MODELLING TECHNIQUES FOR ENGINE VIBRATION ANALYSIS: A REVIEW

[1]Rohit Jain , [2]Gaurav Saxena, [3]Ankit Singh Chauhan
[1]Research Scholar, Department of Automobile Engineering, RJIT BSF Academy, Tekanpur, Gwalior (M. P.)-474005, India
[2] Assistant Professor, Department of Automobile Engineering, RJIT BSF Academy, Tekanpur, Gwalior,(M. P.)-475005, India
[3]Assistant Professor, Department of Mechanical Engineering, ITM University, Gwalior,(M. P.)-475001, India

Abstract:-This review paper is focused on the application of modelling techniques for engine vibration analysis. The application of modeling based analysis is very important to develop and analyze models and predicting different types of engine generated vibrations. It is also useful for predicting the nodal displacements, stress generated, frequencies in term of Eigen values, and Mode shapes. The review initially focuses on finite element analysis as most popularly used investigation technique for the analysis of engine developed vibrations .Recent researches on FEA simulations are performed using ANSYS, HyperMesh ABACUS solver, COMSOL, HyperMesh OptiStruct solver. Besides FEA, mathematical modeling of developed models is done by researchers using tools like MATLAB, Directed Tabu Search (DTS), leap frog method etc. This comprehensive review of the various modeling techniques for engine vibration analysis will be useful for academician, scientist and researchers working for selection of most appropriate and viable technique for engine vibration analysis analysis.

IndexTerms:- Engine Mountings, vibration analysis, simulation, FEA, mathematical modeling.

INTRODUCTION

One of the biggest challenges in today's automotive development which is faced is lowering the fuel consumption of the vehicles. Considering the desirable feature of sports car which primarily consists of speed, racing ability and its performance. One obvious way to increase the fuel efficiency of a vehicle is to reduce its mass. But reduced masses result in increasing problems with vibrations and the dynamical behavior in general. More vibrations mean additional wear and reduction in comfort and the comfort is an important quality characteristic which can influence the customers purchase decision.

The main sources of vibrations in vehicle are the engine induced vibrations. Some of parts are identified by the researchers and engine manufacturers as the vibration producing parts. The prime initiator of vibration source are components like engine block, engine head, piston, connecting rod, crank shaft, flywheel and cam shaft, valves, manifolds, pulleys etc **[26]**, these excites engine vibrations. Due to faulty design and poor manufacturing there is unbalance in the engines which causes excessive stresses in the rotating system because of vibration. The vibration also leads to rapid wear of contact parts such as bearings and gears. Unwanted vibrations may cause loosening of parts, damage frame and engine mounting due to stress, the consequences of which may be premature failure and reduction in service life of the product.
Simulation is basically a process, in which any creation and existing design is analyzed before its practical application. In simulation, analysis is done by static analysis, modal analysis, dynamic analysis for determination of design and application parameters like stress, displacement, fatigue life, thermal conductivity etc. In present, many simulation and CAD software are available for various field, i.e. mechanical engineering, electrical engineering, civil engineering, medical science etc. In IC Engine generally simulation is performed to predict the behavior of combustion chamber, flow of fluid (gas, mixture), temperature and vibration.

II MODELLING TECHNIQUES FOR ENGINE VIBRATION ANALYSIS

Many researchers are working on designing, analysis and optimization of engine mounting by using different types of methodologies. Researchers have applied different kind of simulation software's or mathematical modeling for developing accurate results of their research. Through analysis of significant works available in form of articles published in journals, conference, various working groups' reports and many websites a review is being presented over here. The present review article emphasis mainly on significant research carried out in past years. Here a critical summary of research carried out on analysis of engine mount is presented to get an overview of methodologies adopted by researchers, type of analysis and their significant outcomes. The review is being classified on the basis of modeling technique` used by the researcher for the analysis of engine mount bracket and engine vibration analysis, the classification is done as:-

1) ANSYS Simulation.
2) HyperMesh ABACUS Simulation.
3) HyperMesh OptiStruct Simulation.
4) COMSOL Simulation.
5) Mathematical Modeling
   - MATLAB Simulation
   - Miscellaneous Techniques

2.1 ANSYS SIMULATION

ANSYS Mechanical simulation is a finite element analysis tool for structural analysis, including linear, nonlinear and dynamic studies. This computer simulation product provides finite elements to model behavior, and supports material models and equation solvers for a wide range of mechanical design problems. ANSYS Mechanical also includes thermal analysis and coupled-physics capabilities involving acoustics, piezoelectric, thermal–structural and thermo-electric analysis. ANSYS is computer simulation tool for simulating the response of materials to short duration severe loadings from impact, high pressure or explosions **[27].**

Static structural as well as modal analysis was done by **Chaudhari Pavan B. et al, 2012 [1]** for engine mounting bracket using FEA. Materials viz. aluminium (Al), Magnesium (Mg) and Cast Iron (CI) were analyzed through natural frequency of engine mounting bracket using FEA software ANSYS. The result of the analysis shows that the natural frequency verses stress and stress verses deformations for different materials with various mesh sizes follow the same trend. The result of the analysis concluded that Mg alloy bracket has highest natural frequency followed by Al alloy and CI. Using same FEA tool **GHORPADE UMESH S. et al, 2012 [2]** conducted modal analyses on Engine bracket to obtain natural frequencies for various material viz. Al alloy, Mg alloy and Gray C.I under consideration. They analyzed the material stress and their deformation through Ansys. The results of the analysis concluded that natural frequency of Mg alloy is comparatively better than Al alloy and Gray Cast iron Engine bracket.

