https://dx.doi.org/10.22161/ijaers.4.5.24 ISSN: 2349-6495(P) | 2456-1908(O)

An Experimental Investigation and Modal Analysis of an Engine Supporting Bracket

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*— *Engine supporting bracket playsa vital role in improving the ride comfort and work environments of the vehicle. It is anticipated that Improvement in vibration control can be achieved by the determination of natural frequency of engine supporting bracket system and must be less than self-excitation frequency range.This Paper describes higher stresses induced and its critical region on the component. In static structural analysis design is suggested against deformation, equivalent stress andstrain energy using finite element analysis concept. Existing design is modified and provided with the isolation effectsand on setstress distribution from critical region which concentrates on strengthened body part was suggested. Further scope of this work involves in carrying out vibration study using fast Fourier transform (FFT) analyzer.*

Keywords—Engine bracket, FEA, Static Analysis, Natural Frequency.

I. INTRODUCTION

An engine supporting bracket is very important component of automotive transmission system. An engine is generated undesirable vibration to developing power & transmitted to the skeleton of the vehicle through the supporting members. Engine is rest on the supporting member and it's attached to vehicle body. In the automotive engine bracket is very important in different aspect of vehicle performance. It is significant study of which have depth investigated to understand failure of engine supporting bracket is mainly due to generated crack at high level stress region.It has designed frame work to support the engine. The goal to achieving to minimize stress level on targeted area it means that provided strength on this region or increases negligible thickness. Thus,the process to develop design produce the best structuralcharacteristic and optimum natural frequency performance of engine supporting bracket.

II. METHODOLOGY

Firstly, theoretical study of engine supporting members is done. The main purpose of engine supporting bracket is to supports the main body of vehicle skeleton. An engine which is rest in balanced condition undergoes undesirable vibration induced by engine and tires from uneven road surface. Many possible causes are identified of failure supporting system using Fishbone diagram and modification are identified. The works focused in this study to find optimum frequency of the bracket of two different design models. Best design selected with suited material and same boundary condition.

III. ROOT CAUSE FAILURE ANALYSIS

There is not a single defied methodology Root cause failure analysis; however, there are various ways, tool, processes& philosophies of Root cause failure analysis. In its broader sense it is used to identify, rectify and eliminate component failures & it is most effective when subjected to breakdowns. Even though the failure appears to be contributed by various factors initially, there is always one root cause of the failure. We adopted fishbone diagram methodology to arrive at the logical conclusion at root cause of failure. The diagram presented here is the fishbone diagram constructed, for the component being studied. Following points are considered:

Man: Human being is the factor which would results in weld related defects like incomplete penetration between joints, incomplete fusion, less Co_2penetration 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

https://dx.doi.org/10.22161/ijaers.4.5.24 *ISSN: 2349-6495(P) | 2456-1908(O)*

performed to contribute for manufacturing. Any wrong sequence in Procedure can lead to defects described above.

