FINITE ELEMENT ANALYSIS OF ENGINE MOUNTING BRACKET BY CONSIDERING PRETENSION EFFECT AND SERVICE LOAD

Sandeep Maski[1], Yadavalli Basavaraj[2]

[1]PG student, Department of mechanical engineering, BITM, Bellary, Karnataka, India
[2]Professor & Head, Department of mechanical engineering, BITM, Bellary, Karnataka, India

Abstract

Bracket is one of the important components of an engine mount assembly, heavy performance truck has their engine supported by bracket and this engine mounting brackets assembly is used in chassis front frame which has been designed as a framework to support engine along with transmission member. The main function of the engine mount bracket is to properly balance the engine and transmission on the vehicle chassis, engine mount is an important part of the vehicle to reduce the vibration and noise, by which smooth ride of the vehicle can be achieved. Vibration and strength of engine bracket has been continuously a concern which may lead to structural failure if the resulting vibration and stresses are severe and excessive. The present work focuses on the FEA analysis of engine mount bracket for three materials by using meshing and analysis software which are HYPERMESH and ABACUS, the materials used are cast iron, wrought iron and mild steel, modal analysis and static analysis carried out by which maximum von-misses stress and natural frequency are computed. The main objective is to select the best material from the obtained result under prescribed conditions.

Key Words: *Engine mount bracket, Vehicle chassis, Modal & Static analysis, pretension load, & Service load.*

---***---

1. INTRODUCTION

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. 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. There are number of vibrations and noise equipments which affect the vehicle body. Due to uneven roads and by bad suspension, vibration and noise are generated in a vehicle body, which will result in the high frequency. The noise and vibration generated by the equipments is transferred to the vehicle body points. Body attachment stiffness plays a vital role for NVH performance improvement. Due to disturbance from the road and engine at idle, usually below 30 Hz frequency is developed. To reduce and control the vibrations engine mount bracket should be stiff. In the other case of high frequency range of 30 -250 Hz from the engine has to be absorbed by bracket, so engine mount bracket must satisfy these criteria.

In automobile sector the extremely competitive automotive business needs manufactures to pay a lot of attention to traveling 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 demandfor higher playacting enginemount brackets should not be offset by arise within the production prices and/or development cycle time.

Due to the varying speeds and uneven roads, chassis bodies have underwent uneven balancing force. These tendencies bring about to unwanted vibrations that cause associate uncomfortable ride and conjointly cause extra stresses within the automobile frame and body. An engine mounting system includes a front mount, rear mount, and a transmission mount. Mounting axis of the engine mount, front mount, and rear mount area unit is vertical where as the mounting axis of that transmission mount is lateral. The transmission mount might vary its spring constant throughout vehicle movement. Resilient mounting will offer longer life for frame and block mounting brackets. The front engine together with associate in nursing engine mount bracket that is fastened on one finish thence, the front engine mount mounting the engine with the material endured the opposite finish thence ,front engine mount mounting the engine with the material interposed between the engine and also the engine mount bracket, the material supporting the engine bracket fastened to the front aspect of the engine, wherever area is provided between the front aspect of the engine and engine mount bracket and an off-line equipment disposed in area between the front aspect of the engine and mount bracket, wherever within the front engine mount features a strength against the load applied there to within the longitudinal direction of the vehicle but that of the off-line equipment.

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. By some studies it is observed that brackets saved 38% of mass. 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.

1.2 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. They have considered the three materials for engine mount bracket that is aluminum alloy, magnesium alloy, gray cast iron when modal analysis is carried out, it is found natural frequencies of gray cast iron is low which will prove more hindrance in vibration of engine mount bracket so they have eliminated gray cast iron, in terms of analysis aluminum alloy and magnesium alloy are showing almost near value of natural frequency in practical terms as magnesium alloy is having better strength that is low stress value, so preferably magnesium alloy is selected as better material by study.

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, in this paper the same model and materials has been taken as the paper [1]. 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 they have observed that aluminum alloy have good natural frequency and stresses are also within the yield strength, so by considering the aluminum and reducing its thickness further by 2mm than original component, they found that now von misses stresses are also with in yield stress so they have achieved reduction in the mass of bracket up to 0.43kg when compared to previous one.

