Optimization of Engine Mounting Bracket by Topology Technique for Reduction in Weight

[1]Mr. Shridhar R. Kothawale, [2]Prof. Sanjay A. Pawar

[1]M.Tech. Student, [2]Head of Department
FTC College of Engineering & Research,
Sangola (Maharashtra).

***Abstract*: Engine mounts themselves are small elements that are meant to stabilize, further as properly align, a vehicle's engine. While these mounts are small, they play a big role in the overall functionality of the heart of your vehicle. Moreover, once these allegedly little and minor aspects of the vehicle go dangerous. An automotive engine-body-chassis system is often subjected to unbalanced engine forces, uneven firing forces particularly at the loafing speeds, shaking forces and torques thanks to reciprocator elements, dynamic excitations from gearboxes and accessories, and road excitation. These tendencies produce to unsought vibrations that result in uncomfortable ride and additionally cause further stresses within the automobile frame and body. Vibrations are annoying and their origins are often tough to sight. An engine mounting system includes a front mount, a rear mount, an engine mount, and a transmission mount.**

So, the main aim of our project is to optimize the engine mounting bracket in order that the vibrations are reduced. The 3D model was drawn with the assistance of CATIA V5 software package. The analysis was carried out with the assistance of ANSYS 19 software package. The experimental testing was done at that time the comparative study was done between the Analytical & experimental results and at that time the result & conclusion was drawn.

***Index Terms*: Mounting Bracket, Analytical, CATIA**

I. INTRODUCTION

In this automotive and competitive era the requirement for light-weight weight structural materials is increasing as there is additional concentrate on fuel consumption reduction and improvement in decreasing the emission. The magnitude of production volumes has historically placed severe necessities on the robustness of method employed in the manufacturing. The manufacturers have strong importance on the cost has the demand for the component to enhance the material performance and to deliver these materials at low cost is the requirement.
In automobile sector the extraordinarily competitive automotive business wants manufactures to pay plenty of attention to traveling comfort. Resonant vibration is from unbalanced plenty exist among the engine body, this is often inflicting the designers to direct their attention to the event of fine quality engine mounting brackets therefore to ensure that there is advancement in riding comfort. The demand for higher performing engine mount brackets should not be offset by arise among the assembly costs and development cycle time.
In diesel engine, the engine mounting bracket is that the major downside as there is unthrottled condition and better compression magnitude relation and even there are additional speed irregularities at low speed and low load compared to gasoline engines. Therefore because of this there is additional vibration excitation. By this vibration engine process mount bracket might fail, therefore by optimizing the shape and thickness of engine mount bracket the performance may be improved at initial design stages By some studies it is discovered that brackets saved up to 38% of mass. Structural improvement is important tool for an optimum design; comparison in terms of weight and component performance structural optimization techniques is effective tool to supply higher quality products at lower cost.
Another part holds the engine. An engine is a source of vibration, as it has many moving and rotating parts. The job of an engine mount is not only to hold the engine in place, but to reduce the engine vibration felt inside the car.

II. LITERATURE REVIEW

P.D. Jadhav et. al[1] studied the optimization of engine mount bracket by using different metal bracket with analysis. The analysis of the engine mounting bracket is done with the help of FEA software. This work is contribution to the develop of new material for engine mounting bracket .Which is safe for the required application with the main advantage of light weight.
Monali Deshmukhet et. al[2] the engine mounting plays an important role in reducing the noise and vibration for improving vehicle comfort. Many methods are available for the optimization of mounting bracket .When we optimize the mounting bracket then easy to increasing vehicle comfort. In this work the harmonic response analysis is used for optimization of the engine mounting bracket. Also it is confirmed by harmonic analysis that the harmonic response of modified bracket is within safe limit.
Dr. Yadavalli Basavaraj et. al[3] observed in automotive business industry requires manufacturers to more attention to passenger comfort and riding quality. Comfort is increased by reducing the controlling engine vibration. Hence the design of the engine mount become critical in terms of vehicle crashworthiness. In this work they studied on the engine mount system by six different models. It is created using CAD and Hypermesh and simulation of the engine mount is carried out using LS-Dyna.In that the analysis results which are obtained from six different model obtained from LS-Dyna approach follows experimental test curve. The result included that the rubber used in the engine mount had increased the frequency.

Joong Kim et. al[4] observed proper design of rubber mount may be the most effective engineering approach to improve the ride. A bush type engine mount has type engine mount has been designed using a parameter optimization method. A bush type engine mount being used in a passenger car is chosen for an application model. The shape from the result of the parameter optimization is determined as a final model with some modifications.

