ISSN 2278-2540 | DOI: 10.51583/IJLTEMAS | Volume XIV, Issue XII, December 2025

Investigations on the Mechanical and Vibrational Characteristics of an Electric Vehicle Chassis using Finite Element Analysis (FEA)

[1]Mrityunjay Tiwari, [2]Raja Sekhar Dondapati

[1] School of Advanced Engineering, UPES, Bidholi Via Prem Nagar, Dehradun, Uttarakhand, 248007

[2]School of Mechanical Engineering, Lovely Professional University, Phagwara, Punjab 144411, India

DOI : https://doi.org/10.51583/IJLTEMAS.2025.1412000083

Received: 17 December 2025; Accepted: 23 December 2025; Published: 07 January 2026

ABSTRACT

The chassis of an electric vehicle represents a critical nexus between safety, efficiency, and performance. Its role extends beyond conventional vehicular frameworks, assuming responsibility for housing and protecting vital components, managing thermal dynamics, and providing a robust safety envelope. With the rapid evolution of EV technology, continued advancements in chassis design and engineering will undoubtedly play a pivotal role in shaping the future of sustainable mobility. Structural analysis of the Chassis yields deformation, stress and strain. Similarly, vibration analysis provides the natural frequencies of vibration. Under dynamic conditions, the investigations of these characteristics are done in the present work. Further, failure modes of vibrations are also found using Finite Element Analysis.

From the present study, it is concluded that, with an increase of pressure by 19%, the average percentage increase in the stress, strain and total deformation is observed to lie in the range of 18 to 20%. Further, it is observed that, the amplitude in the Grey Cast Iron is higher as compared to the other materials considered in the present investigation while the amplitude in the Titanium Alloy is lowest. Additionally, vibration frequency in the Structural Steel is higher as compared to the other materials considered in the present investigation while vibration frequency in the Grey Cast Iron is lowest.

Keywords: FEA; Structural and Vibrational Analysis; Phase Response; Harmonic Analysis; Frequency; Deformation; Electric vehicle analysis.

INTRODUCTION

The chassis of an electric vehicle (EV) stands as a critical structural foundation that transcends conventional vehicular frameworks. This thesis delves into the profound significance of the chassis in the context of EVs, emphasizing its pivotal role in ensuring safety, enhancing efficiency, and optimizing overall performance. Unlike internal combustion engine vehicles, where the chassis primarily bears the weight of the vehicle and provides structural integrity, an EV's chassis carries additional responsibilities owing to the unique characteristics of electric propulsion systems.

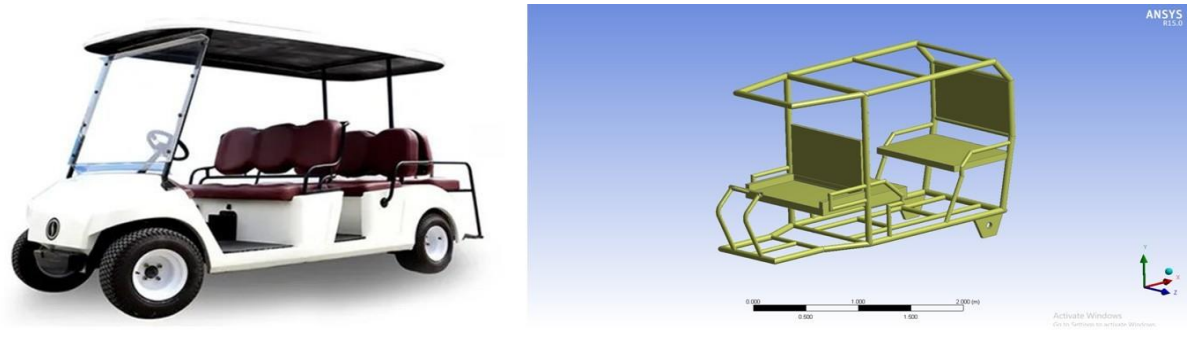


Figure 1: Electric Golf Cart – An example of an Electric Vehicle, along with its Geometric Model

ISSN 2278-2540 | DOI: 10.51583/IJLTEMAS | Volume XIV, Issue XII, December 2025

Firstly, the chassis of an EV serves as an integral component for housing and safeguarding the battery pack and electric drive train. Its design and material composition significantly influence the vehicle's weight distribution, centre of gravity, and overall stability. Therefore, a well-engineered chassis is essential in achieving optimal handling characteristics, especially given the substantial mass of the battery. Moreover, the chassis must also provide adequate protection to the battery against external impacts and environmental elements, safeguarding the integrity of the power source.

Furthermore, the chassis acts as a platform for integrating suspension systems and handling dynamics, contributing to a smooth and comfortable ride for occupants.

Additionally, in terms of safety, the chassis undergoes rigorous testing and engineering to meet stringent crashworthiness standards. Its structural integrity is paramount in safeguarding occupants during collisions, necessitating advanced materials and construction techniques. The use of high-strength materials and innovative manufacturing processes ensures that the chassis effectively absorbs and dissipates energy, minimizing the impact on passengers.

In conclusion, the chassis of an electric vehicle represents a critical nexus between safety, efficiency, and performance. Its role extends beyond conventional vehicular frameworks, assuming responsibility for housing and protecting vital components, managing thermal dynamics, and providing a robust safety envelope. With the rapid evolution of EV technology, continued advancements in chassis design and engineering will undoubtedly play a pivotal role in shaping the future of sustainable mobility.

LITERATURE REVIEW

Structural analysis of the Chassis yields deformation, stress and strain. Similarly, vibration analysis provides the natural frequencies of vibration. Under dynamic conditions, the investigations of these characteristics are done in the present work. Further, failure modes of vibrations are also found using Finite Element Analysis.

