Adıyaman University
Journal of Engineering Sciences
Adyü J Eng Sci 2026;13:e260107
Adıyaman Üniversitesi
Mühendislik Bilimleri Dergisi
Adyü Müh Bil Derg 2026;13:e260107
https://doi.org/10.54365/adyumbd.1775815
Research Article / Araştırma Makalesi

Structural Analysis and Topology Optimization of a Mobile Robot Chassis for STEM Education

STEM Eğitimi için Mobil Robot Şasisinin Yapısal Analizi ve Topoloji Optimizasyonu

Zeynep Yazar[1] **Yusuf Coşkun**[2] * **Lale Özyılmaz**[3]

[1] Yıldız Technical University-Faculty of Science and Letters-Department of Mathematics-Türkiye
[2] Istanbul Health and Technology University-Faculty of Engineering and Natural Sciences-Department of Mechatronics Engineering-Türkiye
[3] Istanbul Gelişim University, Faculty of Applied Sciences, Department of Aviation Electrical and Electronic Engineering - Türkiye

Abstract

This study investigates the structural analysis and topology optimization of a Theo Jansen mechanism-based mobile robot chassis, developed to help primary school students learn engineering fundamentals. Finite element analyses were performed in ANSYS, including static and modal analyses, followed by topology optimization to reduce mass while preserving strength. The pre-optimization model showed 0.2597 mm deformation and 3.5248 MPa stress, whereas the optimized design had 0.4885 mm deformation and 3.7745 MPa stress. Optimization reduced mass by 37% in simulation (15.625 g to 9.8925 g) and 26.3% after 3D printing (10.31 g to 7.60 g). Modal analysis revealed six natural frequencies, used to guide motor selection and avoid resonance. Results were simplified for students through ratio, proportion, and percentage exercises, supported by visuals and models. Future work includes applying the activities to larger groups and evaluating learning outcomes using artificial neural networks and fuzzy logic.

Keywords: *Stem Education, 3D Printing, Topology Optimization, Finite Elements Analysis*

Öz

Bu çalışma, ilköğretim öğrencilerinin mühendislik temellerini öğrenmelerine destek olmak amacıyla tasarlanan Theo Jansen mekanizmalı mobil robot şasisinin yapısal analizini ve topoloji optimizasyonunu ele almaktadır. ANSYS ortamında sonlu elemanlar yöntemiyle statik ve modal analizler yapılmış, ardından topoloji optimizasyonu ile kütle azaltımı sağlanmıştır. Optimizasyon öncesinde model 0.2597 mm deformasyon ve 3.5248 MPa gerilme göstermiştir; optimizasyon sonrası ise deformasyon 0.4885 mm'ye, gerilme 3.7745 MPa'ya yükselmiştir. Kütle, simülasyonda %37 (15.625 g → 9.8925 g), 3D baskıda ise %26.3 (10.31 g → 7.60 g) azalmıştır. Modal analizde elde edilen altı doğal frekans, motor seçiminin rezonansa girmeyecek şekilde yapılmasına rehberlik etmiştir. Sonuçlar, öğrencilere oran, orantı ve yüzde hesaplarıyla basitleştirilerek; görseller ve fiziksel modellerle desteklenmiştir. Gelecek çalışmalarda etkinliklerin daha geniş öğrenci gruplarında uygulanması, öğrenme çıktılarının yapay sinir ağları ve bulanık mantık ile değerlendirilmesi planlanmaktadır.

Anahtar Kelimeler: *Stem Eğitimi, 3D Baskı, Topoloji Optimizasyonu, Sonlu Elemanlar Analizi*

* Corresponding e-mail (Sorumlu yazar e-posta): yusuffcoskun23@gmail.com
Received (Geliş Tarihi):01.09.2025, **Accepted (Kabul Tarihi):** 13.01.2026

1. Introduction

With the advent of Industry 4.0, mobile robots have become widely used in the manufacturing, logistics, service, and defense sectors, evolving into fundamental components of autonomous systems [1–4]. The carrier chassis, which forms the physical infrastructure of these robots, not only provides structural support but also ensures the mechanical strength required for the safe and efficient execution of dynamic tasks such as acceleration, changing direction, and carrying loads. Increasing demands for agility and load capacity make it essential to optimize these chassis structures to be both lightweight and highly durable. In this context, the holistic integration of structural analyses into engineering design processes has become of great importance [5–7]. The structural components of mobile robots must maintain their reliability not only under static loads but also under environmental and operational effects. In the literature, numerous studies have been conducted on the structural evaluation of mobile robot chassis. Most of these studies have been carried out using the finite element method (FEM) and have primarily focused on static and modal analyses. For example, Dong et al. [8] performed static and modal analyses on a robot chassis exposed to electromagnetic effects. Rajurkar et al. [9] compared structural stiffness and deformation behavior in analyses conducted with different material configurations. Chen et al. [10] validated numerical modal analyses with experimental data, analyzing and confirming the structure's tendency to enter resonance. Similarly, Guoqing et al. [11] conducted FEM-based modal and static analyses on the chassis of a tracked test vehicle and proposed improvements to enhance the dynamic performance of the structure. Related studies typically examine stiffness and deformation under static loads and identify natural frequencies via modal analysis; some validate modes experimentally. However, they seldom extend the evaluation to transient operating conditions or re-assess modal characteristics after mass-reducing topology optimization. As a result, the impact of optimization on resonance margins and the practical translation of results to instructional use remains only partially addressed in the literature.

While static analyses determine the stresses and deformations that occur under constant loads, modal analyses identify the natural frequencies of the system and the corresponding vibration modes, allowing for a preliminary assessment of resonance risk [8]. Modal analysis is particularly critical in vibration-sensitive systems, as it enables the determination of frequency ranges that ensure structural safety. However, in their operating environments, mobile robots are exposed not to constant loads but to time-varying transient loads such as acceleration, sudden stops, changes in direction, or surface roughness [12,13]. Such load conditions highlight the necessity of transient analyses, which determine the dynamic responses of the system. These conditions cause time-dependent transient deformations on the chassis and directly affect the structural reliability of the system. Nevertheless, the current literature reveals that transient analyses are addressed only in a limited number of studies and are often neglected. In particular, the interaction between the natural frequencies identified in modal analyses and the transient deformations caused by time-dependent loadings has not been sufficiently investigated [14–16]. This disconnect between modal and transient analyses can lead to the resonance risk being overlooked. In recent years, topology optimization has become a frequently preferred method for reducing the weight of carrier chassis [17]. This method increases energy efficiency and reduces manufacturing costs by decreasing the mass of the chassis. However, it is often observed that newly optimized structures are tested only under static conditions, with their dynamic performance being overlooked [18,19]. Some studies have shown that structural optimization can significantly alter the natural frequencies of a system, potentially increasing the risk of resonance [20–22]. These findings indicate that the dynamic behavior of optimized structures must also be analyzed.

