Tuijin Jishu/Journal of Propulsion Technology
ISSN: 1001-4055
Vol. 45 No. 3 (2024)

Design Optimization and Experimental Validation of a Multipurpose Mounting Bracket for Electric Vehicles

Barath Dhanapal[1], Christian Suhin Raj D[2], Padmanaban Ramasamy[3]
[1,3] *Department of Mechanical Engineering, Amrita School of Engineering, Coimbatore, Amrita Vishwa Vidyapeetham, India*
[2] *TICMPL Murugappa Group, Chennai, India*

Abstract: In electric vehicles, the optimal positioning of components and dedicated space for the battery pack are crucial. To address this, we developed a multipurpose bracket designed to save space by holding both the air compressor tank and the spare wheel. We conducted static analysis under various load conditions to ensure robustness. A significant challenge was choosing a mount location for the bracket, as the air compressor tank supplies air to the brakes, necessitating careful consideration of routings. This new bracket replaces two existing brackets into one made of steel, that is feasible for manufacturing. This work involved modelling the bracket and simulating various load cases, including static, shock, braking, and cornering, using Altair Hyper Mesh to evaluate strength and durability. A baseline model is initially designed, followed by thickness reduction and topology optimization to minimize weight and prevent overdesign. After optimization, a new bracket was modelled, and analysis was reiterated. Additionally, modal analysis was performed to obtain natural frequencies and mode shapes. Physical validations using strain gauges for various road events were conducted to correlate the results, showing good agreement. As a result, a lightweight, robust mounting bracket that meets strength and durability requirements was successfully developed, allowing the saved space to be utilized for the auxiliary battery pack.

Keywords: *Multiple Purpose Bracket, Strength and Durability Analysis, Topology Optimization, Physical Testing*

1. Introduction

In heavy commercial vehicles, brackets play a crucial role in providing structural support and attaching various components to the vehicle's chassis or frame. Given their high usage, replacing multiple brackets with a single multipurpose bracket can significantly reduce bracket consumption and vehicle weight. Electric vehicles, in particular, require dedicated space for battery packs, which occupy substantial volume within the vehicle. Therefore, optimizing component positioning is essential to maximize space utilization and overall vehicle efficiency. Deciding on the battery arrangement is a critical aspect of vehicle design. To address these challenges, we propose the development of a multipurpose mounting bracket that serves a dual function of accommodating both an air compressor tank and a spare wheel. This innovative design aims to conserve valuable space within the vehicle without compromising strength and durability, allowing for better space allocation for battery packs. The bracket design must be feasible for manufacturing and capable of withstanding different load conditions to ensure strength and durability. Choosing the appropriate mount location is challenging, as it should not interfere with the air compressor pathway, which supplies air to the brakes. Additionally, reducing the bracket's weight can lead to cost and weight savings. Many researchers have explored innovative design processes and the functional requirements of bracket systems to meet performance expectations. S.Kandreegula et al., discussed the simulation and validation processes for propeller shaft mounting brackets. The shaft undergoes a complex combination of forces and is subjected to modal, static, and frequency response analyses to thoroughly evaluate its behaviour. This assessment is then validated through physical testing using specialized test rig setups [1]. K. Rohilla et al., discuss the design, simulation, and validation of a mounting bracket for the Front End Accessory Drive, along with an optimization process aimed at significant weight reduction. The resulting design, along with simulation outcomes, is thoroughly validated through physical tests using dedicated test rig setups and live vehicle testing [2]. Aditya Vadrevu et al., explored various optimization techniques to develop a robust automobile console frame

