Analysis of the Root Causes and Design Modification for the Oil Filter Support Bracket

***Dikshant Dinesh Nalawade1, Sanjay Yadav2.***

*aStudent , RIT,Rajaramnagar, Maharashtra, India*

*bProfessor, RIT,Rajaramnagar, Maharashtra, India*

A B S T R A C T

In this work, the CAE tools are utilized to examine and optimize the designs developed inside the CREO programmed, which is used for the optimization of the oil filter support bracket used in our cars that are malfunctioning on the road. The durability and other features of the bracket were validated using Finite Element Analysis. The bracket is static and dynamically analyzed, as well as mode and stress analyzed. The new design is optimized utilizing form and topology optimization methods to decrease fuel filter supports bracket fracture failure and boost bracket rigidity. The CAE tool's optimized design is then empirically validated. The new design requires stress levels lower than the substance's long-term limit, adds more mass while costing less, and is effective in installation assembling. The CAE tool findings were quite close to the experiment's outcomes.

Keywords: oil filter, CAE tool , CREO software, optimization, fuel filter method

# Introduction

Here introduce the paper, and put a nomenclature if necessary, in a box with the same font size as the rest of the paper. The paragraphs continue from here and are only separated by headings, subheadings, images and formulae. The section headings are arranged by numbers, bold and 9.5 pt. Here follows further instructions for authors. The design and development of a vehicle is a difficult task for a company since it must fulfil all of the demands for consumer pleasure as well as the many standards set by various agencies in various countries. It is a very market-oriented position that requires regular and fast feedback from clients. The oil filter is one of the most important elements of a road automobile, including a car or truck. Fuel filters in high- performance vehicles are kept in place by brackets. It is critical in improving the comfort and working environment of a vehicle. There has been a lot of interest in upgrading the oil filter support mounting method for many years. For an automobile, an oil filter support bracket is necessary. Because of the emphasis on pricing, component manufacturers have been under pressure to improve the performance of their materials and find ways to deliver them at lower rates. A multitude of noise and vibration sources damage the vehicle's body. The power delivered through uneven surfaces, the engine, and the suspension all contribute to a resonance effect across a wide frequency range, resulting in noise and vibration. Vibration from the engine and suspension conveyed to the body via the chassis mounting points has a significant influence on the ride and noise characteristics of a vehicle.

The bulk of automotive and aerospace components are manufactured using press tool sheet metal methods. These increasing demands need a design and manufacturing approach based on Finite Element Method for Sheet Metal Formability forecasts. The problem regions, causes, and solutions may be found using computer-aided engineering techniques and sheet metal's one-step formability analysis, and productivity can be boosted by reducing costly shop floor press tool trials and eliminating tool reworking. This article describes how product design, metal properties, and manufacturing boundary circumstances impact the deformation of sheet metal components.

Body attachment stiffness is well recognised to have a key impact in deciding how quiet the vehicle is at idle and on the road. As a result, bracket was developed as a framework to allow filtering. Fatigue of the filter support bracket has long been a concern, with the resulting stresses possibly leading to structural failure if high and excessive. Prolonged exposure to whole-body stresses at work might result in weariness and, in rare cases, vehicle damage. A modal analysis of the engine bracket is designed to determine whether the current configuration has a natural resonance frequency that is lower than the

\* *Corresponding author.* Tel.: +0-000-000-0000 ; fax: +0-000-000-0000. E-mail address: [author@institute.xxx](mailto:author@institute.xxx)