In this context, the primary aim of this study is to redesign the carrier chassis of a mobile robot—developed to enhance primary school students' engineering fundamentals and algorithmic thinking skills—using topology optimization in conjunction with static and modal analyses, and to quantitatively present the structural performance differences that emerge throughout this process. In addition to practical implementations, the mathematical foundation of topology optimization is based on compliance minimization and density-based methods such as the Solid Isotropic Material with Penalization (SIMP) approach. The optimization seeks to minimize structural compliance $C = F^T u$ subject to a volume fraction constraint, ensuring stiffness retention under a prescribed load. Previous studies mainly focused on lightweight design and frequency constraints, yet most neglected educational applications. Hence, this work positions itself by integrating the theoretical principles of topology optimization with an educational context, bridging engineering design and STEM pedagogy.

2. Literature Survey

STEM-based educational practices play a significant role in enhancing algorithmic thinking and problem-solving skills among primary school students. In particular, computational thinking is considered one of the essential 21st-century skills and can be reinforced through interdisciplinary applications. In this context, Liu et al. [23] demonstrated that reverse engineering pedagogy significantly improves the computational thinking skills of primary school students and is more effective compared to traditional lecture-based methods. Similarly, Mensan et al. [24] revealed that unplugged STEM activities implemented in rural areas enhance both algorithmic thinking and scientific inquiry. In a study conducted by Chalmers [25], it was found that classroom projects using commercially available robotics kits not only improved teachers' pedagogical competencies but also had a positive impact on students' computational thinking skills.

In another application using the Scratch program, it was observed that computational thinking strategies improved the geometry learning performance of Colombian students [26]. Similarly, game design, storytelling, and coding-based activities have been reported to enhance both motivation and critical thinking skills among primary school students [27]. Studies conducted in Türkiye, on the other hand, have mostly focused on science-oriented STEM activities. A systematic review covering the years 2019–2023 revealed that most of the studies during this period were conducted at the middle school level, with relatively few studies targeting primary school students [28]. With the adaptation of the international assessment tool TechCheck-2 into Turkish, it has become possible to validly and reliably evaluate the computational thinking skills of primary school students [29]. Furthermore, a study on teachers' beliefs about programming and computational thinking found that teachers in the social sciences field exhibited lower self-efficacy in these areas and required greater support [30]. Another study conducted with mathematics teachers identified a significant relationship between computational thinking and STEM self-efficacy levels; however, it also revealed that teachers' algorithmic thinking skills were low [31]. In this context, our engineering workflow (analysis-optimization-validation) is intentionally coupled with measurable classroom tasks, thereby connecting optimization outcomes to computational-thinking objectives reported in prior STEM studies.

3. Materials and Methods

To support the development of engineering-based thinking skills among primary school students, a mobile robot design that is both easy to assemble and suitable for educational environments was developed. In order to minimize difficulties during the mechanical assembly process, the use of complex fastening elements such as screws and nuts was largely avoided. Instead, specially designed connectors manufactured with 3D printers were utilized, which simplified the assembly and made it more accessible for students.

The designed robot was based on the Theo Jansen mechanism, inspired by natural walking movements, and was structured to help students understand the fundamental principles of mechanical systems through hands-on interaction. In this context, static, modal, and transient load analyses of the robot's carrier chassis were conducted using ANSYS 2021 R1 software. Following these steps, topology optimization was applied, and the results were obtained through the finite element method. PLA material was used during the analyses, and the mass of the structure was calculated as 15.625 grams. The material information for the PLA material is provided in Table 1.

Table 1. PLA material properties

Property	Value	Unit
Density	1.25×10^3	kg/m^3
Young's Modulus	4.0×10^9	Pa
Poisson's Ratio	0.36	-
Yield Strength	65×10^6	Pa
Ultimate Tensile Strength	75×10^6	Pa
Printing Temperature	190-220	°C
Bed Temperature	45-60	°C

Finally, the analysis results were simplified using color gradients and visual indicators to make the data more comprehensible for students. These findings were presented through simulation-based visual educational materials, allowing students to engage with engineering concepts in an intuitive and practical way.

3.1. Static analysis

A static analysis was conducted on the carrier chassis of the robot in its pre-optimization state by applying the necessary boundary conditions, which included fixing the support points and applying the expected load values. The applied boundary conditions are shown in Figure 1, providing a clear representation of the constraints and forces considered during the analysis. This setup allowed for evaluating the structural performance of the chassis under realistic conditions prior to the optimization stage. A 15 N static load was applied to represent the approximate total weight of the motors and supporting components during operation, corresponding to the estimated load obtained from preliminary experimental measurements of the prototype.

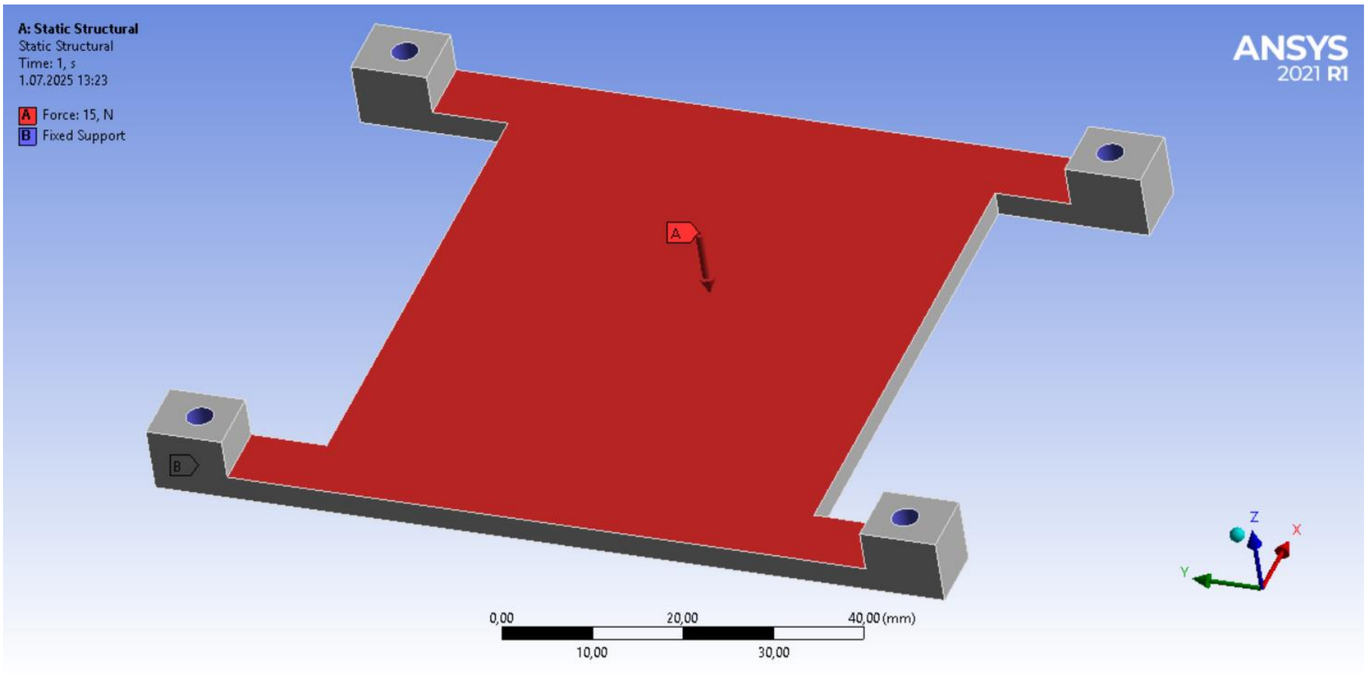

Figure 1. Boundary conditions for the chassis

After the boundary conditions were applied, the resulting deformation, stress, and strain distributions on the chassis were carefully examined to evaluate its structural behavior. The analysis revealed a total deformation of 0.2597 mm, as shown in Figure 2, while stress and strain concentrations were observed in specific regions that are critical for the overall strength of the design. These findings provided the baseline data for comparison with the post-optimization model.

