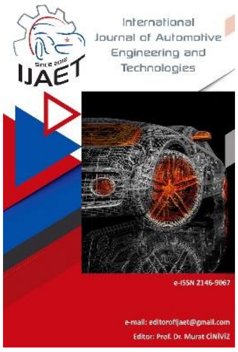
e-ISSN: 2146 - 9067

International Journal of Automotive Engineering and Technologies

journal homepage:

https://dergipark.org.tr/en/pub/ijaet



Original Research Article

Design and structural analysis of a mechanism for positioning heavy vehicle chassis

Tolga Güney[1], Sinan Düzenli[2], Kasım Serbest[3, *]

[1, 2] YEKTAMAK Engineering and Machinery, Sakarya, Türkiye.

[3, *] Sakarya University of Applied Sciences, Department of Mechatronics Engineering, Sakarya, Türkiye.

ARTICLE INFO

Orcid Numbers

1. 0009-0002-8767-3891
2. 0009-0001-4230-2766
3. 0000-0002-0064-4020

Doi: 10.18245/ijaet.1604938

* Corresponding author
kserbest@subu.edu.tr

Received: Dec 20, 2024

Accepted: Feb 12, 2025

Published: 25 Mar 2025

Published by Editorial Board Members of IJAET

To cite this paper: Güney, T., Düzenli, S., and Serbest, K., Design and structural analysis of a mechanism for positioning heavy vehicle chassis, International Journal of Automotive Engineering and Technologies. 2025, 14 (1), 38 – 46. http://dx.doi.org/10.18245/ijaet.1604938

ABSTRACT

Proper and secure positioning of heavy vehicle chassis is crucial in ensuring efficiency and safety, particularly in painting, manufacturing, and maintenance processes. This study focuses on the structural analysis and prototype production of a positioning system capable of rotating heavy vehicle chassis across three axes and setting them in an upright position. To achieve this, a scissor platform was developed to facilitate three-axis rotation, and a dedicated mechanism was designed for upright positioning. Initially, a 3D model of the system was created, and structural strength was evaluated through Finite Element Analysis (FEA), assessing stress distribution under load and deformation during rotation. The analysis revealed a maximum stress value of 190.2 MPa in the chassis tilting mechanism and 204.2 MPa in the scissor platform, with a maximum displacement of 3.1 mm observed in the scissor platform. Following optimization based on these results, a prototype was produced and tested under real working conditions. The findings validate the system’s durability and functionality, demonstrating a reliable and effective solution for the positioning of heavy vehicle chassis.

Keywords: Finite Element Analysis (FEA), Structural Optimization, Three-Axis Rotation, Machine Design

1. Introduction

Due to their excessive weight and large dimensions, the precise positioning of heavy vehicle chassis is a challenging process. Accurate positioning of these chassis is essential for ensuring efficiency and meeting quality requirements in painting, manufacturing, and maintenance processes. Existing systems in the literature typically operate by combining mechanical and hydraulic components to safely and accurately lift, position, and stabilize heavy loads. These systems aim to optimize load distribution, thereby enhancing structural strength and operational efficiency.

The literature surrounding the design and analysis of scissor lift mechanisms is extensive, particularly in the context of heavy vehicle chassis positioning systems. The importance of structural integrity and stability in lifting platforms is underscored by various studies that highlight the critical role of finite element analysis (FEA) in evaluating stress distribution and deformation underload. For instance, Chen et al. conducted a mechanical analysis and finite element simulation of scissor transmission mechanisms, emphasizing the impact of partial loads on the service life and security of these systems [1]. Similarly, Zhang et al. explored the static stability of scissor lifts, providing insights into the energy methods and modeling techniques used to assess stability under operational conditions [2]. When designing a mechanism for positioning heavy vehicle chassis, the integration of robust structural components and optimized load distribution is considered extremely important [3]. These findings are crucial for validating the structural strength of the proposed positioning system.

