Optimization of Engine Mounting Bracket of Tata Ace Mini Truck

Shilpee Ghose[1], Prof. P. R. Shinde[2]

[1]Shilpee Ghose, P. G. Student (Dept of. Design Engineering), Siddhant College of Engineering, SPPU, Pune,
[2]Prof. P. R. Shinde, Mechanical Dept, Siddhant College of Engineering, Pune

---

Abstract

Engine mounts have nice impact on the noise and vibration harshness (NVH) characteristic and needs to face up to massive levels of vibration because of the character of the employment of the machine. This work focuses on style of Engine Mounting Bracket for TATA ACE mini truck. Static and Vibration analysis are performed on the model and failure modes are known. Style iteration study are in serious trouble the betterment of mount life by frequency improvement. the most important objectives of the project square measure to style and analyze structural frequencies and stresses on the engine mounting, and to validate the software package results victimization FFT analyzer testing of element.

In this project we are going to be coming up with the engine mounting bracket for Tata ACE mini truck. Then 3D model victimization modeling software package of mounting bracket are created. Then we are going to be performing arts static, modal and made vibration analysis on the bracket to verify the planning and for locating out the failure modes. With repetitious style study we are going to improve failure modes and natural frequencies of the bracket victimization totally different style parameters. Then we are going to choose the most effective suited style from FEA results for the improvement of vibration purpose. Sorting out the causes and factors contributory within the failure discovered within the current style, modification within the current bracket for improvement of wave failure performance.

***Keywords* – Optimization, Bracket, Fusion 360**

I. INTRODUCTION:

Engine is that the most vital elements of a road vehicle like automotive. High performance sports automotive has their engine supported by bracket. It plays a crucial role in up the comfort & work atmosphere of an automotive. The look improvement of engine bracket system has been the topic of intense interest for several years. It's necessary to style correct engine bracket for an automotive. As such, engine bracket has been designed as a framework to support engine. Vibrations and fatigue of engine, bracket has been ceaselessly a priority which can cause structural failure if the ensuing vibrations and stresses area unit severe and excessive. Prolonged exposure to whole-body vibration within the operating atmosphere might cause fatigue and in some cases it damages the automotive. Generally, the foremost necessary vibration relevant excitations in an exceedingly automotive engine is known as follows:- combustion force; main bearing reaction forces together with mass forces damper operate and regulator whirling, changed b y the front-end damper; piston aspect forces together with secondary motion; rotating shaft bearing reaction forces together with mass forces, gap and shutting impacts and bearing impacts; valve gap and shutting impacts; valve train forces caused by chain/belt movement or gear drive; gear train forces within the transmission; drive train reaction forces and moments. It's well-known from basic Non-linear vibration theory; improvement within the vibration management is achieved by determinant the natural frequency of the engine bracket system well below the band within which excitation exhibits most of the wave energy. It's during this context, the event of engine bracket will build the engine capable of interesting vibration. Automotive engine mounting system should satisfy the first tasks like engine movement, engine rigid body dynamic behavior, and vibration isolation. Style and development of mounting bracket through use of Autodesk Fusion 360 to attain the necessities for mounting system. Limits over the event of the mounting systems thanks to drivability and NVH considerations, provides savings in style resources. NVH is a crucial vehicles characteristic motivating to attain overall client satisfaction. Engine is generally mounted to the front sub frame and once put in in an exceedingly vehicle, engine mounting encompasses an important task in decisive the vehicle vibration characteristics. Optimizing the mounts system in early stages of engine style is feasible by implementation of computer-aided engineering (CAE) tools. CAE results is analyzed with none pricey prototypes. The results is wont to outline strategy for the vehicle mount system and optimize the locations and therefore the rates of mounts. A decent mounting system separates engine input vibration from the vehicle body and suppress the impact of road inputs to the vehicle passengers.