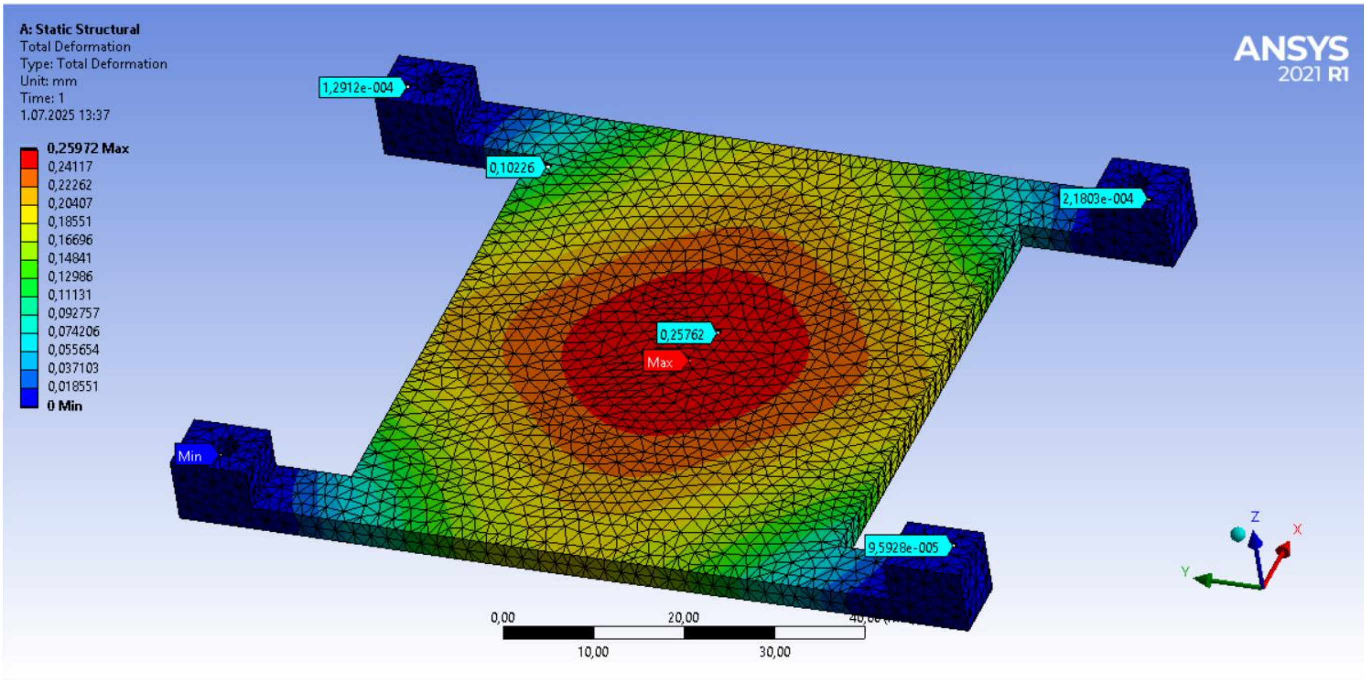

Figure 2. Total deformation of the chassis

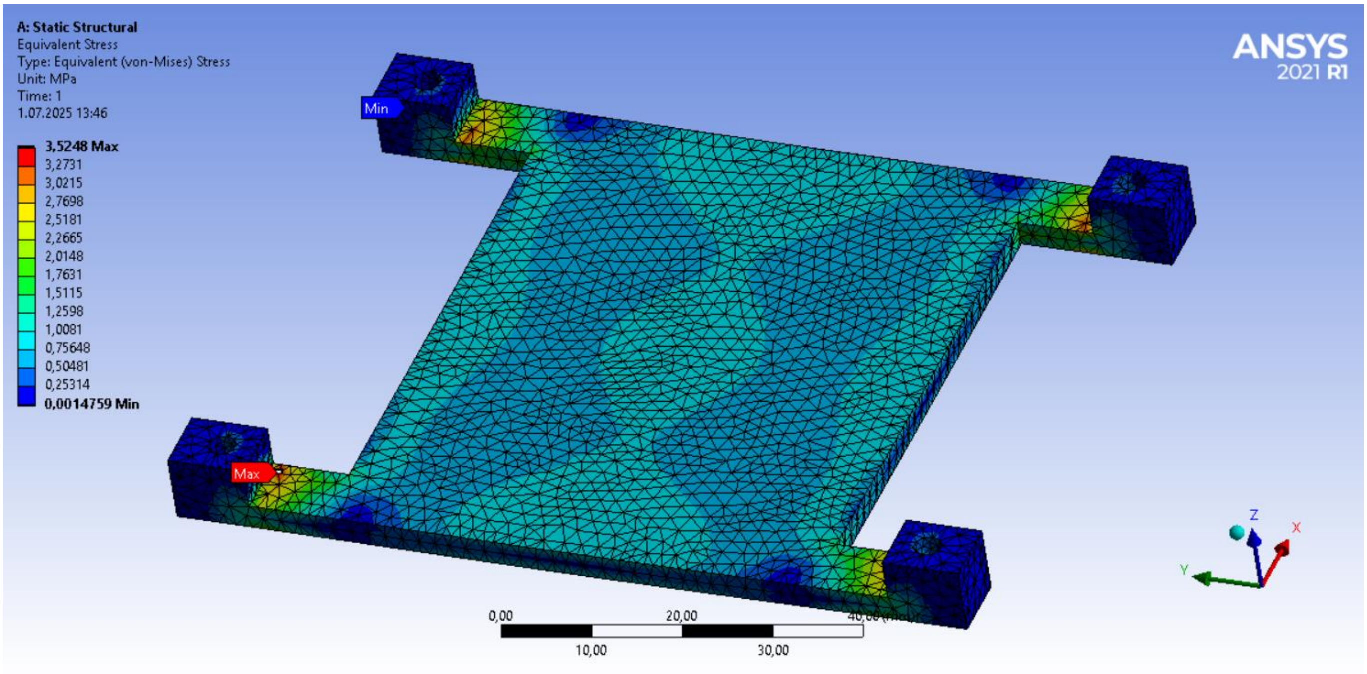

Figure 3. Stress of the chassis

The stress and strain values obtained from the analysis are presented in Figures 3 and 4, providing a detailed view of the load distribution on the chassis. The maximum stress was found to be 3.5248 MPa, while the maximum strain was calculated as 0.0023 mm/mm, indicating the material's response under the applied conditions. These results serve as important reference data for evaluating the structural integrity before optimization.

