IJARSCT

International Journal of Advanced Research in Science, Communication and Technology

International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal

ISSN: 2581-9429 Volume 5, Issue 14, April 2025 Impact Factor: 7.67

A Review on Analysis and Optimization of Engine Mounting Bracket

Rahul J. Zunj[1], Shailesh S. Pimpale[2], Pruthviraj D. Patil[3]

PG Student, Department of Mechanical Engineering[1]

Associate Professor, Department of Mechanical Engineering[2,3]

JSPM's Rajarshi Shahu College of Engineering, Tathawade, Pune, India

[1]rahulzunj@gmail.com; [2]shailesh_pimpale@rediffmail.com; [3]ppruthviraj1983@gmail.com

Abstract: *The engine mounting bracket is a vital component in the automotive industry, serving to support the engine and connect it to the vehicle chassis while mitigating the transmission of vibrations and noise. The engine mounting bracket is a fundamental component in automotive powertrain systems. As automotive design advances toward improved performance, fuel efficiency, and reduced emissions, the need for lighter, stronger, and more efficient engine mounting systems has become increasingly important. This review paper aims to provide a detailed overview of the various techniques employed in the analysis and optimization of engine mounting brackets, focusing on enhancing their mechanical performance and weight reduction. The paper explores various analytical and computational approaches, particularly computer aided design (CAD) and finite element analysis (FEA), which is widely used for modelling and assessing stress distribution, deformation, and modal characteristics. The paper discusses the static analysis and modal analysis performed on the engine mounting bracket. Optimization techniques such as topology optimization, shape and size optimization, and the use of alternative materials are explored for improving strength-to-weight ratio and reducing overall mass without compromising structural integrity. This review aims to guide future research and development efforts toward the creation of more efficient and reliable engine mounting systems.*

Keywords: Engine Mounting Bracket, Finite Element Analysis (FEA), Static Analysis, Modal Analysis, Topology Optimization, Weight Reduction

I. INTRODUCTION

In modern automotive engineering, the engine mounting bracket plays a pivotal role in securing the engine to the vehicle frame while isolating vibrations and maintaining overall structural integrity. As part of the engine mounting system, the bracket not only supports the static and dynamic loads generated by the engine but also contributes significantly to noise, vibration, and harshness (NVH) performance, ride comfort, and safety. With increasing demand for fuel-efficient, lightweight, and high-performance vehicles, automotive manufacturers are under pressure to design engine mounting systems that are robust, reliable, and optimized for both function and cost. Traditionally, engine mounting brackets were designed based on empirical methods and over-engineered to ensure safety. However, the advent of advanced simulation techniques, optimization algorithms, and material science has enabled a more efficient and analytical approach to their design. Finite Element Analysis (FEA), modal analysis, and fatigue analysis have become indispensable tools in evaluating the structural behavior of engine mounting brackets under various loading conditions. Concurrently, optimization techniques such as topology optimization, size and shape optimization, and the use of alternative lightweight materials have made it possible to reduce weight and manufacturing costs while maintaining or improving performance. This review paper aims to provide a comprehensive overview of the research and developments in the analysis and optimization of engine mounting brackets. It covers various analytical and computational methods, recent advancements in material usage, and optimization strategies applied to enhance bracket performance. The paper also highlights key challenges and identifies future research directions to guide the design of next-generation engine mounting systems.

II. LITERATURE REVIEW

Swapnil Taksande, S.M. Fulmali [1] This paper presents an in-depth study on the design optimization of an engine mounting bracket, focusing on improving performance through stress and vibration analysis, and evaluating the use of various materials. The research highlights the essential function of engine mounting brackets in preserving structural integrity and minimizing vibrations. Finite Element Analysis (FEA) is employed to assess the structural strength and vibration damping capabilities of different materials. The proposed material for the engine mounting bracket demonstrates superior strength and durability compared to mild steel. Furthermore, the bracket's life cycle shows a significant improvement. The static analysis results indicate that the von Mises stress in wrought iron is lower than in the other evaluated materials, remaining within safe limits and below the yield strength. Modal analysis reveals that cast iron exhibits the highest first natural frequency at 236 Hz, outperforming the other materials. Consequently, cast iron is identified as a more suitable material for the engine mounting bracket, offering improved vibration damping characteristics.

