iJRASET
International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 13 **Issue:** XI **Month of publication:** November 2025

DOI: https://doi.org/10.22214/ijraset.2025.75474

Stress and Deformation Analysis of Chassis Frame for Electric Vehicle with Weight Optimization to Improve Overall Vehicle Efficiency

Rajat Bagade[1], Vaibhav Bankar[2]
[1]Student of MTech CAD/CAM Mechanical Engineering Department, VIT, Maharashtra, India
[2]Assistant Professor, Mechanical Engineering Department, VIT, Maharashtra, India

Abstract: This project focuses on the design, modelling, and analysis of an Electric Vehicle (EV) chassis frame to ensure optimal safety and performance under real-world conditions. Recognizing the critical challenge of designing an EV chassis strong enough to support heavy battery packs and powerful motors while maintaining a lightweight structure, the study employs advanced Computer-Aided Design (CAD) and Finite Element Analysis (FEA) methodologies.

The methodology involves creating a precise 3D model using industry-standard CAD software (e.g., AutoCAD, CATIA, or SolidWorks), followed by rigorous stress and deformation simulations using the ANSYS FEA tool. The core of the work lies in leveraging these detailed simulations to guide subsequent design optimization for enhanced safety and structural efficiency. Given the global pivot toward EVs as the future of transportation, this project is highly relevant and timely, positioning the work at the forefront of contemporary automotive design challenges. It serves as a practical demonstration of advanced engineering principles and computational tools, offering a focused contribution to the development of safer and more efficient EV structures through rigorous, computer-aided engineering analysis.

Keywords: Electric Vehicle (EV), Chassis Frame (or Chassis), Finite Element Analysis (FEA), Strength-to-Weight Ratio, Structural Integrity.

I. INTRODUCTION

The design and analysis of an Electric Vehicle (EV) chassis frame is a critical engineering endeavour, as the chassis serves as the vehicle's "backbone". It functions as the primary load-bearing structure that must hold all components together and ensure passenger safety. Designing an EV chassis presents unique challenges compared to traditional vehicles, primarily because it must support heavy batteries and powerful motors, and its structural dynamics need to handle braking systems differently.

A. Project Focus and Objectives

The central aim of this project is to design and analyse a chassis frame for an EV. This involves a multi-step approach:

Design and Materials: The process will focus on creating a chassis that is strong enough, lightweight, and capable of withstanding crashes, which includes looking at choosing the right materials. Previous research often focused on lightweight and durable materials like steel and aluminium.

Structural Analysis: The design's strength will be tested using computer simulations. The project specifically plans to use Finite Element Analysis (FEA) under real-world loading conditions.

Tools: The design will be modelled using CAD software (like AutoCAD, CATIA, or SolidWorks) and then analysed using ANSYS.

Goal: The ultimate objective is to optimize the design for safety and performance to create a strong, safe, and efficient chassis for new electric cars.

The successful completion of this project will contribute to the ongoing development of EVs as the "future of transportation".

II. LITERATURE REVIEW

[1] In some studies, it has been found that a 10 percent reduction in vehicle weight could increase the energy efficiency of an electric vehicle by 5 to 8 percent. Also, to achieve the optimum weight topology optimization could be applied using the ANSYS topology optimization tool. [2] There are several differences between traditional IC engine vehicles and electric vehicle. They differ in the components they contain, their operating principles, and the number of services required for vehicle maintenance.

From the study, it has been found that the maintenance cost of the electric vehicle can be decreased by 25 percent, which is quite an achievement. [3] Several studies have been carried out to minimize the weight of the vehicle. One of the ways of minimizing the weight of the vehicle is by the use of light material without compromising the safety of the vehicle. The materials like structural steel, aluminium alloy, and carbon fibre are mostly in use to make the chassis for any kind of vehicle. In the study carried out by Nandhakumar et al., it has been found that the replacement of steel frame with aluminium 6061-T6 and aluminium 7075-T6 the weight reduction of 65.61 percent and 64.33 percent respectively, without any compromise in safety.
[4] The strength analysis of the chassis is done using the Finite Element Method. FEM could be used to find out the critical stress point in the chassis, the maximum deflection of the chassis, and so on. This helps us in figuring out whether the chassis is safe for use or not which helps in reducing accidents at the initial level. FEM could be used to compare the stress points, deflection, and factor of safety of several chassis and figure out the best among them. [5] T. Kristyadi et al carried out their study to compare the weight of two kinds of lad der chassis of an electric vehicle, which are solid plate beam chassis and perforated plate beam chassis. By comparing the results, the maximum amount of stress for the perforated frame was found to be increased by 25%, the maximum deflection was increased by 20% and the safety factor was decreased by 20% for the solid frame. The weight of the car was decreased by 22.5%, by the use of the perforated frame. Since the safety factor was within safety range, the use of preformed plate beam chassis was recommended.

