Research Article
Volume 14:1, 2025

Advances in Robotics & Automation

ISSN: 2168-9695 Open Access

Design and Analysis of Manual Pallet Stacker by Using FEM Method

Rohit Pandey[1*] **and Ashish Kumar Shrivastava**[2]

[1]*Department of Mechanical Engineering, ATLAS SkillTech University, Mumbai, India*

[2]*Department of Mechanical Engineering, Sagar Institute of Science, Technology and Research (SISTec-R), Bhopal, India*

Abstract

The design and analysis of a manual pallet stacker play a crucial role in enhancing material handling efficiency in industries. This study focuses on the structural design and performance evaluation of a manual pallet stacker using the Finite Element Method (FEM). The primary objective is to optimize the pallet stacker's design to ensure its mechanical reliability and safety under various load conditions. The manual pallet stacker consists of key components such as the frame, lifting mechanism, forks, and wheels, all of which are subjected to complex stress, strain, and deformation during operation. FEM is employed to model the structural behavior of these components, simulating real-world operational scenarios including lifting, lowering, and horizontal movement. The analysis helps in identifying critical stress points, potential failure zones, and areas for design improvement. Results from the FEM analysis are used to refine the stacker design, ensuring enhanced performance, reduced material usage, and improved durability. The findings offer valuable insights for engineers and manufacturers, leading to the development of more efficient, cost-effective, and safe manual pallet stackers. This study demonstrates the potential of FEM in optimizing manual material handling equipment, contributing to advancements in industrial ergonomics and productivity.

Keywords: Manual pallet truck • Manufacturing • Pallet • Lifting • Static analysis • ANSYS • CATIA V5 R20

Introduction

Material handling refers to the development, storage, management and security of information, materials and products, from assembly, distribution measures and their use and disposal. MH is the craft and study of the movement, storage, stability and control of materials. Product stewardship means designing and manufacturing products at the lowest possible cost using legal equipment and hardware [1]. Product handling is the craft and study of shipping, moving, placing, handling, packaging and storing information: The added value (for customers) of short-term shipping bales (such as FedEx) is comparable to traditional mail handling. In any case, regular mail will be used next time, which is more important or compared to the additional cost of assistance [2,3]. The additional cost of keeping the product close to the bottle machine is the budget associated with using the machine to reduce the cost of keeping the product in the machine. The electric pallet stacker is a flimsy, profoundly adaptable lift that praises almost any basically indoor application. Adjusted like a customary forklift and without base legs, the counter-balanced electric stacker can find a way into restricted spaces [4]. Very strong and spending amicable, the Toyota counter-balanced stacker can help increase both your uptime and your main concern. The use of simulation models for structural analysis and motion behavior (CAE) is widely used in many industries for virtual product development. Increased investment costs, advanced user skills, complex application identification, and a significant number of powerful modeling tools are barriers to the use of CEA [5]. While accessibility and affordability are important factors in using CAE effectively, the focus is on technology. These determining factors include a smooth transition to intralogistics. VDI 2209 provides only general information about modeling techniques and does not provide information about modeling tools. Recommendations and rules for logistics modeling processing are related to material flow (throughput) modelling [6]. It has nothing to do with CAE or engineering. There are many mobile material handling equipment manufacturers and suppliers across the world. According to design standards, it is necessary to predict, analyse, and calculate the maximum allowable stress of the structure [7,8]. For this purpose, the FEM1 software package is widely used [9-10]. The importance of FEA2 lies in its ability to easily handle complex structures and provide insight into the

*Address for Correspondence: *Rohit Pandey, Department of Mechanical Engineering, ATLAS SkillTech University, Mumbai, India; E-mail: pandeyrohit1988@gmail.com*

Received: 04 November, 2023, Manuscript No. ARA-23-119241; **Editor assigned:** 07 November, 2023, PreQC No. ARA-23-119241 (PQ); **Reviewed:** 21 November, 2023, QC No. ARA-23-119241; **Revised:** 03 January, 2025, Manuscript No. ARA-23-119241 (R); **Published:** 13 January, 2025, DOI: 10.37421/2168-9695.2025.14.308

structural behaviour and reliability of the structure. Using FEA, engineers gain valuable information about weaknesses, potential errors, and poor design features (Figure 1) [11].