Diwakar k. et. al[5] studied design of vehicle structures, it is always challenging to achieve higher stiffness and strength and simultaneously reduce weight. To reduce the weight of engine mounting bracket by two technique which is topology technique and material optimization. Final comparison between the existing model and optimized model in terms of stress, weight and the natural frequencies of the component.

A Prasad Babu et.al[6] topology optimization is often used in the early design process to define the optimum part layout. The Engine mounting bracket has been used to for mounting support as well as reduction of the vibration created by the engine. The engine mounting bracket is made up of different materials Aluminum alloy, Aluminum Silicon Carbide and Grey Cast Iron. In this paper the weight reduction engine mounting bracket is taken under the consideration without varying the performance of the component.

Vijay Kalantre et. al[7] studied structural optimization is one application of optimization. Here the purpose is to find the optimal material distribution according to some given demands of a structure. The common functions to minimize the mass. Topology is scientific tool that provides guideline about removing of inefficient material from the structure.

Mahendra Giakwad et. al[8] explained design engine mount bracket of a automotive and centered on to see natural frequencies of car engine mount bracket. they had thought of the three materials for engine mount bracket that's atomic number Al alloy, Mg alloy, grey cast iron whose modal analysis is carried out, it is found natural frequencies of grey cast iron is low which can prove additional hindrance in vibration of engine mount bracket so that they have eliminated gray cast iron, in terms of analysis of Al alloy and Mg alloy are showing virtually near value of natural frequency in practical terms as magnesium alloy has higher strength that is low stress value, thus ideally Mg alloy is chosen as better material by study.

Mr. Pramod Walunje et. al[9] during this work they need principally targeted on the utilization of weight material for bracket and conjointly to reduce the weight of the bracket. Here the weight of the material is reduced and pre-processing and post-processing is administrated and even with this an experimental setup is additionally used to find the strain level of the materials are ascertained that aluminium alloy have practical natural frequency and stresses are at intervals the yield strength, thus by considering the aluminum and reducing its thickness by 2mm than original component, they found that currently von misses stresses also are within yield stress so that they have achieved reduction within the mass of bracket up to 0.43kg in comparison to previous one.

Sandeep Maski et. al[10] observed vibration and strength plays a vital role within the design of engine mount bracket, thus during this paper special attention has been given for choice of appropriate material for engine mount bracket in order that it will withstand high strength and vibrations. It is seen from the obtained results and discussion that for modal analysis first fundamental natural frequency of mild steel is high which is 65Hz compared with cast iron and iron and just in case of static analysis displacement of mild steel is 1.6mm that is a smaller amount compared to different two materials and therefore the maximum von-misses stress coming on mild steel is lesser than yield point, thus mild steel is taken into account as better material to design engine mount bracket.

III. PROBLEM STATEMENT:

In this automotive era the requirement for light weight structural materials is increasing as there is a more focus on fuel consumption reduction and improvement in decreasing the emission. Currently, bracket contains excess material, leads to increase in weight of the vehicle. Directly affects the mileage and cost. In this modeling of present bracket in CAD software and analyzing it for induced stresses and deformation in CAE software will be done. Then using topology optimization material will be removed. Again, analysis will be done on optimized model for stresses and deformation. It is also tested experimentally and results were correlated it with analysis results.

IV. OBJECTIVES:

- Modeling original Engine mounting bracket.
- Analyzing for stresses and deformation of original Engine mounting bracket.
- Topological optimization for the Engine mounting bracket
- Analyzing for stresses and deformation.
- Experimental testing and correlating results.

V. SCOPE:

Finite Element method along with experimental techniques is used. The weight is most important criteria while considering the engine mounting bracket, so reduce the weight of engine mounting bracket is necessary. For that topology optimization method used for weight reduction.

VI. METHODOLOGY:

CAD modeling of existing engine bracket using CATIA

↓

Finite Element Discretization using ANSYS

↓

Bending load calculations for applying boundary conditions.

↓

FEA for mount to obtain stress & strain.

↓

Using ANSYS solver to achieve mass optimization.

↓

Experimental stress analysis of existing mount using Strain gauging & applying Bending loading through UTM.

