A REVIEW ON DESIGN AND ANALYSIS OF TRUCK CARGO BED USING DIFFERENT MATERIALS FOR WEIGHT REDUCTION

Mr. Rohan Chandrakar*, Mr. A. P. Dhawan#

*M Tech. Student, #Assistant Professor

School of Engineering & Technology, Sanjay Ghodawat University, Kolhapur

ABSTRACT: -

This review Paper examines a project aimed at reducing the weight of the truck cargo bed without compromising structural integrity. The study outlines the methodology followed, beginning with a detailed assessment of the existing cargo bed and the development. Literatures are reviewed for Structural performance under multiple loading scenarios using ANSYS. The review emphasizes the replacement of conventional heavy wooden and steel components with lightweight alternatives. A parametric optimization of essential structural members—such as horizontal supports, vertical elements, and floor panels studied for overall weight and material usage. This review underscores the effectiveness of material substitution and structural optimization in commercial vehicle design.

KEYWORDS: -

Truck cargo bed design; Weight reduction; Lightweight materials, Load-bearing performance; Vehicle lightweighting

INTRODUCTION: -

Trucks constitute a fundamental element of modern logistics and transportation systems, serving as the primary mode for moving goods across short and long distances. Within these vehicles, the cargo bed plays a critical role as it carries substantial loads and is subjected to significant mechanical stresses. The mass of the cargo bed directly influences payload capacity, fuel efficiency, and overall vehicle performance. Consequently, the selection of appropriate materials and structural designs has become increasingly important in efforts to reduce weight while maintaining adequate strength and durability.

Historically, wooden cargo beds were widely used due to their favourable combination of strength, durability, and low cost. However, rising concerns regarding fuel efficiency, sustainability, and regulatory pressures have encouraged the transition toward advanced lightweight materials. Contemporary alternatives—such as aluminium alloys, high-strength steel, and engineered structures like honeycomb panels—offer significant weight reductions without compromising stiffness or structural integrity.

The chassis of a vehicle serves as the main supporting framework, functioning analogously to a skeletal structure on which major components, including the engine and transmission system, are mounted. The vehicle body must support its own weight in addition to that of the cargo, with the chassis ensuring stability and load distribution under diverse operating conditions. Beginning in the decades following the 1970s oil crisis, vehicle size and weight increased notably due to the addition of safety, comfort, and entertainment features. Systems such as ABS, airbags, and reinforced body structures contributed to this trend, further intensifying the need for weight-saving innovations.

The global truck industry continues to expand in response to evolving economic and industrial demands. Sectors including logistics, agriculture, and manufacturing rely heavily on trucks for efficient transportation. Yet, in many regions, truck development and production remain dependent on imported technologies, which may not always meet local requirements for cost-effectiveness, driving performance, and operational efficiency. The cargo bed, typically constructed from wood or aluminium, constitutes a substantial portion of the vehicle's weight; thus, reducing its mass represents a cost-effective strategy for enhancing overall transport efficiency.

The primary functions of the cargo bed include:

1. **Carrying static and dynamic loads**,
2. **Supporting the upper vehicle body**, and
3. **Withstanding forces generated during transportation**, such as vibration, impact, and fatigue loads.

The review in this paper focuses on weight reduction of the cargo bed system for achieving significant weight reduction without compromising structural performance or manufacturability.

LITERATURE REVIEW: -

Pankaj Prakash Ande [1], in this research study the author has designed honeycomb safety structures, analyzed for safe road transport application point of view and the use of materials in these structures were addressed. Honeycomb structure was constructed using honeycomb core, adhesive and aluminum panels and Compression test was performed on it. CAD model of the honeycomb structure was designed using PTC Creo parametric design software. Following which, ANSYS Workbench - Static Structural simulations are carried out on different material grades of aluminum on honeycomb structures. These compression test and simulations are carried out with the aim to compare and understand the impact load capacity on these materials of honeycomb structures. Apart from that, the energy absorption ability of honeycomb structure with the normal block structure has been analyzed using ANSYS Workbench- Explicit Dynamics.

