Journal of
Experimental & Applied Mechanics

https://journals.stmjournals.com/joeam

ISSN: 2230-9845 (Online)
ISSN: 2321-516X (Print)
Volume 16, Issue 2, 2025
May–August
DOI (Journal): 10.37591/JoEAM

Research JoEAM

Design and Structural Evaluation of a Dual-Axle Trailer Frame with a Focus on Increasing Strength While Aiming to Reduce Overall Weight

Lalam Srinu[1,]*, Bazani Shaik[2]

Abstract

The primary objective of this project is to evaluate various cross-sections and materials for the structural frame. The frame includes elongated side components and a central longitudinal member, which is connected to several cross-supports. The design focuses on optimizing both the cross-support layout and the choice of materials, incorporating composites to enhance performance. Load conditions are also altered to achieve improved efficiency. Static simulations are performed using ANSYS, while CATIA is used for design modeling. Additionally, modal analysis is conducted to assess the trailer's dynamic behavior. Weight reduction is taken into account to further boost productivity. Ultimately, the most effective design is produced as a prototype using 3D printing techniques. Weight optimization is an integral part of the design process, with the goal of improving the trailer's fuel efficiency, ease of towing, and overall productivity without compromising structural integrity. The integration of lightweight composites plays a pivotal role in this regard. After finalizing the most effective configuration based on performance metrics and simulation results, a scaled prototype is fabricated using 3D printing techniques to validate the design's manufacturability and structural viability. This project aims to contribute to the development of high-performance trailer frames that meet the growing demands of the transportation and logistics industry.

Keywords: Automobile trailer, chassis frame, 2-axle frame, vibrational analysis, composite material, weight management

INTRODUCTION

A trailer is typically defined as an unpowered vehicle that is designed to be towed by a powered vehicle. Trailers are primarily used for transporting goods and come in various types, depending on their intended application. Over time, trailers have played a crucial role in the development of towable transport, and historically, many vehicles were designed specifically for towing. Today, trailers have evolved to include mobile homes and other specialized functions. Their sizes vary significantly, from single-axle models to modern trailers featuring six or even nine axles to accommodate heavier loads.

While some trailers are designed for personal or small business use and can be towed by standard vehicles equipped with a proper hitch, others are built as part of larger freight systems, such as semi-trailer trucks used in commercial cargo transport. Recreational and utility trailers, such as enclosed toy haulers and motorcycle trailers, can often be pulled by everyday vehicles like pickup trucks or vans without requiring a special license. Smaller trailer types, including bicycle trailers and open-air

***Author for Correspondence**
Lalam Srinu
E-mail: mech.sseducationals@gmail.com

[1]Student, Department of Mechanical Engineering, Ramachandra College of Engineering, Eluru, Andhra Pradesh, India
[2]Professor, Department of Mechanical Engineering Ramachandra College of Engineering, Eluru, Andhra Pradesh, India

Received Date: May 20, 2025
Accepted Date: June 20, 2025
Published Date: July 12, 2025

Citation: Lalam Srinu, Bazani Shaik. Design and Structural Evaluation of a Dual-Axle Trailer Frame with a Focus on Increasing Strength While Aiming to Reduce Overall Weight. Journal of Experimental & Applied Mechanics. 2025; 16(2): 9–17p.

motorcycle trailers, are accessible to even compact automobiles. Utility trailers and campers are available in single- and multi-axle formats, suited to various towing capacities.

LITERATURE REVIEW

Nandan *et al.* [1]

Investigations into how certain materials influence chassis performance under applied stress have drawn significant academic attention. The outcomes from such analyses suggest that further exploration into chassis systems is necessary to provide foundational data for the advancement and refinement of vehicle chassis engineering. A chassis primarily serves to bear the weight of connected systems and transported goods. In developing a heavy-duty truck frame, various considerations such as choice of construction material, structural integrity, rigidity, and mass were evaluated. This study examines the static and dynamic response characteristics of a ladder-type frame structure for a utility vehicle prototype. The frame was digitally constructed using Autodesk Fusion 360, which was also utilized for conducting the structural assessments. For mesh generation during simulation, the platform's automatic meshing functionality is employed.