Fig. 1: Engine mounting Bracket of Car

II. LITERATURE REVIEW

Mallikarjun B Patil et.al [1] conferred their analysis on "Finite part Analysis and Natural Frequency optimization of Engine Mounting Bracket." during this paper the author says that gift automotive market demands low value and lightweight weight part to fulfill the requirement of fuel economical and value effective vehicle. This successively given the increase to simpler use of materials for automotive elements which might scale back the mass of car at an equivalent time enhance the performance. Reducing weight provides multi-disciplinary challenges like strength, stiffness, fatigue life and NVH demand. Noise, Vibration and Harshness are the most parameters that increase the discomfort to vehicle ride and these are originated from the engine. The engine mounting bracket plays the necessary role in reducing NVH and will increase comfort. Because the brackets supports to the engine and these are exposed to the high vibrations, forces and stresses (static in addition as dynamic). Vibrations are often reduced or dissipated and keeping the stresses underneath the planned level of safety are often done by careful style, analysis and choosing material of brackets. During this paper the modal analysis of engine mounting bracket is finished by victimization ANSYS thirteen.0. The look of the bracket is unbroken same however the fabric is modified to carbon fibers from steel. The results were generated by victimization ANSYS R15.0. With the projected new material the burden of the bracket is reduced by approx. seventy fifth and therefore the stresses are within the permissible stress level. M. V. Aditya Nag "Material choice and process Analysis on DOHC V16 Engine's Mounting Bracket victimization COMSOL Multiphysics package." during this paper the analyzers have told that reduction of the engine vibration and therefore the dynamic forces sending from engine to the automotive anatomical structure has continuously been a crucial a part of automotive research. Automobile sector face the task of making a mechanism to soak up these vibrations and supply a sleek ride. The usage of Motor Mounts is that the best answer for moistening the consequences of vibrations and oscillations. This study deals with the fabric choice and finite part analysis for the engine accent part. This main objective of the analysis is to attenuate the burden of the mount bracket. This experiment conjointly deals with overall weight reduction of vehicle's engine mounting bracket victimization differing types of materials. The most parameter thought of for variation within the behavior of the bracket is material variations. For various materials, the stresses are calculated and compared to arrive for the simplest model underneath prescribed conditions. Within the method of simulation, initial the CAD model is formed victimization CATIA V16 tool. The model are going to be foreign to Consul Multiphysics package for performing arts FEA on the model for appropriate loading conditions and constraints. The results obtained are used for analysis of best appropriate material for the engine's mounting bracket victimization the D.F.M.A. methodology.

A. S. Adkine et.al [3] submitted their work on "Static Behavior of Engine Mounting Bracket." during this work an endeavor was created to investigate the engine mounting bracket. Style includes the modeling of the engine mounting brackets by taking under consideration all packaging constraints. Analysis includes Static Analysis of engine mounting bracket. The most purpose of this study is to investigate the natural frequency of by analytically and thru developing the model and self-excitation frequency of engine bracket. An endeavor was created to examine whether or not the natural frequency of engine mounting bracket is a smaller amount than self-excitation frequency of engine bracket. Thus this work is carried by victimization ERW-1, ERW-2, metal and atomic number 12 alloys for the engine mount bracket. The results are analyzed for stresses and deformations.

Umesh S. Ghorpade et.al [4] conferred their work on "Finite part Analysis and Natural Frequency optimization of Engine Bracket." They aforementioned that the automotive engine mounting systems are vital because of totally different aspects of car performance. Early in improvement of the engine mounting system ought to be quickly checked and exactly analyzed, while not sample of a vehicle authorization. Engine bracket has been style as a framework to support engine. Fatigue and vibration of engine bracket has been incessantly a priority which can result in structural failure if the ensuing vibration and stresses are severe and excessive. It's a big study which needs in-depth investigation to grasp the structural characteristics and its dynamic behavior. This paper presents and focuses on some FEA of a typical engine bracket of a automobile are going to be disbursed and natural frequency are going to be determined.

A. S. Adkine conferred the work on "Design and Analysis of Engine Mounting Bracket victimization ANSYS Tool." during this paper the author says that engine mounting bracket plays terribly vital role in reducing vibration, noise and harshness caused because of engine and therefore has terribly effective role in up vehicle comfort. This current article accounts for the investigation of engine mounting bracket by victimization ANSYS. Static and modal analysis of engine mounting bracket was wiped out order to analyze whether or not the present natural frequency of engine mounting bracket is less than that of self-excitation frequency of bracket. It absolutely was found that circular cross section having stress elicited 128.47MPa is additional reliable than sq. cross section. The results were analyzed for stresses and deformations. The look was tested for various materials like ERW-1, atomic number 12 and ERW-3 at the side of suitableness of fabric. Stresses elicited in atomic number 12 bracket were sixty four.07 MPa with the deformation of one.20 mm. It are often assumed that atomic number 12 brackets are corrosion resistant and may be thought of for desired application. Stiffness of ERW-3 material was found higher than atomic number 12 and may be used for needed application.

N. S. Vele, N. K. Kamble, Dr. Et.al [7] given the work on "Finite component Analysis of Engine Mounting Bracket." The researchers say that engine is one in all the foremost necessary elements of a road vehicle like automotive. High performance sports automotive has their engine supported by bracket. It plays a crucial role in rising the comfort & work setting of an automotive. The advance of engine bracket system has been the topic of intense interest for several years. It's necessary to style correct engine bracket for an automotive. As such, the engine mounting bracket has been designed as a framework to support engine. Fatigue of engine bracket has been unceasingly a priority which can result in structural failure if the ensuing stresses area unit severe and excessive. Excessive exposure to whole-body stresses within the operating setting could result in fatigue and in some cases it damages the automotive. Optimizing the mounts system in early stages of train style is feasible by computer-aided engineering tools. CAE results are often analyzed with none pricey prototypes. The results thought-about to outline strategy for the vehicle mount system and optimize the locations and therefore the rates of mounts.