↓

Comparative analysis between FEA & Experimental results

↓

Conclusion & Future scope

VII. EXPERIMENTAL TESTING AND ANALYSIS

CATIA MODEL

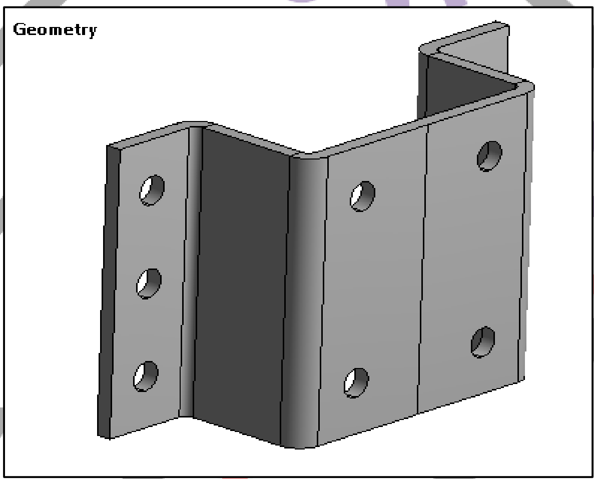


Fig. 1 Catia model

Topology Optimization of Engine Bracket

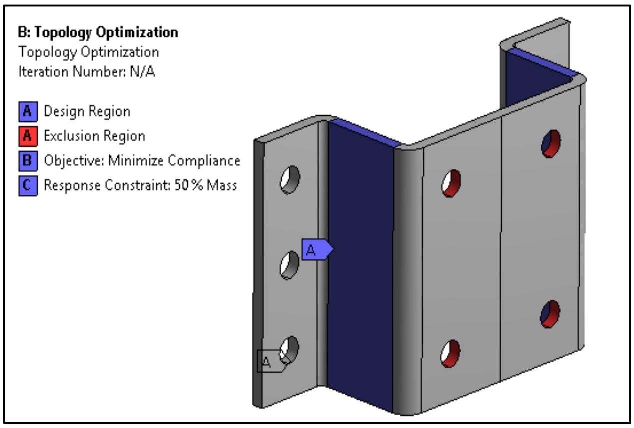


Fig. 2 Topology optimization

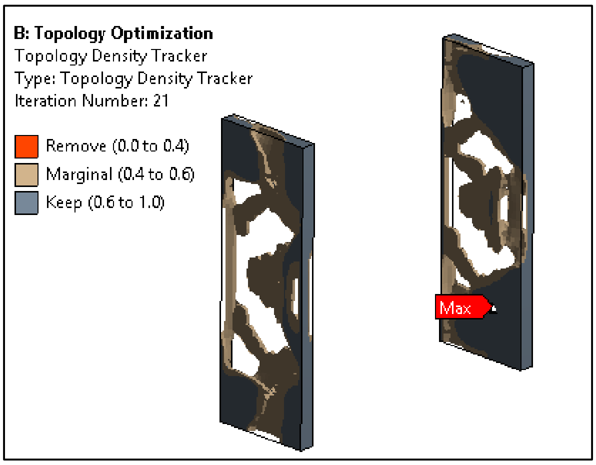


Fig. 3 Topology density tracker

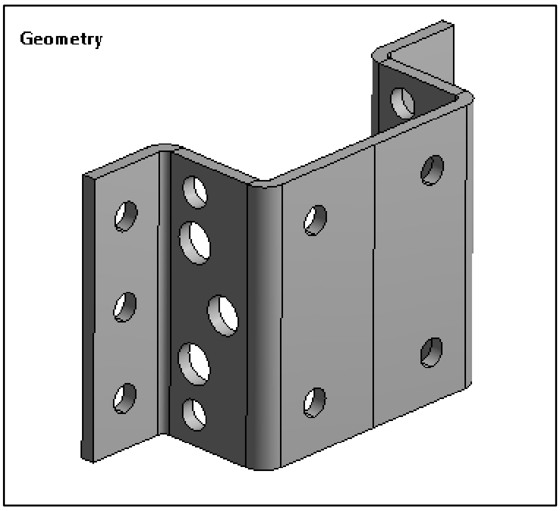


Fig. 4 Optimized engine bracket

Mesh component

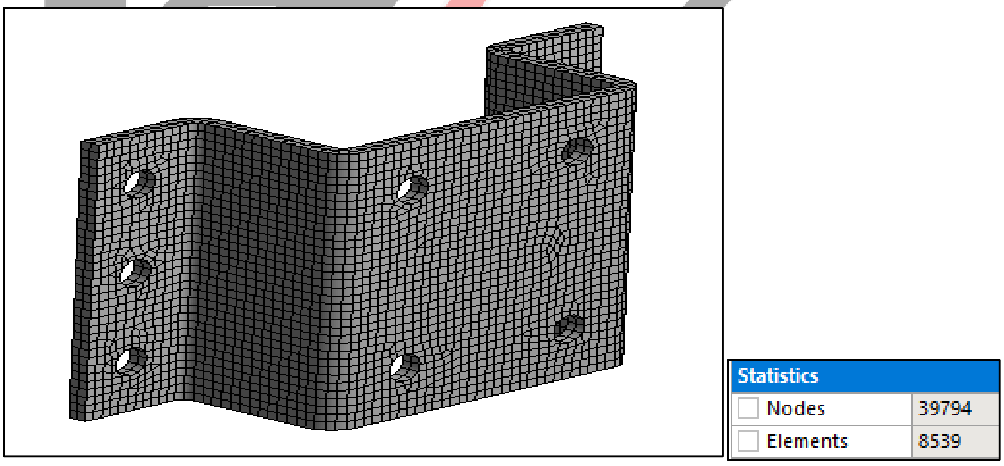
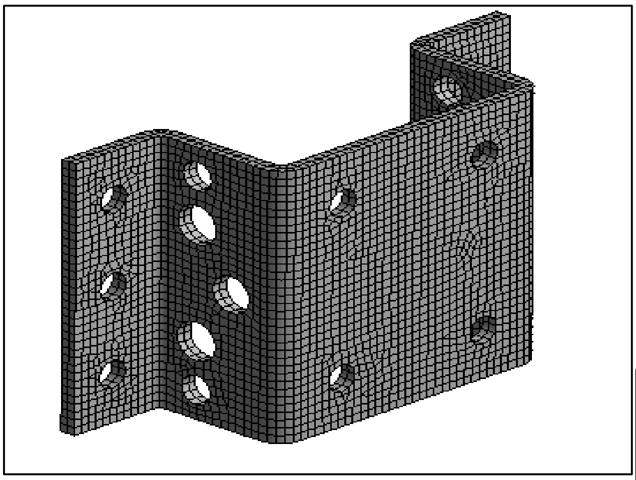


Fig. 5 Meshing of original model



Statistics	
Nodes	39794
Elements	8539

Fig. 6 Meshing of optimized model

Boundary Condition

A boundary condition for the model is that the setting of a well-known value for a displacement or an associated load. For a specific node you'll be able to set either the load or the displacement but not each.

