International Journal of Research Publications in Engineering, Technology and Management (IJRPETM)

|www.ijrpetm.com | ISSN: 2454-7875 | editor@ijrpetm.com |A Bimonthly, Peer Reviewed & Scholarly Journal|

||Volume 2, Issue 2, March- April 2019||

DOI:10.15662/IJRPETM.2019.0202001

# Finite Element Analysis and Structural Optimization of Electric Vehicle Chassis Design

**Karan Mehta Swati**

Malnad College of Engineering, Hassan, Karnataka, India

**ABSTRACT:** In 2018, the design of electric vehicle (EV) chassis leveraged Finite Element Analysis (FEA) and structural optimization to meet competing demands of lightweight construction, safety, and dynamic performance. This study surveys two representative FEA applications from that year: one involving a lightweight electric bus frame and another analyzing an electric car chassis. Yang et al. performed FEA under full-load bending, torsion, emergency braking, and steering conditions to optimize an electric bus frame using structural steel and aluminum alloy combinations, ultimately reducing weight without sacrificing structural integrity Matec ConferencesResearchGate. Meanwhile, Mardji & Prasetiyo used ANSYS 18.1 along with Autodesk Inventor 2018 to model an electric car chassis, revealing key metrics: equivalent stress of ~59.98 MPa, equivalent elastic strain ~$3.325\times10^{-4}$ mm/mm, total deformation ~2.43 mm, and a safety factor of ~3.55 Matec ConferencesMendeley. These works demonstrate that FEA was being systematically applied for both lightweight and structural reliability goals in EV chassis design. This paper builds on those 2018 studies: it examines methodologies for lightweight optimization and strength validation, compares results and approaches, and discusses their implications for EV chassis engineering. The findings highlight the viability of FEA-driven design adjustments in achieving weight savings, ensuring safety margins, and addressing diverse load scenarios. Structural optimization via material substitution and geometry adjustments emerges as a powerful strategy. This study concludes with insights on how 2018-era techniques laid the groundwork for further advancements in operational efficiency, design methods, and integrated optimization workflows in EV chassis development.

**KEYWORDS:** Electric vehicle chassis, Finite Element Analysis, Structural optimization, Lightweight design, Bus frame, ANSYS 2018

## I. INTRODUCTION

By 2018, electrification had reshaped priorities in automotive chassis design: efficiency and range became paramount. Weight reduction featured prominently, alongside structural integrity under diverse loading scenarios. FEA emerged as the primary simulation tool enabling engineers to validate and refine chassis designs before physical prototyping.

One case from 2018 illustrates this shift: researchers applied FEA to an electric bus frame combining structural steel and aluminum alloy. They analyzed the structure across full-load bending, torsion, emergency braking, and steering conditions, achieving weight reduction while preserving mechanical performance Matec ConferencesResearchGate. In parallel, another study involved modeling an electric car chassis in Autodesk Inventor 2018 and analyzing it with ANSYS 18.1. The simulation produced results—equivalent stress (~59.98 MPa), deformation (~2.43 mm), safety factor (~3.55)—that validated the structural adequacy of the design under static loads Matec ConferencesMendeley.

These 2018 developments exemplify two key FEA use cases in EV chassis design: (1) lightweight optimization through material and geometry changes while maintaining load-bearing capabilities, and (2) rigorous validation of chassis models under static load conditions to ensure safety and reliability.

This paper aims to build on post-2018 perspectives by reconstructing the era's research landscape, techniques, and design rationales. Specifically, it will examine the applied FEA methodologies, assumptions, and performance outcomes in the context of 2018's computational and engineering standards. Through comparative analysis, we seek to uncover best practices, design trade-offs, and future pathways for integrating FEA-driven optimization in EV chassis engineering workflows.

|www.ijrpetm.com | ISSN: 2454-7875 | editor@ijrpetm.com |A Bimonthly, Peer Reviewed & Scholarly Journal|

||Volume 2, Issue 2, March- April 2019||

DOI:10.15662/IJRPETM.2019.0202001

## II. LITERATURE REVIEW

The 2018 literature on EV chassis design through FEA and structural optimization reveals two prominent studies, each serving different design emphases.

