Design And Development of In-House EV'S Chassis

Vyanktesh Sharma[1], Vikas Chandrawanshi[2], Tejas Bagde[3], Vishu Sharma[4], Prof. Girish Kumar Khare[5], Asst. Prof. Priyanka Shukla[6]
[1234]*Mechanical Department, Oriental Institute of Science & Technology, Bhopal Madhya Pradesh, 4662022*
[5,6]*Correspondence author*

Abstract: **Present study focusses on modelling, designing and performance analysis of Automotive chassis system under moderate loading conditions. CAD modelled Automotive Chassis system for in house built EV is analyzed using FEA approach for structural integrity under static loading using Fusion 360 and ANSYS software. Mild steel is chosen as the key element for fabricating chassis frame for its ease of availability and low cost. In-house fabricated chassis frame is tested to endure vehicle load. Results from the study shows that the frame is capable of supporting load in the range 700 to 850 kgs. Results from our analytical approach supports the physical results for our designed chassis system.**

The chassis is a vital part of any vehicle, providing the necessary support and stability while handling various forces from vehicle weight and payload of vehicle. In electric vehicles, designing chassis becomes even more critical, therefore during developing an in-house EV's chassis we had to optimize the weight in a way that was feasible for more readily available materials like mild steel. During the designing phase we utilized Fusion 360 and Ansys to check on various load condition, and were able to get our chassis capable of carrying weight in range of 700 to 850 kilograms.

Keywords: **Chassis design, FEA (Finite element method), Mild Steel chassis, Structural Optimization, Weight feasibility**

INTRODUCTION

The chassis is the main foundation of a vehicle that supports essential parts like body, engine, suspension system, and drivetrain. It is critical for guaranteeing the stability, safety, and overall performance of the car under various circumstances. Effective weight management occupants safety during impacts, rigidity and ease of manufacturing are among several technical factors that are taken under account during chassis design. Testing of modern designs which mostly use efficient mild steel—is being driven by high demand for lighter and more energy-efficient result.

EV design differs greatly from traditional internal combustion engine (ICE) vehicle chassis, which are constructed on mechanical components such as engines and gearboxes. A novel chassis design that emphasizes lightweight construction must be developed for electric vehicles in order to accommodate high-voltage batteries, electric motors and power electronics which will help in increasing the range.

This paper proposes a design method that combines computer simulations, various materials and iterative prototyping to tackle the various challenges faced during the production of vehicle.

The primary objectives are to:
(1) design a robust yet lightweight chassis using high-strength materials
(2) maximize space utilization for battery placement to enhance stability
(3) guarantee scalability for future vehicle models
(4) validate the design through finite element analysis (FEA) and physical testing.
The paper combines the theoretical research with actual engineering techniques and also showcases the completed EV, with the designed and developed chassis.

METHODOLOGY

The methodology is structured into three key primary phases, specifically: Design Concept, Development of Physical Prototype and Finite Element Analysis.

Phase 1:
Design Concept: To find best practices in EV chassis design, a comprehensive literature analysis and benchmarking study were conducted in the first phase. This influenced the creation of a chassis design that aimed to strike a compromise between strong structural performance and lightweight construction. Design elements like joint

arrangements, cross-sectional profiles, and material choice were carefully taken into account. Fusion 360 was used to produce the chassis' intricate 3D models and 2D technical drawings. As the blueprint for the fabrication process, these drawings—which included orthographic projections and isometric views ensured exact component integration and dimensional precision.

Phase 2:

Development of the Physical Prototype: An actual EV chassis prototype was created using the intricate CAD designs. The fabrication process included cutting mild steel tubes, plates, to size, using ARC welding to precisely weld the individual frame pieces together, checking alignment and squaring with calipers and level. Because of the thorough documentation of the fabrication process, quality control and conformance to the design parameters were guaranteed.