In the above references the authors carried out the FEA of engine mount bracket for the static type of loading and analysis carried out was static and modal analysis with optimization of materials,

By information from the above survey it is decided the task to be carried out

1. Understanding the loading conditions on the engine mount bracket.
2. Selecting the type of materials which can be used in manufacturing of engine bracket.
3. Selecting the proper designing tools for modeling
4. Selecting the suitable meshing and analysis tool.

2. FINITE ELEMENT ANALYSIS

The Finite Element Method is a numerical approximation method, in which the complex structure is divided into number of small parts that is pieces and these small parts are called as finite elements. These small elements are connected to each other by means of small points called as nodes. As the finite element method uses matrix algebra to solve the simultaneous equations, so it is also known as structural analysis and it's becoming primary analysis tool for designers and analysts.

The three basic FEA process are

a) Pre processing phase
b) Processing or solution phase
c) Post processing phase

2.1 HyperWorks

HYPERWORKS is one of the most used software to solve finite element analysis which will provide solid meshing, surface meshing then the processors or solvers used are MSC Nastran, Marc, Abacus, LS-DYNA, ANSYS, and Pam-Crash.

Altair hyper works is having many tools for the creation of ready models and to analyze for non-linear, linear, thermal, explicit dynamics and other types of finite element problems. To deal with gaps in a model there is a tool called geometry cleanup which will make engineers easy to modify. For the creation of models from scratch there is a tool called solid meshing tool so it is very easy to create FE models by using hyper mesh. Meshing can be easily done on the surfaces by using automated meshing tool and even there are manual methods which provide more control. In this finally boundary conditions, loads and analysis is built in for most FE solvers, which will minimize the need to edit input decks.

2.2 Abaqus

Abacus is important simulation program in this any range of problems can be solved from simple linear analysis to non linear simulations. Abacus is having library of elements which is used for studying structural problems and more (stress/displacements). By this problems can be simulated in areas such as heat transfer, thermal management, mass diffusion, acoustics, solid mechanics and piezoelectric analyses. In most simulation even including non linear ones, the user needs to provide engineering data information such as geometry of model, material behavior, boundary conditions and load applied on it. But for non linear analysis in abacus, it chooses automatically appropriate load increments and convergence tolerance.

Finite element mesh is the main goal of preprocessing, after this material properties have to be assigned and boundary conditions to be applied in the form of restraints and loads. Geometry is subdivided into elements by finite element mesh, which contains nodes. In space nodes are just point locations which are present at each side of element corner. In this 2D-elements or 3D-elements may be present. Plane stress, axis symmetric and plane strain are conditions for

2D-elements and physical thickness in three dimensions is present for 3D element. Three translation degree of freedom per nodes are present for solid element. Depending upon the class of analysis degree of freedom is assigned, for example only temperature or many temperature of degree of freedom are present at each node in thermal analysis. As meshing is most time consuming job in FEA so free meshing is carried out, which automatically subdivides meshing region into elements thus doing easy meshing. With the type of solution material properties will vary. Linear static analysis requires elastic modulus and Poisson's for each material type, after this analysis the input file is run in abacus to get the onp file which will give the complete result and is viewed in hyperview

2.3 Scope of Project

This thesis is part of a research project whose aim is to design engine mount bracket. The overall aim of the project is to develop models of engine mount bracket by using CATIA software. In order to create computational tools that facilitates the design of engine mount bracket with improved performance. This model has been developed in order to create a finite element (FE) model of an available automotive engine mount bracket, so that it's current static and modal behavior could be analyzed, several materials can be tested in this model and, perhaps, improvements to the current performance specifications can be achieved.

Computational modeling and simulation of an available engine mount are performed to gain such an understanding and to evaluate the effectiveness of current solutions available in the market. Special attention is given to the correct modeling of nonlinear effects on the static behavior of the engine mount bracket. The FE analyses in this thesis have been performed in ABAQUS. The choice of using a commercial FE code makes it easier to focus on the engineering problem rather that a detailed description of complex FE models. It also results in methods that can be put directly to use in industry.

2.4 Objective of Project

To do static structural and modal analysis of engine mounting bracket for three different materials viz. Mild steel (MS), Wrought Iron (WI) and Cast Iron (CI) and suggest best material for the bracket.

