DOI:10.5937/IIZS25049J

APPLICATION OF CAD TOOLS IN DESIGNING COMPLEX MECHANICAL ASSEMBLIES

Ratko Joksimović[1], Miroslav Milutinović[1]
[1]University of East Sarajevo, Faculty of Mechanical Engineering, East Sarajevo, Bosnia and Herzegovina
e-mail: ratko.joksimovic92@gmail.com, miroslav.milutinovic@ues.rs.ba

Abstract: This work presents the application of CAD tools on the example of a hydraulic lifting table. The mechanism was modeled in Autodesk Inventor and analyzed through motion simulation and static calculation using the finite element method. The results indicate that CAD software accelerates development, reduces costs, and enables timely corrections within the virtual model.
Key words: *stress analysis, motion simulation, finite element method (FEM), Computer-aided design (CAD)*

INTRODUCTION

The development of modern industry is unimaginable without computer tools that enable fast and precise design, analysis, and optimization of technical systems. According to Budynas and Nisbett, the integration of computer-aided methods has fundamentally transformed engineering design by enabling more reliable and efficient solutions [1]. A key role in this process is played by CAD (Computer-Aided Design) software, which has become a standard in mechanical engineering. Its application includes creating 3D models, generating technical documentation, and connecting with tools for motion simulation and stress analysis. As Norton emphasizes, the combination of CAD modeling with analytical methods significantly improves the ability of engineers to predict system behavior during the design stage [2]. This significantly reduces product development time, increases construction accuracy, and improves quality control.

Designing complex assemblies with many interconnected parts requires ensuring functionality, reliability, and cost-effectiveness at the same time. Modern CAD software provides engineers with the ability to test such assemblies virtually, thereby reducing the need for physical prototypes and lowering development costs [3]. In this context, CAD software is an indispensable tool, allowing quick validation of ideas and precise engineering decisions.

The aim of this paper is to present the importance of CAD programs in the development of constructions with complex mechanical assemblies. As a case study, a hydraulic lifting table, shown on Fig. 1, was modeled using CAD tools, followed by motion simulation and static stress analysis. This type of lifting table is used for raising heavy loads to an ergonomic height, allowing the operator to work on them more efficiently. Based on the obtained results, the advantages of this approach are discussed in comparison to traditional design methods.

Fig. 1. *Hydraulic lifting table*

CAD TOOLS AND THEIR ROLE IN ENGINEERING DESIGN

CAD tools occupy a central role in modern engineering design, as they enable the creation, analysis, and modification of complex constructions within a digital environment. Their development and widespread adoption have significantly accelerated the design process and improved the quality of technical documentation. By using CAD software, engineers can develop 3D models, quickly implement design changes, and visually demonstrate the operation of complex assemblies, which was previously difficult to achieve using traditional methods.

A particularly important feature of CAD tools is the integration of models with calculations and simulations. Within CAD software and integrated environments, it is possible to perform motion analyses, load calculations, as well as detailed stress and deformation analyses using the FEM [1]. This allows the design to be evaluated during the development phase, reducing the need for physical prototypes and accelerating product development.

Modern CAD tools, such as Autodesk Inventor, SolidWorks, CATIA, and PTC Creo, offer modules for kinematic and dynamic simulations of assemblies, as well as static and modal analyses [4]. Stress and deformation analyses allow engineers to identify critical areas in the design and optimize geometry or materials before production [2]. This approach not only increases the safety and reliability of the construction but also contributes to cost reduction and shorter development time. Fig. 2 illustrates the main stages of construction development using CAD tools.

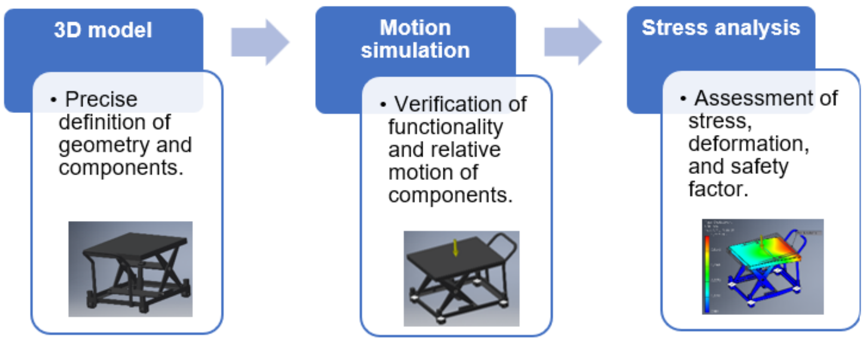


Fig. 2. *Main stages of construction development using CAD tools*

In industrial practice, the use of CAD tools combined with simulations and calculations provides multiple advantages. It allows testing of various design variants under different operating conditions, enabling performance optimization and extending the product's service life [5, 6]. In this way, CAD becomes a fundamental tool not only for design but also for the validation and optimization of complex mechanical assemblies.

