ISSN 2395-1621

# Static Structural Analysis and Optimization of Engine Mounting Bracket

[#1]Chandrashekhar B Tayde, [#2]Prof. D. H. Burande

[1]chandrashekhar.tayde@gmail.com
[2]dhburande.scoe@sinhgad.edu

[#12]Department of Mechanical Engineering
Sinhgad College of Engineering
Pune 411 041.

## ABSTRACT

**One of the main causes of vibration that produce by a car is engine and transmission. Engine and transmission location determination is an important thing that must be considered for a car. Automobile industry continued improving from many years with the efforts conducted for the purpose of modification of the mechanical parts of vehicles in order to improve their performance and vibration response. The most effective way of increasing automobile mileage while decreasing emissions is to reduce vehicle weight. In this paper a study has been done to model the engine mounting bracket using CAD tool to do static structural analysis for stresses, deformation in CAE tool & second part of project is topology optimization for mass reduction without much altering strength using CAE tool.**

**Keywords: Bracket, Topological Optimization, CATIA, Hypermesh, Ansys**

ARTICLE INFO

**Article History**

Received: 22nd July 2016

Received in revised form : 22nd July 2016

Accepted: 25th July 2016

**Published online : 25th July 2016**

## I. INTRODUCTION

In current automotive era the need for light weight structural materials is increasing as there is a more focus on fuel consumption reduction and improvement in decreasing the emission. The magnitude of production volumes has traditionally placed severe requirements on the robustness of process used in the manufacturing. The manufacturers have strong importance on the cost has the demand for the component to improve the material performance and to deliver these materials at low cost is the requirement.

In automobile sector the extremely competitive automotive business needs manufactures to pay a lot of attention to travelling comfort. Resonant vibration is from unbalanced masses exist within the engine body, this is causing the designers to direct their attention to the event of top quality engine mounting brackets so as to confirm that there is improvement in riding comfort. The demand for higher play acting engine mount brackets should not be offset by arise within the production prices and/or development cycle time.

In diesel engine, the engine mounting bracket is the major problem as there is unthrottled condition and higher compression ratio and even there are more speed irregularities at low speed and low load when compared to gasoline engines. So due to this there are more vibration excitation. By this vibration engine mount bracket may fail, so by optimizing the shape and thickness of engine mount bracket we can improve the performance at initial design stages. Structural optimization is an important tool for an optimum design; comparison in terms of weight and component performance structural optimization techniques is effective tool to produce higher quality products at lower cost.

## II. PROBLEM STATEMENT

In this automotive era the need 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 thesis, modeling present bracket in CAD software and analyzing it for induced stresses and deformation in CAE software. 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.

## III. OBJECTIVE

1. Modeling current bracket.
2. Analyzing for stresses and deformation.
3. Topological optimization for the model.
4. Analyzing for stresses and deformation on optimized model.
5. Experimental testing and correlating results.

## IV. LITERATURE REVIEW

Umesh S Ghorpade, D.S Charan, Vinay Patil and Mahendra Giakwad [1], in this paper they have designed engine mount bracket of a car and focused on to determine natural frequencies of car engine mount bracket.

Mr. Pramod Walunje, Prof. V.K. Kurkute [2], in this work they have mainly focused on the use of light weight material for bracket and also to reduce the weight of the bracket. Here the weight of the material is reduced and pre processing and post processing is carried out and even with this an experimental setup is also used to find the stress level of the materials.

P.D. Jadhav, Ramakrishna [3], this work is a contribution to the development of new material for engine mounting bracket. The results obtained for the static structural and modal analysis have shown that the magnesium is better than aluminum.

Sandeep Maski, Yadavalli Basavaraj [4], conducted test on bracket and concluded that the maximum von-misses stress coming on mild steel is lesser than yield point, hence mild steel is considered as better material to design engine mount bracket.

B. Sreedhar, U. Naga Sasidhar [5], on topology optimized model finite element analysis has carried out for normal mode, FRF and strength analysis and complete parametric study has carried out by using Hyper Study. In this paper, topology optimization approach is presented to create an innovative design of an engine mount bracket. Final comparison in terms of weight and component performance illustrates that structural optimization techniques are effective to produce higher quality products at a lower cost.

## V. FINITE ELEMENT ANALYSIS

Engine mounts themselves are small parts that are meant to stabilize, as well as properly align, a vehicle's engine. So, even though these mounts are small, they play a large role in the overall functionality of the heart of your vehicle.

An engine mounting system includes a front mount, a rear mount, an engine mount, and a transmission mount. Installation heights and spring constants of each mount are predetermined such that the majority of the weight of the power train is supported by the front and rear mounts.

Fig.1 Existing Bracket

Dimensions of bracket are found physically with help of measuring instruments. Bracket is modeled with help of CATIA (V5R20) software.