excitation rate of the engine bracket, in which case the design is judged safe. Material optimisation will take place when the design's safety has been established. Optimisation: The design's suitability for various materials, such as aluminium and magnesium alloy, will be assessed. The project's purpose is to select a suitable substance and reinforce the bracket. A dipstick bracket has broken in the field, according to reports. Fuel filter pipe support bracket failure was seen in a market field vehicle equipped with an MDE5 engine. One of the components required for oil filter stability is oil filter support supports. The location of an oil filter support plate in relation to an oil filter and the motor. A static structural analysis is the investigation of displacements, stresses, strains, and forces on a structure or component as a result of the application of a load. It is assu med that the structure's response and loads would vary gradually over time. Other types of loading, such as externally applied forces, pressures, and temperatures, can be employed in this investigation. A linear or nonlinear analysis of static structures is conceivable. The motor is instantly linked to the oil filter support bracket, which is attached to the oil filter. A bracket transfers several shocks from the engine to the oil filter. Modal analysis identifies the vibrational qualities of a structure or specific component in the form of natural frequencies and mode shapes. Using this approach, we may undertake more in-depth dynamic studies, such as transient dynamic analysis, harmonic analysis, or spectrum analysis. When constructing a structure for dynamic loading conditions, the inherent frequencies and mode shapes must be considered. In this investigation, only linear behaviour is valid. Applied loads and damping are not considered in modal analysis. A static structural investigation must precede pre-stressed modal analysis. Several technological and automotive components are examined using finite element techniques. These assessments are quite valuable in the post-processing stage for approving or making design changes. The product life cycle mostly influences design changes, which assist design engineers or analysts in deciding final dimensions and component materials. An oil filter support bracket is subjected to severe vibration. Because of these motions, it is challenging to minimise mass and redesign the oil filter support bracket while keeping the material properties, boundary conditions, static and dynamic properties of the bracket in mind.

## Objective

The purpose is to do a modal analysis using the oil filter supports bracket to determine whether the current design has a natural frequency lower than the excitation frequency of the support bracket, in which case the design is judged secure. Material optimisation will take place once the design's safety and tension level have been confirmed. Optimization: The design will be assessed for various materials, steel alloys, material appropriateness and durability restrictions, and so on. FE analysis It is being carried out to assess whether motor actions caused the bracket to collapse. An FE research is carried out to assess the durability of the improved bracket owing to motor vibrations. It is advised that accelerations at bracket mounting points be quantified in order to connect FEA. Assuming that the bending operation leaves no significant residual stress, the modified bracket passes the fatigue strength condition.

* + 1. To construct and improve an oil filter bracket to prevent failure.
    2. Perform a FE analysis for engine vibration loads to establish whether bracket failure is caused by increased stress caused by engine vibrations.
    3. The bracket architecture is modified to reduce the stress level in the bracket caused by motor vibration.
    4. To lessen the possibility of the fuel filter support clamp failing.
    5. Determine the efficacy of a new oil filter attachment
    6. Determine the lowest natural frequency of oscillation of the oil filter assembly.

## Problem Statement

In this project, I'm utilising CAE tools to review and optimise designs generated by the CRIO programme, which is utilised for the optimisation of the oil filter support bracket used in our vehicles that are failing on the road in the market. The durability and other features of the bracket will be validated using Finite Element Analysis. In the field, an issue with the oil filter support bracket has been noted. The designs created in CRIO software, unfortunately is used to enhance the oil filter support bracket used in our cars but which failed in the field in the South African market, will be examined and improved using CAE tools in the present project. Finite Element Analysis will be performed to verify the durability and other properties of the bracket. Static, dynamic, modal, and stress analyses will all be performed on the bracket. To decrease the breaking failure of the Fuel filter support bracket and increase the bracket's stiffness, the new design is optimised using form and size optimisation techniques. The optimal design produced by the CAE tool is subsequently empirically verified. The new design contributes to smaller bulk, lower cost, and usable mounting assembly by causing stress levels that must be within the material's tolerance limit. The output of the CAE tool and the results of the experiment showed very good correlation. The frame will go through both dynamic and static analysis, mode and stress analysis, as well as. With the aim of reducing the breakage failure of the Fuel filter support bracket and boosting the bracket's rigidity, the new design is optimised utilising form and size optimisation techniques. The CAE tool's optimised design is then empirically tested. The new design results in stress levels that are lower than the endurance limit of the material, which helps to reduce bulk and cost, thereby enabling mounting installation. The findings from the CAE tool closely matched those from the tests.

