International Journal of Innovative Technology and Exploring Engineering (IJITEE)
ISSN: 2278-3075 (Online), Volume-9 Issue-10, August 2020

Stress Optimization of Engine Support Bracket

Paresh G Chaudhary, G.R.Selokar

***Abstract**: This paper briefs about the several facts about the Engine Bracket which plays a vital role in any Automobile. Engine bracket is sustaining heavier shocks on roads so it's being very crucial to analyses this component on stress point of view and minimize the stress to maximum possible extent. An attempt is made in this paper to reduce the stress occurring in the component by varying the materials and by changing the geometrical conditions and to find out best combination of material as well as geometry which will ultimately satisfy all engineering conditions. The design part of the component has covered with the help of CATIA software and the whole analysis part is done with the help of ANSYS software. The current paper will also cover some historical references of the relevant data and conclusions from them so that future researches can easily grab those references. Ultimately this paper will result in suggesting the best combination of material and geometry which will sustain at mentioned engineering frequency.*

Keywords: Finite Element Analysis, Stress, Stress Reduction, Non Linear FEA, Engine Mounting.

I. INTRODUCTION

Engine is one of the most significant component of a road vehicle such as car. High performance sporting vehicle has their engine supported by bracket. It plays an important role in improving the comfort & work environment of a car. The enhancement of engine bracket system has been the topic of deep interest for many ages. It is essential to design proper engine bracket for passenger vehicles. For instance such, engine bracket has been designed as a structure to support engine. Fatigue and shakings of engine bracket has been endlessly a concern which may lead to structural failure if the resultant vibrations and stresses are severe and disproportionate. Lengthy discovery to whole-body vibration in the functioning surroundings may lead to fatigue and in some cases it indemnities the car. Generally, the most essential vibration significant excitations in a car engine can be identified as ignition force; core bearing reaction forces including flywheel whirling and mass forces damper function, modified by the front-end damper; piston side forces including secondary motion; camshaft bearing reaction forces including mass forces, opening and closing impacts and bearing impacts; valve opening and closing impacts; valve train forces caused by chain/belt movement or gear drive; gear train forces inside the transmission; drive train reaction forces and moments.

Revised Manuscript Received on August 30, 2020.
* Correspondence Author
Paresh G Chaudhary, Department. of Mechanical Engineering, SSSUTMS, Sehore, Bhopal, M.P, India
Dr.G.R.Selokar, Professor and Registrar in SSSUTMS, Sehore, Bhopal, M.P, India.

It is well-known from basic Non-linear vibration theory; improvement in the vibration control can be achieved by determining the natural frequency of the engine bracket system well below the frequency band in which excitation exhibits most of the vibratory energy. It is in this environment, the advance of engine bracket can make the engine proficient in absorbing vibration. Mounting system of an automotive engine must fulfil the primary tasks such as engine movement, engine rigid body dynamic performance, and vibration isolation. The project and development of Mounting bracket by use of Ansys software to achieve the requirements for mounting system. Bounds over the development of the mounting systems because of drivability and NVH concerns, provides savings in design resources. [1] The mounting of the engine should be well constrained and the mount brackets need to be lightweight and designed to safely bear the inertial loads and maximize vibration-transmission. If the brackets have their resonance frequencies close to the operating engine frequencies, then the large amplitude of vibration may cause its fatigue failure or breakage, thus reducing its estimated or desired life. [4]

II. LITERATURE SURVEY

M.V. Aditya Nag had inspected the procedure for material collection and computational investigation on Double Overhead Camshaft Engine i.e. DOHC V16 Engine's Mounting Support Using cross-platform finite element analysis, solver and Multiphysics simulation software COMSOL Multi physics Software. In this research he carried out experimentation for weight reduction of engine mounting bracket by keeping performance constant. Modelling and analysis of the Engine Mounting Bracket had done according to the standards given by the Yun he Yu and V. Rant Kiran by applying force of 150N and keeping fixed at other end. The Similar bracket had further processed by using material selection with the help of COMSOL Multi physics software. After all Experimentation Weight reduction found to be 60 % of the original without hampering the performance. The materials focused for conducting the analysis were GCI and AL Sic MMC composite material. COMSOL Multiphysics 4.2 version software is helped in carrying out computational testing on the component at isotropic state through the application of Thermo-mechanical Vibration Analysis. [2]