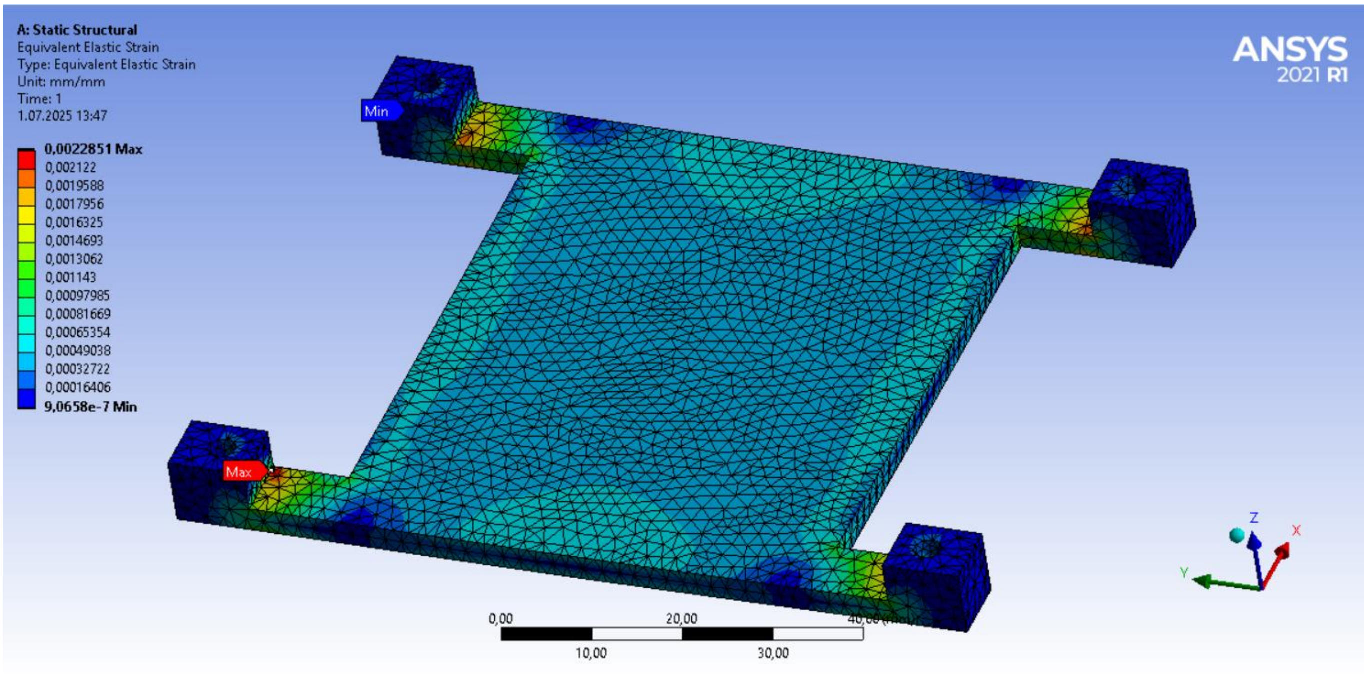

Figure 4. Strain of the chassis

3.2. Modal analysis

A modal analysis was performed to determine whether the vibrations generated by the motors to be mounted on the chassis during operation would cause any structural effects. Within the scope of this analysis, six mode shapes and their corresponding frequencies, representing the natural frequencies of the system, were obtained. These data allow for an evaluation of the chassis against resonance risk and assist in selecting motors whose vibration frequencies are compatible with these values. Only the first six modes were considered because higher-order modes exhibited frequencies above 1.6 kHz, well beyond the operational frequency range of the drive motors, and thus were neglected as they have negligible influence on practical vibration behavior. One of the total deformation results obtained from the modal analysis is presented in Figure 5.

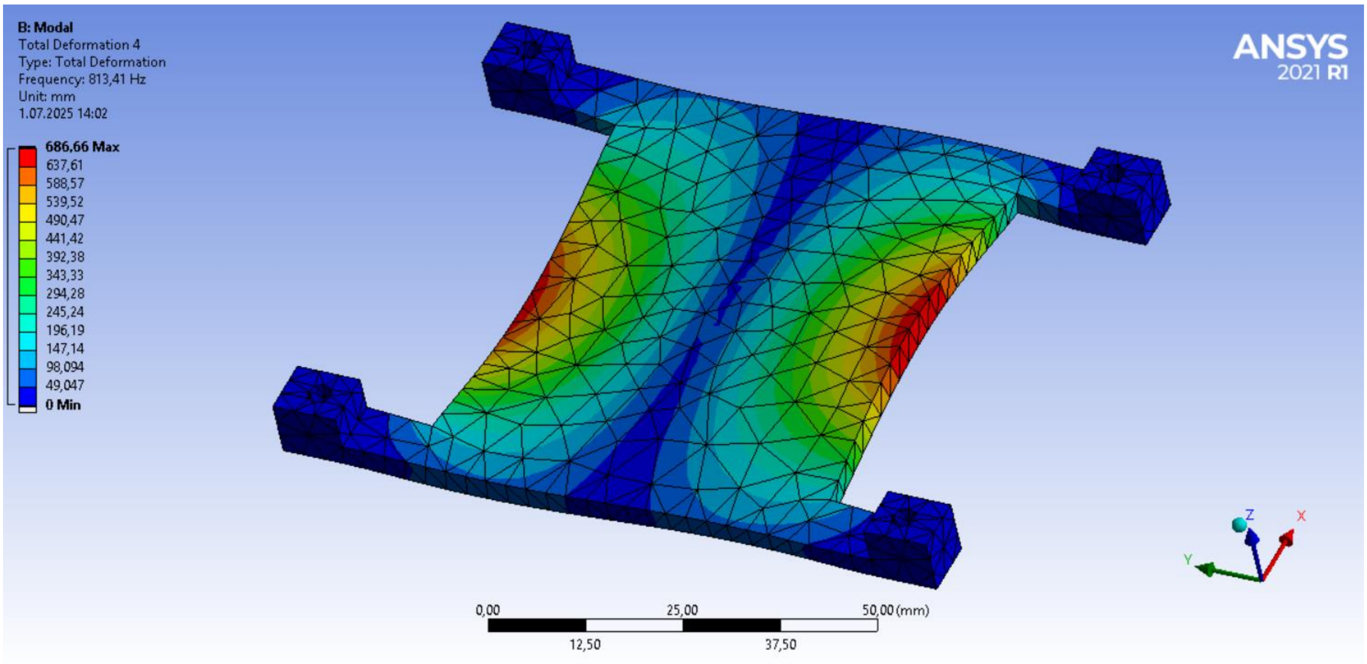

Figure 5. Result of modal analysis

As a result of the modal analysis, a total of six natural frequencies were identified. The deformation values corresponding to these frequencies are presented in Table 2.

