STM JOURNALS

Journal of
Polymer & Composites

ISSN: 2321-2810 (Online)
ISSN: 2321-8525 (Print)
Volume 11, Special Issue 8, 2023
DOI (Journal): 10.37591/JoPC

http://engineeringjournals.stmjournals.in/index.php/JoPC/index

Research JoPC

Design and Analysis of Engine Mount Bracket to Enhance the Crash Performance of the Engine Mounting System

K.V.P.T. Jagannadha Rao[1,*], G. Diwakar[2]

Abstract

The gradual evolution of automobile industries has led to rise in the competition in providing the customer with quality while riding and comfort. This has compelled the designers of each component to pay close attention towards the production of high-quality components. In the present scenario of the engine mount bracket designing, the dominant parameters like design change and material change play a crucial role. In the current generation of automobiles, the chances of changing the material are getting leaner as the strength to weight ratio of different materials that can be procured in bulk has reached their peak. Hence, the permutations with design of the engine mount bracket to enhance the crash worthiness is considered to be a better alternative. The current study deals with the observation of the engine mounting bracket characteristics like crash worthiness and other mechanical aspects during loading conditions. This objective is expected to be achieved by engineering the engine side suspension mount bracket. This involves designing of the new bracket, meshing, converting it into a solid finite element method (FEM) model and application of different kinds of loads on the solid bracket model to analyze the behavior of the bracket. The model is designed in CATIA, cleaning, meshing and solid FEM model is generated using ANSA, crash-related analysis is done in PAMCRASH solver, and the results are viewed in ANSA meta post to inspect the breaking behavior of the engine mount bracket.

Keywords: Engine mount bracket, crash worthiness, finite element method (FEM) model, analysis

INTRODUCTION

The most essential component of a vehicle is the engine since it is the source of power generated through consumption of different kinds of fuels. For a general-purpose road transport facility, the engine must be supported by the suspension mounts to have a relative attachment of engine to the body of the transport facility. There are three fundamental functions of a suspension mounting system, to support the engine, to dampen the vibrations from the source of vibrations (i.e., engine), to isolate the engine system from the vibrations of body due to road conditions. Over the years of automobile engineering, the suspension mounts have undergone various changes in the sense of change in materials, change in geometry, change in system of suspension and they can be varied till date. As Kagde and Gandigude [1] indicate, when mount fails to perform effectively, the vibrations and stresses can adversely affect the vehicle's structural integrity and will lead to discomfort of passenger. Kamble and Bhalerao [2] report that it is crucial that the frequency of the bracket is higher than the engine's frequency which indicates the need of materials that have a direct impact on natural frequency of the bracket along with its structure. However, this leads to the main objective of these modifications over the years which is to achieve optimal durability and weight of the bracket. Though having a suspension mount with high durability increases the weight of the bracket, this may not affect the performance of the powertrain system by a noticeable scale and hence can be negligible. Having a suspension mount of light weight may meddle with the durability of the mount which may result in premature failures during its life cycle. So, it is safer and mechanically beneficial to have a stronger bracket with a slight

*Author for Correspondence
K.V.P.T. Jagannadha Rao

[1]M.Tech (Student), Department of Mechanical Engineering, KLEF University, Vijayawada, Andhra Pradesh, India
[2]Associate Professor Department of Mechanical Engineering, KLEF University, Vijayawada, Andhra Pradesh, India

Received Date: August 18, 2023
Accepted Date: September 12, 2023
Published Date: September 22, 2023

Citation: 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. 2023; 11(Special Issue 8): S122–S128.

increase in mass that improves the crash characteristics of the bracket without change in material of the bracket.

