https://dx.doi.org/10.22161/ijaers.4.3.9 ISSN: 2349-6495(P) | 2456-1908(O)

Modal Analysis of Engine Supporting Bracket using Finite Element Analysis

A.S. Adkine[1], Prof.G.P.Overikar[2], Prof. S .S. Surwase[3]

[1]PG Student Department of Mechanical Engineering, Shree Tuljabhavani College of Engineering, Tuljapur, India
[2]Assistant Prof. Department of Mechanical Engineering, Shree Tuljabhavani College of Engineering, Tuljapur, India
[3]Associate Prof. Department of Mechanical Engineering, Shree Tuljabhavani College of Engineering, Tuljapur, India

Abstract— *The Engine supporting bracket plays a vital role by reducing noise, vibration and harshness. The current work describes the finite element approch for modal and static structural analysis of engine supporting bracket. CAD model of engine supporting bracket was created in CATIA software and same was analyzed for stress and vibration analysis using ANSYS workbench 15.0. Both initial design and modified design was compared for output responses in terms of equivalent Von-Mises stress, deformation and strain energy absorbed. In modal analysis, bracket was considered for vibration studies. The sole aim of modal analysis was to check whether the self excitation frequency of engine supporting bracket was less than natural frequency. Four alternative materials (Gray C.I., Aluminum alloy, Magnesium alloy and ERW-1) were analyzed. Stress analysis results suggest that deformation and Von-Mises stresses observed in EEW-1 materials was less (0.495 mm and 164.87 MPa). Further natural frequency of modified design was found to be 257.83 Hz which was well within the range below self-excitation frequency and less than the natural frequency (268.59 Hz) of initial design. It was found that aluminum bracket limit its use for the said application due to greater deformation and less stiffness. Magnesium bracket can be the option to ERW-1 steel for the Engine supporting bracket application but it cannot be deployed as it is highly susceptible to corrosion. From the results, it can be concluded that ERW-1 material best suit the requirement of the desired application and can be deployed with some safety standards.*

Keywords— Engine Supporting bracket, Finite element analysis, Modal Analysis.

I. INTRODUCTION

Engine supporting assembly includes support members which are attached to a main frame vehicle body. A good mounting system separates engine generating vibration from vehicle body and suppresses the effect uneven road surface inputs to the vehicle driver. Noise, vibration and harshness are important vehicles characteristics and decrement in these characteristics motivates manufacturer for achieving customer satisfaction. The automotive engine bracket is very important due to different aspect of vehicle performance. The finite element model of engine mount is created using CAD & Hypermesh and simulation of the engine mount is carried out using LS-Dyna. Based on the finite element model created, the mountings frequency response function curves are determined and multi-dimensional effects in the mounts responses are observed. Engine mounts in which the design of four arm symmetry engine mount curve is obtained from LS-Dyna approach follows exactly the experimental test curve and also this design has the highest natural frequency amongst all of design iterations. The engine mounts time dependent response study is compared to the ones obtained using a damping material model suggested by the automotive constructors. The results indicated that the rubber used in the engine mount had increased the frequency from 1.2Hz (Basic design) to 1.8Hz (four arm symmetry) [1].
Model of Engine Mount bracket assembly was performed using FEA and modal analysis techniques. Mg alloy bracket has highest natural frequency followed by Al alloy & Cast Iron gives less frequency so it is rejected. The results are compared with experimental results and it was found that the bracket manufactured with Mg alloy gives optimized frequency. In addition, Al can be used in few applications but CI alloy gives minimum frequency. Mg and Al are preferred [2]. 3-D modeling of the engine mount has been designed using CATIA V5 software. The computational testing on the component is done at isotropic state through the application of Thermo-mechanical Vibration Analysis using the Comsol Multiphysics 4.2 version software. The main objective of the research is to reduce weight without changing impact of the component. The Engine mounting bracket is been used to reduction of the vibration created by the engine. The results obtained states that 60% of the weight reduction is done to the component. The weight is reduced 60% through the usage of the optimized AlSiC MMC composite material made component. The future work focuses on the cost reduction of the material without varying the weight of the component [3].