Cicek Karaoglu, N. Sefa Kuralay [2], in this study, stress analysis of a truck chassis with riveted joints was performed by using FEM. The commercial finite element package ANSYS version 5.3 was used for the solution of the problem. Determination of the stresses of a truck chassis before manufacturing is important due to the design improvement. In order to achieve a reduction in the magnitude of stress near the riveted joint of the chassis frame, side member thickness, connection plate thickness and connection plate length were varied. Numerical results showed that stresses on the side member can be reduced by increasing the side member thickness locally.

Y. Kiran Kumar Reddy et. al. [3], in this paper the author has studied the Honeycomb structures which offer high strength to weight ratio and act as energy absorbers in impact analysis. In this work authors have also studied the different types of honeycomb structure. The honeycomb structures are studied under various types of loading and boundary conditions. Initially, three different shapes of honeycomb structures are studied. These are considered as reference models. According to the results of initially built models, the hexagonal cell model has been considered as the best model. After those two types of hexagonal honeycomb structures are modeled, one is isotropic model and the other, orthotropic model. The isotropic material properties are taken from literature. Equivalent orthotropic material properties are calculated based on various available models in literature. Both models are analyzed under same loading and boundary conditions. Primary goal is to develop an equivalent model by replacing the actual model with orthotropic model. ANSYS programming is used for the analysis. The results are compared to find out the best performing equivalent model.

Teo Han Fui, RoslanAbd. Rahman [4], the authors have worked on the Statics and Dynamics, Structural Analysis of a 4.5 Ton Truck Chassis, they determined the dynamic characteristic, of the truck chassis, investigating the mounting locations of components on the truck chassis and observing the response of the truck chassis under static loading conditions. The maximum stress of the truck chassis is 490 MPa while the maximum translation is 33.6 mm. These values are acceptable as compared to the yield strength of the chassis material and the tolerance allowed for the chassis.

O Kurdi et al [5], works on the Stress Analysis of Heavy-Duty Truck Chassis Using Finite Element Method, he mainly focuses on the important steps in development of a new truck chassis is the prediction of fatigue life span and durability loading of the chassis frame. Fatigue study and life prediction on the chassis is necessary in order to verify the safety of this chassis during its operation. Stress analysis using Finite Element Method (FEM) can be used to locate the critical point which has the highest stress. This critical point is one of the factors that may cause the fatigue failure.

Karaoglu et al [6], in this paper the authors have investigated the stress analysis of a truck chassis with riveted joints using FEM. Numerical results showed that stresses on the side member can be reduced by increasing the side member thickness locally. Fermer et al investigated the fatigue life of Volvo S80 Bi-Fuel using MSC Fatigue.

Shaik Nazeer et al [7], in this literature the author has studied the Honeycomb structures. They are natural or man-made structures that have the geometry of a honeycomb to allow the minimization of the amount of used material to reach minimal weight and minimal material cost. There are different types of honeycomb core structures like square, hexagonal, pentagonal, tetrahedral, pyramidal etc. In this project author is comparing the structural analysis for square and hexagonal honeycomb structures and thermal analysis of square and hexagonal honeycomb structures. Structural analysis is the determination of the effects of loads on physical structure. To perform an accurate analysis, information as structural loads, geometry, support conditions, and materials properties are important. The results of analysis include deformation, stresses and displacements. This

information is then compared to criteria that indicate the conditions of failure. Thermal analysis calculates the temperature distribution and related thermal quantities in the system or component.

M. Ravi Chandra, S. Sreenivasulu, Syed Altaf Hussain [8], in this paper author have designed and analyzed the vehicle chassis of a TATA 2515EX vehicle. In this work three different composite heavy vehicle chassis have been modeled by considering three different cross-sections namely C, I and Box type cross sections. For validation the design is done by applying the vertical loads acting on the horizontal different cross sections. Software is used in this work PRO – E 5.0 for modeling, ANSYS 12.0 for analysis.

