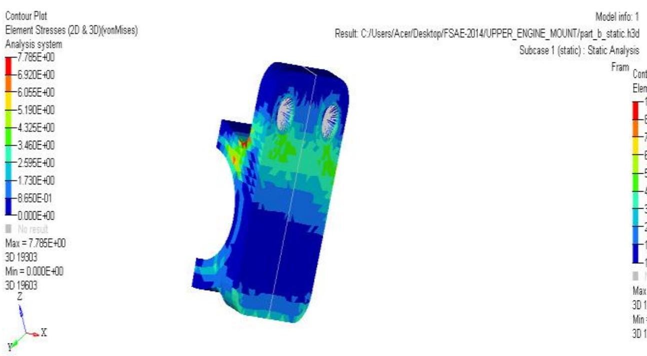

Fig.16. Stress-plot 1

Static and dynamic forces were applied on the meshed model and the following stress and displacement plots were obtained. Max stress was about 8 Mpa in the MS Component, hence was found to be sufficiently safe.

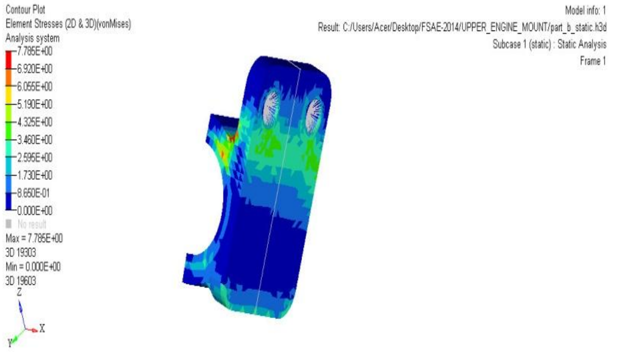

Fig.17. Stress plot 2

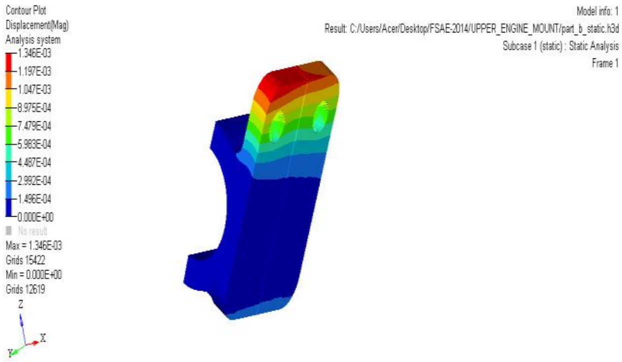

Fig.18.Displacement plot

Maximum deflection was found to be 0.001346 mm. Hence it was very much within permissible limits.

V. DESIGN MODIFICATION

From the above results of the preliminary design the following modifications were made-

a.) Addition of fillets at sharp edges due to stress concentration.

Also, the max displacement of 1.577 mm being high, an additional rib was added on the bracket.

b.) Since, addition of an additional rib meant addition of weight, mass optimization of the bracket was also done. The results were as follows :