OBJECTIVE

A new design development is meant to be achieved to have better crash worthiness which results in improvement of impact load adsorption capacity of the engine mounting system which is a desired effect. According to Park et al. [3], the existing simplified models have limitations, as they can produce incomplete and large-scale results, particularly when subjected to complex crash loading conditions, and they lack failure criteria and consideration of other components assembled with them. Also, Basavaraj and Manjunatha [4] report that the brackets that support the engine endure substantial static and dynamic stresses along with a large number of vibrations. Hence, it is crucial to design and analyze these mount brackets carefully to dissipate vibrations and ensure stresses remain within predetermined safety levels. Jacob et al. [5] advise that design and crash performance play a crucial role in determining the overall safety and efficiency of an all-terrain vehicle (ATV) when studied which can be implied to the engine mounting structures. Montalvão and Moore [6] have introduced the concept of EURONCAP tests, which results in implication of testing loads on the bracket having the testing standards as reference. Deshmukh and Sontakke [7] advise that the frequent change of material for subsystems in real-world automotive manufacturing can turn out to be challenging. Thus, change in material is not the concern of the current study. Agharkakli and Wagh [8] helped in understanding of mount performance in crash events and potentially contributed to improved safety and crash worthiness of the current bracket.

METHODOLOGY AND APPROACH

The engine mount bracket is created in CATIA V6 software and the pre-processing is done in ANSA where cleaning, meshing, solid part generation, assembly with the neighboring components and application of different types of loads is done based on the study requirement. The solution for crash analysis is obtained by using PAMCRASH software. The results/behavior of the engine mount bracket obtained from PAMCRASH are observed in ANSA-Meta post. Sakurai and Suzuki [9] created a finite element method (FEM) model of the vehicle's body structure to evaluate crashworthiness. FEM is a numerical method commonly used to analyze the behavior of structures under various loads and boundary conditions; this statement led to the use of FEM model of the bracket with boundary conditions. Maski and Basavaraj [10] suggested the correlation of finite element analysis (FEA) model, since the reference has physically validated results, we can use the data to correlation for the bracket of current study.

Computer-Aided Design (CAD) Model

Figure 1 represents the engine side suspension mounting bracket, which was initially created using CATIA V6 software. The design and geometry of the bracket were defined and detailed in CATIA V6 to meet the specific requirements of the vehicle's suspension system. Following the design phase, the CATIA V6 model of the engine side suspension mounting bracket was then imported into ANSA software for pre-processing tasks. ANSA is a powerful pre-processing tool used in FEA to prepare and set up models for simulation.

Model Setup

The pre-processing in ANSA involved various essential steps to ensure the model is ready for the subsequent analysis using FEA software (e.g., PAMCRASH). Some key pre-processing tasks carried out in ANSA include the following:

Geometry Cleaning: Ensuring the imported CAD geometry is clean and free from any imperfections, overlaps, or gaps that could adversely affect the analysis.

Mesh Generation: Creating a finite element mesh of the bracket's geometry. The mesh discretizes the solid model into smaller elements, enabling the FEA software to perform calculations accurately.

Solid Part Generation: Defining and organizing the different parts of the bracket assembly as separate entities for the analysis.

Assembly: Assembling the engine side suspension mounting bracket with other relevant components or subsystems to simulate its real-world behavior and interaction within the vehicle.

Application of Loads and Constraints: Applying the necessary boundary conditions, loads, and constraints to represent the actual operating conditions and loading scenarios that the bracket will experience during use.

Material Assignment: Assigning appropriate material properties to the bracket's mesh elements to accurately represent its mechanical behavior.

Contact Definition: Defining contact regions and interactions between the bracket and other components to accurately capture their contact behavior during the analysis.

Verification and Quality Check: Conducting thorough checks and verification to ensure the accuracy and integrity of the pre-processed model before exporting it to the FEA software.

By converting the CATIA V6 model to an ANSA model and performing pre-processing tasks, the bracket's geometry and behavior are effectively prepared for the subsequent crash analysis using FEA software. The pre-processing stage is crucial in generating reliable simulation results and ensuring the analysis accurately represents the real-world performance of the engine side suspension mounting bracket. The model setup is as shown in Figure 2.

