

Finite Element Analysis of Engine Mounting Bracket

Rahul S Kadolkar¹, Anand C Mattikalli²

¹M Tech, Maratha Mandal Engineering College, Belgaum, Karnataka

²Asst. Professor, Maratha Mandal Engineering College, Belgaum, Karnataka

Abstract - One of the main component of the automobile or the heart of the automobile is Engine. As it is one of the main components its mounting with the Chase is done on the Engine mounting bracket with a rubber vibration isolator in between the bracket and the engine. Hence the design of the engine mount becomes the critical aspect in terms of vehicle crashworthiness. In this particular study, different designs of the engine mount system have been evaluated for damageability starting from the base model. In the process of optimizing the damageability, study has been performed repetitively on engine mount with design configurations.

This project deals with the design and material optimization of Engine mounting bracket. The results were found to be around vicinity and within the working limits. The stress distribution and deformation of the engine mounting bracket of Finite Element Analysis were closer to the theoretical results and the nature of the graph of Theoretical and FEA were Co-relating.

Key Words: Static Load, Cantilever Beam, Hyper Mesh, SG 500/7, Analysis, Optimization.

1. INTRODUCTION

Engine mounting bracket is a very critical component in any automotive vehicle, upon which the engine is mounted. It takes the various forces produced by the engine. Engine of any automobile is not directly mounted onto the chassis but upon an engine mounting bracket, which is fastened to the chassis with the help of fasteners. Vibration isolators are provided between the engine mounting bracket and the engine so as to absorb undesirable vibrations. [1]

1.1 Problem Definition

The engine mount racket is subjected to various stresses. The current project work deals with the static load condition and for the L type of engine mounting bracket when vehicle moves on a plain road surface and when the vehicle passes across the potholes and also determining the stresses and deflection in the different cases of mesh sizes Further in this project more optimization is done by selecting different material grades and further analyze.

1.2 Objectives

To analyze and optimize the Engine Mounting Bracket for different mesh size case and optimize by using different material grade.

To determine the optimal and critical point having highest stress and optimized in terms of reducing weight and reducing stresses.

1.2 Methodology

First of all we define the objective of the project and then we study the loading conditions of the bracket that involves studying the different cases for which we determine the loading conditions after which we do the modeling in CATIA as the model is provided by the company. Then we convert the model from its parent format to IGES after which geometry clean-ups is done and do the meshing in hyper mesh using proper elements after which we give the boundary conditions and put it in the solver to get the results which is compared with theoretical calculations. Later comes to the documentation and report. [2]

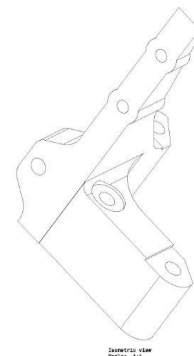


Fig- 1: Engine Bracket

2. FINITE ELEMENT ANALYSIS

The finite element analysis (FEA) is a numerical technique for solving problems which are described by fractional differential equations. Approximating functions in finite elements are determined in terms of nodal values of a physical field which is sought after. An uninterrupted physical problem is changed into a discretized finite element problem with unknown nodal values. For a rectilinear problem, a system of linear algebraic equations

must be solved. Values inside finite elements can be noted using nodal values. [3]

2.1 Meshed Model

The Finite Element model prepared in hyper mesh and meshed with an element size of 4 and the quality criteria as given above in the table of quality criteria. At first we organized the entire assembly and given each and every component an appropriate name with which have similar thickness. Then we extract the mid surface and in each and every component and save it in the same component. After which we start meshing we constantly check for quality criteria so as to see the elements which have failed and repair them so attain required mesh quality then finally we check for duplicates and do the further analysis.[4]

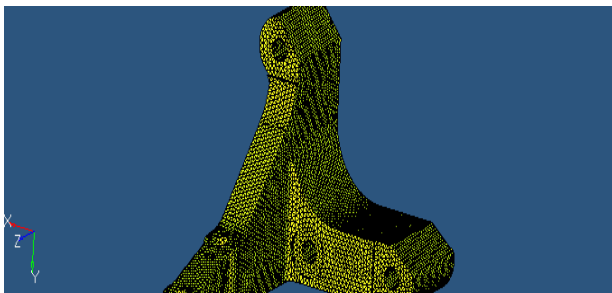


Fig -2: Meshed Model

3. ANALYSIS AND CALCULATION

3.1 Theoretical Calculation

Material – SG 500/7

Load on One hole = 4905 N

Depth, d = 57 mm

b = 23.5mm

$I = 362669.625 \text{ mm}^4$

$Z = 12725.25 \text{ mm}^3$

Taking moment at B,

$$M_B = 382590 \text{ N-mm}$$

$$\sigma_{nom} = 30.07 \text{ N/mm}^2$$

$$\sigma_{max} = 63.15 \text{ N/mm}^2$$

Deflection will be maximum at free end.