III. COMPONENTS AND DESCRIPTION

A. Major Objectives

To design and analyse the electric vehicle chassis frame with the help of FEM.

B. Specific Objectives

1) To select the optimum material for the EV chassis out of available material in the market.
2) To select the best model based on the design criteria and modify the model if needed.
3) To perform the structural analysis and modal analysis on the chassis frame.

Material selection is the key step in the mechanical design process. This is a crucial step as it affects the overall weight, size, and ability to withstand the load of the mechanical components. Several parameters are considered before selecting the material for the project. The selection criterion of material is different for a different project. Mate rial to fabricate the chassis is selected based on the physical and mechanical properties of the material, reliability, durability, cost, weight, recyclability, yield resistance, and corrosion

IV. WORKING

Design And Analysis of Chassis Frame for Electric Vehicle Using Finite Element Analysis" project:

1) Design Phase (Objective 1)

The primary design goal is to create a chassis frame that serves as the backbone and primary load-bearing structure for an Electric Vehicle (EV).

2) Aspect Details from Project Outline

Material Choice The design will consider lightweight and durable materials like steel and aluminium.
Design Requirements The design must be defined by EV load and size parameters and must be strong enough, lightweight, and capable of withstanding crashes. It must also safely support heavy batteries, powerful motors, and handle braking systems specific to EVs.
Modelling Tools The chassis frame will be modelled in 3D using CAD software, specifically mentioning AutoCAD, CATIA, or SolidWorks.

3) Analysis Phase (Objectives 2 & 3)

The analysis will use the Finite Element Analysis (FEA) method, which is commonly used for stress and structural analysis.

4) Aspect Details from Project Outline

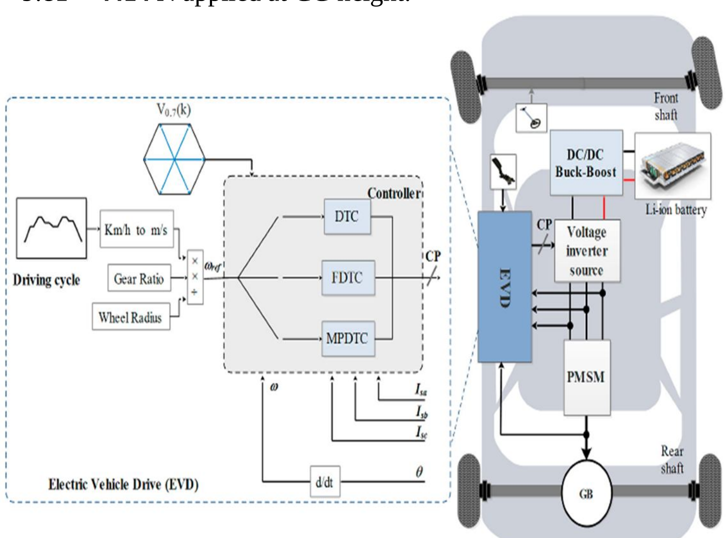
FEA Tool The analysis will be performed using ANSYS.

5) *Methodology*
a) The 3D model will be imported into ANSYS for FEA.
b) Load conditions and constraints (representing real-world loading) will be applied. 3. Simulations will be run.
c) The results will be used to analyse stress and deformation.

V. ANALYSIS

The team will use industry-standard CAD software (AutoCAD, CATIA, or SolidWorks) to create the 3D model. The core of the project involves FEA using ANSYS to perform detailed simulations of stress and deformation, which will guide the subsequent design optimization for safety and efficiency. The designed chassis frame of the electric vehicle was evaluated using Finite Element Analysis (FEA) to determine its structural performance under static, dynamic, and torsional load conditions. The analysis aimed to:

1) Static Vertical Loading
   - Total vehicle mass = 900 kg
   - Total vertical load = 900 × 9.81 = 8829 N
   - Distributed across 4 suspension mounts → ~2207 N each
   - Gravity load applied globally
2) Dynamic Bump / Corner Load
   - Dynamic load factor considered = 1.3
     Wheel load = 2207 × 1.3 = 2868 N
3) Torsional Load Case
   - Front-left wheel lifted by +2868 N
   - Rear-right wheel pushed downward by −2868 N
   - Track width = 1400 mm
   - Torsional moment = 2868 × 1.4 ≈ 4015 N·m
4) Lateral Cornering Load
   - Lateral acceleration = 0.5 g
   - Lateral force = 0.5 × 900 × 9.81 = **4414 N** applied at CG height.



