Scientific Journal of Impact Factor (SJIF): 5.71

e-ISSN (O): 2348-4470
p-ISSN (P): 2348-6406

International Journal of Advance Engineering and Research Development

Volume 6, Issue 01, January -2019

STATIC ANALYSIS AND OPTIMIZATION OF ENGINE MOUNTING BRACKET

Mr. Swapnil S. Patil[1], Mr. N.S. Dharashivkar[2]

[1,2]*department Of Mechanical Engineering, TatyasahebKore Institute Of Engineering And Technology, Warananagar*

Abstract-*An engine mount is the part that holds the engine to the body or chassis of the car. In a typical car, the engine and transmission are attached together with the help of three or four mounts. The mount that holds the transmission called transmission mount and others are referred to as engine mounts. To reduce the engine vibration encountered inside the car, engine mounts are filled with rubber so that there is no direct metal to metal contact between engine and the car body. Vibrations and fatigue of engine bracket has been continuously a concern which may lead to failure of bracket in case of vibrations and stresses are severe and excessive. Prolonged exposure of system to vibration in the working condition may lead to fatigue and in some cases it damages the car. The paper deals with experimental and finite element analysis of engine mounting bracket and optimization of bracket. And results obtained by these two method were compared each other.*

Keywords-*Engine Mounting Bracket, Finite Element Analysis, Stress, Deformation, Universal Testing Machine,*

I. INTRODUCTION

Design engineers always aims for improvement in each part of automobile system. Automobile industry continued improving from many years for the purpose of modification of the mechanical parts of vehicles in order to improve their performance. These characteristics have a vital impact on the overall performance of system. In addition, redesigning the mechanical model plays an important role in improving the sustainability of the system against the resultant stresses and strains; therefore, significant consideration should be taken for this, when designed by engineers.

In an automotive vehicle, the engine rests on brackets which are connected to the chassis of the car. Hence, during its operation, the undesired vibrations generated by the engine and road conditions can get directly transmitted to the chassis through the brackets. This may lead to discomfort to the passenger and more vibrations might even damage the chassis.

Engine mounts are small parts that are used to stabilize, as well as align, a vehicle's engine. So, even though these mounts are small in size, they play a important role in the overall function of your vehicle. Moreover, when these supposedly small and minor components of the vehicle go bad, the mounts can cause a variety of problems for your vehicle.

Fig.1. Engine Mounting Bracket

II.LITERATURE REVIEW

Umesh S. Ghorpade et al. worked on static structural and modal analysis of engine bracket using finite element analysis. The need for light weight structural materials in automotive applications is increasing for improvement in emissions and fuel economy. The effective way to increase automobile mileage is to reduce vehicle weight. The incorporation of aluminum and magnesium alloys into automotive structures has steadily increased to meet all these requirements. This paper deals with Finite Element (FE) analysis of a typical engine bracket of a car and natural frequency will be determined.

P.D. Jadhav et al. worked on finite element analysis of engine mount bracket. The engine mounting plays an important role in reducing the vibrations and harshness of vehicle for improving vehicle ride comfort. The function

of an engine mounting bracket is to properly balance the engine on the vehicle chassis for good motion control as well as good isolation. this work deals with FEA analysis of engine mounting bracket. It includes the modeling of the engine mounting brackets using 3D modeling software.. Materials selected are Aluminum alloy and magnesium alloy. Static and Modal Analysis of engine mounting bracket done by using Square Cross -section. The study shows that this bracket will have weight reduction with compare to standard aluminum alloy material which withstand high stress.

Po Wu et al. worked on topology optimization design of automotive engine bracket. According to the structural parameters of the automobile engine bracket, the finite element model of the bracket is established. As the engine mount is connecting part between the engine and the chassis, the performance requirements of the automobile engine bracket affect the comfort and the safety of the vehicle directly. Under the premise of ensuring its reliability, with the help of OptiStruct software optimization of bracket carry out to get the optimal material distribution of the bracket and the final design will meet the performance requirements.