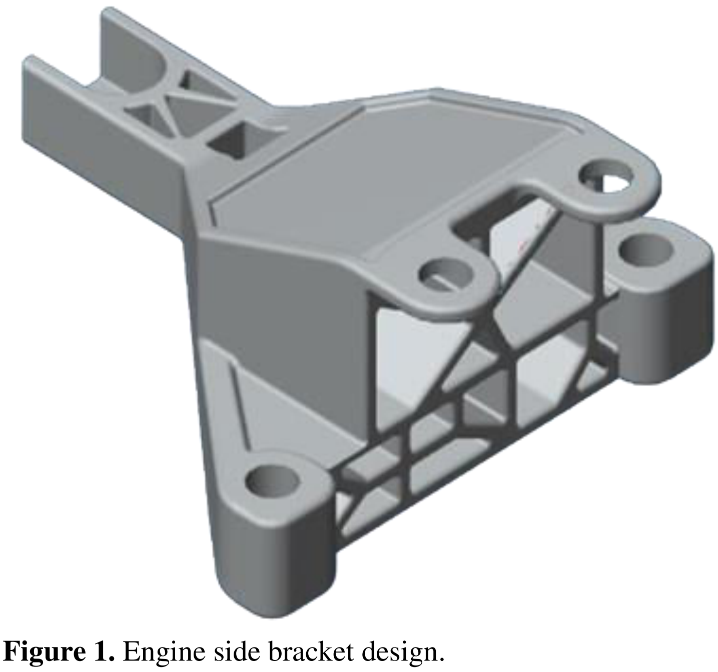

Figure 1. Engine side bracket design.

Loads and Boundary Conditions

In the loaded model setup shown in Figure 3, the engine side suspension mounting bracket, assembled with neighboring components, is subjected to various loads and constraints to simulate real-world operating conditions. The setup includes the following boundary conditions and loading conditions:

Constant Application of Velocity: The jig parts, represented in the model, are subjected to a constant application of velocity along the horizontal axis (*X*-axis). This loading simulates the effects of dynamic forces and movements on the bracket during vehicle operation.

Tightening Force on Nuts: The nuts used in the assembly are preloaded with tightening forces. These forces are applied to simulate the clamping effect of the nuts on the bracket, ensuring a secure and stable connection between components.

Reaction Force on Bolts: The bolts used to fasten the bracket and neighboring components experience reaction forces. These forces represent the reaction generated by the tight bolts and are essential to accurately simulate the interactions between connected parts.

Attachment to Spring Material: The entire assembly, including the bracket and neighboring components, is attached to a spring material. This spring material is used to represent the stiffness behavior of the vehicle's body. It allows the model to simulate the effects of the vehicle's structural rigidity on the bracket's performance and behavior.

By incorporating these boundary conditions and loading conditions, the model can simulate the dynamic behavior of the engine side suspension mounting bracket and its response to external forces during vehicle operation. This comprehensive setup enables engineers to evaluate the bracket's structural integrity, stress distribution, deformation, and overall performance under realistic conditions.

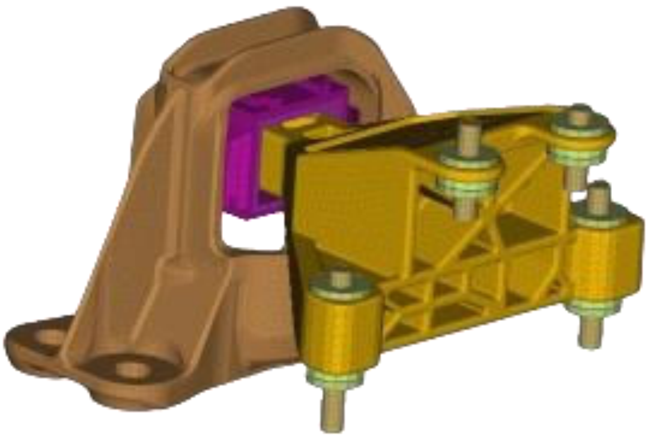

Figure 2. Engine side bracket assembly with the neighboring components.