Arimadla *et al.* [2]

Research has been carried out on vehicle frameworks, which predominantly consist of steel structures providing foundational support for the engine and body. The word "chassis" denotes the essential framework forming the backbone of a vehicle. This study aims to explore the design process and perform static structural evaluation of a heavy-duty chassis using different alloy compositions subjected to varying optimal load conditions. The design phase will involve modeling in Fusion 360 and CATIA V5, while structural assessments will be conducted using Ansys Workbench. The objective of the project is to develop an improved frame that contributes to overall weight reduction and enhances large vehicle efficiency under ideal operational scenarios.

Jha *et al.* [3]

A study was conducted using the dimensions and mass parameters of the Audi A8 as a baseline. For the powertrain and battery specifications, data from the Mahindra e2o electric vehicle, sharing comparable size, was referenced. The core aim of this research is to create a digital model and execute static structural analysis on the frame of a four-passenger automobile. Initially, a space frame structure was considered due to its high stiffness and minimal displacement under load. Subsequently, a ladder frame was adopted as the final design, being the most widely utilized chassis format in regions like Nepal and India. The deviation in displacement between the two configurations was minimal, recorded at approximately 0.3235 mm, with a notable mass reduction nearing 120 kg. Simulation tasks were executed using ANSYS software. Based on both computational results from ANSYS and theoretical assessments, the study concludes with the identification of the most appropriate material for chassis application.

Roslan Abd Rahman, et al., 2008 [4]

This study presents a stress evaluation of a heavy-duty truck frame. Stress assessment plays a vital role in fatigue analysis and in predicting component lifespan by identifying the location subjected to maximum stress levels. The findings indicate that the region experiencing peak stress is located around the chassis aperture interfacing with the bolt. At this critical location, the stress value reaches approximately 386.9 MPa. This area is considered the origin point for potential failure, as fatigue-related damage typically initiates from zones exhibiting the highest stress concentrations.

OBJECTIVES

- Designing a 3D model of tractor trolley Chassis in CATIA.
- Static and Vibrational analysis using ANSYS.
- Comparison of the static analysis and vibrational analysis results for C and I sections and mixed C and I sections of the Chassis.

- Analyzing the better materials in terms of traditional vs. alloy vs. composite material.
- Comparing the weights of the product.
- Suggesting the optimum cross section of the Chassis.
- Best output model sent to 3D printing for prototype model.

PROBLEM STATEMENT

As we see on Indian roads, the major mode of transportation in small villages and towns is trailers. These trailers differ based on the requirement of the usage. However, our analysis has revealed several practical problems, such as, the insufficient loading capacity and inadequate strength of the trailer chassis. Additionally, the profile shape of the trailers differs a lot. Since trailers are mainly used for commercial purposes, such as civil material transportation and for the goods, it is important to address these issues. So, to overcome this issue, a trailer needs to be developed using a better profile and with better with material, while also reducing the weight of the trailer.

Ansys on Original Model Using HSS-S420m Material Geometry

In ANSYS, importing geometry from external CAD software is a fundamental step in setting up a simulation. Engineers often create complex models in CAD tools such as SolidWorks, CATIA, NX, or AutoCAD, and then transfer these designs into ANSYS for finite element analysis (FEA). This process helps streamline the design and analysis workflow by avoiding the need to rebuild the geometry from scratch.

ANSYS Workbench supports a wide range of geometry formats including STEP, IGES, Parasolid, and native files from various CAD systems. When a geometry file is imported, it is brought into the *DesignModeler* or *SpaceClaim* environment within ANSYS, where it can be cleaned, modified, or simplified for meshing and analysis. These tools allow users to fix geometry errors, remove unnecessary details, or perform defeaturing to reduce computational load.

The imported geometry must be carefully reviewed for compatibility with simulation requirements. Common issues include small edges, gaps, or overlapping surfaces that can interfere with mesh generation. ANSYS provides tools to detect and repair such problems during the import process (Tables 1–8). Once the geometry is prepared, it is used to create a finite element mesh and apply boundary conditions, loads, and material properties. Maintaining the fidelity of the original design while adapting it for simulation is essential to ensure accurate analysis results. This import capability makes ANSYS a versatile platform that integrates well with the broader design and engineering ecosystem (Figures 1–7).