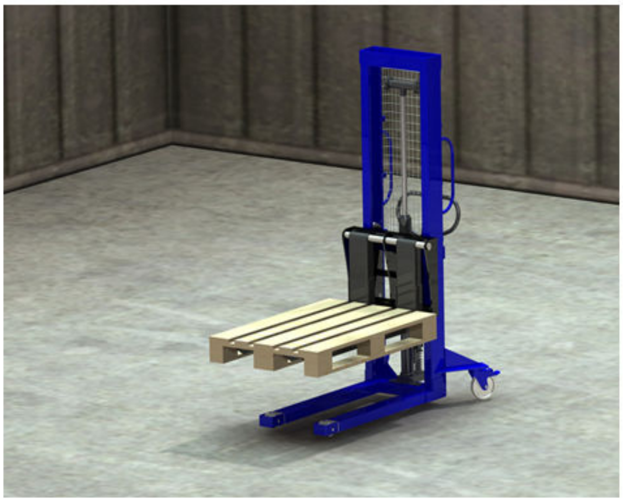

Figure 1. Manual pallet stacker.

Materials and Methods

Modeling and simulation

The packaging machine model was created in CATIA VER 5R 20 software, and the simulations were performed in the ANSYS software package. CATIA IGES model data is recorded and later imported into ANSYS modeling software. Consider a manufacturing company, such as a beverage company, that produces many products. Once a seller receives an order, the company packages and ships the various products to the seller using the most efficient packaging and shipping methods [12]. The factory's logistics system plans how to meet special requirements. The packaging configuration generated by the scheduler details how many pallets should be produced and how each product should be arranged on a specific pallet [13,14]. Packages are sorted in a pre-sorting stage using conveyors and stop systems. Packages are sorted so that each package is sent to the appropriate palletizing station [15]. A palletizing system consists of several robots that remove packages from conveyors and place them into one or more stacks. A particular warehouse system may consist of multiple palletizing systems. At the packaging stage, all products are placed on pallets. Storage and storage steps are usually separate processes [16]. However, some organizations also incorporate storage and retrieval heuristics into their plans. Common finite element analysis software, ANSYS Mechanical or ANSYS Multi physical, originally went by the name ANSYS (and is still widely used). Although ANSYS Inc. Perhaps the best known are ANSYS Mechanical and ANSYS Multi physical, which produce a variety of CAE products [17]. Structural loads include gravity, normal forces, and moments (playing conditions) applied to the slewing ring. The magnitude and direction of rotational support depends on operating conditions (idling, loading and unloading) and boom angle (boom swing and reach). ANSYS academic research, ANSYS academic, teaching advanced, beginning, etc. are the names of educational models in the industry [18,19]. These products are standalone, general-purpose finite element analysis tools that combine preprocessing tools (meshing and geometry), solvers, and post processing modules into a single Graphical User Interface (GUI). In this application, "ANSYS" refers to the FEA capabilities of products formerly known as ANSYS Inc. He was tested. To obtain the stress distribution, the geometry was discretized using the FEM ANSYS package. ANSYS Workbench is an interactive environment for modeling and solving all types of engineering and scientific problems using Partial Differential Equations (PDEs) [20]. The entire geometry was discretized using approximately 120,000 tetrahedral 2D elements (Figures 2-11). Solid modeling is used because of its simplicity and accuracy compared to shell modeling.

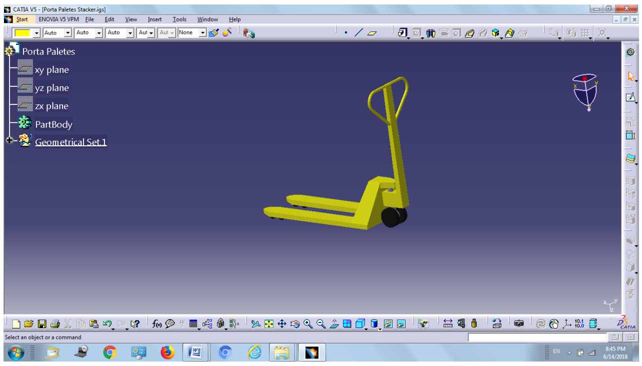

Figure 2. CATIA model of stacker.