Alternative materials for Static Structural Analysis of Bracket Using Materials are Cast Iron, Aluminum & Magnesium & Equivalent (von misses) stresses of

https://dx.doi.org/10.22161/ijaers.4.3.9 *ISSN: 2349-6495(P) | 2456-1908(O)*

Aluminum, Magnesium. Modal analysis of bracket using materials Cast Iron, Aluminum and Magnesium was done in earlier stages. Experimental Analysis for Natural Frequency Analysis & Static analysis of modified Bracket with Aluminum material. After reducing the thickness of the Bracket the von-misses stresses coming on the bracket are well within the yield strength (280 MPa) of the Aluminum. So, after selecting the light weight material Aluminum & reducing its thickness by 2 mm the reduction in the mass up to 0.43 Kg [4]. It has been developed durability test on vehicles in the end- user environment to reduce failure and warranty cost in the end- user hands. The failure analysis of muffler mounting brackets of three-wheeler vehicles was done for durability test. Cracks at the weld location between the engine cradle and bracket were observed in all the vehicles at an observed in all the vehicles at an average distance of 10,000 km. Many possible causes of the failure are identified using fishbone diagram.

Further investigations were carried out on the design using finite element method (FEM). FEM model developed for the engine cradle assembly in which assembly in which engine and muffler were modeled as point mass. Results show high magnitude of stresses and strain energy at the weld location. Results show high magnitude of stresses and strain energy at the weld location. Analysis of the design suggested that bracket was acting as a cantilever beam with one plane welding mounting on the engine cradle modified design through eliminated the above failure shifted the failure mode to the bush –bracket region[7].

II. METHODOLOGY

Firstly, the theoretical study of engine supporting bracket is done. The main purpose of the bracket is to supports the engine. To modify the existing design of engine supporting bracket and to carry out static structural analysis for various materials Gray C.I., Aluminum alloy, Magnesium alloy and ERW-1. Modal Analysis of engine mounting bracket for determines the modified design has a natural frequency of component lower than the self-excitation frequency. Best material is selected for engine supporting bracket.

2.1 Position and Design of Engine Supporting Bracket

Fig.1: Position of ESB

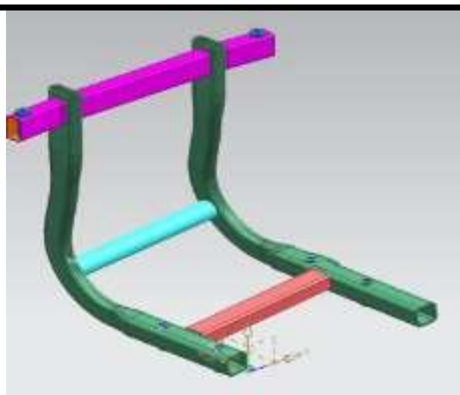


Fig.2: 3-D CAD model of ESB Initial design

Engine is most essential or an important component of automotive vehicle and which is rest on this structural L-shaped frame is known as Engine supporting bracket and these are connected to the skeleton of vehicle. Hence, during its operation engine is produced undesirable vibration and road roughness can directly transmit through the frame to bracket. Due to that reason discomforts ride as well as passenger or might even damage the bracket. Engine supporting bracket is to resist the vibration response to engine vehicle. Vehicle is running on the road dynamic response of the bracket to calculate the Static structural characteristics of engine supporting bracket.

2.2 Finite Element Analysis

Finite element analysis is most important software tool that can solve many kinds of engineering problem with high degree of precision as necessary. FEM is computational technique ANSYS 15.0 is used to analyses a wide range of engineering problem.in this analysis a complicated shapes or complex region define a continuum is discretized into simple geometrical shape called as finite element.

Modern mechanical design involves Complex or complicated shapes, sometimes made of different materials that as a whole cannot be solved by existing mathematical tools. Engineers need the FEA to evaluate their designs. The process of dividing the complicated shapes model into small pieces is called **meshing**. The behavior of each element is well-known under all possible support and load scenarios. Unknown value of element is called as **nodes.**

2.2 General Procedure of FEA

Any Analysis Involved main three Step

Pre processing

- Create or import model geometry
- Mesh the Geometry

Solution

- Apply the load
- Solve

Post processing

- Review Results
- Check the Solution.

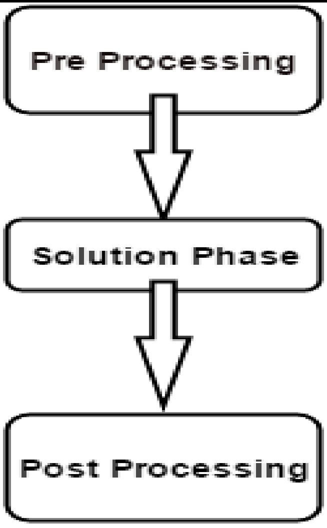


Fig.3: Typical FEA Procedure

2.3 Alternative Materials

Designed will test for different Materials. They are gray cast iron; Aluminum Alloy, Magnesium Alloy & ERW-1 best suited material are found out.