---

design. Their study focused on minimizing design space and achieving optimal material distribution along with appropriate wall thickness [3]. Li Zhengmao et al., discuss the design and weight reduction of vehicle cabin mounting brackets. They employed FEM analysis to enhance weight reduction while meeting strength, stiffness, and vibration performance requirements. The study introduces both topology optimization and size optimization. The findings reveal that topology optimization is more effective for reducing bracket weight, whereas size optimization provides a quicker solving time [4]. Basem Alzahabi et al., have optimize the transmission mount bracket for improved stiffness, strength, and mass efficiency while shortening the development cycle. Comparative analyses between simulation data and experimental data [5]. Mekalke G., have discussed the Engine mounting brackets. Their design involves performing static and modal analysis, along with weight reduction, to ensure they are lightweight and robust. Experimental validation confirms that materials such as Mild Steel, Aluminum 6063, Carbon Fiber Composite, and Glass Fiber Composite are effective in withstanding inertial loads and maximizing vibration isolation [6]. Gavade, S et al., have explores the transition from aluminum to Metal Matrix Composite (MMC) for engine accessory brackets, involving structural requirement analysis, MMC process adaptation, and CAE-based topology optimization [7]. Vibhay Kumar et al., discussed on Oil Strainer Mounting Bracket it presents dynamic analysis, including constrained modal analysis and frequency response analysis for different engine exciting frequencies and service loads. The frequency response analysis, conducted using the modal superposition method with base excitation loads, showed good correlation between natural frequencies and stress responses with test results [8]. Kursat Celik H et al., have studied damaged engine mounting bracket at initial FEA identified stress concentrations and deformation behaviours leading to failure due to insufficient geometry thickness and notch effects under torsional loading from the electric engine. The redesign focused on improving these geometric features, and subsequent FEA confirmed the redesigned bracket's ability to operate safely [9]. Ankit Vyas et al., studied on Topology Optimization, specifically Evolutionary Structural Optimization, applied to optimize the driver cabin mounting bracket of a heavy commercial vehicle. The research aims to achieve mass reduction and compliance minimization [10]. Vijay Kalantre et al., have reviewed various approaches such as shape, size, and topology optimization highlighting their application in designing automotive components without compromising strength. Using ANSYS and Evolutionary Structural Optimization (ESO), the study optimizes a Leaf Spring Mounting Bracket, achieving a significant weight reduction while maintaining structural integrity and cost efficiency [11]. Mandar Omprakash Shete., studied on replacing traditional sheet metal brackets with thermoplastic materials. The transition aims to reduce weight and production costs while comparing performance between sheet metal and polypropylene brackets, highlighting the potential benefits of 3D printing technology in automotive applications [12]. Shivaram Srikanth et al., focused on optimizing the lift arm of a self-loading concrete mixer, aiming to reduce weight without compromising functionality. By modelling and simulating the existing lift arm assembly, the analysis identifies stress points under impact loads, guiding design modifications for weight reduction. The refined designs are validated through further testing under operational conditions [13]. Swathikrishnan S et al., have enhanced the design of a swingarm for a prototype performance electric motorcycle, to improve stiffness and strength while reducing weight. Analysis led to the development of two optimized designs using AISI 4340 steel and Aluminium alloy 6061 T6 respectively. Demonstrated significant weight reduction with improved stiffness and safety factors compared to the original design [14]. Padmanaban Ramasamy et al., studied FSW of dissimilar aluminum alloys AA2024 and AA7075, focusing on optimizing process parameters for maximizing joint tensile strength. Through varying tool rotation speed and welding speed, the study develops a mathematical model using response surface methodology to predict joint strength. Optimization using the particle swarm optimization algorithm identifies optimal FSW parameters and achieving robust welds in dissimilar aluminum alloys through systematic parameter optimization [15]. Harihara Kumaran S et al., explores the application of topology optimization to design an aircraft hatch lock, aiming to reduce weight while maintaining structural integrity and operational efficiency. By identifying and optimizing material areas with minimal stress, the study achieves weight reduction without compromising safety factors [16]. Navaneeth Krishnaan K et al., study investigates the effectiveness of basalt fiber as a retrofitting material in concrete structures, emphasizing finite element analysis over traditional experimental methods. the study models the interaction between concrete and basalt fiber, validating results against experimental data. Analytical modeling demonstrates a cost-effective approach for assessing stress-strain behavior in composite materials, enhancing structural performance and durability [17]. Pooja More et al., have investigates the dynamic vibration characteristics of radiator brackets in generator systems, crucial for mitigating noise and structural damage caused by vibrations. Using finite element modeling, sensitivity analysis, and design of experiments, the study derives transfer functions for the first two bending frequencies of the brackets. Validation against analytical and FEM results confirms the accuracy of the transfer functions, demonstrating their utility in cost-effective and efficient design optimizations for robust generator structures [18]. In this paper, the integrated mounting bracket is modelled as shown in the figure 1 and subjected to strength and durability analysis. Furthermore, the design undergoes an optimization process to achieve maximum possible weight reduction advantage. The software tools employed for this study include Altair HyperWorks and Optistruct. The resulting design and simulation results are validated through physical correlation.