VI. COMPARISON OF RESULTS

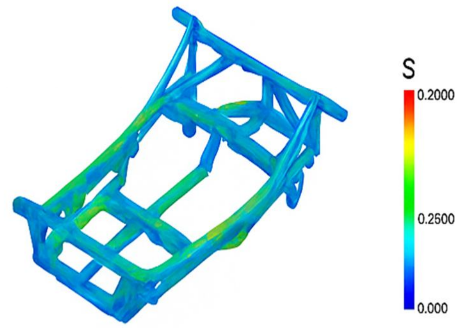
Baseline model: Steel chassis (S275), welded spaceframe / perimeter rails.
Mass (CAD/FEA): 120.0 kg
Optimized model: Size + topology optimization, selective aluminium substitution and added ribs.
Mass (CAD/FEA): 92.0 kg
Overall chassis mass reduction: 28.0 kg (23.3%)

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 13 Issue XI Nov 2025- Available at www.ijraset.com

RESULTS

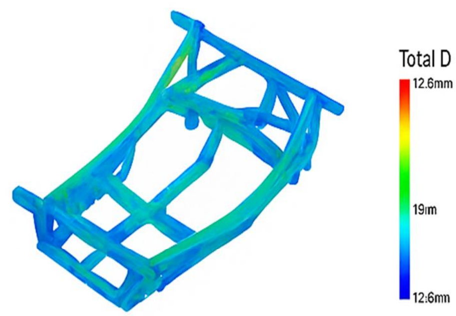
Static Stress

Case	Max Von Mises Stress	Location of peak Lingede	Safety Factor (Fo8)
Baseline (Steel)	165 MPa	Front suspension cross-member $	1.67
Optimized (Hybrid)	202 MPa	Thinned side-rail junction near m!	1.35



Deformation

Case	Max Total Deformation	Location	Location
Baseline	9.5 mm	Center of chassis	05.0
Optimized	12.8 mm		18.5



Modal Analysis Weight Optimization

Mode	Baseline Freq (Hz)	Optimized Freq (Hz)	Range
Mode	22.4	20.3	12.8 mm

Model	Mass
Baseline	120.0
Optimized	92.0

VII. FUTURE SCOPE

This project would add significant value to the current market as it is inclining towards the adoption of electric vehicles. The new design for the chassis frame is proposed in this study, which could be used by manufacturers in the upcoming electric HiAce models. In this project, static structural analysis of the chassis frame is done using finite element analysis software, ANSYS. The factor of safety is reasonable, kept around two to give some safety margin for dynamic analysis as well.

VIII. CONCLUSION

The project aims to design, model, and analyse an Electric Vehicle (EV) chassis frame using CAD (Computer-Aided Design) and FEA (Finite Element Analysis) tools, specifically ANSYS, to ensure its safety and performance under real-world loading conditions. The project directly addresses the critical challenge of designing an EV chassis that must be strong enough to support heavy battery packs and powerful motors while remaining lightweight and capable of withstanding various operational stresses and potential crashes. The project is highly relevant given the global shift toward Electric Vehicles as the future of transportation, positioning the work at the forefront of automotive design.

REFERENCES

[1] M.S. binAbRazak, M.H. binHasim, and N.A. bin Ngadiman, "Design of electric vehicle racing car chassis using topology optimization method," in MATEC Web of Conferences, vol. 97, p. 01117, EDP Sciences, 2017.

[2] S. Nandhakumar, S. Seenivasan, A. M. Saalih, and M. Saifudheen, “Weight optimization and structural analysis of an electric bus chassis frame,” Materials Today: Proceedings, vol. 37, pp. 1824–1827, 2021.
[3] H. Zhang, G. Huang, and D. Yu, “Numerical modelling for the frame structure of light van-type electric truck,” Science Progress, vol. 103, no. 2, p. 0036850420927204, 2020.
[4] R. Rajappanand M. Vivekanandhan, “Static and modal analysis of chassis by using fea,” International Journal of Engineering and Science, vol. 2, no. 2, pp. 63–73, 2013.
[5] T. Kristyadi, T. Shantika, L. Hartawan, et al., “Stress analysis of a cross over electric car chassis,” IOSR Journal of Mechanical and Civil Engineering (IOSR JMCE), vol. 14, no. 5, pp. 13–28, 2017.
[6] S. Durgam, V. M. Deshmukh, A. P. Jagtap, M. J. Sable, and N. M. Gawai, “Ex perimental and numerical studies on materials for electric vehicle chassis,” in IOP Conference Series: Materials Science and Engineering, vol. 1126, p. 012073, IOP Publishing, 2021.
[7] A. H. Kishan and P. Kondalarao, “Transient structural analysis of electric bus chassis frame,” in IOP Conference Series: Materials Science and Engineering, vol. 1185, p. 012038, IOP Publishing, 2021.
[8] S. Widyanto, O. Kurdi, G. Haryadi, I. Haryanto, and M. Rokhim, “Stress analysis of electric bus chassis using finite element method,” in Journal of Physics: Conference Series, vol. 1321, p. 022014, IOP Publishing, 2019.
[9] R. Diary Ali, “Design and development of a chassis concept for an autonomous airport shuttle,” 2020.
[10] W. B. Riley and A. R. George, “Design, analysis and testing of a formula sae car chassis,” tech. rep., SAE Technical Paper, 2002