Gray cast iron: These are alloys of iron, carbon and silicon, containing more than 2% carbon (as flake graphite), up to 3% silicon and less than total 1% of alloying elements (mainly chromium, copper, magnesium, molybdenum, nickel, phosphorous, silicon, sulphur, titanium and vanadium). They are used to make heavy machine beds, camshafts, cranks, elevator buckets, and railway wheels

Al Alloy: It is a soft silvery white metal with about one third of density of ferrous metals and low tensile strength in its pure form. It is an excellent conductor of heat and electricity and has corrosion resistance in most environments including seawater, oils and many chemicals. To improve its strength, hardness and fluidity, silicon, copper, magnesium and zinc are added. Aluminum has only about one third the density of steel and the most commercial aluminum alloys posses substantially higher specific strength than steel. A vehicle weight reduction would not only result in higher oil savings, but also gives a significant reduction in emission. For these reasons there is preference to use more aluminum and replace steel in automotive applications.

Magnesium Alloy: Magnesium is the lightest of all metals used as the basis for constructional alloys. It is this property due to which automobile manufactures has to replace denser materials, not only steels, cast irons and copper base alloys but even aluminum alloys by magnesium base alloys. The advantages of magnesium alloys are listed as follows, lowest density of all metallic constructional materials. It posses high specific strength, good cast ability, which suitable for high pressure die casting good welding properties, higher corrosion resistance. Also compared with polymeric materials it posses better mechanical properties, better electrical and thermal conductivity and it is recyclable.

ERW-1: Electrical resistance welded steel tube & seamless tube are often used in application with automobiles ERW to their convention properties should also innovation which reduces cost, weight & to their high strength. Mechanical Properties of ERW-1are

Table.1: Mechanical Properties of ERW-1

IS 3074 ERW-1	
Young's modulus N/m^2	2.1 e+10 N/m^2
Poisson's ratio	0.29
Density Kg/m^3	7800 Kg/m3
Yield strength N/m^2	2.4e+8 N/m^2

2.4 Fishbone Diagram

The study and analysis of the problem present here would be easy to visualize by adopting the cause effect method for fault finding. The Fishbone diagram provides a visualization of a problem with respect to factors affecting it. Such as, Measurement, Methods, Man, materials, design, etc. Arranging all such factors in the form of the fishbone diagram, provides a reasonable and logical guideline for analysis and fault finding. The diagram present here is the fishbone diagram constructed for the component being studied. Following points are considered:

https://dx.doi.org/10.22161/ijaers.4.3.9 **ISSN: 2349-6495(P) | 2456-1908(O)**

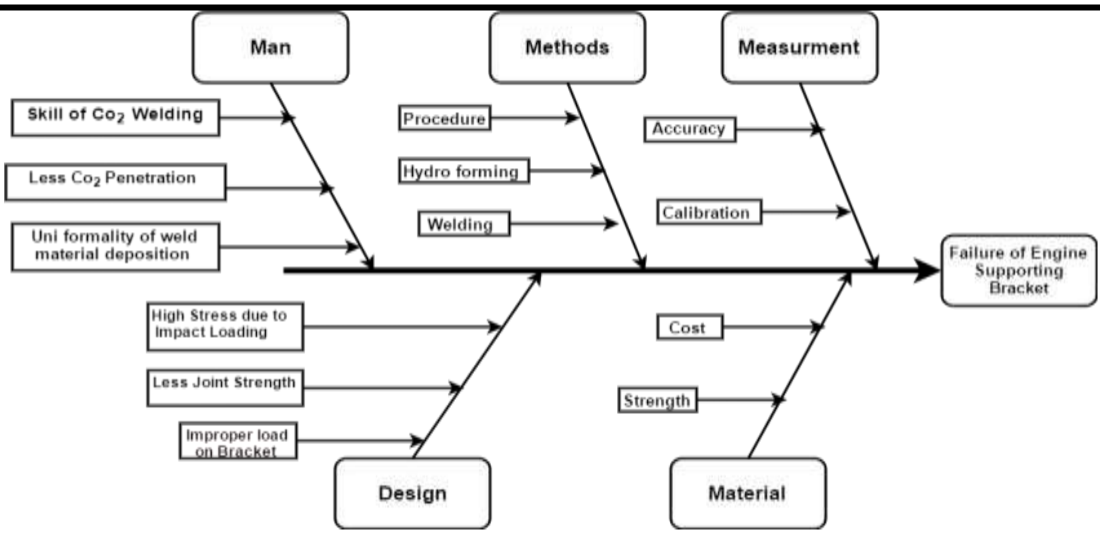


Fig.4: Fishbone diagram

Many possible causes are identified of failure system using fishbone diagram.