Chen Shuxun et al, 2012 [3] conducted an analysis through FEA software using ANSYS for engine Power-train mounting system which is equivalent to a composite body of one inertial body and one elastic body. In equivalent analysis model method, component of mounting are simplified as one spring along its three principal elastic axes and the power-train mass and rotational inertia are accurately embodied. The results of analysis obtained for vibration modes and energy decoupling degrees in given directions are shown in table 1.

Table1: Energy Decoupling Degrees in various Directions **[3].**

Direction	**X**	**Y**	**Z**	**RX**	**RY**	**RZ**
Frequency (Hz)	9.1	8.2	11.3	15.6	11.3	15.5
Decoupling Degree (%)	87	48	46	90	49	84

Adopting the same methodology of FEA static simulation and experimental analysis was done by **Mr. Walunje Pramod et al, 2013[4]** on conventional materials such as Gray Cast Iron, Al alloy and Mg alloy. In this case also ANSYS was used for static and modal analysis. Experimental analysis followed the simulation based analysis considering same materials under investigation. The results of the analysis concluded that Al alloy is optimum material whose stress is within yield value and obtained frequencies is more compared to both materials. Later an optimization through weight reduction by 21.8% of the mounting bracket is achieved.

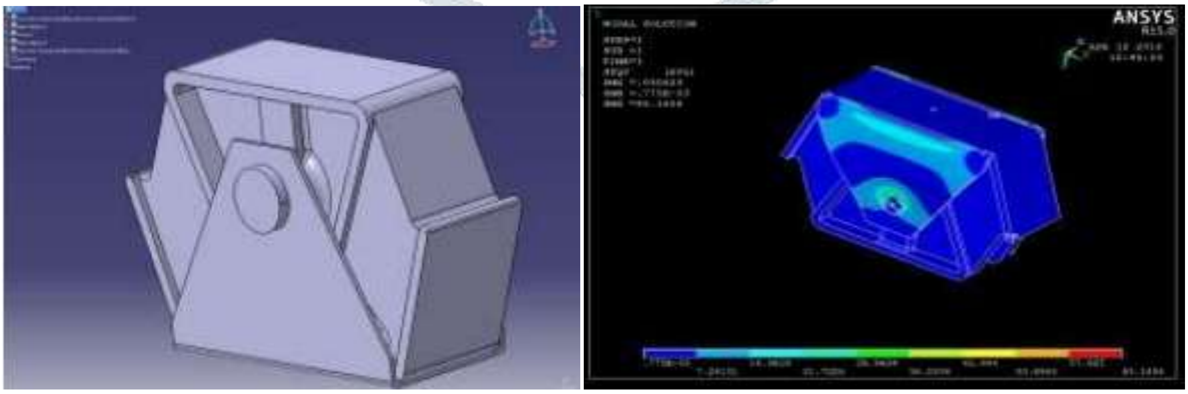
Later a significant static and modal analysis on engine mounting bracket using square cross section was performed by **Jadhav P.D. et al, 2014 [5]**. The analysis was done using FEA software ANSYS of the engine mounting brackets by changing the material (Al alloy & Mg alloy) of component. The outcome from their research showed Mg alloy as best opted material for square cross section engine.

Choudhury, Promit et al, 2014 [6] carried out basic research on the mathematical modeling and shape optimization of front damper mount of Ashok Leyland 1612 truck. The researchers worked towards the optimization of the damper bracket with constraints of stiffness, strength and natural frequency. The geometric model was developed on modeling software SOLIDWORKS and further analysis was carried out on ANSYS. The obtained result showed that the displacement and stress were reduced in the new optimized design. The optimized mass of bracket gets reduced up to 21.15%.

Similarly, static and modal analysis was done by **Adkine A.S. et al, 2015 [7]** of engine mounting bracket. They examined the natural frequency of engine bracket analytically and through developing the model and self excitation frequency of engine bracket. ERW-1, ERW-2, aluminum and magnesium alloys for the engine mount bracket. For analysis Ansys was used as FEA solver. Shape optimization and material optimization was done on different materials. The results showed that best material for engine bracket was ERW-2, which is higher in excitation frequency .

The researcher **Deshmukh, Monali et al, 2015[8]** conducted static as well as modal analysis of engine mounting bracket. The FEA analysis was done on ANSYS 15.0. The result obtained through analysis were reduction in weight of proposed model of the engine mounting bracket by 12.5%, maintaining an acceptable level of yield stress and harmonic response.