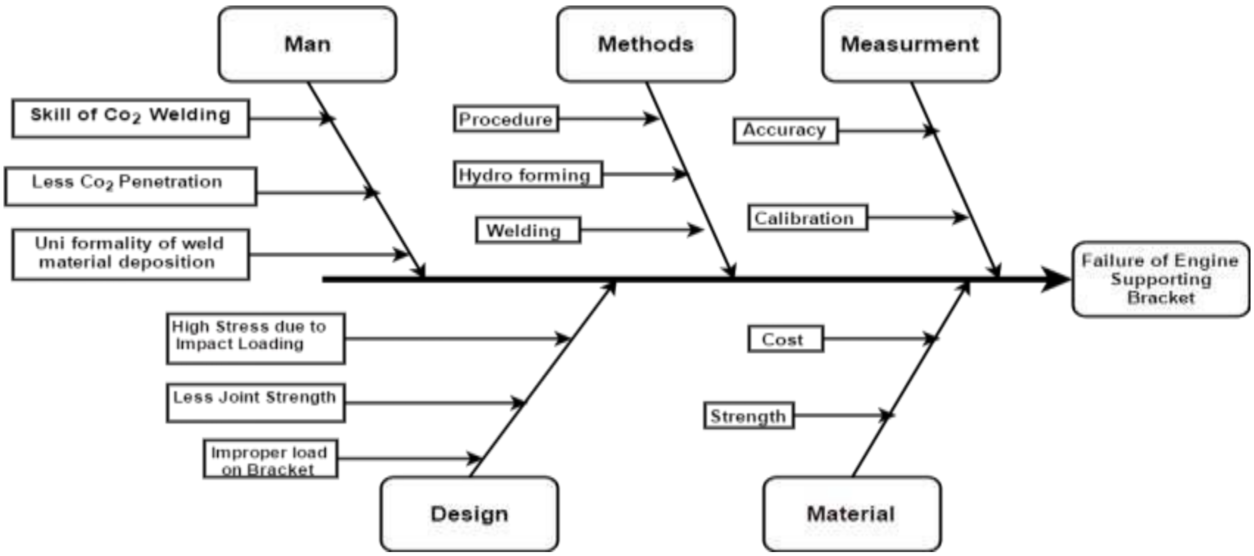


Fig.1: Schematic view of Fishbone diagram

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.

- **Static & Modal Analysis of Engine supporting bracket:**

Finite element analysis is most important software tool with the capability to find a wide range of different problem and gives approximate solution for any complex shape design. The engine supporting bracket is prepared 3-D Part modeling using CATIA V5 software & imports this CAD Geometry in ANSYS workbench R 15.0. Select their material properties from engineering material data (like young modulus, density, poisons ratio, yield strength of materials). At that time only one material is selected. The process for dividing the complex shape into small pieces is called as discretization or mesh generation The pre-processing of the engine supporting bracket is the purpose of the dividing the problem into node, elements and to developing the equation for an element matrices. Further apply the boundary conditions & select area for fixed supports. In static structural analysis is analyzed deformation, equivalent stress & Strain energy of the components. In modal analysis to determine natural frequency of the component. Generating results and save the file.