Man: Human being is the factor which would results in welds related defects like incomplete penetration between joints, incomplete fusion, less Co_2 penetration undercutting and bad weld design.

Material: Here materials being used for production of component are decided. The criterion for selection of material is the yield strength .In that sense low yield strength of material will lead to insufficient strength and poor weldability.

Methods: This factor summarizes the setup and different production processes used for manufacturing the component. Process like hydroforming & welding are performed to contribute for manufacturing. Any wrong sequence in Procedure can lead to defects described above.

Measurement: this factor deal with various measuring system used for calibration of instrument. Tolerances and allowances are to be studied. Any misfortune in inspection would definitely affect the accuracy and ultimately the final quality of the component.

Design: The major problems associated with factor design are bracket acting as cantilever beam, high thermal stress due to radiation effect, ergonomic consideration; improper load on bracket has strong influences on weld failure. Meanwhile the above factors will govern the range of dimensions of component.

III. RESULTS AND DISCUSSION

3.1 Static Structural and Modal Analysis of Initial Model

The analysis of engine supporting bracket is done with the help of FEA commercial software ANSYS workbench. First stage prepares the CAD model of Engine supporting bracket using CATIA Software & model importing into the geometry in ANSYS workbench. Further that the material selection is done from in engineering data sheet of FEA package software. Meshing or discretization done as hex mesh a solid element is generated is to required small pieces. Static structural analysis engine supporting bracket weight of the engine is distributed in equilibrium condition. Stress analysis of engine mounting bracket upper end is connected to main frame of the vehicle and then the engine weight load is applied on bracket is 800N & silencer weight is act at silencer rest is 20N as a point load. Process flow of ANSYS Workbench.

- Preparation of CAD model.
- Import in ANSYS Workbench 15.0.
- Define Material Properties (young's modulus, poison ratio, density, yield strength) Discretization (Mesh generation).
- Apply Loading Condition (Define Boundary Condition).
- Solution (Static Structural & to find deformation, stress, strain energy Results)
- Modal Analysis (Find out Frequency Hz with mode results)

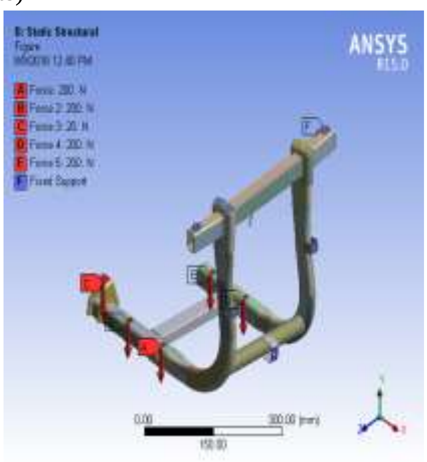
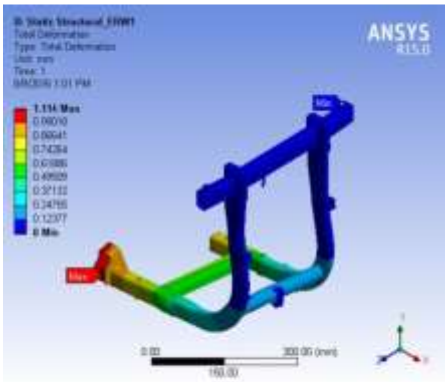
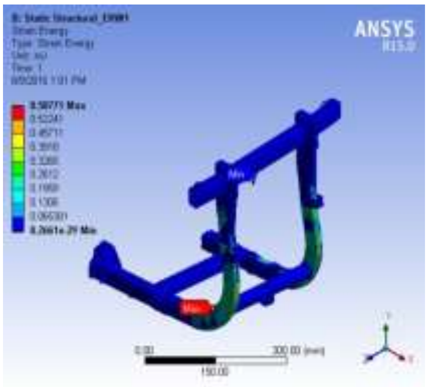