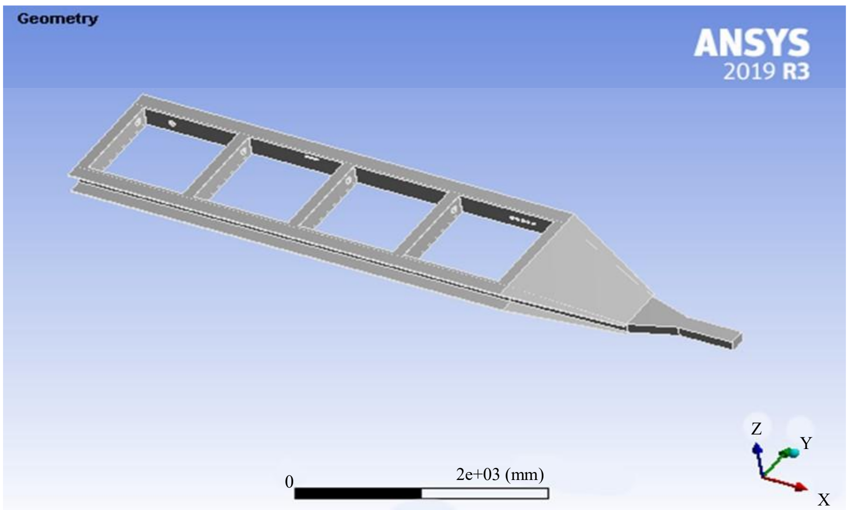

Figure 1. Input model of chassis in Ansys.

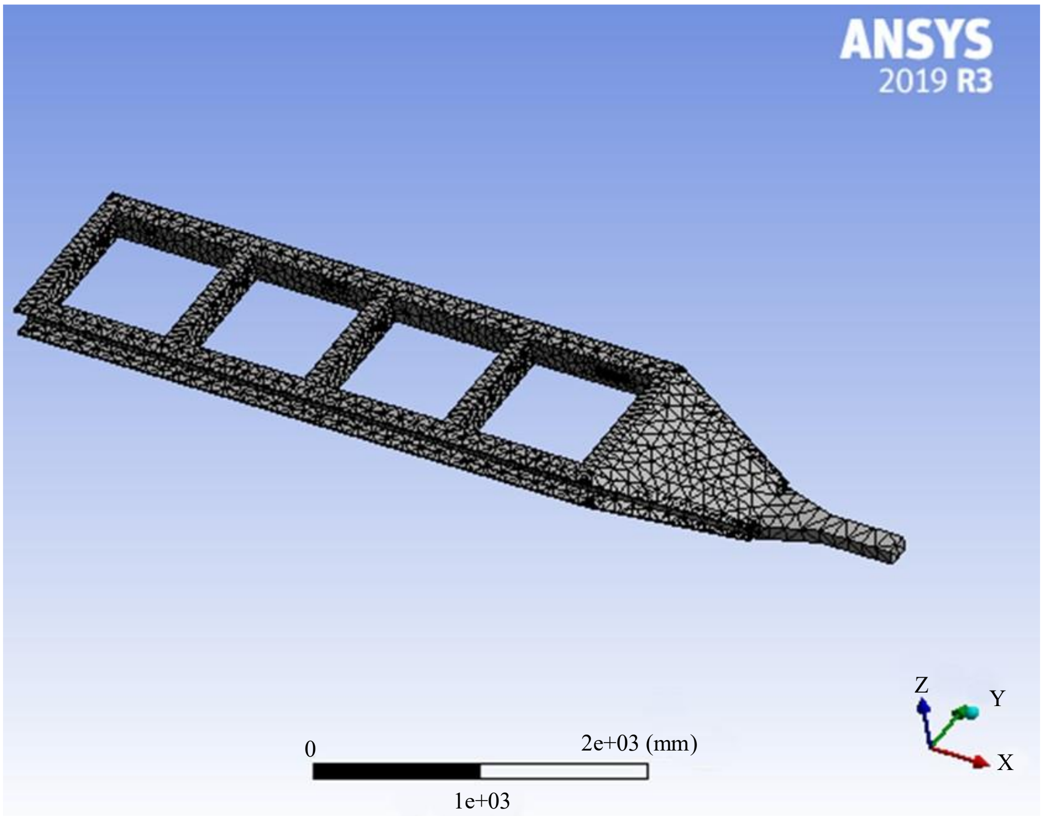

Figure 2. Mesh image of chassis system in Ansys.