Sameer U. Kolte et al. worked on structural durability analysis of power train mounting bracket. Structural analysis is performed to check durability of specified part for a given load conditions. The engine mounting bracket is subjected to loads primarily due to weight of the engine, the unbalanced torque. The CAD model is imported in CATIA V5R21 software. The bracket is subjected to different load conditions. And this model is optimized so that stresses generated and deflection of the bracket is within permissible limit. Effect of bolt preloads is also considered. The results of the analysis are compared with Hyper mesh results and are found to be in good correlation.

III. FININTE ELEMRNT ANALYSIS

The finite element method is a numerical analysis technique which is used to obtain approximate solutions to a wide variety of engineering problems. The method originated in the aerospace industry as a tool to study stresses in complex structures. This method is general enough to handle any complex shape or geometry, any material properties, any boundary conditions and any loading conditions. It is an effective design tool by which designers can perform parametric design studies by considering various design cases, analyzing and optomizing them The finite element method is consist of three phases. Pre-processing, in which the analyst generates a element mesh to divide the subject geometry into subdomains for mathematical analysis and applies material properties and boundary conditions. Second is solution, during which the program derives the governing matrix equations from the model and solves for the primary quantities. And third one is post- processing, in which the analyst checks the validity of the solution, examines the values of primary quantitates.

A.Modeling

In this phase we prepare 3D model of a engine mount as per specification by using CATIA V5 (computer aided three dimensional interactive application) software. Imported CATIA assembly in ANSYS software using STEP (stp) files.

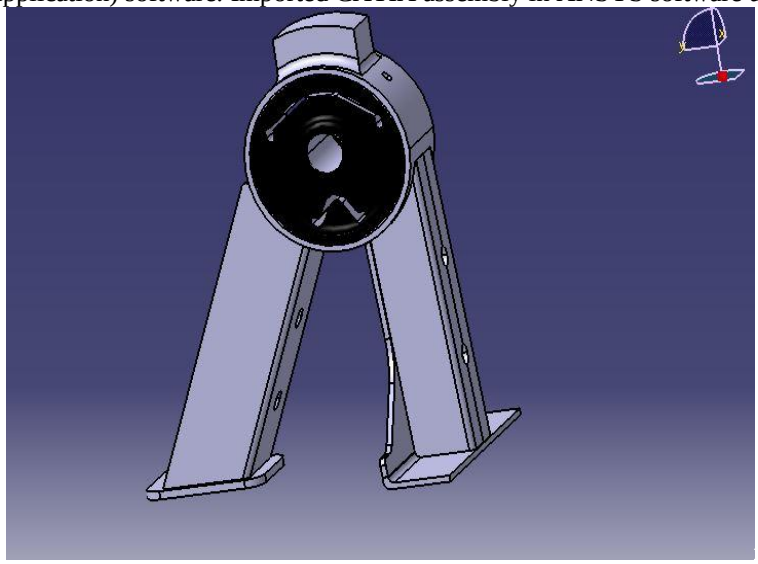

Fig.2.*Engine mounting bracket*

B. Pre-processor

Meshing- Type of element used is tetrahedron element solid.

Element size is 3 mm.

Number of elements are 133110.

Number of nodes are 220464.

Define material properties :Density - 7845 kg/m^3

Young's Modulus (E) - 2 x 10^5 N/mm^2

Poisons Ratio - 0.3

C.Post processing

After completion of procedure for finite element analysis of a engine mounting bracket, we get results in the form of stress and deformation. As per our interest we estimate the portion of bracket where stresses and deformation are maximum. Stresses are estimated from the colour at portion of bracket and corresponding value for colour in window gives the stress value for it