https://dx.doi.org/10.22161/ijaers.4.5.24 *ISSN: 2349-6495(P) | 2456-1908(O)*

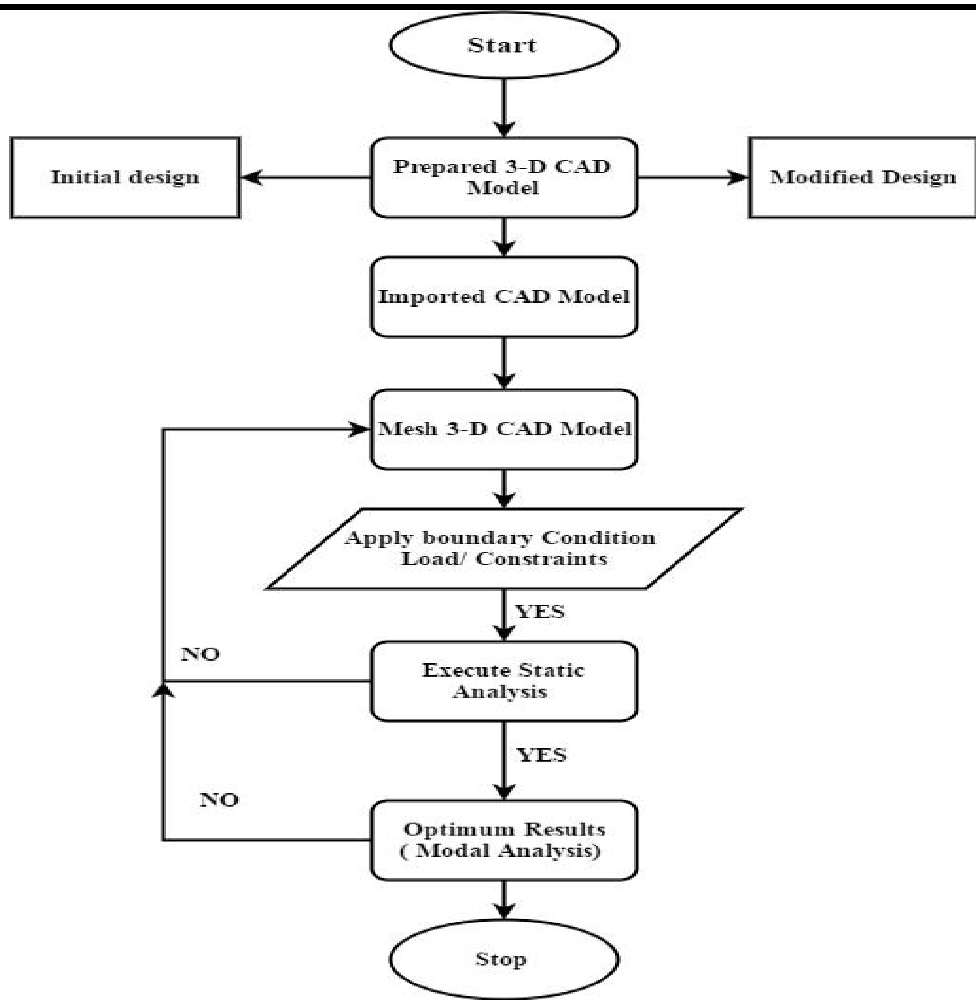


Fig.2: Typical Design process flow

IV. MATERIAL & MATERIAL PROPERTIES

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

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

V. BOUNDARY CONDITION

Basic model of engine supporting bracket is prepared by using CAD Software modelling CATIA V5. CAD Model geometry is imported in Ansys workbench and next step are part, coordinate system, Contact & contact region. Engine is rest on the engine supporting structural link is known as Engine supporting bracket. Weight of Baja Mega Max dieselEngine is 800N is acted on four mounting position on the structure bracket in equilibrium condition and Silencer weight is 20N is applied on end section. This force is acted vertically download on the bracket and the force that acted upon externally when the system is not under motion in static structural analysis. Weight of engine is equally distributed on the bracket component and upper end is connected to the main skeleton of the vehicle. In finite element analysis is very important for deciding boundary condition.

Meshing: Engine supporting bracket component is dividing into small pieces is known as meshing or discretization. Meshing is done using FEA package Ansys Workbench and meshing are dividing three group are coarse, medium and fine option are available. Meanwhile node is 112312 and element 37782.