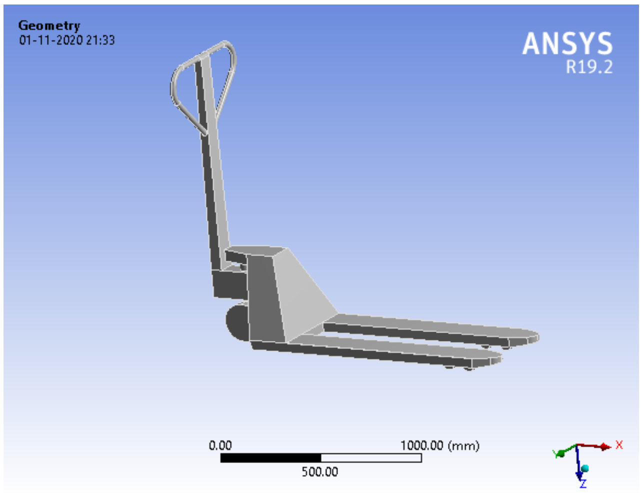

Figure 3. Import geometry in ANSYS.

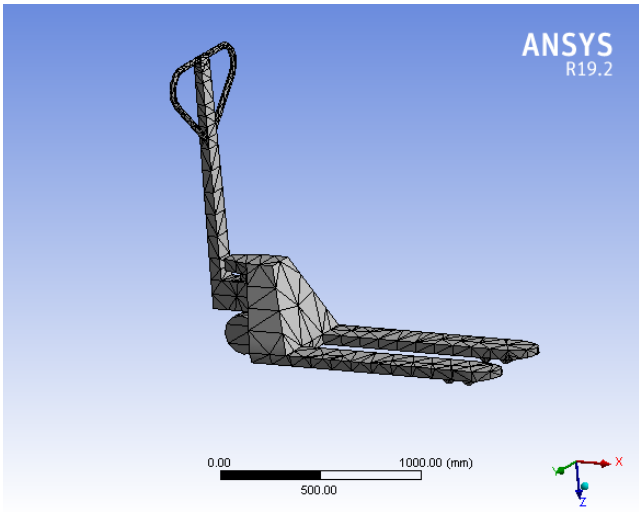

Figure 4. Geometry meshing.

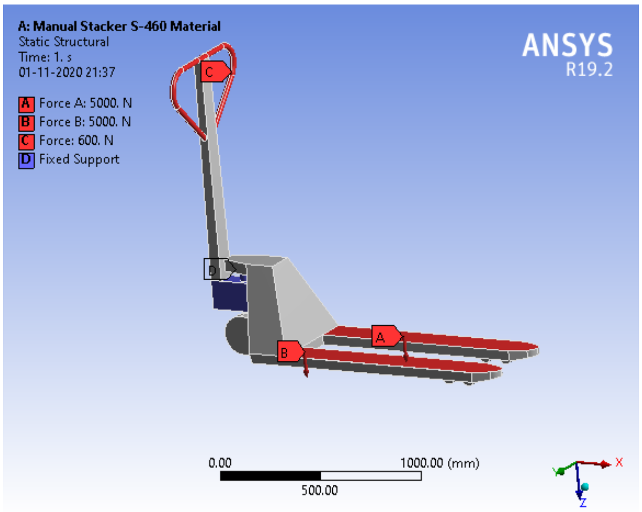

Figure 5. Boundary conditions (structural steel).

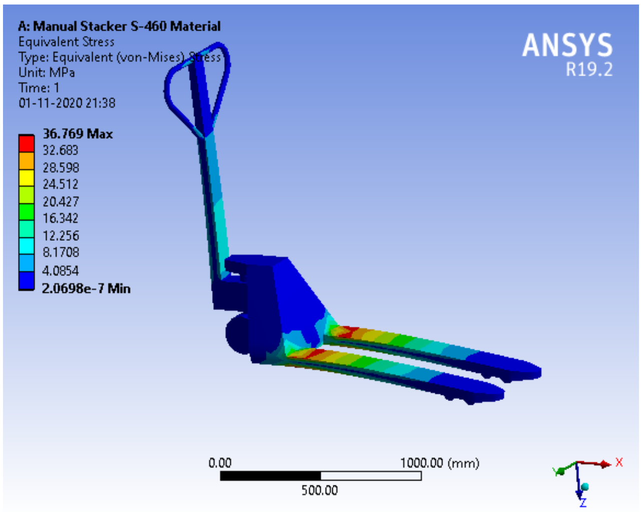

Figure 6. Equivalent stress (structural steel).