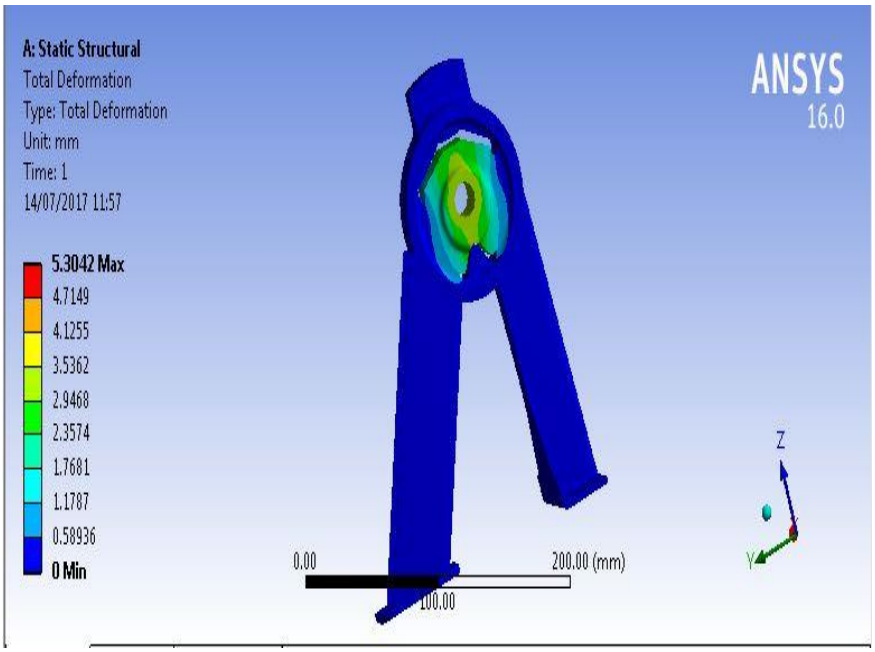

Fig.3. Deformation in bracket

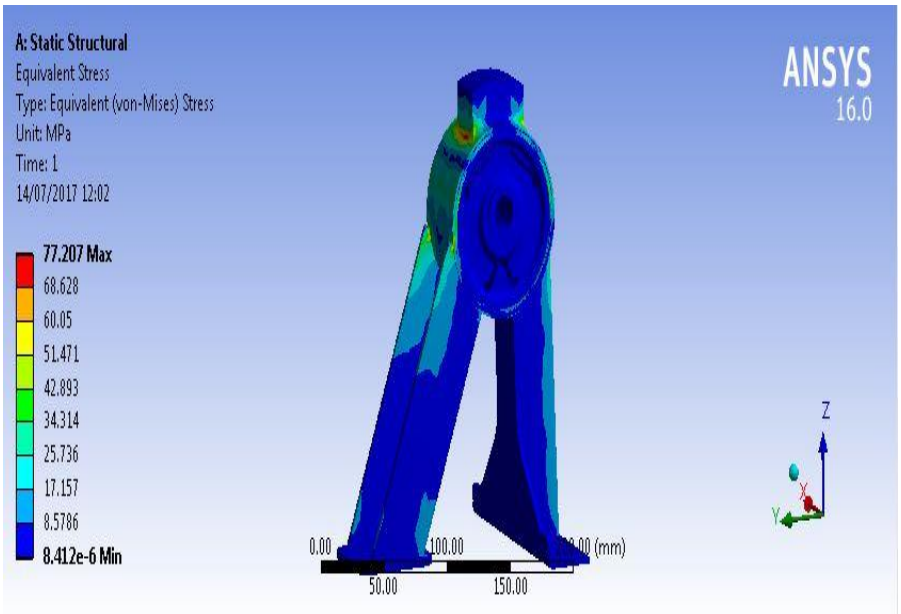

Fig.4. Equivalent stress in bracket

From above analysis deformation and stress in bracket for 3650 N load are 4.12 mm and 77.20 Mpa respectively.

IV. EXPERIMENTAL ANALYSIS

Engine mounting bracket used for experimentation is of Suzuki swift. Measurements of stress and deformation are carried out by universal testing machine.

A) Design and development of fixture

A fixture is a work holding or support device used in manufacturing industry. For experimental analysis a fixture is prepared which contains one flat platform on which loading is done. This platform is supported by two columns which are connected to rod passing through hole of engine mounting bracket.