Indu Gadagottu, M V Mallikarjun [9], in this paper the authors have described the design and analysis of heavy vehicle chassis by using honeycomb structure. In this paper traditional materials are replaced with composite materials [S-glass epoxy and E-glass epoxy] using reverse engineering method. Existing model, modified model, honey comb model are compared and studied. For validation the design is done by applying a single vertical load acting on the chassis. And then Structural and, fatigue analysis will be carried out on three models to all materials and select the best material Impact analysis can also be done for the selection material in all models Software's used in this work solid works for modelling ANSYS 14.5 for analysis.

Ch. P. V. S. SAI Trinadh et al [10], in current paper authors have completed static and dynamic analysis of both C and I sections of tractor trolley chassis is performed using Finite Element Analysis. The 3D finite element model of the chassis is achieved through NX CAD software. Later, the finite element model of the chassis is imported to ANSYS. The Finite Element Analysis is carried out in commercial finite element software named ANSYS. By conducting the static structural analysis of the finite element model of the chassis in ANSYS, deflection, Von Mises stress, Normal stress, shear stresses are found out for both C and I sections. Dynamic analysis is also performed on the finite element model of the chassis with C and I section and natural frequencies are found out. Comparison of the results of the static and dynamic analysis for both C and I sections of the chassis is presented. From this comparison, the optimum shape of cross section of chassis is suggested.

Siddhesh Chavan et al [11], in this paper authors have worked on the weight reduction of the cargo bed by using the sandwich materials. The load carrying parts like vertical and horizontal supports and cargo area will be optimized. The material saving of overall system was achieved upto 57.26 %. Material saved in system is of 1677 Kg. The use of honeycomb structure material decreases the weight of the cargo bed upto 51.61 %. As the dead weight of the cargo bed decreases, the fuel consumption of the truck will decrease.

Arun S. Shinde et al [12], This paper aims to perform the stress analysis of an actual Tractor trolley chassis structure consisting of C section beams design application of 6 tons. The material of structure is Mild Steel with 248 MPa of yield strength. The scope of this study concern on structural design of the C section beams for information and data gathering, which will be used for further design improvement. Finite element modelling (FEM) and analysis are performed using a modelling software i.e., Pro-E Wildfire 5. Firstly, a 3-D model of Tractor trolley chassis based on design from workshop is created by using Pro-E.

P. H. Meshram et al [13], In this study authors have worked on redesign of existing trolley axle considering the static and dynamic load conditions. A CAD model is prepared using CATIA V5 as a tool. Minimum cross section for the axle is calculated which resulted in the 24.8 % reduction in the weight of the axle. The axle dimensions are redesigned to 75 X 75 mm which is smaller than the old axle. The design is optimized based on the manufacturing cost of the axle. The failure analysis is performed on the axle of trolley used in agricultural area. These results provide a technical basis to prevent future damage to the location axle.

Mr. Suraj S. Jadhav et al [14], in this paper the authors have worked on existing rear axle shaft which is used in tractor trolley. They studied that existing axle has greater factor of safety so unwantedly heavy axle is used for trolley in existing condition which increases the weight of axle as well as cost of axle. The newly designed axle with different cross section and different material shows that can maximally reduce the 31.95 % weight as compared to the existing axle. Also reduces the cost of trolley axle as the weight of the axle reduces. The cost of axle reduced approximately up to Rs. 1696 per axle and the deformations as well as stresses developed in new designed axle are within limits.

Vishal M Bidve et al [15], In this paper the author has work on the survey of sugarcane tractor trolley, during survey it is found that most of axles having bending, deformation, weight, strength problems, and common problem are bending. Further analysis is done on axle by finite element analysis using Ansys for checking bending, deformations, and weight optimization by using various materials like MS, SAE1020, Inconel, etc. Bending problem of trolley axle is reduced by using SAE1020 material axle, because based on the strength and

rigidity of material. Deflections induced in SAE 1020 material axle is less as compared to MS axle hence failure of trolley axle minimizes. On reducing 2mm thickness of axle weight of SAE 1020 axle get optimized. On making fine slots instead of reducing thickness weight will further optimizes, and stress concentration reduces due to abrupt change in area by slots. This method gives the optimum results, but it slightly increases the machining cost compared to previous method, but it reduces the running cost. As bending stress for SAE1020 trolley axle reduced, ultimately the strength increases.