The development of scissor lift platforms has also seen innovations aimed at enhancing their operational efficiency and durability. For example, Solmazyigit et al. discussed the design and prototype production of a scissor lift platform with a capacity of 25 tons, which incorporated advanced materials to minimize wear and improve stability [4]. This aligns with the findings of Petru et al., who noted that the symmetrical distribution of forces on lifting platforms is essential for reducing bending moments and enhancing overall stability during operation [5]. Such design considerations are vital for ensuring that the positioning system can withstand the stresses encountered during the rotation and upright positioning of heavy vehicle chassis. Therefore, the successful design and implementation of a mechanism for positioning heavy vehicle chassis require a comprehensive understanding of fundamental principles and a thorough addressing of structural and operational challenges [6].

Moreover, the integration of hydraulic systems in scissor lifts has been shown to significantly improve their functionality. Kart's research on hydraulic walking power steering-controlled scissor lift platforms highlights the advantages of hydraulic systems in enhancing the mobility and control of lifting mechanisms [7]. This is particularly relevant for the proposed system, which requires precise control during the three-axis rotation and upright positioning of heavy vehicle chassis. The optimization of hydraulic components can lead to improved performance, as noted by Yimer and Wang, who explored the design and manufacturing of hydraulic cylinder-based scissor lifts, demonstrating their effectiveness in the automotive sector [8]. In summary, the design and development of a mechanism for positioning heavy vehicle chassis requires a multifaceted approach, encompassing structural analysis, stability considerations, and the integration of advanced hydraulic systems.

Unlike previous studies that primarily focus on the design, analysis, or operational improvements of scissor lift mechanisms in isolation, this study aims to develop a comprehensive positioning system specifically tailored for heavy vehicle chassis. By integrating a scissor lift platform capable of precise three-axis rotation and an upright positioning mechanism, this research addresses the challenges posed by the excessive weight and large dimensions of heavy vehicle chassis. The proposed system is designed not only to optimize load distribution and structural strength but also to enhance maneuverability, operational efficiency, and stability during positioning operations. Through finite element analysis (FEA) and prototype testing, the study validates the system's structural integrity, ensuring its capacity to handle significant mechanical stresses and deformations. Additionally, the system's modular design facilitates easier maintenance and future upgrades, contributing to long-term operational reliability. The incorporation of hydraulic components further enhances precision, enabling seamless control during chassis rotation and positioning tasks. The findings of this research provide a practical and scalable solution for automotive and heavy vehicle industries, potentially setting a benchmark for similar applications in the manufacturing and maintenance sectors. By addressing key engineering challenges such as load distribution, structural resilience, and system adaptability, this study contributes a novel and efficient approach to heavy vehicle chassis positioning technology.