M.V. Aditya Nag in his research on Topology optimization of engine mounting bracket had achieved the topology optimization of the model using OPTISTRUCT module of Hyper works 9 with effective use of Design for Manufacture and Assembly. The maximum stress observed on chassis because of unbalanced engine forces and uneven engine balancing forces at the idling speeds, torques due to reciprocating parts and shaking forces. And proposed that these are the main reason for the vibration along with that these vibration are disturbing and creating noise and it is very difficult to find out the origin of the same.

The conditions for analysis were taken as Force of 150N and constant frequency at 1000 N. [3]

III. BRACKET DESIGN

A. Specification

The geometry is created using the CATIA software and further analyzed with Ansys.

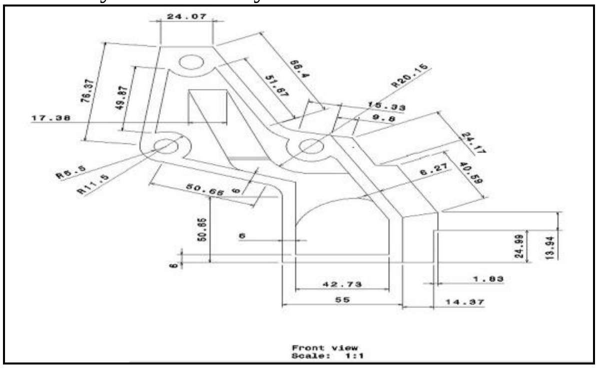


Figure 1: Upper Part Front View.

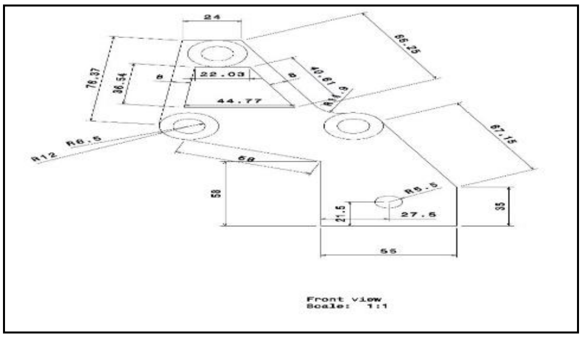


Figure 2: Bottom Part Front View.

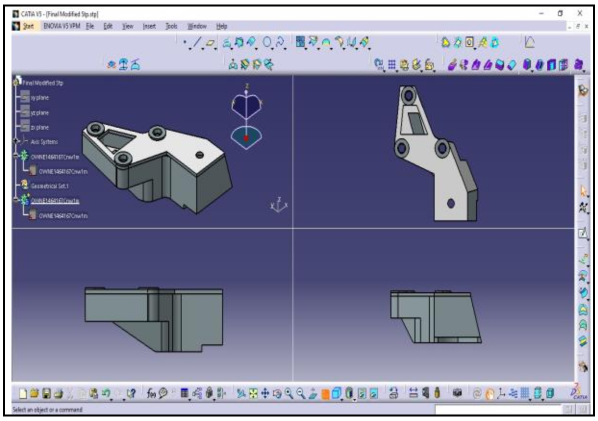
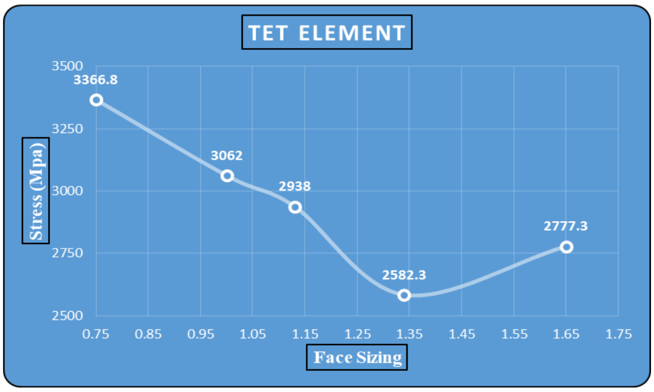
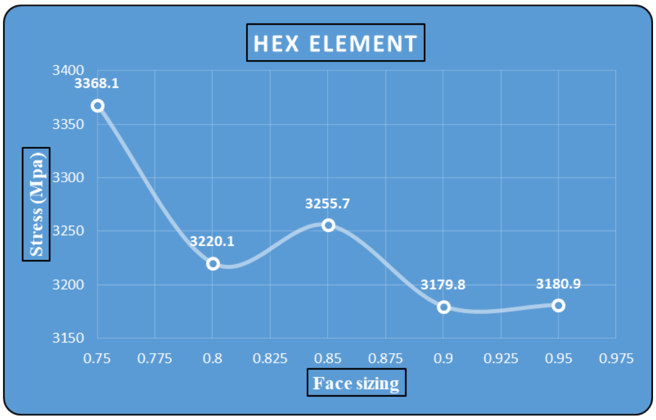