The main kinds of loading obtainable in FEA include force, pressure and temperature. These may be applied to points, surfaces, edges, nodes and components or remotely offset from a feature.

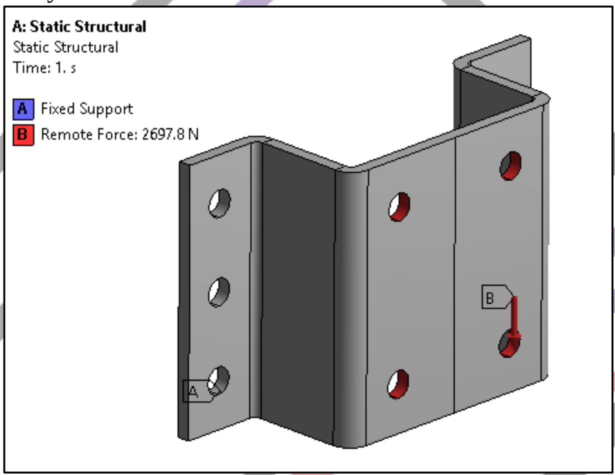


Fig. 7 Boundary condition of original model

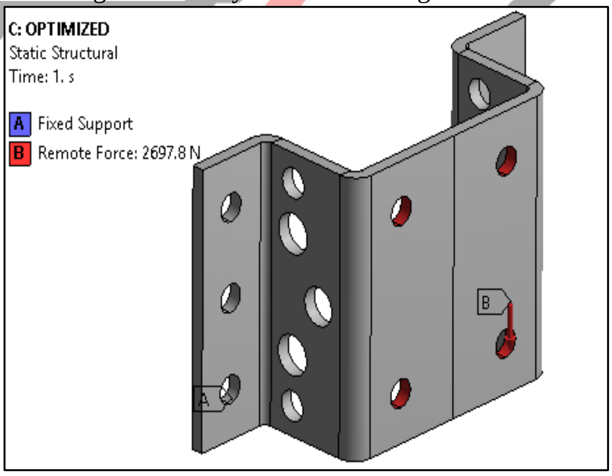


Fig. 8 Boundary condition of optimized model

Total Deformation

In finite element method the total deformation and directional deformation are general terms irrespective of software being used. Directional deformation may be place because the displacement of the system in a very particular axis or user defined direction. Total deformation is that the vector sum of all directional displacements of the systems.