**Computer Aided Design (CAD):**

CATIA (computer aided three-dimensional interactive application) is a multi – platform computer - aided design (CAD) / computer - aided manufacturing (CAM) / computer - aided engineering (CAE) software suite developed by the French company Dassault Systems.

CATIA delivers the unique ability not only to model any product, but to do so in the context of its real-life behaviour: design in the age of experience. Systems architects, engineers, designers and all contributors can define, imagine and shape the connected world.

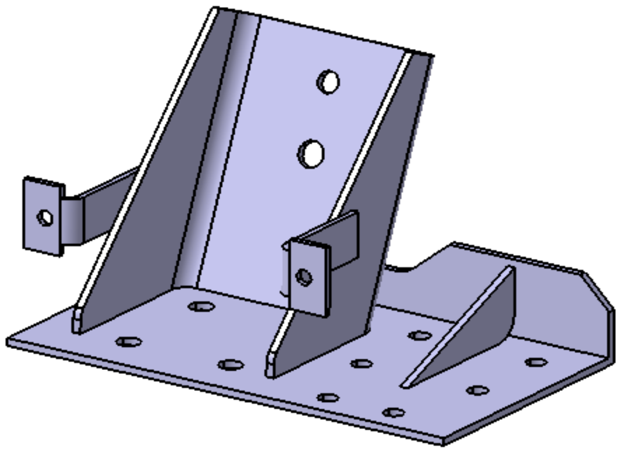

Fig.2 Modeled in CATIA (V5R20) Tool

**Finite Element Modeling (FEM):**

It is a numerical technique for finding approximate solutions to boundary value problems for partial differential equations. It is also referred to as finite element analysis (FEA). FEM subdivides a large problem into smaller, simpler, parts, called finite elements. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem. FEM then uses variational methods from the calculus of variations to approximate a solution by minimizing an associated error function.

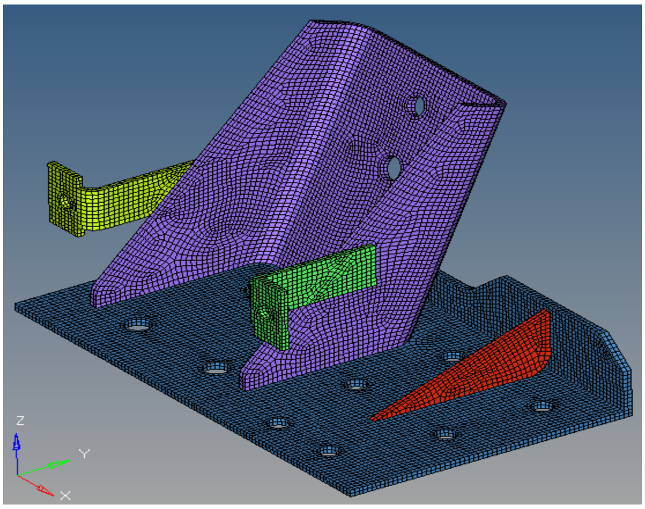

Fig.3 Solid Meshing

**Discretization:**

A solid element mesh is required to be generated. The meshing of the bracket is done in HYPERMESH software.

Type of Elements – Hexahedron, Pyramid, Wedge
Element Order- First order
Number of Elements- 26558
Number of Nodes- 40140

**Static Structural Analysis:**

A static structural analysis is done to analyze displacements, stresses, strains and forces on structure or a component due to load application. The structures response and loads are assumed to vary slowly with respect to time. There are various types of loading that can be applied in this analysis which are externally applied forces and pressures, and temperatures.

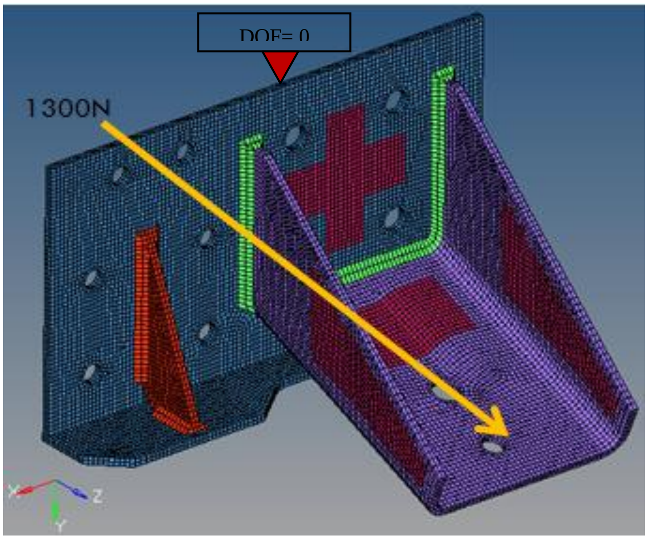


Fig.4 Load & Constraint

**Material Properties:**
Material- Structural Steel
Young's Modulus- 200 GPa
Poisons Ratio- 0.3
Density- 7850 kg/m^3
Yield Strength- 250 MPa