Monali Deshmukh et.al [6] submitted analysis paper on "Analysis and improvement of Engine Mounting Bracket." during this work the authors same that the engine mounting plays a crucial role in

reducing the noise, harshness and vibrations for rising vehicle ride comfort. The brackets on the frame that support the engine endure high static and dynamic stresses likewise as large quantity of vibrations. Thus, dissipating the vibrations and keeping the stresses underneath a pre-determined level of safety ought to be achieved by careful planning and analysis of the mount brackets. Keeping this in mind during this paper, Static Analysis, modal analysis of engine mounting is finished likewise as harmonic response analysis of bracket as a section of dynamic analysis is performed with the FEA computer code package ANSYS fifteen.0.The existing model is optimized and a completely unique model was projected to cut back the load of the rib of the engine mounting bracket likewise because the harmonic response in term of acceleration is checked to make sure that the projected model won't end
in to strident operation. The results of the stresses, deformations and harmonic response for the models of the engine mounting bracket were compared to every alternative. With the projected model of the engine mounting bracket twelve.5%weight reduction is achieved maintaining an appropriate level of yield stress and harmonic response. Sameer U. Kolte et.al[8] submitted analysis on "Structural sturdiness Analysis of train Mounting Bracket." the students say that structural analysis is performed to examine sturdiness of nominal half for a given load and support conditions. For the element to be safe, in any domain, the stresses generated shouldn't exceed the yield strength of the fabric. However, considering chance of fatigue life failure, the element is optimized such stresses generated don't exceed the endurance strength of the fabric. The engine mounting bracket is subjected to hundreds primarily thanks to weight of the facility train, the unbalanced force. During this analysis, structural mechanics library of Comsol is employed. The CAD model is foreign from CATIA®V5R21. Polyhedron is chosen because the meshing component. The fabric is chosen from 'built in' materials in Comsol-library. The bracket is subjected to totally different loading conditions like 3-2-1g weights, 3g downward unbalanced force etc. and is optimized so stresses generated and deflection of the bracket is among permissible limit. Result of bolt preloads is additionally thought-about. The results of the analysis area unit compared with hyper mesh results and area unit found to be in best correlation.

III. OBJECTIVES OF THE PROJECT

1. To design and model of engine mounting bracket for TATA ACE using 3 D modeling tool.
2. To perform static, dynamic and modal analysis on the designed bracket.
3. To optimize the design of the mounting bracket for weight reduction.
4. To validate the software results using FFT analyzer testing of component.

IV. METHODOLOGY OF THE PROPOSED WORK

1. To find out-design specification for engine mounting bracket.
2. Designing the engine mounting bracket, and modeling in modeling software.
3. Perform static, modal and forced vibration analysis on the bracket to verify the design and for finding out the failure modes.
4. Iterative design study to improve failure modes and natural frequencies of the bracket using different design parameters.
5. To select the best suited design from FEA results for the optimization of vibration purpose.
6. To manufacture the optimized bracket as per modified parameters.

Conducting FFT analyzer test on the bracket model and comparing the results with FEA.

V. PROBLEM STATEMENT

Most of the engine mounting bracket for the four wheeler engines is designed with Mild Steel material and with conventional standard shape and dimensions. In this study focus will be on designing the alternate Engine mounting bracket for the TATA ACE engine using finite element analysis technique and weight reduction in the current design needs to be achieved.

VI. THEORETICAL ANALYSIS

Table No.1.0 Specification

Vehicle selected for study	TATA Ace

Engine Type	Four Strokes, Two Cylinders, 702 cc
Maximum Power	16 HP (12 KWatt) at 3200 rpm
Vehicle Kerb Weight	885 Kg
Vehicle Gross Weight	1550 Kg
Engine Weight	151Kg + 49Kg (Part of transmission weight)
Total Design Weight	200 Kg (2000 N Approx.)

Bracket Design for thickness –
Material selection for bracket –
Optimum Material Selection by Johnson Method:
Optimum design of mechanical element is the selection of material and values for independent geometrical parameters with explicit objective of either minimizing most significant undesirable effect or maximizing a most significant functional requirement, making in the design procedure that the mechanical element satisfies other functional requirements and that other undesirable effects are kept within tolerable limits.