2. Materials and Methods

2.1. 3D modeling and system design

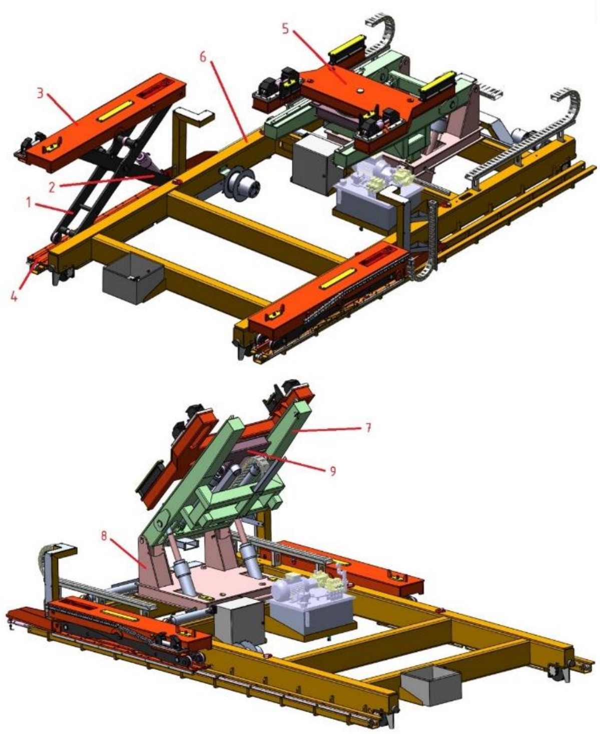
The 3D modeling process was carried out

using CAD software, which enabled the creation of precise models for all components of the system. The software allows detailed geometric accuracy and facilitated assembly and motion analysis to ensure compatibility between parts. During this phase, the system was also simulated under different conditions to anticipate and address potential issues. In the system design phase, key factors such as functionality, durability, and ease of use were prioritized. The design focused on structural integrity, with components arranged to withstand the forces encountered during operation. Modular structures were used to simplify maintenance and repairs, while ergonomic considerations were made to ensure operator's safety and comfort. Energy efficiency was another critical aspect, particularly in the hydraulic systems that powered the three-axis rotation and upright positioning. By optimizing fluid dynamics and system layout, the design aimed to improve operational efficiency while reducing energy consumption. The final design also emphasized safety, ensuring that all components were capable of withstanding high loads without failure. The 3D model of the system was refined through an iterative review process, resulting in an optimized design ready for prototype production. The complete model, shown in Figure 1, includes all key components such as the chassis tilting mechanism and scissor lift platform.

2.2. Finite element analysis for structural integrity

The Finite Element Analysis (FEA) was conducted to evaluate the structural integrity of the system under operational conditions. In this analysis, standard gravitational acceleration (9.81 m/s²) was applied to simulate real-world loading conditions. Steel material properties were assigned to all components, with an elasticity modulus of 210 GPa, Poisson's ratio of 0.3, and a density of 7850 kg/m³, which are typical for structural steel used in heavy machinery applications [9].

In this study, the structural analyses of the developed system were performed using a nonlinear finite element model. The Second Piola-Kirchhoff stress method was utilized to model deformations. The Piola-Kirchhoff tensor is used to represent stress in the reference configuration for finite deformations. The force in the reference configuration is expressed in relation to the relative orientation between the direction of the force and the normal vector of the surface. The Second Piola-Kirchhoff stress is expressed as follows.



1; outer scissors, 2; inner scissors, 3; scissor lift upper platform, 4; scissor lift lower platform, 5; chassis bracing upper platform, 6; lower chassis, 7; vertical positioner, 8; horizontal positioner, 9; rotation group.

Figure 1. 3D model of the proposed system.

$$S = \frac{FL_0}{A_0 L} = \frac{\sigma_E}{(1+\epsilon_E)} = \frac{\sigma_E}{\lambda} \quad (1)$$

where F represents the force, L_0 denotes the original length of the part, L represents the final length of the part, A_0 indicates the initial cross-sectional area, σ_E refers to the engineering stress, ε_E represents the engineering strain, and λ denotes the stretch ratio.

Additionally, the matrix compression method [10] was used to reduce the degrees of freedom of each element and optimize the calculations. Mesh convergence analysis was conducted to ensure the accuracy and reliability of the structural analysis results for the chassis positioning mechanism. The analysis aimed to refine the mesh size and increase the number of elements, as shown in Table 1. Despite these refinements, the observed stress values exhibited minimal differences. For instance, the difference in stress values between the 7 mm and 8 mm mesh sizes was less than 1%. This indicates that the solution is effectively converged. The mesh convergence process was applied to all analyzed components to ensure consistency and optimize computational efficiency.

Table 1. Mesh and stress variations for the model

Mesh size (mm)	**Stress value (MPa)**	**Number of elements**
15	198	400
13	212	736
11	215	1404
9	220	3036
8	221	3496
7	221	3956

The system was evaluated in two separate load groups to simulate different operational scenarios. For the chassis tilting mechanism, a distributed load of 5000 kg was applied, considering the weight of the chassis during tilting. For the scissor platform, which operates symmetrically, a distributed load of 2500 kg was applied to each platform. These loads were carefully chosen to reflect the real working conditions of the system.

To ensure the accuracy of the analysis, the structural components were meshed using both 3D and 2D grids, depending on the thickness of the parts. Quadrilateral elements were used for meshing, which are particularly suited for thin-walled components and provide precise results. For the analysis of critical components, triangular elements were preferred for more precise analysis on cylindrical surfaces. Additionally, RBE2 and RBE3 elements were used to model welds and bolt connections, ensuring that the structural integrity of these critical junctions was properly represented [11]. Some examples of the developed finite element model and its mesh structure are presented in Figure 2. The component subjected to the highest load in the developed system is the axle shown in Figure 3. Mesh optimization was performed on this axle, and a mesh size of 8 mm was selected for structural analyses. The results of this optimization were also applied to other critical structural components.