Recently, a structural analysis has been done [1] and reported the total deflection, equivalent stress, safety factor, and first six mode shapes for the chassis design. Previously, a computer-aided analysis was done [2] for spray boom and battery-operated vehicle sprayer working together as it's crucial to account for the entire load while designing the system. Further, Design and Vibration Analysis of Go-kart Chassis [3] showed change in material doesn't cause significant change in vibration. Also, monocoque-type chassis frame and was examined under static loading, frontal impact, side impact, rear impact, front rollover, and side rollover incidents for safety analysis [4]. In addition, literature review about the characteristics of a variety of materials [5] - including carbon fiber, aluminium alloy, and titanium, used for chassis, has been investigated and compared to those of normal mild steel. Moreover, the design and vibration analysis of a heavy-duty vehicle (trailer) chassis utilizing finite element method (FEM) has determined that vibration-induced deformation is the main cause of chassis failure over time [6]. Formerly, it was inferred that steel with an 'I' section has superior strength to withstand high loads and induced low deformation and stress distribution when compared to other cross sections [7]. In past works, we have considered the problem of chassis mode shapes and natural frequency. The analysis results shows that the frequency range varies with different vibration modes like torsional and bending has been identified [8]. Model analysis is conducted on a truck chassis to optimize effect of vibration on chassis. As chassis always undergoes to continuous uniform loading and it is inferred chassis must have high natural frequency so that while working in vibrations it should no bend or deform permanently [9]. Prior to it, through FEM analysis, the vibrations affecting the gearbox was calculated and harmonic analysis has been conducted to find maximum and minimum amplitude against frequency [10]. Determining the truck chassis' dynamic properties, including its natural frequencies and mode shapes and watching how the truck chassis reacts to static loads was worked upon previously [11].

Recent studies on EV chassis optimization and analysis include Scurtu & Moldovanu (2024), who applied topology optimization to minimize chassis mass while maintaining structural integrity, demonstrating practical lightweighting strategies [15]. Zamzam et al. (2025) conducted finite element analysis on EV chassis structures to assess stress and deformation under static conditions, providing empirical simulation benchmarks [16]. Wang

ISSN 2278-2540 | DOI: 10.51583/IJLTEMAS | Volume XIV, Issue XII, December 2025

et al. (2025) integrated modal analysis with optimization to improve vibrational performance of EV chassis frameworks, offering insights into NVH-oriented design [17]. These recent studies underscore the trend toward hybrid optimization strategies, experimental coupling, and cost-performance analyses, motivating the present work's expanded focus.

Finite Element Method (FEM) analysis is vital in chassis design, ensuring structural integrity, weight optimization, and safety. It enables engineers to simulate stress, deformation, and heat transfer, aiding material selection and crash testing for improved durability and passenger protection. FEM also reduces prototyping costs and ensures compliance with industry regulations, making it a cornerstone in the development of efficient, reliable chassis systems across automotive and aerospace sectors. Literature review has been done to identify the materials and loads that can be applied on the Electric Vehicles. FEM's versatility and accuracy have revolutionized the way we approach structural analysis and optimization, ultimately leading to safer, more efficient, and innovative products and structures. As technology continues to advance, FEM analysis will remain a cornerstone of modern engineering practices, driving progress and innovation across numerous disciplines.

Problem Description

Structural Analysis

For the intended analysis, varying pressure is applied vertically on the chassis, ranging from 3000Pa to 7000Pa and the Stress, Strain, Total Deformation and Directional Deformation (in X, Y & Z directions) are recorded.

Table 1: Property Table of different materials used for the study

	Structural Steel	**Stainless Steel**	**Grey Cast Iron**	**Aluminium Alloy**	**Titanium Alloy**
Density (kg/m^3)	7850	7750	7200	2770	4620
Compressive Ultimate Strength (Pa)	0	0	8.2e+08	0	0
Compressive Yield Strength (Pa)	2.5e+08	2.07e+08	0	2.8e+08	9.3e+08
Tensile Yield Strength (Pa)	2.5e+08	2.07e+08	0	2.8e+08	9.3e+08
Tensile Ultimate Strength (Pa)	4.6e+08	5.86e+08	2.4e+08	3.1e+08	1.07e+09
Young's Modulus (Pa)	2.e+11	1.93e+11	1.1e+11	7.1e+10	9.6e+10
Poisson's Ratio	0.3	0.31	0.28	0.33	0.36
Bulk Modulus (Pa)	1.666e+11	1.693e+11	8.333e+10	6.960e+10	1.142e+11
Shear Modulus (Pa)	7.692e+10	7.366e+10	4.296e+10	2.669e+10	3.529e+10

ISSN 2278-2540 | DOI: 10.51583/IJLTEMAS | Volume XIV, Issue XII, December 2025

Vibrational Analysis

By choosing the desired modes to extract, such as in this study, first 6 modes are selected for analysis. And the natural frequencies obtained in these modes are recorded.

Table 2: Frequency of Vibration of different materials at varying modes

Modes	Frequency (Hz)				
	Structural Steel	Stainless Steel	Grey Cast Iron	Aluminium Alloy	Titanium Alloy
1	3.0998	3.0654	2.3994	3.1119	2.8055
2	4.9710	4.9147	3.8497	4.9871	4.4937
3	10.6220	10.5040	8.2225	10.6630	9.6128
4	12.6080	12.4600	9.7720	12.6330	11.3680
5	15.8650	15.6840	12.2870	15.9130	14.3320
6	16.9870	16.8040	13.1430	17.0700	15.4120

Harmonic Analysis

A periodic force of 3000Pa is applied on chassis to calculate the vibrations it will experience. And frequency of the desired range is set to obtain the harmonic response, which is further recorded. The maximum and minimum frequency obtained in frequency analysis is used for harmonic analysis with a hundred intervals between them.

Table 3: Maximum and Minimum Frequency of Vibration for different materials

Modes	Frequency (Hz)				
	Structural Steel	Stainless Steel	Grey Cast Iron	Aluminium Alloy	Titanium Alloy
Minimum	3.0998	3.0654	2.3994	3.1119	2.8055
Maximum	16.9870	16.8040	13.1430	17.0700	15.4120

RESEARCH METHODOLOGY

Performing an analysis of a chassis in ANSYS involves several steps to ensure accurate results. Here's a general overview of the process:

- **Preprocessing:**

Geometry Import: Start by designing the 3D CAD model of the chassis in ANSYS DesignModeler[12]. Common file formats for import include STEP, IGES, or native CAD formats.

