Design and Analysis of Engine Mounting Bracket

Mr. Sachin Walke [1], Dr. Vikramsingh Mane [2]
[1,2]TPCT college of engineering osmanabad

***Abstract*—Engine mounting bracket plays very significant role in reducing noise, vibration and harshness caused due to engine and thus has very effective role in improving vehicle comfort. This current work accounts for the investigation of engine mounting bracket by using hyper mesh and Optistruct approach. Static analysis of engine mounting bracket was done in order to check design of existing and modified bracket. The results were analyzed for stresses and deformations. The design was tested for different design of Mild Steel with different thickness. From Design and analysis, it is considered that stresses induced in the bracket were 260.00 Mpa and deformation 9.6 mm. It can be anticipated that modified brackets can be considered for desired application.**

***Index Terms*—Hyper mesh, Optistruct, Mild Steel etc.**

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. [1] 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. [2] 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. 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. [2] 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. 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.

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. It is known that body attachment stiffness is an important factor of idle noise and road noise for NVH performance improvement. The problem of vibration has two classes, first is of forced isolation and other is of motion isolation. In later one, blunders can be avoided by controlling natural frequency of bracket lower than the self-excitation frequency. So as a counter part of this law of transmissibility is considered. [3]

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.

Working of Engine Mounting Bracket Engine mounts are what separate you from the nasty vibrations and harmonics created by the internal combustion engine. Without these simple items, you would feel exactly how rough the engine in your vehicle really is, even when running properly. Engine mounts are basically rubber isolators that are mounted between the engine in a vehicle and the frame. They hold the engine in place while absorbing the vibrations caused by the engine, creating a quiet, smooth feeling inside the vehicle.

Fig 1.1 Engine Mounted on Engine Mount [2]

They are made of rubber to absorb vibration without transferring it, but some manufacturers have tried using a liquid (oil) filled mount to dampen vibration with some success. In some performance applications polyurethane or solid steel mounts are used, these mounts transfer vibration, but can withstand the abuse and high-horsepower applications seen in racing where a comfortable, smooth ride isn't really an issue.

II. LITERATURE REVIEW

A.S. Adkine et.al [1] ". Design and Analysis of Engine Mounting Bracket Using Ansys Tool" International Journal of Innovation in Engineering, Research and Technology [IJIERT] 2016. This current paper accounts for the investigation of engine mounting bracket by using ANSYS. Static and modal analysis of engine mounting bracket was done in order to investigate whether the current natural frequency of engine mounting bracket is lower than that of self-excitation frequency of bracket. The obtained results were also examined for cross section of bracket. It was found that circular cross section having stress induced 128.47MPa is more reliable than square cross section. The results were analyzed for stresses and deformations. The design was tested for different materials like Magnesium, ERW-1 and ERW-3 along with suitability of material. Stresses induced in magnesium bracket were 64.07 MPa with the deformation of 1.20 mm. It can be anticipated that magnesium brackets are corrosion resistant and can be considered for desired application.

Jasvir Singh Dhillon, et.al [2] *"Design of Engine Mount Bracket for a FSAE Car Using Finite Element Analysis" Int. Journal of Engineering Research and Applications.2015.* 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 ensure 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.

Sebastian C S, et.al [3] "*Design and Optimization Of Engine Mount Bracket" International Journal of Application or Innovation in Engineering & Management (IJAIEM)2016.*

The study of a jet engine mounts by FEA usually deals with the stress analysis of the mount, a very few papers deal with the displacement analysis. It is futile to believe that mounts that only stress analysis is necessary, as the displacement of the mount(elongation) is also an important factor when real life scenarios come into play. For the study the mount was designed using CATIA V5 and was analyzed using ANSYS. The ends that were to be fixed onto the chassis were fixed and then suitable load of 1000[4]N was applied to the load bearing region. Study was initially done using Al alloy and Mg as materials and then titanium alloy(Ti 6Al 4V) was also used on the final model. The initial models had a stress in the range of 10 and displacement was in the range of 20-30 meters which was unacceptable and suitable modification were made to the model in geometry and its dimensions so as to reach the optimal value of 5 mm displacement and stress of 2.5 KN for Al and 10 mm displacement and 3 KN stress for Mg alloy. Later the study was done using Titanium alloy (Ti 6Al 4V) and it is seen that the stresses and displacement were well within the permissible limit. From the study it is very evident that of all materials Titanium alloy had the minimal deviation from ideal behaviour, and it is safe to assume that Titanium alloy (Ti 6Al 4V) is the most suited alloy to make engine mounts.