Figure 1: Bracket and Assembly

2. Numerical Analysis

The structural performance of the mounting bracket can be accurately predicted using CAE techniques. By analyzing CAE results, engineers can identify and resolve potential issues by adjusting CAD geometry parameters such as thickness, material selection, and reinforcement. This method significantly streamlines the design process for structural support brackets.

Table 1: Component weight

Components	Mass (Kg)
Spare Wheel	110
Air Tank (1+1)	15+15

CAD data is imported and discretized, utilizing a 2D mesh configuration for the components, as shown in Figure 2. Each component is assigned specific material properties, with the integrated bracket specified as Mild Steel E250. Ensuring that component weights match their physical counter parts detailed in Table 1 is critical.

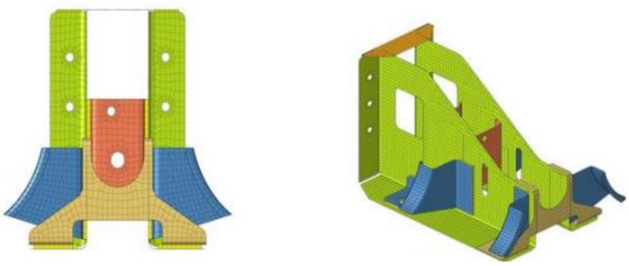

Figure 2: Component level meshing

The model is then constrained across all degrees of freedom at mounting frame members. Testing procedures involve applying specific loads and constraints to simulate real-world conditions. For instance, braking tests assess the bracket's resilience during sudden stops, while cornering tests evaluate its structural integrity during vehicle turns. The goal is to ensure the bracket can withstand these events without compromising strength or durability. Testing scenarios include static, shock, braking, and cornering load cases. The loading location are shown in figure

3. Static load refers to constant weight applied to components when the vehicle is stationary, typically from mounted components. Shock load involves sudden and intense forces, mimicking collisions or impacts. Braking load encompasses longitudinal and vertical forces during deceleration, while cornering load includes

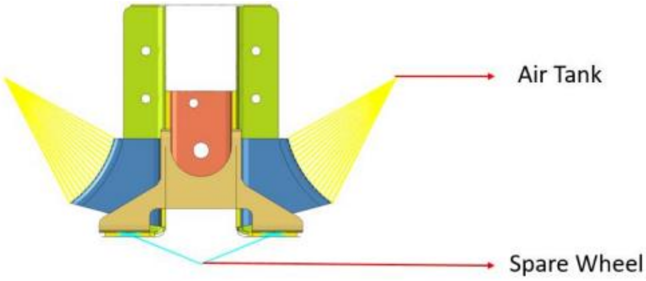

Figure 3: Loading location

Lateral and vertical forces from turns and dynamic load shifts. Figure 4 illustrates the system-level validation process flow used for the bracket. The bracket features a thickness of 6 mm and weighs 23.47 kg. It has been subjected to various loading conditions including static, shock, braking, and cornering as depicted in Figure 5.

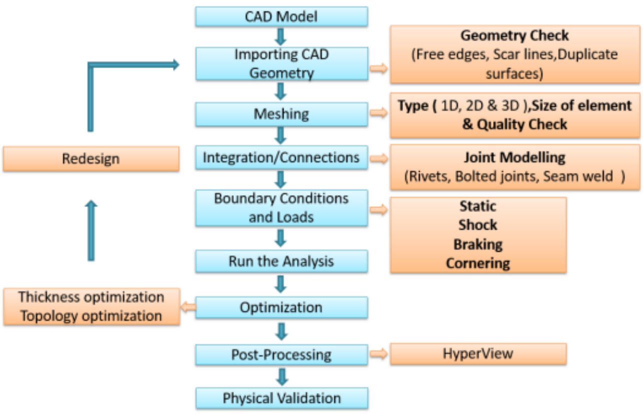

Figure 4: System level validation process flow for Bracket.