Guoqing Zhang, Xueyan Li , Junxin Li , Xiaoyu Zhou, Yongsheng Zhou [2] Consumers are decreasingly seeking electric vehicles with long range and affordable prices. Featherlight construction is an effective approach for dragging range and lowering costs. To achieve this, featherlight design optimization for battery classes using 3D printing technology was conducted. Rhino software was used for 3D modeling, followed by Altair Inspire software for topology optimization. Manufacturing and assembly examination were completed using a 3D printer. Results showed a maximum relegation of 3.20 mm after topology optimization, a maximum Mises fellow stress of 240.7 MPa, and a livery stress distribution at the bottom of the type. The minimal factor of safety met design conditions at 1, and the mass dropped to 0.348 kg, representing a 49.2 drop compared to pre-optimization situations. The 3D- published type achieved mass abatement, with a bright face, low roughness, and no screwing or distortion blights. Assembly results showed tight collaboration of factors, icing dimensional delicacy and fit match product conditions. These findings lay the root for mass product of high- performance battery pack classes.

K.V.P.T. Jagannadha Rao, G. Diwakar [3] The automobile industry has evolved, leading to increased competition for quality and comfort. Designers are now focusing on engine mount bracket design, with design changes and material changes playing crucial roles. As the strength to weight ratio of materials in bulk increases, permutations with engine mount bracket design to enhance crashworthiness are considered a better alternative. This study aims to observe engine mounting bracket characteristics, such as crashworthiness and mechanical aspects during loading conditions, by engineering the engine side suspension mount bracket. This involves designing the new bracket, meshing it, converting it into a solid finite element method (FEM) model, and applying different loads to analyze the bracket's behaviour.

Hicham Fihri Fassi, Hadji Aniyou [4] The machine, a pivotal element of a vehicle, is attached to the main frame via a machine mounting type, supporting weight and operating loads. This composition aims to find the most optimal model considering resistance, environmental impact, and manufacturing cost. It involves optimizing the support by reducing its original mass by 30 and opting suitable accoutrements and manufacturing processes with minimum environmental impact. Topology optimization, environmental assessment, and manufacturing cost analysis will be combined, with four accoutrements tested and estimated. A cost analysis will compare conventional processes and 3D printing.

H. Kursat Celik, Hakan Ersoy, Ayla Doğan, Gokhan Eravci, Allan E.W. Rennie, Ibrahim Akinci [5] This study presents a strength-based design analysis approach using Finite Element Analysis (FEA) for a damaged engine mounting bracket in a converted electric vehicle. The bracket in question was specifically developed and manufactured for use in such a vehicle but experienced failure during normal operation. Consequently, the research focused on redesigning the bracket geometry to improve its performance. To prevent similar failures, it is crucial to thoroughly define strength-related characteristics such as deformation behaviour and stress distribution under expected loads. However, accurately describing these characteristics often presents a complex experimental challenge for designers. The study applied FEA to redesign the bracket, focusing on the effects of torsional loads generated by the electric motor which was identified as the primary cause of failure. Simulation results, both visual and numerical, provided clear insights into the bracket's failure behaviour, guiding the redesign process. Initial FEA results identified the regions of damage and showed stress levels that exceeded the material's strength limits. A comparison between simulation results and physical observations confirmed that the FEA accurately represented the bracket's deformation behaviour. The analysis concluded that failure was primarily due to insufficient material thickness and stress concentrations caused by geometric notches. To address these issues, the bracket geometry was revised, targeting these specific weaknesses. Final FEA results confirmed that the redesigned bracket could operate safely under defined loading conditions.