III. PROBLEM DEFINITION AND OBJECTIVE

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. At first the theoretical study of bracket is done. The key areas for modification are identified. The main task in this study is to do static analysis of bracket and to reduce deflection and stress for mounting bracket. The 3-Dimensional model is prepared for Bracket. Different modification in shape and design of bracket is done and static analysis is carried out in hyper mesh.

- The main objective of this research paper is to do static analysis of Engine mounting bracket.
- To redesign bracket for easy manufacturing.

IV. THEORETICAL ANALYSIS

A. The dimensions of Mounting Bracket

Dimension of Lower Control Arm is as follows:

Overall length is 187mm, width is 150and thickness is 4mm

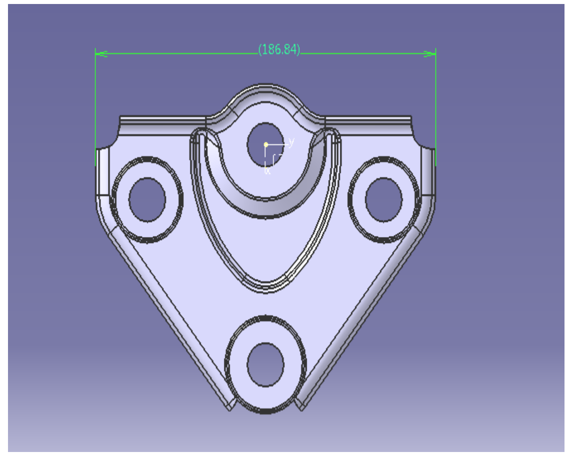

Fig. 2. Dimension of Mounting Bracket

B. The material properties of steel

The material is AISI 1040, which is having all these characteristics.

TABLE -IMATERIAL PROPERTIES

Material	AISI 1040
Young's modulus	2.1e5MPa
Poisson's ratio	0.3
Density	7850Kg/m3
Yield strength	415MPa
Tensile strength	620 MPa

C. Static Load calculation of Lower Control Arm

Gross Weight of Engine=300 Kg (considering passengers and accessories weight)

Total Weight in Newton = 300 x 9.81

W =2943 N

It is assumed that 52% of weight taken by front axle, due to mounting of engine on front side and remaining 48% weight taken by rear axle. Therefore,

Weight on Front axle = 0.52 x 13243.5 N

F_1= 6886.62 N

Weight on Rear axle = 0.48 x 13243.5 N

F_2 = 6356.88 N

Reaction at each front wheel

R_w= Weight on Front axle/2

R_w = 6886.62/2

R_w =3443.31 N

This load is constituted by spring, stub axle and lower control arm. While stub axle of the wheel takes 50% of the total load acting on each wheel. Therefore, force acting on the stub axle of wheel is given by,

F= 1721.6 N

Following line diagram is a representation of the spring, and lower arm.

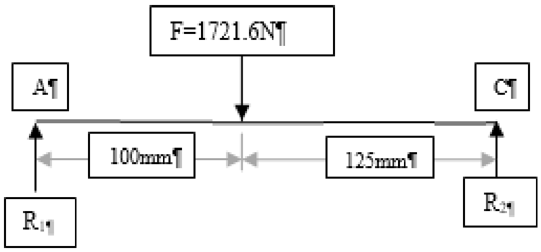

Fig.3. Line diagram for force distribution

Where,
R_1=Reaction for spring in Newton.
R_2=Reaction for lower arm in Newton.
F = Force acting on stub axle in Newton
Therefore, from equilibrium condition, taking moment at A is equal to zero.

$\sum M_A = 0$

$Fx100-R_2x225=0$ (1)

$1721.6 \text{ x } 100 = R_2 \text{ x } 225$

$R_2 = (1721.6 \text{ x } 100)/225$

$R_2 = 765.15$ N

This is vertical load acting on the lower control arm.

Now, $R_1 + R_2 = F$

$R_1 = F - R_2$ (2)

$R_1 = (1721.6-765.15)$N

$R_1 = 956.45$ N

This reaction is acting vertically upward at spring.
Therefore, the Reaction R_2 = 765.15 N
Approximately taken as $R_2 \approx 765$N, which is acting in vertically downward direction on lower control arm.