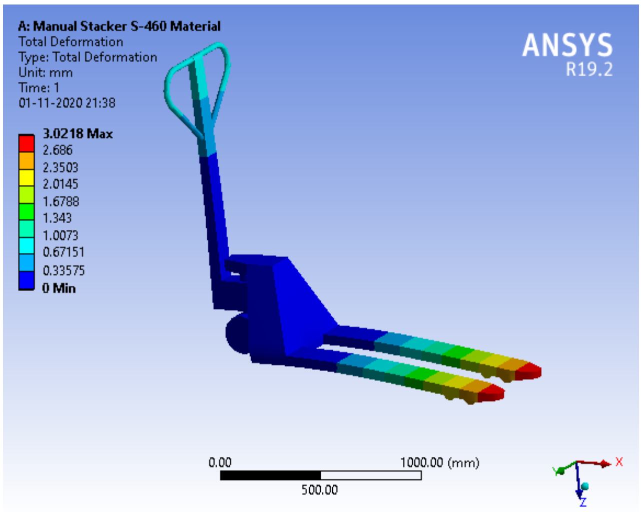

Figure 7. Total deformation (structural steel).

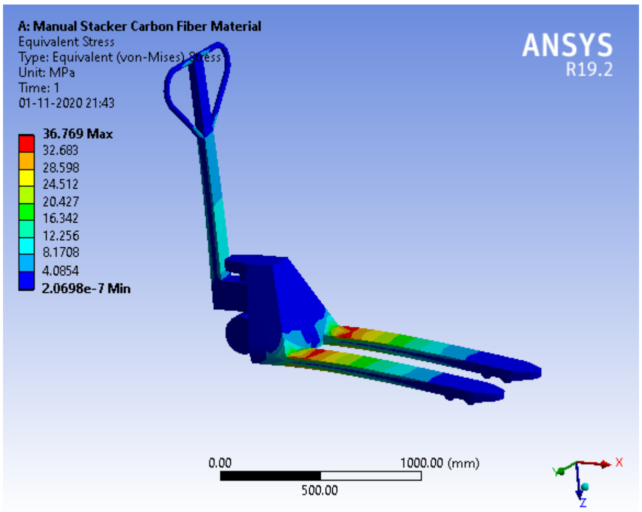

Figure 8. Equivalent stress carbon fiber material.

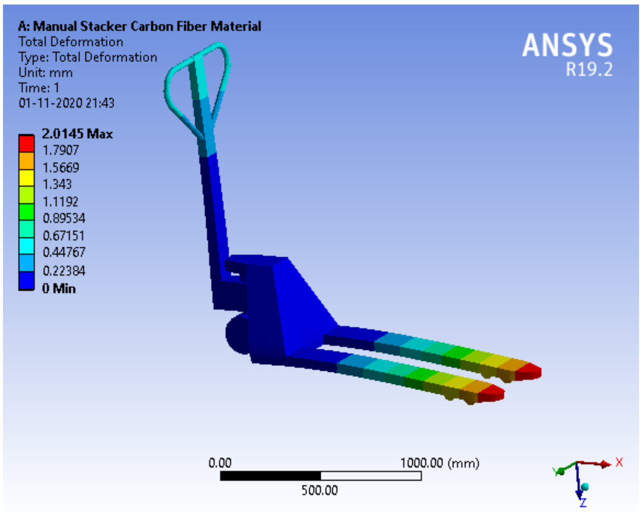

Figure 9. Total deformation on carbon fiber material of the stacker.