MODELING AND ANALYSIS OF THE HYDRAULIC LIFTING TABLE STRUCTURE

Creation of the 3D Model

In the first phase of the work, it is essential to create a 3D model of the structure, which serves as the foundation for all subsequent simulations and calculations. The model was developed using Autodesk Inventor, one of the most widely used CAD tools in industry today, due to its intuitive interface and the ability to integrate with various analysis modules. At this stage, the structure is precisely defined through all necessary dimensions, shapes, and the relationships between components. Special attention is given to accurately representing joints, as these areas often constitute critical zones in terms of loads and stress.

Characteristics of the hydraulic lifting table:

- Work surface: 800 x 1000 mm
- Working height: 850 mm

- Load capacity: 500 kg
- Structure weight: 220 kg
- Minimum height: 250 mm
- Lifting/lowering time: 15 seconds

The main components of the hydraulic lifting table are shown on Fig. 3.

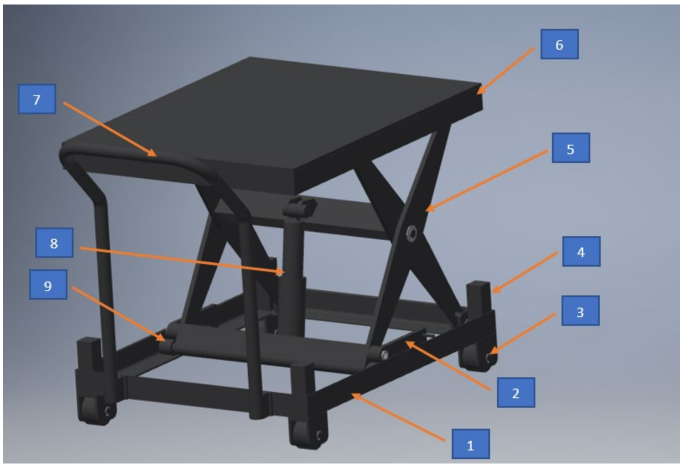


Fig. 3. *Main components of the hydraulic lifting table*

1. Lower frame
2. Safety brake
3. Wheels
4. Minimum height limiter
5. Scissor mechanism
6. Upper frame with plate
7. Handles for moving the hydraulic table
8. Hydraulic cylinder
9. Lower rollers

Preparation for Analyses

After creating the 3D model, the next step is to prepare the structure for analyses. This stage involves checking the geometry and removing any unnecessary details that do not affect the simulation results but could increase the complexity of the finite element mesh and thus prolong computation time [6]. Additionally, the materials of all components are defined, with emphasis on using realistic mechanical properties (elastic modulus, yield strength, Poisson’s ratio, etc.) to ensure reliable results. Autodesk Inventor allows easy selection of materials from the built-in library, with the option to enter custom data when needed. For the hydraulic lifting table, different materials were applied: standard structural steels were used for the supporting frame, while the moving parts (rollers and shafts) were made of improved steel to increase resistance to load and wear.

Motion Simulation and Static Analysis Setup

During the simulations, special attention is given to motion analysis and static calculation of the structure. Motion simulation allows verification of the mechanism’s functionality and analysis of its operating dynamics, enabling the identification of potential collisions, blockages, or irregularities in system operation [7].