V. MODELLING OF ENGINE BRACKET

CAD software like CATIA is higher end software which is feature based solid modeling systems. CATIA V5R30 is used for modeling of Lower Control.

A. 2DModeling" of Existing Engine Mounting *Bracket*

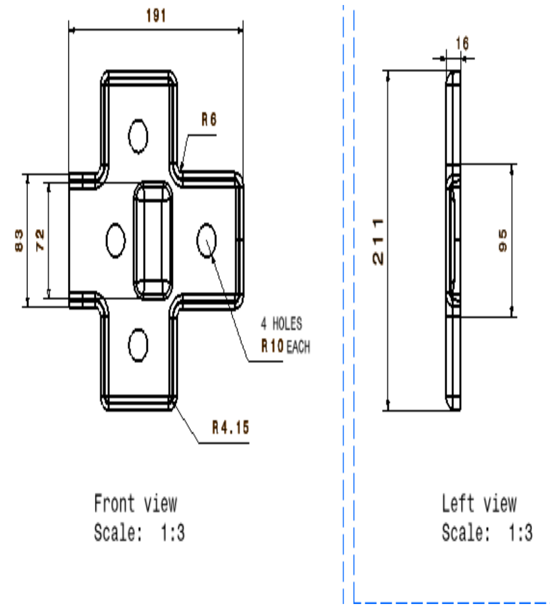

Fig. 4. '2D' Views of Mounting Bracket

B. "3DModeling" of Engine Mounting Bracket

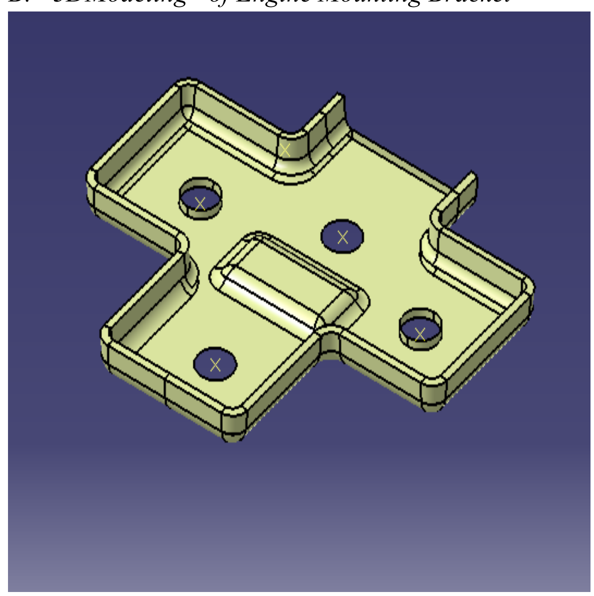

Fig. 5. Top View 3D of Mounting Bracket

VI. FINITE ELEMENT ANALYSIS

HYPERMESH with Optistruct approach is used to mesh the solid model. Assembled CAD model which is in IGES format is imported to HYPERMESH.

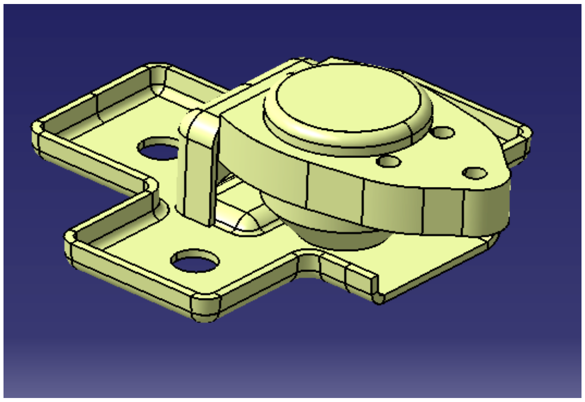

Fig. 6. Baseline Assembled Engine Bracket for FEA

A. Meshing of baseline geometry

The conventional model which was developed in CATIA software has to be meshed for analysis. For this HYPERMESH workbench software is used. It is a high-performance finite element pre-processor that provides a highly interactive and visual environment to analyze product design performance. With the broadest set of direct interfaces to commercial CAD and CAE systems. The solid tetrahedron elements are used to generate the meshing of the control Arm.