Enrico Armentania, Venanzio Giannella, Antonio Parente, Mauro Pirelli [6] In recent years, Noise, Vibration, and Harshness (NVH) have been recognized as major factors contributing to customer dissatisfaction. This study adopts the Design For X (DFX) approach, focusing specifically on Design for NVH. The primary objective was to perform a Topology Optimization (TO) of an engine bracket support based on its vibrational behavior, with the aim of minimizing vibration transmission from the engine to the chassis and thereby enhancing passenger comfort. The TO process was guided by a target function aimed at increasing the bracket’s first natural frequency, while mass reduction served as a constraint. The vibrational performance of the bracket support was evaluated using Frequency Response Function (FRF) analysis conducted through the Finite Element Method (FEM), which helped identify the resonant frequencies of various design iterations generated during the optimization. The FEM models incorporated the cylinder head to constrain the engine bracket support under analysis, along with a secondary bracket where the simulation load was applied. Results demonstrated that the TO method successfully reduced the bracket's weight by approximately 20% and increased its first natural frequency by around 10%, leading to a measurable improvement in passenger comfort.

Lv Lin, Wang Jizhong, Chen Shuai [7] The automotive industry is rapidly developing, and improving the reliability of products has become a top priority for enterprises. However, the reliability of engine mounting brackets for commercial vehicles cannot be reduced without affecting the structure's mechanical strength. Topology optimization is carried out to find the most reasonable material distribution, and structural optimization is demonstrated through conceptual design, finite element analysis, and strength tests. Product design is the most fundamental factor affecting structural reliability, and defects left in design cannot be completely solved in production and use. A commercial vehicle's engine mounting bracket broke during road testing, leading to a rapid fracture due to excessive load and stress concentration. The finite element model of powertrain mounting bracket is established, and the structure of the engine bracket is improved according to stress distribution and processing requirements. The improved strength and weight reduction make it an important reference for strength performance design.

M. V. Aditya Nag, Dareddy Ramana Reddy [8] This paper focuses on optimizing the topology of engine accessory components using finite element analysis software. The main objective is to minimize the weight of the mount bracket, considering its shape as a parameter for variation in behaviour. The stress is computed and compared for the best model under prescribed conditions. The simulation process involves creating a CAD model using SolidWorks 2016 tool, performing F.E.A. analysis for suitable loading conditions, and optimizing the topology using the Optimize module of Solidthinking Inspire 2017. The results are used to evaluate the best structure for the engine mounting bracket.

Deepak Kumar, R. A. Savanur [9] The machine mounting type assembly in the lattice front frame supports the machine and transmission member. This study focuses on understanding its structural characteristics and dynamic gist through Finite Element (FE) analyses, similar as normal mode analysis. The most suitable model is named for farther analysis, and static analyses give the maximum structural stress condition under static lading. The most maximum stresses are set up at the type, challenging structural fatigue analysis for the suspense mounting assembly.

Sandeep Maski, Yadavalli Basavaraj [10] The bracket is a vital component of the engine mount assembly. In high-performance trucks, the engine is supported by brackets that are part of an engine mounting system installed on the front chassis frame. This section of the chassis is structurally designed to accommodate and support both the engine and the transmission unit. The primary role of the engine mount bracket is to ensure the proper alignment and balance of the engine and transmission on the vehicle's chassis. Engine mounts are essential for minimizing vibrations and noise, thereby contributing to a smoother and more stable driving experience. The strength and vibration behavior of engine brackets have been ongoing concerns, as excessive vibrations and stress can result in structural failure. This study focuses on the Finite Element Analysis (FEA) of an engine mount bracket using three different materials i.e., cast iron, wrought iron and mild steel. The analyses were conducted using meshing and simulation tools such as HYPERMESH and ABAQUS. Both modal and static analyses were performed to determine the maximum von Mises stress and natural frequency for each material. The primary objective of this research is to identify the most suitable material based on the results obtained under specific conditions. Given that vibration resistance and strength are key factors in the design of engine mount brackets, this study places special emphasis on material selection. The results indicate that in the modal analysis, mild steel exhibits the highest fundamental natural frequency at 65 Hz, compared to cast iron and wrought iron. In the static analysis, mild steel also shows the lowest displacement of 1.6 mm, and its maximum von Mises stress remains below the material's yield strength. Therefore, mild steel is considered the most suitable material for the design of engine mount brackets.