ISSN 2278-2540 | DOI: 10.51583/IJLTEMAS | Volume XIV, Issue XII, December 2025

Material Properties: Define the material properties for the chassis components, including Young's modulus, Poisson's ratio, and density. ANSYS provides a material library for common materials, but you can also input custom material properties. All analysis mentioned below are run for five different materials. Namely, Structural Steel, Stainless Steel. Grey Cast Iron, Aluminium Alloy and Titanium Alloy.

Mesh Generation: Create a finite element mesh for the chassis geometry. The mesh should have an appropriate level of refinement, with finer mesh near areas of interest like load application points or stress concentrations. ANSYS offers various meshing tools, including automatic meshing and manual mesh refinement.

The steps followed in the preprocessing stage remain constant for all the three analysis (Static Structural, Vibration & Harmonic). Further, different boundary conditions and analysis settings are defined for each study as follows.

Static Structural Analysis

Boundary Conditions:

Constraints: Define boundary conditions by specifying where the chassis is fixed or constrained. Commonly, certain nodes or faces to represent the actual mounting points of the chassis are fixed.

Fixed constraints were provided at the base of the chassis.

Loads: Apply the loads that the chassis will experience during its intended use. These may include forces, pressures, or accelerations. Ensure that loads are applied accurately and in the correct directions.

For the intended analysis, varying pressure is applied vertically on the chassis, ranging from 3000Pa to 7000Pa.

Analysis Settings:

Analysis Type: Choose the appropriate analysis type for your chassis. In this study, we consider variation in Stress, Strain, Total Deformation & Directional Deformation (X, Y & Z) under different pressure application.

Solver Settings: Configure solver settings, including convergence criteria, time steps (for dynamic analysis), and solution methods.

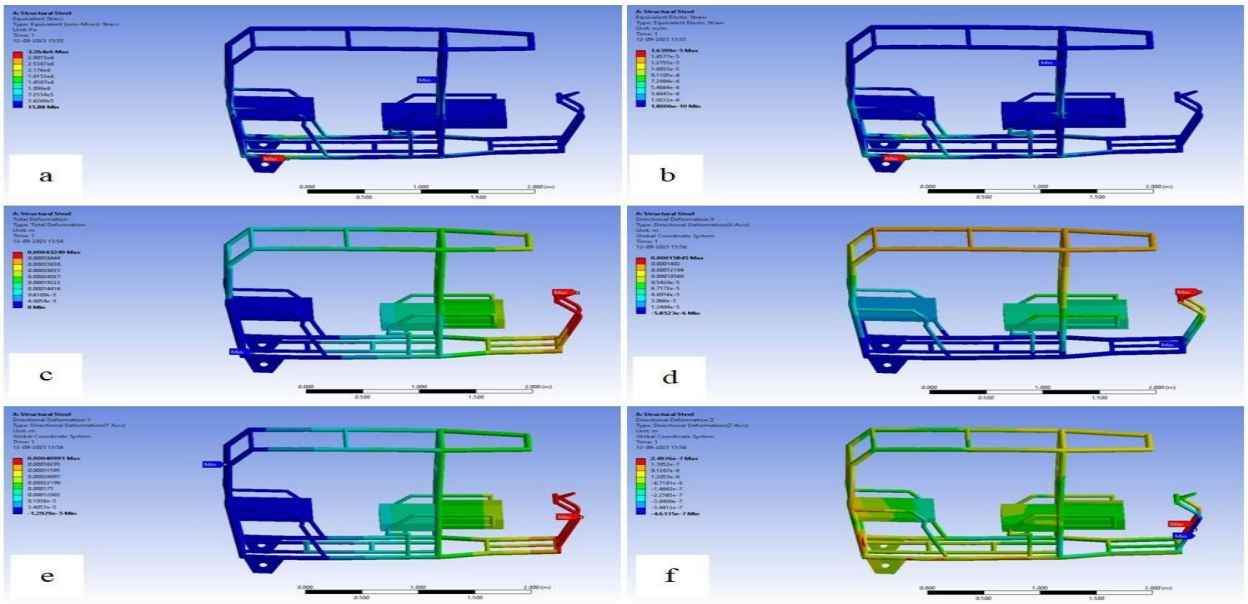


Figure 2: Contours of (a)Equivalent Stress (b)Equivalent Strain (c)Total Deformation (d)Directional Deformation in X Direction (e) Directional Deformation in Y Direction & (f) Directional Deformation in Z Direction for Structural Steel at 7000Pa

ISSN 2278-2540 | DOI: 10.51583/IJLTEMAS | Volume XIV, Issue XII, December 2025

Solution:

a) **Run Analysis:** Initiate the analysis solver to compute the results. ANSYS will perform calculations and simulations based on the inputs provided.

b) **Monitor Progress:** Monitor the solver's progress and check for any convergence issues or errors.

- **Post Processing:**

a) **Results Visualization**: Once the analysis is complete, visualize the results using ANSYS post processing tools. Contours of Stress and Strain distributions, deformation plots, and other relevant data will be illustrated.

b) **Generate Report:** Generate reports or documentation summarizing the analysis results, including stress values, pressure distribution, and deformation profiles.

Vibration Analysis

- **Boundary Conditions:**

a) **Constraints:** Define boundary conditions by specifying where the chassis is fixed or constrained. Commonly, certain nodes or faces to represent the actual mounting points of the chassis are fixed.

Fixed constraints were provided at the base of the chassis.

b) **Modal Analysis Settings:** Set up the analysis type as a modal analysis. In ANSYS, this is often done in a separate analysis system called the "Mechanical APDL" or using the Workbench environment, depending on ANSYS version.