The loading and boundary conditions required for the FEA were defined based on the mechanism's limit position. Figure 4 illustrates the limit loading position of the mechanism. Here, the load was determined using a point mass of 5000 kg concentrated at the center of gravity of heavy vehicle chassis. Boundary conditions were defined by considering the operational characteristics of the components. For the moving parts, single-degree-of-freedom rotational and translational motions were assigned to accurately simulate their functional behavior under real-world conditions. All other parts of the system were assumed to be fixed to ensure stability and represent their stationary nature during operation. A distributed load was applied across the entire system to simulate the external forces acting on the mechanism.

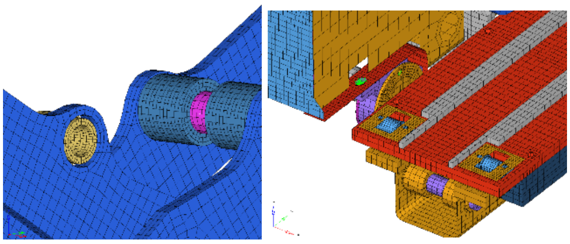


Figure 2. Finite element model and mesh types.

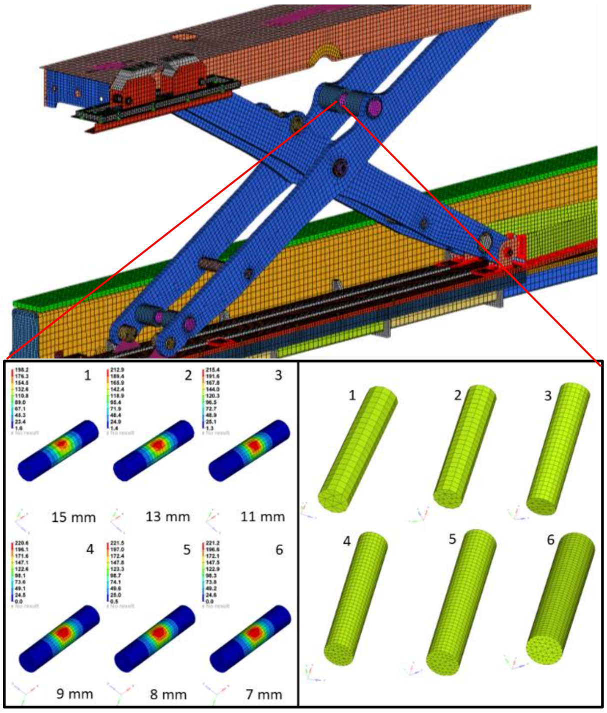


Figure 3. Mesh optimization for the critical component.

The FEA results identified key stress concentrations and potential failure points, which will guide the design improvements needed for enhanced durability and safety. By analyzing stress distribution across various components, such as the chassis tilting mechanism, the scissor platform, and the connecting joints, the FEA provides invaluable insight into the system's performance underload. The results from this analysis form the basis for further optimization of the system's design to improve its strength, reduce material fatigue, and ensure safe operation.

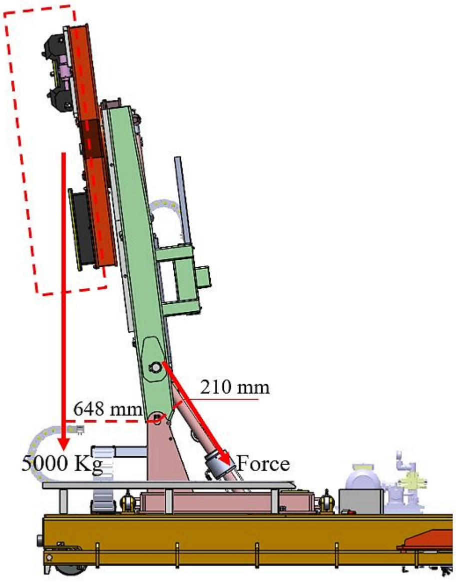


Figure 4. Limit position and loading conditions.

2.3. Prototype production

The prototype production phase was essential for validating the design and functionality of the system developed during the design and modeling phases. This phase involved the physical manufacturing of the components and their integration into a working system. Various production techniques were employed, including CNC machining for precision component fabrication, welded manufacturing for structural integrity, and hydraulic assembly to simulate operational conditions.