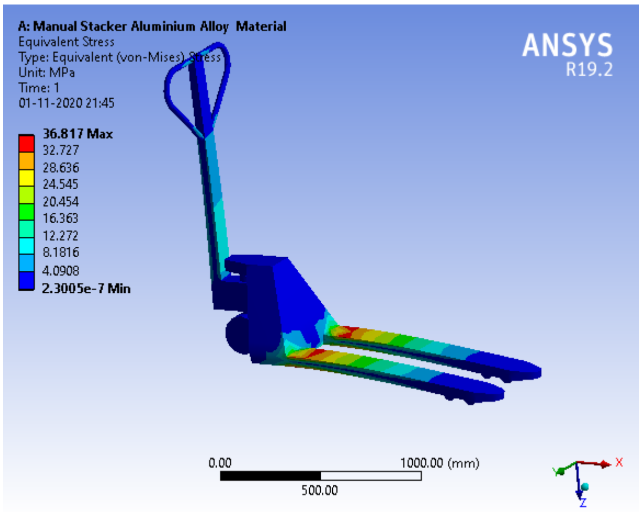

Figure 10. Equivalent stress aluminum alloy material.

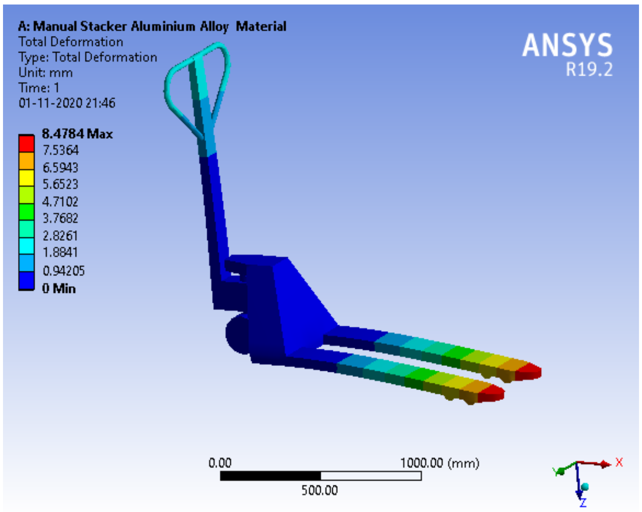

Figure 11. Total deformation aluminum alloy material.

Results and Discussion

In this study, we found that the stress of S-460, aluminum alloy, and carbon fiber were 36.7 MPa, 36.81 MPa, and 36.76 MPa, respectively. The total deformations of S-460, aluminum alloy, carbon fiber and other materials are 3.02 mm, 8.4 mm and 2.0 mm, respectively. Check out our different products from all materials, we will choose the composite material because it is light, heavy and has very high deformation and stress under 1000 kg load. This is a simple structural static analysis using one loading step of the angle iron shown in Figure 12. The bottom of the right pin hole is subjected to conical high load, while the entire perimeter of the left pin hole is constrained (welded). The purpose of this question is to introduce ANSYS analysis techniques. This picture shows the length of a right angle. The bracket is made of A36 steel with a Poisson's ratio of 0.27 and a Young's modulus of 30E6 psi. Because the bracket is only in the xy plane and is thin in the z direction compared to the length of the x and y cross-sections (1/2 inch thick), the problem can be considered a plane stress. A network is created using a 2D model, nodes, points, and the model used to create it. (Creating nodes and points directly in ANSYS is optional). In this study, the stresses of S-460, aluminum alloy, and carbon fiber were found to be 36.7 MPa, 36.81 MPa, and 36.76 MPa, respectively. The overall variations of S-460, aluminum alloy, CFRP and other materials are 3.02 mm, 8.4 mm and 2.0 mm respectively. Check out our wide range of products in all materials. I will choose composite materials because they are light and heavy and have very high strains and stresses under 1000 kg load. This is a simple structural static analysis using a single angular load step shown in below Figure 13. A high conical load is applied to the bottom of the right pinhole, and the entire perimeter of the left pinhole is fixed (welded). The purpose of this question is to introduce ANSYS analysis methods. This picture shows the length of a right angle. The bracket is made of A36 steel with a Poisson's ratio of 0.27 and a Young's modulus of 30E6 psi. Since the bracket is only in the xy plane and is long and thin in the z direction compared to the x and y parts (1/2 inch thick), the problem can be considered a plane stress. 2D models are assembled using nodes and elements, as well as the model used to create them. (Creating nodes and points directly in ANSYS is optional).