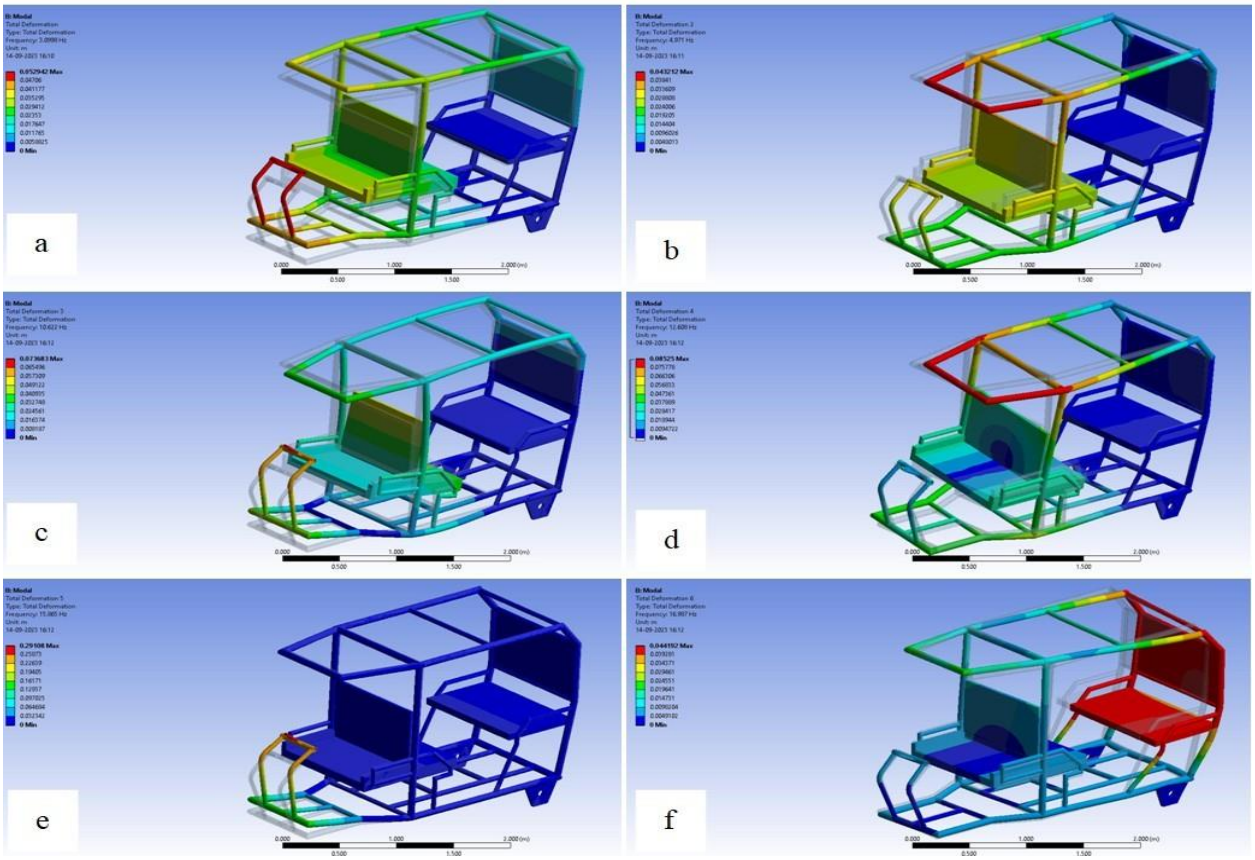


Figure 3: Contours of Total Deformation of Structural Steel at (a)Mode 1 (b) Mode 2 (c) Mode 3 (d) Mode 4 (e) Mode 5 & (f) Mode 6 under Vibrational Analysis

ISSN 2278-2540 | DOI: 10.51583/IJLTEMAS | Volume XIV, Issue XII, December 2025

Analysis Settings:

c) **Solver Settings:** Configure solver settings, including the number of modes to extract (eigen modes) for solution.

d) **Mode Selection:** Choose the desired modes to extract, such as the first few modes or a specific mode range. In this study, 6 modes are selected or analysis.

- **Solution:**

a) **Run Modal Analysis:** Initiate the modal analysis solver to compute the natural frequencies and mode shapes of the chassis.

b) **Review Convergence:** Check the convergence of the analysis and ensure that the solver successfully calculates the modes.

- **Post Processing:**

a) **Mode Shapes Visualization:** Visualize and examine the mode shapes of the chassis. These represent how the chassis vibrates at each natural frequency.

b) **Natural Frequencies:** Analyze the natural frequencies to identify any critical modes that may affect the chassis's performance.

c) **Mode Participation Factors:** Calculate and review mode participation factors to understand which mode shapes contribute the most to specific responses or vibrations.

Harmonic Analysis

- **Boundary Conditions:**

a) **Constraints:** Define boundary conditions by specifying where the chassis is fixed or constrained. Commonly, certain nodes or faces to represent the actual mounting points of the chassis are fixed.

Fixed constraints were provided at the base of the chassis.

b) **Load Application:** Define the harmonic loads or excitations applied to the chassis. A periodic force of 3000Pa is applied to chassis to calculate the vibrations it will experience.

- **Analysis Setting:**

a) **Solver Settings:** Configure solver settings for harmonic analysis. The maximum and minimum frequencies obtained from vibration analysis are specified as frequencies for harmonic excitation and a hundred numbers of harmonic frequencies are set to be analyzed.

b) **Frequency Range:** Set the frequency range for the analysis based on the expected harmonic frequencies of interest.