Primary Design Equation (P.D.E):
Weight (W) per unit length = weight density × volume of plate
W= w×(b×l×t) ……………………(1)
Subsidiary design equation (S.D.E):
Bending stress = F× L × Yb12t3……………… (2)
Put Y = t2
Bending stress = 6 × F × L b × t2
t2 = 6 × F × L b ×σ

t =√6 × F × L b ×σ…………………….....(3)
Limit equation
Σb≤ SbNf …………………..... (4)
Combining S.D.E with P.D.E we get,
W= w × b × l ×√6 ×F ×L b ×σ ……………..... (5)
Therefore,
W α 1√σ
So weight will be minimum if bending stress will be maximum i.e. σb will be in its upper limit.
i.e
Σb= SbNf = 0.6 SyNf
By putting σb= 0.6 SyNfin eq. (5)
W= 3.162 √F × L × b ×$w\sqrt{Sy}$
This is the desired equation:

By considering,
3.162 √F × L × b term constant say K, we get,
W= K w√Sy

Table 2.0 Different material properties

Sr. no.	Material	Weight Density W	Yield Strength S_y	w/√Sy
1	MS (AISI 1018)	77204.7	370	4013.681 23
2	SS 310	78480	205	5481.280 96
3	SS 309	78480	310	4457.364 48
4	Tool steel	22106.6	600	3147.86

From above equation it is possible to evaluate weight density of various steels which directly affects the weight of material. Among above tabulated results, weight factor ($w\sqrt{Sy}$) for tool steel is lower than any other type of steel so it is justified that alloy steel is optimal material for bracket. But if we consider cost of the tool steel and manufacturing processes are very hard we will select the next best option which is MS AISI 1018 as the material for the bracket.

Mechanical Properties for material –MS AISI 1018 Material
Syt – 370 MPa
Sut – 440 MPa
Elongation – 15%
Modulus of Elasticity (E) – 205 GPa
Poisson's Ratio (μ) – 0.29
Considering bending formula,
$MI=\sigma y$
Now, calculating bending moment

$Mb=P\times l$
=70000 Nmm
Moment of inertia of minimum area at two hole section,
I =$Iplate$− 2($Ihole$)
=$w\times t$312−2 ($d\times t$312)
I=3.67t^3
Position of Neutral Axis, $y=t^2$
Now putting value in bending equation,
$MI=\sigma by$
∴t2=27.67
∴t=5.26 mm
Selecting next grade of thickness as 6 mm
Checking bracket for shear stress –

Minimum area will be at maximum stress and it is at the hole section, $AMin$ = (60×6) − 2(8×6)
=264 $mm2$
Calculating shear stress induced in plate, $\tau = P\ AMin$
=7.58 N/mm^2

Now calculating allowable shear stress for bracket material, τall=0.5 Syt
=0.5× 370
=175 N/mm^2
$\tau \leq \tau all$

VII. FINITE ELEMENT ANALYSIS

The finite component technique may be a numerical technique for gaining Associate in Nursing approximate Associate in Nursingswer to the matter by representing the article by an assembly of rods, plates, blocks, bricks. Finite component analysis may be a technique of determination, sometimes or so, sure issues in engineering and science. It's used chiefly for issues that no actual a nswer, utter able in some mathematical kind, is obtainable. As such, it's a numerical instead of Associate in nursing analytical technique. Strategies of this sort are required as a result of analytical strategies cannot address the important, sophisticated issues that are met with in engineering. for instance, engineering strength of materials or the mathematical theory of physical property are often wont to calculate analytically the stresses and strains during a bent beam, however neither are terribly winning find o ut what's happening partly of an automotive suspension throughout cornering. One amongst the primary applications of FEA was, indeed, to seek out the stresses and strains in engineering parts beneath load. FEA, once applied to associate in nursing realistic model of an engineering part, needs a vast quantity of computation and also the development of the tactic has relied on the supply of appropriate digital computers for it to run on. the tactic is currently applied to issues involving a good vary of phenomena, together with vibrations, heat conductivity, mechanics and physics, and a good vary of fabric properties, like linear-elastic (Hookean) behavior and behavior involving deviation from law (for example, physical property or rubber-elasticity). Model of the mounting bracket for engine is formed exploitation modelling code CATIA as shown within the image below.

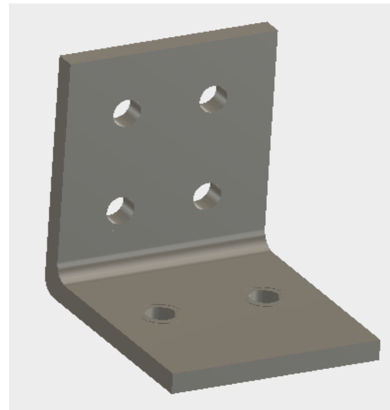

Fig. 1.1: CAD Model for Engine mounting bracket

Model shown in the figure above is imported in the Autodesk Fusion 360 software. Bracket is modelled as a solid. Standard size of 1 mm is used for the fine meshing and good results of FEA analysis. Remote force of 1000 N is applied remotely to the mounting location of the engine.

