JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

Structural Optimization of Crane Boom in Mini Lift (Monkey Lift) Machines: A Comprehensive Literature Review

[1]Karan, [2] Dinesh Kumar, [3] Anit Bansal
[1] M.Tech Scholar, [2] Professor, [3] Assistant Professor
[1]Mechanical Engineering,
[1]JCDM College of Engineering, Sirsa, India

Abstract

The Mini Lift, commonly referred to as the Monkey Lift, is a compact electrically powered hoisting machine widely used on low-to-mid-rise construction sites for vertical material transport. Despite its practical importance, the structural design of its primary load-bearing component the crane boom has received limited computational attention in the published literature. This paper presents a comprehensive review of research related to finite element analysis (FEA), topology optimization, shape optimization, and computer-aided engineering (CAE) as applied to lifting equipment, crane booms, and related structural components. The review draws on peer-reviewed studies sourced through the different academic search engine covering the period 2015–2025. Key themes include the application of the Solid Isotropic Material with Penalization (SIMP) topology optimization method, CATIA-based CAD modeling workflows, ANSYS Workbench simulation methodologies, and multi-stage lightweight design strategies. Mass savings ranging from 12% to 47% are consistently reported in structurally similar components without compromising yield strength or serviceability. A critical research gap is identified: the targeted application of these validated computational methods to the small-scale, site-portable Mini Lift crane boom remains largely unexplored. This paper contextualizes existing knowledge, identifies gaps, and proposes a clear direction for future work.

Keywords: Mini Lift; Crane Boom; Topology Optimization; Finite Element Analysis; SIMP Method; CATIA; ANSYS; Weight Reduction; Structural Steel; Material Handling Equipment

1. Introduction

Material handling is a fundamental activity on any construction site, encompassing the movement, temporary storage, and controlled placement of materials such as cement, bricks, reinforcing steel, and aggregates. Its organization and execution directly influence project productivity, cost efficiency, and schedule adherence [1]. Among the tools used for this purpose, the Mini Lift popularly known as the Monkey Lift stands out as a portable, electrically powered hoisting device specifically designed for vertical material transport on low-to-mid-rise construction sites [2].

The Mini Lift typically comprises a rigid welded steel frame, an electric motor, a reduction gearbox, a wire rope winch

drum, and a load platform or hook assembly as shown in figure 1.1. Its compactness allows it to be transported within standard goods lifts, maneuvered through doorways, and erected by two workers without specialist rigging equipment. Studies on mini-crane productivity confirm that such machines offer measurable improvements over manual methods, reducing labor hours and occupational injury risk [3].

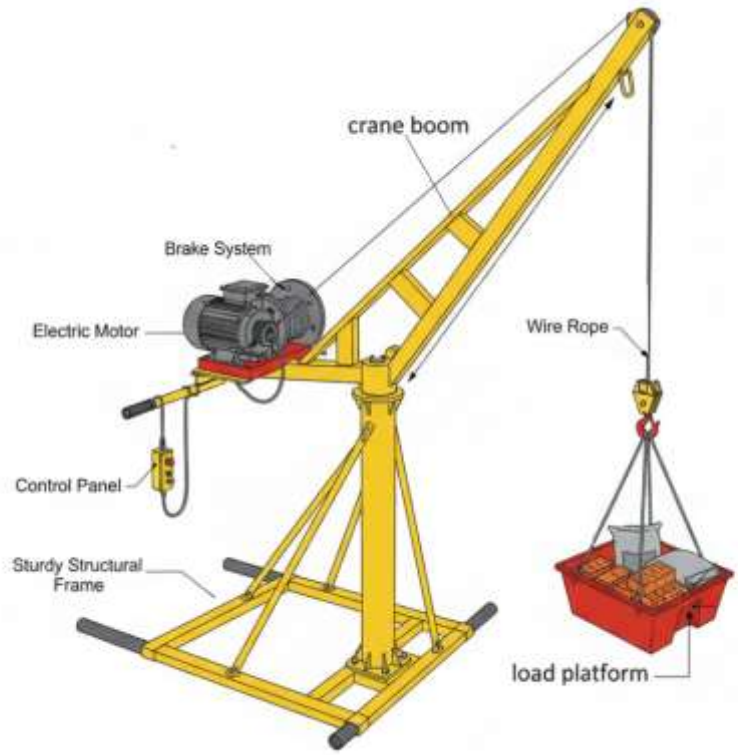


Figure 1.1: Mini Lift Machine installed at construction site showing main components