Roslan Abd Rahman et al [16], this paper presents the stress analysis of heavy-duty truck chassis. The stress analysis is important in fatigue study and life prediction of components to determine the critical point which has the highest stress. The analysis was done for a truck model by utilizing a commercial finite element packaged ABAQUS. The model has a length of 12.35 m and width of 2.45 m. The material of chassis is ASTM Low Alloy Steel A 710 C (Class 3) with 552 MPa of yield strength and 620 MPa of tensile strength. The result shows that the critical point of stress occurred at the opening of chassis which is in contact with the bolt. The stress magnitude of critical point is 386.9 MPa. This critical point is an initial to probable failure since fatigue failure started from the highest stress point.

Mohd Azizi Muhammad Nor et al [17], in this paper authors had aimed to model, simulate, and perform the stress analysis of an actual low loader structure consisting of I-beams design application of 35 tonne trailer designed in-house by Sumai Engineering Sdn. Bhd, (SESB). The material of structure is Low Alloy Steel A 710 C (Class 3) with 552 MPa of yield strength and 620 MPa of tensile strength. The scope of this study concern on structural design of the I-beams for info and data gathering, which will be used for further design improvement. Finite element modelling (FEM), simulations and analysis are performed using modelling software i.e., CATIA V5R18.Firstly, a 3-D model of low loader based on design from SESB is created by using CATIA. Stress and displacement contour are later constructed, and the maximum deflection and stress are determined by performing stress analysis. Computed results are then compared to analytical calculation, where it is found that the location of maximum deflection agrees well with theoretical approximation but varies on the magnitude aspect. Safety factor for the low loader structure has also been calculated. In the end, the current study is important for further improvement of the current low loader chassis design.

P. Ravi et al [18], In this paper authors have carried out Design and structural analysis of heavy vehicle chassis (TATA 2518TC) by considering three different cross-sections, namely C, I, and hollow rectangular (Box) type cross sections subjected to the same conditions, made of four different materials viz., Steel ST37, boron/Epoxy, Carbon/Epoxy, E-glass/Epoxy composites. A three-dimensional solid Model is generated using CATIA and analysed in ANSYS 15.0. The numerical results obtained are validated with analytical values considering the stress distribution and deformation. The results are then compared to finalize the best among three cross sections. Boron/epoxy with Box section has the minimum deflection and Stress is observed.

Abhishek Singh et al [19], This paper describes Structural analysis & optimization of vehicle chassis with constraints of maximum shear stress and deflection of chassis under maximum load. The authors have taken higher strength as the main issue, so the dimensions of an existing vehicle chassis of a TATA LP 912 Diesel BS4 bus is taken for analysis with materials namely Steel alloy (Austenitic) subjected to the same load. The four different vehicle chassis have been modelled by considering four different cross-sections. Namely C, I, Rectangular Box (Hollow) and Rectangular Box (Intermediate) type cross sections. For validation the design is done by applying the vertical loads acting on the horizontal different cross sections.

Monika S. Agrawal [20], in this paper authors have done the study to produce results to rectify problems associated with structures of a commercial vehicle such as strength, stiffness and fatigue properties along with stress, bending moment and vibrations. This can be achieved by static and dynamic analysis, combining existing theoretical knowledge and advanced analytical methods. The static and modular analysis is done in this paper and results are proved.