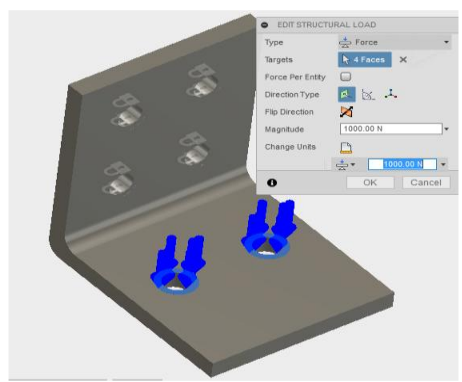

Fig. 1.2: Boundary Conditions of the Static Analysis

Fixed holes are highlighted with locks and blue color highlights the loading attachment area in the above figure. Total of 1000 N force applied remotely to the bracket. Below are given the results of the analysis performed on the bracket.

Von mises stress plot shows the maximum stress of 221.5 MPa which is due to stress concentration at the fixing boundary condition. We can see in the stress plot that maximum stress in the area is not crossing 221.5 MPa limit as there is no significant area above or in the green color band of contour which represent stresses higher than 221.5 MPa.

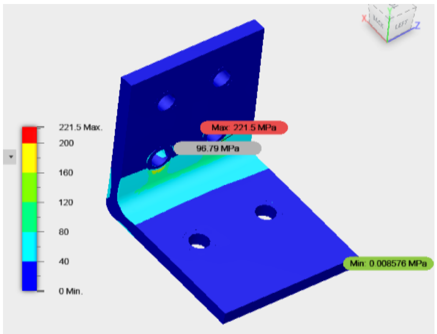

Fig. 1.3: von Mises stress plot

This plot shows the stress distribution of high stress is really local and average stress near the maximum stress region cross section is 96.79 MPa which means design is safe and it can be iterated to save some material by redesigning it.

FEA Results Modal Analysis

After static analysis on the bracket modal analysis is performed on the bracket to find out mode shapes and modal frequencies of the bracket at the given boundary conditions of fixing. First mode shape is plotted and list of first 6 natural frequencies is given in the table.

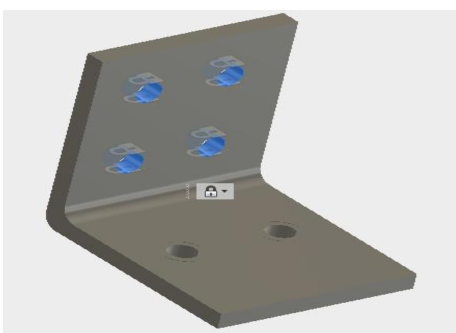

Fig. 1.4: Boundary condition for the modal analysis

Baseline model's first mode shape plot is shown in the image below.

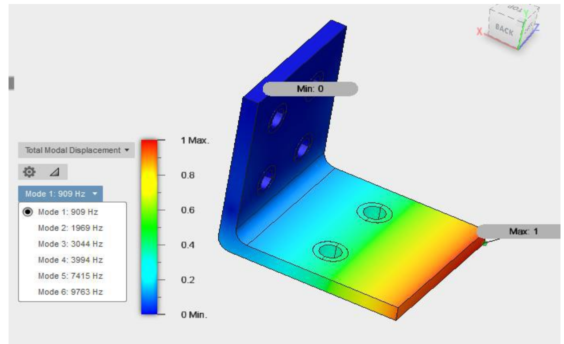

Fig: 1.5: Mode 1 @ baseline model 909 Hz

Mode shape shows that mode is due to loading in the vertical direction. Engine has maximum speed of 6000 RPM which will say that engine's second order frequency will not cross value of 200 RPS that is 200 Hz. Modal frequency of the bracket is much higher for getting affected by engine load variations. Which shows us that there is scope of removing material and still won't affect vibrational life of the engine mounting bracket. Table below shows the list of first 6 modal frequencies of the engine mounting bracket baseline module.

Table: 3.0- First 6 natural frequencies table baseline

Mode No.	Natural Frequency (Hz) Steel bracket
1	909
2	1969
3	3044
4	3994
5	7415
6	9763

Now we will use the optimization technique for finding the possibility of weight optimization. The model and meshing of the model is similar in the optimization module. Only one thing added is the optimization settings.

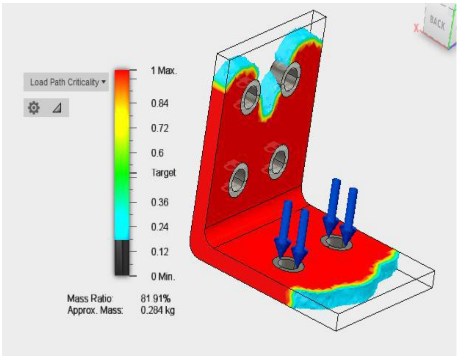

Fig. 1.6: The optimized model and the load criticality path

The above figure shows the load criticality path and also the areas from where the material can be removed. Now we perform iterations to remove material from the basic current model and solve the iteration for the load and boundary conditions.
Iteration 1: Making a 25 mm hole on the horizontal component of the bracket. The sketch used to make the hole is as shown below.