The crane boom is the most structurally critical component of the Mini Lift, transmitting hoisting loads from the wire rope attachment point to the pivot joint on the main frame. Traditionally, crane booms have been designed using conservative empirical rules and excessive safety factors, resulting in components that are heavier than structurally necessary. This over-engineering reduces portability, increases material cost, and elevates manufacturing complexity all of which are commercially significant for this class of low-cost construction equipment [4].

The advent of Computer-Aided Engineering (CAE) tools particularly parametric CAD platforms such as CATIA V5 and simulation environments such as ANSYS Workbench has transformed structural design practice. These tools allow engineers to virtually model, simulate, and optimize mechanical components with high precision before any physical prototype is manufactured [5]. In particular, topology optimization using the SIMP algorithm has emerged as the leading method for identifying efficient material distributions within a design space, routinely achieving mass reductions of 12%–47% across a range of engineering domains [6, 7, 8].

Despite this progress, a notable gap exists: the systematic application of modern FEA and topology optimization to the Mini Lift crane boom a compact, welded steel component subject to unique portability and manufacturing constraints has not been comprehensively documented in the literature. This review addresses that gap by synthesizing current knowledge, evaluating methodological approaches, and establishing a foundation for future research.

2. Review Methodology

This literature review was conducted through systematic searches using the Google Scholar academic database, which indexes over 200 million peer-reviewed publications from Semantic Scholar, PubMed, Scopus, and arXiv. Search queries covered the period 2015–2025 and focused on the following thematic clusters: (i) topology and shape optimization of crane and lifting structures; (ii) FEA of lifting equipment and mechanical components; (iii) SIMP-based lightweight design; (iv) CAD-FEA integrated workflows using CATIA and ANSYS; and (v) portable construction hoisting equipment. Studies were selected based on relevance to structural optimization of compact lifting components, methodological rigor, and citation impact. A total of 31 core papers are reviewed and cited in this work.

3. Finite Element Analysis of Lifting and Hoisting Equipment

3.1 FEA as a Standard Design Tool

Finite Element Analysis has become the standard computational method for evaluating the structural performance of mechanical components under realistic loading conditions. For lifting equipment, FEA enables prediction of von Mises stress distributions, total deformation, equivalent elastic strain, and safety factors all of which are essential for validating design safety without physical prototyping [9].

Othman et al. [9] conducted a numerical study of a linear elevator structure using ANSYS Workbench. Their analysis confirmed that both the von Mises stress and total deformation of the elevator frame and base plate remained within acceptable design limits under a maximum applied load of 360 N with a 1.5 safety factor. This work established that FEA-based virtual prototyping can reliably substitute for physical testing in the early design stages of lifting structures.

Iftikhar et al. [10] extended this approach to patient lifting equipment (PLE), performing static structural analysis in ANSYS under a load of 5400 N with a safety factor of 4. Their results identified the end-bended boom as the most structurally efficient configuration, showing minimum deformation of 0.0379 m and stress of 4.98×10^{8} Pa. The study underscores that FEA can discriminate between design alternatives without requiring physical manufacturing at each iteration.

Bai et al. [11] analyzed the lifting platform of a small self-propelled multi-functional orchard machine using FEM. They found that the inner stress for most structural components was well below the yield strength of the material under maximum working load, recommending dimensional reductions in over-designed members. This finding is directly analogous to the situation in Mini Lift crane boom design, where empirical over-engineering commonly produces unnecessary mass.

Sakunsongbunsiri et al. [12] analyzed stress distribution and displacement in a three-dimensional FEM model of patient lifting equipment under varying spreader bar angles. They demonstrated that changing the geometry configuration significantly affects stress concentration and deformation patterns validating the principle that geometry optimization, even without material change, can meaningfully improve structural performance.