CNC machining was used to produce components with high accuracy and tight tolerances, ensuring that the parts fit together seamlessly during assembly. The use of welded manufacturing provided a strong and durable connection for structural parts, particularly for areas subjected to high stress, such as the chassis tilting mechanism and the scissor lift platform. Hydraulic systems were integrated to simulate the real operational conditions of the lift mechanism, allowing for testing of the system's functionality and performance underload.

Once the individual components were produced, the system was assembled, and a series of functional and structural tests were carried out. These tests were designed to assess the system's performance under various operational conditions, ensuring that it met the required standards for stability, durability, and safety. Durability tests involved subjecting the system to repetitive loading and unloading cycles to evaluate wear resistance, while safety tests focused on the performance of the system under extreme loads and operational stresses.

The findings from these prototype tests provided valuable insights into the system's behavior, highlighting areas where design improvements were needed. Modifications were made to enhance the system's performance, such as reinforcing critical stress points and optimizing the hydraulic components for better efficiency and control. An illustration of the prototype and testing processes is shown in Figure 5, highlighting the real-world implementation and functionality of the system.

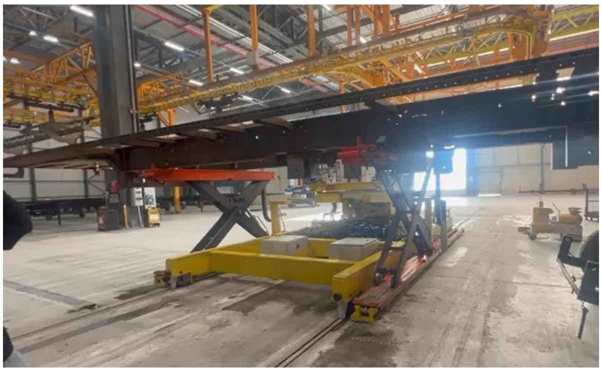


Figure 5. Final version of the system for positioning heavy vehicle chassis.

3. Results and Discussion

This section presents the structural analysis results of the developed system, conducted using the finite element method. The structural analysis results of the developed system, conducted using the FEM, reveal critical stress points in various components of the chassis tilting mechanism. The highest stress observed in the upper structure of the chassis tilting mechanism was 31.4 MPa, as shown in Figure 6(a), indicating a key point that affects overall stability. In the chassis rotation structure, the highest stress value of 32.8 MPa, presented in Figure 6(b), highlights the importance of optimizing this area for improved rotational stability. The tilting arm experienced a stress of 152.6 MPa, shown in Figure 6(c), which suggests a need for reinforcement to prevent deformation under heavy loads.