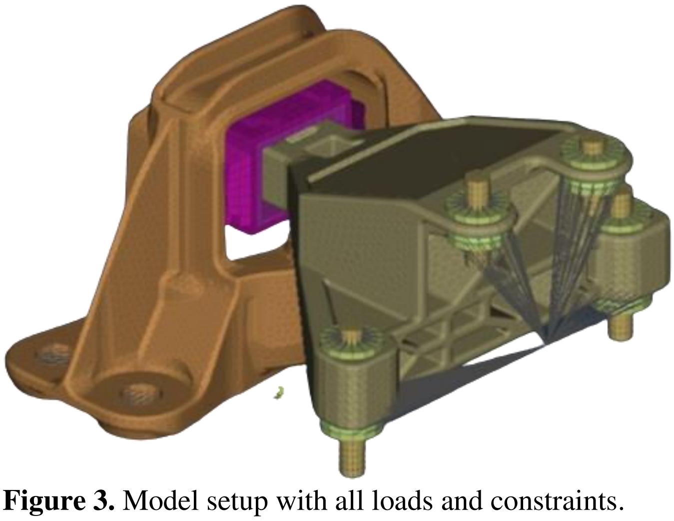

Figure 3. Model setup with all loads and constraints.

The FEA software (e.g., PAMCRASH) will analyze the model based on the specified loads and constraints, providing valuable insights into the bracket's mechanical behavior and performance. The results obtained from this analysis will assist in identifying areas of concern, potential weaknesses, and opportunities for design optimization to ensure the bracket meets safety, performance, and durability requirements in the real-world application.

RESULTS AND DISCUSSION

In this project, the crash analysis of the new engine mount bracket was successfully conducted using PAMCRASH, and the results were visualized in the ANSA METAPOST pre-processor. Figure 4 displays the distribution of equivalent stress, revealing that the stress concentration is predominantly in the desirable region. Moreover, the bracket's breaking occurs at the neck, which satisfies the required failure zone criteria. The load versus time graph in Figure 5 depicts the dynamic behavior of the bracket when subjected to a constant velocity. This graph showcases how the bracket responds to the applied load over time, providing valuable insights into its performance during various loading scenarios. Overall, the crash analysis, supported by PAMCRASH and visualized using ANSA METAPOST, has substantiated the new engine mount bracket's improved crashworthiness. The stress distribution in the desirable region and the failure mode at the bracket's neck indicate a robust design capable of withstanding high impact forces. These simulation results are instrumental in validating the design and engineering efforts, confirming the enhanced structural integrity and load-carrying capacity of the new bracket. The successful analysis and dynamic behavior assessment serves as essential contributions to the overall safety and reliability of the vehicle's engine mount system.