**1. Lightweight Optimization of Electric Bus Frame+**

Yang et al. conducted finite element analysis on an electric bus frame, considering four critical driving conditions: full-load bending, torsion, emergency braking, and emergency steering. By strategically altering frame structure and material composition (steel and aluminum alloy), they achieved considerable weight reduction while enhancing mechanical performance under simulated conditions Matec ConferencesResearchGate. This study underscores how FEA enables engineers to navigate the trade-off between weight and load-bearing capability in large EV platforms.

**2. Structural Analysis of Electric Car Chassis**

Mardji & Prasetiyo conducted strength analysis of a small electric car chassis using ANSYS 18.1. Modeling was performed in Autodesk Inventor 2018, and FEA yielded quantifiable results: equivalent stress of 59.983 MPa, strain of $3.325\times10^{-4}$ mm/mm, deformation of 2.43 mm, and a safety factor of 3.55 Matec ConferencesMendeley. The study emphasizes the role of FEA in verifying structural integrity and safety in lightweight chassis materials and assembly.
These studies contribute to establishing a 2018 baseline where FEA was firmly integrated into structural design cycles of EV chassis. Key themes emerge:

- **Multi-condition loading scenarios:** addressing real-world events like braking and steering torsion.
- **Material and geometric optimization:** leveraging composite designs for mass reduction.
- **Quantitative validation metrics:** using stress, strain, deformation, and safety factor outcomes.

However, literature gaps include limited exploration of dynamic loading, modal behavior, or full-scale topology optimization. The 2018 focus remained primarily on static and quasi-static conditions within limited design scopes.
This review underscores that while FEA was already a critical part of EV chassis design in 2018, integrating dynamic analysis and advanced optimization methods remained areas ripe for further exploration.

## III. RESEARCH METHODOLOGY

This study retrospectively reconstructs and compares the FEA and structural optimization methodologies applied in 2018 EV chassis research.

**Case Selection and Comparative Framework**

We select two representative 2018 studies:
Electric bus frame lightweight optimization using structural-steel and aluminum alloy under multiple static load conditions Matec ConferencesResearchGate.
Electric car chassis strength analysis using Autodesk Inventor 2018 modeling and ANSYS 18.1-based static stress assessment Matec ConferencesMendeley.

**Data Extraction and Analytical Criteria**

For each study, methodological components are cataloged:

- Modeling approach (software, level of detail).
- Materials used and their properties.
- Loading conditions simulated (bending, torsion, braking, steering).
- Optimization strategies (e.g., material substitution, geometry changes).
- Output metrics (stress, strain, deformation, safety factor, weight reduction).

**Comparative Analysis**

The extracted data are organized into a comparative matrix to elucidate similarities and distinctions:

- Scope (bus vs. car chassis).
- Objectives (lightweight optimization vs. validation).
- Techniques (multi-condition FEA vs. static load FEA).
- Outcomes and implications for design reliability and efficiency.

|www.ijrpetm.com | ISSN: 2454-7875 | editor@ijrpetm.com |A Bimonthly, Peer Reviewed & Scholarly Journal|

||Volume 2, Issue 2, March- April 2019||

DOI:10.15662/IJRPETM.2019.0202001

**4. Synthesis and Interpretation**

- We interpret how each methodology reflects 2018's engineering capabilities:
- The bus frame study exemplifies multi-constraint optimization mindset within FEA frameworks.
- The car chassis analysis reflects precision in validating safety factors within design limits.

**5. Limitations and Reflection**

We acknowledge limitations in the 2018 approach such as limited dynamic or fatigue analysis, absence of modal or topology optimization techniques within these studies. We propose that subsequent methodological progression could have included advanced optimization algorithms and broader load-case considerations.

This methodology reconstructs the engineering reasoning and technical workflows of 2018-era FEA applied to EV chassis, providing insight into prevailing practices and the emerging potential in structural design optimization.

## IV. RESULTS AND DISCUSSION

**Bus Frame Lightweight Optimization**

FEA under four critical load conditions (load bending, torsion, emergency braking, steering).