ISSN 2278-2540 | DOI: 10.51583/IJLTEMAS | Volume XIV, Issue XII, December 2025

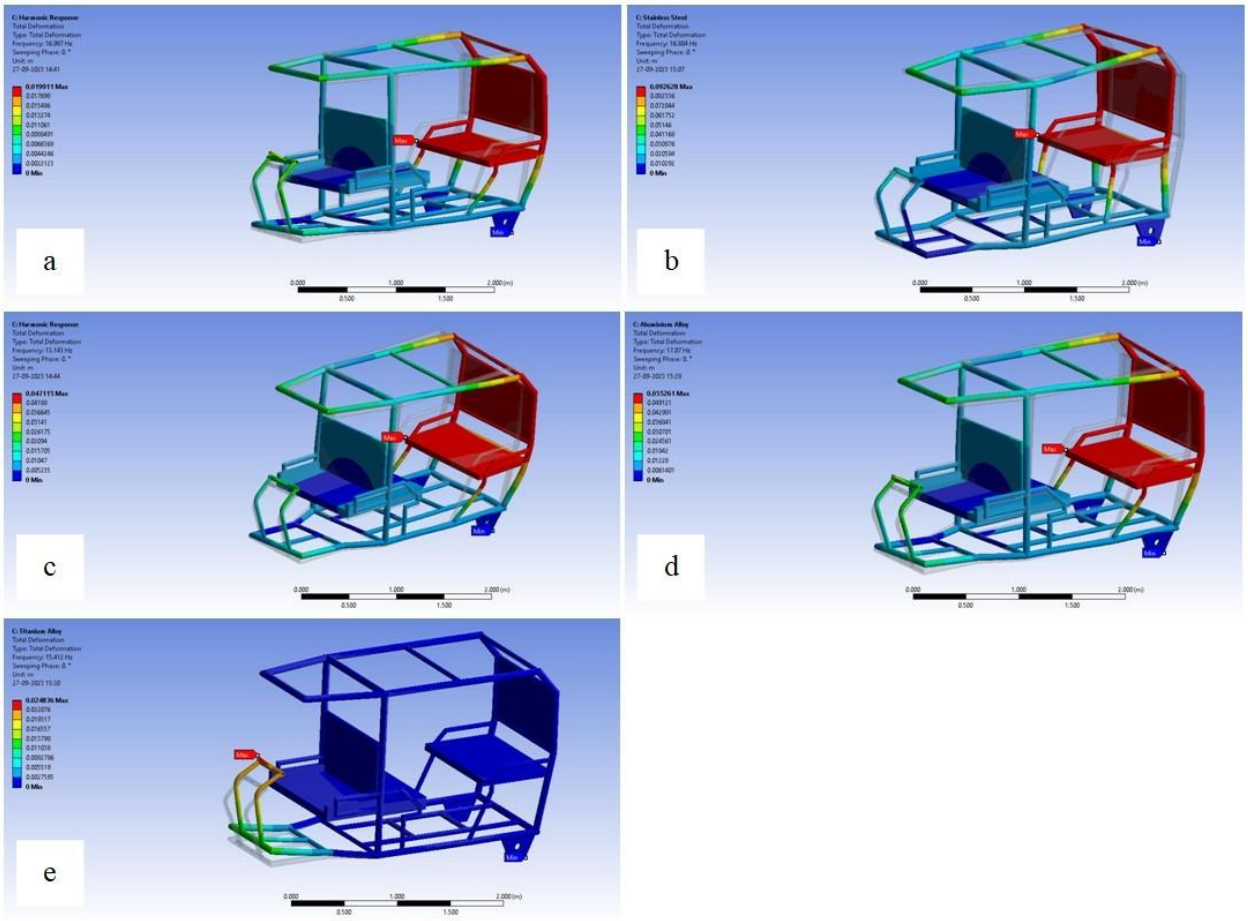


Figure 4: Contours of Total Deformation in under (a)Structural Steel (b)Stainless Steel (c) Grey Cast Iron (e)Aluminium Alloy & (f)Titanium Alloy Harmonic Analysis

- **Solution:**

a) **Run Harmonic Analysis:** Initiate the harmonic analysis solver to compute the response of the chassis to the harmonic excitations. ANSYS will calculate the response at specified frequency intervals within the defined frequency range.

b) **Monitor Convergence:** Check the convergence of the analysis and ensure that the solver successfully calculates the response for each harmonic frequency.

- **Post Processing:**

a) **Results Visualization:** Visualize and examine the response of the chassis to harmonic excitations. Plot displacement, stress, and other relevant results as a function of frequency.

b) **Frequency Response Analysis:** Analyze the amplitude and phase of the chassis's response at each harmonic frequency. This helps identify resonant frequencies and potential issues.

c) **Stress and Displacement Analysis:** Examine stress and displacement distributions to assess the chassis's structural integrity under harmonic loading.

Mesh Sensitivity Analysis:

To ensure that simulation results are independent of finite element discretization, a mesh sensitivity study was conducted on the reference chassis model. Three mesh densities were evaluated as seen in Table 4:

ISSN 2278-2540 | DOI: 10.51583/IJLTEMAS | Volume XIV, Issue XII, December 2025

Table 4: Frequency of Vibration of different materials at varying modes

Mesh Level	Element Size (mm)	Nodes	Max Stress (MPa)
Coarse	10	85000	214
Medium	5	230000	198
Fine	2.5	610000	195

The maximum stress results converge between the medium and fine meshes (<2% difference), indicating sufficient result stability at the medium mesh level used in this study. Based on these findings, all subsequent simulations apply the medium mesh as the optimal balance between accuracy and computational cost. A similar trend was observed for modal frequencies, further validating the mesh independence of the key outcomes.

Cost Benefits and Manufacturing Feasibility Considerations

In automotive mass production, mechanical performance alone does not determine material selection; cost and manufacturability are equally critical.

Cost Analysis:

c) Aluminium alloys offer favourable strength-to-weight ratios and established mass-production casting/extrusion methods, but raw material cost is higher than conventional steel.

d) Conventional steel remains cost-effective with mature fabrication infrastructure, yet its higher density negatively impacts overall vehicle weight.

e) Titanium, while exhibiting superior mechanical properties, is prohibitively expensive for cost-sensitive EV segments.