Fig. 5. Fixture connected to bracket

B) Measurement of deformation and stress

Calculation for load on engine mounting bracket is done as follows
Weight of the engine =136 Pounds =62Kg
As we known,

Factor of safety for engine componats is varies from 3-6 [17]. Here we take factor of safety 6.
maximum load = FOS *working load
= 62*6
= 372Kg
Total weight = 372 * g = 372 * 9.81 = 3650 N
Load of 3650 N is applied on engine mounting bracket by UTM.

Fig. 6. UTM in loading condition

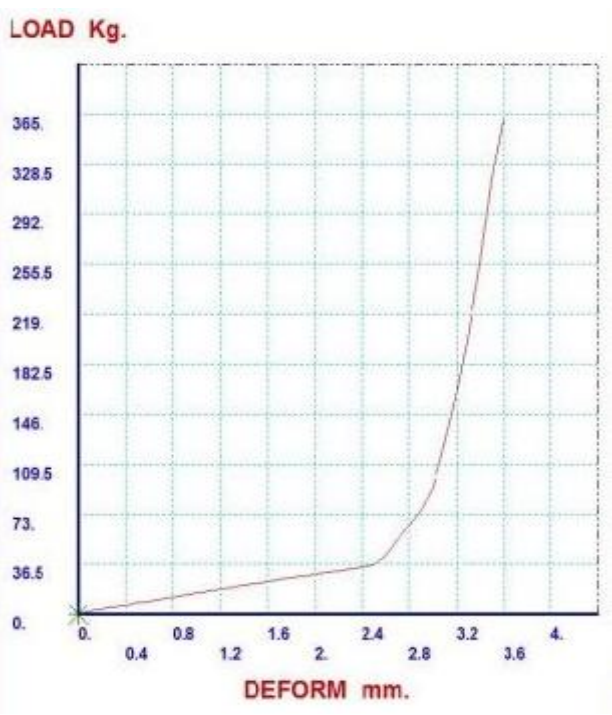

Fig.7. Compression test graph

From above graph of load Vs deformation, deformation for load 3650 N is 3.6 mm. Using hook's law stress is calculated which is 72 MPa.

V. OPTIMIZATION

Optimization analysis is used to optimize the geometric parameters and shapes of over or under designed components. Optimization for geometry parameters, work well at individual component level rather than complicated assemblies. Software is not useful to add or remove the geometry but it works only within specific limits.
The optimization is done with changing the material rubber is replaced by neoprene rubber and the steel is changed to titanium alloy. Properties of structural steel and natural rubber are replaced by titanium alloy and neoprene rubber inansys software.
Properties for titanium alloy are

1) Density - 4506 kg/m^3
2) Young's Modulus (E) - 120 GPa
3) Poisons Ratio - 0.31

And the results are shown below:

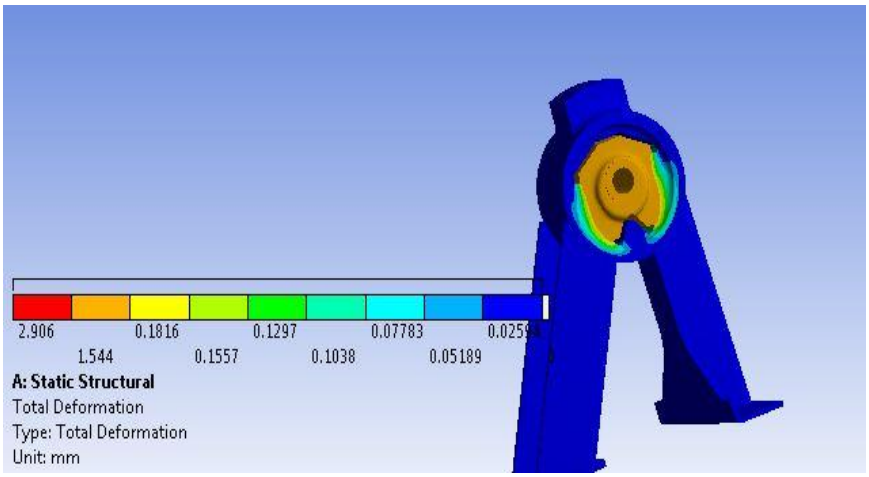

Fig.8. Total deformation of optimized model

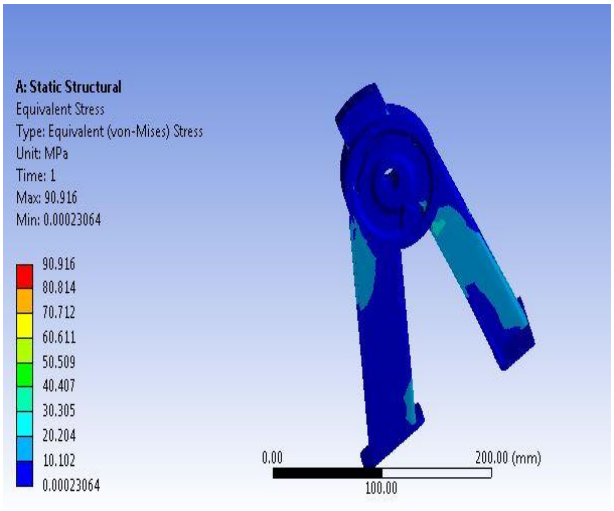

Fig.9. Equivalent stress of optimized model

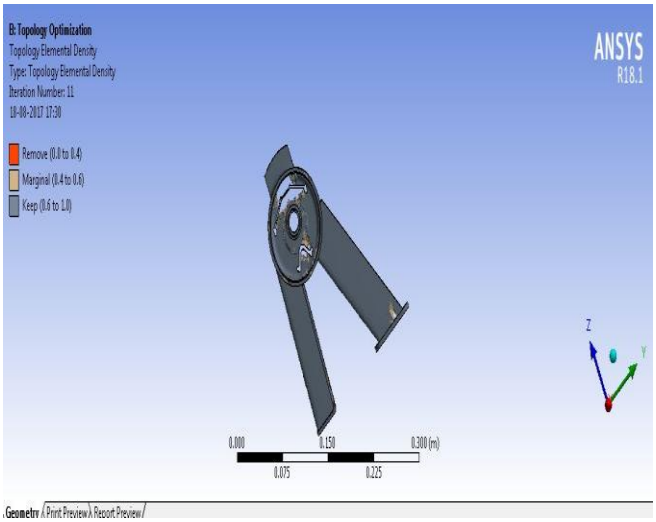

Fig.6.5. Optimized model for weight reduction

In topology optimization base metal is replaced by another efficient metal which will help to reduce stress concentration , weight of model, size and shape of model. Here we use titanium alloy instead of steel and rubber in engine mounting bracket is replaced by neoprene rubber. As a result of optimization in software we get 20% of weight reduction.
Original weight of engine mounting bracket is 2200 gm.
After 20% reduction weight is 1750 gm
Reduction in weight is 450gm
By reducing weight of engine mounting bracket we can reduce the weight on chassis of vehicle which is beneficial for the vehicle.

VI. RESULT AND DISCUSSION

To find out deformation and stress in engine mounting bracket under static loading we have done experimentation and analysis with software. Experimentation is done with universal testing machine by applying static load on bracket. Finite element analysis is done in ANSYS software .results of both are tabulated below.

Load under static condition (N)	Deformation		Deviation (mm)	Stress (Mpa)		Deviation (Mpa)
	UTM	FEA		UTM	FEA	
3650	3.6	4.12	0.52 (12%)	72	77.20	5.2 (7%)

Table.1. Comparison between both results for original model

Load under static condition (N)	Before Optimization		After Optimization		Deviation (mm)	Deviation (Mpa)
	Deformation (mm)	Stress (MPa)	Deformation (mm)	Stress (MPa)	Deformation	Stress
3650	4.12	77.20	1.5	50.50	2.72	26.7

Table.2. Comparison between both results before and after optimization

As per results we can see that there is deviation in stress and deformation between experimental and finite element analysis. There is 0.52 mm (12%) deviation in deformation between experimental and finite element analysis and 5.2 Mpa (7%) deviation in stress between experimental and finite element analysis. Finite element analysis method is an approximate method to estimate stress and deformation under loading hence there is deviation between experimental and analysis results.
The material of the present model and the material used in the optimized model differ; the result of optimized model is better comparing with the present model. The change in deformation of optimized model is about 2.60 mm from the present model. There is 34% reduction in deformation and 65 % stress reduction after optimization.