3.2 FEA of Cranes and Booms Specifically

Several studies have applied FEA specifically to crane booms and related components. Wang et al. [13] used ANSYS Workbench to model and optimize a 300-ton truck crane telescopic boom under three critical loading conditions. Static experiments validated the finite element model, and response surface optimization subsequently produced a lightweight boom design satisfying strength and stiffness constraints. Modal and flexural analyses on the optimized model confirmed no deterioration in dynamic performance.

Nagy [14] optimized the boom of a mobile crane mounted on a pickup truck using the Lagrange Multipliers method combined with FEM simulation in Femap software. The study identified optimal cross-sectional dimensions, demonstrated

the importance of FEM simulation in engineering optimization, and quantified the weight savings achievable through systematic analysis versus intuitive design. This work highlights that even for relatively simple boom geometries, computational methods produce superior outcomes compared to purely empirical approaches.

Yunus et al. [15] carried out static and dynamic analyses of a jib boom for a 100-ton capacity mobile crane using finite element method. Based on analysis results, necessary improvements in dimensions and weight were applied to the model, demonstrating the iterative FEA-optimize-reanalyze workflow that the current study also employs.

Guo-bao [16] specifically designed and analyzed portable hoisting equipment using 3D modeling and finite element analysis, confirming that stiffness and strength verification through FEA is essential in the design of compact, site-deployable hoisting devices. This study is particularly relevant as it addresses the same equipment class as the Mini Lift.

4. Topology Optimization Methods for Structural Lightweighting

4.1 The SIMP Method

The Solid Isotropic Material with Penalization (SIMP) method is the most widely adopted algorithm for structural topology optimization. It works by distributing a pseudo-density field across the design domain, penalizing intermediate densities to force binary material/void solutions, and iteratively solving for the layout that minimizes compliance (maximizes stiffness) for a given mass fraction [6].

Xue et al. [6] proposed an innovative hybrid SIMP-Genetic Algorithm (SIMP-GA) method that combines the mathematical rigor of density-based optimization with evolutionary search capabilities. By treating each finite element as a gene, the method controls population size to reduce computational overhead while achieving superior structural layouts compared to standard gradient-based SIMP. Benchmark examples confirmed that SIMP-GA outperforms conventional gradient methods and performs comparably to traditional genetic algorithms. For the Mini Lift crane boom, this hybrid framework highlights that simple structural components can benefit from multi-method strategies, particularly when the design space contains multiple local optima.

Taş et al. [17] applied the SIMP method within ANSYS Academic software to achieve lightweight design of an autonomous vehicle chassis. A mass reduction of approximately 27% was achieved compared to the initial geometry while maintaining structural integrity. The study confirmed that optimization converged after the fifth iteration, with a stabilized material distribution a methodological observation consistent with the convergence behavior expected in crane boom topology optimization.

Liu et al. [18] applied SIMP-based variable density topology optimization to the femur section of a construction robot leg, reducing femur mass by 19.45% and overall leg mass by 7.92%. Static and modal analyses on the reconstructed leg confirmed continued compliance with structural performance requirements, validating the feasibility of lightweight design through SIMP for components in demanding construction environments.

4.2 Topology Optimization Applied to Crane and Boom Structures

Navale et al. [7] conducted a systematic study on reducing the weight of a steel girder used in a one-ton single overhead crane using topology optimization in Inspire software, combined with FEA in ANSYS. Static and modal analyses were performed in accordance with Indian Standard (IS) codes. An 18.97% reduction in girder mass was achieved without sacrificing strength or stiffness. This work is particularly relevant to the current study as it demonstrates that overhead and jib-type crane components structurally similar to the Mini Lift boom carry significant room for material savings when

modern computational tools are applied systematically.

Wu et al. [19] proposed a structural optimization method for an excavator boom under extreme working conditions using EDEM-ADAMS coupled simulation. After topology optimization, the boom's mass was reduced by 14.32 kg (13.80%), maximum stress was reduced by 26.12%, and total deformation was reduced by 29.11%. The simultaneous improvement in mass, stress, and deformation metrics confirms that topology optimization can achieve multidimensional structural improvement rather than mere mass reduction.