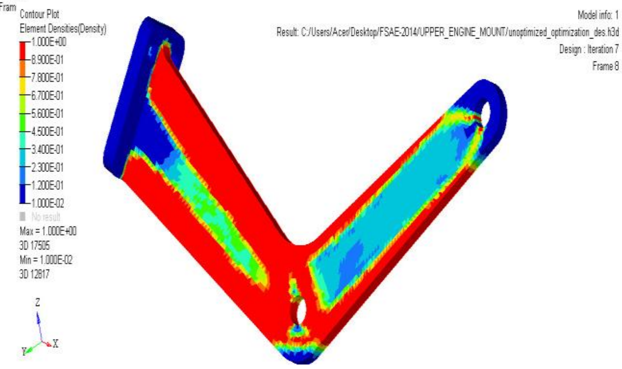

Fig.19. Optimization

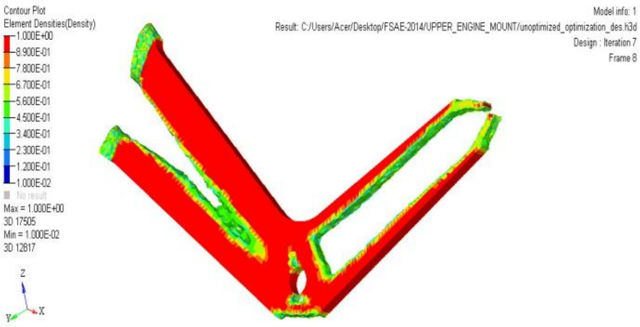

Fig.20. Optimization with removed material

In the given image, the red coloured patches represent the required or necessary material to safely bear the stresses, whereas the blue areas show the non-required material. Hence, we concluded that the middle portions of the bracket were not important in bearing the loads. Hence the modified design was made as follows.
Material was removed in the form of holes instead of rectangular slots, since by using holes we get better stiffness and less deflection comparatively. Material in between the rib and the bracket was removed too in order to reduce weight.

VI. MODIFIED DESIGN

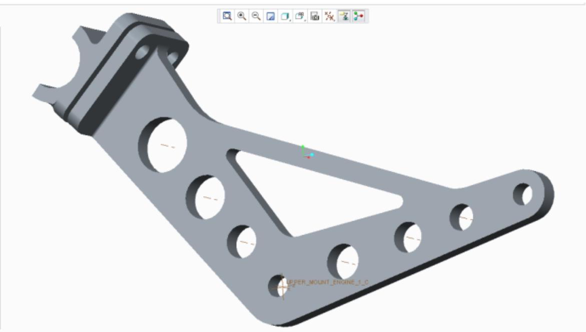

Fig.21. Modified bracket

a) MESH OF NEW MODEL

The new CAD model was tetra-meshed again as shown in the figure below:

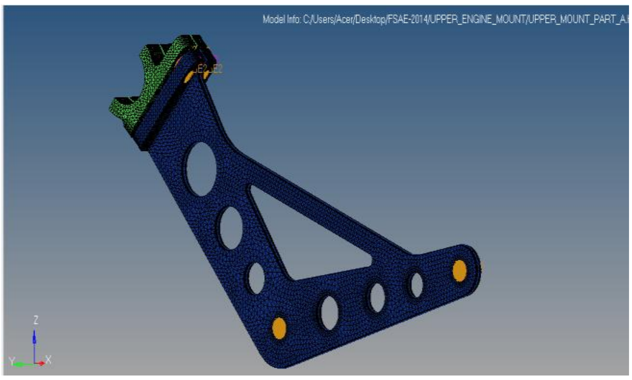

Fig.22.New meshed model

b) STATIC AND DYNAMIC FORCE ANALYSIS RESULTS

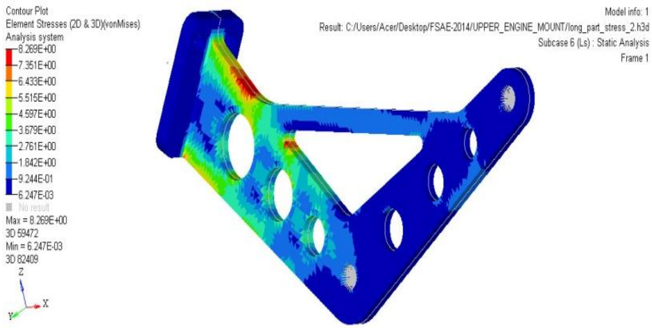

Fig.23. Optimized stress plot

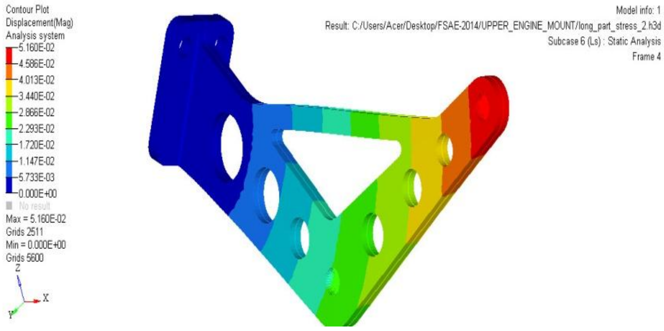

Fig.24. Optimized displacement plot

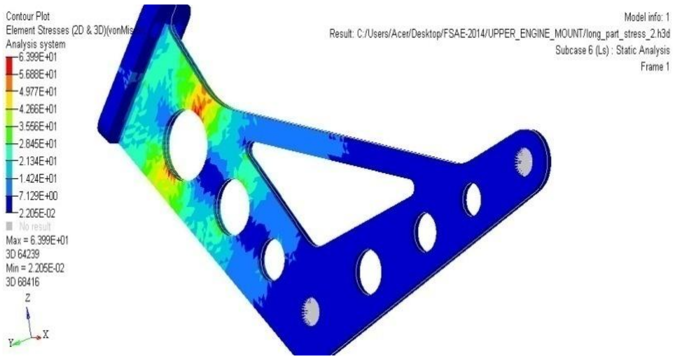

Fig.25. Optimized stress-plot for cornering and braking condition

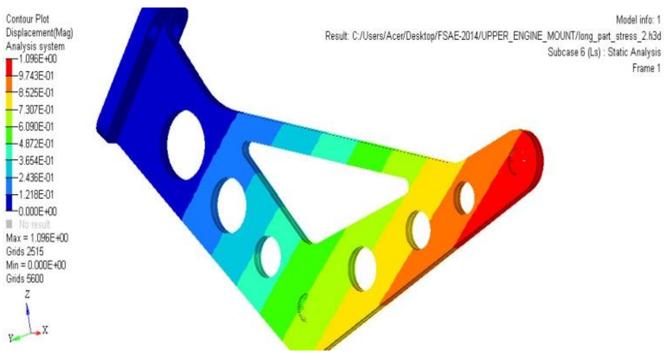

Fig.26. Displacement plot

The observations were:
Maximum stress in static condition- 8.3 MPa
Maximum displacement in static condition- 0.0516 mm
Maximum stress due to dynamic loading- 63.99 MPa
Maximum displacement with max loading- 1.096 mm
Hence, compared to the preliminary design, the maximum displacement has been reduced by about 30.5%. Also, the factor of safety obtained was 3.3.
Weight of new bracket – 356 grams. Hence, weight has been reduced, thus saving material.

c) MODAL ANALYSIS

Modal analysis was also performed on the new bracket. The mode shapes obtained were similar to that of the preliminary bracket. Only the values of natural frequencies and amplitudes changed. A tabular comparison is given as follows:

Mode Shapes(1-10) of Engine Bracket		
Frequency	Un-optimized bracket (Hz)	Optimized bracket (Hz)
1st	15.59	14.1
2nd	36.19	34.57
3rd	63.38	63.1
4th	162.04	258.78
5th	520.066	440.2
6th	1082.7	887.63
7th	1466.4	1200.5
8th	2063.03	1672.7
9th	3640	3012.1
10th	4692.4	3653.3

Fig.27. Comparison of natural frequencies

From the table, the first six frequencies shown are called as the natural frequencies. Hence, if the external disturbance frequency matches these, resonance takes place. In both the cases the 4th natural frequency lies in the operating/exciting frequency range.

It is observed that the optimized bracket has lower natural frequency up to engine-idling frequency. Hence, this is desired since resonant amplitude of vibrations is very high at low frequencies. The frequencies above 410 Hz are of less significance because the engine seldom operates at RPM's corresponding to those frequencies. Also, the amplitude of vibration greatly reduces in the higher range, but noise becomes a major concern. Higher the resonant frequency, higher is the level of noise disturbances.

Given below are two superimposed graphs of *frequency (Hz) Vs Amplitude (mm)* for the un-optimized and the optimized brackets.