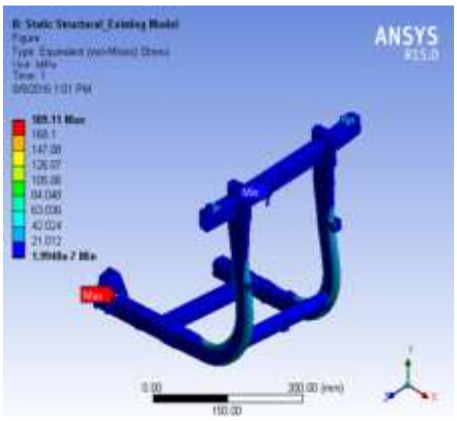
Fig.5: Boundary condition of initial design

https://dx.doi.org/10.22161/ijaers.4.3.9 *ISSN: 2349-6495(P) | 2456-1908(O)*

Fig.6: Meshed Model of ESB

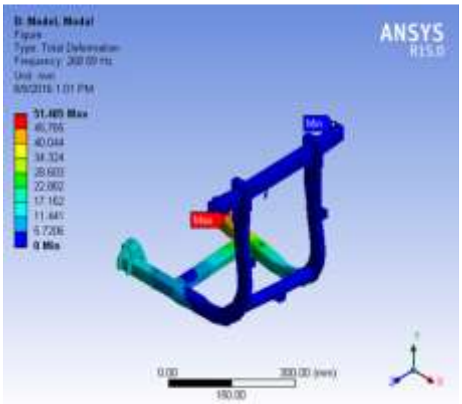






Static structural Analysis of Engine Supporting Bracket ERW-1. (Initial Design)	
Total Deformation mm	1.114
Von-Mises Stress Mpa	189.11
Strain Energy mJ	0.58771

Fig 7 Total deformation, Equivalent (Von- Mises) Stress & Strain Energy for Initial design of ESB of Initial Model



Mode	**Frequency [Hz]**
1	148.12
2	167.32
3	268.59
4	500.92
5	630.92
6	700.3

Fig.8: Modal Analysis of Initial Engine supporting Bracket.

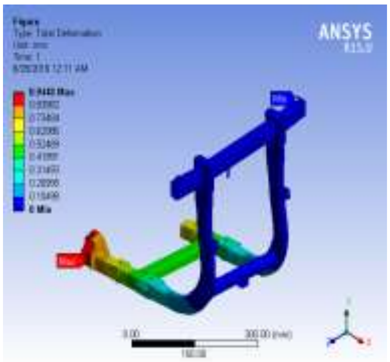
Static structural of Engine supporting frame using ERW-1 material for initial design analysis by ANSYS Workbench R15.0 and getting results of total deformation is 1.114 mm, Equivalent (Von- Mises) Stress is 189.11 MPa & Strain Energy is 0.58771 mJ**.**

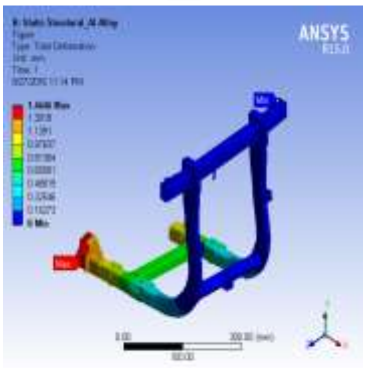
3.2 Static Structural and Modal Analysis of Modified Design

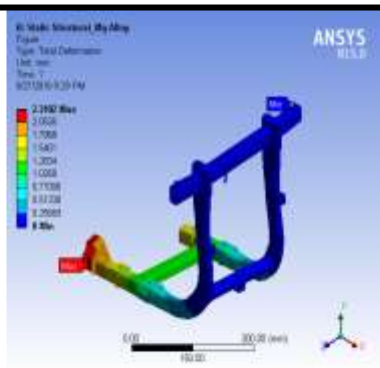
Static Analysis deals with the conditions of the equilibrium of the bodies acted upon by forces .A Static analysis is used to determine the total deformation in mm with alternative materials Gray cast Iron, Al Alloy, Mg Alloy & ERW-1.

https://dx.doi.org/10.22161/ijaers.4.3.9 ISSN: 2349-6495(P) | 2456-1908(O)