https://dx.doi.org/10.22161/ijaers.4.5.24 *ISSN: 2349-6495(P) | 2456-1908(O)*

VI. RESULTS AND DISCUSSION

6.1 Static Structural and Modal Analysis of Initial Design

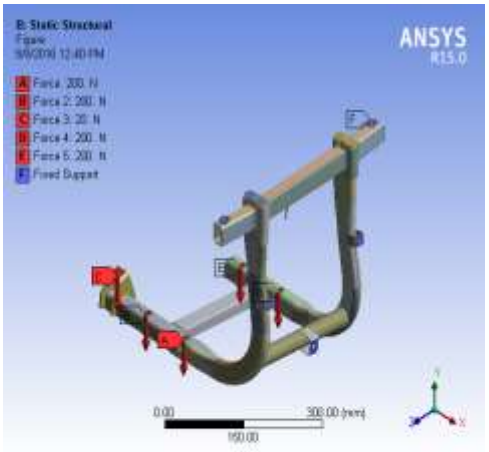


Fig.3: Boundary condition of initial design

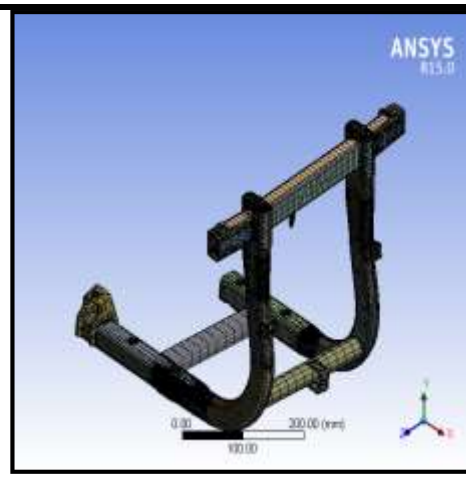
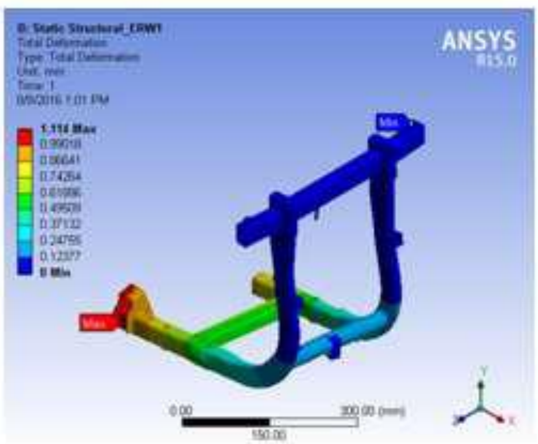
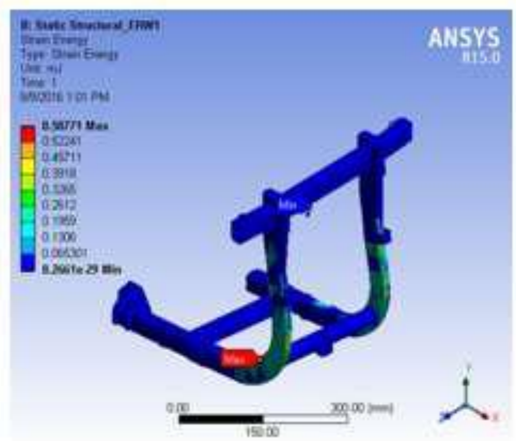
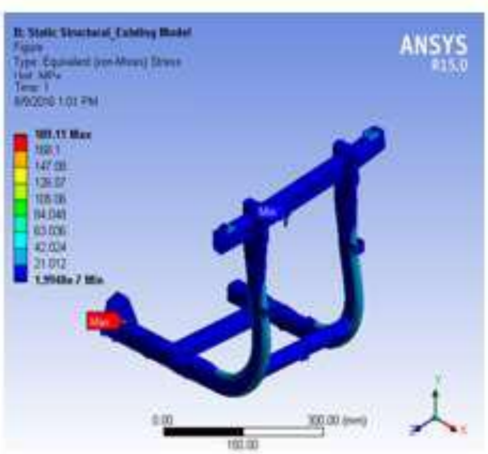


Fig.4: Meshed Model of ESB





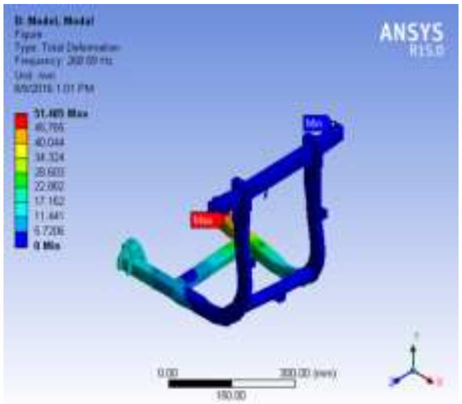


Static structural Analysis of Engine Supporting Bracket ERW-1. (Initial Design)	
Total Deformation mm	**1.114**
Von-Mises Stress Mpa	**189.11**
Strain Energy mJ	**0.58771**