Vinchurkar Sanket et al, 2016 [9] performed optimization of engine mounting bracket of Mahindra vehicle. The researcher designed and analyzed engine mounting bracket without changing the supporting locations and types of support of original engine. Geometric modeling was done by CATIA V5 and post processing through FEA software Ansys in all conditions. The study concluded that the maximum stress is 65.16 N/mm^2 within the safety limit and total weight gets reduced by 15% without affecting its strength. The model of the engine mounting bracket of Mahindra Scorpio developed by the researcher is shown in **Fig.1**



(a) **(b)**

Fig.1: (a) Geometric Model of Engine Mounting Bracket in CATIAV5R19, (b) Stress contours using Ansys.

2.2 ABACUS SIMULATION

ABAQUS is a HyperMesh solver software, which is used for the modeling and analysis of mechanical components and assemblies for visualizing the finite element analysis result. A subset of ABAQUS only the post-processing module can be launched independently in the ABAQUS/Viewer product. ABAQUS is used in the automotive, aerospace, and industrial products industries **[28]**.

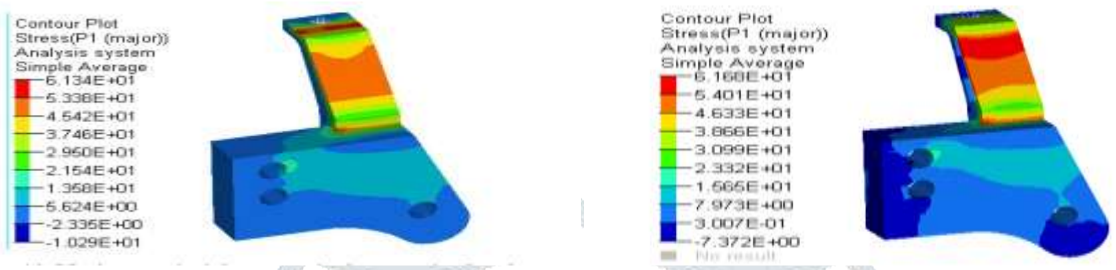
Maski Sandeep et al, 2015 [10] carried out modal and static analysis by FEA tools of engine mount bracket. They performed analysis for three materials by using Ansys .After assigning of material was discretization as done by using HYPERMESH and then analysis by solver ABACUS. The materials used are cast iron, wrought iron and mild steel for engine mount bracket. The results show that mild steel is better than other two materials under examination.

2.3 HyperMesh OptiStruct solver Simulation

OptiStruct is the HyperMesh modern solver, which is used for Structural Analysis (linear and non-linear) under static and dynamic loadings. It provides solution for structural design and optimization. This solver is used worldwide to analyze and optimize structures for their strength, durability and NVH (noise, vibration and harshness) characteristics **[29]**.

Static and Vibration Analysis of Engine Mounting Bracket for TMX 20-2 done by **Koushik . S, 2013, [11]** using OptiStruct solver. The engine mounting is modeled in Unigraphics (NX7.5) and imported to HyperMesh. Strain Gauges were placed where the brackets was failing in the field. The result obtained from the stress difference between physical testing and simulation approach shows deviation of about 18% and failure occurs where the ribs end on the bracket.

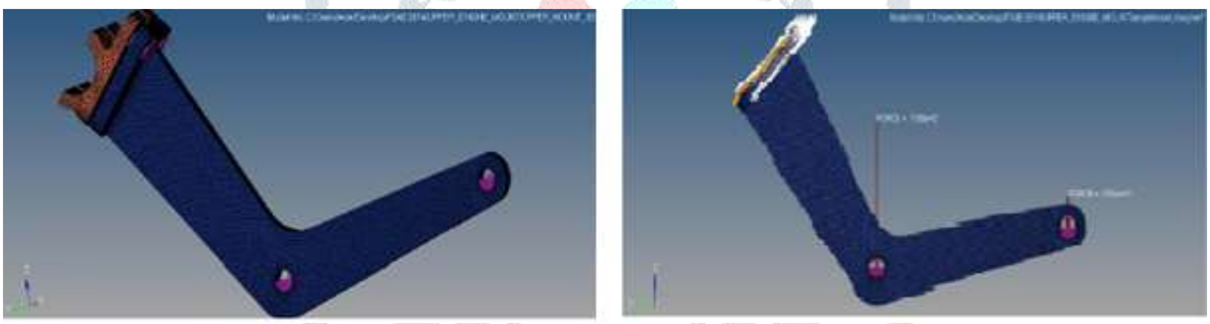
Another researchers **Reddy M. Vijay Kumar et al, 2014 [12]** conducted topology optimization in engine mounting bracket through Finite Element Analysis using solver Optistruct10.0. Results obtained after topology optimizations gives stress of 61.34 N/mm^2 and displacement up to 0.357mm, satisfying designing criteria. Further researcher also reduced mass by 0.22kgs per component. The maximum stress of the mounting bracket before and after optimization is shown in **Fig.2: (a), (b)** respectively.