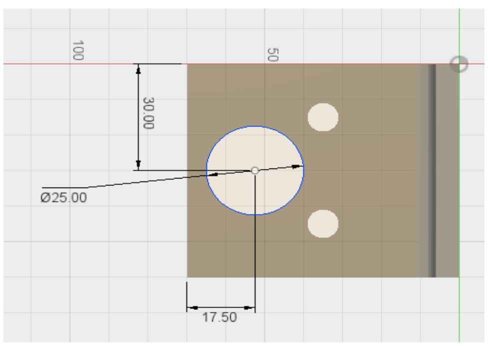

Fig. 1.7: Sketch for iteration 1, 25 mm hole

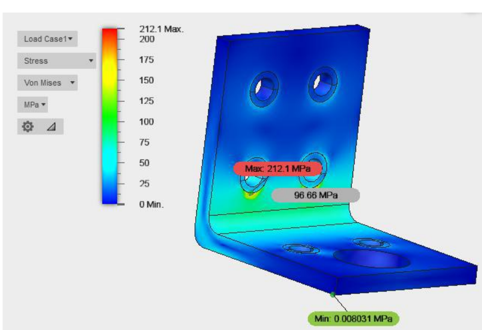

Fig. 1.8: Von Mises stress plot for iteration 1

Iteration 2:
Making a cut in the horizontal component of the bracket as shown in the below sketch.

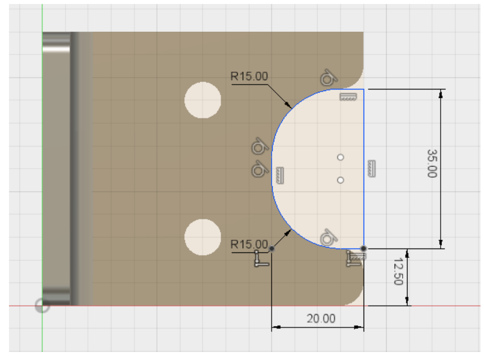

Fig. 1.9: Sketch for iteration 2

In the above image, the maximum stress value is shown as 212.1 Mpa. This value is due to the stress concentration in the region. If we see the value of stress in the adjacent element, it is too low i.e. 96.66 Mpa. Therefore the design is safe. Now we try for another iteration with more material removal.
Now we cut the material from the basic model based on the above shown sketch. So the model looks like the image shown below.

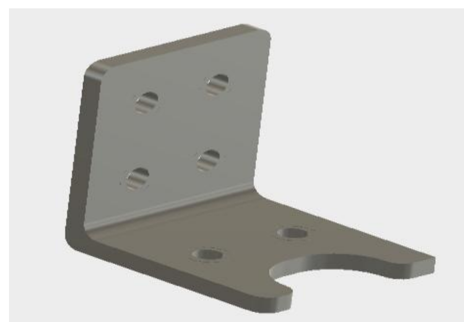

Fig. 2.0: Model for iteration 2

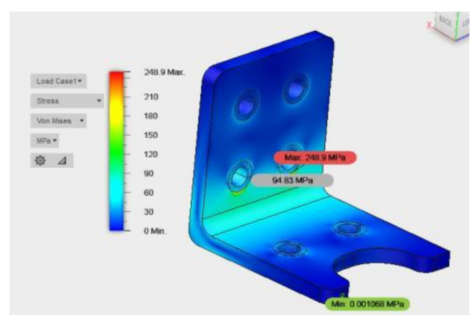

Fig. 2.1: Von Mises stress plot for iteration 2

Iteration 3:

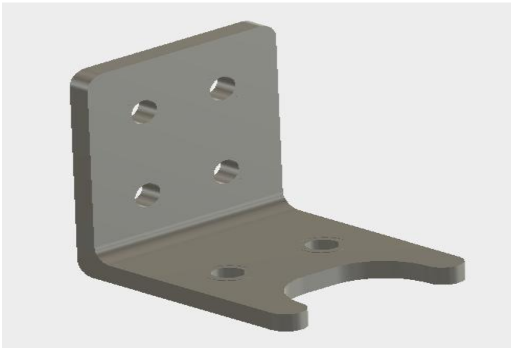

Fig. 2.2: Model for iteration 3

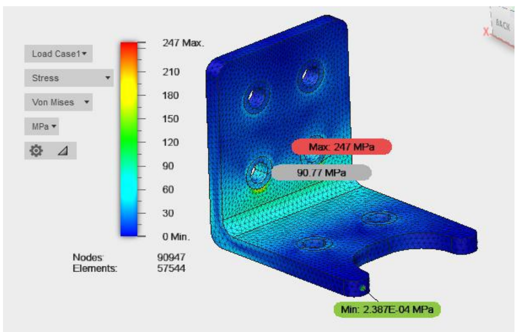

Fig. 2.3: Von Mises Stress Plot for iteration 3

In the above image, the maximum stress value is shown as 247 Mpa. This value is due to the stress concentration in the region. If we see the value of stress in the adjacent element, it is too low i.e. 90.77 Mpa. Therefore the design is safe. Now we try for another iteration with more material removal.