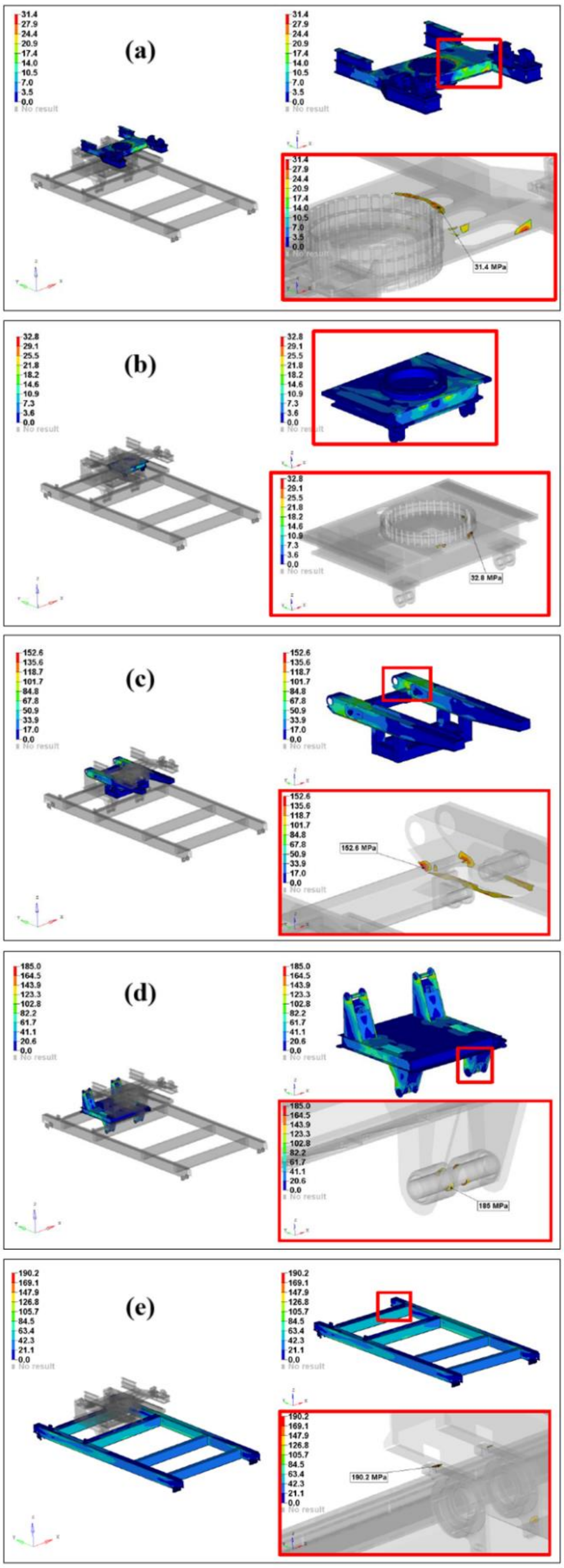


Figure 6. Stress analysis results.

The horizontal sliding module, with a maximum stress of 185.0 MPa as seen in Figure 6(d), also indicates a potential area for design optimization to enhance wear resistance. Finally, the fixed chassis module, exhibiting the highest stress of 190.2 MPa as depicted in Figure 6(e), underscores the need for strengthening to ensure stability and prevent failure under operational conditions. These results provide valuable insights into future design adjustments to improve the system's performance and durability.

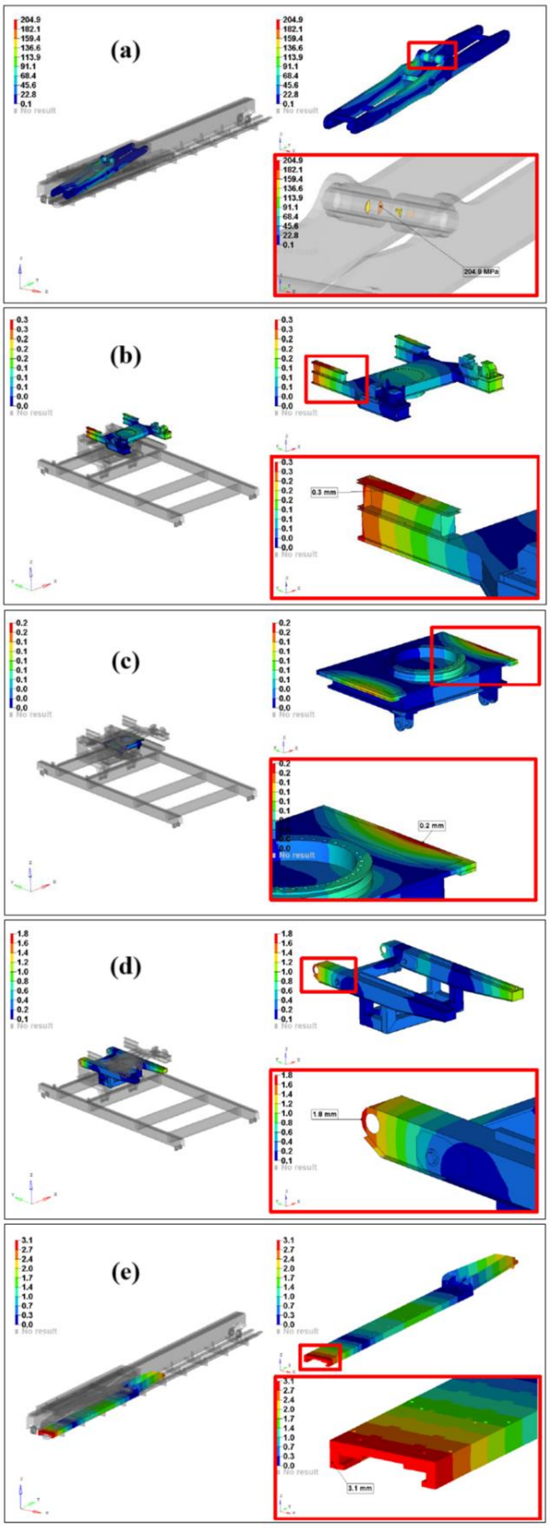


Figure 7. Stress and displacement analysis results.