(a) (b)

Fig.2: Maximum principle stresses in the mounting bracket (a) before optimization and (b) after optimization, as reported by **[13]**

Modal analysis and mass optimization was done by **Jasvir Singh Dhillon et al, 2014 [13]** on engine mount bracket for a FSAE Car. CAD modeling done by FEA software Creo 2.0 and discretization was done on HyperMesh 11 followed by analysis on FEA solver Optistruct, After designing optimization was done on engine mount bracket. The obtained results for engine mounting bracket showed 30.5% maximum deflection in worst loading case. The maximum von-mises stress increased from 29.65% maintained factor of safety 3.3. The researcher optimized the weight by 11.64%. Fig.3: (a) and (b) shows the meshed and boundary condition model of engine mounting bracket assembly.



(a) (b)

Fig.3: (a) and (b) for discretized model and applied boundary conditions on engine bracket respectively.

2.4 COMSOL SIMULATION

COMSOL Multiphysics is a multiphysics simulation software and finite element analysis, tools also. It uses conventional physics-based user interfaces and coupled systems of partial differential equations (PDEs). The analysis done by this software is useful in structural mechanics module, which included mechanical structures that are subject to static or dynamic loads. It can be used for a wide range of analysis types, including stationary, transient, eigen mode/modal, parametric, quasi-static, frequency-response, buckling, and pre-stressed **[30]**.

To reduce weight the researcher **Nag M.V. Aditya, 2012 [14]** conducted FEA analysis on different material of engine Mount bracket. Geometric model of engine mount bracket done on CATIA V16 tool. The researcher carried out computational testing of the component at isotropic state through the application of thermo-mechanical vibration analysis using the COMSOL Multiphysics 4.2 version software. For analysis two materials viz. AlSiC Composite Material and Conventional Grey Cast Iron (GCI) were compared for same model. The result shows that the AlSiC Composite material on the Engine Mounting Bracket provides the greater stability, consistent material behavior and the better performance. It also reduces weight of the component by 60%. **Fig.5: (a)** shows relation between the stress and the frequency and **Fig.4: (b)** shows relationship between the displacement and the frequency during the thermo- mechanical vibration analysis using COMSOL multi-physics software.