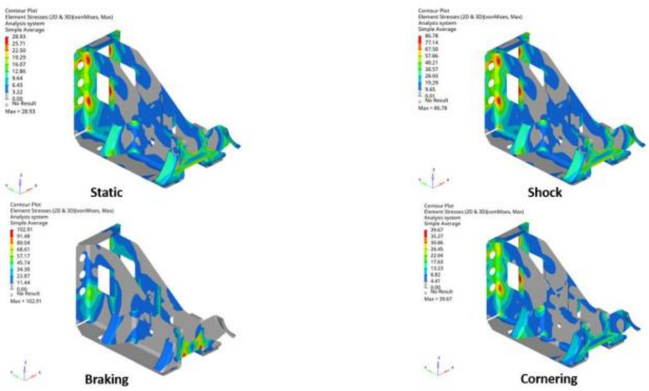

Figure 5: Bracket under static, shock, braking, and cornering load

The stress values obtained are 28.93 MPa, 86.78 MPa, 102.91 MPa, and 39.67 MPa, respectively. These values are well below the yield strength of 250 MPa, with a factor of safety above 2, indicating overdesign. Further optimization is necessary to reduce weight and cost of the model.

3. Optimization

The bracket was modelled with varying thicknesses, specifically 5mm, 4mm and 3mm as shown on Table 2. These designs underwent durability analyses, including Static, shock, braking, and cornering conditions. Subsequently, comparisons were drawn between these designs and a baseline design. The design that exhibited a compromised factor of safety was identified as the optimized design, ensuring both durability and structural integrity under diverse operating conditions. Among these conditions, the braking condition experiences higher stress level. In the 4mm bracket, stress reaches 185 MPa under braking conditions, near to the yield point of 250 MPa. Conversely, the 5mm bracket exhibits a stress level of 126 MPa under braking conditions, which is below the yield point. With a safety factor of 2, the 5mm bracket emerges as the optimized solution and the weight of the bracket is 21.31 Kg.

Table 2: Bracket with varying thickness

Design	Size (mm)	1g conditions	3g conditions	Cornering conditions	Braking conditions
Baseline	6 mm	28.93 MPa	86.78 MPa	39.67 MPa	102.91 MPa
Design - I	5 mm	43.98 MPa	131.93 MPa	54.28 MPa	126.34 MPa
Design - II	4 mm	73.68 MPa	221.05 MPa	90.92 MPa	185.23 MPa
Design - III	3 mm	136.02 MPa	408.05 MPa	168.13 MPa	309.76 MPa

Topology optimization using Altair HyperWorks was performed on a 5mm thick bracket, employing a systematic approach to enhance structural efficiency. The model was segmented into distinct design (blue) and non-design (green) regions, as depicted in Figure 6, enabling focused analysis through collector options. After meshing, the bracket underwent testing under various loading conditions including static, shock, braking, and cornering to simulate real world scenarios. During the optimization process, special attention was given to scenarios with high stress concentrations, particularly observed under shock and braking conditions as shown in Figure 7. Detailed analyses, including volume fraction and weighted compliance response analyses, were conducted to further refine the bracket's design.

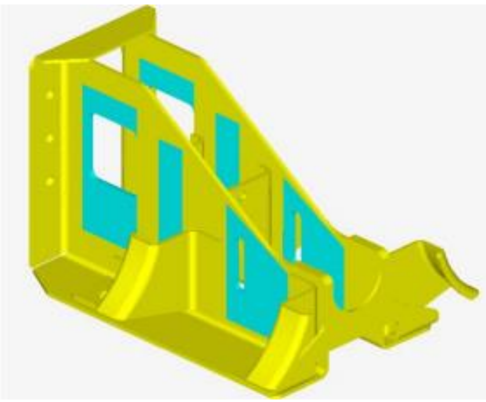

Figure 6: Design (blue) and non-design (green) regions

The volume fraction analysis aimed to optimize material distribution by selectively removing material from low-stress regions, thereby reducing unnecessary weight while maintaining structural integrity. Meanwhile, the weighted compliance response analysis aimed to minimize deflection under loading conditions, ensuring the bracket's ability to withstand dynamic forces encountered during vehicle operation.