Total Deflection = Deflection at free end due to load (W1) + Deflection at free

Endue to load (W2)

$$W_1 = 4905 \text{ N}$$

$$W_2 = 4905 \text{ N}$$

$$Y_1 = \frac{(L - a)W_1 a^2}{2EI}$$

$$Y_2 = \frac{(L - a)W_2 a^2}{2EI}$$

Total Deflection at free end

$$Y_{max} = 0.067 + 0.082$$

$$Y_{max} = 0.149 \text{ mm}$$

3.2 Finite Element Analysis of SG500/7

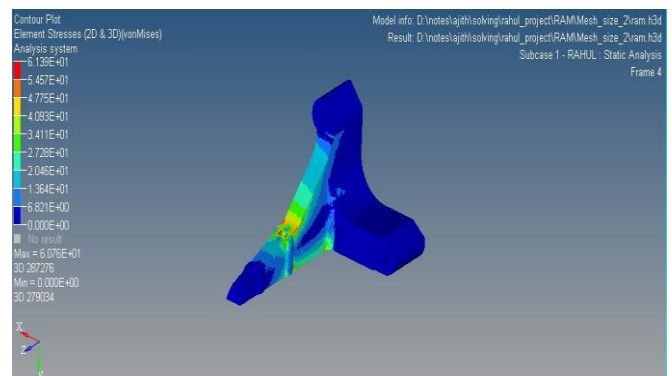


Fig-3.1: Contour Plot of Stress for 2mm size

The analysis is done by considering mesh size 2mm The Fig shows stresses generated in the front section of the bracket due to static case which is 61.39MPa.

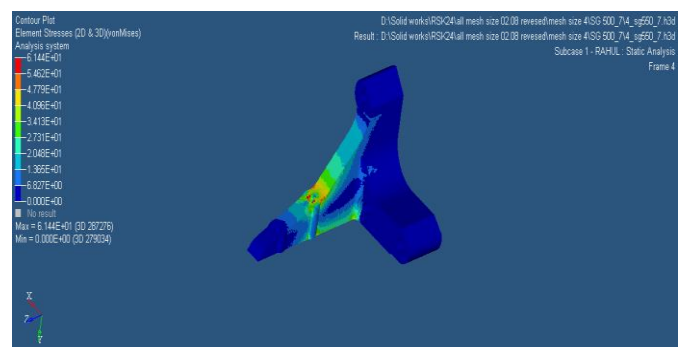


Fig-3.2: Contour Plot of Stress for 4mm size

The analysis is done by considering mesh size 4mm The Fig shows stresses generated in the front section of the bracket due to static case which is 61.44MPa.

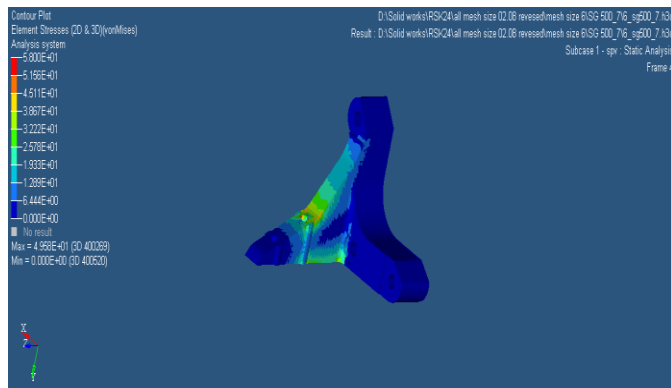


Fig-3.3: Contour Plot of Stress for 6mm size

The analysis is done by considering mesh size 6mm The Fig shows stresses generated in the front section of the bracket due to static case which is 58.01MPa.

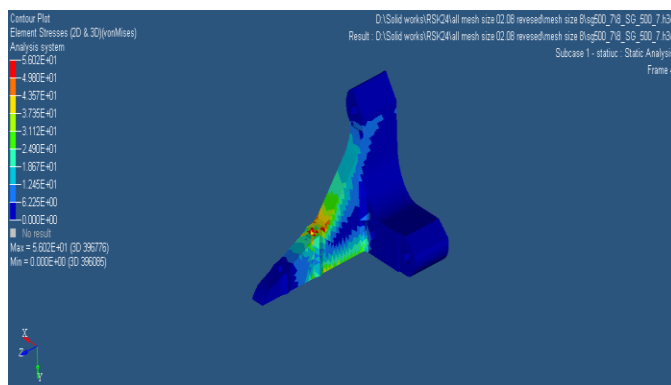


Fig-3.4: Contour Plot of Stress for 8mm size

The analysis is done by considering mesh size 8mm The Fig shows stresses generated in the front section of the bracket due to static case which is 56.02MPa. [4]

3.3 Modal Analysis

Modal analysis controls the vibration characteristics of a particular component in the form of natural frequencies and mode shapes. The natural frequencies and mode shapes are important in the design of a structure for dynamic loading conditions. Damping is excluded and applied loads are not taken in to consideration in modal analysis. This analysis was done to find out the natural frequencies and mode shapes of the bracket.

Figure shows the mode of vibration. The frequency of vibration for the particular mode is 6.292×10^1 Hz here the model is subjected to maximum deformation near the top end.

