Journal of Artificial Intelligence and Emerging Technologies
Volume 2, Issue 2, pp 12-15, February 2025
ISSN (online): 3049-1967
DOI: https://doi.org/10.47001/JAIET/2025.202003
Available Online at https://www.jaiet.com

Finite Element Based Structural Analysis of a Two-Seater Electric Vehicle Frame

[1]Muhammad Fajar Nugroho, [2]Andi Saputra Wijaya

[1,2]Mechanical Engineering, Institut Teknologi Sepuluh Nopember, Surabaya 60111, Indonesia

Abstract: The increasing demand for sustainable transportation has significantly accelerated the development of compact electric vehicles (EVs) for urban mobility applications. A crucial component of any electric vehicle is its chassis or frame, which serves as the primary load-bearing structure supporting passengers, battery packs, drivetrain components, and suspension systems. This study presents the design and finite element-based structural analysis of a two-seater electric vehicle frame aimed at achieving optimal strength-to-weight ratio while ensuring safety and structural integrity. The frame is modeled using advanced computer-aided design (CAD) software and evaluated through finite element analysis (FEA) under various static loading conditions including passenger load, battery load, and impact considerations. Parameters such as total deformation, equivalent (Von Mises) stress, strain distribution, and factor of safety are examined to validate the structural performance. The numerical results confirm that the proposed frame design satisfies permissible stress limits while maintaining lightweight characteristics, thereby enhancing vehicle efficiency and driving range. This research contributes to the development of structurally efficient and economically feasible EV chassis systems suitable for low-speed urban transportation.

Keywords: Electric Vehicle (EV); Chassis Design; Finite Element Analysis (FEA); Structural Analysis; Lightweight Structure; Von Mises Stress; Factor of Safety; Vehicle Frame.

I. INTRODUCTION

Electric vehicles have emerged as a promising alternative to conventional internal combustion engine vehicles due to growing environmental concerns, rising fuel costs, and stringent emission regulations. The structural design of an electric vehicle differs significantly from traditional vehicles because of the integration of heavy battery packs and electric drivetrain systems. The vehicle frame must be designed to withstand static and dynamic loads while minimizing overall weight to improve battery efficiency and range. In compact two-seater electric vehicles intended for urban commuting, space constraints and lightweight requirements present additional design challenges. A well-designed frame must ensure passenger safety, structural rigidity, and durability under operating conditions such as acceleration, braking, cornering, and uneven road surfaces. Finite Element Analysis (FEA) has become a powerful computational tool for evaluating structural performance during the design phase, reducing the need for costly physical prototypes. This study focuses on the structural evaluation of a two-seater EV frame using numerical simulation techniques to ensure mechanical reliability and safety compliance.

In recent years, the shift toward electrified transportation has not only been driven by environmental sustainability but also by the need for energy efficiency and urban mobility optimization. Compact two-seater electric vehicles are particularly suitable for short-distance commuting, campus mobility, and intra-city transportation due to their reduced footprint and lower energy consumption. However, the structural design of such vehicles must address challenges associated with battery integration, crashworthiness, torsional rigidity, and load transfer mechanisms. Unlike conventional internal combustion engine vehicles where the engine mass contributes to structural stiffness, electric vehicles distribute heavy battery packs along the chassis floor, altering stress paths and bending characteristics. Therefore, the frame must be engineered to maintain stiffness while avoiding excessive material usage. The integration of numerical simulation tools such as Finite Element Analysis (FEA) allows engineers to predict structural behavior under multiple loading conditions, thereby reducing development time and fabrication cost while ensuring compliance with automotive safety standards.

II. OBJECTIVES

The primary objective of this research is to design and structurally analyze a two-seater electric vehicle frame capable of supporting operational loads while maintaining lightweight characteristics. Specific objectives include minimizing

Journal of Artificial Intelligence and Emerging Technologies
Volume 2, Issue 2, pp 12-15, February 2025
ISSN (online): 3049-1967
DOI: https://doi.org/10.47001/JAIET/2025.202003
Available Online at https://www.jaiet.com

deformation, ensuring stress levels remain within allowable limits of the selected material, and achieving an adequate factor of safety.

The methodology involves developing a 3D CAD model of the vehicle frame using modeling software. The material selected for the frame is structural steel due to its favorable strength-to-cost ratio and weldability. The CAD model is imported into FEA software for meshing and boundary condition application. Static structural analysis is performed by applying distributed loads representing passenger weight, battery mass, motor weight, and road reaction forces. Results such as deformation contours, stress distribution, and safety factors are evaluated and compared with material yield strength to determine structural adequacy.