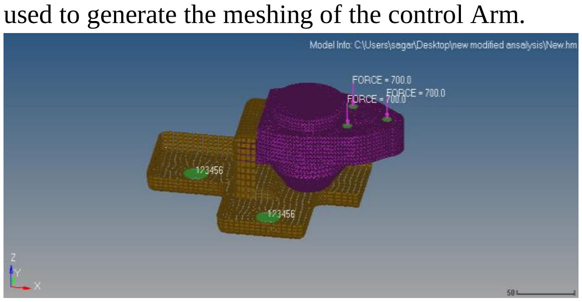

Fig. 7. Meshing of Engine Mounting Bracket

TABLE-II DETAILS OF MESHING

Sr. No.	Description	Values
1	Number of Nodes	53002
2	Number of Elements	26694
3	Element Size	Maximum 5 mm Minimum 3 mm

B. Design parameters

In case of vehicle in actual running conditions forces acting on it are of dynamic in nature and changes as per driving conditions. In order to make preliminary analysis steady state operating conditions are assumed.

C. Boundary conditions of baseline geometry

Fig.8. Connection of Mounting Bracket

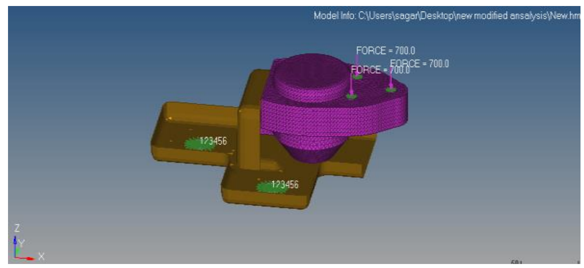

Fig. 9. Boundary Conditions of Engine Mounting Bracket.

In this base bracket is connected to chassis with bolting further supporting bracket is welded to base bracker and an isolator is bolted to end bracket. The load of engine along with vibration coming from road is directly transferred to mounting bracket.

D. Analysis Result of Baseline model

After Finite Element analysis on HYPERMESH workbench 14.0 following results have been find out. The displacement contour plots are shown in the below figure10.

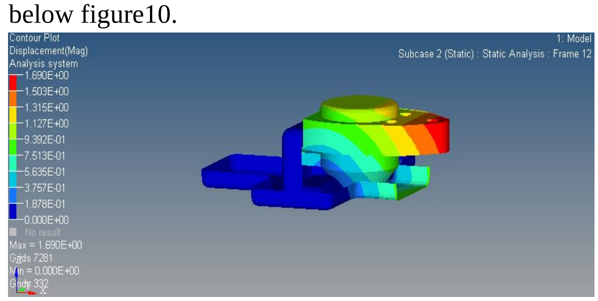

Fig. 10. Maximum Deformation Plot

The maximum displacement shown by the baseline lower control arm is 16.9 mm.

The following figure11 shows contour plot of the von-Mises stress

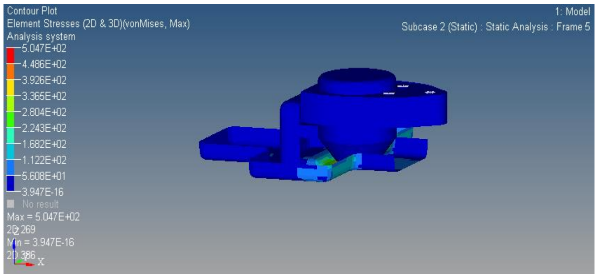

Fig.11. Equivalent Stress Plot

As per distortion energy theory, the maximum equivalent stress observed in the lower arm model is 504 MPa.

The tensile strength of the material is 620 MPa. According to results, the von-Mises stress 504 MPa is lower tensile strength of the material. The factor of safety of the baseline lower Arm is 1.21. Mass of the Baseline design = 1.2 Kg

D. Topology Optimisation by FEA

Topology Optimization is defined as finding out the best possible solution of problem by considering the given sets of

objective and number of constraints. For solving any topology optimization problem it have to specify three parameters that is Design Variables (material density), Design objective (Weight reduction) and design constraints (Volume)[6].Topology optimization is performed on a model to create a new topology for the structure, removing any unnecessary material. The resulting structure is lighter and satisfies all design constraints.

The topology optimisation of control arm model is carried out in OPTISTRUCT software. The Material data for carbon steel remain same as used in the static structural analysis. The optimisation model includes same boundary conditions as used in the static analysis of baseline model. Topology optimization carried out for the following objective

Objective	To minimize volume (reduce weight)
Constraints	von-Mises stress < 620 MPa (Tensile strength of material)
Design Variables	The density of each element in the design space.