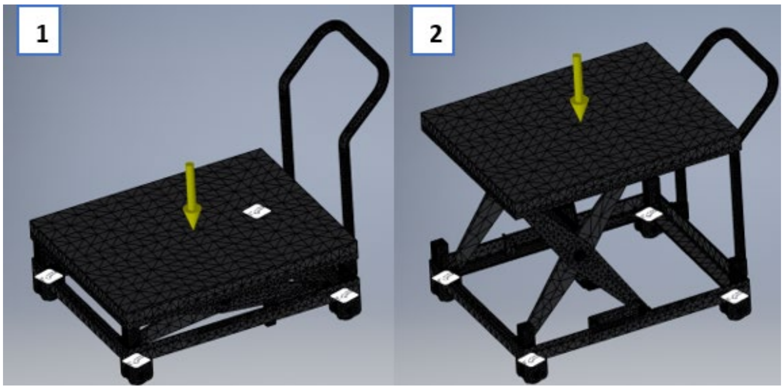


Fig. 4. *Fixed points and force load of the hydraulic table in the lower (1) and upper (2) positions*

For the purpose of static stress analysis, the structure is fixed at the support points, while appropriate loads corresponding to real working conditions are applied to relevant components, as shown on Fig. 4. In our case, the structure must support a load of 500 kg, corresponding to a force of 5000 N. It is also necessary to define a suitable FEM, where a finer mesh in critical areas provides more accurate results [8, 9]. The software then calculates the distribution of stress, deformation, and displacement.

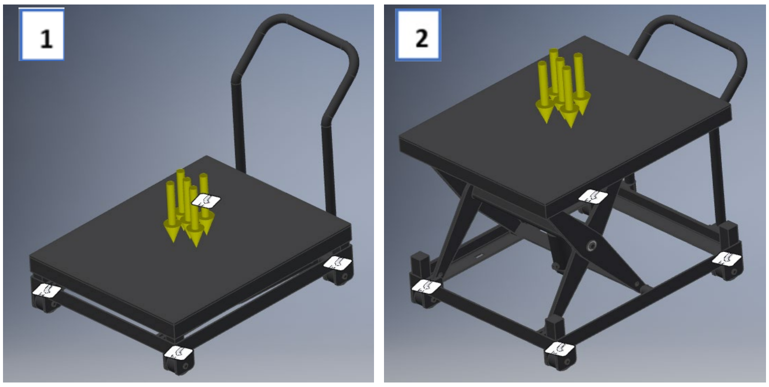


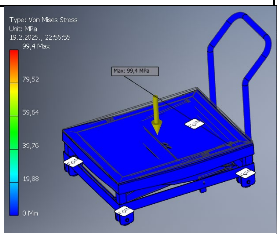
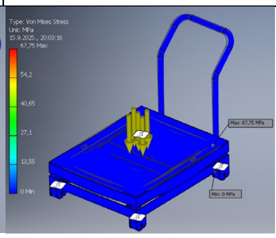
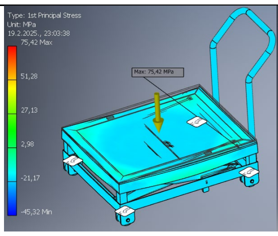
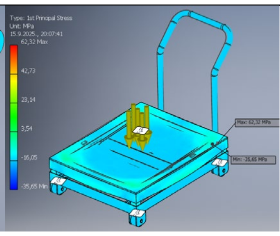
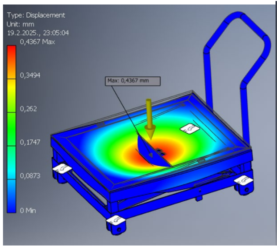
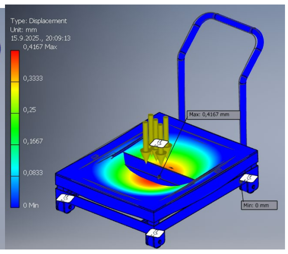
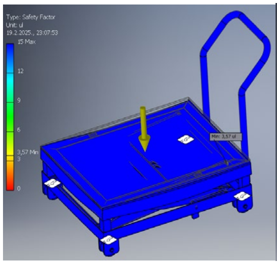
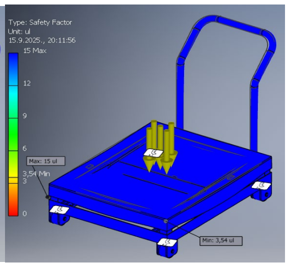
Fig. 5. *Fixed points and distributed loads of the hydraulic table in the lower (1) and upper (2) positions*

In addition to the force load applied directly to the platform surface, an alternative approach was carried out using distributed load, represented as surface pressure equivalent to the weight of 500 kg acting on the working platform of the hydraulic lifting table, as shown on Fig. 5. This enabled a comparative assessment of the results obtained under the two loading methods, providing a more comprehensive evaluation of the structural behavior.
The structure is also subjected to the weight of the hydraulic lifting table components, which is applied in the CAD program using the "Gravity" function.