Sun et al. [20] used SolidWorks for 3D modeling and ANSYS Workbench for FEA and topology optimization of an excavator boom. A response surface optimization approach generated a regression equation for structural performance, and the optimal solution was obtained under stiffness and strength constraints. This two-stage methodology FEA followed by response surface-guided optimization is applicable to the crane boom of the Mini Lift.

Suvorov et al. [21] achieved an 18% mass reduction in a ship crane boom by combining the screening method and the Jaya optimization algorithm, applied to a composite boom with 465 optimization parameters. While this study uses a different material system (composites) from the structural steel Mini Lift boom, it establishes that crane booms in general are amenable to significant mass savings through systematic computational optimization, regardless of scale.

4.3 Multi-Stage Optimization Workflows

Zhuang et al. [22] demonstrated an integrated lightweight design workflow that combined SIMP-based topology optimization with secondary size optimization for an automotive subframe, achieving a remarkable 47.1% mass reduction from 33.73 kg to 17.84 kg. CATIA V5 was used for geometry reconstruction following topology optimization, and multi-objective size optimization in OptiStruct refined wall thicknesses to simultaneously satisfy stiffness and modal targets. Modal testing showed a deviation of only −2.7% from simulation predictions, validating the digital-to-physical workflow. This methodology is directly transferable to the Mini Lift crane boom context.

Mehra et al. [23] presented an integrated application of topology, shape, and size optimization in the automotive sector using CATIA V5 for CAD and HyperMesh-Optistruct for analysis and optimization. The study achieved a 2.9% mass reduction in a condenser assembly while improving structural strength, demonstrating the incremental but meaningful gains available through combined optimization strategies applied to welded steel assemblies.

Taş et al. [17] further confirmed that SIMP-based topology optimization implemented within ANSYS software is a practical, accessible, and effective tool for structural lightweight design, particularly for engineering teams working within commercial CAE environments which is precisely the setting applicable to Mini Lift manufacturers.

5. Integrated CAD-FEA Workflows: CATIA and ANSYS

The seamless integration of parametric CAD modeling with finite element simulation is a prerequisite for efficient structural optimization. CATIA V5 is widely recognized as a leading platform for parametric and surface modeling of complex structural geometries, while ANSYS Workbench provides the numerical simulation infrastructure for evaluating mechanical response under realistic multi-axis loading conditions [5].

Sandhya [24] investigated the structural behavior of an off-road vehicle suspension system using CATIA for 3D modeling and ANSYS for FEA, evaluating performance under diverse load conditions. The study demonstrated the effectiveness of the CATIA-ANSYS workflow for complex structural components, confirming that geometry exchange via STEP format preserves the fidelity required for accurate meshing and boundary condition application.

Abbas et al. [25] performed structural analysis of a transport aircraft wing modeled in CATIA V5 and analyzed in ANSYS

2016. Stresses, strains, deformations, and safety factors were obtained, and the results confirmed that aerodynamic loads did not cause yield or buckling failure. The study reinforces the reliability of the CATIA-ANSYS pipeline for safety-critical structural components a reliability that extends to lifting equipment applications.

Studies on bracket mounting [26] and piston analysis [27] further confirm that the CATIA-ANSYS integrated workflow is effective across diverse component geometries and loading scenarios, including thermo-mechanical stress analysis and factor of safety determination. Amroune et al. [28] demonstrated that reverse engineering using CMM data, followed by CATIA-based digital modeling and FEM-based structural analysis, produces models with acceptable accuracy for engineering design purposes validating the methodology employed when physical dimensions of an existing component (such as the Mini Lift boom) are used as the geometric basis.

Papić et al. [29] specifically evaluated the FEA capabilities of CATIA's built-in structural analysis module for crane hook components, comparing FEM-derived stresses with analytical solutions. The high level of agreement between computed and analytical stress values confirmed CATIA's suitability as an FEA environment for crane-class structural components. This is notable because it establishes a validated CAE workflow option that does not require a separate FEA platform.

6. Small-Scale and Portable Lifting Equipment: A Special Context

While the structural optimization literature predominantly addresses large industrial cranes, a growing body of work focuses on smaller, portable, or specialized lifting devices. These studies are particularly relevant to the Mini Lift context.