The structural analysis results also revealed key displacement values observed in various components of the system. The highest stress observed in the outer scissor module of the scissor platform was 204.9 MPa, located at the position indicated in Figure 7(a). Regarding displacement, the highest value in the upper structure of the chassis tilting mechanism was 0.3 mm, as shown in Figure 7(b). The highest displacement in the chassis rotation module of the chassis tilting mechanism was 0.2 mm, located at the position indicated in Figure 7(c). In the tilting arm of the chassis tilting mechanism, the maximum displacement value was 1.8 mm, as shown in Figure 7(d). Finally, the highest displacement observed in the module where the scissor platform slides was 3.1 mm, located at the position indicated in Figure 7(e). These displacement values highlight areas of potential movement and deformation, which are critical for ensuring the system's stability and performance underload.

The results obtained from the FEA and prototype testing provide valuable insights into the performance and structural integrity of the proposed heavy vehicle chassis positioning system. The FEA results revealed critical stress points in various components of the system, such as the chassis tilting mechanism, scissor platform, and connecting modules. These findings align with previous studies on scissor lift systems, which emphasize the importance of optimizing load distribution and stress concentration to ensure the system's durability and safety under operational conditions.

In particular, the high stress observed in the fixed chassis module (190.2 MPa) and the tilting arm (152.6 MPa) indicate that these components are subjected to significant forces during operation. These results suggest that the design of these parts should be reinforced to prevent potential failures due to excessive stress. The stress values in other components, such as the horizontal sliding module (185.0 MPa) and the chassis rotation structure (32.8 MPa), also highlight areas where optimization could reduce wear and enhance long-term reliability [12].

The displacement analysis further corroborates the findings from stress analysis, with areas of the system, such as the scissor platform sliding module (3.1 mm displacement), exhibiting noticeable movement underload. This displacement is within acceptable limits for functionality but could lead to long-term issues if not addressed, particularly in terms of wear and potential misalignment over time. The displacement observed in the tilting arm (1.8 mm) and upper structure (0.3 mm) also provides useful data for assessing the system's operational stability and guiding future design improvements [13].

The prototype production phase was crucial for testing the real-world applicability of the design. The functional and structural tests confirmed the system's ability to handle the stresses and displacements predicted by the FEA, but they also revealed areas where further refinements were needed. The feedback from the prototype tests highlighted the importance of optimizing hydraulic components to improve performance and reduce operational inefficiencies. Additionally, reinforcing critical components, such as the tilting arm and the fixed chassis module, could further enhance the system's longevity and load-bearing capacity.

One notable contribution of this study is the development of a comprehensive positioning system that integrates both scissor lift mechanisms and an upright positioning mechanism. This integrated system addresses the unique challenges posed by heavy vehicle chassis, which require precise, stable, and reliable positioning during various operational tasks [14]. The ability to rotate and position the chassis along three axes adds versatility and control, making the system an effective solution for industries such as manufacturing, maintenance, and painting, where accurate positioning of large, heavy components is essential.

Despite the promising results, some limitations were identified during testing. For instance, the system's overall efficiency could be improved by further optimizing the hydraulic and mechanical components to reduce energy consumption and enhance operational speed. Additionally, long-term testing under varied conditions is required to assess the system's performance in real-world environments and ensure its reliability over time.

4. Conclusion

This study presents the design and analysis of a positioning system for heavy vehicle chassis, incorporating a scissor lift platform with three-axis rotation and upright positioning mechanisms. Finite element analysis (FEA) identified critical stress points and displacement values, guiding design improvements for structural integrity and performance. Prototype testing confirmed the system's functionality, with minimal deformation under load, ensuring operational efficiency and safety. The prototype tests were based on field trials under the system's actual working conditions. Observation-based results were obtained after extended periods of operation. One of the future studies involves preparing an appropriate experimental setup to compare experimental data with FEA results.