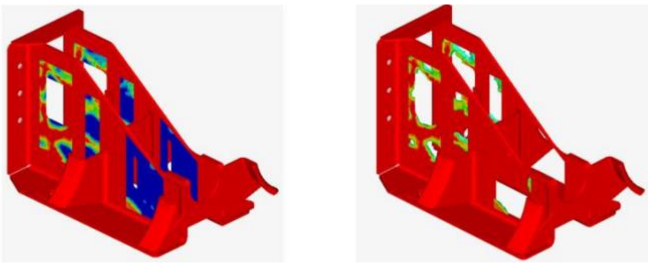

Figure 7: Topology optimization

This comprehensive optimization approach ensured that the final bracket design not only met stringent performance criteria but also achieved significant weight reduction without compromising safety or functionality.

4. Optimized Design

The bracket was redesigned based on the results of topology optimization, as depicted in Figure 8. The model underwent iterative processes to ensure structural integrity. Evaluation of the bracket included testing with both steel and alloy steel materials to verify a factor of safety above 2. Details of the material properties are listed in Table 3.

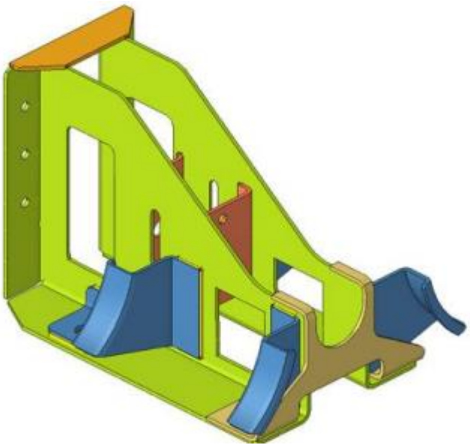

Figure 8: Bracket after optimization

Table 3: Property of Steel and Alloy Steel

Steel		Alloy Steel	
Properties	Value	Properties	Value
Young's modulus	207 GPa	Young's modulus	213 GPa
Density	7830 kg/m3	Density	8030 kg/m3
Poisson's ratio	0.3	Poisson's ratio	0.3
Yield strength	250 MPa	Yield strength	470 MPa
Ultimate strength	448 MPa	Ultimate strength	745 MPa

The optimal bracket was analyzed under static, shock, braking, and cornering load conditions as shown in Figure 9, with comparisons made between steel and alloy steel. The stress values and factors of safety were identified and are presented in Table 4.

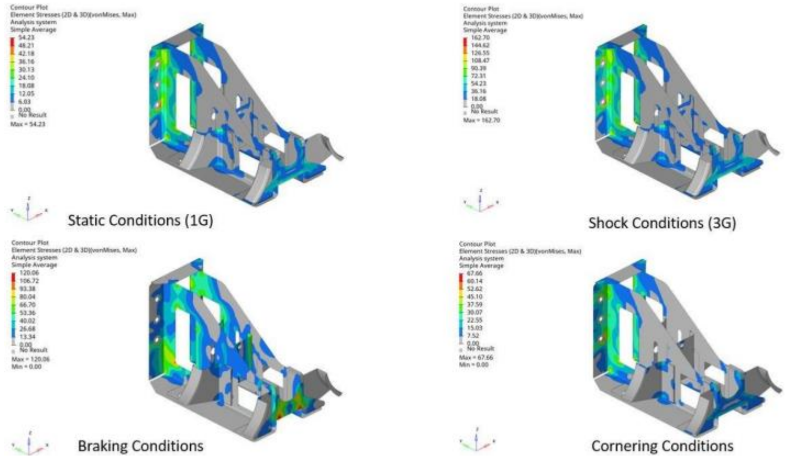

Figure 9: Optimal bracket under static, shock, braking, and cornering load conditions

Initial stress values obtained for steel were found to be high and did not meet the required factor of safety and yield point criteria. Subsequently, the analysis was repeated using alloy steel, which resulted in stress values of 54.23 MPa, 162.70 MPa, 120.06 MPa, and 67.66 MPa for static, shock, braking, and cornering conditions, respectively.

Table 4: Comparison of Steel and Alloy Steel

Conditions	Steel	FOS	Alloy Steel	FOS
Static load (1G)	54.25 MPa	3.6	54.23 MPa	8.6
Shock load (3G)	162.76 MPa	1.2	162.70 MPa	2.8
Braking	120.06 MPa	1.6	120.06 MPa	3.9
Cornering	67.66 MPa	2.9	67.66 MPa	6.9

Comparatively, alloy steel has a yield point of 470 MPa, providing a factor of safety above 2. With stress values comfortably below the yield point and ensuring sufficient safety margins, the design was deemed safe and reliable.