3. Engine Mount Bracket

The engine mounting bracket is most important part of vehicle in reducing vibrations and harshness for the smooth ride of the vehicle. The important function of an engine mounting bracket is to balance the engine & transmission properly on the vehicle chassis for good balance control when vehicle is in motion. In this engine mount bracket of a truck is modeled using CATIA.

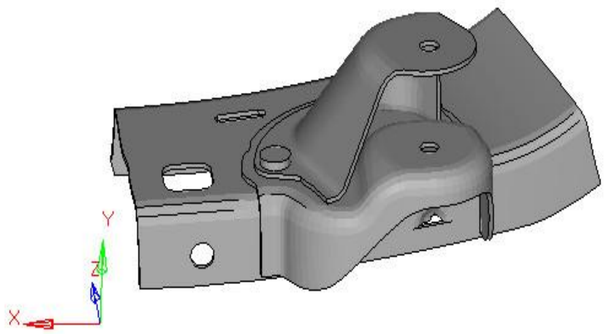


Fig -1: CATIA model of Engine mount bracket

3.1 Meshing of Engine Mount Bracket

For meshing the model solid mesh is generated by keeping mesh size as 5 mm.

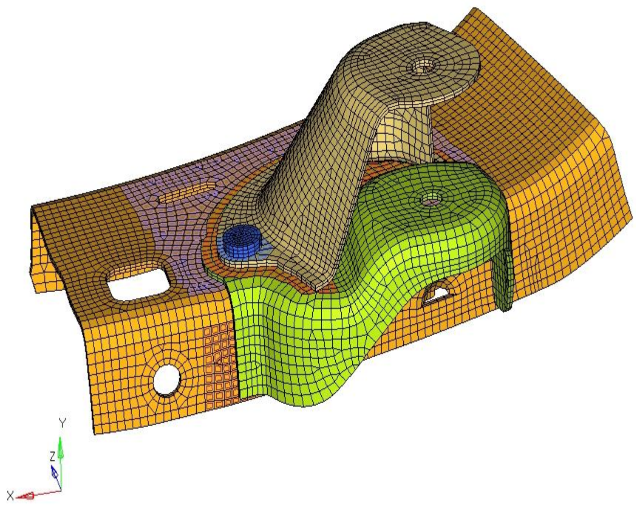


Fig -2: Meshed model of Engine mount bracket

3.2 Boundary and Loading Conditions

Constraints: The element which is used to fix engine mounting bracket and body of the vehicle is fixed and used as a rigid element. In this chassis frame which is fixed and connected to the bracket and is made constrained at the both ends, further after constraining analysis is carried out.

Loads: Specific values of load are applied on engine mounting bracket. The load is taken as 1000N which is considerable as the distributed weight of the engine; this load is taken as service load as shown below in fig. Pretension Load of 24000N is applied on the bolt, which is the standard pretension load calculated for the M12 bolt, to calculate the pretension effect as shown below in fig.