Iteration 4:

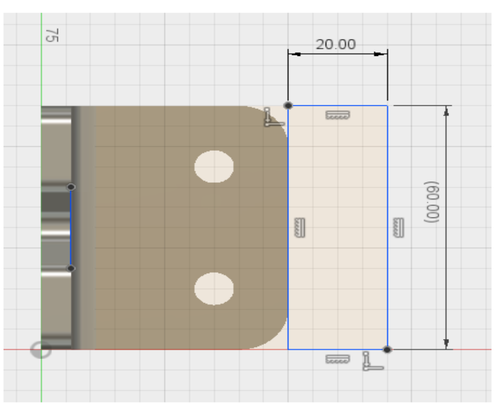

Fig. 2.4: Sketch for removing material from horizontal component

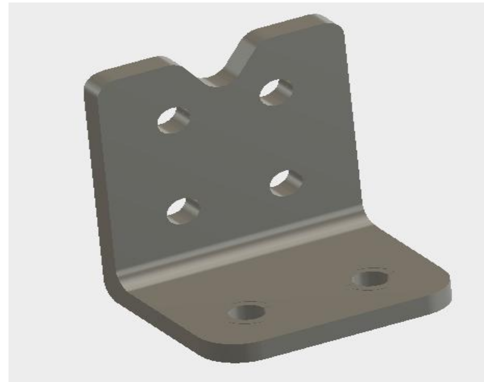

Fig. 2.5: Model for iteration 4

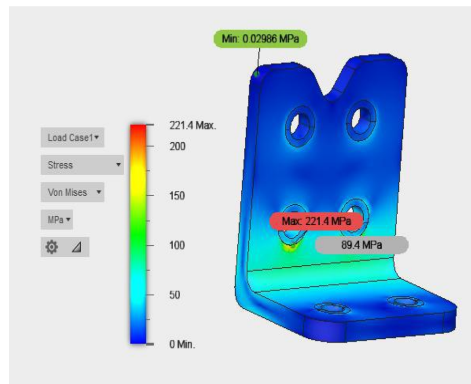

Fig. 2.6: Von Mises Stress plot for iteration 4

In the above image, the maximum stress value is shown as 221.4 Mpa. This value is due to the stress concentration in the region. If we see the value of stress in the adjacent element, it is too low i.e. 89.4 Mpa. Therefore the design is safe. Now we try for another iteration with more material removal.

Modal Analysis for iteration 4

Fig. 2.7: Boundary conditions for modal analysis for iteration 4

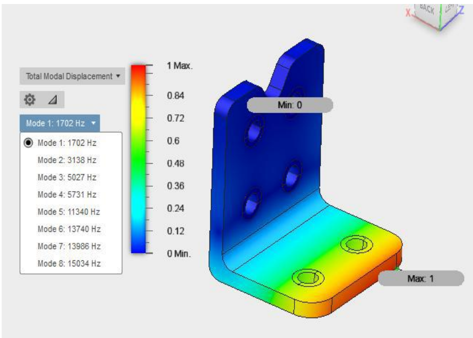

Fig. 2.8: Modal Analysis plot for 1st mode @1702 Hz

Mode shape shows that mode is due to loading in the vertical direction. Engine has maximum speed of 6000 RPM which will say that engine's second order frequency will not cross value of 200 RPS that is 200 Hz. Modal frequency of the bracket is much higher for getting affected by engine load variations. Which shows us that there is scope of removing material and still won't affect vibrational life of the engine mounting bracket. Table below shows the list of first 6 modal frequencies of the engine mounting bracket baseline module.

Table: 3.0- First 6 natural frequencies table for iteration 4

Mode No.	Natural Frequency (Hz) Steel bracket
1	1702
2	3138
3	5027
4	5731
5	11340
6	13740

The design for iteration 4 is safe in static as well as modal analysis and therefore we can go for manufacturing as per the changes suggested in the iteration 4.

VIII. MANUFACTURING

For the manufacturing of engine mounting bracket mild steel material is used. The manufacturing is done on the laser cutting machine for that 6 mm thickness sheet is used and from the last iteration develop length of bracket is calculated. From develop length drafting is done for laser cutting drawing, after laser cutting operation bending operation is done.