Phase 3:

Finite Element Analysis (FEA) for Analytical Validation: Fusion 360, Simulations was used to perform Finite Element Analysis (FEA) in order to assess the chassis design's structural performance. The CAD design served as the basis for the creation of a comprehensive FEA model that included boundary conditions and material attributes that faithfully represented the expected operational scenarios. To evaluate stress distribution and deformation under different loading situations, such as Loading situations, e.g., maximum payload, cornering forces, braking forces, impact loads, etc., static structural analysis was carried out. The inherent frequencies and mode shapes of the chassis were ascertained by modal analysis, which is essential for assessing its vibration properties and avoiding resonance. To make sure the chassis would remain stable under compressive stresses, buckling analysis was done.

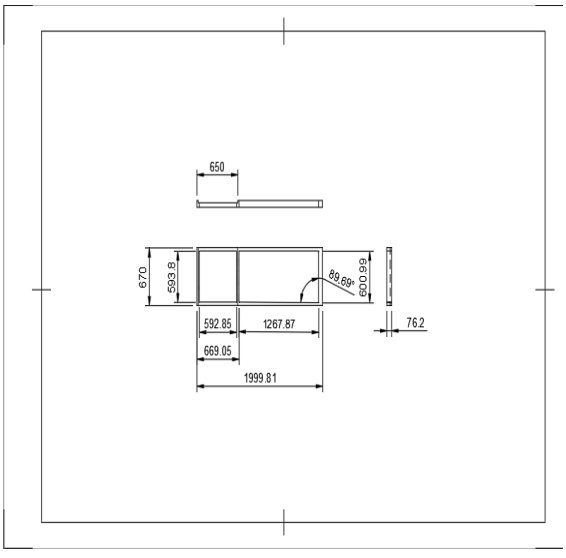
A clear comprehension of the chassis' structural behavior was made possible by the use of color-coded stress contour plots and deformation diagrams with exaggerated displacement to show the FEA results.

Integration and Validation: Experimental data from the prototype's physical testing was used to compare and validate the FEA results. Impact testing using a drop tower to evaluate energy absorption, vibration testing with accelerometers to identify natural frequencies, static load testing with the software. The correctness and dependability of the research findings were guaranteed by the combination of computational analysis and experimental validation, which resulted in well-informed choices about material choice, structural alterations, and overall chassis optimization.

Additionally, by emphasizing regions of high stress concentration and deformation, these visualizations provide a visual depiction of the chassis' behavior under varied loading situations. For example, color-coded stress graphs make it easy to determine the chassis' load-bearing capacity and structural integrity. The inherent frequencies and mode shapes of the chassis are also shown in modal analysis visualizations, which are essential for evaluating its vibration properties and guaranteeing operational stability.

These illustrations and visualizations are included as an essential component of the study technique, not just for aesthetic reasons. They make it easier to comprehend the structural analysis, the design process, and the physical prototype's confirmation. The report improves clarity and accessibility by graphically presenting the design and analysis results, which helps readers understand the intricate structural dynamics of the EV chassis and recognize the thoroughness of the study process.

2D DRAWING



DESIGN AND ANALYSIS REPORT

Analyzed File	Static Analysis 1 v1
Version	Autodesk Fusion (2601.0.90)
Creation Date	2025-04-01, 22:50:02
Author	hp

Report Properties

Title	Studies
Author	Lenovo

Simulation Model 1

Study 1 - Static Stress

Study Properties

Study Type	Static Stress
Last Modification Date	2025-04-01, 22:42:20

Settings

General

Contact Tolerance	0.10 mm
Remove Rigid Body Modes	No

Mesh

Average Element Size (% of model size)	
Solids	10
Scale Mesh Size Per Part	No
Average Element Size (absolute value)	-
Element Order	Parabolic
Create Curved Mesh Elements	Yes
Max. Turn Angle on Curves (Deg.)	60
Max. Adjacent Mesh Size Ratio	1.5
Max. Aspect Ratio	10
Minimum Element Size (% of average size)	20