5. Virtual Validation and Test Correlation

The physical validation process for the bracket aimed to establish correlation between CAE results and testing outcomes. Strain gauges were strategically placed at hotspot locations to collect strain data, which was compared and validated against CAE predictions. This validation method ensures accuracy in predicting structural performance under various loading scenarios. Two specific locations were identified as critical hotspots, as illustrated in Figure 10, where strain gauges were affixed. Strain measurements at these points, gathered during rigorous vehicle road tests are detailed in Table 5. By precisely positioning strain gauges at these locations, engineers aimed to comprehensively assess the bracket's structural behavior under real-world conditions. Utilizing the strain data from testing, stress values were calculated using Young's modulus formula to establish a correlation between strain and stress. These stress values were then compared with corresponding data from CAE simulations, which included static, shock, braking, and cornering conditions representing worst-case scenarios for the vehicle.

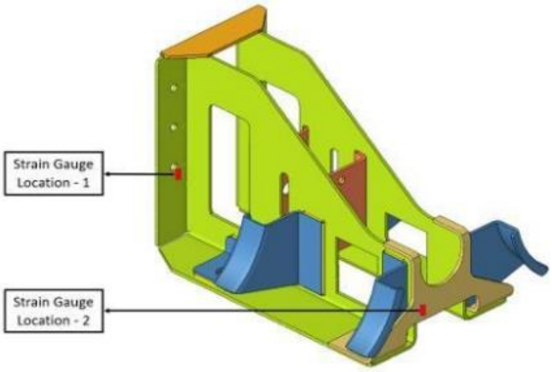

Figure 10: Location of strain gauge

Table 5: Strain value from Testing

Conditions	Strain value at location-1	Strain value at location-2	Equivalent max stress
Idle/Static State	0.00005518	0.00005394	11.753 MPa
Shock load	0.00037366	0.00025135	79.589 MPa
Braking	0.00029754	0.00020913	63.376 MPa
Cornering	0.00015516	0.00012747	33.049 MPa

The stresses induced were calculated using hook's law and are 11.753 MPa, 79.589 MPa, 63.376 MPa and 33.049 MPa respectively for static condition, shock condition, braking condition, and cornering condition. The calculation revealed that stress values predicted by CAE tended to be higher than those observed in road testing. Hence the bracket is safe and durable under loading conditions

6. Conclusion

This work successfully tackled the challenge of designing a lightweight and durable mounting bracket for a vehicle, specifically for supporting spare wheel and air compressor tanks. Comprehensive strength and durability analyses were pivotal throughout the design phases, covering various real-world loading scenarios including static loads, shock from rough roads, braking forces, and cornering maneuvers. Initially opting for steel with a 6 mm thickness, the baseline design met safety requirements with a factor of safety exceeding 2 but weighed 23.47 kg, prompting the need for weight reduction. Thickness optimization resulted in 21.31 kg with a 5mm thickness, and further topology optimization brought it down to 20.04 kg while maintaining structural integrity. Adopting alloy steel with higher yield strength addressed heightened stress demands identified during optimization. Validation included stress analysis confirming a safety factor above 2 under all conditions for the alloy steel design. Physical testing using strain gauges under varied road conditions corroborated CAE simulations, demonstrating alignment between predicted and actual stress values. Effectively reducing the initial weight of the mounting bracket from 23.47 kg to an optimal weight of 20.04 kg achieved a weight savings of 3.43 kg, approximately 14.6%. The space saved by reducing weight was utilized for an auxiliary battery pack, enhancing vehicle functionality. Future advancements could explore lighter materials like high-strength aluminum alloys or composites, aiming for further weight reduction without compromising performance. The methodologies and insights gained here are applicable for optimizing other vehicle components, contributing to overall weight reduction and improved fuel efficiency.

Acknowledgement

I express my sincere gratitude to TICMPL-Murugappa Group Chennai for granting me the valuable Internship Opportunity in CAE. I extend my heartfelt thanks to Mr. Christian Suhin Raj D for his unwavering moral support. Additionally, I would like to thank Mr. Balamurugan M, CAE Head, Mr. Balaji K, Durability Manager and Dr. Padmanaban Ramasamy, Associate Professor, Amrita School of Engineering, Coimbatore, for their valuable guidance and support.