Manufacturing Feasibility:

High-volume steel and aluminium parts benefit from standardized processing (stamping, robotic welding). Titanium fabrication requires specialized tooling and heat treatment control, increasing production cycle times and capital costs. These trade-offs are critical for OEMs targeting sub-$40K EV segments. This brief cost-benefit analysis, in Table 5, suggests that while advanced materials such as titanium show structural advantages in simulation, their adoption must be weighed against cost and manufacturing scalability.

Table 5: Cost & Manufacturability Analysis Table of different materials used for the study

Material	Approx. Cost (USD/Kg)	Manufacturability
Steel	1.0-1.5	Excellent (welding, stamping)
Aluminium Alloy	2.0-3.5	Very Good (extrusion, casting)
Titanium Alloy	30-40	Challenging (specialized forming)

ISSN 2278-2540 | DOI: 10.51583/IJLTEMAS | Volume XIV, Issue XII, December 2025

RESULTS AND DISCUSSION

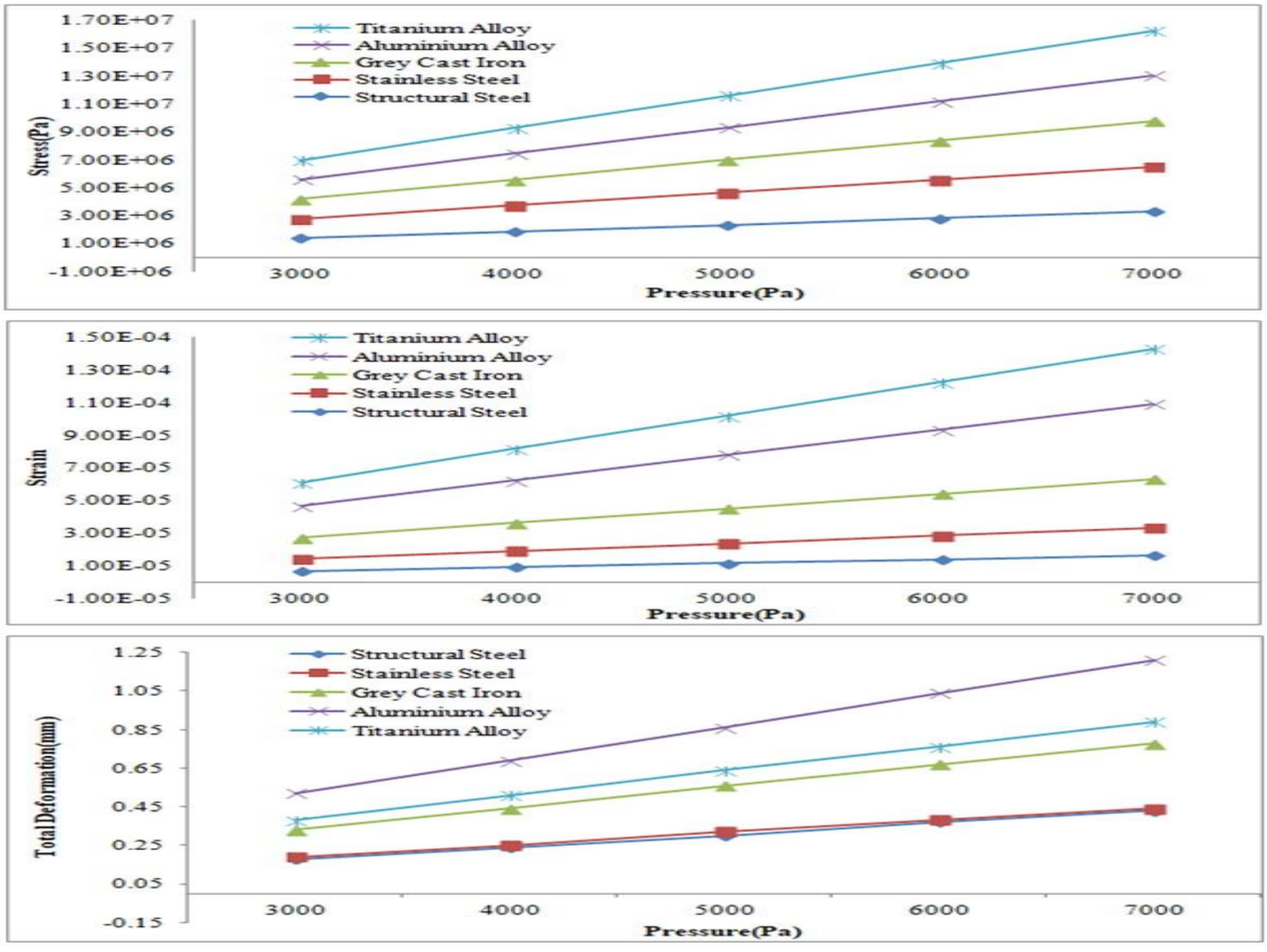
The present work is focused on investigating the variation in the effect of various pressure loads and vibrations on chassis made up of different materials. The following inferences were made from the simulations run;

From Figure 5, it can be observed that the stress, strain and total deformation increase with the applied pressure. Further, it can be concluded that,

- The Stress in the Titanium Alloy is higher as compared to the other materials considered in the present investigation while Structural Steel is lowest.
- The Strain in the Titanium Alloy is higher as compared to the other materials considered in the present investigation while Structural Steel is lowest.
- The Total Deformation in the Aluminium Alloy is higher as compared to the other materials considered in the present investigation while Structural Steel is lowest.

Figure 6 shows the variation of Frequency with respect to the modes. It can be observed that the frequency increases from first to last mode. Further, it can be concluded that, the frequency of the Aluminium Alloy is higher as compared to the other materials considered in the present investigation while Grey Cast Iron is the lowest.

Figure 7 shows the variation of Amplitude with respect to Frequency. It can be observed that the amplitude of the Grey Cast Iron is higher as compared to the other materials considered in the present investigation while Titanium Alloy is the lowest.