Ramesh kumar S. et al [21], in this paper the analysis of chassis is done by using the ansysis software. The modelling is done by using Catia software. The stress analysis and fatigue analysis are the main analysis carried out on the chassis using ANSYS software. This is particularly done for the identification of the weak points and the life of the chassis. Further even more research must be continued to carry out in the nearby. All the design aspects are analyzed carefully based on the required parameters. The analysis clearly suggest that the Structural Steel material is the best material for the chassis manufacturing based on the design, when compared with Aluminium Alloy and Titanium Alloy.

B. Narayana Swamy et al [22], In this paper authors have tried to replace the steel chassis with a composite material, the composite materials considered for the analysis work E-glass epoxy and S-glass epoxy subjected to the similar conditions as that of steel chassis. The modelling of chassis has been carried out by Pro-E Software V5.0 Package and static analysis has been carried out by ANSYS V 11.0 Software.

V. Vamsi Krishnam Raju et al [23], in this paper authors have completed Modeling and Structural Analysis of Ladder Type Heavy Vehicle Frame of a TATA 1109 EX2 vehicle. The analysis is done in this work with three different composite materials namely Carbon/Epoxy, E-glass/Epoxy and S-glass/Epoxy subjected to the same pressure as that of a steel frame.

CONCLUSION

The review highlights the growing importance of lightweight cargo bed structures in modern truck design, driven by the need for improved fuel efficiency, higher payload capacity, and compliance with environmental and regulatory standards. Traditional wooden cargo beds, while historically effective, are no longer adequate for meeting the performance and sustainability requirements of today's transportation sector. Advances in materials engineering—particularly the use of aluminium alloys, high-strength steel, and honeycomb composite structures—offer substantial benefits in terms of weight reduction without compromising structural integrity.
Overall, the review underscores that adopting lightweight materials and optimized structural designs is essential for modern truck development. Continued research in material innovation, structural optimization, and integration of simulation tools will play a critical role in advancing cargo bed design and meeting future demands of the logistics and transportation industries.

REFERENCE

[1]. Pankaj Prakash Ande, "Honeycomb Safety Structure: Design, Analysis and Applications in Safe Road Transport", International Journal of Science and Research (IJSR), ISSN: 2319-7064.
[2]. Cicek Karaoglu, N. Sefa Kuralay, "Stress analysis of a truck chassis with riveted joint", Journal of Finite Elements in Analysis and Design 38 (2002), Elsevier Science, page no- 1115–1130.
[3]. Y. Kiran Kumar Reddy, N. Venkatramana Reddy, "Design and Analysis of Sandwich Honey Comb Structures", International Journal of Research in Engineering, Science and Management Volume-2, Issue-1, January-2019.
[4]. Teo Han Fui, RoslanAbd. Rahman, Faculty of Mechanical Engineering, University Teknologi Malaysia, "STATICS and Dynamics Structural Analysis of a 4.5 Ton Truck Chassis" December, 2007
[5]. O Kurdi, R Abd- Rahman, M N Tamin, Faculty of Mechanical Engineering University Teknologi Malaysia 81310 UTM Skudai, Johor, Stress Analysis Of Heavy Duty Truck Chassis Using Finite Element Method
[6]. Karaoglu, C. and Kuralay, N.S., 2000, "Stress Analysis of a Truck Chassis with Riveted Joints", Elsevier Science Publishers B.V. Amsterdam, the Netherlands, Vol. 38, 1115-1130.
[7]. Shaik Nazeer, Shaik Allabakshu, "Design and Analysis of Honey Comb Structures with Different Cases", International Journal of Engineering Development and Research Volume 3, Issue 4, ISSN: 2321-9939.
[8]. M. Ravi Chandra, S. Sreenivasulu, Syed Altaf Hussain,,,, Modeling and Structural analysis of heavy vehicle chassis made of polymeric composite material by three different cross sections, "International Journal of Modern Engineering Research (IJMER) Vol.2, Issue.4, July-Aug. 2012 pp-2594-2600 ISSN: 2249-6645.
[9]. Indu Gadagottu and M V Mallikarjun, structural analysis of heavy vehicle Chassis using honey comb structure, International journal of Mechanical Engineering & Robotics Research 2015, ISSN 2278 – 014 Vol. 4, No. 1, January 2015
[10]. Ch. P. V. S. SAI Trinadh, Srinivas Pavan kumar Adivi, "Design and Analysis of Tractor Trolley Chassis Using Finite Element Method", May 2021, DOI:10.13140/RG.2.2.17539.58404
[11]. Siddhesh Chavan, Dr. S. L. Ghodake, "Design and Development of Cargo Bed of Truck Using Honeycomb Sandwich Material", International Research Journal of Engineering and Technology (IRJET), Volume: 07 Issue: 06 | June 2020, e-ISSN: 2395-0056
[12]. Arun S. Shinde, Prof. J. Y. Mule, "Stress Analysis of Tractor Trolley Chassis with Effect of various Thickness and Design Optimization for Weight Reduction", IJARIIE-ISSN(O)-2395-4396, Vol-2 Issue-2 2016