Fig.5: Total deformation, Equivalent (Von- Mises) Stress &Strain Energy for Initial design of ESB of Initial Model

https://dx.doi.org/10.22161/ijaers.4.5.24 *ISSN: 2349-6495(P) | 2456-1908(O)*

6.2 Modal Analysis of Initial Engine supporting Bracket.



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

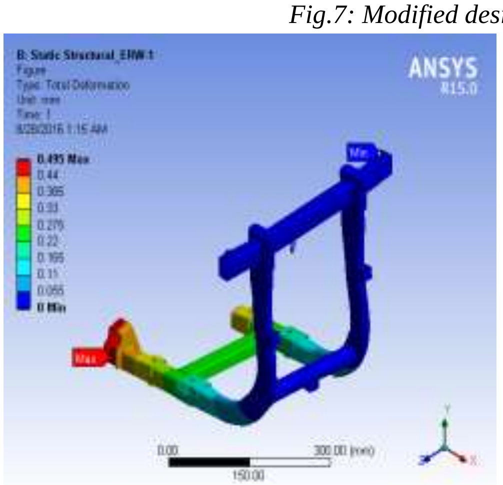
Fig.6: Modal Analysis of Initial Engine supporting Bracket.

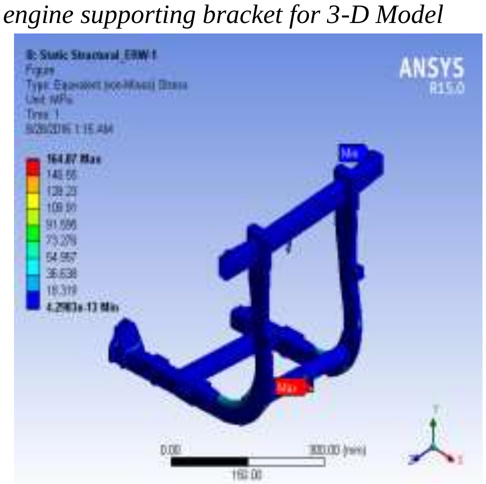
Static structural of Engine supporting frame using ERW-1 material for initial design analysis by ANSYS Workbench R15.0 and getting results of total deformation is 1.114 mm, Equivalent (Von- Mises) Stress is 189.11 MPa& Strain Energy is0.58771 mJ**.**

6.3 Static Structural and Modal Analysis of Modified Design

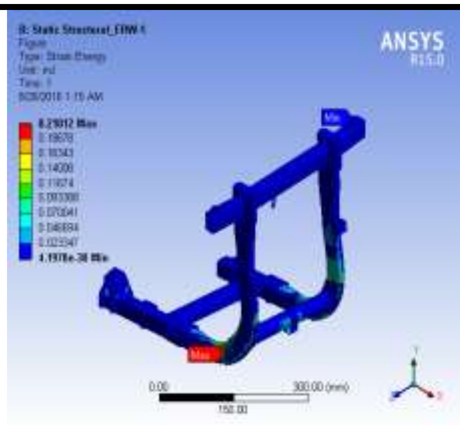
Static Analysis deals with the conditions of the equilibrium of the bodies acted upon by forces .A Static analysis is used to determine the total deformation in mm with Same material i.eERW-1.

Fig.7: Modified design of engine supporting bracket for 3-D Model





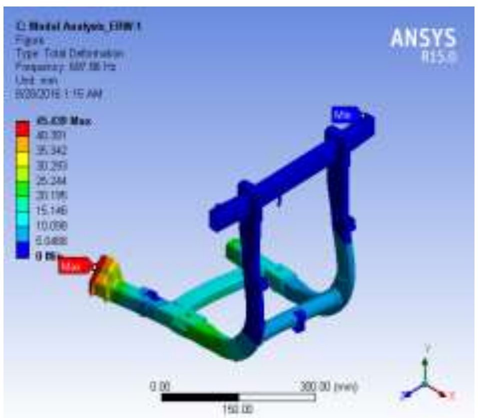
https://dx.doi.org/10.22161/ijaers.4.5.24 *ISSN: 2349-6495(P) | 2456-1908(O)*



Static structural Analysis of Engine Supporting Bracket ERW-1. (Modified Design)	
Total Deformation mm	**0.495**
Von-Mises Stress Mpa	**164.87**
Strain Energy mJ	**0.21012**

Fig. 8: Total deformations, Equivalent (Von- Mises) Stress &Strain Energy for Modified design of ESB.