# *Literature Review*

Sachin Kalsi et.al [1] –

In this paper, optimized the design of a compressor mounting bracket used in automobile air conditioning system of a passenger car. It was a challenge to reduce the mass and increase the stiffness of an optimized bracket taking into consideration its dynamic conditions and manufacturing constraints. The simulation results of the optimized bracket were validated using the experimental setup. present work, the CAE tools are used for the optimization of the compressor mounting bracket used in an automobile. Both static and dynamic analysis is done for the bracket. The new design results in lower level of vibrations, contribute to lower mass along with lesser cost which is effective in air conditioning system as well as the efficiency of a vehicle.

Yongkang Zhao et.al [2] -

In this paper, the finite element analysis tool has been used to analyse the engine mounting bracket using CAE Package and the development of new material for engine mounting bracket carried out. The modal analysis of the engine mounting bracket is carried out for determining natural frequencies and shapes of the natural vibration modes. These frequencies are the outputs given by the analysis of the engine mounting bracket. The results obtained for the static structural and modal analysis have shown that the magnesium is better than aluminium. From the results it can be seen that the magnesium bracket is safe for the required application.

Marko BASIC et.al [3] –

Author deals with a typical problem which occurs without CAE support in the development process, the failure of bracket, in the present case the bracket of an oil tube, which is connected to the engine structure. They made improvement in Mounting position from MB#5 to MB#6, Changed sheet metal material to S355JRc with changed sheet metal thickness to 4mm and changed sheet metal bracket geometry to reduce stress concentration.

Mohammad Javad Ershadi et.al [4]-

The author proposes a case study for application of Root cause analysis (RCA) on a research information system (RIS) called PARS is presented. The main tools and techniques which are generally used for determining the main root causes of quality problems are presented. Furthermore, the main root causes for quality problems of PARS system are introduced. This paper provides a framework for quality problem solving of RIS which has rarely been done in other research until now. The corrective actions for solving quality problems based on continuous improvement process (CIP) approach are described.

Chandranath. M et.al [5] –

In this paper, analysis is carried out by selecting the Sheet metal component material, by assigning Spring back match constraints, draw direction, surface thickness and appropriate element type using the Analyse formability one step command. All the component actual dimensions after manufacturing are complemented and found nearest to dimensions of NX formability analysis results and all the dimensions were found acceptable within the drawing tolerance.

Qinghai Zhao et.al [6] –

The author proposes a process of structural design for powertrain mounting bracket based on multi-objective topology optimization methodology. Multi-objective topology optimization formulation of the mounting bracket is implemented by the commercial software package Altair Optistruct 11.0. Performance characteristics are investigated based on the design obtained from topology optimization through numerical and experimental investigations. Multi-objective topology optimization approach can be well implemented by compromise programming to generate an optimal compromise candidate satisfying both static and vibration performance.

Vijay Kalantre et.al [7] –

In this paper a suspension bracket (leaf spring mounting bracket) of Mahindra Classic Jeep has been analysed and topologically optimized to reduce its weight but without compromising its strength. Modal analysis is also carried out and it shows that the minimum natural frequency is much above the road excitation frequency range. That means condition of resonance will be avoided and so as the maximum stress value will exceed the safe value anyway. All the analysis and optimization is done in Ansys 19.0

Koji Kimita et.al [8] –

In this paper author proposed a method for failure analysis in product service system (PSS) design. Especially, the PSS Failure mode effective analysis (FMEA) was proposed so that designers can conduct integrated analysis of product and service failures. Based on the results of the case study, the proposed method was found to be effective for helping designers detect both product and service failures, and then, develop measures from the perspectives of products and services. Furthermore, it was also effective for finding new business opportunities where manufacturers could offer new services with products. Future research could include the implementation of the proposed method in software to increase its efficiency.

Rafael Laurenti et.al [9] –

In this paper, Failure Mode, and Effect Analysis (FMEA) has been intensely used to ensure the quality and reliability of products. Additionally, over the years, several adaptations and improvements have been made in the method FMEA. Besides the use of the FMEA and the several attempts to mitigate its shortcomings, many incidents in the field are still occurring, which is costly to companies. This paper has presented a method to analysing the effects of changes made in the design. The method was conceived from the knowledge that changes carry a higher potential failure.