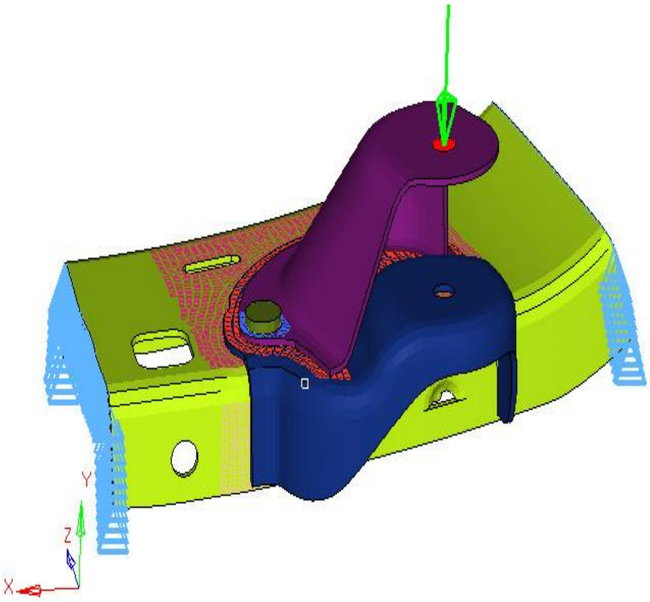


Fig -3: Service load acting on bracket

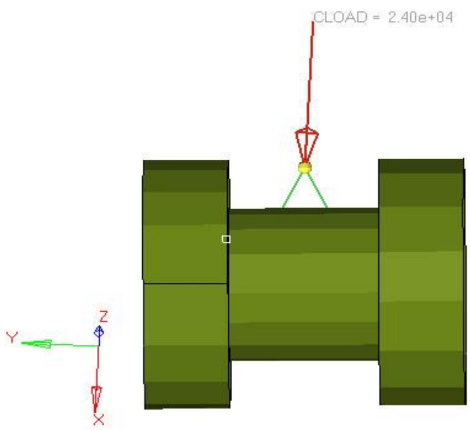


Fig -4: pretension load acting on bolt

Table -1: Mechanical properties of materials used for engine mount bracket

Sl. No	Mechanical Property	Mild Steel	Cast iron	Wrought iron
1	Young's modulus (Mpa)	$2.1e^{+5}$	$1.2\ e^{+5}$	$1.9\ e^{+5}$
2	Poisson's ratio	0.30	0.28	0.30
3	Density(Kg/mm^3)	$7.89e^{-9}$	$7.20e^{-9}$	$7.75e^{-9}$
4	Yield Strength (N/mm^2)	370	130	210
5	Ultimate Strength(N/mm^2)	440	220	320

3.3 Analysis

Modal analysis -
Modal analysis is used to determine the vibration features of a component in the form of mode shapes and natural frequency. The normal modal analysis is carried out for engine mount brackets, in three design iterations. The three design iterations are carried out for different materials with constant of thickness which is mentioned as per the company standards. Our objective is to perform modal analysis to determine the natural frequencies and mode shapes for natural vibration modes for three structures in which the damping and applied load is neglected.

Static analysis -
A static structural non linear analysis is the time independent analysis where we do check for displacements, stress and strain of a component. Static analysis for all the above materials is processed using Altair Hyper works and solved using the software Abacus. Here static analysis is carried out to all the above events to determine the stress concentration and the displacement. The stress concentration and the displacement are carried out for both service load and the pretension load. The results obtained are shown in the figures.