6.4 Modal Analysis of Modified design Engine supporting Bracket.



Mode	Frequency [Hz]
1	145.86
2	164.62
3	257.83
4	489.68
5	607.07
6	145.86

Fig. 9: Modal Analysis of Engine Supporting bracket Modified Design.

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

Comparative study ofinitial and modified design for ERW-1(Static Structural Analysis).It can be anticipated from the above analysis that the modified Engine Supporting bracket is safe and suitable for further operation**.**

VII. VIBRATION STUDY USING FAST FOURIER TRANSFORM

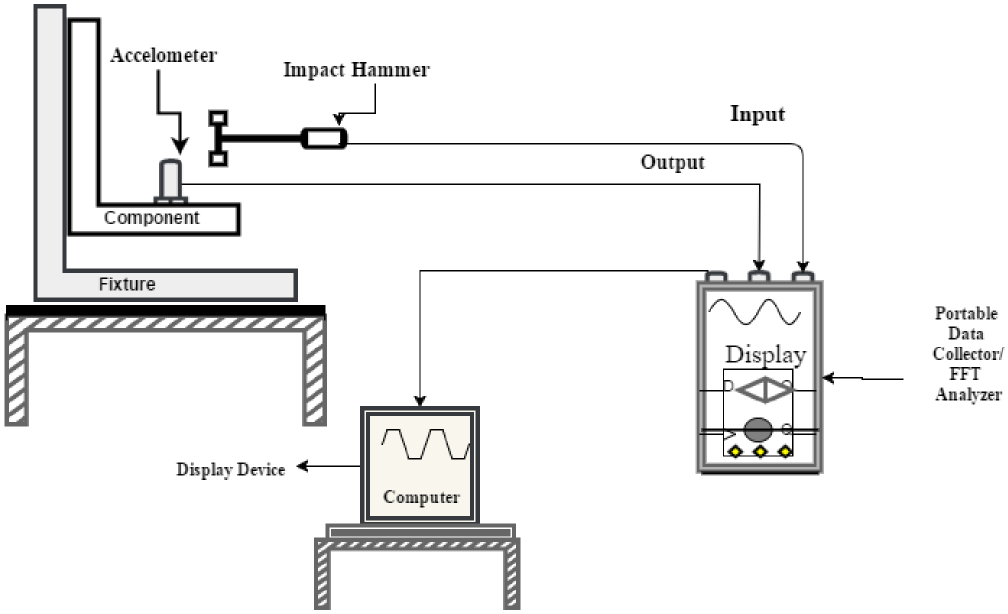


Fig.10: Experimental set up connections

- Engine Supporting bracket placed over the rubber pad as isolation on the table
- An accelerometer (with magnetic base) was placed at one end of the bracket & is connected to FFT Analyzer.
- The Impact Hammer and Accelerometer is connected to FFT, the output of FFT is given to PC.
- Applied the load by using Impact Hammer as excitationat Different points & the Power is supplied to FFT and PC.
- The Result is displayed on PC.
  - Vibration analysis of Initial Model by using FFT Analyzer the result are shows in graphical representation are as below:-