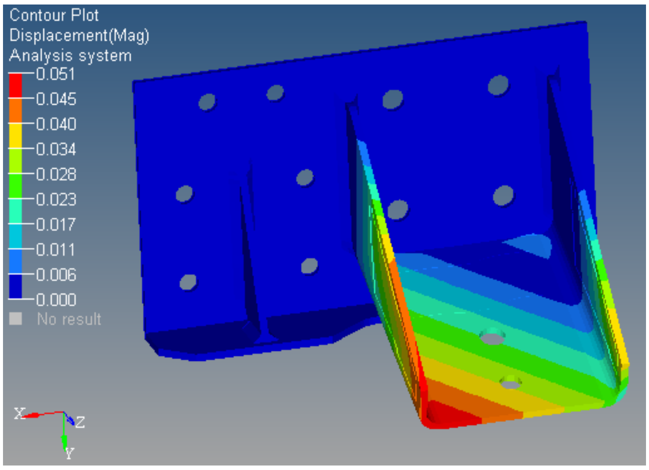


Fig.5 Maximum Deflection of existing Design

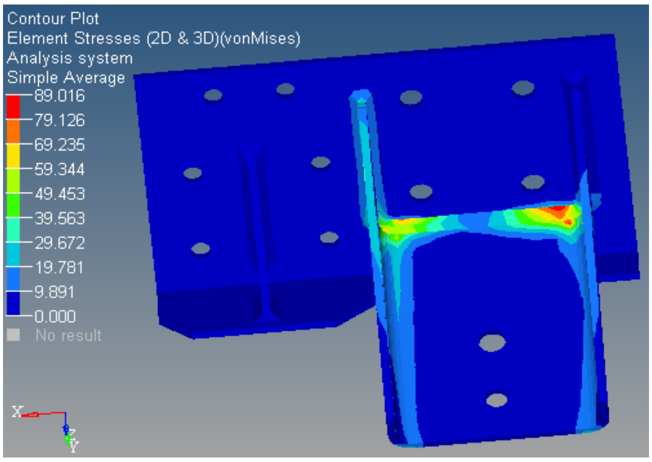


Fig.6 Maximum Von-Mises stress for given boundary

## VI. TOPOLOGICAL OPTIMIZATION

Topology optimization is a mathematical approach that optimizes material layout within a given design space, for a given set of loads and boundary conditions such that the resulting layout meets a prescribed set of performance targets.

There are three kinds of structure optimization,

- Size Optimization
- Shape Optimization
- Topology Optimization

The topology optimization consists of the following sequence of steps.

- Define the design space
- Define optimization parameters
- Material removal process and detail design