3.4 Modal Analysis Results

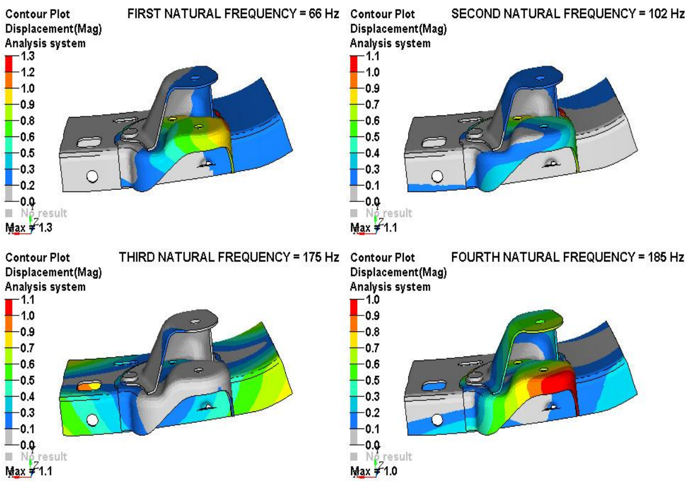


Fig -5: Modal analysis of mild steel bracket

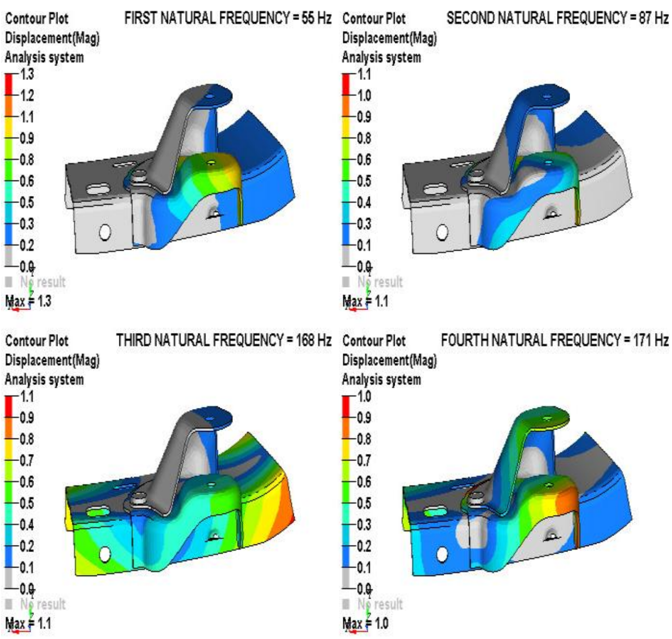


Fig -6: Modal analysis of cast iron bracket

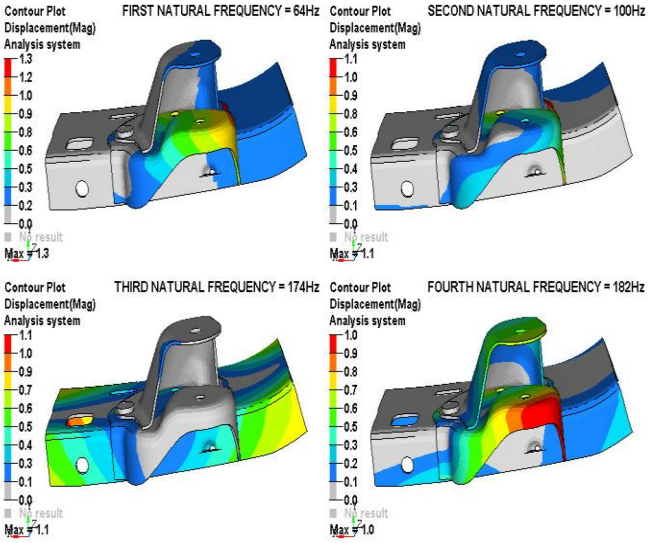


Fig -7: Modal analysis of wrought iron bracket

Table -2: Comparison of Modal analysis Results for three materials of engine mount bracket

Design iteration	Materials Thickness 3.4mm	Mode frequency (HZ)			
		1	2	3	4
01	Mild steel	66	102	175	185
02	Cast iron	55	87	168	171
03	Wrought iron	64	100	174	182

3.5 Discussion

By comparing normal modal analysis for all materials of engine mount bracket that is the first fundamental natural frequency of mild steel is 66Hz which is found to be good compared with two materials. To select the suitable material for the engine mount bracket, we can't decide only on the basis of modal analysis, so further static analysis is carried out to know the stress and displacement of the materials which will be helpful to choose the suitable material for the engine mount bracket.