Fig.9: Modified design of engine supporting bracket for 3-D Model







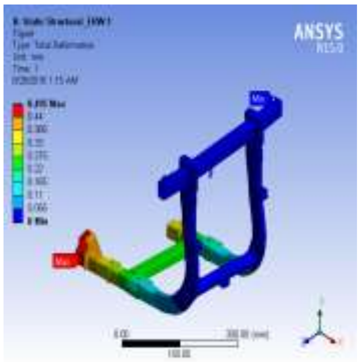
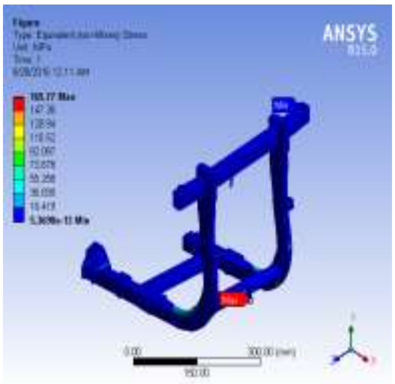
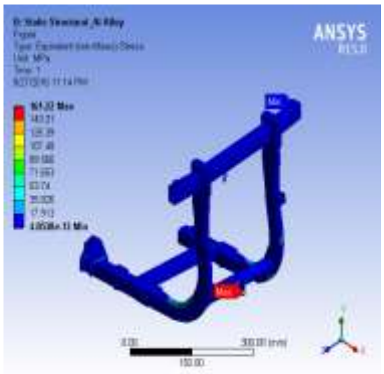
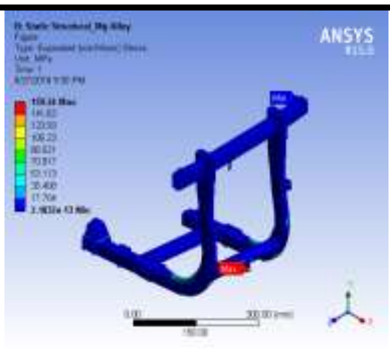


Fig.10: Comparative Study of displacement between Gray C.I., Aluminum, Magnesium and ERW-1.





https://dx.doi.org/10.22161/ijaers.4.3.9 ISSN: 2349-6495(P) | 2456-1908(O)



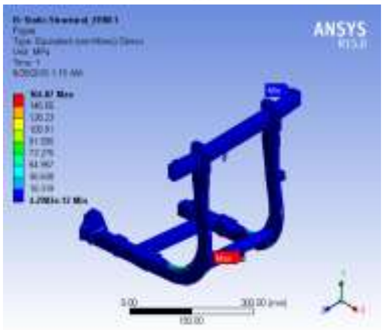
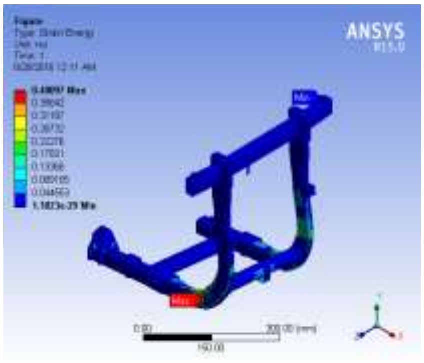
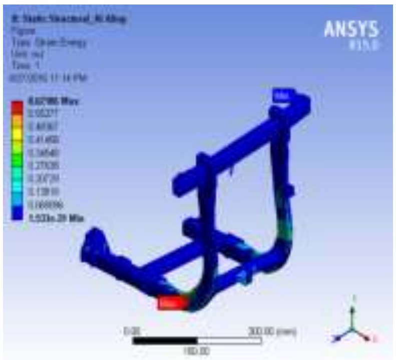
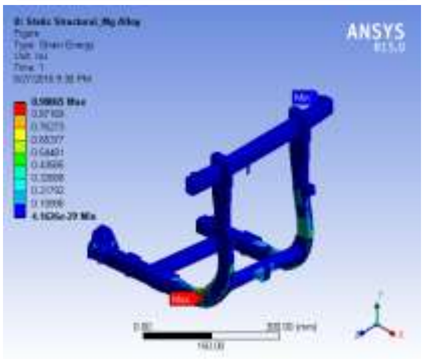


Fig.11: Equivalent Von-Mises Stress observed during stress analysis in Cast iron, Aluminum, Magnesium and ERW-1 Engine Supporting bracket respectively







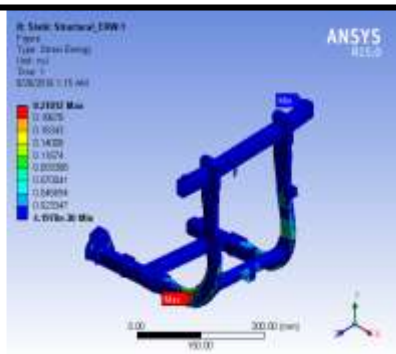


Fig.12: Strain Energy observed during stress analysis in Cast iron, Aluminum, Magnesium and ERW-1 Engine supporting bracket respectively.