Table 2. Results of modal analysis

Mod	Deformation	
	(mm)	(Hz)
1	370.97	378.93
2	621.81	657.66
3	686.66	813.41
4	714.22	960.42
5	290.04	1470.7
6	486.2	1610

3.3. Topology optimization

Topology optimization was performed on the chassis structure using ANSYS software, after which a new static analysis was carried out to compare the results before and after optimization. The topology optimization was conducted using the density-based SIMP method in ANSYS. The objective function was the minimization of total strain energy (compliance), subject to a 60 % mass-retention constraint. Convergence was achieved when the change in compliance between successive iterations fell below 1 %. Boundary conditions identical to the static analysis were maintained, and the optimization was terminated after 50 iterations once the design reached convergence. This comparative approach allowed the structure to be evaluated not only in terms of mass reduction but also in relation to its ability to maintain sufficient structural strength under loading conditions. The geometric configuration of the optimized part, together with the boundary conditions applied during the analysis, is illustrated in Figure 6, providing a clearer understanding of the changes introduced by the optimization process.

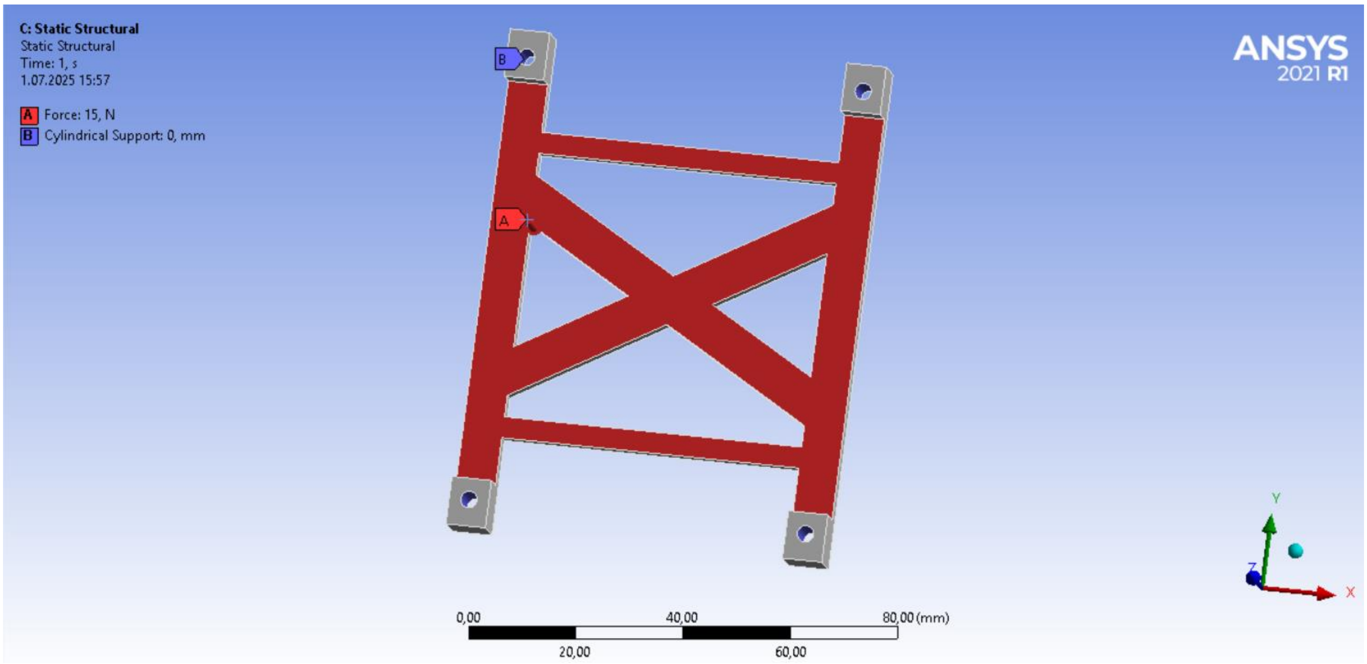

Figure 6. Model and boundary conditions created with topology optimization

According to the results of the static analysis performed on the part generated after topology optimization, the total deformation was calculated as 0.4885 mm, as shown in Figure 7.

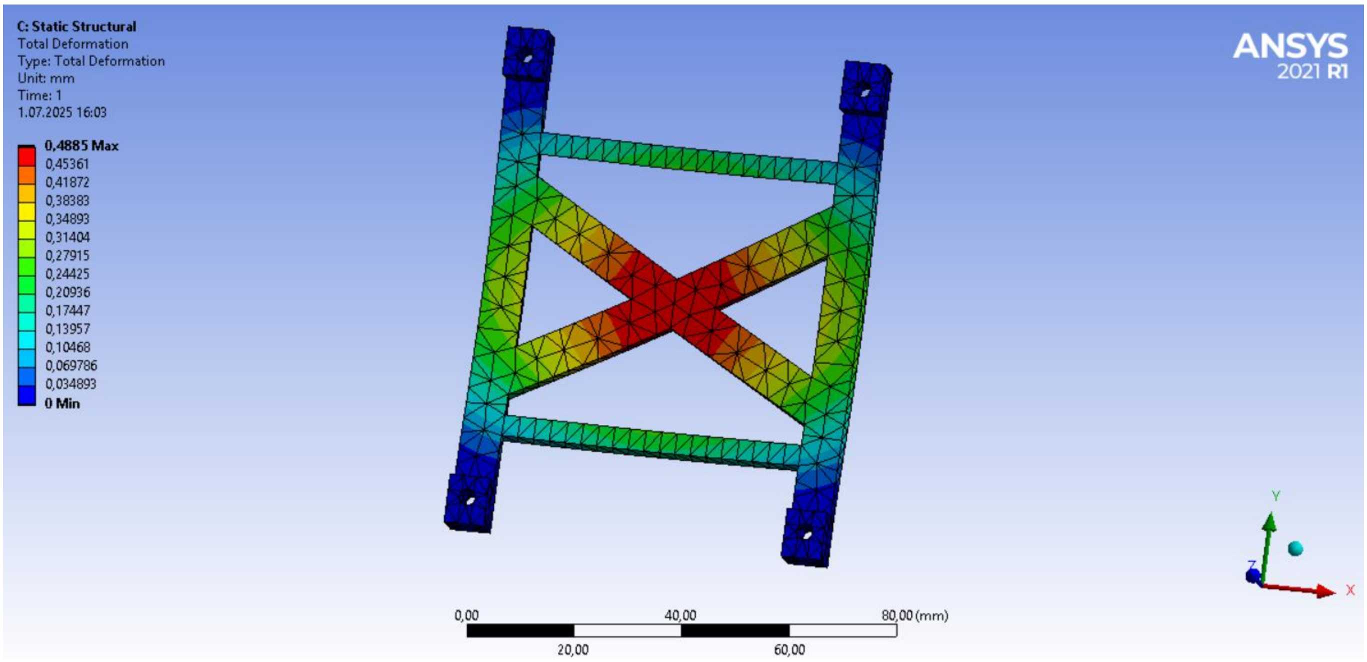

Figure 7. Total deformation amount after topology optimization

The stress and strain values of the model generated after optimization were also calculated. The maximum stress was found to be 3.7745 MPa, as presented in Figures 8.