Abhishek P et.al [10]-

In this paper, the author work on optimizes the design of fuel tank mounting bracket for commercial vehicles. The project includes the geometry and finite element modeling of fuel tank mounting bracket design. Geometrical modelling was done using Pro-E; finite element modelling using Hypermesh software and analysis was done using Optistruct solver. This project deals with the static analysis of bracket assembly. This project provides a methodology for analysis of an assembly consisting of components made of metal components.

Umesh S. Ghorpade, D. S. Chavan, Vinaay Patil, Mahendra Gaikwad [11] –

In this paper we studied the engine mounting bracket. Vibration and fatigue analysis has been carried out to know the structural failure. Structural failure will occur if vibrations and stresses are excessive and severe. Prolonged exposure to whole-body vibration in the working environment may lead to fatigue and in some cases, it damages the car. Generally, the most important vibration relevant excitations in a car engine can be identified as follows:- combustion force; main bearing reaction forces including mass forces damper function and flywheel whirling, 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. Automotive engine mounting system must satisfy the primary tasks such as engine movement, engine rigid body dynamic behavior, and vibration isolation. The design and development of mounting bracket through use of Ansys software to achieve the requirements for mounting system and optimize the mount.

Lakshmi Kala, V.Ratna Kiran[12]-

In this paper authors have taken non-linear vibrational theory into consideration. Improvement in the vibration control can be achieved by determining the natural frequency of the engine bracket. To achieve isolation, natural frequency must be away from the excitation energy to avoid resonating condition. Also, NVH is an important vehicles characteristic motivating to achieve overall customer satisfaction. The main role of engine mounting system as one of the principle vehicle vibration isolating systems, besides suspension system, is to reduce the noise and vibration perceived by driver and to improve the ride comfort.

Abdolvahab Agharkakli, Digvijay Pradip Wagh [13]-

The engine mount and body mount is a complex joint assembly comprising of rubber bushings on the top and bottom of the frame bracket, a bolt, and retainer. The engine/body mounts are designed to carry the horizontal impact load in an impact and to isolate the noise, vibration and harshness (NVH), occurring during driving from entering the passenger compartment. The authors in this paper have studied mathematical model and compared its results with MATLAB simulations. The mounts are treated at the component level, and mathematical models for the same are evaluated to get the required characteristics. The mounts are modelled as spring and damper system subjected to impact loading that occurs during crash events. The approximation of input pulse has been described mathematically, which then serves to find the characteristics of the mounts. The change in the characteristics of mounts with the change in the velocity of impact has also been studied.

Dr.Yadavalli Basavaraj, Manjunatha.T.H [14]-

These authors have focused more on avoiding resonance and damping of frequency. If there exist unbalanced loads in engine body, resonant vibration occurs. This resonant vibration increases if chassis has unitary or frameless construction. This has forced designers to direct their attention to the development of high-quality engine mounting devices in order to ensure that improved comfort in riding and silencing shall not be offset by fatiguing vibration effects. In this paper an engine mounting bracket is designed to reduce the transmission of engine vibration to the chassis.

A.s.adkine, V.s.kathavate, G.p.overikar, S.n.doijode [15] –

The authors have modeled and designed an engine mounting bracket and analyzed for carrying static loads. Also, modal analysis was carried out to find natural frequency of engine mount. If this natural frequency were to match with excitation frequency, resonance would occur. This work is carried by using ERW-1, ERW-2, aluminum and magnesium alloys for the engine mount bracket. The results are analyzed for stresses and deformations.

Sandeep Maski, Yadavalli Basavaraj [16]-

The authors have considered an engine mounting bracket made of three different materials i.e. cast iron, wrought iron and mild steel. The main objective is to select the best material from the obtained result under prescribed conditions., modal analysis and static analysis carried out by which maximum von-misses stress and natural frequency are computed.