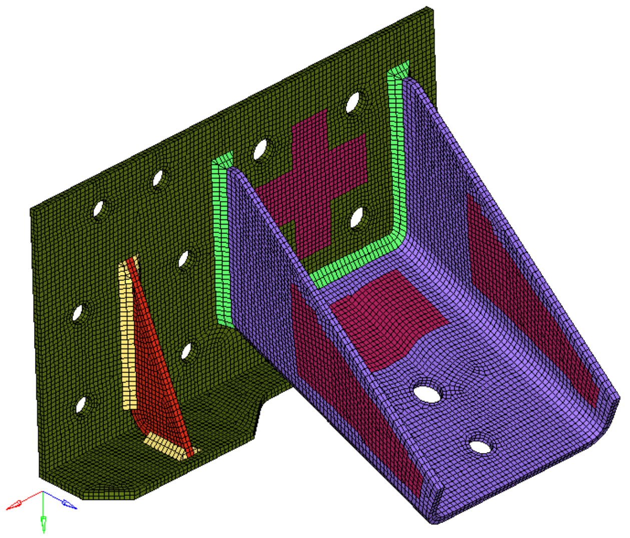
Fig.7 Bracket With Extra Material

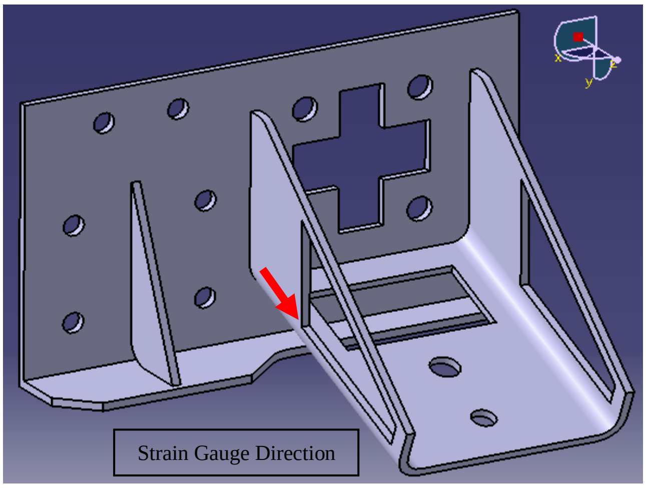

Fig.8 Optimized Model

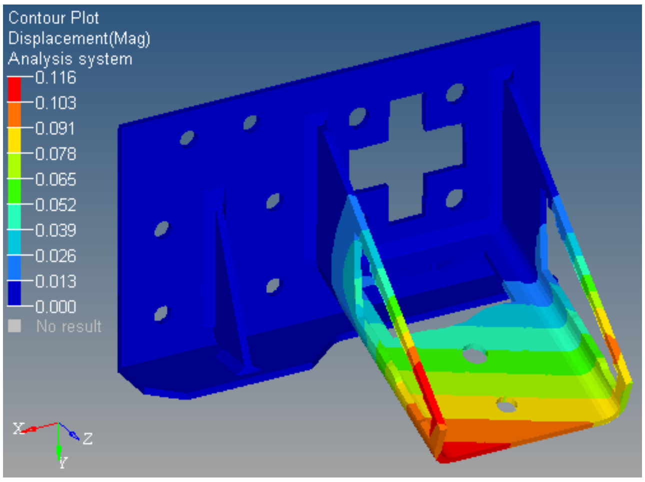

Fig.9 Maximum Deflection Of optimized model

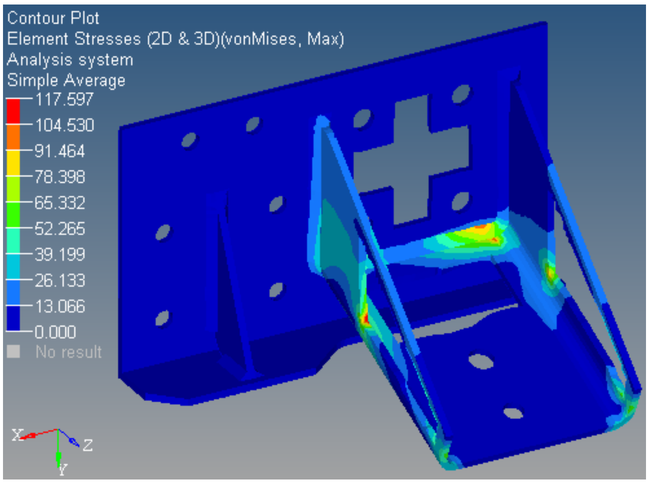

Fig.10 Maximum Von-Mises stress of optimized model for given boundary conditions

## VII. EXPERIMENTAL

Existing bracket is machined on Electronic Discharge Machine (EDM) to get the required accuracy.


Fig.11 Machined Part as per Optimized Model


Fig.12 Experimental Setup

Experiment is done on UTM having capacity of 100ton. Fixture is designed as per testing feasibility. Load applied is 1300N. Strain from experimental is 160microns.

 

## VIII. CONCLUSION

From results of finite element analysis it is observed that the maximum stress value is within the safety limit. There is a great potential to optimized this safety limit which can be done by removing material from low stressed region thus optimizing its weight without affecting its structural behavior. The maximum displacement value is also very less. So, if the material from low stressed region is removed this mass can be reduced without affecting its strength and within the yield strength.

- Von-mises stress of existing and optimized components are having 15% variation compared to material yield strength.
- Deflection measured and found on existing and optimized model is very less.
- With this topological optimization a weight saving of 23% from existing to optimized component.

## ACKNOWLEDGEMENT

It is my great pleasure to acknowledge sense of gratitude to all, who have made it possible for us to complete this seminar with success. It gives me great pleasure to express my deep gratitude to my seminar guide Prof. D. H. Burande for his support and help from time to time during seminar work.

It is my pleasure to acknowledge sense of gratitude to our Head of Department Prof. V. N. Kapatkar, Vice Principal Dr. Y. P. Reddy and Principal Dr. S. D. Lokhande for their great support and encouragement in seminar work.

Finally yet importantly I would like to thank all Staff Members and all our colleagues for their valuable suggestions and support.

## REFRENCES

[1] 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.

[2] Mr.Pramod Walunje, Prof.V.K.Kurkute, “Engineering Optimization of Engine Mounting Bracket Using FEA” Indian Journal of Research ISSN-2250-1991 volume:2, Issue:12, Dec 2013.

[3] P.D. Jadhav , Ramakrishna , Finite Element Analysis of Engine Mount Bracket, Internation Journal of Advancement In Engineering Technology, Managament And Applied Science, Volume 1, Issue 4, Issue no. 2349-3224,Setember 14

[4] 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.

[5] B. Sreedhar,U. Naga Sasidhar, Optimization of Mounting Bracket, htc2012

 