E. Optistruct Model

The Optimised CAD is prepared in the CATIA software. The redesigned model prepared in CATIA is exported in .step format and imported in HYPERMESH. Following is the CAD imported in HYPERMESH for meshing. The prepared CAD is divided into design space and nondesign Space. The design space is the region where design optimisation will be carried out. The nondesign space is the region where, no design change will be done by the software.

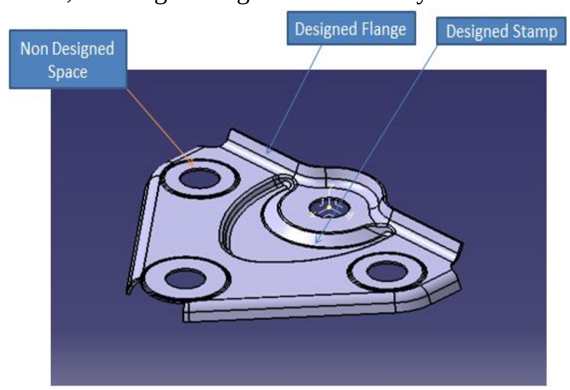

Fig.13. Optimized geometry

The optimised CAD model is meshed in HYPER MESH with CTETRA elements.

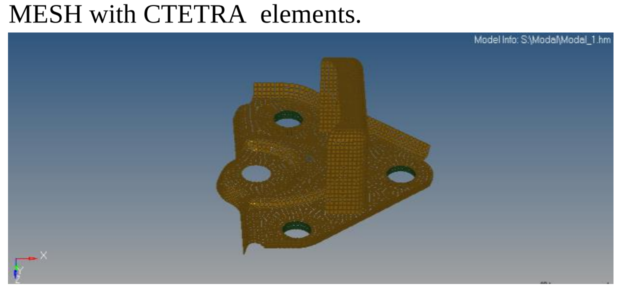

Fig.14. Mesh model for Optimization

TABLE- IIIDETAILS OF MESHING

Sr. No.	Description	Values
1	Number of Nodes	32536
2	Number of Elements	98131
3	Element Size	2 mm

The rigid body motion of the Lower control arm are restrained by constraining the faces of the holes where it is fixed with the screw connections as shown in the figure below in red colour region. The x, y, z translation and ROTX, ROTY, ROTZ rotations are fixed in all directions.

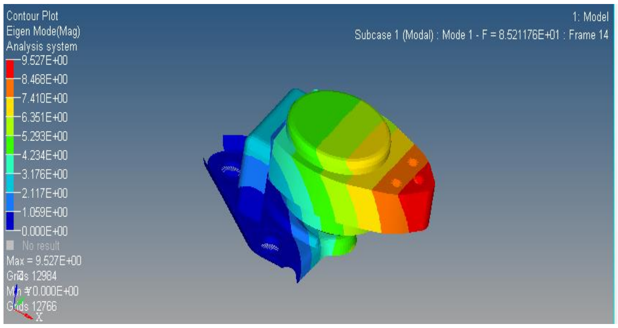

Fig.15. Boundary conditions for Topology optimization

F. Analysis result for optimised model

The element density plots shows the optimized pattern of the model. The white region in lower control arm indicates the unnecessary material to be removed from the design. The optimized design is extracted from the raw design obtained through analysis. The optimized design is prepared in the CATIA software.