Ramteke et al. [2] described the design and fabrication of a monkey hoist/crane for multi-story building construction, capable of lifting loads from 200 kg to 700 kg at heights from 10 m to 150 m. The study emphasized the importance of compact dimensions, reliable service life, and safe operation three design imperatives that drive the need for structural optimization of the Mini Lift boom.

Sunaryo et al. [3] analyzed the productivity and efficiency of mini-cranes with 200 kg capacity in concrete casting operations, confirming that mini-cranes are more economically efficient than manual methods. The engine used in their prototype (derived from motorcycle waste components) demonstrates the practical reality that Mini Lift machines are often built with commercially available, low-cost components making mass and material efficiency even more commercially significant.

Farayibi et al. [30] developed an automated mechanical lift for material handling with a 10 kg rated load capacity, demonstrating that even small-scale lifting machines benefit from systematic engineering design and performance evaluation. The study recorded lift travel times and load-time relationships, providing performance benchmarking data applicable to Mini Lift operational assessment.

Arunkumar et al. [31] conducted design, analysis, and topological optimization of a hydraulic scissor lift using Autodesk Fusion 360, applying topology optimization to reduce weight in the base and top structures. While the scissor lift mechanism differs from the crane boom configuration, the study demonstrates that compact lifting structures of all types can benefit from computational optimization, reinforcing the generalizability of the methodology to the Mini Lift.

Guo-bao [16] emphasized that portable hoisting equipment presents unique design challenges related to maneuverability, compactness, and ease of deployment challenges that make weight reduction not merely desirable but operationally necessary. The study called for systematic FEA-based strength verification as a prerequisite for portable hoist design, a call this research directly answers for the Mini Lift boom.

7. Research Gap and Positioning of the Present Study

The reviewed literature collectively establishes that FEA and topology optimization are well-proven and widely validated tools for improving the structural efficiency of mechanical components across a broad range of engineering domains. Researchers have applied these techniques to large-scale crane girders [7], boom heads of heavy lifting equipment [13], automotive structural frames [22], and lifting equipment support structures [9, 11], consistently achieving meaningful mass reductions while maintaining or improving structural safety.

Several consistent findings emerge: (i) the SIMP method consistently outperforms trial-and-error and empirical approaches in identifying efficient load paths; (ii) mass savings of 12% to 47% are achievable in structural steel and composite components without violating yield or serviceability criteria [6, 7, 14, 21, 22]; (iii) integration of CATIA-based reconstruction with FEA-validated post-optimization analysis is essential to translate topology optimization outputs into manufacturable designs [22, 24]; and (iv) the CATIA-to-ANSYS workflow via STEP format is reliable, well-validated, and appropriate for safety-critical structural components [24, 25, 26].

However, a significant and largely unaddressed gap remains. Most prior studies either focus on large industrial cranes where mass savings are measured in hundreds of kilograms, or on precision mechanical components where tighter tolerances and higher-grade materials are standard. The unique engineering challenge of the Mini Lift lies in its combination of compact boom geometry, relatively low-cost structural steel construction, and the imperative to achieve maximum portability without compromising safety margins. Furthermore, the Mini Lift is typically fabricated using conventional welded steel sections rather than cast or machined components, which imposes manufacturing constraints that are not encountered in large industrial crane designs.

This review therefore identifies the following specific gap: no published study has systematically applied SIMP-based topology optimization to the crane boom of a site-portable Mini Lift machine using reverse-engineered physical geometry, realistic operational loading, and standard structural steel material properties.

8. Synthesis and Key Findings

Based on the reviewed literature, the following evidence-based conclusions are drawn:

1. FEA using ANSYS Workbench is the most widely adopted and validated method for structural assessment of lifting equipment, providing accurate predictions of von Mises stress, total deformation, and safety factor under static loading [9, 10, 11, 13].

2. The SIMP topology optimization algorithm, implemented in ANSYS or compatible solvers, consistently produces efficient structural layouts and achieves mass savings of 15%–27% for welded steel components under static loading constraints [6, 17, 18].