The results validate the proposed system's effectiveness, providing a reliable solution for heavy vehicle chassis positioning. In another future work, additional improvements could focus on reducing the weight of the system's components, increasing modularity for easier maintenance, and enhancing the user interface for better control and monitoring of the system during operation. Further research could also explore alternative materials for component fabrication, which could reduce wear and improve energy efficiency. Ultimately, the goal is to refine the system to offer a more sustainable and cost-effective solution for heavy vehicle chassis positioning.

CRediT authorship contribution statement

Tolga Güney: Conceptualization, Investigation, Methodology, Writing-Original Draft.
Sinan Düzenli: Conceptualization, Investigation, Methodology, Writing-Original Draft.
Kasım Serbest: Conceptualization, Investigation, Writing-Final Draft, Supervision.

Declaration of Competing Interest

The authors declare no conflict of interest.

Acknowledgement

This study was conducted at the design center of YEKTAMAK Engineering and Machinery Co. Ltd.

5. References

1. Chen, J., Xue, Y., and Zhang, F., Mechanical analysis and finite element simulation of scissor transmission mechanism under partial load, Key Engineering Materials, 667, 518-523, 2015.
2. Zhang, W., Zhang, C., Zhao, J., and Du, C., A study on the static stability of scissor lift, The Open Mechanical Engineering Journal, 9(1), 954-960, 2015.
3. Podkowski, K., Okruch, Ł., Jasiński, P., and Stańko-Pająk, K., Strength numerical analyses of the construction of the PAWO autonomous high mobility platform, XII International Science-Technical Conference Automotive Safety, 2020.
4. Solmazyiğit, İ., Başkurt, R., Ovalı, İ., and Tan, E., Design and prototype production of scissor lift platform 25 tons capacity, The European Journal of Research and Development, 2(4), 326-337, 2022.
5. Petrů, M., Lepšík, P., Novák, O., and Lufinka,A., Numerical modelling and experimental measurement of lifting platform construction for car relocation, Applied Mechanics and Materials, 732, 219-222, 2015.
6. Zhao, X., Li, T., Fang, X., Liang, P., Yang, Z., Liu, J., and Yang, Y., Design Optimization of an Electric Positioning Mechanism, Journal of Engineering Mechanics and Machinery, 7(4), 44-52, 2022.
7. Kart, S., Conceptual design and prototype production of innovative hydraulic walking power steering controlled scissor lift platform, The European Journal of Research and Development, 3(4), 195-204, 2023.
8. Yimer, W., and Wang, Y., Design, analysis and manufacturing of double scissors lift elevated by one hydraulic cylinder, International Journal of Engineering Research & Technology, 8(11), 709-713, 2019.
9. Dao, N.D., and Ryan, K.L., Computational simulation of a full-scale, fixed-base, and isolated-base steel moment frame building tested at E-defense, Journal of Structural Engineering, 140(8), A4014005, 2014.
10. Chang, S., Liu, K., Yang, M., and Yuan, L., Theory and implementation of sub-model method in finite element analysis, Heliyon, 8(11), 2022.
11. Legrain, G., Geuzaine, C., Remacle, J.F., Moës, N., Cresta, P., and Gaudin, J., Numerical simulation of CAD thin structures

using the eXtended Finite Element Method and Level Sets, Finite Elements in Analysis and Design, 77, 40-58, 2013.
12. Wang, W., Huang, B., Du, J., and Cheng, S., Structural optimization for stiffness and stochastic fatigue life improvement of a sport utility vehicle chassis, IOP Conference Series: Materials Science and Engineering, 692(1), 012030, 2019.
13. Tang, C., He, L., and Khajepour, A., Design and analysis of an integrated suspension tilting mechanism for narrow urban vehicles, Mechanism and Machine Theory, 120, 225-238, 2018.
14. Cui, Z., Xu, H., Chen, Z., Yang, H., Huang, S., and Gong, M., Design of a novel AGV with automatic pick-and-place system based on scissor lifting platform, Chinese Automation Congress (CAC), 4435-4440, 2020.