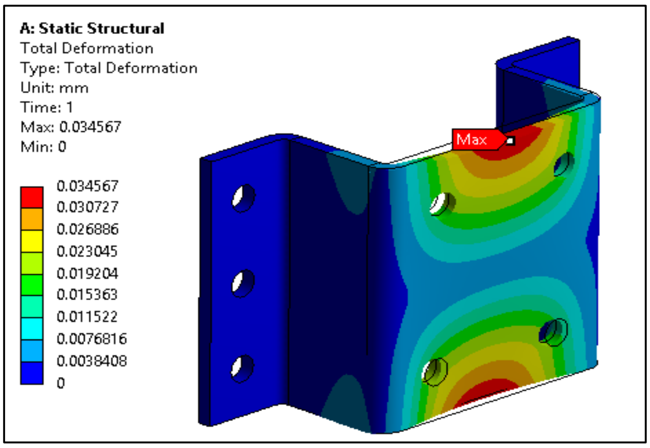


Fig. 9 Total deformation of original model

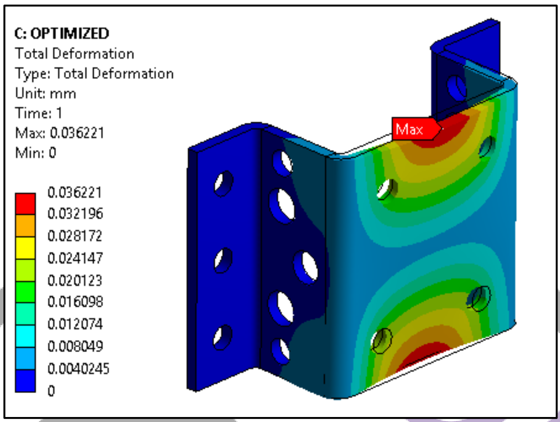
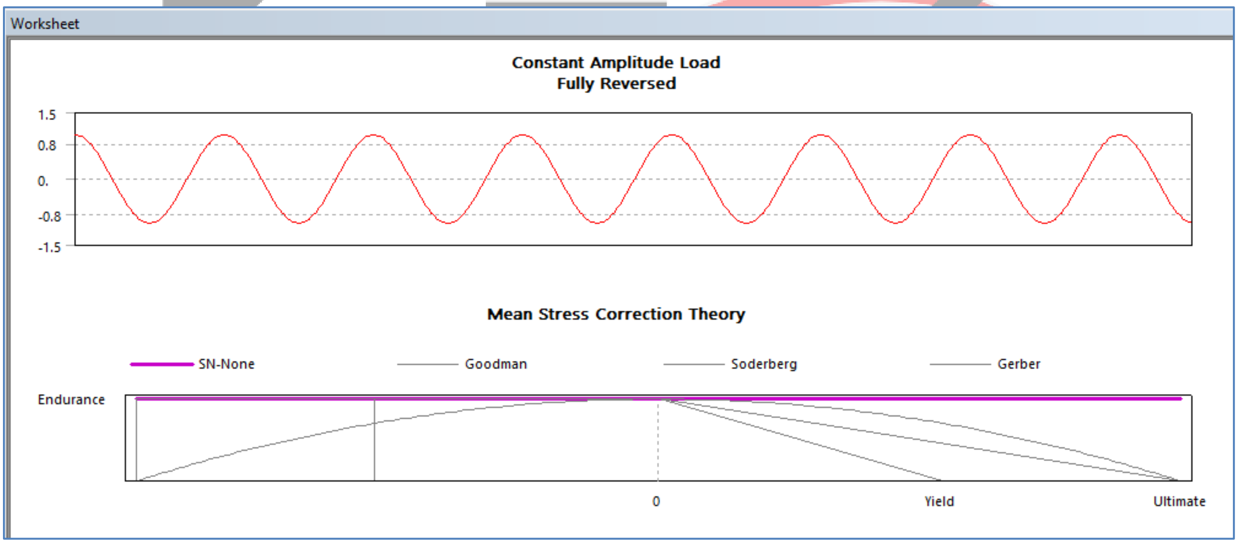


Fig. 10 Total deformation of optimized model

FATIGUE LIFE:



Details of "Fatigue Tool"	
Domain	
Domain Type	Time
Materials	
Fatigue Strength Factor (Kf)	1.
Loading	
Type	Fully Reversed
Scale Factor	1.
Definition	
Display Time	End Time
Options	
Analysis Type	Stress Life
Mean Stress Theory	None
Stress Component	Equivalent (von-Mises)
Results Input (Beta)	Stress

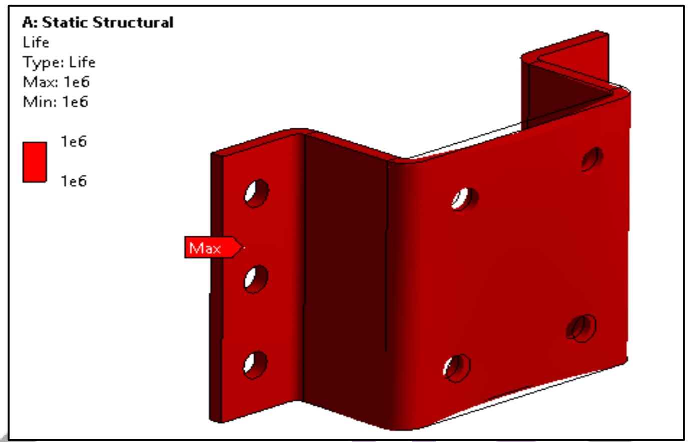
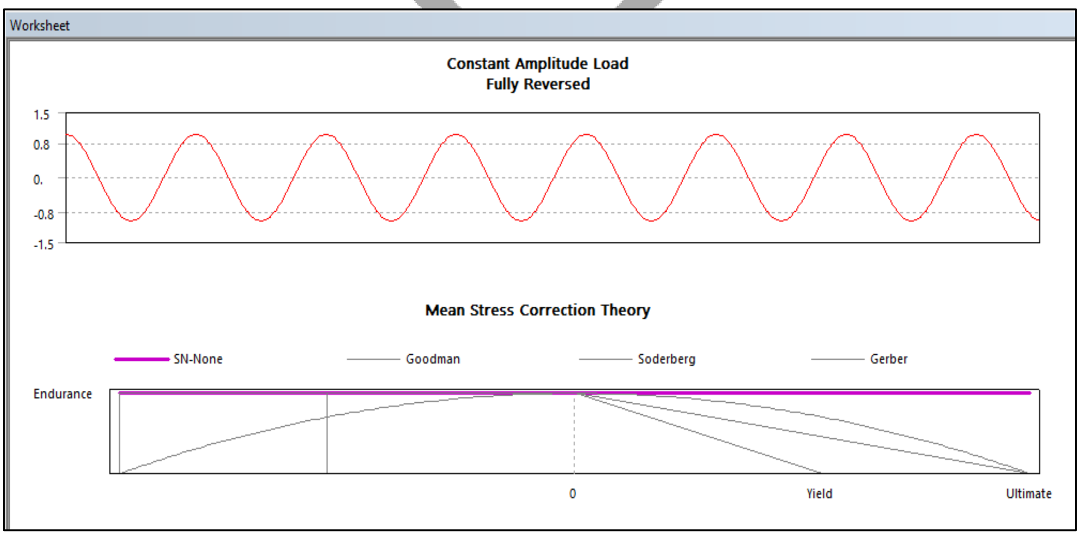


FIG. 11 Fatigue life of original model

Details of "Fatigue Tool"	
Domain	
Domain Type	Time
Materials	
Fatigue Strength Factor (Kf)	1.
Loading	
Type	Fully Reversed
Scale Factor	1.
Definition	
Display Time	End Time
Options	
Analysis Type	Stress Life
Mean Stress Theory	None
Stress Component	Equivalent (von-Mises)
Results Input (Beta)	Stress