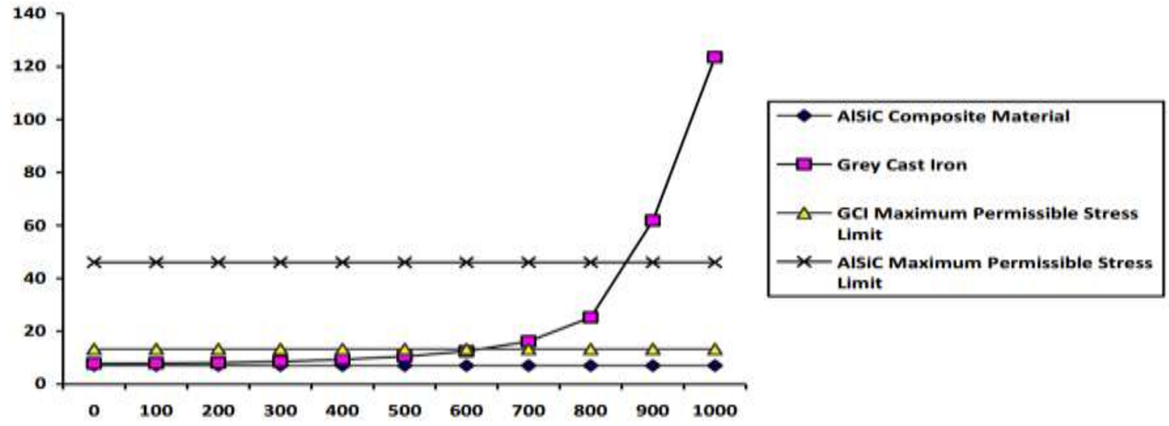


Fig.4: (a) Relationship between the stress and the frequency

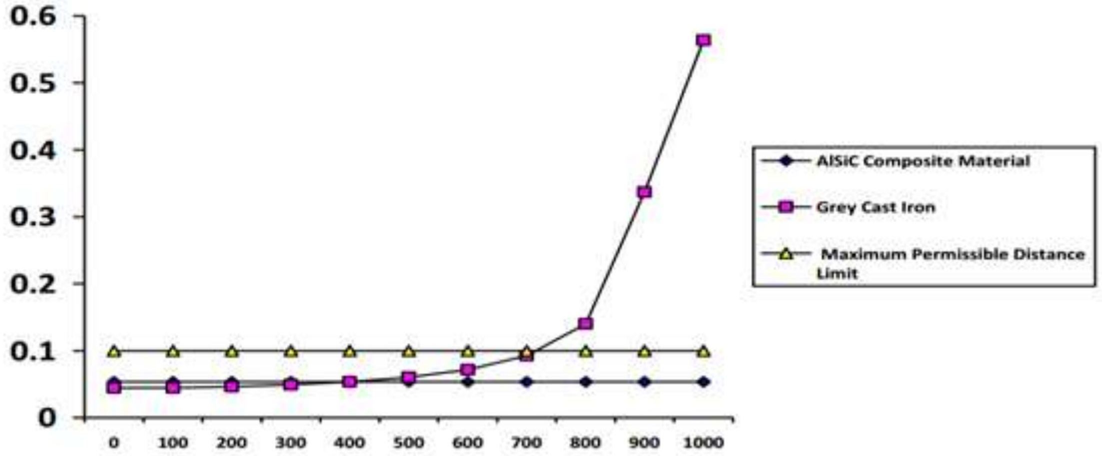
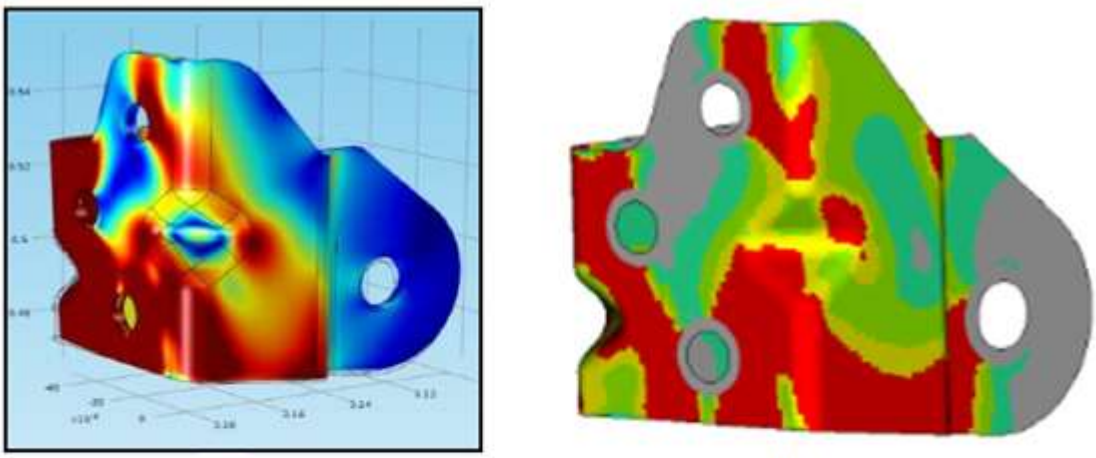


Fig.4: (b) Relationship between the displacement and the frequency.

Sameer U. Kolte [2013] [15] performed structural analysis of power train mounting bracket using COMSOL Multiphysics. The COMSOL analysis starts with a CAD model imported from CATIA V5R21. Tetrahedron is selected as the meshing element. The material is selected from 'built in' materials in COMSOL library. The bracket is subjected to different load conditions like 3-2-1g weights, 3g downwards unbalanced torque etc and is optimized so that stresses generated and deflection of the bracket is within permissible limit. Effect of bolt preloads is also considered. Results obtained from COMSOL were validated with the results from Hypermesh analysis. The correlation was extremely good in terms of values of stresses, stress pattern and displacement. The comparison of stress plot for both the analysis are shown in **Fig.5:** (a) and (b).



(A) (B)

Fig.5: Stress plot (a) COMSOL Mutiphysiscs Model (b) Hypermesh Model

2.5 MATHEMATICAL MODELING

Description of any system which is defined by mathematical concepts and language is known as mathematical model. The development of mathematical model is termed mathematical modeling. Mathematical models are used in the natural sciences and engineering disciplines. A model may help to give details a system and to study the effects of different components, and to make predictions about behavior.

2.5.1. MATLAB Simulation

MATLAB consist of two word matrix and laboratory, MATLAB is a multi-paradigm numerical computing environment and fourth-generation programming language. A proprietary programming language developed by , MATLAB allows matrix manipulations, plotting of functions and data, implementation of algorithms, creation of user interfaces, and interfacing with programs written in other languages, including C, C++, C#, Java, Fortran and Python **[31]**.

Wang Qingzheng et al, 2011[16] developed Optimization program by using MATLAB for design of specific vehicle mounting system. Dynamic model of Power-train mounting system was established and optimized on energy decoupling. Modal simulation analysis was done by using FEA tool viz. MSC.ADAMS/VIEW. After optimal design, the first-order mode and the energy distribution of the frequency bands by the optimized system were within a reasonable range. The obtained results show the maximum frequency of 17Hz and the lowest frequency of 7Hz.Therefore, a high degree decoupling of the system was achieved.

Static and Modal Analysis of engine mounting bracket is done by **Naghate Sahil et al, 2012 [17]**, they used Square Cross section for Aluminium alloy and magnesium alloy. Analysis was done using FEA tool followed by comparison for both material of engine mounting bracket. They conclude magnesium alloy is better than aluminium alloy for various operations.

The researchers **Agharkakli, Abdolvahab et al, [Dec 2013] [18]** studied linear characterization of engine mount and body mount for crash analysis by the help of MATLAB Simulations. To obtain linear mount characteristics with the help of mathematical models and comparison between their results. The mounts are treated at the component level, and mathematical models for the same are evaluated to get the required characteristics. The mounts are modeled as spring and damper system subjected to impact loading that occurs during crash events. The approximations of input pulse were 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 was also studied.