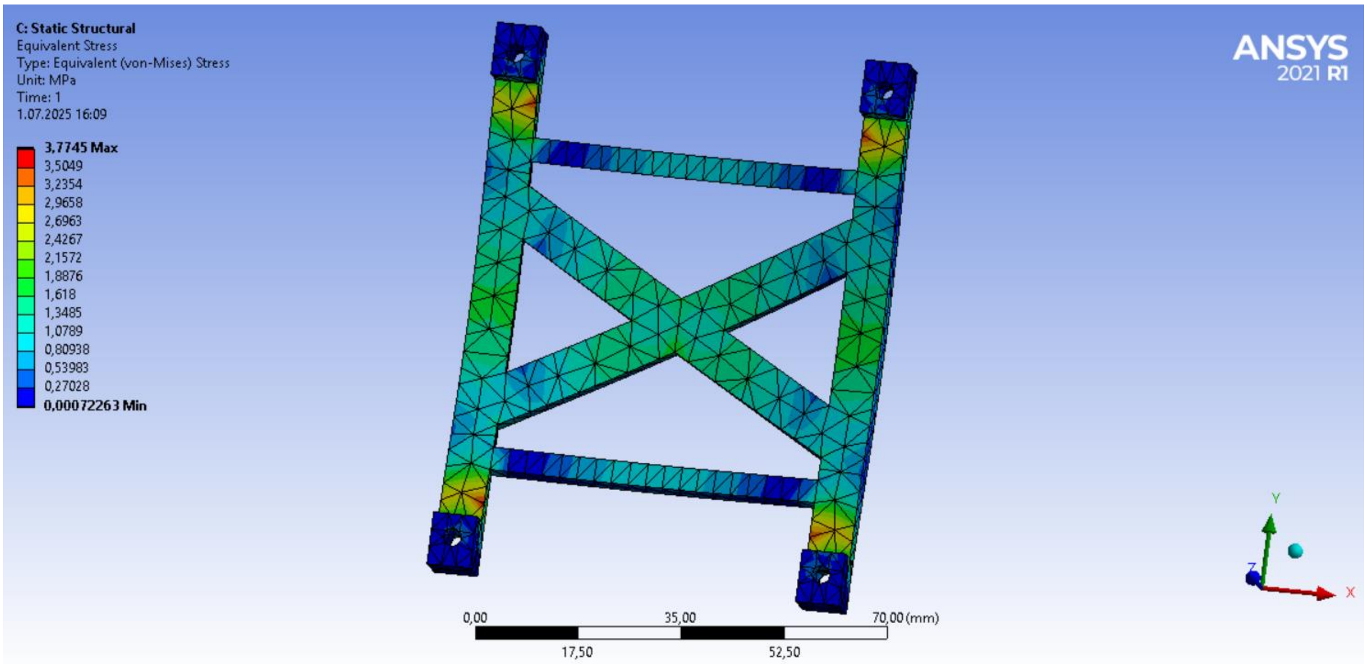

Figure 8. Stress amount generated after topology optimization

A detailed comparison of the static analysis results obtained before and after the application of topology optimization, including the changes in total deformation, stress, and strain values, is presented in Table 3 to illustrate the structural performance differences between the two design stages. The factor of safety (FOS) for the optimized design was calculated to evaluate the structural reliability of the PLA material under the applied loading conditions. The FOS is defined as the ratio of the material's yield strength to the maximum von Mises stress obtained from the finite-element analysis. Considering the yield strength of PLA as 60 MPa and the maximum stress after optimization as 3.77 MPa, the calculated safety factor was approximately 15.9. This high value indicates that the optimized chassis remains well within the elastic region of the material and can safely withstand potential overloads or minor assembly imperfections during operation. Consequently, the design ensures sufficient structural robustness while achieving the targeted weight reduction through topology optimization.

Table 3. Comparison table of static analysis

Analysis	Before Topology Optimization	After Topology Optimization	Percentage Change (%)
Total Deformation, mm	0.2597	0.4885	88
Stress, Mpa	3.5248	3.7745	7
Strain, mm/mm	0.0023	0.002348	2

The maximum stress of 3.7745 MPa was adopted as a practical design limit based on the expected load conditions of the educational robot. Since the structure is intended for low-speed operation and light payloads, this value represents a conservative stress level that ensures mechanical reliability while achieving significant mass reduction through topology optimization. Topology optimization was performed on the chassis structure using ANSYS software, followed by a new static analysis to compare the results before and after optimization. According to the optimization results, although mass reduction was achieved, no adverse effect on the structural strength of the part was observed. This evaluation revealed the performance of the structure in terms of both mass reduction and structural strength. The chassis mass, which was 15.625 grams before optimization, was reduced to 9.8925 grams after optimization, resulting in approximately a 37% weight reduction.

3.4. Printing with a 3D printer

Upon completion of the analyses, both the pre- and post-topology optimization designs of the robot chassis were produced using 3D printing technology with PLA (Polylactic Acid) material. During the production process, a 20% infill density, 0.2 mm layer thickness, 45 °C bed temperature, and 220 °C nozzle temperature were used. After physical production, the pre-optimization design was measured at 10.31 grams, while the post-optimization design weighed 7.60 grams. These results indicate that topology optimization achieved approximately a 26.3% reduction in chassis mass. The produced physical models were utilized not only to allow the mass difference between the pre- and post-optimization designs to be observed directly but also to enable a visual and tactile evaluation of the structural changes in detail, providing a tangible means for comparing the two design stages. Specifically, Figure 9 presents the 3D-printed version of the part modeled before optimization, while Figure 10 illustrates the physical print of the design obtained after the optimization process.

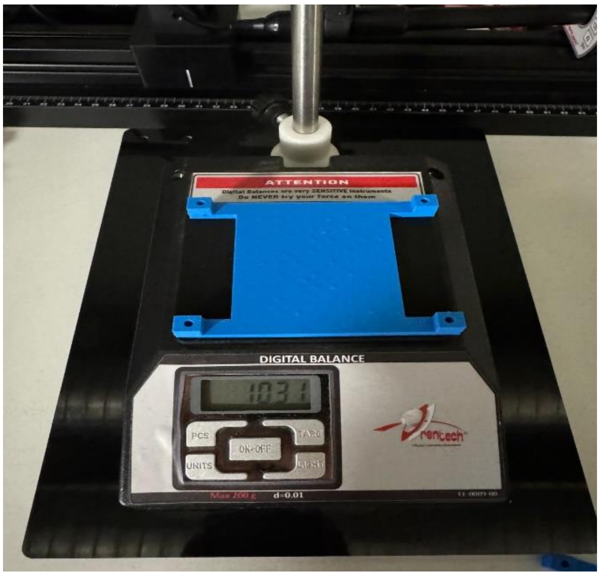

Figure 9. The 3D-printed version of the robot chassis modeled prior to topology optimization

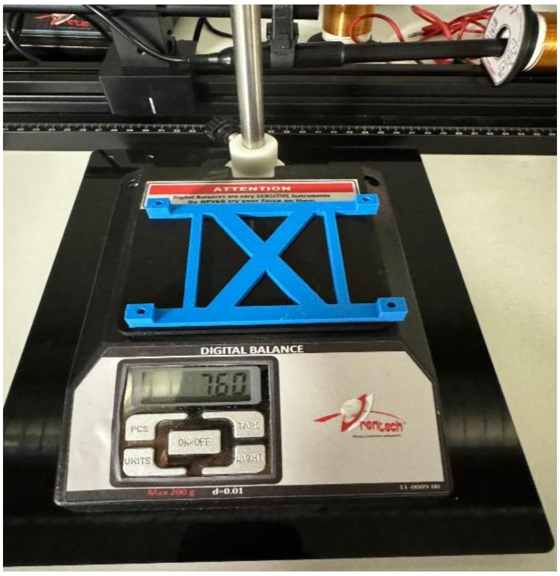

Figure 10. The robot chassis design obtained after topology optimization, produced using a 3D printer

A slight difference was observed between the 37% mass reduction obtained in the simulation environment and the 26.3% reduction measured after physical production. The primary reason for this discrepancy is that, in the finite element analysis, the part was assumed to be fully solid, whereas in the 3D printing process, production-related parameters such as a 20% infill density were applied. In addition, internal structure arrangements automatically generated by the printer software during production and material tolerances also contributed to this difference.