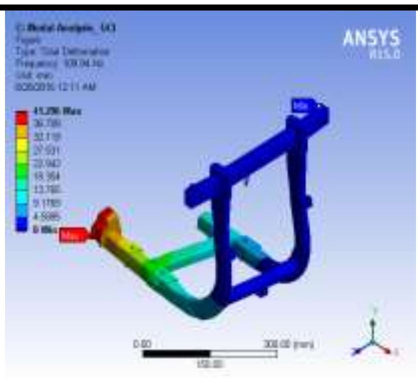
The figure above shows the comparative study of Strain Energy observed during stress analysis in Cast iron, Aluminum, Magnesium and ERW-1 Engine. This Cast iron, Aluminum, Magnesium and ERW-1 materials were tested for alternative materials for suitable application and results were summarized. Stress induced in magnesium material was found out to be lower than ERW-1 and aluminum. It was probably due to high specific strength of magnesium and its ability to get denser. It can be anticipated that ERW-1 is the best for the desired application (figure 12).

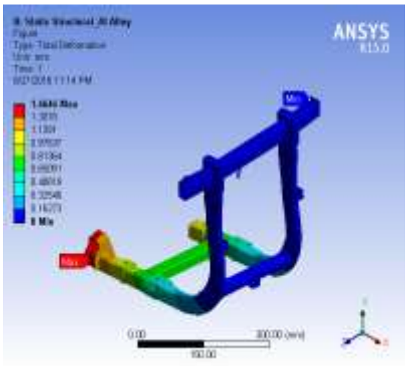
Table.2: Static Structural of Engine Supporting bracket Modified Design

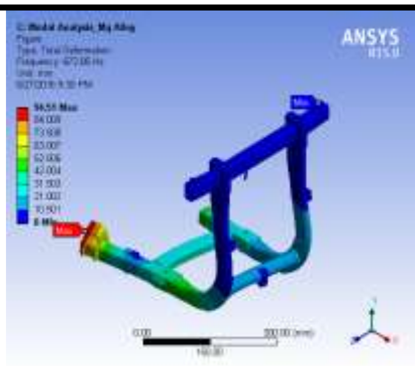
Materials	*Gray C.I*	*Al Alloy*	*Mg Alloy*	*ERW-1*
Total Deformation mm	*0.9448*	*1.4646*	*2.3102*	*0.495*
Equivalent (Von-Mises) Stress MPa	*165.77*	*161.22*	*159.34*	*164.87*
Strain Energy mJ	*0.40097*	*0.62186*	*0.98065*	*0.21012*

3.3 Modal Analysis of Modified Model

The Stress analysis of Engine Supporting bracket was carried out using ANSYS. Modal analysis was done for obtaining the different frequencies for Cast Iron, Aluminum Magnesium, ERW-1, and Table-1 Frequency Cast Iron vs. Aluminum vs. Magnesium vs. ERW-1.







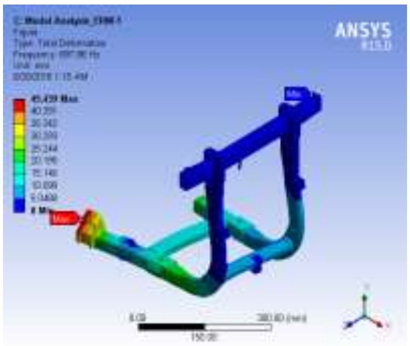


Fig.13: Frequency of various materials as Cast Iron vs. Aluminum vs. Magnesium vs. ERW

Table.3: Modal Analysis of Engine Supporting bracket Modified Design

Mode	**GCI in Hz**	**Al Alloy in Hz**	**Mg Alloy in Hz**	**ERW-1 in Hz**
1.	109.94	142.06	140.06	145.86
2.	124.09	160.38	158.32	164.62
3.	194.25	251.63	248.62	257.83
4.	368.96	477.6	471.68	489.68
5.	457.39	592.33	585.14	607.07
6.	525.71	681.05	672.85	697.86

As the mode of frequency was changed, initially no change was observed in ERW-1 and Magnesium material. This is probably due to the varying composition and superior mechanical properties of ERW-1 material.