3.6 Static Analysis Results

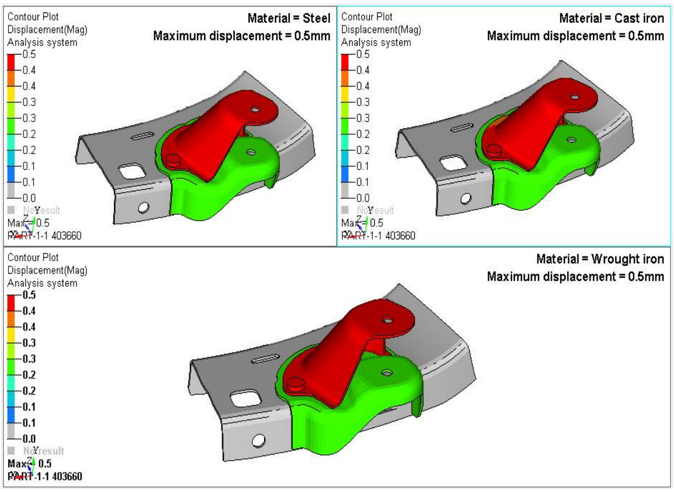


Fig 8: Displacement for pretension load acting on bolt

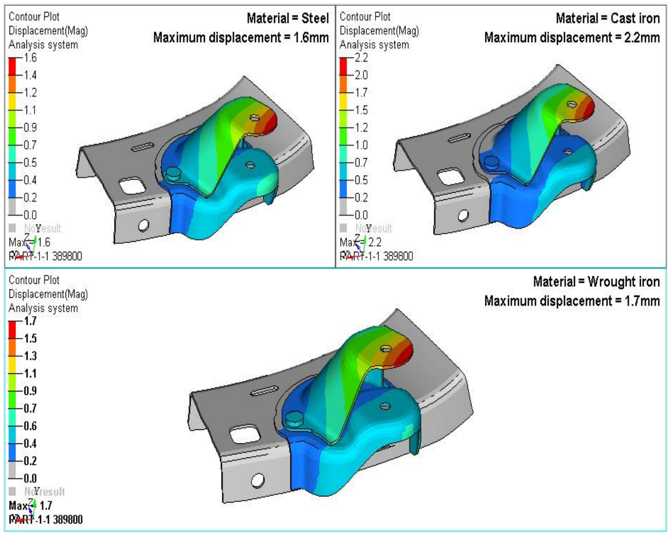


Fig -9: Displacement for service load acting on bracket

Table -3: Maximum displacement of bracket

Material displacement (mm)	Mild steel	Cast iron	Wrought iron
Pretension load	0.5	0.5	0.5
Service load	1.6	2.2	1.7