References

[1] S. Kandreegula, K. Rohilla, N. Sukumar, and K. Kamal, "Simulation and validation of propeller shaft mounting brackets for heavy duty commercial vehicles," SAE Technical Paper 2017-28-1947

[2] K. Rohilla, S. K. Kandreegula, S. Agrawal, J. Bisht, and others, "Design, simulation and validation of front end auxiliary drive (FEAD) mounting bracket for electric powertrain application," SAE Technical Paper 2019-26-0279.

[3] Vadrevu et al., "Wall thickness optimization for centre console frame," presented at the Altair Technology Conference, 2013.

[4] Z. Li, D. Wang, and X. Wang, "Weight-reduction design on mounting bracket of commercial vehicle cabin," *SAE Technical Paper* 2019-17-0463

[5] B. Alzahabi, S. Simon, and L. Natarajan, "Optimization of Transmission Mount Bracket," SAE Technical Paper 2003-01-1460.

[6] G. Mekalke, "Design, analysis and optimization of engine mounting bracket for SAE Supra car using finite element analysis," *SAE International Journal of Advances and Current Practices in Mobility*, vol. 3, no. 2, pp. 802-805, 2021.

[7] S. V. Gavade, G. Soni, S. Momin, and A. Bhargava, "Design & development of metal matrix based mounting bracket for commercial vehicle application," *SAE Technical Paper* 2020-28-0463.

[8] V. Kumar and G. Uppala, "Correlation study of oil strainer mounting bracket for dynamic loading," *SAE Technical Paper* 2013-01-2782.

[9] H. K. Celik, H. Ersoy, A. Dogan, G. Eravci, A. Rennie, and I. Akinci, "Strength-based design analysis of a damaged engine mounting bracket designed for a commercial electric vehicle," *Journal of Failure Analysis and Prevention*, vol. 21, no. 4, pp. 1315-1322, 2021.

[10] A. Vyas, S. Varkhede, G. Patil, D. More, and D. Malgar, "Design & analysis of cabin mounting bracket," International Research Journal of Engineering and Technology (IRJET), e-ISSN: 2395-0056, 2021.

[11] V. Kalantre, K. H. Munde, and A. Pawar, "Topology optimization of front leaf spring mounting bracket," International Journal of Scientific Development and Research (IJSDR), vol. 3, no. 7, 2018.

[12] M. O. Shete, "Optimization of sheet metal brackets using 3D printing technology," *International Journal of Engineering Research & Technology (IJERT)*, vol. 07, no. 12, ISSN: 2395-0072, 2020.

[13] S. Srikanth, Z. Hussain, and T. Mohanavelu, "Design and optimization of lifting arm of self-loading concrete mixer through finite element analysis," *International Journal of Recent Technology and Engineering (IJRTE)*, vol. 8, no. 1, 2019.

[14] S. Swathikrishnan, P. Singanapalli, and A. S. Prakash, "Design and analysis of swingarm for performance electric motorcycle," *International Journal of Innovative Technology and Exploring Engineering (IJITEE)*, vol. 8, no. 8, ISSN: 2278-3075, 2019.

[15] R. Padmanaban, R. Vaira Vignesh, A. P. Povendhan, and A. P. Balakumharen, "Optimizing the tensile strength of friction stir welded dissimilar aluminium alloy joints using particle swarm optimization," *Materials Today: Proceedings*, vol. 5, pp. 24820-24826, 2018.

[16] S. Harihara Kumaran, V. Jayakumar, M. Ajmal Sheriff, A. Pandiyan, M. Shajin Majeed, and G. Ragul, "Topology optimization of aircraft hatch lock," *International Journal of Mechanical Engineering and Technology*, vol. 8, no. 9, pp. 86-94, 2017.

[17] K. N. Krishnaan, R. Hari, and K. M. Dr. Mini, "Finite element analysis on the performance of strengthening concrete structures confined using natural and artificial fibers," *Journal of Physics: Conference Series,* vol. 1451, p. 012001, 2020.

[18] Pooja More, B. Santhosh, and Amol Jain, "Analysis of Radiator Mounting Bracket Design and Derivation of Transfer Function to Predict Modal Frequencies Based on Parametric Study," *Materials Today: Proceedings*, vol. 5, pp. 23945-23954, 2018.