RESULTS AND DISCUSSION

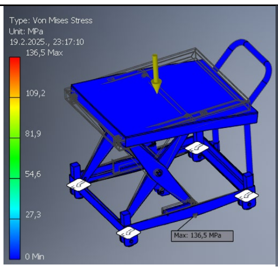
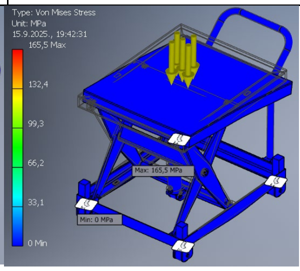
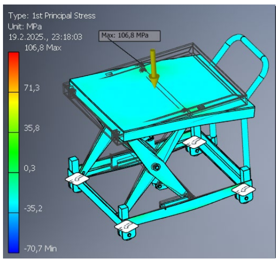
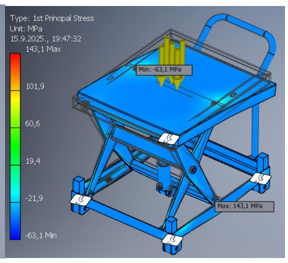
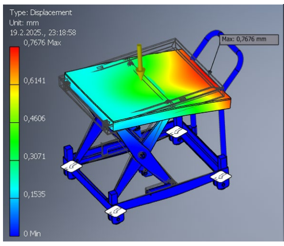
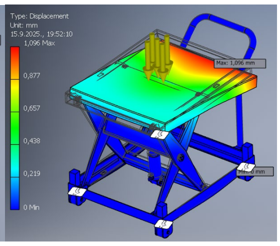
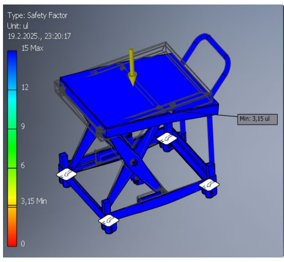
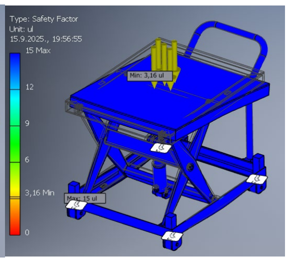
The results obtained from the static analysis in Autodesk Inventor provide a detailed insight into the behavior of the structure under load. Based on these results, critical areas requiring additional reinforcement or geometry modifications can be identified. Comparing the results with the allowable material limits confirms whether the structure meets load-bearing and safety requirements [10]. The obtained data serve as a basis for design optimization and cost reduction, as they allow corrections on the virtual model instead of a physical prototype. In this way, the use of CAD software, such as Autodesk Inventor, directly impacts product quality and development efficiency.
The results of stress and deformation analysis for the hydraulic lifting table in the lower position are presented in Table 1, showing the results for force load (1) and pressure load (2).

Table 1. *Static analysis results for the lowered position of the structure*

Static analysis results for the lowered position of the structure				
Characteristic	**Load image**		**Values**	**Comment**
	1. Force method	*2. Pressure method*		
1.Von Mises Stress (Strain in relation to the yield stress of the material):			Max. stress: 1. 99.4 MPa 2. 67.75 MPa	The max. stress was detected on the upper roller.
2. 1st Principal Stress			Max. stress: 1. 75.42 MPa 2. 62.32 MPa	The max. stress was detected on the upper roller.
3.Displacement (Deflection - deformation)			Max. deflection: 1. 0.437 mm. 2. 0.42 mm	The max. deflection was detected in the middle of the platform. Max. allowed deflection: 3 mm.
4.Safety Factor:			Min. safety factor: 1. 3.57 2. 3.54	The min. safety factor for the force method was detected in the upper roller zone, while for the pressure method it was in the upper scissor shaft (limit: 3).

The results of stress and deformation analysis for the hydraulic lifting table in the upper position are presented in Table 2, showing the results for force load (1) and pressure load (2).