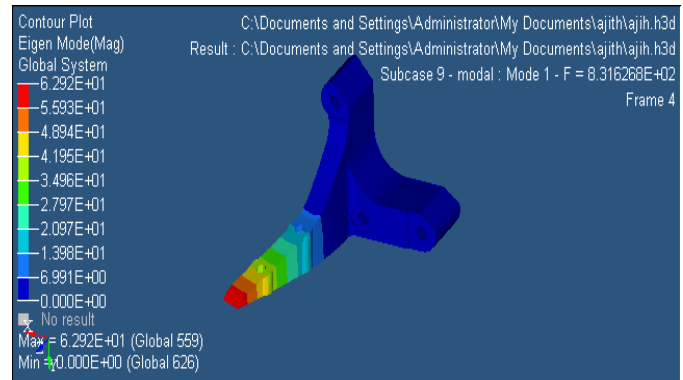


Fig 3.4: Mode of Vibration

This analysis determines the response to externally applied transient vibrations. Modal analysis gives us the form or mode shapes corresponding to the natural frequencies without considering any applied forces. The mode shapes obtained are important because they show us the direction of deflection and free amplitude of vibration for each natural frequency. [5]

4. CONCLUSIONS

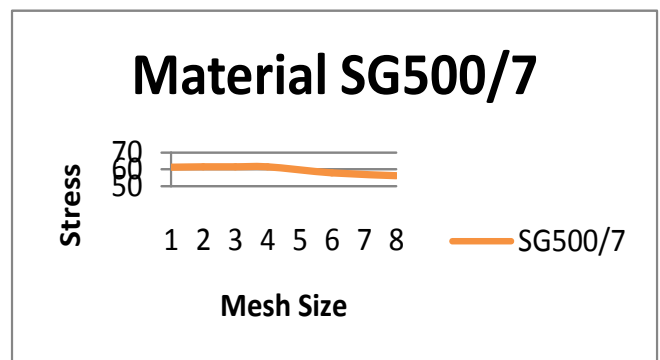


Fig-4.1: Stress v/s Mesh Size

The above graph depicts the stress v/s mesh size for the material SG500/7 for the elemental mesh size 2mm the stress is 61.39 Mpa and for the elemental mesh size 4mm the stress is 61.44 Mpa same further for the elemental mesh size 6mm is 58.01 Mpa.

Up to some extend for the elemental mesh size 4mm the corresponding stress slightly varies and it starts to change

the behaviour after the mesh size 4mm.from this point onwards as the mesh size goes on increasing the stress will decrease and the behaviour of the graph falls down as it depicts in graph.

SL. NO.	MATERIAL	COST /KG	Actual stress Mpa
1	SG 500/7	100	61.44
2	FG 150	50	61.83
3	FG 200	55	61.83
4	FG 220	58	61.83
5	FG 260	60	61.83

Chart-1: Actual Stress and Cost

The values of stresses are almost same and mainly they are within the Permissible limit but the Material SG 500/7 is having more cost than the other material when we compared as per the economic point of view.

ACKNOWLEDGEMENT

I would also like to thank Prof. Kallappa Khannukar an Prof. Ramnath Lhase department of Mechanical Engineering, Maratha Mandal Engineering College, Belgaum, Karnataka , for the help, guidance and continuous support given to me in completing my project work.

I like to thanks Dr. Ashok Hulagabali for his valuable advice.

REFERENCES

- [1] Dr.Yadavalli Basavaraj, Manjunatha. T. H "Design Optimization of Automotive Engine Mount System", International Journal of Engineering Science Invention, ISSN (Print): 2319 - 6726, Volume 2 Issue 3 || March. 2013 || PP.48-53.
- [2] P.D. Jadhav, Ramakrishna, "Finite Element Analysis of Engine Mount Bracket", IJAET ISSN No: 2349-3224,Volume 1 Issue 4 September 2014.
- [3] Prof.M.V.Kavade, Prof.D.S.Chavan, "Static Structural And Modal Analysis Of Engine Bracket Using Finite Element Anlysis", International Journal of Engineering Research & Technology, ISSN: 2278-0181, Vol. 1 Issue 10, December- 2012.
- [4] P.Lakshmi Kala1, V.RatnaKiran "Modeling and Analysis of V6 Engine Mount Bracket", International Journal of Innovative Research in Science,Engineering and Technology, Vol.4, Issue7, July2015.
- [5] Umesh S. Ghorpade, D. S. Chavan "finite element analysis and natural frequency Optimization of engine bracket", International Journal of Mechanical and Industrial

Engineering (IJMIE) ISSN No. 2231 –6477, Vol-2, Iss-3, 2012.

BIOGRAPHIES



M Tech (Machine Design), under VTU, Belgaum. Areas of interest are composites, Strength of Material and Experimental Design.



M. Tech (PhD), has a vast teaching experience of 15 years an 1 year industrial experience. Paper National level-02, International journal-04. Area of Interest are Machine Design, Composites, Experimental stress Analysis.