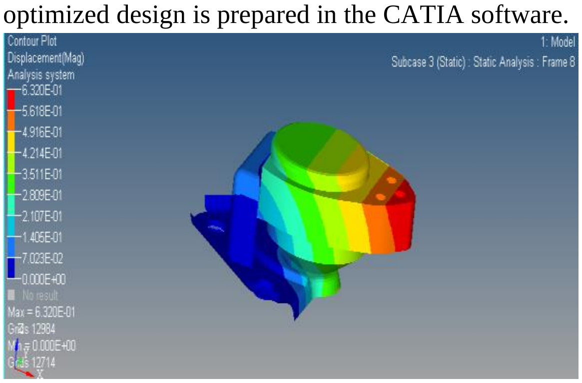

Fig.16. Displacement Analysis of Modified Bracket

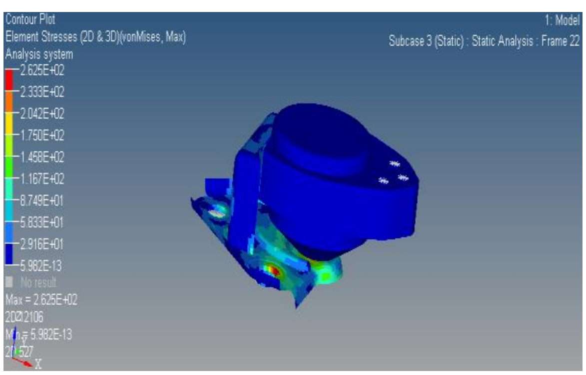

Fig.17. Stress Analysis of Modified Bracket

Above figure shows that low stess blue region can be removed from the design space while keep the red region in the design space as it is. Mass of the optimised design is obsereved to be 0.99 Kg.. As per Element Density plot new optimized LCA model is designed in CATIA and it is analysed in ANSYS for same boundary and loading conditions. The maximum deformation for the optimised design is observed up to 9.5 mm. The von-Mises stress is observed upto 262 Mpa for optimised model.

VII. EXPERIMENTAL ANALYSIS

Universal testing machine also known as Universal tester or material testing machine which is used to test the tensile strength and compressive strength of materials. For mounting of LCA on UTM for testing, proper fixture has been design. Following figure shows assembly of LCA and fixture

Fig.18 Fixture

To verify the deflection and stress values of Lower control arm, experimental testing of both the arm structure is done on universal testing machine in metallurgical laboratory. The readings from the machine are used to verify with the Finite element analysis results. Figure 22 shows the experimental setup for lower control arm. Load is applied on arm by using of load cells of universal testing machine.

Fig.19 Experimental Setup

The peak 765N load is applied on both arm models to find out deflection value on that peak load. A deflection value for the baseline arm is 7.7 mm and for new model is 10.4 mm.

VIII. RESULTS AND DISCUSSION

Static analysis of existing and modifbracket ied is carried out by experimentally and Finite element method. Following graph shows the results of both model by Finite element method.

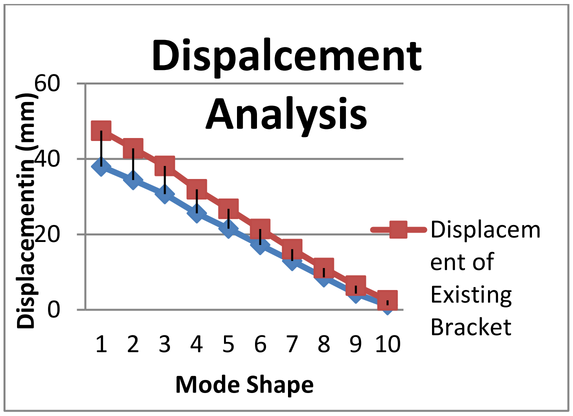

Fig.20. Graph plot of Force vs. Deflection

The same above compassion we will see by graphical way, from that we come to know that displacement value decreases as mode shape increases. From the above graph it is observed that deformation of a optimized model as compared to baseline model is varies upto 9.5 mm with the gradual application of load.

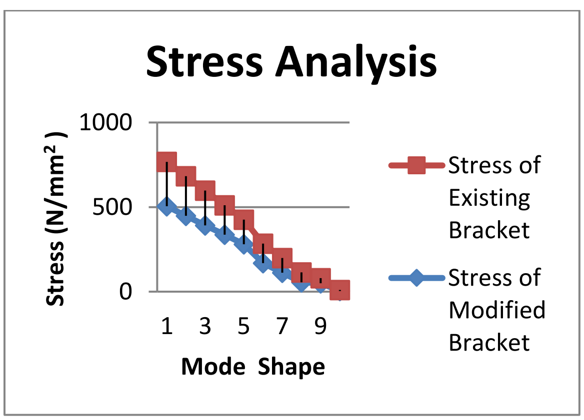

Fig.21. Graph plot of Stress vs. Mode Shape

This indicates that design is safe for the applied load. The same above compassion we will see by graphical way, from that we come to know that displacement value decreases as mode shape increases. From above it can be seen that stress for modified designed bracket reduces as mode shape increases. The maximum value of stress from graph is 500N/mm^2 which is less than Stress in the existing bracket. From the above graph it is observed that there is a increase in stress occur in optimized model due to reduction in mass but this increased stress is below the ultimate limit. Above graph shows that maximum stress induced in both model is below the ultimate tensile strength of material.