Through analytical computations for no excitation, half-sine excitation and Fourier equivalent wave, following results were obtained:-

1) The displacement predicted by half-sine excitation is equivalent to Fourier equivalent wave
2) The breakage of the mount may occur after it has displaced as predicted by half-sine excitation.
3) The boundary conditions are well satisfied by the model, and thus the mathematical model can be used in future as standard tool to predict the deformations and velocity of the mounts at the initial stage.
4) The maximum displacement always occurs when the velocity is zero, which is absolute theoretical condition.

2.5.2. Miscellaneous Techniques

Some of the researchers used distinct and customized type mathematical model for analysis and optimization of engine mountings.

Yadollah Rasekhipour et al, [July 2011] [19] determined the model which is most appropriate for the optimization of Hydraulic Engine Mounts (HEMs). Hydraulic engine mounts are optimized to improve the vehicle ride comfort using a global optimization method called Directed Tabu Search (DTS) method. A full-vehicle model was used to evaluate the optimization results, and to determine the optimized HEMs in the simplified models. Through the analysis they concluded that instead of the complex full-vehicle model which is too hard to be thoroughly modeled, a model of a 1DOF body mounted on the ground via one HEM can be used. Researchers used 13 DOF model, 6 DOF model and 1DOF model as shown **Fig.6:** (a),(b) and (c) respectively.

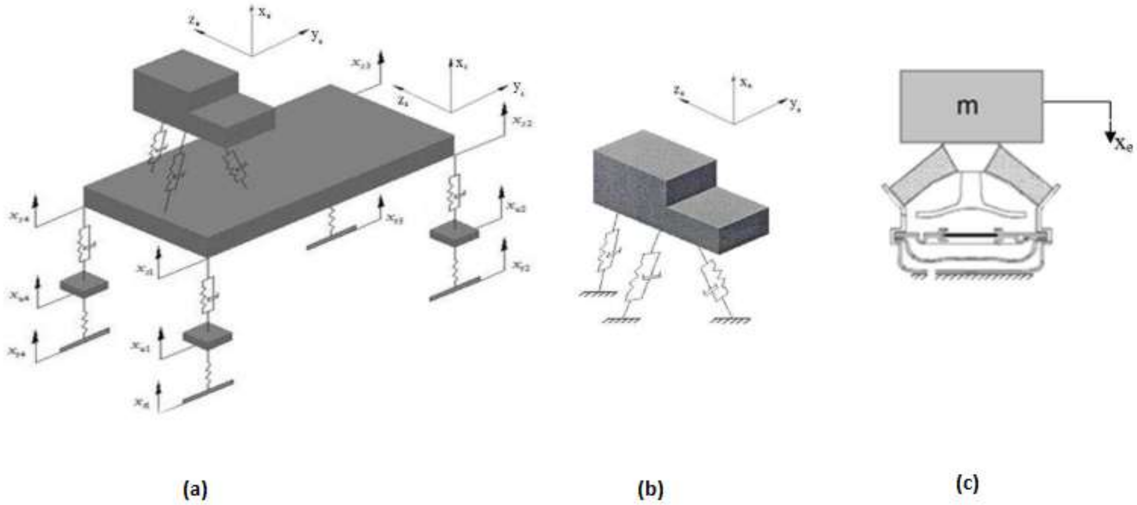


Fig.6: Optimization models, (a) Model No.1 of 13 DOF full-vehicle (reference model),(b) Model No.2 of 6DOF engine mounted to the ground via three HEMs, (c) Model No. 3 of 1DOF engine supported on a HEM.

M K.A. Zakariaet at el [2011] [20] has done fatigue damage assessment of the engine mount bracket using a statistical based approach. The researchers analyzed two types of strain signals from road loading and compared effect on fatigue damage of the engine mount bracket. Strain gauge were attached to the engine mount bracket and was connected to the data acquisition set in order to capture the actual strain signals when an automobile is driven on two different road surfaces. The strain signals were then analyzed using global signal statistic and integrated *kurtosis* based algorithm for *Z-filter (I-kaz)* method. Damage of the engine mount bracket was evaluated using finite element commercial software. The conclusion drawn by researcher shows that larger strain ranges amplitude produces more data scatter, higher I-kaz coefficient and larger damage values. This finding shows that *I-kaz* coefficient used to assess the total damage of the engine mount bracket. The result obtained from analysis is shown fig.7: fatigue damage showed an increment with the respective statistical values of the strain signals.

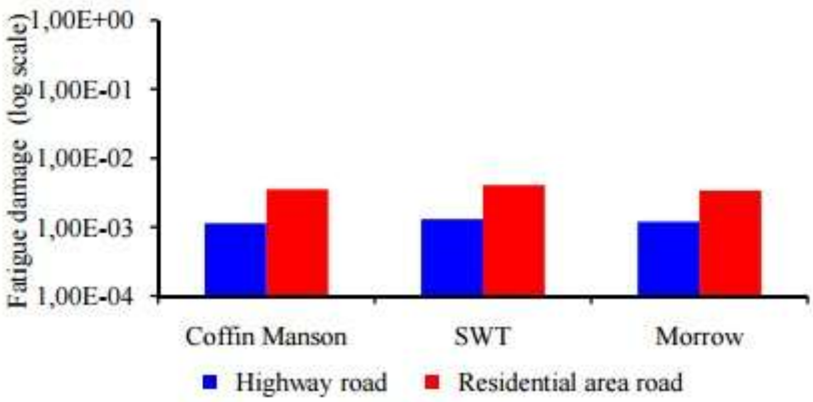


Fig.7: Fatigue damages an increment with the respective statistical values of the strain signals.