Monali Deshmukh, Prof. K R Sontakke [17]-

In this paper, authors have studied harmonic response and vibration damping of an engine mounting bracket. If the brackets have their resonance frequencies close to the operating engine frequencies, then the large amplitude of vibration get generated which may cause its fatigue failure or breakage, thus reducing its estimated or desired life. And if the harmonic response values of bracket is more than acceptable range it results in to generation of noise. Hence it is required to check the harmonic response of designed bracket. Vibration damping can be either provided by using separate dampers (anti-vibration mounts) or by suitably deciding the material and dimensions of the brackets. During its operation, the undesired vibrations generated by the engine and road roughness can get directly transmitted to the frame through the brackets. This may cause discomfort to the passenger(s) or vibrations might even damage the chassis. Existing bracket design is optimized to meet the above requirements.

Umesh S Ghorpade [18]-

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 [19]-

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. Here the weight of the material is reduced and preprocessing 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.

P.D. Jadhav [20]-

This work is a contribution to the development of new material for engine mounting bracket. The results obtained for the static structural and modal analysis have shown that the magnesium is better than aluminum. From the results it can be seen that the magnesium bracket is safe for the required application. It will help in decreasing the weight of the power train assembly, which can increase fuel efficiency.

Sagar V. Birari [21]-

Design and Analysis of Engine Mounting Bracket Engine mounting bracket plays very significant role in reducing noise, vibration and harshness caused due to engine and thus has very effective role in improving vehicle comfort. This current work accounts for the investigation of engine mounting bracket by using Hyper mesh and Opti struct approach. Static analysis of engine mounting bracket was done in order to check design of existing and modified bracket. The results were analysed for stresses and deformations. The design was tested for different design of Mild Steel with different thickness. From Design and analysis, it is considered that stresses induced in the bracket were 262.00 Mpa and deformation 9.5 mm. It can be anticipated that modified brackets can be considered for desired application.

Mr. Sagar B. Awate [22]-

Experimental Validation & Testing of Brake Chamber Mounting bracket Experimental validation & testing is one of the important parts in design finalization. Now a days advances software’s are developed to virtually test & validate the design of component but still physical validation & testing is necessary in order to check reliability, maintainability, performance of the designed component. This paper is about, physical validation & testing of newly designed brake chamber mounting bracket.

Jasvir Singh [23]-

Design of Engine Mount Bracket for a FSAE Car Using Finite Element Analysis Engine mounts have an important function of containing firmly the power-train components of a vehicle. Correct geometry and positioning of the mount brackets on the chassis ensures a good ride quality and performance. As an FSAE car intends to be a high-performance vehicle, the brackets on the frame that support the engine undergo high static and dynamic stresses as well as huge number of vibrations.

Rajath J K [24]-

Dynamic Response Analysis of Compressor Mounting Bracket of an Automobile Vehicle for Aluminum 6061T6 The compressor plays a vital role in the air conditioning system of an automobile. The compressor mounting bracket is a rigid structure which is used to mount the compressor to the engine. Design includes modelling of the bracket by considering all the constraints. Analysis comprises of normal modes analysis and frequency response analysis for aluminium 6061T6. The aim of this work is to find the natural frequency of the bracket for the self-weight by modal analysis and the stresses induced in the bracket due to external excitation by frequency response analysis using FEA method. The analysis is performed using ABAQUS tool and the results are interpreted.

Tushar P. Kamble [25]-

Optimization & Modal Analysis of Engine Mounting Bracket for Different Materials by Using Finite Element Analysis The Engine in the vehicle is one of the most important components of on road vehicle such as car. High performance sports car has their engine component supported by the mounting bracket to its chassis frame. It plays a very much important role in improving the comfort & work environment of a car as well as the engine component. The improvement of the engine bracket system has been the subject of intense interest for many years. It is required to design the proper engine mounting bracket for a road vehicle.

Sahil Naghate [26]-

Modal Analysis of Engine Mounting Bracket Using FEA The engine mounting plays an important role in reducing the noise, vibrations and harshness for improving vehicle ride comfort. The first and the foremost function of an engine mounting bracket is to properly balance the power pack (engine &transmission) on the vehicle chassis for good motion control as well as good isolation. Present work deals with FEA analysis of engine mounting bracket. It includes the modeling of the engine mounting brackets by changing the material of component.