Figure 3: Catia Assembled Exploded view.



Graph.1: Stress v/s Face Sizing of TET Element.



Graph.2: Stress v/s Face Sizing of HEX Element.

From aforesaid graphs it can be clearly seen that Hex dominant element shows convergence at upper body sizing-2, lower body sizing-1 & face sizing 0.85. Hence finalizing and continuing further analysis with face sizing 0.85 we get its relevant models. The similar model is considered for further analysis with remote force of 1000N, which is equally divided on respective holes. [8]

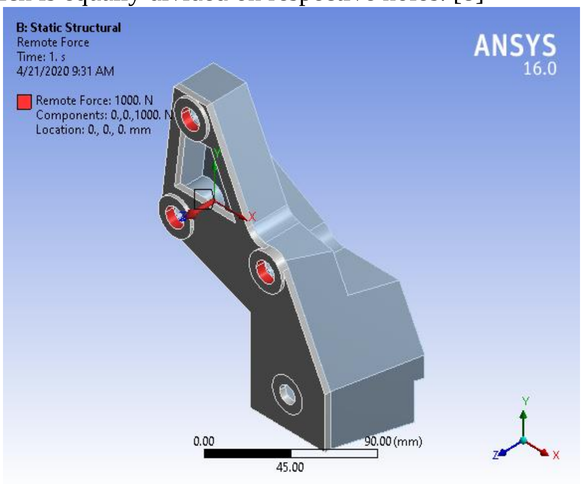


Figure 4: Finalized Model with Remote Force

Retrieval Number: J74550891020/2020©BEIESP
DOI: 10.35940/ijitee.J7455.0891020
Journal Website: www.ijitee.org

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

International Journal of Innovative Technology and Exploring Engineering (IJITEE)
ISSN: 2278-3075 (Online), Volume-9 Issue-10, August 2020

IV. RESULT

With the selected elemental constrains the analysis is further continued and the results are obtained as follows for the regular stress values and linearized stress. As the component contains two major parts with different dimensions hence by varying the dimensions stress values are obtained and that are compared for obtaining the solution. Three materials are used as Aluminum, Magnesium, and Gray Cast Iron for comparison.

Table- I: Thickness variation of top part and bottom part.