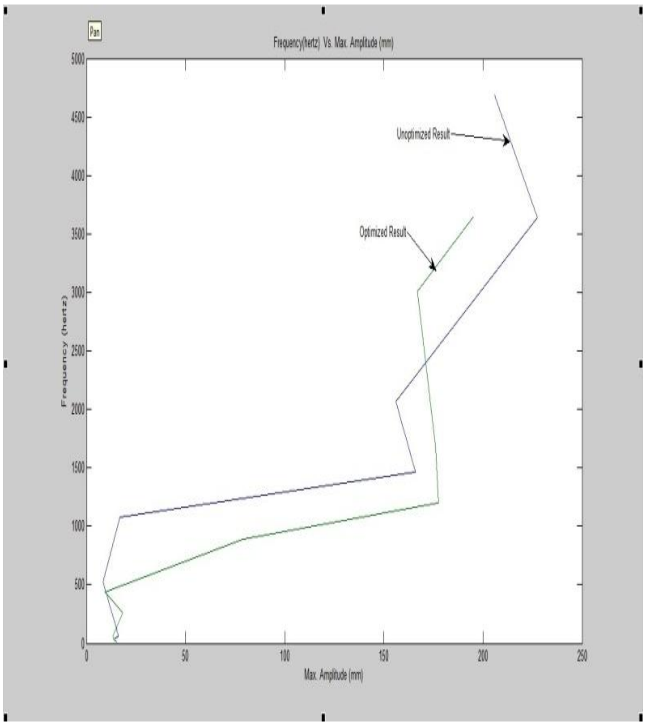

Fig.28. Comparison of Freq. vs. Ampl.
For the two designs

The amplitudes of free vibration at a particular frequency are more for the preliminary or un-optimized bracket whereas it is lesser for the modified or optimized bracket. Hence the modified bracket's response to free-vibrations is better than the older design.

d) FATIGUE LIMIT

From the stress results obtained, the mean and amplitude stresses came to be about 33 MPa. The endurance limit calculated for the bracket was calculated to be 71MPa. Hence, using S-N curve the life in cycles was more than 10^6.
Considering the weld strength, the no. of life cycles calculated were more than the required vehicle-performance engine cycles.

VII. EXPERIMENTAL VALIDATION

After the actual installation of the bracket and set-up, the vehicle was tested rigorously. Cornering and braking forces of up to 0.8 G's were achieved during the continuous testing period of 5 days. The performance of the bracket was satisfactory as the driver(s) felt minimal vibrations. No signs of surface failure or cracks in the bracket as well as the weld were seen when checked. There was no significant noise disturbance resulting from the mounts too.

Since, the 4^{th} natural frequency of 258.8Hz lied in the operating range, the magnitude of vibrations were checked using an accelerometer at the corresponding 7764 RPM of the engine. Maximum amplitude of vibration was found to be 0.115 mm in the vertical direction. Also, according to the mode shape the direction of amplitude-magnification at resonance was in the vertical direction as well. But the vertical motion had been constrained with the help of the other three mounting brackets and hence the effect had been neutralized.

Fig.29. Installed Mount Bracket

VIII. RESULTS

Result Table			
	Maximum Displacement (mm)	Maximum Von-Mises stress(MPa)	Weight (grams)
Preliminary Design	1.577	45.02	403
Modified Design	1.096	63.99	356

IX. CONCLUSION

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.

REFERENCES :

[1] Karl D. Hammond, "*Frequency Response Analysis*", January 2008.

[2] Rogers Corporation," *Materials Design: Understanding Load Vs Frequency Curves*".

[3] Nitin S Gokhale, Sanjay S Deshpande, Sanjeev V Bedekar, Anand N Thite, - "*Practical Finite Element Analysis*".

[4] 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 (IJMIE) ISSN No**.** 2231 –6477, Vol-2, Iss-3, 2012.

[5] Sahil Naghate, Sandeep Patil, "*Modal Analysis Of Engine Mounting Bracket Using FEA*", International Journal of Engineering Research and Applications (IJERA), Vol. 2. Issue4. July-August-2012,pp.1973-1979.

[6] Koushik S.," *Static and Vibration Analysis of Engine mounting Bracket of TMX-20-2 using OptiStruct*", Altair Technology Conference, India -2013.

[7] R Singh, "*Dynamic Design of Automotive Systems: Engine Mounts Structural Joints*", SaÅdhanaÅ, Vol. 25, Part 3, June 2000, pp. 319±330.

[8] Dr. N.K. Giri, "Automobile Mechanics", Khanna Publications, ISBN No. 81-7409-216-1, Edition-2006.