III. PROBLEM STATEMENT

The design of a two-seater electric vehicle frame presents challenges related to weight reduction, structural rigidity, and load distribution. Excessive weight reduces battery efficiency and driving range, whereas insufficient structural strength may compromise passenger safety. Improper load distribution can lead to localized stress concentrations, causing structural failure. Furthermore, electric vehicles require accommodation of battery packs that significantly alter the center of gravity and load path compared to conventional vehicles. Therefore, there is a need to design a structurally optimized frame that can withstand static loads and moderate impact conditions while maintaining lightweight properties. The problem addressed in this study is to evaluate whether the proposed frame design can meet safety and strength requirements using finite element-based numerical analysis before prototype fabrication.

IV. BACKGROUND STUDY

Previous research on automotive chassis systems highlights the importance of structural optimization for improving vehicle performance and safety. Traditional ladder frames and monocoque structures have been widely studied in automotive engineering. With advancements in computational tools, FEA has become standard practice for evaluating chassis strength and stiffness. Studies indicate that lightweight tubular space frames offer superior torsional rigidity and weight reduction compared to conventional ladder frames. Additionally, research on electric vehicle chassis design emphasizes battery placement and its influence on stress distribution. Researchers have also investigated the use of alternative materials such as aluminum alloys and composite materials; however, structural steel remains a practical choice for low-cost prototypes and small-scale production. Numerical analysis methods such as static structural analysis and modal analysis are commonly used to predict frame behavior under operational loads. These studies provide the theoretical foundation for applying finite element techniques in the present work.

V. PROPOSED METHODOLOGY

The proposed methodology begins with conceptual frame design based on dimensional requirements for two passengers and battery accommodation. A tubular space frame configuration is selected to balance strength and weight. The 3D model is developed using CAD software with accurate geometric representation of cross members, longitudinal beams, and support brackets.

After modeling, the geometry is imported into finite element software for preprocessing. Meshing is performed using tetrahedral elements to ensure accurate stress representation. Boundary conditions are defined by fixing suspension mounting points and applying vertical loads corresponding to passenger weight (assumed 75 kg per passenger), battery weight, and motor assembly weight. The material properties of structural steel, including Young’s modulus, Poisson’s ratio, and yield strength, are assigned.

Static structural analysis is then conducted to determine deformation and stress distribution. The results are validated by comparing maximum stress values with allowable stress limits. If necessary, design modifications such as reinforcement of cross members are incorporated to improve performance.

VI. RESULTS AND EVALUATION

The finite element simulation results were analyzed using contour plots and numerical data extracted from critical regions of the frame. The maximum equivalent (Von Mises) stress was observed near the front suspension mounting points, where load transfer from road reaction forces is significant. However, the stress values remained below the yield strength of structural steel, confirming elastic behavior under applied static loads. The total deformation results indicated that deflection was primarily concentrated at mid-span longitudinal members, which is typical in frame structures subjected to distributed loading. The computed factor of safety exceeded the recommended minimum value for automotive structural components, demonstrating adequate reliability. Furthermore, stress gradients were found to be smooth without abrupt discontinuities, indicating proper geometric transitions and absence of sharp stress risers. These

Journal of Artificial Intelligence and Emerging Technologies
Volume 2, Issue 2, pp 12-15, February 2025
ISSN (online): 3049-1967
DOI: https://doi.org/10.47001/JAIET/2025.202003
Available Online at https://www.jaiet.com

results validate the structural integrity of the proposed frame and confirm the effectiveness of the numerical approach in predicting mechanical performance prior to physical prototyping.

The finite element analysis results indicate that the maximum deformation occurs at the mid-span region of the frame under full load conditions. The observed total deformation remains within acceptable limits for safe operation. The equivalent (Von Mises) stress distribution shows higher stress concentration near suspension mounting brackets and motor support regions. However, the maximum stress value is below the yield strength of structural steel, ensuring elastic behavior under applied loads.

The calculated factor of safety is greater than the minimum recommended value of 2 for automotive structural components, indicating adequate design reliability. The stress contour plots confirm uniform load distribution with no critical failure points. These findings demonstrate that the proposed frame design is structurally stable and suitable for fabrication.

A. Testing Equipment for Efficiency

To validate numerical results experimentally, physical testing equipment can be used after fabrication. Universal Testing Machines (UTM) can evaluate material tensile properties and weld strength. Strain gauges may be mounted on critical frame locations to measure real-time strain under load conditions. Load cells can be employed to simulate passenger and battery weight distribution. Additionally, vibration testing equipment can assess structural stability under dynamic conditions. These experimental validations enhance the reliability of finite element predictions.

B. CAD Model

The CAD model of the two-seater EV frame consists of tubular structural members arranged in a space frame configuration. The design includes longitudinal side members, cross members, battery mounting platform, motor support bracket, and suspension mounting points. The geometry ensures proper weight distribution and ergonomic seating arrangement. The tubular sections are selected based on standard pipe dimensions to simplify fabrication and welding. The CAD model serves as the foundation for finite element simulation and manufacturing drawings.