ISSN 2278-2540 | DOI: 10.51583/IJLTEMAS | Volume XIV, Issue XII, December 2025

Figure 5: Variation of Equivalent Stress (a) Equivalent Stress (b) Equivalent Strain & (c) Total Deformation with varying Pressure for different materials

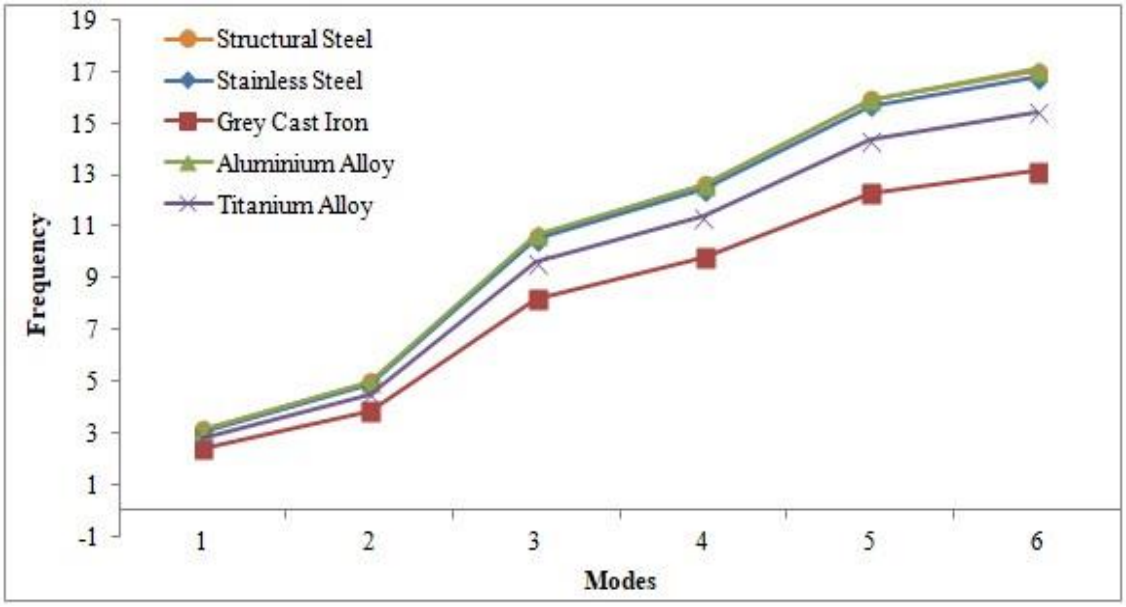


Figure 6: Variation of Frequency at different Modes in different materials

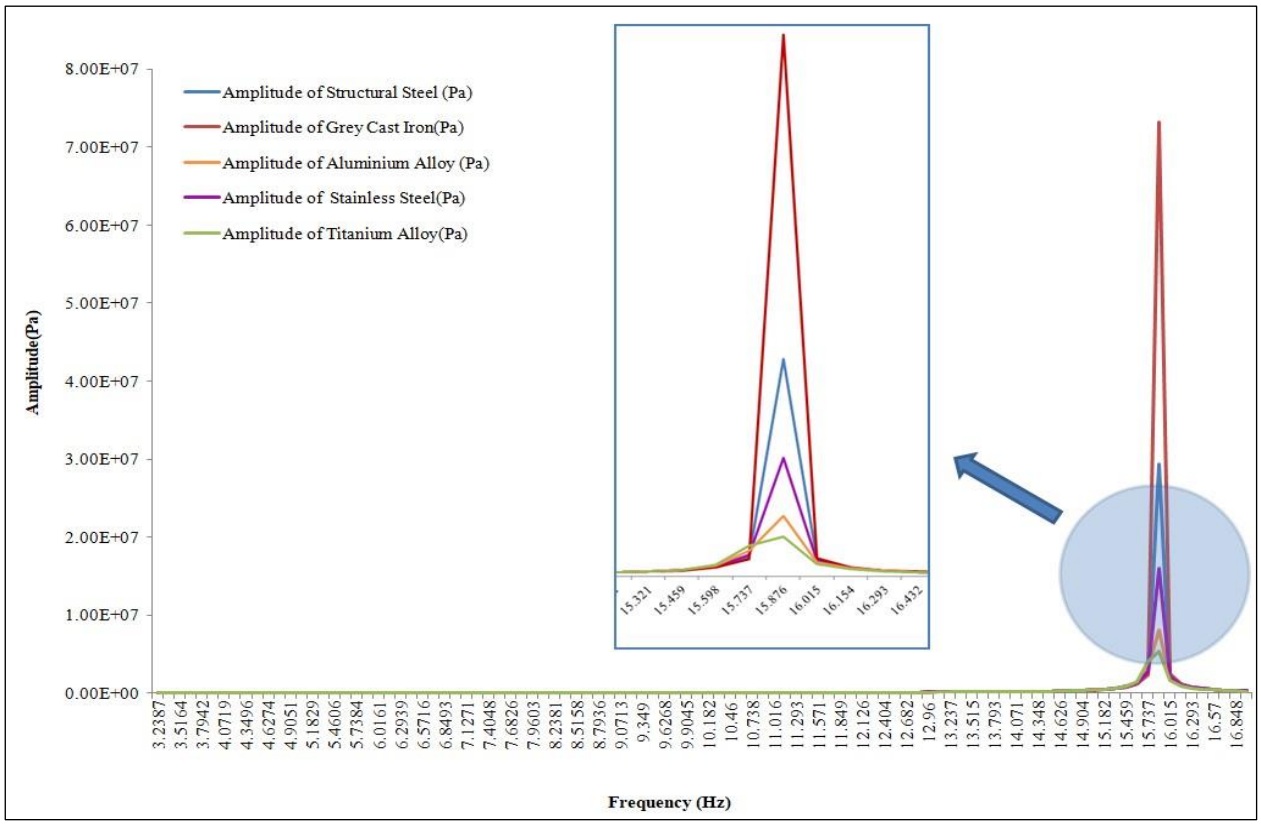


Figure 7: Variation of Amplitude with Frequency in different materials

Experimental Validation and Literature Comparison

To enhance confidence in the simulation results, a targeted comparison between our FEA outcomes and experimental/empirical findings reported in recent literature was performed. Direct experimental testing on the current chassis prototype was not feasible at the time of revision; however, similar structural evaluations of EV chassis components provide meaningful benchmarks. For example, Zhang et al. (2023) conducted bending and torsion tests on a medium-duty EV frame, reporting natural frequency ranges and displacement magnitudes