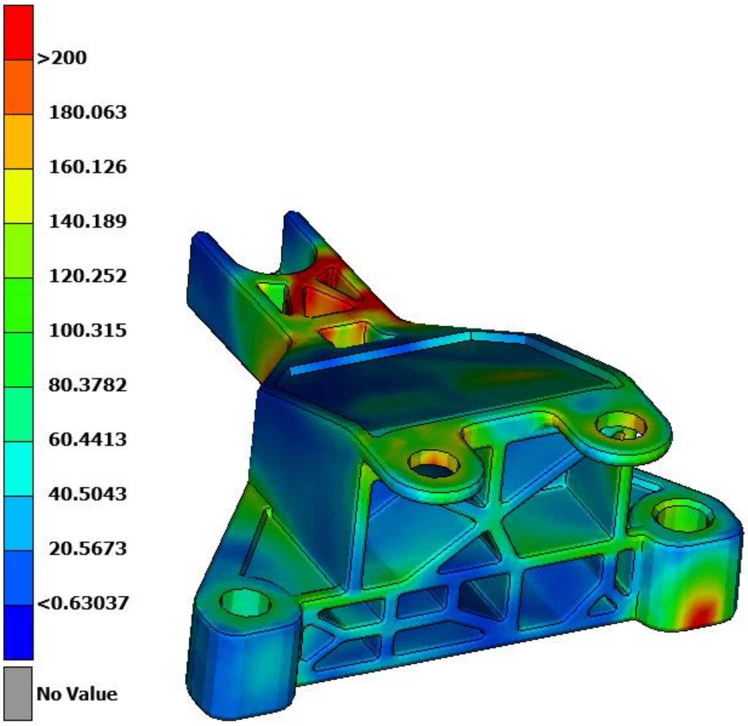

Figure 4. Equivalent stress on the bracket.

The comparison between the reference and the current models is presented in Table 1.

Table 1. Comparison between the reference and the new brackets.

S.N.	Model	Maximum Stress (MPa)	Breaking Load (N)
1	Reference bracket	311 [334*]	28288 [26011*]
2	New bracket	416	50597

* Indicates the physical validations from the referred source catalog.

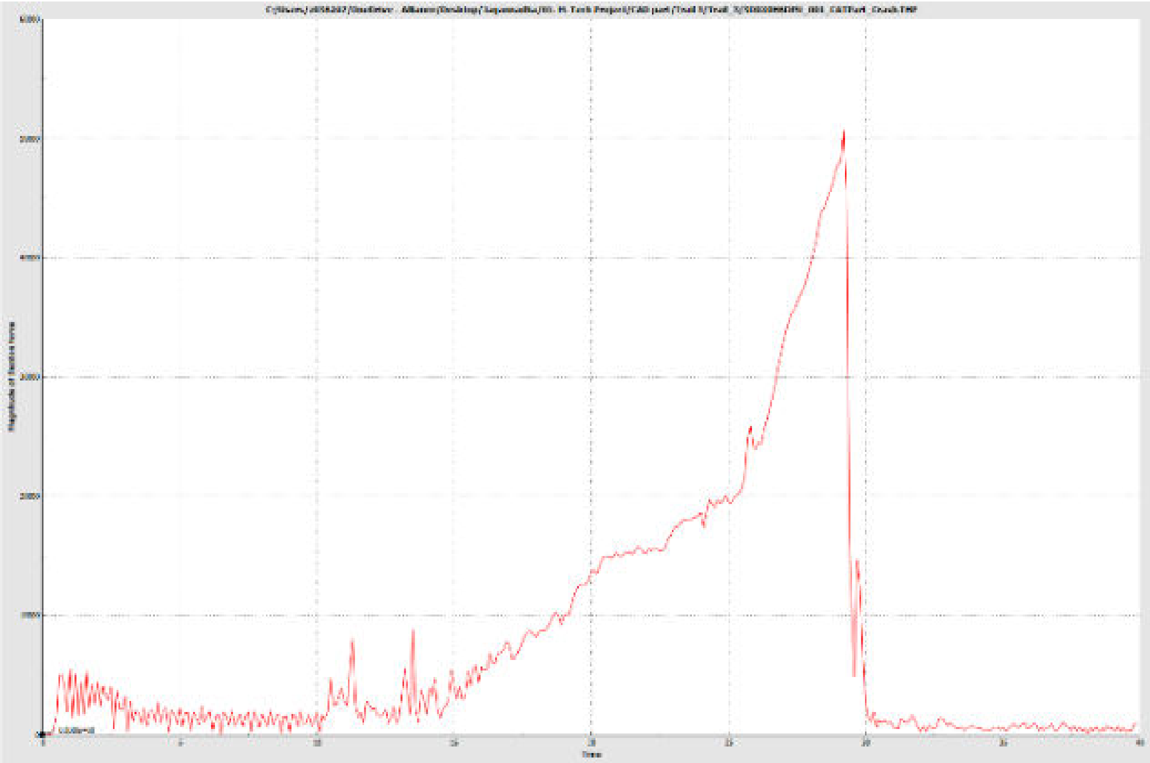

Figure 5. Load versus time graph of the bracket under application of constant velocity.