[13]. P. H. Meshram, R. M. Thakare and R. U. Hedau, "Modeling and Analysis of Tractor Trolley Axle using Finite Element Analysis", International Journal of Research in Biosciences, Agriculture and Technology, Vol. II, Issue (7), Nov 2015: 462-467
[14]. Mr. Suraj S. Jadhav, Prof. E. N. Aitavade, Mr. Azhar S. Mulla, "Design and Analysis of Tractor Trolley Axle Using FEA", International Journal of Engineering Sciences & Research Technology, ISSN: 2277-9655, DOI: 10.5281/zenodo.3256081
[15]. Vishal M Bidve, Prof. Swami M. C, "Analysis & Optimization of Sugarcane Trolley Axle Using FEA and Experimental", International Journal of Engineering Development and Research, Volume 4, Issue 4 | ISSN: 2321-9939.
[16]. Roslan Abd Rahman, Mohd Nasir Tamin, Ojo Kurdi, "Stress Analysis of Heavy-Duty Truck Chassis as A Preliminary Data for Its Fatigue Life Prediction Using Fem", Journal Mechanical December 2008, No. 26, 76 – 85
[17]. Mohd Azizi Muhammad Nor, Helmi Rashid, Wan Mohd Faizul Wan Mahyuddin," Stress Analysis of a Low Loader Chassis", International Symposium on Robotics and Intelligent Sensors 2012 (IRIS 2012), Procedia Engineering 41 (2012) 995 – 1001
[18]. P. Ravi, M. Kumara swamy, "Topology Optimization Design and Finite Element Analysis of Heavy-Duty Truck Chassis Frame", International Journal of Advance Engineering and Research Development Volume 4, Issue 10, October -2017
[19]. Abhishek Singh, Vishal Soni, Aditya Singh, "Structural Analysis of Ladder Chassis for Higher Strength", International Journal of Emerging Technology and Advanced Engineering Website: www.ijetae.com (ISSN 2250-2459, ISO 9001:2008 Certified Journal, Volume 4, Issue 2, February 2014)
[20]. Monika S. Agrawal, "Design and Analysis of Truck Chassis Frame", IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e- ISSN: 2278-1684, p-ISSN: 2320-334X PP. 76-85
[21]. Ramesh kumar. S, Dhandapani. N. V, Parthiban. S, Kamalraj. D, Meganathan. S, Muthuraja S., "Design and Analysis of Automotive Chassis Frame Using Finite Element Method", International Journal of Pure and Applied Mathematics Volume 118 No. 20 2018, 961-972, ISSN: 1311-8080 (printed version); ISSN: 1314-3395 (on-line version)
[22]. B. Narayana Swamy, C. Lakshmaiah, Dr. K. Tirupati Reddy. " Modeling and Analysis of Light Vehicle Chassis Made of Composite Material", International Journal of Engineering Science and Computing, March 2017
[23]. V. Vamsi Krishnam Raju, B. Durga Prasad, M. Balaramakrishna, Y. Srinivas, "Modeling and Structural Analysis of Ladder Type Heavy Vehicle Frame" International Journal of Modern