4. Educational Integration of Results

The engineering analyses and physical production outputs obtained in this study were not only presented as technical performance evaluations but were also restructured as instructional materials for primary school students. Adapted to the students' age and cognitive levels, the analysis results were visualized using color gradients, with critical regions of stress and deformation highlighted, as exemplified in Figure 11.

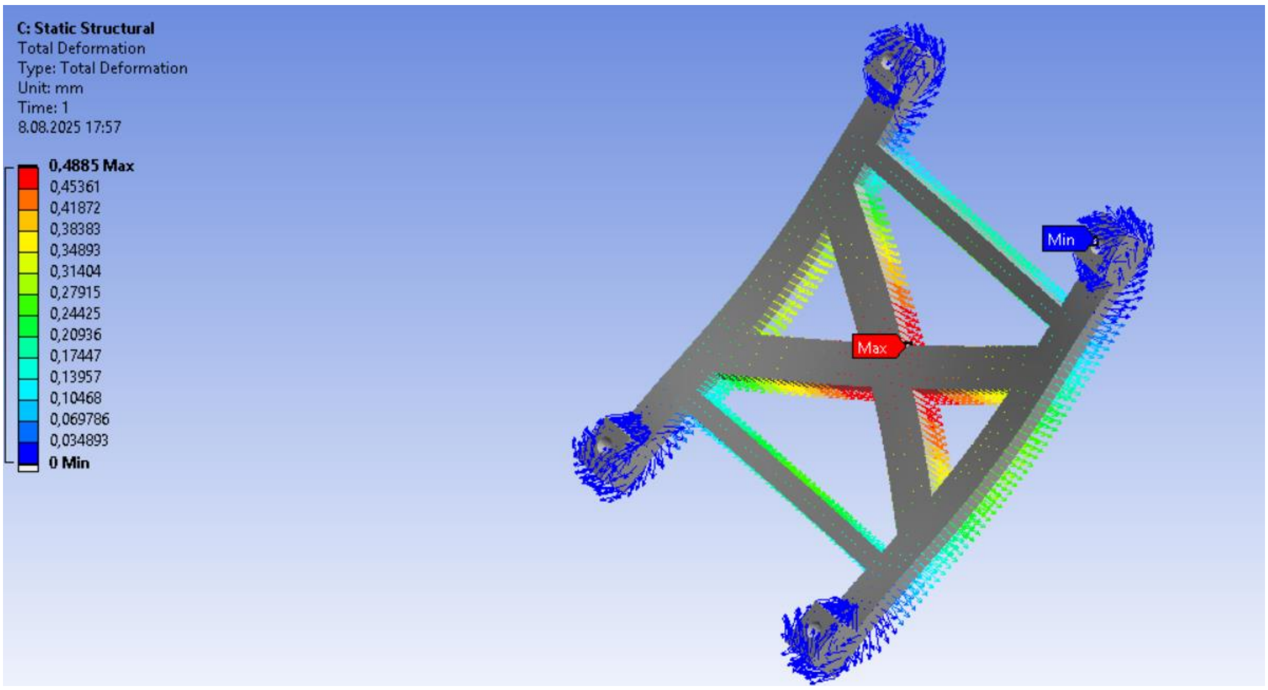

Figure 11. Visual work prepared for students

The mass difference and strength data obtained in the simulation environment were supported by the comparison of physically produced parts, enabling students to relate abstract data to tangible experiences. In particular, the engineering calculations were presented in conjunction with basic mathematical concepts such as ratio–proportion and percentage calculation. For example, the information that the mass decreased from 15.625 grams to 9.8925 grams after topology optimization was discussed with the students through questions such as "How much did the mass decrease?" and "What is the percentage reduction?". As a result, the 37% weight reduction was calculated together, ensuring active student participation. Similarly, when the mass of the physical models decreased from 10.31 grams to 7.60 grams after printing, the 26.3% reduction was calculated collaboratively with the students, thereby improving their inference and ratio formulation skills. For such calculations, the students recorded and evaluated their results as shown in Table 4.

Table 4. Table used by students for mathematical calculations

Condition	Amount	
	(gr)	(%)
Before Simulation	15.625	-
After Topology Optimization (Simulation)	9.8925	37%
Before 3D Printing	10.31	-
After 3D Printing	7.60	26.3%

The advantages of topology optimization were presented in a scenario as "reducing weight by removing unnecessary parts while maintaining strength" and discussed with the students using examples. Discussions on which regions of the part might have been removed and the possible reasons for their removal were conducted to encourage critical thinking.

Through all these approaches, students were enabled to apply engineering concepts, algorithmic thinking, and basic mathematical operation skills in an integrated manner, thereby creating a STEM-based comprehensive learning environment. In this respect, the study is considered to provide not only a technical contribution but also an educational one.

5. Conclusion and Recommendations

The results obtained in this study are consistent with previous findings in the literature [6, 17, 21], which reported mass reductions between 30 % and 40 % with negligible stress increases. The 37 % simulation-based and 26 % experimental mass reductions achieved here confirm the reliability of the applied optimization methodology. Unlike most prior works focusing solely on industrial applications, this study extends the discussion to an educational framework, highlighting the pedagogical potential of topology optimization in STEM learning environments. This study encompasses the structural analyses and topology optimization of a mobile robot chassis with a Theo Jansen mechanism, developed to support primary school students in learning engineering fundamentals. In the static analyses, pre-optimization results showed 0.2597 mm deformation and 3.5248 MPa stress, while post-optimization results indicated an increase in deformation to 0.4885 mm and a stress value of 3.7745 MPa. In the simulation environment, a 37% mass reduction was achieved (from 15.625 g to 9.8925 g), whereas after 3D printing, the reduction was 26.3% (from 10.31 g to 7.60 g), with no adverse effect observed on structural strength. The natural frequencies obtained from the modal analysis were evaluated to prevent resonance risk during motor selection.

In future studies, the activities will be conducted with a larger group of students, and the extent to which these engineering-based trainings are understood will be assessed through exam-like evaluation methods. The collected data will be analyzed using artificial neural networks (ANN) and fuzzy logic methods, thereby quantitatively revealing the relationship between students' conceptual learning levels and their practical performance. This approach will enhance both the technical and pedagogical contributions, enabling the development of the study as an innovative model in education.