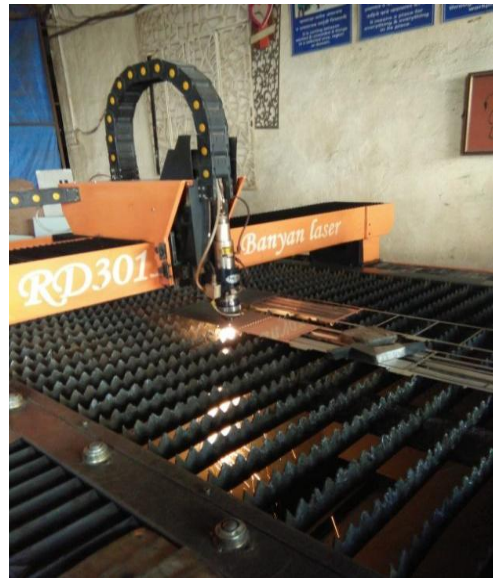

Fig. 2.7: Laser Cutting

Fig. 2.8: Bending Operation

IX. EXPERIMENTAL VALIDATION

Fig. 3.0: compression loading

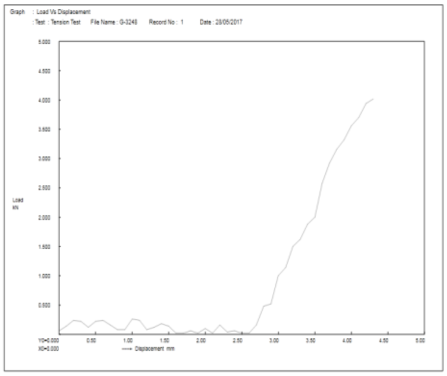

Fig. 3.1: Load vs. Displacement

X. RESULT

From the above iterations done on the Finite Element Analysis, we have got the following results for the weight reduction of the engine mounting bracket.

Iteration	Mass (gms)	% Reduction in mass
Basic model	364	
Iteration 1	341	6%
Iteration 2	335	8%
Iteration 3	330	9%
Iteration 4	299	18%

The maximum weight reduction is obtained in the fourth iteration i.e. 18%.

XI. CONCLUSION

Weight of the optimized bracket is observed as 299 grams which was 364 grams in the basic part. So total of 65 gram mass is reduced, 18% weight is reduced from the total weight of the bracket without affecting the load carrying capacity. Therefore we have manufactured the engine mounting bracket as per the dimensions of the fourth iteration and successfully tested it on Universal Testing Machine.

XII. REFERENCES

1. Mallikarjun B Patil, Prof. S. S Kelkar, “Finite Element Analysis and Natural Frequency Optimization of Engine Mounting Bracket”, International Engineering Research Journal Page No 981-989.
2. M.V. Aditya Nag, “Material Selection and Computational Analysis on DOHC V16 Engine’s Mounting Bracket Using COMSOL Multiphysics Software”, COMSOL Conference Bangalore, 2012.
3. Static Behaviors of Engine Mounting Bracket, A. S. Adkine, V. S. Kathavate, G. P. Overikar, S. N. Doijode, International Advanced Research Journal in Science, Engineering and Technology Vol. 2, Issue 4, April 2015.
4. Finite Element Analysis And Natural Frequency Optimization Of Engine Bracket, Umesh S. Ghorpade, D. S. Chavan, Vinay Patil & Mahendra Gaikwad, International Journal of Mechanical and Industrial Engineering (IJMIE) ISSN No. 2231 –6477, Vol-2, Iss-3, 2012.
5. A. S. Adkine, Design and Analysis of Engine Mounting Bracket using ANSYS Tool, DOI:10.13140/RG.2.1.4946.4406, December 2015.
6. Monali Deshmukh, Prof. K R Sontakke, Analysis and Optimization of Engine Mounting Bracket, International Journal of Scientific Engineering and Research (IJSER), 2347-3878, Impact Factor (2014): 3.05.
7. N. S. Vele, N. K. Kamble, Dr. S. V. Joshi Finite Element Analysis of Engine Mount Bracket, International Journal of Research In Advance Technology(IJRAT), 2013-14.
8. Sameer U. Kolte, David Neihguk, and Abhinav Prasad, Structural Durability Analysis of Powertrain Mounting Bracket, October 2013 DOI:10.13140/RG.2.1.1414.6722
9. P.D. Jadhav, Ramakrishna, “Finite Element Analysis of Engine Mount Bracket”, International Journal of Advancement in Engineering Technology, Management And Applied Science, Volume 1, Issue 1 Sept 2014, Issn: 2349-3224
10. Koushik . S, “Static and Vibration Analysis of Engine Mounting Bracket of TMX 20-2 using OptiStruct”, Altair Technology Conference, 2013 India
11. Yunhe Yu, Nagi G Naganathan, Rao V Dukkipati, A literature review of automotive vehicle engine mounting systems, Mechanism and Machine Theory, Volume 36, Issue 1, 1 January 2001, Pages 123-142, ISSN 0094-114X, 1016/S0094-114X(00)00023-9.
12. V.Ratna Kiran, M.V. Aditya Nag, “Design optimization for engine mount bracket for V6 engine”, Proceedings of 2nd International Conference on Advances in Mechanical and Building Sciences, V.I.T.U., India, pp no 814819.