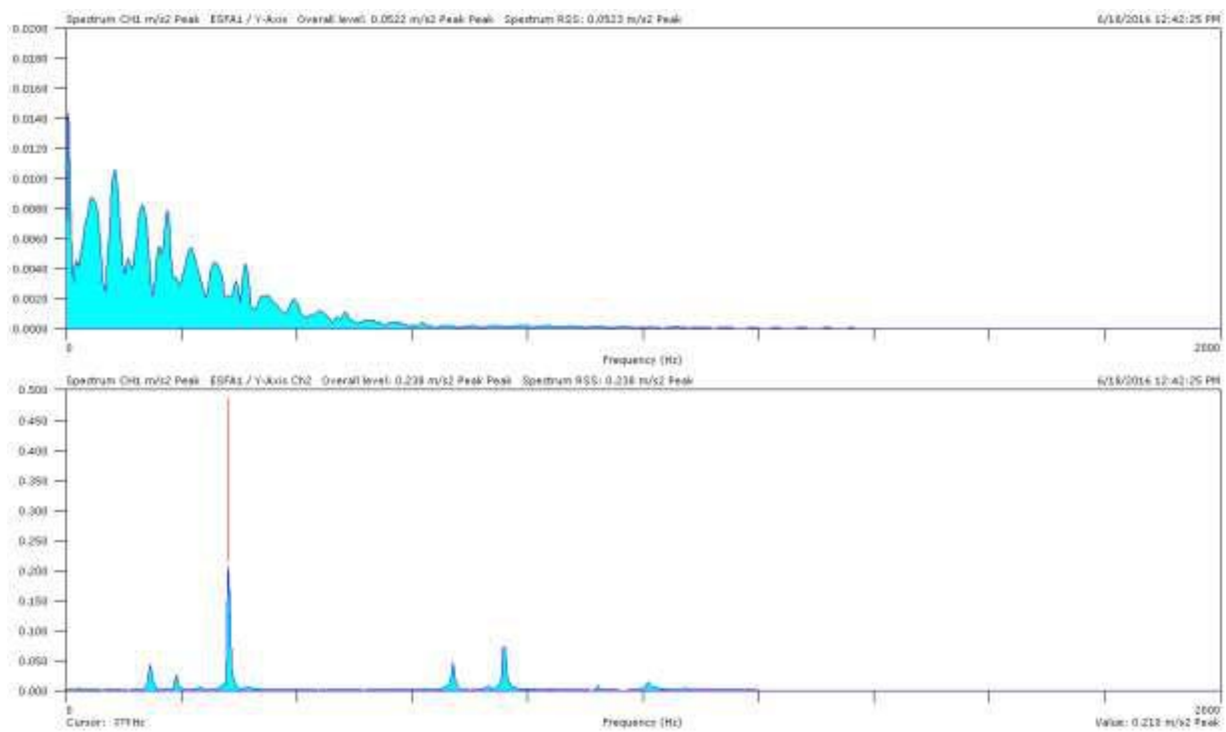


Fig. 11: Vibration study under the consideration of acceleration vs. frequency.

The vibration study for investigating whether the self-excitation frequency of the Engine Supporting bracket is greater than or equal to the natural frequency of the Engine Supporting bracket.

- Vibration analysis of Modified Model by using FFT Analyzer the result are shows in graphical representation are as below:-