Acknowledgments

This study was supported by the TÜBİTAK under the 2209-A – Research Project Support Programme for Undergraduate Students, Project No. "1919B012406034".

Authors' Contribution

All authors contributed equally to the execution of the project work and the preparation of the manuscript.

The Declaration of Conflict of Interest/ Common Interest

No conflict of interest or common interest has been declared by authors.

References

[1] Demir N, Sucuoğlu H, Böğrekci İ, Demircioğlu P. Topology optimization of mobile transportation robot. Int J 3D Print Technol Digit Ind. 2021.
[2] Darminto HWA. Advancements in cooperative mobile robots control strategies for large-scale material transport. KINETIK: Game Technol Inf Syst Comput Netw Comput Electron Control. 2024;9(4):317–326.
[3] Marques B, Junqueira G, Alves J, Pedrosa E. Mobile robots meet augmented reality technologies: transforming human-robot interaction in industry 4.0 scenarios. In: Companion of the 2024 ACM/IEEE Int Conf Human-Robot Interaction; 2024 Mar; pp. 740–744.
[4] Ghodsian N, Benfriha K, Olabi A, Gopinath V, Arnou A. Mobile manipulators in industry 4.0: a review of developments for industrial applications. Sensors. 2023;23(19):8026.
[5] Wang Y, Zhang X, Wu X. Dynamic numerical computation and optimum analysis of robot chassis using finite element method. Key Eng Mater. 2008;392–394:25–29.
[6] Coşkun Y, Aydın DC, Aktoz H, Özyılmaz L. Finite element analysis and production of a balance robot designed to be used in the field of advertising. Int J 3D Print Technol Digit Ind. 2025;9(1):114–121.
[7] Koçak MT, Bayraklılar MS. Mechanical shaft optimization: a study on static structural analysis and topological optimization in ANSYS. Int J 3D Print Technol Digit Ind. 2021;7(3):541–549.
[8] Dong X, Li J, Zhou D, Li J, Li Y. Finite element analysis of mobile chassis for the maintenance robot with charged used in substation under idle condition. IOP Conf Ser Earth Environ Sci. 2019;233(5):052024.
[9] Rajurkar A, Dangra K, Deshpande A, Gosavi M, Phadtare T, Gambhire G. Comparative finite element analysis of a novel robot chassis using structural steel and aluminium alloy materials. Eng Innov. 2023;7:61–73.
[10] Chen K, Yuan Y, Zhao B, Jin X, Lin Y, Zheng Y. Finite element modal analysis and experiment of rice transplanter chassis. Int J Agric Biol Eng. 2022;15(5):91–100.
[11] Guoqing G, Shaojun L, Yihui Z, Lingping C. Finite element analysis on chassis of tracked test vehicle. In: 2012 Int Conf Comput Distributed Control Intell Environ Monit; 2012 Mar; pp. 334–337.
[12] Hong CW. Dynamic simulation of road vehicle performance under transient accelerating conditions. Proc Inst Mech Eng Part D J Automob Eng. 1996;210(1):11–21.
[13] Pan TC, Li J. Dynamic vehicle element method for transient response of coupled vehicle-structure systems. J Struct Eng. 2002;128(2):214–223.
[14] Wang C, Wang Z, Hu H, Li L. Innovative design and kinematic characteristics analysis of floating mobile chassis of inspection robot. Machines. 2022;11(1):24.
[15] Gagnon M, Dollon Q, Nicolle J, Morissette JF. Operational modal analysis of Francis turbine runner blades using transient measurements. IOP Conf Ser Earth Environ Sci. 2021;774(1):012082.
[16] Egusquiza M, Valentín D, Valero C, Presas A, Egusquiza E. Transient analysis of Pelton turbine prototypes. IOP Conf Ser Earth Environ Sci. 2021;774(1):012120.
[17] Sha L, Lin A, Zhao X, Kuang S. A topology optimization method of robot lightweight design based on the finite element model of assembly and its applications. Sci Prog. 2020;103(3):1–16.
[18] Coşkun Y, Aydın M, Koçak MT. Topology optimization of snake-like robot limb using ANSYS. J Mater Electron Devices. 2023;5(1):1–8.

[19] Kusay DIB, Aydın M, Coşkun Y. Topology optimization for flexible robotic gripper using ANSYS. J Mater Electron Devices. 2024;3(1):30–36.
[20] Siqueira LO, Silva KES, Silva ECN, Picelli R. Topology optimization of stationary fluid–structure interaction problems considering a natural frequency constraint for vortex-induced vibrations attenuation. Finite Elem Anal Des. 2024;234:104137.
[21] Wei W, Qingguo T, Fengbin W, Yesen F, Shikun Z, Wenhui Z. A multi-objective topology optimization method used in simultaneous constraints of natural frequency and static stiffness. In: 2022 Asia Conf Algorithms Comput Mach Learn (CACML); 2022 Mar; pp. 7–12.
[22] Meng Z, Yang G, Wu Q, Xiao S, Li Q. Reliability-based topology optimization for fundamental frequency maximization with frequency band constraints. Mech Syst Signal Process. 2023;195:110295.
[23] Liu X, Wang X, Xu K, Hu X. Effect of reverse engineering pedagogy on primary school students' computational thinking skills in STEM learning activities. J Intell. 2023;11(2):36.
[24] Tracy M, Osman K, Majid NAA. Promoting algorithmic thinking through unplugged primary science activity in rural schools. Int J Mod Educ. 2024;6(23):793–800.
[25] Chalmers C. Robotics and computational thinking in primary school. Int J Child Comput Interact. 2018;17:93–100.
[26] Echeverría L, Cobos R, Morales M, Moreno F, Negrete V. Promoting computational thinking skills in primary school students to improve learning of geometry. In: 2019 IEEE Global Eng Educ Conf (EDUCON); 2019 Apr; pp. 424–429.
[27] Chiazzese G, Fulantelli G, Pipitone V, Taibi D. Engaging primary school children in computational thinking: designing and developing videogames. Educ Knowl Soc (EKS). 2018;19(2):63–81.
[28] Sakallı R, Usta N. STEM eğitim yaklaşımının bilgi işlemsel düşünme becerisine etkisi üzerine yapılan çalışmaların incelenmesi: sistematik literatür taraması. Sinop Univ Eğitim Fak Derg. 2024;1(1):21–45.
[29] Küçükaydın MA, Akkanat Ç. Adaptation into Turkish of the computational thinking test for primary school students. Probl Educ 21st Century. 2022;80:765–776.
[30] Günbatar MS, Bakırcı H. STEM teaching intention and computational thinking skills of pre-service teachers. Educ Inf Technol. 2019;24:1615–1629.
[31] Yitmez BG, Mol D, Kabakçı DA, Yılmaz S. Matematik öğretmenlerinin bilgi işlemsel düşünme ve STEM öz yeterlikleri arasındaki ilişkinin incelenmesi. DEU Buca Eğitim Fak Derg. 2023;58:3103–3120.