Adaptive Mesh Refinement

Number of Refinement Steps	0
Results Convergence Tolerance (%)	20
Portion of Elements to Refine (%)	10
Results for Baseline Accuracy	von Mises Stress

Materials

Component	Material	Safety Factor
Body2	Steel, Mild	Yield Strength

Steel, Mild

Density	7.850E-06 kg / mm^3
Young's Modulus	220000.00 MPa
Poisson's Ratio	0.275
Yield Strength	207.00 MPa
Ultimate Tensile Strength	345.00 MPa
Thermal Conductivity	0.045 W / (mm C)
Thermal Expansion Coefficient	1.200E-05 / C
Specific Heat	480.00 J / (kg C)

Contacts

Mesh

Type	Nodes	Elements
Solids	1427	566

Load Case1

Constraints

Fixed1

Type	Fixed
Ux	Fixed
Uy	Fixed
Uz	Fixed

Loads

Force1

Type	Force
Magnitude	8000.00 N
X Value	0.00 N
Y Value	0.00 N
Z Value	-8000.00 N
Force Per Entity	No

Results

Result Summary

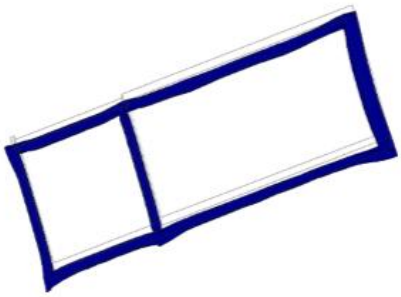
Name	Minimum	Maximum
Safety Factor		
Safety Factor (Per Body)	15.00	15.00
Stress		
von Mises	0.019 MPa	0.042 MPa
1st Principal	-0.014 MPa	0.008 MPa
3rd Principal	-0.052 MPa	-0.03 MPa
Normal XX	-0.017 MPa	0.006 MPa
Normal YY	-0.017 MPa	0.008 MPa
Normal ZZ	-0.048 MPa	-0.03 MPa
Shear XY	-0.004 MPa	0.004 MPa
Shear YZ	-0.009 MPa	0.009 MPa
Shear ZX	-0.009 MPa	0.008 MPa
Displacement		
Total	0.00 mm	1.450E-05 mm
X	-3.821E-06 mm	4.205E-06 mm
Y	-4.786E-06 mm	4.807E-06 mm
Z	-1.324E-05 mm	0.00 mm
Reaction Force		
Total	0.00 N	125.22 N
X	-26.123 N	25.297 N
Y	-21.674 N	20.692 N

Z	-2.542 N	125.203 N
Strain		
Equivalent	9.332E-08	2.288E-07
1st Principal	0.00	9.043E-08
3rd Principal	-2.678E-07	-1.140E-07
Normal XX	-1.028E-08	8.346E-08
Normal YY	-8.369E-09	8.999E-08
Normal ZZ	-1.872E-07	-1.066E-07
Shear XY	-4.529E-08	4.514E-08
Shear YZ	-1.091E-07	1.094E-07
Shear ZX	-1.101E-07	8.913E-08
Contact Force		
Total	0.00 N	0.00 N
X	0.00 N	0.00 N
Y	0.00 N	0.00 N
Z	0.00 N	0.00 N

Safety Factor

Safety Factor (Per Body)