C. Shaft Design

The design phase of the two-seater electric vehicle frame involved iterative optimization of geometry, cross-sectional dimensions, and load-bearing members. Initially, conceptual sketches were developed considering ergonomic requirements, wheelbase dimensions, and battery placement. A tubular space frame configuration was selected due to its high strength-to-weight ratio and ease of fabrication. Circular hollow sections were chosen as primary structural members because they provide uniform stress distribution and superior torsional resistance compared to rectangular sections. The battery pack was positioned at the lower central region of the chassis to reduce the center of gravity and improve stability. Suspension mounting brackets and motor support structures were reinforced to prevent localized stress concentration. The design also incorporated triangulated members to enhance rigidity and minimize bending deflection under vertical loading. Iterative simulations were conducted to refine member thickness and reduce unnecessary mass while maintaining structural safety margins.

The vehicle frame includes a transmission shaft connecting the electric motor to the rear axle. The shaft is designed considering torsional loading conditions. The torque transmitted by the motor is calculated using:

$$T = \frac{P \times 60}{2\pi N}$$

where

P is motor power and
N is rotational speed.

The shaft diameter is determined using torsion theory:

$$T = \frac{\pi}{16} \tau d^3$$

where

τ is allowable shear stress and
d is shaft diameter.

Based on calculated torque and allowable shear stress for mild steel, the shaft diameter is selected to ensure safe operation under maximum load conditions.

VII. CONCLUSION

This study presented the design and finite element-based structural analysis of a two-seater electric vehicle frame. The

Journal of Artificial Intelligence and Emerging Technologies
Volume 2, Issue 2, pp 12-15, February 2025
ISSN (online): 3049-1967
DOI: https://doi.org/10.47001/JAIET/2025.202003
Available Online at https://www.jaiet.com

CAD model was developed and evaluated under static loading conditions to determine stress distribution and deformation behavior. The results confirmed that the proposed frame design maintains stress levels below the material yield strength and achieves an adequate factor of safety. The lightweight tubular configuration enhances structural efficiency while ensuring passenger safety. The study demonstrates the effectiveness of finite element analysis as a reliable tool for pre-fabrication structural validation. Future work may include dynamic analysis, crash simulation, and material optimization using advanced lightweight alloys or composites.

REFERENCES

[1] S. Rajasekaran and G. Sankarasubramanian, "Design and analysis of automotive chassis using finite element method," International Journal of Engineering Research & Technology (IJERT), vol. 2, no. 7, pp. 1254–1260, 2013.

A. R. Abu Bakar, M. M. Rahman, and M. M. Noor, "Finite element analysis of a space frame chassis structure," Journal of Mechanical Engineering and Sciences, vol. 2, pp. 178–188, 2012.

[2] P. K. Mallick, Vehicle Body Engineering and Design, SAE International, 2014.

[3] R. C. Hibbeler, Mechanics of Materials, Pearson Education, 2014.

[4] T. D. Gillespie, Fundamentals of Vehicle Dynamics, SAE International, 1992.

[5] S. S. Rao, The Finite Element Method in Engineering, Butterworth-Heinemann, 2011.

[6] V. B. Bhandari, Design of Machine Elements, McGraw Hill, 2010.

[7] J. Reimpell, H. Stoll, and J. W. Betzler, The Automotive Chassis: Engineering Principles, Butterworth-Heinemann, 2001.

[8] N. Jazar, Vehicle Dynamics: Theory and Application, Springer, 2008.

[9] A.Sharma and S. Chauhan, "Structural performance evaluation of electric vehicle chassis using FEA," International Journal of Automotive Engineering and Technologies, vol. 6, no. 4, pp. 182–190, 2017.

[10] Z. Marciniak, J. Duncan, and S. Hu, Mechanics of Sheet Metal Forming, Butterworth-Heinemann, 2002.

[11] M. B. Topac and S. Ercan, "Stress analysis of a vehicle chassis by finite element method," Journal of Engineering and Natural Sciences, vol. 28, pp. 111–120, 2010.

[12] J. Fenton and R. Hodkinson, Lightweight Electric/Hybrid Vehicle Design, Elsevier, 2001.

[13] W. D. Callister, Materials Science and Engineering: An Introduction, Wiley, 2007.

[14] M. F. Ashby, Materials Selection in Mechanical Design, Elsevier, 2011.

[15] K. J. Bathe, Finite Element Procedures, Prentice Hall, 2006.

Citation of this Article:

Muhammad Fajar Nugroho, & Andi Saputra Wijaya. (2025). Finite Element Based Structural Analysis of a Two-Seater Electric Vehicle Frame. *Journal of Artificial Intelligence and Emerging Technologies (JAIET).* 2(2), 12-15. Article DOI: https://doi.org/10.47001/JAIET/2025.202003

***** End of the Article *****