Table 1. Comparison of stress results.

Models	HSS-S420 m	Magnesium Alloy	CFRP-5%
Original model	13.097	12.782	13.112
Model 1	32.09	18.062	19.003
Model 2	33.644	17.952	18.442

Table 2. Comparison of strain results.

Models	HSS-S420m	Magnesium Alloy	CFRP-5%
Original model	0.092294	0.0003761	0.0031821
Model 1	0.21805	0.00072846	0.0063516
Model 2	0.20534	0.00067021	0.0058143

Table 3. Comparison of deformation results.

Models	HSS-S420 m	Magnesium Alloy	CFRP-5%
Original model	235.9	0.9978	8.1519
Model 1	255.48	0.97601	7.934
Model 2	233.79	0.87111	7.0634

Table 4. Comparison of directional deformation results.

Models	HSS-S420m	Magnesium Alloy	CFRP-5%
Original model	21.3	0.090714	0.76213
Model 1	16.119	0.048954	0.39548
Model 2	18.541	0.052598	0.42652

Table 5. Comparison of modal analysis results: Original Model.

Models	Parameters	HSS-S420 m	Magnesium Alloy	CFRP-5%
Original model	Frequency	2.161	85.621	36.753
	Deformation - 1	3.1543	7.5392	9.2092
	Frequency	2.5505	96.666	41.324
	Deformation - 2	2.6854	7.7892	8.6617
	Frequency	2.6082	101.71	42.9
	Deformation - 3	3.0695	3.533	5.8682
	Frequency	2.9432	112.27	48.167
	Deformation - 4	4.11	11.968	14.568
	Frequency	3.2603	117.76	50.39
	Deformation - 5	5.8562	5.4113	6.5982

Table 6. Comparison of modal analysis results: Model-1.

Models	Parameters	HSS-S420m	Magnesium alloy	CFRP-5%
Original Model	Frequency	0.39011	75.308	32.275
	Deformation - 1	2.9795	7.7138	9.4298
	Frequency	1.4035	100.06	42.366
	Deformation - 2	2.3404	5.7302	7.0256
	Frequency	2.2153	100.71	42.719
	Deformation - 3	3.76	3.0839	3.8394
	Frequency	3.1482	104.35	44.768
	Deformation - 4	4.8222	8.9629	11.213
	Frequency	3.2211	131.89	56.644
	Deformation - 5	2.9169	11.371	13.926

Table 7. Comparison of modal analysis results: Model-2.

Models	Parameters	HSS-S420 m	Magnesium Alloy	CFRP-5%
Original Model	Frequency	1.2817	70.517	30.212
	Deformation - 1	2.9729	8.0663	9.8641
	Frequency	1.6023	100.11	42.368
	Deformation - 2	1.9545	5.7816	7.0895
	Frequency	2.079	101.51	43.043
	Deformation - 3	3.8839	3.1541	3.898
	Frequency	3.196	106.27	45.624
	Deformation - 4	2.8825	8.6449	10.686
	Frequency	3.223	132.56	56.938
	Deformation - 5	4.6826	11.61	14.217

Table 8. Comparison of weight (kg's) results.

Models	HSS-S420m	Magnesium Alloy	CFRP-5%
Original model	1534.7	356.13	238.08
Model 1	1573.2	365.07	244.06
Model 2	1555.4	360.94	241.29