Snyman et.al [21] used "*Leap frog*" optimization algorithm to minimize the objective function of engine vibration in the mounted 4-cylinder internal combustion engine. They analyzed the mathematical model, for which balancing mass and the lead angles were taken as the design variables. The objective function used in this research was the vibratory forces from the engine, transmitted to the engine mounts. The objective function was minimized to minimize the vibratory output of the engine.
Chung–Ha et al, [22], presented simplified method to determine the vibrational amplitude developed by a 4-cylinder engine when supported on visco-elastic mounts. They modeled the engine parts as rigid bodies connected to the rubber mounts which were modeled with spring and damping elements. The location, orientation and stiffness of the mounts can easily be optimized to reduce vibration and noise in the engine design.
Hoffman et al, [23] developed a seven degree-of-freedom model for low frequency engine vibrations that utilizes two way coupling assumption. They compared results of the two way coupling model with the one way coupling model. Also they identified that the new model properly conserves energy and account for gravitational forces.
Deana. M. Winton et al, [24] conducted an experimental study to determine the rigid body modal content of engine block vibration on a modern heavy-duty inline six-cylinder Diesel engine. They used three engine mounts fitted with multi-axis force transducers and utilized standard modal analysis to determine rigid body modal characteristics and engine mount forces signatures of the engine vibration modes of engine block.
Hoffman et al, [25] conducted an experiment on heavy duty in-line six-cylinder Diesel Engine to measure all the three orthogonal vibration force components at the each of the three engine mounts during standard impact-excitation. Modal identification tests on the quiescent-engine and during engine operation were conducted.

OUTCOMES OF REVIEW

This work provides a comprehensive literature review of existing research carried out in terms of design, stress, modal, optimization analysis of the engine mounting bracket. An effort has been made to comprise all the important contributions to this area and highlighting the most pertinent literature available for investigating the engine mounting bracket. The concluding remarks from the current review are as follows:-

a) From the review of available research on engine mounting bracket, it is apparent that most of the research conducted is purely simulation based on Finite Element Method for economical consideration and validated through experimentation.
b) Out of all the simulation techniques discussed in the present article, Ansys simulation package is most widely used by the researches, due to its ease of use and accurate results to access strength of the engine mounting bracket. The same is also applied for shape optimization and material selection of engine mounting bracket.
c) Both static and modal analyses are conducted in all the mentioned simulation techniques on engine mounting bracket considering engine as a rigid model using.
d) The review shows that engine mounting bracket are designed using different types of materials through simulation for determination of most suitable material.
e) Simulation through COMSOL multiphysics and mathematical modeling also provides results similar to conventional FEA tools.

REFERENCES

[1] Chaudhari Pavan B., Panchagade Dr.D. R., "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), Vol. 2, Issue 5, ISSN: 2248-9622, September- October 2012, pp.1016-1020.
[2] Ghorpade Umesh S., Chavan D. S., Patil Vinaay And Gaikwad Mahendra., "FINITE ELEMENT ANALYSIS AND NATURAL FREQUENCY OPTIMIZATION OF ENGINE BRACKET", International Journal of Mechanical and Industrial Engineering (IJMIE), Vol-2, Iss-3, ISSN No.2231 –6477, 2012.
[3] Chen Shuxun, Wei Qifeng, Huang Jincheng, Yin Guobao and Li Zhiqiang, "An Equivalent Model for Modal Analysis of Engine Mounting System", Electronic & Mechanical Engineering and Information Technology (EMEIT), 2012, Pp. 1128-1132.
[4] Mr. Walunje Pramod and Prof. Kurkute V.K., "Optimization of Engine Mounting Bracket Using FEA", PARIPEX - INDIAN JOURNAL OF RESEARCH, Vol- 2, Issue : 12, ISSN - 2250-199,Dec 201.
[5] Jadhav P.D. and Ramakrishna., "Finite Element Analysis of Engine Mount Bracket", International Journal of Advancement of Engineering Technology, Management & Applied Science, Vol-1, Iss-4, ISSN No. 2349-3224, 4 September2014.
[6] Choudhury Promit and Krishna Chinmaya Hemanth, "Mathematical Modelling and Shape Optimisation of Front Damper Mount of Ashok Leyland 1612 Truck Using 3d Finite Element Method", The International Journal Of Engineering And Science (IJES),Vol-3, Issue-4, ISSN 2319 – 1805, 2014, Pp.70-76.

[7] Adkine A.S., Kathavate V.S., Overikar G.P. and Doijode S.N., "Static Behaviour of Engine Mounting Bracket", International Advanced Research Journal in Science, Engineering and Technology, Vol. 2, Issue 4, ISSN No- 2394-1588, April 2015.