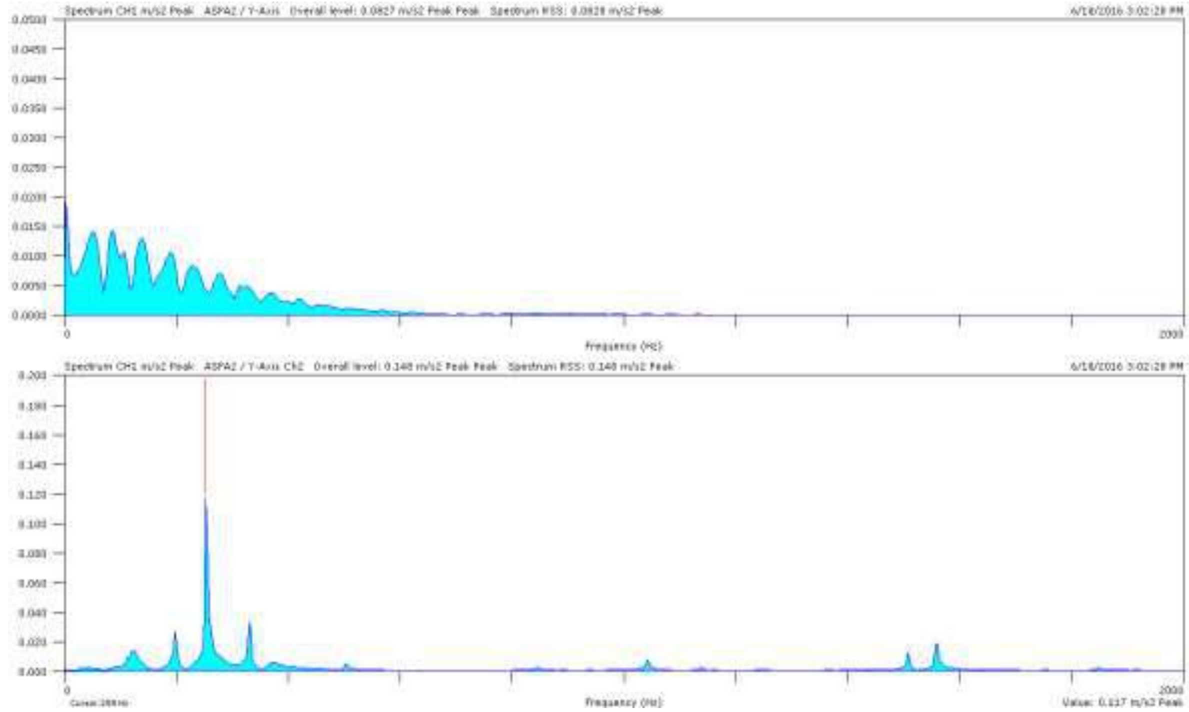


Fig. 12: Vibration study under the consideration of acceleration vs. frequency

It was found that for the initial design of Engine Supporting bracket the frequency mode 273 Hz the acceleration was 0.209729 m/s^2. But for modifies design it was found that at 259 Hz frequency the acceleration 0.116604 m/s^2 which was much less than the earlier one.

VIII. CONCLUSIONS

Evaluation of modified engine Supporting bracket was done using static structural, modal and fast Fourier transform analysis. Use of Finite Element method was found out to be very significant in this work as experimental work and time reduced by huge margin.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 displacement and equivalent von-Mises stress 189.11MPa.Further natural frequency of modified design was found to be 257.83 Hz which well within the range below self-excitation frequency and less than the natural frequency 268.59 Hz of initial design. From the results obtained for initial design and modified design, it can be anticipated that ERW-1 material best suit the requirement of the desired application.

REFERENCES

[1] Walunje Prasant .S.V. N. chougule, AnibanC.Mitra "Investigation on modal parameter of rectangular cantilever beam using Experimental Modal analysis" 4th international conference on material processing and characterization Science Direct Material Today: Proceeding 2 (2015) 2121-2130, doi:10.1016

[2] Mohammad Vaziri, Ali Vazari, Prof. S.S. Kadam "Vibration Analysis of Cantilever Beam By Using F.F.T Analyser" International Journal of Advanced Engineering Technology (IJAET) ISSN: 0976-3945/ Volume-IV/ Issue-II/ April-June, 2013.

[3] Senthilnathan Subbian, O.P. Singh, Srikanth K. Mohan, Arockia P, Jeyaraj,"Effect of muffler mounting bracket design on durability" ELSEVIER at Science Direct Engineering Failure Analysis 18 (2011) 1094-1107 doi: 10.1016

[4] Chetan D. Gaonkar*"Modal Analysis of Exhaust System to Optimize Mounting Hanger Location"* International Journal of Engineering Research & Technology (IJERT) ISSN: 2278-0181 Vol. 4 Issue 03, March-2015.

[5] Umesh S. Ghorpade, Prof. D. S. Chavan, Prof..M.V.Kavade, *" Static Structuraland modal analysis of engine bracket using Finite Element analysis"* International journal of engineering research & technology (IJERT) ISSN:2278-0181, Vol.I,Issue 10, December-2012,pp1-6.

[6] 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"* International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 Vol. 2, Issue 5, September- October 2012, pp.1016-1020).

[7] 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, (2014), vol. 4, Issue 9, pp 74-81.