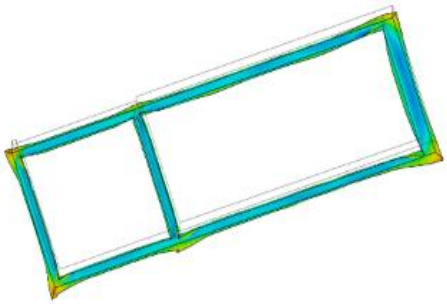
0.00 8.00



Stress

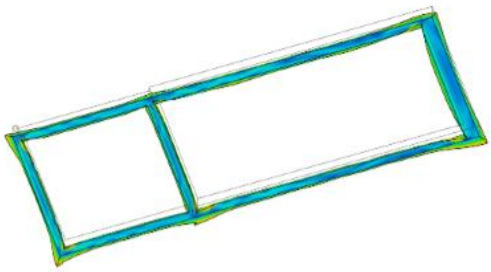
von Mises

[MPa] 0.019 0.042



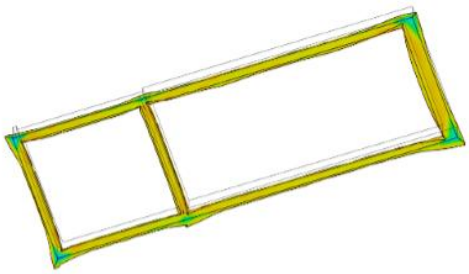
1st Principal

[MPa] -0.014 0.008



3rd Principal

[MPa] -0.052 -0.03



Analyzed File	Static Analysis 1 v4
Version	Autodesk Fusion (2601.0.90)
Creation Date	2025-04-01, 23:13:47
Author	hp

Report Properties

Title	Studies
Author	Lenovo

Simulation Model 1

Study 4 - Structural Buckling

Study Properties

Study Type	Structural Buckling
Last Modification Date	2025-04-01, 23:10:47

Settings

General

Contact Tolerance	0.10 mm
Number of Modes	3

Mesh

Average Element Size (% of model size)	
Solids	8
Scale Mesh Size Per Part	No
Average Element Size (absolute value)	-
Element Order	Parabolic
Create Curved Mesh Elements	Yes
Max. Turn Angle on Curves (Deg.)	60
Max. Adjacent Mesh Size Ratio	1.5
Max. Aspect Ratio	10
Minimum Element Size (% of average size)	20

Adaptive Mesh Refinement

Number of Refinement Steps	0

Results Convergence Tolerance (%)	20
Buckling Mode	1

Materials

Component	Material	Safety Factor
Body2	Steel, Mild	Yield Strength

Steel, Mild

Density	7.850E-06 kg / mm^3
Young's Modulus	220000.00 MPa
Poisson's Ratio	0.275
Yield Strength	207.00 MPa
Ultimate Tensile Strength	345.00 MPa
Thermal Conductivity	0.045 W / (mm C)
Thermal Expansion Coefficient	1.200E-05 / C
Specific Heat	480.00 J / (kg C)

Contacts

Mesh

Type	Nodes	Elements
Solids	1642	664

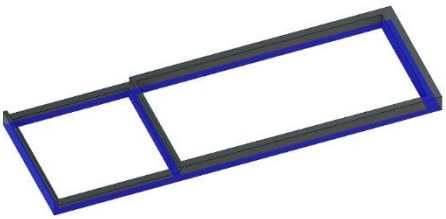
Load Case1

Constraints

Fixed1

Type	Fixed
Ux	Fixed
Uy	Fixed
Uz	Fixed

Selected Entities

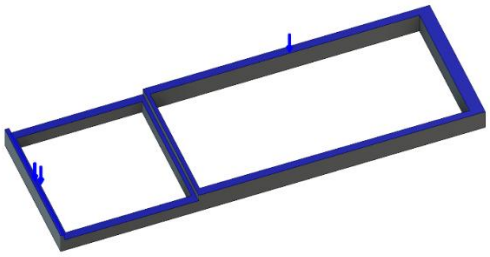


Loads

Force1

Type	Force
Magnitude	8000.00 N
X Value	0.00 N
Y Value	0.00 N
Z Value	-8000.00 N
Force Per Entity	No

Selected Entities



Results

Result Summary

Load Multiplier
Buckling Mode 1: 274810.00 x Load
Buckling Mode 2: 302280.594 x Load
Buckling Mode 3: 327389.688 x Load

RESULTS AND DISCUSSION

This work utilizes Fusion 360 to recreate a big vehicle's chassis. The chassis was subjected to a structural study using Fusion 360. Moderate-strength structural steel is one of the three materials used in the analysis. We have used the FE approach to analyse both static and dynamic loads.