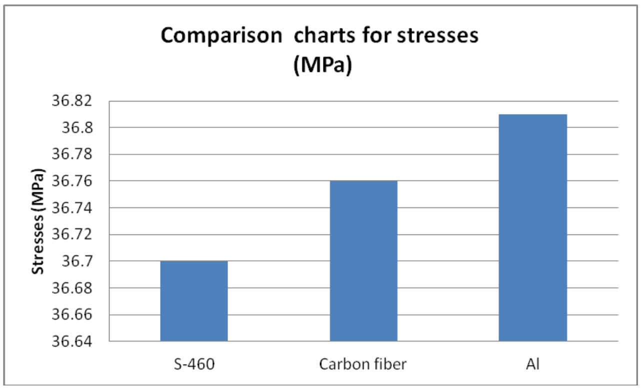

Figure 12. Stress comparison with different materials.

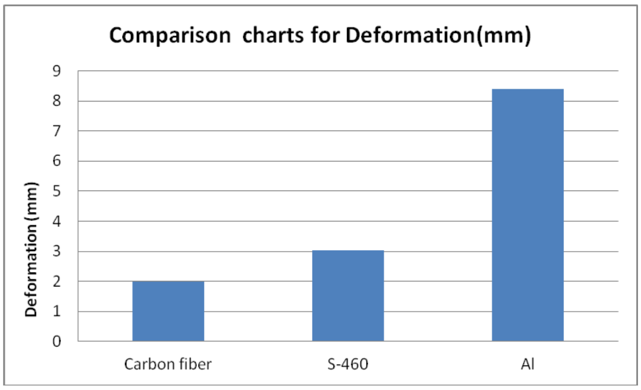

Figure 13. Deformations comparison with different materials.

Conclusion

In this paper we can see that we have used three different materials in all materials we will be selected composite material to other than because it is light weight and heavy duty its deformation and stresses range are considerable under 1000 kg loading condition and its very light weight compare to other than materials here we have optimize the unit weight of Pallet Truck 30% and its simple in construction, convenient lifting operating system and special design is available according to customers' requirements. With the help of Ansys' structural analysis software, engineers with varying specializations and backgrounds can solve challenging structural engineering issues more quickly and effectively. Engineers may do Finite Element Analyses (FEA), automate and customize solutions for structural mechanical problems, and examine various design scenarios with our set of tools. Businesses may save money, cut down on design cycles, and accelerate product launches by utilizing our tools early in the process. The CAD model's numerical stress analysis was carried out using the finite element-based software program ANSYS R19.2. The equivalent stress that resulted was calculated. The maximum load applied was three tonnes to prevent distortion on the mechanisms and base support. PTO Cero was used to create the prototype's CAD model, which was then loaded into ANSYS R19.2 for analysis. Mild steel was chosen as the material for the mode due to its availability, affordability, machinability, and other factors. The mode analysis was done to determine the weight at which the design can tolerate deformation. A force of three tonnes was applied to the base support model, and the prototype proved able to bear the force.

Future Scope

For further research work can be extended by using different materials and in future we can do work on stacker machine with composite materials and optimize weight, cost parameter and performed dynamic analysis with different loading conditions. A popular technique for modelling and examining how systems and structures behave in various scenarios is Finite Element Analysis (FEA). The need for FEA engineers who can use this technology to develop and optimize structures and systems is rising in tandem with the complexity of engineering designs. It has several benefits. Flexibility: Complex geometry, material characteristics, and boundary conditions are all difficulties that FEM can solve. Numerous physical phenomena, including fluid dynamics, electromagnetics, heat transfer, structural mechanics, and more, can be modelled with it. The FEM is a technique for using computer simulation to address engineering issues like stress analysis, heat transfer, fluid flow, and electromagnetics.