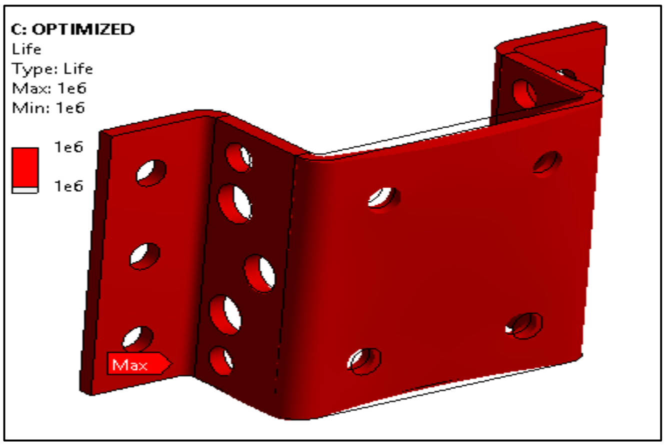


Fig. 12 Fatigue life of optimized model

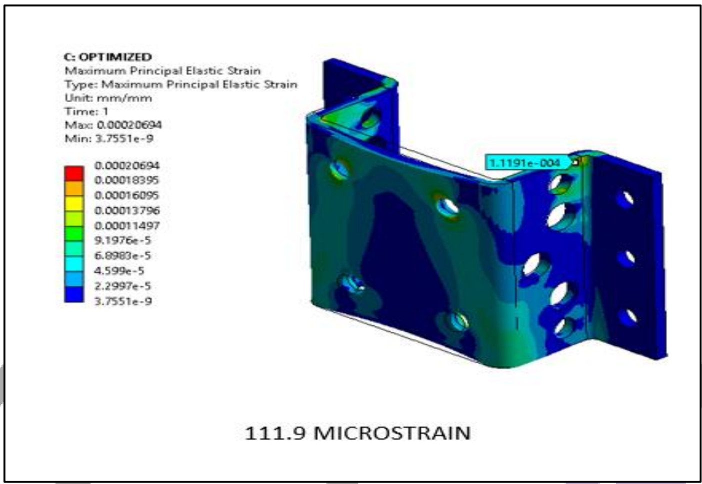


Fig. 13 Principle strain of optimized model

A. EXPERIMENTAL TESTING:

There are various types of experimental methods to analyse strains and stresses at a point. Strain gauge methods use either electrical or mechanical means to measure strains. In these types of strain gauges, electrical resistance strain gauges are the most accurate and widely used ones. This experiment consists of three parts, all utilizing electric resistance strain gauges.

Historically, the development of strain gauges has followed many paths and various methods have been developed based on mechanical, optical, electrical, acoustic and pneumatic principles. In spite of the very wide variations in the strain gauge designs, they all have four basic common characteristics. These are gauge length, gauge sensitivity, measuring range, and, accuracy and reproducibility. Gauge Length: Strains cannot be measured at a point with any type of gauge, and as a consequence non-linear strain fields and local high strains are measured with some degree of error being introduced.

Strain Gauge:

A strain gauge is a device used to measure strain on an object. The most common type of strain gauge consists of an insulating flexible backing which supports a metallic foil pattern. The gauge is attached to the object by a suitable adhesive, such as cyanoacrylate. As the object is deformed, the foil is deformed, causing its electrical resistance to change. This resistance change, usually measured using a Wheatstone bridge, is related to the strain by the quantity known as the gauge factor.

There are different types of commercial strain gauges; these are:

1. Unbonded wire gauges
2. Bonded wire gauges
3. Bonded foil gauges
4. Piezo-resistive gauges
5. Semi-conductive gauges

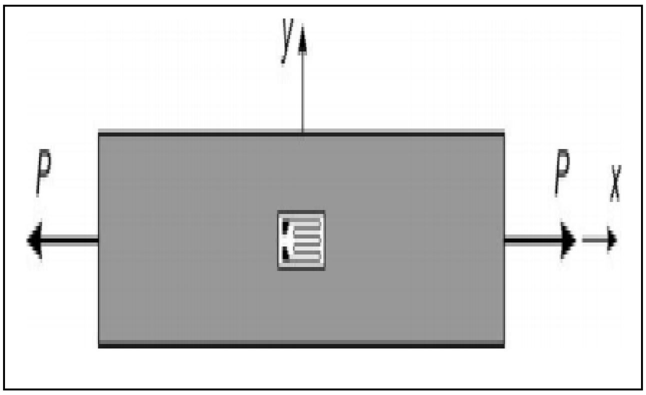


Fig.14. Bonded Foil strain gauges

About **UTM**: A universal testing machine (UTM), also known as a universal tester, materials testing machine or materials test frame, is used to test the tensile strength and compressive strength of materials. The "universal" part of the name reflects that it can perform many standard tensile and compression tests on materials, components, and structures (in other words, that it is versatile)