Modified structure combining steel and aluminum alloy achieved weight reduction while maintaining or enhancing structural performance Matec ConferencesResearchGate.

**Electric Car Chassis Strength Validation**

Detailed modeling and static FEA (Autodesk + ANSYS 18.1).

Obtained equivalent stress ≈ 59.98 MPa, equivalent elastic strain ≈ $3.325 \times 10^{-4}$ mm/mm, deformation ≈ 2.43 mm, safety factor ≈ 3.55 Matec ConferencesMendeley.

**Discussion:**

- **Weight vs. Structural Performance:** The bus frame analysis illustrates that through material selection and FEA-guided geometry adjustments, designers could achieve lightweight architectures without compromising safety—a crucial consideration for extending EV battery life.
- **Validation and Safety Assurance:** The car chassis study highlights how FEA provides essential validation data (stress, strain, deformation, safety factor) to ensure chassis integrity, especially under static load conditions.
- **Methodological Emphasis:** Both studies reflect a predominantly static-load FEA approach prevalent in 2018. The bus frame analysis added complexity through multiple load scenarios, but dynamic analysis and optimization methods remained largely untapped in the EV context.
- **Foundations for Advanced Techniques:** While adequate for establishing design reliability, these methods lacked incorporation of topology optimization, modal analysis, fatigue simulations, or dynamic loading—areas that modern EV chassis design increasingly emphasizes.
- **Engineering Implications:** The consistent safety margins and weight reductions confirmed by these studies underscore the effectiveness of FEA-driven methodologies in EV chassis design. Moreover, these 2018 examples served as benchmarks and learning prototypes for subsequent developments in integrated design and analysis workflows.

## V. CONCLUSION

In 2018, Finite Element Analysis played a pivotal role in the structural design and optimization of electric vehicle chassis. Through examples involving a lightweight bus frame and a validated electric car chassis, FEA demonstrated its capability to navigate the balance between reducing mass and ensuring structural integrity. Key outcomes included effective weight reduction strategies, quantifiable safety margins, and multi-scenario validation under static loads.

However, the methodologies of that period were largely confined to static or quasi-static analyses, with limited exploration of dynamic, modal, or topology-optimization-based approaches. These limitations mark latency in methodological evolution, despite FEA's central role.

Collectively, the 2018 studies laid a foundational groundwork: FEA had become indispensable in EV chassis engineering, enabling agile iteration on materials, geometry, and design validation. They served as stepping stones toward more integrated, dynamic, and comprehensive structural optimization frameworks that would follow in subsequent years.

|www.ijrpetm.com | ISSN: 2454-7875 | editor@ijrpetm.com |A Bimonthly, Peer Reviewed & Scholarly Journal|

||Volume 2, Issue 2, March- April 2019||

DOI:10.15662/IJRPETM.2019.0202001

## VI. FUTURE WORK

Building on the 2018 landscape, future research directions include:

1. **Dynamic and Modal Analysis:** Incorporate vibration analysis and modal assessments to ensure chassis resilience under real driving conditions.
2. **Topology and Advanced Structural Optimization:** Leverage topology optimization techniques to identify material-efficient chassis configurations beyond manual design adjustments.
3. **Fatigue and Crash Loading Simulations:** Extend FEA to cover life-cycle durability under repetitive loading and crash scenarios.
4. **Material Innovation:** Explore advanced composites and hybrid materials for weight reduction and stiffness enhancement.
5. **Integrative MDO Workflows:** Implement multidisciplinary design optimization blending structural, thermal, and material constraints to achieve holistic EV chassis design.
6. **Experimental Correlation:** Conduct physical testing for validation of FEA results, reinforcing the simulation's reliability.

## REFERENCES

1. Yang, Z., Deng, B., Deng, M., & Sun, G. (2018). A study on Finite Element Analysis of electric bus frame for lightweight design. *MATEC Web of Conferences, 175*, 03049 Matec ConferencesResearchGate.
2. Mardji, A., & Prasetiyo, D. (2018). Strength analysis chassis of UM electric cars using finite element method. *MATEC Web of Conferences, 204*, 07017 Matec ConferencesMendeley.