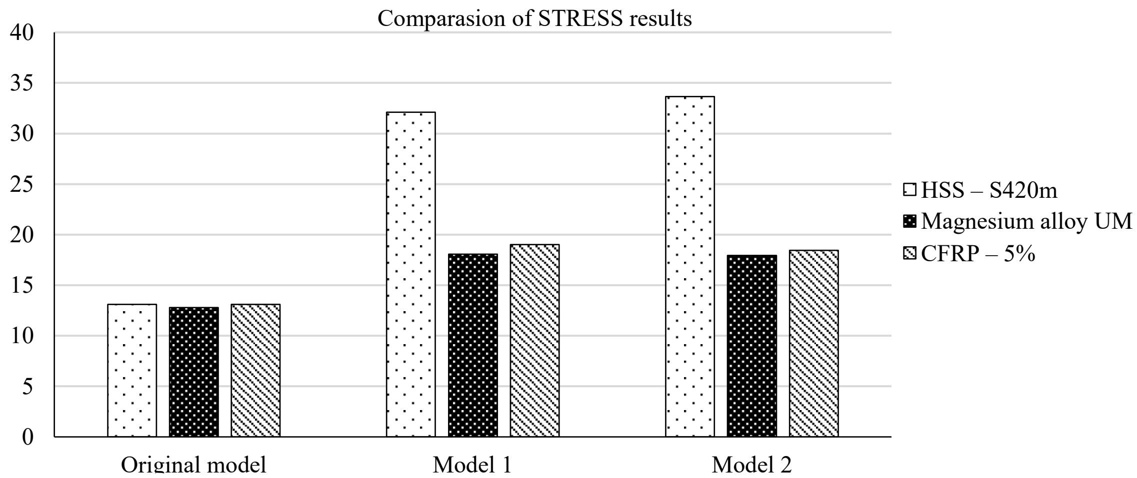

Figure 3. Comparison of Stress results.

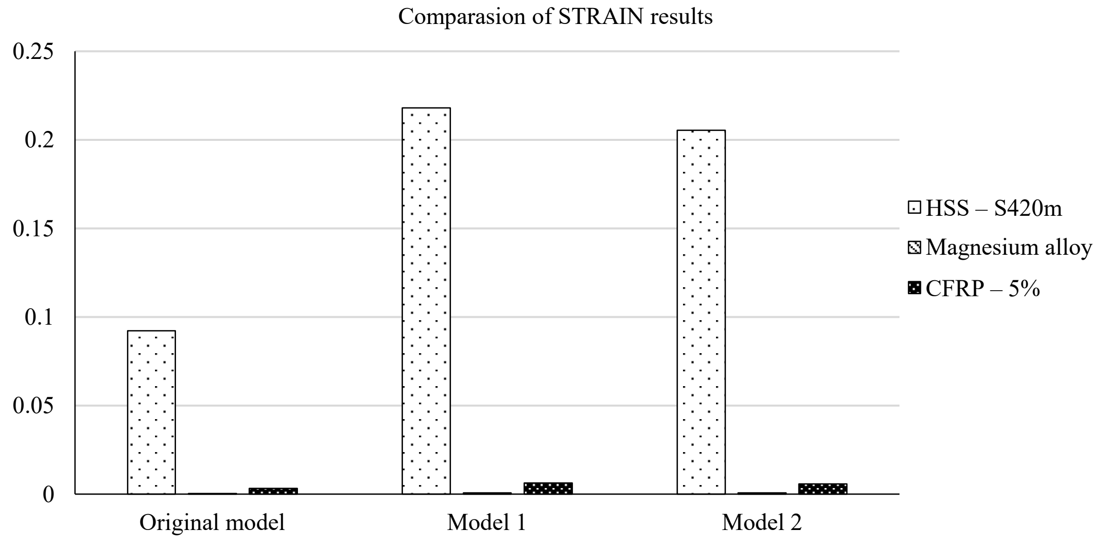

Figure 4. Comparison of Strain results.