References

1. VamsiKrishna, K., and S. Porchilamban. "Development and Structural Analysis of Masthead for a Twin Boom Stacker." *Int J Eng Dev Res* 2 (2014).
2. Sivasubramanian, A., M. Jagadish, and C. Sivaram. "Design and Modification of Semi-Automatic Stacker." *Ind J Appl Res* 4 (2011):178.
3. van Vianen, Teus, Jaap Ottjes, and Gabriël Lodewijks. "Simulation-based Rescheduling of the Stacker–Reclaimer Operation." *J Comput Sci* 10 (2015): 149-154.
4. Miao, Sen Chun, Yong JiSun, and Ting Ting Wang. "The Transient Dynamic Analysis of multi-stage Fork of Stacker." *J Appl Sci Eng Innov* 1 (2014).
5. Sahu, Niraj Kumar, and Ram PrakashBhatele. "Stacker/Reclaimer Long Travel Drive Operation with Vfd-A Performance Study." *Int J Sci Eng Technol* Res 33 (2014).
6. Tian, Ming Hua, and Shi Cheng Hu. "Optimization of the Hinge Point Position of Luffing Mechanism in Reach Stacker for Container." *Adv Mater Res* 694 (2013): 142-147.
7. Xiao, Yanjun, Diming Guo, Xinyu Liang, and Yuming Guan. "The Finite Element Analysis of Trolly Frame of Stacker-Reclaimer Running Mechanism Based on ANSYS." *Res J Appl Sci Eng Technol* 6 (2013): 3015-3017.
8. fenn Daniel, Oscar, and A. Hussainlal. "Stress Analysis in Pulley of Stacker-Reclaimer by Using Fem Vs Analytical." *IOSR J Mech Civil Eng* 52-59.
9. Singh, Sarabjeet, and Yash Parikh. "Finite Element Analysis of Stacker Mechanism used in Bearing Manufacturing." *Int J Innov Res Adv Eng* 2 (2015).
10. Lalge, R. V., Kiran Godse, Aashreya Deolalikar, Nilesh Padalwar, and Mahesh Nivangune. "Design and Analysis of Two Wheel Drive Forklift for Industrial Warehouses." *New Arch-Int J Contemp Arch* 8 (2021): 825-830.
11. Sequeira, Anil A., Saif Mohammed, Avinash A. Kumar, Muhammed Sameer, Yousef A. Kareem, and Krishnamurthy H. Sachidananda. "Design and Fabrication of Battery Operated Forklift." *JESA* 52 (2019): 569-575.
12. Talele, Manoj P., and Ashok J. Keche. "Design Optimization and simulation Study of the Engine Transportation Metal Pallet for Stationary and Movable Condition." *Int J u and e-Service Sci Technol* 8 (2015): 2169–2230.
13. Yaman, Hande, and Alper Şen. "Manufacturer's Mixed Pallet Design Problem." *Eur J Oper Res* 186 (2008): 826–840.
14. Gzara, Fatma, Samir Elhedhli, and Burak C. Yildiz. "The Pallet Loading Problem: Three-Dimensional Bin Packing with Practical Constraints." *Eur J Oper Res* 287 (2020): 1062–1074.
15. Clarke, John W, Thomas E. McLain, Marshall S. White, and Philip A. Araman. "Evaluation of metal connector plates for repair of wood pallet stringers." *Forest Prod J* 43 (1993): 15–22.
16. Kabir, Mohammed F, Daniel L. Schmoldt, Philip A. Araman, Mark E. Schafer, and Sang-Mook Lee. "Classifying defects in pallet stringers by ultrasonic scanning." *Wood Fiber Sci* 35 (2003): 341–350.
17. Adamakos, Konstantinos, Stefano Sesana, and Ioannis Vayas. "Interaction between Pallets and Pallet Beams of Steel Storage Racks in Seismic Areas." *Int J Steel Struct* 18 (2018): 1018–1034.
18. Safronov, Eugene, and Andrey Nosko. "A Method to Determine Allowable Speed for a Unit Load in a Pallet Flow Rack." *Acta Mech Automatica* 13 (2019): 80-85.
19. Shah SNR, NH Ramli Sulong MZ Jumaat, and Mahdi Shariati. "State-of-the-art Review on the Design and Performance of Steel Pallet Rack Connections." *Eng Fail Anal* 66 (2016): 240–258.
20. Lyu, ZhiJun, Jie Zhang, Ning Zhao, Qian Xiang, YiMing Song, and Jie Li. "A Comparative Study on the Performance of Fem, Ra and Ann Methods in Strength Prediction of Pallet-Rack Stub Columns." *Int J Steel Struct* 20 (2020): 1509–1526.

How to cite this article: Pandey, Rohit and Ashish Kumar Shrivastava. "Design and Analysis of Manual Pallet Stacker by Using FEM Method." *Adv Robot Autom* 14 (2025):308.