Table 2. *Static analysis results for the raised position of the structure*

Static analysis results for the upper position of the structure				
Characteristic	**Load image**		**Values**	**Comment**
	1. Force method	*2. Pressure method*		
1.Von Mises Stress (Strain in relation to the yield stress of the material):			Max. stress: 1. 136.5 MPa 2. 165.5 MPa	The max. stress was detected on the lower roller.
2. 1st Principal Stress			Max. stress: 1. 106.8 MPa 2. 143,1 MPa	The max. stress for the force load method was detected on the upper roller, while for the pressure load method it was detected on the lower roller.
3.Displacement (Deflection - deformation)			Max. deflection: 1. 0.76 mm. 2. 1,1 mm	The max. deflection was observed on the platform edge farthest from the scissors mechanism (limit: 3 mm).
4.Safety Factor:			Min. safety factor: 1. 3.15 2. 3,16	The min. safety factor was detected in the upper roller zone. Allowed min. safety factor: 3.

The stress and deformation analysis of the hydraulic lifting table shows that the structure meets all safety and functional requirements. In the first case, with a concentrated force of 5000 N, maximum stress and deflection remain within allowable limits. Similar results were obtained under a distributed load simulating a 500 kg mass on the work platform.
Comparing the lower and upper positions of the hydraulic mechanism, stress and deflection values differ slightly, but in both positions the structure satisfies the required safety factor and maximum deflection limits. These results indicate that the table design provides sufficient strength and stability.

CONCLUSION

This paper presented the development process of a hydraulic lifting table structure using Autodesk Inventor. To obtain a structure that meets all required criteria, stress analysis was performed multiple times, identifying critical components. These components were optimized by modifying dimensions or materials; for example, the rollers were made from stronger steel with additional heat treatment, improving their mechanical properties.

The final static calculation is presented, in which the structure was already optimized to satisfy the prescribed safety factor and allowable deformations. Stress and deformation analyses were conducted for two types of loads: a concentrated force (Force load) and a distributed load over the platform (Distributed load). In both cases, the structure meets the minimum requirements regarding safety factor and maximum deflection. In addition, kinematic simulations of the hydraulic table's movement were performed to verify that all components function correctly throughout the lifting range. The results, shown through principal stresses, maximum deformation, and safety factor, confirm that the structure can safely carry a load of 500 kg.

The use of CAD software enabled detailed simulation of the mechanism's behavior under load, eliminated potential errors from manual calculations, and significantly accelerated the design and verification process. As a direction for future research, further optimization of the structure is suggested by reducing material thickness or using lighter materials only in areas where the safety factor is significantly higher than 3, which would result in production savings and a reduction of the overall mass of the hydraulic lifting table.

REFERENCES

[1] Budynas, R.G., Nisbett, J.K., *Shigley's Mechanical Engineering Design*, 9th Edition, McGraw-Hill, New York, 2011.
[2] Norton, R.L., *Machine Design: An Integrated Approach*, 5th Edition, Pearson, New Jersey, 2013.
[3] Autodesk Inc., *Autodesk Inventor Professional 2022 – Help Documentation*, Autodesk, 2022.
[4] Beer, F.P., Johnston, E.R., DeWolf, J.T., Mazurek, D.F., *Mechanics of Materials*, McGraw-Hill, New York, 2015.
[5] Hibbeler, R.C., *Mechanics of Materials*, 10th Edition, Pearson, Boston, 2017.
[6] Tabaković, S., *Simulacija kinematike virtuelnog prototipa i primjena CAD/CAE alata*, Mašinski fakultet Univerziteta u Beogradu / Centar za razvoj, Beograd, Srbija, 2016.
[7] Pahl, G., Beitz, W., Feldhusen, J., Grote, K.-H., *Engineering Design: A Systematic Approach*, 3rd ed., Springer, London, 2007.
[8] Kovacevic, D., *MKE modeliranje u analizi konstrukcija*, Belgrade, 2006.
[9] Basa, B.B., *Primjena CAD/CAM sustava u konstrukciji i obradi kućišta motora*, diplomski rad, Fakultet strojarstva i brodogradnje, Sveučilište u Zagrebu, Zagreb, Hrvatska, 2017.
[10] Karagülle, H., *Investigation on the design of double-stage scissor lifts using FEM*, Machines (MDPI) / European research, 2023.