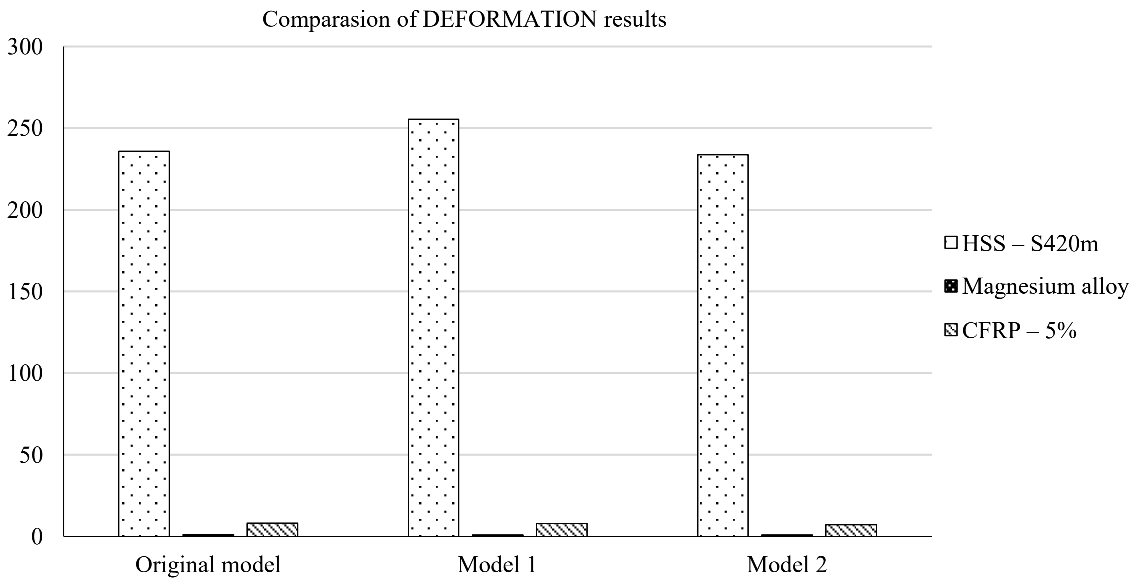

Figure 5. Comparison of deformation results.

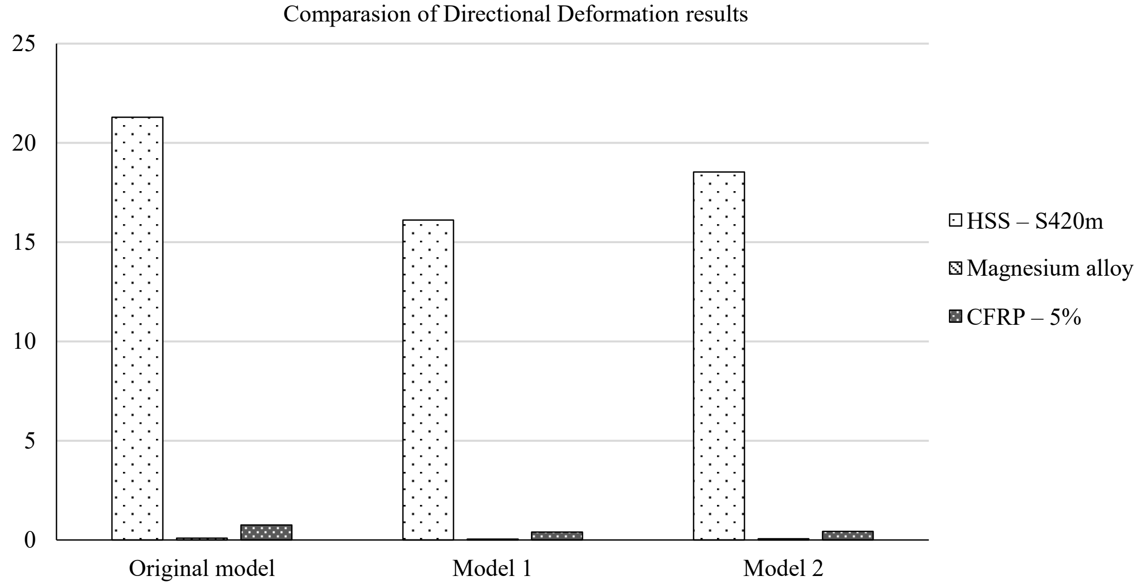

Figure 6. Comparison of directional deformation results.

Figure 7. Comparison of estimated weight of trailer results.

CONCLUSION

This study involves the evaluation of various cross-sectional geometries and material selections for the trailer frame. The structural configuration includes two longitudinal side members and a central main member, interconnected by multiple cross-members. Optimization efforts focus on refining the arrangement of cross-members and substituting conventional materials with advanced composites to enhance overall structural performance. Load conditions are varied to achieve an optimized structural layout under different operational scenarios.

The initial design modelling is executed using *CATIA,* while *ANSYS* is employed for static structural analysis. To further assess the dynamic behavior, *modal analysis* is conducted to identify the natural frequencies and mode shapes of the trailer frame. In addition to structural performance, weight reduction is evaluated as a key factor in enhancing productivity. The most effective design configuration, based on performance metrics and weight efficiency, is selected and fabricated as a prototype using 3D printing technology.

As we verify the results of the model here, we can observe that the output of stress is less for the new materials when compared with the traditional materials, but as we compare the models, there is a slight increase in the stress for the new suggested models.