Although hefty and economically, sectioned chassis with different thicknesses ranging from 38.1 mm to 76.2 mm is dependable. In order to increase the box type chassis profile's variable thickness, structural thickness optimizations were carried out for 38.1 mm, 50.8 mm, and 76.2 mm.

The overall displacements for the frame thickness are under the range of 6 mm, 38.1 mm (original model) thickness, and 50.8 mm and 76.2 mm box-type chassis profile were found in the static analyses of a one-ton load-bearing automobile chassis.

The stress values in finite element analysis may exceed these values. The manufacturer's specifications like a linear, elastic, and static analysis are the cause. The stress value increases in the static analysis without running into any opposition. Furthermore, just one vertical direction and axis are used to process the static testing. There are various ways that reality burdens might be impacted. This circumstance makes the most. The maximum von-misses stresses in the chassis analysis are 0.042 MPa for 38.1 mm × 50.8 mm.

CONCLUSION

This study presented a practical approach to designing a lightweight and durable chassis for

electric vehicles, combining advanced simulations with hands-on prototyping. By focusing on the unique needs of EVs, such as battery placement and weight distribution, the research achieved a chassis that maintains strength without unnecessary bulk. Finite Element Analysis (FEA) confirmed the design's reliability, showing low stress levels (0.042 MPa) and minimal deformation under load. The use of mild steel, along with strategic thickness adjustments (ranging from 38.1 mm to 76.2 mm), helped optimize performance while keeping costs manageable. The project highlights how modern engineering tools—like CAD modeling and FEA—can streamline chassis development, ensuring safety and efficiency. Moving forward, collaboration across engineering disciplines will be essential to refine these designs further and meet the growing demands of next-generation electric vehicles.

ACKNOWLEGDEMENT

We would like to express the essential gratitude to all those who contributed to the successful completion of this research paper. Specifically, we would like to acknowledge the invaluable role of Autodesk Fusion 360 and designing and developing a chassis described in this paper. The software has been very useful in providing parametric modelling to simulation and in providing an iterative design process which help us in providing a precise optimisation of the chassis structure. We would like to particularly be thankful for the online sources and support communities associated with Fusion 360 and Ansys Mechanical which provides us with the conceptual design, robust precision & accuracy. This research would not have been possible without the availability and accessibility of this powerful design platform

REFERENCE

[1] International Journal of Scientific and Technological Research www.iiste.org ISSN 2422-8702 (Online) Vol 1, No.6, 2015 1 | P a g e www.iiste.org Design, Analysis and Optimization of Heavy Vehicle Chassis Using Finite Element Analysis.

[2] ISSN: 2319-5967 ISO 9001:2008 Certified International Journal of Engineering Science and Innovative Technology (IJESIT) Volume 5, Issue 1, January 2016 DESIGN & ANALYSIS OF AUTOMOBILE CHASSIS.

[3] The International Journal Of Engineering And Science (Ijes) ||Volume|| 2 ||Issue|| 2 ||Pages|| 63-73 ||2013|| Issn: 2319 – 1813 Isbn: 2319 – 1805 www.theijes.com The IJES Page 63 Static and Modal Analysis of Chassis by Using FEA.

[4] STRESS ANALYSIS OF HEAVY DUTY TRUCK CHASSIS USING FINITE ELEMENT METHOD O Kurdi, R Abd- Rahman, M N Tamin.

[5] https://hdl.handle.net/20.500.14540/18886

[6] https://www.researchgate.net/publication/290315133 Design, Analysis and Optimization of Chassis for an Electric Vehicle.

[7] Design And Analysis Of Chassis For Electric Vehicle International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 Volume: 08 Issue: 08 | Aug 2021 www.irjet.net p-ISSN: 2395-0072.