CONCLUSIONS

The successful creation of the design and FEA model for the new engine mount bracket has yielded promising results. The subsequent crash analysis, conducted on the assembly of the newly generated FEM model and neighboring components, has provided valuable insights into the bracket's performance. Comparisons between the aspects of the new and reference brackets indicate significant improvements in the new design. The maximum stress that the new bracket can absorb is 33% higher than the reference, demonstrating its enhanced strength and structural integrity. Moreover, the load at which the new bracket breaks is 79% more than the reference, indicating a substantial increase in the bracket's load-carrying capacity. The findings suggest that the new engine mount bracket exhibits improved crashworthiness, signifying its ability to withstand higher forces and impacts during collisions or accidents. Based on the correlation between computational and physical values from the reference bracket, it is anticipated that the stress range for the new bracket will be between 435 and 450 MPa, and the expected breaking load range will be within 42.5 and 50 kN. These positive results validate the efficacy of the new design and its potential to outperform the reference bracket under crash conditions. The increased absorption capacity and load-carrying capability of the new bracket enhance the overall safety and durability of the vehicle, making it a valuable advancement in the design and engineering of engine mount brackets for future automotive applications.

Acknowledgments

We would like to express our sincere gratitude to Dr. B. Nageshwar Rao and Dr. S. N. Padhi for their invaluable assistance and guidance throughout the duration of this project. Their expertise and support have greatly contributed to the successful completion of our work. Additionally, we would like to extend our appreciation to the developers and team behind CATIA, ANSA and PAMCRASH software. Their powerful and user-friendly software played a crucial role in our project, enabling us to simulate and analyze complex phenomena efficiently. Finally, we would like to thank all individuals who directly or indirectly contributed to this project, including our colleagues and research team members, for their valuable insights and discussions. We are truly grateful for the contributions and assistance provided by everyone involved, as they have significantly enhanced the quality and impact of this project.

REFERENCES

1. Kagde RR, Gandigude AU. Design analysis and optimization of engine mounting bracket. Int Res J Eng Technol. 2021; 8 (4): 5051–5056.
2. Kamble TP, Bhalerao RA. Optimization and modal analysis of engine mounting bracket for different materials by using finite element analysis. Int Res J Eng Technol. 2016; 3 (10): 120–125.
3. Park SU, Koka MR, Thomson KR, Robbins JL. A FE modeling and validation of vehicle rubber mount preloading and impact response. In: 8th International LS-DYNA Users Conference, Crash/Safety, Detroit, MI, USA, 2004. Pp. 5-23–5-32.
4. Basavaraj Y, Manjunatha TH. Design optimization of automotive engine mount system. Int J Eng Sci Invent. 2013; 2 (3): 48–53.
5. Jacob S, Thiruvarasan V, Surendhar S, Senthamizh R. Design, analysis and optimization of all terrain vehicle chassis ensuring structural rigidity. Mater Today Proc. 2021; 46: 3786–3790.
6. Montalvão D, Moore M. Computational analysis of a car chassis frame under a frontal collision. Int J Eng Res Technol. 2014; 3 (2): 39–43.
7. Deshmukh M, Sontakke KR. Analysis and optimization of engine mounting bracket. Int J Sci Eng Res. 2015; 3 (5): 131–136.
8. Agharkakli A, Wagh DP. Linear characterization of engine mount and body mount for crash analysis. Int J Eng Adv Technol. 2013; 3 (2): 37–43.
9. Sakurai T Suzuki T. Crashworthiness of electric vehicles. World Electric Vehicle J. 2010; 4 (1): 41–48.
10. Maski S, Basavaraj Y. Finite element analysis of engine mounting bracket by considering pretension effect and service load. Int J Res Eng Technol. 2015; 4 (8): 327–333.