CONCLUSION

Owing to the results obtained by different methods used to carry out static structural analysis of engine mounting bracket, following conclusions can be drawn at the end of dissertation work
UTM if effectively used for estimation of stress and deformation under static loading

There is good agreement between experimental and FEA results. Error for deformation by UTM amd FEA is below15% and error for stress by UTM and FEA is below 10%.

With optimization stress and deformation is reduced up to certain extent.

Base metal is replaced by more efficient metal which will give 20% weight reduction of engine mounting bracket.

Optimization also gives reduction in stress and deformation of engine mounting bracket.

REFERANCES

[1] 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, ISSN No. 2231 –6477, Vol-2, Iss-3, 2012.
[2] Abdolvahab Agharkakli, Digvijay Pradip Wagh, **"Linear Characterization of Engine Mount and Body Mount for Crash Analysis"**, International Journal of Engineering and Advanced Technology ISSN: 2249 – 8958, Volume-3,Issue-2, December 2013.
[3] Dr.Yadavalli Basavaraj, Manjunatha.T.H, **"Design Optimization of Automotive Engine Mount System",** International Journal of Engineering Science Invention ISSN (Online): 2319 – 6734, ISSN (Print): 2319 – 6726.
[4] P.D. Jadhav, Ramakrishna, **"Finite Element Analysis of Engine Mount Bracket"**, International journal of advancement in engineering technology, management and applied science, ISSN no. 2349-3224.
[5] Vikas Doundkar, Dhananjay Ghatage, Manish Madkaikar, AniruddhaKulkarni, **"Topology Optimization for Engine Mounting Arm with Fatigue",** HTC2011
[6] Koushik S., **"Static and Vibration Analysis of Engine Mounting Bracket of TMX 20-2 using OptisStruct"** Altair Technology Conference 2013, India.

[7] Sanket C. Vinchurkar, P. M. Khanwalkar, **"FEA and Topology Optimization of an Engine Mounting Bracket"**, International Journal of Current Engineering and Technology E-ISSN 2277 – 4106, P-ISSN 2347 – 5161

[8] A.S.Adkine ,V.S.Kathavate, G.P.Overikar, S.N.Doijode, **" Static Behaviour of Engine Mounting Bracket"**, International Advanced Research Journal in Science, Engineering and Technology, Vol. 2, Issue 4, April 2015

[9] Sameer U. Kolte ,DavidNeihguk , and Abhinav Prasad, **"Structural Durability Analysis of Powertrain Mounting Bracket"**, Excerpt from the Proceedings of the 2013 COMSOL Conference in Bangalore, Mahindra and Mahindra Ltd.

[10] 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

[11] Ritesh Bhairappa Dhamoji, Dr.Santosh V Angadi, Prof. Anand S N, **"strength and stiffness analysis of an engine bracket"**, International Journal For Technological Research In Engineering Volume 3, Issue 10, June-2016

[12] Jasvir Singh Dhillon, PriyankaRao, V.P. Sawant, **"design of engine mount bracket for a fsae car using finite Element Analysis"**, Int. Journal of Engineering Research and Applications, ISSN : 2248-9622, Vol. 4, Issue 9(Version 6), September 2014, pp.74-81

[13] AbburiLakshmankumar, KolliBalasivaramareddy, A. Sathiskumar, N. Kuthus, **" stress analysis and optimization of engine mount"** ,International Journal of Mechanical Engineering and Technology (IJMET) Volume 8, Issue 6, June 2017, pp. 148–155 Article ID: IJMET_08_06_016