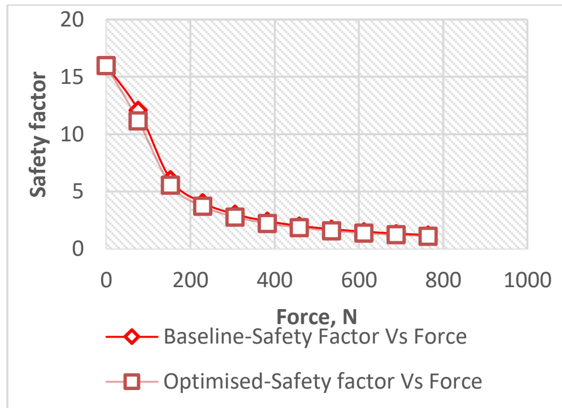

Fig.26. Graph plot of Safety factor vs. Force

Above graph shows that there is slight variation in factor of safety by 10% can leads to 17.5% reduction of mass of single lower control arm.

Total weight reduction in one control arm is found to be 210gm. There is presence of two lower control arm in Mac pherson suspension system. So overall weight reduction in front suspsension system is 420gm.

The cost of AISI 1040 is Rs.51.36 per Kg. According to this baseline lower control arm(LCA) of 1.2Kg requires material of Rs.61.95 whereas optimized LCA of 0.99 Kg requires material of Rs.51.11. So total cost saving in material of one arm is as follows

$C_T = C_B - C_O = 61.95 - 51.11 = \text{Rs.}10.84$

Where,

C_T = Total cost saving in material of one bracket

C_B = Cost of material for baseline bracket

C_O = Cost of material for optimized bracket.

Let say in mass production company manifactures 1000 parts per week, so total cost saving per 1000 parts will be Rs.10842.75 which is good achievement in company prospective.

TABLE- IVRESULT ANALYSIS

Sr. No.	Method	Description	Baseline Design	Optimised Design
1	Experimental Method	Deflection, mm		
2		Von-Mises stress, MPa		
3		Mass, Kg		
4	FEA Method	Deflection, mm	36mm	11mm
5		Von-Mises stress, MPa	505	263
6		Mass, Kg	1.20	0.99
7	Theoretical analysis	Factor of Safety	1.22	1.12

IX. CONCLUSION

As deflection and stress of modified LCA is within the range. Thus, the modified design is safe. Weight of the final optimized model is 0.99 kg. The total reduction in mass is observed 17.5% by keeping Factor of safety for optimized design within permissible limits. Thus the objective of weight reduction of unsprung mass and cost reduction has been achieved.

X. ACKNOWLEDGMENT

I take this opportunity to thanks Prof. Dr. Vikramsingh Mane and Prof. Dr. Patel for valuable guidance and for providing all the necessary facilities, which were indispensable in completion of this work. I am very much thankful to Radhe Engineering Services and Solutions and entire team of Radhe for active and timely cooperation thereout working of project.

REFERENCES

[1] A.S.Adkine et.al [1] ".Design And Analysis Of Engine Mounting Bracket Using Ansys Tool" International Journal of Innovation in Engineering, Research and Technology [IJIERT] 2016

[2] Jasvir Singh Dhillon, et.al [2] " Design of Engine Mount Bracket for a FSAE Car Using Finite Element Analysis" Int. Journal of Engineering Research and Applications.2015

[3] Sebastian C S, et.al [3] "Design And Optimization Of Engine Mount Bracket" International Journal of Application or Innovation in Engineering & Management (IJAIEM)2016

[4] Vikas Palve. et.al Manager - CAE [3] "Vibration Fatigue Analysis of Sheet Metal Fender Mounting Bracket & It's Subsequent Replacement With Plastic" Altair Technology Conference

[5] Khanehmasjedi et.al [3] "Comparative evaluation of shear bond strength of metallic brackets bonded with two different bonding agents under dry conditions and with saliva contamination" Journal of the Chinese Medical Association xx (2016)

[6] Eliades T, Brantley et.al WA. The inappropriateness of conventional orthodontic bond strength assessment protocols. Eur J Orthod 2000;22:13e23.

[7] Oztoprak MO, Isik et.al . Effect of blood and saliva contamination on shear bond strength of brackets bonded with 4 adhesives. Am J Orthod DentofacialOrthop2007;131:238e42.Schiradelly, R. and Diekman, F. J. "Cryogenics – The Racer's Edge", Heat Treating Progress, pp. 43-50, November 2001.

[8] P. Paulin, "Frozen gears", The Journal of Gear Manufacturing, pp.26-29, Mars/April 1993.