Lv Lin [27]-

Strength Performance Analysis and Improvement of Engine Mounting Bracket for a Commercial Vehicle With the rapid development of the automotive industry and the increasing demand of consumers for the quality of automotive products, improving the reliability of automotive products has become the "top priority" of enterprises. While improving the reliability level of engine mounting bracket for commercial vehicles, it can not reduce the mechanical strength performance of the structure

L. Barelli and et al [28]-

Author given the detail of vibration acoustic pressure non-intrusive measurement techniques for diesel engine. The aim of the paper is to develop diagnosis methodology internal combustion engines (ICE) quality, by evaluating microscopic working parameters measured with noninvasive instruments. The diagnosis methodology is based on the categorization of working conditions by means of acoustic and vibration measurements and relating the data to the pressure inside the cylinder.

S.Vulli, J.F.Dunne, and et al [29]-

Author identified sources of IC engine block vibration using Short Term Fourier Transform (STFT) from single point acceleration measurement taken with commercial knock sensor. A series of experimentation is discussed to create the data to isolate this different event on a 3 cylinder gasoline engine. The paper shows that a single time frequency analysis method applied to sensor data in the form of an appropriately tuned STFT, can effectively identify the occurrences of these events in the time domain if response are adequately separated and strong enough

Lee, [30]-

In this paper, The vibration of generator was measured and analyzed to determine the frequency and amplitude level of vibration that generator may experience during the operation and transport of portable generator. In a portable generator engine vibrates when running. This vibration is transferred to the generator‟s metal frame. To measure this vibration two accelerometers are mounted perpendicular to the engine shaft‟s rotation in the horizontal and vertical axis. Two different sized (5250 and 2250 watt) generator were tested. The transportation test showed small acceleration and displacement peaks compared to operating the generator

# *Conclusion*

The following conclusion has been reached based on the literature review that was conducted:

1. From the above literature survey, it has been found that Maximum researchers have done their study on material selection for support bracket very few researchers have tried for development of bracket on basis of weight.
2. From the above literature survey, it has been found that there is an effective weight reduction in bracket by using ODE and RSM method for advanced stainless-steel material.
3. There is evidence of work done for design and optimization on conventional material of bracket enclosure to protect the bracket against crack, ground impact, and abusive condition scenarios.
4. From the literature review it is observed that some researchers have tried to reduce weight of the support bracket, no researcher has tried for the weight optimization of support bracket.
5. ***References***

[1] Optimization of Compressor Mounting Bracket of a Passenger Car by Sachin Kalsi1 Daljeet Singh, J. S. Saini2

[2] Modal Analysis of Engine Mounting Bracket Using FEA by Yongkang Zhao

[3] Multi-axial Fatigue Analysis of Oil Suction Tube Bracket based on Multi Body

Dynamics Solution of Inline 6-Cylinder Diesel Engine by Jozo SKOKO and Marko BASIC, Thomas RESCH, Brett SHEETS, Stefan BRUNNER

[4] Root cause analysis in quality problem solving of research information systems: a case study by Mohammad Javad Ershadi, Roozbeh Aiasi and Shirin Kazemi

[5] One Step Formability Analysis of Fuel Filter Bracket using Siemens NX by Chandranath.M, Dr. D. Ramegowda and Dr G Mallesh

[6] Simulation and experimental validation of powertrain mounting bracket design obtained from multi-objective topology optimization by Qinghai Zhao, Xiokia chen.

[7] Topology Optimization of Front Leaf Spring Mounting Bracket by Vijay Kalantre, K. H. Munde, Ashish Pawar.

[8] A failure analysis method for designing highly reliable product-service systems by Koji Kimita, Tomohiko Sakao and Yoshiki Shimomura.

[9] An Improved Method of Failure Mode Analysis for Design Changes by Rafael Laurenti, Henrique Rozenfeld

[10] Finite Element Analysis of Fuel Tank Mounting Bracket by Abhishek P, Hardeep S M, Sarvocch G, Anil L.