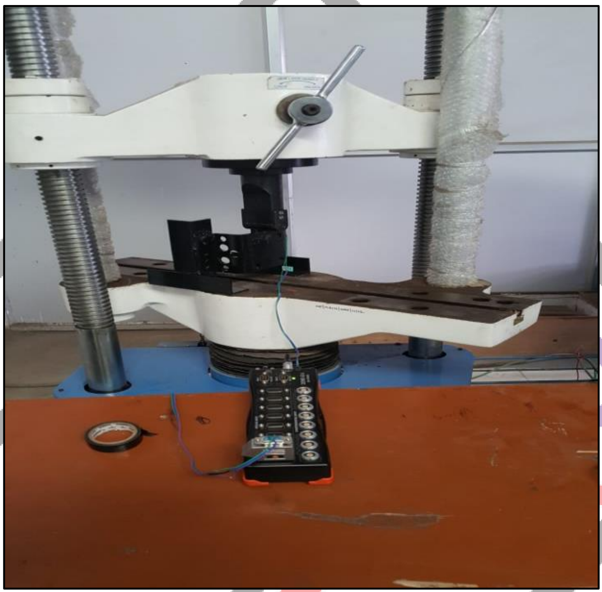


Fig. 15 Experimental testing of Engine mounting bracket

VIII. RESULT AND DISCUSSION

S. No	Parameters	Original Engine bracket		Optimized Engine bracket
1	Deformation (mm)	Max	0.345 mm	0.362 mm
		Min	0 mm	0 mm
2	Von-mises stress (Mpa)	Max	39.414 MPa	39.749 MPa
		Min	0.0012 MPa	0.0014 MPa

Weight Reduction of engine mounting bracket:

Engine mounting bracket	Existing	Optimized	Weight reduction	% Weight reduction
Weight	2.55kg	2.4kg	0.17kg	6.53%

IX. CONCLUSION:

The following conclusions are drawn from the present work.

- Topology optimization of engine mounting bracket.
- From FEA result it conclude that weight of engine mounting bracket after optimization reduces 6.07% than original engine mounting bracket .
- The strain value after the analysis is 111.9 micro strain and strain value after the testing is 122 micro strain.
- The results obtained after the testing & analysis are nearly equal. So, validation of optimized engine mounting bracket is done.

X. ACKNOWLEDGMENTS

I take this opportunity to express a deep sense of gratitude towards my guide Prof. Sanjay A. Pawar, for providing excellent guidance, encouragement and inspiration throughout the project work. Without his invaluable guidance, this work would never have been a successful one. I would also like to thank all my classmates for their valuable suggestions and helpful discussions.

REFERENCES:

[1] P.D.Jadhav, Ramakrishna, "Finite Element Analysis of Engine Mount Bracket,"IJAETMAS: volume 1, Issue 4 September 2014.
[2] Monali Deshmukh, Prof. K.R.Sontakke, "Analysis and Optimization of Engine Mounting Bracket," IJSER: Volume 3 Issue 5, May 2015
[3] Dr.Yadavalli Basavaraj, Manjunatha.T.H , "Design Optimization of Automotive Engine Mount System," International Journal of Engineering Science Invention: Volume 2 Issue 3 March. 2013.
[4] Jong Kim and Heon Kim, "Shape Design of an Engine Mount by a Method of Parameter Optimization," Computer and structure: Volume65, No 5,pp. 725-731,1997.
[5] Diwakar K, Surendra. A, Bhaskar. NandMallikarjuna. P, "Static Structural Analysis and Weight Optimization of Engine Mounting Bracket Using Topology,"Vol-3, Issue-6, 2017.
[6] A.Prasad Babu, Y. Vijaya Kumar, Dr.C.Udaya Kiran, "Topology Optimization in Design of Engine Mounting Bracket," ICEME: VOLUME- 1, 24th, 25th February 2014.
[7] Vijay Kalantre, K. H. Munde and Ashish Pawar, "Topology Optimization of Front Leaf Spring Mounting Bracket," IJSDR: Volume 3, Issue 7 July 2018.
[8] Umesh S. Ghorpade, D.S.Chavan, Vinay Patil & Mahendra Gaikwad, "Finite Element Analysis and Natural Frequency Optimization of Engine Bracket" (IJMIE) ISSN No. 2231 –6477, Vol-2, Iss-3, 2012.
[9] Pramod Walunje, V.K.Kurukute "Engineering Optimization of Engine Mounting Bracket Using FEA" Indian Journal of Research ISSN-2250-1991 volume:2, Issue:12, Dec 2013.
[10] Sandeep Maski, Yadavalli Basavaraj, "Finite Element Analysis of Engine Mounting Bracket by Considering Pretension Effect and Service Load" IJRET: International Journal of Research in Engineering and Technology, e-ISSN: 2319-1163, p-ISSN: 2321-7308, Volume: 04 Issue: 08, August-2015.