ISSN 2278-2540 | DOI: 10.51583/IJLTEMAS | Volume XIV, Issue XII, December 2025

under comparable loading conditions. This experimental modal analysis indicated first three bending mode frequencies between 25–35 Hz, which align with the trends observed in our harmonic analysis (Mode 1 ≈ 28 Hz, Mode 2 ≈ 32 Hz) [13]. Likewise, Lee and Kumar (2022) documented stress distribution patterns in automotive lightweight frames under dynamic load that closely resemble the stress contours in our ANSYS simulations [14].

While not direct one-to-one experimental validation, these literature-driven comparisons corroborate the overall structural behaviour predicted by our model, particularly for modal responses and stress localization trends. Incorporating full-scale experimental verification is planned in follow-up work.

CONCLUSIONS

The present work evaluated the variation in the effect of various pressure loads and vibrations on chassis made up of different materials to identify the advantages of certain materials over other materials used in the study.

From the results of static structural analysis, it is concluded that, with increasing of pressure by 19%, the average percentage increase in the stress, strain and total deformation is observed to lie in the range of 18 to 20%. Also, it was inferred that, in terms of response to increase in load, Structural Steel is the most preferred material for construction of chassis, since out of all materials analysed, it had the minimum equivalent stress, equivalent strain and total deformation.

Further, it is observed that, though Grey Cast Iron had the lowest modal frequency, it shows the highest spike in amplitude during harmonic analysis compared to other materials used in the study.

This study enables the engineers to identify stress, strain, deformation, and modal response of the chassis for improved durability and passenger protection. The literature review has shown the revolutionized versatility and accuracy in the way we approach structural analysis and optimization, ultimately leading to safer, more efficient, and innovative products and structures. As technology continues to advance, FEM analysis remains a cornerstone of modern engineering practices, driving progress and innovation across numerous disciplines.

Conflict of Interest

The author declares that there are no conflicts of interest regarding the publication of this manuscript.

Funding

The author received no financial support for the research, authorship, or publication of this article.

Data Availability

This research is not supported by any open-source, extensive or specific dataset. However, some considerations and parameters used in this analysis and research are available from the author upon reasonable request.

REFERENCES

1. D. Sammangi, L. Kayathi, J. Kanaparthi, R. Kutcherlapati, and R. Sarimalla, “A study on structural behaviour and vibrational characteristics of a go-kart chassis,” E3S Web Conf., vol. 391, 2023.
2. D. Karale, S. Thakare, V. Khambalkar, M. Desmukh, and R. Walke, “Design and Analysis of Battery Electric Vehicle Sprayer Chassis by Using Finite Element Method,” International Journal of Innovations in Engineering and Science, vol. 5. 2020.
3. N. Saini, M. Panwar, A. Maurya, and P. A. Singh, “Design and Vibration Analysis of Go-kart Chassis,” Int. J. Appl. Eng. Res., vol. 14, pp. 53–57, 2019.
4. C. Bulgu, “Design a Space Frame Chassis for Light Weight Automobiles for Improved Safety and Reliability,” Mech. Eng. Res., vol. 9, p. 36, 2020.
5. H. H. Rathod, S. Kumar, and V. Goel, “A Review on Analysis and Design of Vehicle Chassis and its Materials,” 2018.

ISSN 2278-2540 | DOI: 10.51583/IJLTEMAS | Volume XIV, Issue XII, December 2025

6. D. Sharma and Y. D. Vora, "Design and Analysis of Heavy Duty Vehicle (Trailer) Chassis through FEM Software," 2017.
7. K. Subburathinam and V. S N, "Finite Element Analysis of Automotive Chassis with Various Cross-Section," 2016.
8. H. Jaiswal, A. Kumar, A. Anand, and P. P. Patil, "Free Vibration Mode Shape Analysis of Space Frame Chassis of a Sports Car based on FEA," 2014.
9. B. E. Gajbhiye, S. V Bhalerao, and K. B. Salve, "Study of Vibrational Effects on Heavy Duty Truck Chassis with the Help of Model Analysis Using Fem Technique," Int. J. Eng. Res. Technol., vol. 2, 2013.
10. S. Rosbi, H. Desa, K. Saifullah, M. Abdul Rahman, K. Salleh, and I. Ibrahim, A study of Vibration Analysis for Gearbox Casing Using Finite Element Analysis. 2009.
11. R. Rahman and H. Teo, "Statics and dynamics structural analysis of a 4.5 ton truck chassis," Jan. 2007.
12. Ansys Inc, Academic Research, Release 16. 2014.
13. Zhang, L., et al. (2023). Experimental Modal Analysis of EV Chassis Structures, Journal of Automotive Engineering.
14. Lee, S., & Kumar, P. (2022). Dynamic Stress Evaluation of Lightweight Automotive Frames, International Journal of Vehicle Design.
15. L. Scurtu and D. Moldovanu, "Conceptual design of an electric vehicle chassis using topology optimization method," IOP Conf. Ser.: Mater. Sci. Eng., vol. 1311, 012022, 2024.
16. O. Zamzam, A. A. Abdelaziz, T. Elnady, and A. Ayman, "Structural analysis of an electric vehicle chassis using finite element analysis," J. Eng. Adv. Technol. Sustain. Appl., vol. 1, pp. 48–54, 2025.
17. K. Wang, P. Wang, Y. Lin, and X. Wang, "Modal analysis and optimization of electric vehicle chassis structure," World Scientific Research J., vol. 11, no. 9, pp. 11–19, 2025.