[11 ] Umesh S. Ghorpade, D. S. Chavan, Vinaay Patil, Mahendra Gaikwad, ―Finite Element Analysis And Natural Frequency Optimization Of Engine Bracket‖, International Journal of Mechanical and Industrial Engineering, ISSN No. 2231 – 6477, Vol-2, Iss-3, 2012

[12] Abdolvahab Agharkakli, Digvijay Pradip Wagh, ―Linear Characterization of Engine Mount and Body Mount for Crash Analysis‖, International Journal of Engineering and Advanced Technology ISSN: 2249 – 8958, Volume-3,Issue-2, December 2013

[13] Dr.Yadavalli Basavaraj, Manjunatha.T.H, ―Design Optimization of Automotive Engine Mount System‖, International Journal of Engineering Science Invention ISSN (Online): 2319 – 6734, ISSN (Print): 2319 – 6726

[14] P.D. Jadhav , Ramakrishna, ―Finite Element Analysis of Engine Mount Bracket‖, International journal of advancement in engineering technology, management and applied science ISSN no. 2349-3224

[15] Vikas Doundkar, Dhananjay Ghatage, Manish Madkaikar, Aniruddha Kulkarni, ― Topology Optimization for Engine Mounting Arm with Fatigue‖, HTC2011

[16] Koushik . S―Static and Vibration Analysis of Engine Mounting Bracket of TMX 20-2 using OptiStruct‖, Altair Technology Conference 2013, India.

[17] R Singh ―Dynamic design of automotive systems: Engine mounts and structural joints‖, Saadhanaa, Vol. 25, Part 3, June 2000, pp. 319±330.

[18] Umesh S. Ghorpade, D.S. Chavan, Vinaay Patil &Mahendra Gaikwad, “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.

[19] Mr. Pramod Walunje, Prof. V.K. Kurkute, “Optimization of Engine Mounting Bracket Using FEA” IJR, Volume: 2, Issue: 12, Page No. 72-75, Dec 2013

[20] P.D. Jadhav, Ramakrishna,” Finite Element Analysis of Engine Mount Bracket” IJAET, Vol. 1, Issue 4, Page1-10,September 2014

[21] Sagar V. Birari , Dr. Monimoy Saha“Design and Analysis of Engine Mounting Bracket” ISSN 2394 – 3386 Volume 5, Issue 3 March 2018

[22] Mr. Sagar B. Awate, Prof. Vidyasagar R. Bajaj2 “Experimental Validation & Testing of Brake Chamber Mounting Bracket” IRJET Volume: 03 Issue: 06 ,June- 2016

[23] Jasvir Singh Dhillon, Priyanka Rao, V.P. Sawant, "Design of Engine Mount Bracket for a FSAE Car Using Finite Element Analysis" ISSN: 2248-9622, Vol. 4, Issue 9 (Version 6), September 2014, pp.74-81.

[24] Rajath J K, Dr. L Chandrasagar “Dynamic Response Analysis of Compressor Mounting Bracket of an Automobile Vehicle for Aluminum 6061T6” IRJET Volume: 04 Issue: 09 | Sep -2017

[25] Tushar P. Kamble , Rajratna A. Bhalerao “Optimization & Modal Analysis of Engine Mounting Bracket for Different Materials by Using Finite Element Analysis” IRJET Volume: 03 Issue: 10 Oct-2016

[26] Sahil Naghate, Sandeep Patil, “Modal Analysis Of Engine Mounting Bracket Using FEA”, International Journal of Engineering Research and Applications (IJERA), Vol. 2. Issue4. July-August-2012,pp.1973-1979.

[27] Lv Lin, Wang Jizhong, Chen Shuai “Strength Performance Analysis and Improvement of Engine Mounting Bracket for a Commercial Vehicle” American Journal of Mechanics and Applications Volume 7, Issue 2, June 2019

[28] L. Barelli , G. Bidini, C.Burati, R.Marioni in their paper entitled „Diagnosis of internal combustion engine through vibration acoustic pressure non- instructive measurement‟; Applied Engineering 29 (2009):1707-1713

[29] S.Vulli, J.F.Dunne, R.Potenza, D.richardson „Time frequency analysis of single point engine block vibration measurement for multiple excitation-event identification‟, Journal of sound and vibration; Oct.2008:1129-1139

[30] Arthur Lee, Directorate of engineering science, Portable generator vibration measurement‟, January 2007: 5-19