Table.4: Static Structural Analysis of engine supporting bracket

Parameter	**Initial Design**	**Modified Design**
Total deformation mm	**1.114**	**0.495**
Von Mises stress Mpa	**189.11**	**164.87**
Strain energy mJ	**0.58771**	**0.21012**

Results are in tabulated form of Comparative study of initial and modified design for ERW-1.Initial model having less % of carbon was having more von-mises stress may fail in design criteria. But the design was fabricated and analyzed for new existing alternative materials aluminum, magnesium and ERW-1. It was found that the stress generated in ERW-1 material was less than initial design. Again the frequency and vibration studies through Fast Fourier Transform show good little convergence of frequency well within stipulated permissible limit. So, it reveals in natural frequency of Engine Supporting bracket of modified design matches is less than self-excitation frequency. It can be anticipated from the above analysis that the modified Engine Supporting bracket is safe and suitable for further operation**.**

https://dx.doi.org/10.22161/ijaers.4.3.9 ISSN: 2349-6495(P) | 2456-1908(O)

Table.5: Comparative Study of Initial and Modified Design

Mode	Initial Design		Modified Design	
	Deformation mm	**Frequency Hz**	**Deformation mm**	**Frequency Hz**
1.	31.974	148.12	39.651	145.86
2.	34.628	167.32	32.521	164.62
3.	51.485	268.59	47.839	257.83
4.	35.781	500.92	34.254	489.68
5.	34.976	630.92	32.792	607.07
6.	45.02	700.3	45.439	697.86

Mode shape is obtained were similar to that initial design of bracket. Thus change only the values of natural frequencies and amplitudes. First six frequencies are shown are called as Natural frequencies. Since applied load and damping effect are ignored in this analysis. A static structural analysis is required pre stressed modal analysis. This modal analysis was done to find out the vibration characteristic of the structure or particular component in the form of the natural frequency and mode shape of the bracket. Results are shown in tabulated from with comparison of all frequencies values and it is observed that third mode of vibration and here bracket to subject to maximum deformation at the end section of bracket.

IV. CONCLUSIONS

- Evaluation of modified engine mounting bracket was done using static structural analysis and modal analysis.
- It was found that, for the modified design deformation was 0.4950 mm with equivalent von-Mises stress 164.87 MPa which was very less than initial design with 1.14 mm deformation and equivalent von-Mises stress 189.11 MPa.
- Further natural frequency of modified design was found to be 257.83 Hz which was well within the range below self-excitation frequency and less than the natural frequency (268.59 Hz) of initial design.
- It was found that aluminum bracket limit its use for the said application due to greater deformation and less stiffness.
- Magnesium bracket can be the option to ERW-1 steel for the Engine supporting bracket application but it cannot be deployed as it is highly susceptible to corrosion.
- From the results obtained for initial design and modified design, it can be concluded that ERW-1 material best suit the requirement of the desired application and can be deployed with some safety standards.

REFERENCES

[1] Dr. Yadavalli Basavaraj, Manjunatha.T.H, "Design Optimization of Automotive Engine Mount System" IJESI, (2013), vol.2 Issue 3 pp 48-53.

[2] Pavan B. Chaudhari, Dr.D. R. Panchagad "Comparison of Magnesium, Aluminium and Cast Iron to obtain Optimum Frequency for Engine Bracket using Finite Element Analysis" IJERA, ISSN: 2248-9622 Vol. 2, Issue 5, September- October 2012, pp.1016-1020.

[3] M.V. Aditya Nag "*Material Selection and Computational Analysis on DOHC V16 Engine's Mounting Bracket Using COMSOL Multiphysics Software"* Excerpt from The Proceedings of Comsol Conference in Bangalore 2012.

[4] Mr. Pramod Walunje and Prof. V.K. Kurkute, *"Optimization of Engine Mounting Bracket Using FEA"* (2013), Indian Journal of Research, Volume 2, pp 72

[5] Umesh S. Ghorpade, D. S. Chavan, Vinaay Patil and Mahendra Gaikwad, *"Finite Element Analysis and Natural Frequency Optimization of Engine Bracket"* (2012) IJMIE, ISSN No. 2231 –6477, Vol-2 pp 1-6.

[6] Jasvir Singh Dhillon, Priyanka Rao, V.P. Sawant, *"Design of Engine Mount Bracket for a FSAE Car Using Finite Element Analysis"* Int. Journal of Engineering Research and Applications, (2014), vol. 4, Issue 9, pp 74-81.

[7] Senthilnathan Subbian, O.P. Singh, Srikanth K. Mohan, Arockia P, Jeyaraj, *"Effect of muffler mounting bracket design on durability"* Engineering Failure Analysis 18 (2011) 1094-1107.