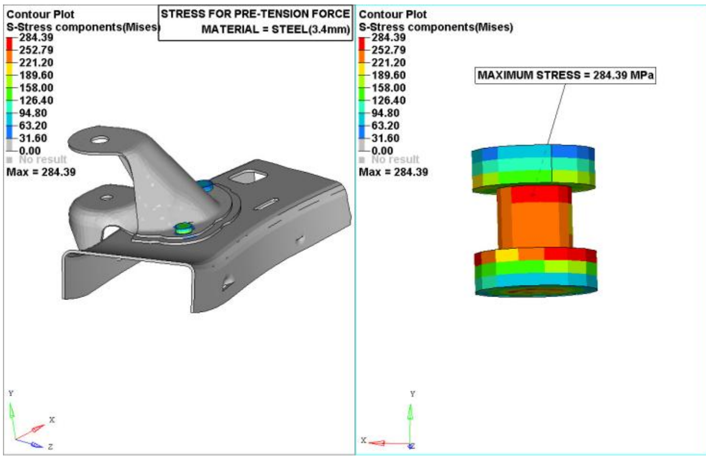


Fig -10: Maximum stress for pretension load of mild steel bracket

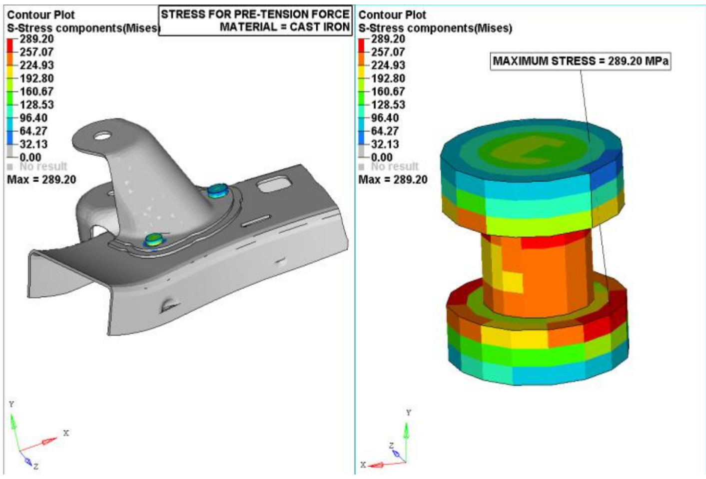


Fig -11: Maximum stress for pretension load of cast iron bracket

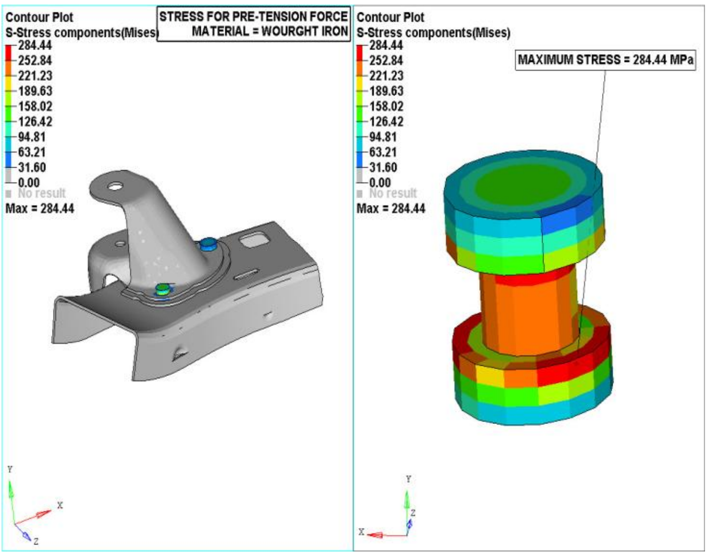


Fig -12: Maximum stress for pretension load of wrought iron bracket

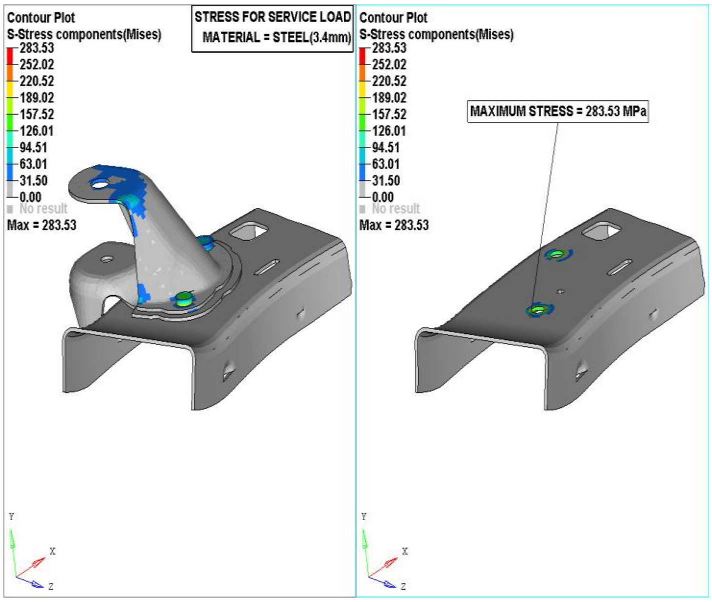


Fig -13: Maximum stress for service load of mild steel bracket

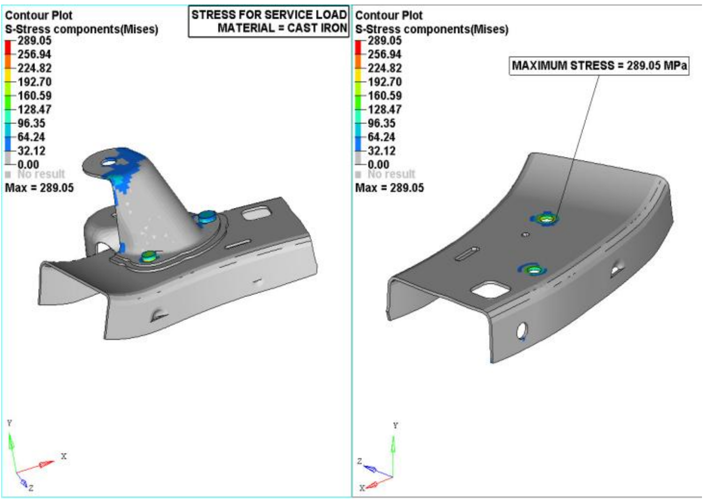


Fig -14: Maximum stress for service load of cast iron bracket

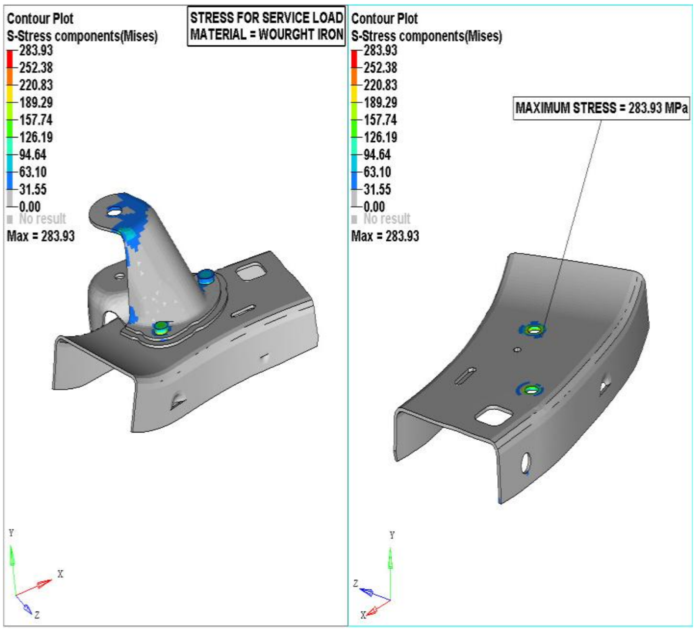
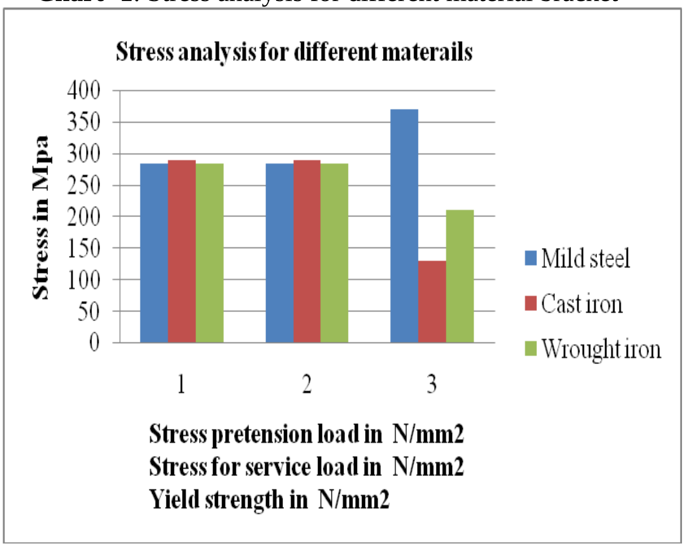


Fig -15: Maximum stress for service load of wrought iron bracket

Table -3: comparison of stress of engine mount bracket for different materials

Different materials	Stress for pretension load in N/mm^2	Stress for service load in N/mm^2	Yield strength in N/mm^2	Ultimate strength in N/mm^2
Mild steel	284.39	283.53	370	440
Cast iron	289.20	289.05	130	220
Wrought iron	284.44	283.93	210	320

Chart -1: Stress analysis for different material bracket



3.7 Discussion

In the comparison of material variation, From the above plot of figures we can clearly see the deformation of engine mount bracket is taking place in all the materials for the applied service load and displacement is varied in all the materials, but the maximum deformation is 2.2mm, which is in the cast iron engine bracket, this is due to the fact that density of the material is different for all the materials. The minimum deformation is in the mild steel bracket that is 1.6mm because mild steel is having more density when compared to cast iron and wrought iron.