When we compare the results between the strain, here also the two new materials obtained a better output when compared with the original material used. But when we compare with the deformation results, here the materials have performed the very best outputs when compared with the present material and even the modified models also obtained the better output results.

As we compare the weight of the product, the original design has no difference in the weight when we change the design of the model using the original material, but when we use the new materials as suggested with the change of designs, there is a lot of change in the weight of the product too.

Future Scope

1. *Crash and Impact Analysis:* Future studies could include *crashworthiness assessments* to evaluate energy absorption characteristics during impact scenarios. This is critical for improving safety standards in transport vehicles.
2. *Fatigue Life Prediction:* Incorporating *fatigue and lifecycle analysis* under varying road and load conditions will enable better understanding of long-term durability and maintenance scheduling.
3. *Integration with Smart Systems:* The chassis can be embedded with *IoT-based sensors* to monitor strain, stress, temperature, and vibration in real-time. This facilitates *predictive maintenance* and *health monitoring* of the structure.
4. *Electrification Compatibility:* With the rise of electric mobility, chassis designs can evolve to accommodate *battery packs, wiring harnesses,* and *cooling systems* needed for electric tractor trailers, ensuring structural and thermal compatibility.

REFERENCES

1. Nandan, Reuben George Oommen, Shashank SK, Irwin D'Souza, Swaroop Gannamani, Anil Kumar Gujjal. A Journal on “Design and Optimization of Vehicle Chassis for Harsh Road Conditions”. Int Res J Eng Technol. 2022 Jun; 09(06): 2725–2734.
2. Sravan Arimadla, Anil Kumar P, Jhansi P, Vivek M, Susheel PV. Design and Static Analysis of Heavy Vehicle Chassis with Different Alloy Materials at Different Optimum Load Conditions. Int Res J Eng Technol. 2022 Mar; 09(03): 1203–1209.
3. Murli Jha, Sujan Raut, Sanjip Pokhrel, Nitesh Kumar Yadav. Design and Analysis of Chassis of Four-Seater Car. Int J Res Appl Sci Eng Technol. 2022 Aug; 9(8): 1661–1668.
4. Rahman R, Tamin N, Kurdi O. Stress analysis of heavy duty truck chassis as a preliminary data for its fatigue life prediction using FEM. Jurnal Mekanikal. 2008; 26:76–85.
5. Xiong Xin, Tan Gangfeng, Guo Xuexun, Chen Menghuna. The study of Automobile Chassis Design and Development Using Digital Mock-Ups. 2011 International Conference on Electric Information and Control Engineering, Wuhan. 2011; 2814–2817.
6. Wang Gouging, Leo Shaojun, Zeng Yihui, Lingping C. Finite Element Analysis on the Chassis of a Tracked Test Vehicle. 2012 International Conference on Computer Distributed Control and Intelligent Environmental Monitoring, Zhangjiajie, China. 2012; 334–337.
7. Abdulkadir Yasar, Ali Birkin. Design, Analysis, and Optimization of Heavy Vehicle Chassis Using Finite Element Analysis. Int J Sci Technol Res. 2015; 1(6): 1–9. ISSN: 2422-8702.
8. Mooli Harish, Bhaskar K. Modeling and Analysis of a Heavy Vehicle Chassis Using Composites E-Glass Resin & S-2 Glass. Int J Recent Technol Eng. 2019 Mar; 7(6): 1–5. ISSN: 2177-3868.
9. Patil Kamlesh Y, Deore Eknath R. Stress Analysis of Ladder Chassis with Various Cross Sections. IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE). 2015; 12(4): 111–116.
10. Aayush Chugh, Rachit Ahuja, Sukriti Ranjan. Shape Optimization of Automobile Chassis. Int J Eng Res Technol. 2017; 6(2): 644–648.

11. Shrinidhi Rao, Ajay Bhattu. Dynamic Analysis and Design Modification of a Ladder Chassis Frame Using Finite Element Method. Amity Journal of Computational Sciences (AJCS). 2017; 1(2): 16–22.
12. Hari Kumar A, Deepanjali V. Design & Analysis of Automobile Chassis. Int J Eng Sci Innov Technol. 2016; 5(1): 187–196.
13. Shubham Agrawal, Arun Patel. A Review on Design and Analysis of Ladder Chassis. International Journal of Scientific Research and Engineering Trends (IJSRET). 2021; 7(5): 3012–3015.