III. RESEARCH GAP

Despite progress in the analysis and optimization of engine mounting brackets, key gaps remain. The application of advanced and composite materials is still limited due to insufficient understanding of their fatigue behaviour, vibration damping, and long-term performance in real-world conditions. Additionally, the effects of various manufacturing processes such as casting, forging, and additive manufacturing on material properties and bracket performance are not thoroughly investigated. There is also a lack of integrated studies combining material selection, process influence, and design optimization, which are essential for developing lightweight, cost-effective, and high-performance engine mounting brackets.

IV. CONCLUSION

In conclusion, the analysis and optimization of engine mounting brackets are essential for improving vehicle performance, safety, and efficiency. While conventional materials and methods have been widely studied, the adoption of advanced and composite materials offers promising opportunities for weight reduction and enhanced mechanical properties. However, their application remains limited due to challenges in cost, manufacturability, and long-term durability. Future research should focus on exploring the full potential of these materials, supported by robust simulation, experimental validation, and manufacturing integration. Embracing advanced and composite materials can lead to the development of lighter, stronger, and more efficient engine mounting systems.

V. ACKNOWLEDGMENT

I would like to take this time to thank Dr. Shailesh S. Pimpale, Dr. Pruthviraj D. Patil, and Dr. P. R. Sonawane for their invaluable advice in getting the work done. Additionally, I want to express my gratitude to the administration of JSPM's Rajarshi Shahu College of Engineering, Tathawade, Pune, for giving all the resources required to complete this work.

REFERENCES

[1] Swapnil Taksande, S.M. Fulmali, "Design Modification of Engine Mounting Bracket of Car", International Research Journal of Modernization in Engineering Technology and Science, Volume 06, Issue 05, e-ISSN: 2582-5208, May-2024.

[2] Guoqing Zhang, Xueyan Li , Junxin Li , Xiaoyu Zhou & Yongsheng Zhou, "Optimization Design of Battery Bracket For New Energy Vehicles Based On 3D Printing Technology", Scientific Reports, 14:13461, 2024, https://doi.org/10.1038/s41598-024-64393-x.

[3] K.V.P.T. Jagannadha Rao, G. Diwakar, "Design and Analysis of Engine Mount Bracket to Enhance the Crash Performance of the Engine Mounting System", Journal of Polymer & Composites, Volume 11, Special Issue 8, ISSN: 2321-2810 (Online), 2023, DOI (Journal): 10.37591/JoPC.

[4] Hicham Fihri Fassi, Hadji Aniyou, "Towards an Optimal Motor Mounting Bracket Using Topology Optimization Combined with Sustainability and Manufacturing Cost Analysis", Management and Production Engineering Review, Volume 14, Number 4, PP 109-116, December 2023, DOI: 10.24425/mper.2023.147208.

[5] H. Kursat Celik, Hakan Ersoy, Ayla Doğan, Gokhan Eravci, Allan E.W. Rennie, Ibrahim Akinci, "Strength Based Design Analysis Of A Damaged Engine Mounting Bracket", Journal of Failure Analysis and Prevention, 21:1315-1322, 2021, https://doi.org/10.1007/S11668-021-01177-9.

[6] Enrico Armentania, Venanzio Giannella, Antonio Parente, Mauro Pirelli, "Design for NVH: Topology Optimization of an Engine Bracket Support", ScienceDirect, Procedia Structural Integrity 26, 211–218, 2020.

[7] 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, 7(2): 21-29, July 2019.

[8] M. V. Aditya Nag, Dareddy Ramana Reddy, "Design Optimization of DOHC V16 Engine's Mount Bracket", International Journal of Pure and Applied Mathematics, Volume 118, No. 24, ISSN: 1314-3395 (on-line version), 2018.

[9] Deepak Kumar, R. A. Savanur, "Design & Analysis of Truck Engine Mount Brackets of a Truck to Improve its Performance of Life using FEA", Asian Journal of Convergence in Technology, Issue I, Volume 04, 2018.

[10] Sandeep Maski, Yadavalli Basavaraj, "Finite Element Analysis of Engine Mounting Bracket by Considering Pretension Effect and Service Load", International Journal of Research in Engineering and Technology, eISSN: 2319-1163, pISSN: 2321-7308, Volume 04, Issue: 08, August-2015.