From the table-3 it can be seen that von misses stress for all three materials are almost nearer, this is due to fact that area and force for all materials is same but stress coming on the mild steel bracket are well within the yield strength (370Mpa) but von misses stress of cast iron and wrought iron is greater than yield strength, so we cannot select cast iron and wrought iron as suitable for the bracket.

4. CONCLUSIONS

As vibration and strength plays an important role in the design of engine mount bracket, so in this paper special attention has been given for selection of suitable material for engine mount bracket so that it can withstand high strength and vibrations.

It can be seen from the obtained results and discussion that for modal analysis first fundamental natural frequency of mild steel is high that is 65Hz compared with cast iron and wrought iron and in case of static analysis displacement of mild steel is 1.6mm which is less when compared to other two materials and 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

REFERENCES

[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]. Pavan.B.Chaudhari, Dr.D.R.Panchagade "Comparison of Magnesium, Aluminum 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.

[4]. A.S.Adkine, V.S.Kathavate, G.P.Overikar, S.N.Doijode, "Static Behavior of Engine Mounting Bracket" International Advanced Research Journal in Science, Engineering and Technology ISSN (Online) 2393-8021Vol. 2, Issue 4, April 2015.

BIOGRAPHIES

Sandeep Maski received the B.E degree in mechanical engineering in 2012 from BITM, Bellary affiliated to Visvesvaraya technological university Belagum and worked as Mechanical engineer in Hospet steels ltd, presently pursuing MTECH 2nd year from BITM, Bellary.