Sr.No	Base Plate Size(mm)	Inner Thickness Reduction by(mm)	Material	Eq.Stress (Mpa)	Eq.Stress-2 (Mpa)
1	1	1	Al	6249.4	1560
			Mg	5821.9	1450
			GCI	5959.2	1990
2	1	2	Al	6221.3	1590
			Mg	6303.7	1570
			GCI	6136.2	2050
3	1	3	Al	6315.1	1575
			Mg	6495.5	1610
			GCI	6441.1	2150
4	1	4	Al	5874.2	1465
			Mg	5881.3	1460
			GCI	5894	1980
5	2	1	Al	1772.7	445
			Mg	1981.7	490
			GCI	1653.6	555
6	2	2	Al	1766.9	440
			Mg	1801.8	450
			GCI	2062.4	700
7	2	3	Al	1753.2	445
			Mg	1863.4	460
			GCI	2179.4	730
8	2	4	Al	1618.1	400
			Mg	1646.3	410
			GCI	1600.1	535
9	3	1	Al	838.15	210
			Mg	831.53	205
			GCI	930.39	315
10	3	2	Al	837.91	210
			Mg	844.42	210
			GCI	896.29	300
11	3	3	Al	931.12	235
			Mg	885.5	220
			GCI	968.06	325
12	3	4	Al	815.7	255
			Mg	914.86	230
			GCI	887.28	300
13	4	1	Al	571.98	145
			Mg	564.51	140
			GCI	591.41	200
14	4	2	Al	610.33	155
			Mg	602.01	150
			GCI	631.88	210
15	4	3	Al	653.21	165
			Mg	645.5	160
			GCI	673.08	225
16	4	4	Al	706.32	180
			Mg	686.73	170
			GCI	758.29	255

V. CONCLUSION

Acutely analysis has been carried out with the help of Ansys software and stress is observed. Out of all combinations the set of 4mm base and its internal variation gives the minimum stress. The 4mm base with 4 mm reduction in thickness fails in engineering frequency criteria, rest all can be picked up for further analysis. Currently out of all selected materials, Magnesium material shows the lowest stress among all. Though currently magnesium is showing minimum stress in the engine bracket, further stress reduction can be achieved by implementing the Titanium Material. It is always suggested that materials are chosen by considering every aspect related to it such as availability, durability and manufacturing process and cost involved in the process.

REFERENCES

1. Kim K., Choi I., Design optimization analysis of body attachment for NVH performance improvements, SAE technical paper series 2003-01-1604.
2. M.V. Aditya Nag, Material Selection and Computational Analysis on DOHC V16 Engine's Mounting Bracket Using COMSOL Multiphysics Software. Proceedings of the 2012 COMSOL, conference in Bangalore.
3. M.V. Aditya Nag, Topology optimization of engine mounting bracket, in HTC Technical Presentations 2012, hyper work technology conference
4. Umesh S. Ghorpade, D. S. Chavan, Vinaay Patil, Mahendra Gaikwad, "finite element analysis and natural frequency optimization of engine bracket" IJMIE, Vol-2, Iss-3, Page no. 1-6, 2012.
5. P.D.Jadhav, Ramakrishna, "Finite Element Analysis of Engine Mount Bracket", International Journal of Advancement in Engineering Technology, Management & Applied Science, ISSN 2349–3224, Volume-1 Issue -4,Sept.2014, pp-1-10.]
6. Haval Kamal Asker, comparison of the mechanical properties of different models of automotive engine mounting, ARPN Journal of Engineering and Applied Sciences, VOL. 8, NO. 6, JUNE 2013.
7. Abdolvahab Agharkakli, Dig Vijay Pradip Wagh, Linear Characterization of Engine Mount and Body Mount for Crash Analysis, International Journal of Engineering and Advanced Technology (IJEAT), ISSN: 2249 – 8958, Volume-3, Issue-2, December 2013.
8. Analysis of Engine Support Bracket for Frequency and Stress Optimization, International Journal of Innovative Technology and Exploring Engineering (IJITEE), ISSN: 2278-3075, Volume-9 Issue-6, April 2020. DOI: 10.35940/ijitee.F4281.049620,PP-996-1000.

AUTHORS PROFILE

Paresh G Chaudhary is PhD Scholar in Dept. of Mechanical Engineering, SSSUTMS, Sehore, Bhopal, M.P, India and is also author of International book also filed Patent in Mechanical Domain. He has published various papers in various domains.

Dr.G.R.Selokar is Professor and Registrar in SSSUTMS, Sehore, Bhopal, M.P, India. Has contributed in various domains of Engineering with full dedication to improve the quality of research, also published many research papers with ultimate outcomes & sets standard for new era of Technology.