[8] Deshmukh Monali and Prof. Sontakke K. R., "Analysis and Optimization of Engine Mounting Bracket", International Journal of Scientific Engineering and Research (IJSER), Vol-3, Issue 5, ISSN 2347-3878, May 2015, Pp. 131-136.

[9] Vinchurkar Sanket, Prof. Khanwalkar P.M., "A Review on Optimization of Engine Mounting Bracket", International Journal of Engineering Trends and Technology (IJETT), Vol-35 No- 1, ISSN: 2231-5381, May 2016, Pp. 47-49.

[10] Maski Sandeep and Basavaraj Yadavalli, "FINITE ELEMENT ANALYSIS OF ENGINE MOUNTING BRACKET BY CONSIDERING PRETENSION EFFECT AND SERVICE LOAD", International Journal of Research in Engineering and Technology (IJRET), Vol-04, Issue-08, ISSN: 2321-7308, August-2015.

[11] S Koushik, "Static and Vibration Analysis of Engine Mounting Bracket of TMX 20-2 using OptiStruct", Altair Technology conference India, 2013.

[12] Reddy M. Vijay Kumar, Babu P. Chitti and Bharathi K. C. K., "Analysis of Light Motor Vehicle Component Using Topology Optimization Method", International Journal of Emerging Technology and Advanced Engineering, Vol-4, Issue-1, ISSN 2250-2459, January 2014, Pp.211-215.

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

[14] Nag M.V. Aditya, "Material Selection and Computational Analysis on DOHC V16 Engine's Mounting Bracket Using COMSOL Multiphysics Software".

[15] Kolte Sameer U., Neihguk David, and Prasad Abhinav, "Structural Durability Analysis of Powertrain Mounting Bracket", COMSOL conference in Bangalore, 2013.

[16] Wang Qingzheng, Wang Liangmo and Tan Li, "Automotive Vehicle Powertrain Mounting System Optimum Design and Simulation Analysis", IAJC-ASEE International Conference, ISBN 978-1-60643-379-9, 2011.

[17] Naghate Sahil and Patil Sandeep., "Modal Analysis of Engine Mounting Bracket Using FEA", International Journal of Engineering Research and Applications (IJERA), Vol. 2, Issue4, ISSN: 2248-9622, August 2012, Pp.1973-1979.

[18] Agharkakli Abdolvahab and Wagh Digvijay Pradip, "Linear Characterization of Engine Mount and Body Mount for Crash Analysis", International Journal of Engineering and Advanced Technology (IJEAT), Vol-3, Issue-2, ISSN: 2249 – 8958, December 2013,Pp.37-43.

[19] Rasekhipour Yadollah and Ohadi Abdolreza, "Optimization of Hydraulic Engine Mounts through Simplified and Full Vehicle models", World Congress on Engineering (WCE) London, U.K, Vol-3, ISSN: 2078-0958, July 2011.

[20] Zakaria K.A., Abdullah S., Ghazali M.J., Nuawi M.Z. and Padzi M.M., "Fatigue Damage Assessment of the Engine Mount Bracket Using a Statistical Based Approach", Advanced Materials Research Trans Tech Publications Switzerland, Vol-197-198, 2011, pp. 1631-1635.

[21] Snyman J.A., Heyns P.S., and Vermeulen P.J., "Vibration isolation of a mounted engine through optimization", Mech. Mach. Theory, Vol- 30, 1995, Pp.109-118.

[22] Suh Chung-Ha and Smith Clifford G., "Dynamic simulation of engine mount system", SAE ISSN No 971940, 1997.

[23] Hoffman D.M.W. and Dowling D.R., (1999) "Limitations of rigid body descriptions for heavy-duty diesel engine vibration", ASME Journal of Engineering for Gas Turbine and power, ISSN No-197-204.

[24] Winton Deana M. and Dowling David R., "Modal content of Heavy-duty diesel engine block vibration",SAE, 1997, Paper#971948.

[25] Hoffman D.M.W. and Dowling D.R., "Limitations of rigid body descriptions for heavy-duty dieselengine vibration", ASME Journal of Engineering for Gas Turbine and power, 1999,Pp.197-204.

[26] Ramachandran T. and Padmanaban K. P.,"REVIEW ON INTERNAL COMBUSTION ENGINE VIBRATIONS AND MOUNTINGS", International Journal of Engineering Sciences & Emerging Technologies, Vol-3, Issue 1,ISSN: 2231 – 6604, August 2012, Pp.63-73.

[27] https://en.wikipedia.org/wiki/Analysis, assessed on dated 15 December 2017.

[28] https://en.wikipedia.org/wiki/Abaqus, assessed on dated 15 December 2017.

[29] http://www.altairhyperworks.in/product/OptiStruct, assessed on dated 15 December 2017.

[30] https://en.wikipedia.org/wiki/COMSOL_Multiphysics, assessed on dated 15 December 2017.

[31] https://en.wikipedia.org/wiki/MATLAB, assessed on dated 15 December 2017.