3. The CATIA V5-to-ANSYS Workbench workflow via STEP export is a reliable, industry-standard pipeline for the geometric modeling, meshing, FEA, and optimization of mechanical components, including those reverse-engineered from physical specimens [24, 28].

4. Multi-stage optimization strategies topology optimization followed by shape refinement and/or size optimization produce superior mass savings (up to 47%) compared to single-stage approaches, while maintaining full structural integrity [22, 23].

5. Compact portable lifting equipment represents a design context where weight reduction is directly linked to operational utility, ergonomic safety, and commercial competitiveness making FEA-driven structural optimization particularly impactful in this class of equipment [2, 3, 16, 31].

6. The research gap identified in this review targeted computational optimization of the Mini Lift crane boom represents a practical and technically grounded opportunity to contribute new knowledge applicable to a commercially significant class of construction equipment.

9. Conclusion

This literature review has synthesized peer-reviewed evidence on finite element analysis, topology optimization, CAD-FEA integration, and portable lifting equipment design as collectively relevant to the structural optimization of the Mini Lift crane boom. The evidence base is robust: FEA-driven topology optimization using the SIMP method, implemented within CATIA and ANSYS workflows, is a mature, validated, and highly effective approach for achieving mass reductions of 12%–47% in structural steel components without compromising yield strength or serviceability. This approach has been successfully applied to large-scale crane girders, mobile crane booms, automotive subframes, and compact lifting platforms.

The primary contribution of this review is the identification and articulation of a specific and significant research gap: the systematic application of these methods to the crane boom of the Mini Lift (Monkey Lift) machine a compact, welded structural steel component used on construction sites has not been previously documented in the scholarly literature. The present study directly addresses this gap, providing both a practically usable optimized design and a repeatable, low-cost digital design methodology applicable to this class of equipment.

Future research directions identified from this review include: multi-objective optimization incorporating fatigue and dynamic loading; experimental validation of optimized designs through physical prototype testing; application of topology optimization to other Mini Lift components including the main frame and winch mounting; and extension of the methodology to higher-capacity portable hoist variants.

References

[1] Glock, C. H., Grosse, E. H., Kim, T., Neumann, W. P., & Sobhani, A. (2020). Assistive devices for manual materials handling in warehouses: a systematic literature review. International Journal of Production Research.

[2] Ramteke, A., et al. (2021). Fabrication of Material Handling or Material Lifting Crane. International Journal of Advanced Research in Science, Communication and Technology.

[3] Sunaryo, C., et al. (2020). Productivity analysis and efficiency of concrete casting using mini-cranes with a capacity of 200 kg based on appropriate technology. Revista De La Construccion.

[4] Nagy, B. (2020). Optimization of mobile crane boom using the finite element method. Thesis. **[5]** Sandhya, K. (2024). Structural Analysis of an Off-Road Vehicle Suspension System Using CATIA and ANSYS. International Journal of Scientific Research in Engineering and Management.

[6] Xue, H., et al. (2021). A Novel Method for Structural Lightweight Design with Topology Optimization. Energies.

[7] Navale, R. S., et al. (2023). Design Optimization of Single Overhead Crane Girder using Finite Element Analysis. International Journal on Recent and Innovation Trends in Computing and Communication.

[8] Wang, C., et al. (2023). Optimization of Crane Telescopic Boom Structure Based on Finite Element Software. SAE International Journal of Materials and Manufacturing.

[9] Othman, S. A., et al. (2021). Numerical Analysis of Linear Elevator Structure Using Finite Element Method. Engineering and Technology Journal.

[10] Iftikhar, S., et al. (2025). Finite Element Analysis to Enhance the Design of Patient Lifting Equipment for Clinical Applications. Pakistan Journal of Rehabilitation.

[11] Bai, Z., et al. (2019). Finite Element Analysis for the Lifting Platform of a Self-Propelled Multi-Functional Machine Used in Orchard. IOP Conference Series: Materials Science and Engineering.

[12] Sakunsongbunsiri, M., et al. (2020). Three-Dimensional Finite Element Analysis of Stress Distribution and Displacement to Design of Patient Lifting Equipment. IOP Conference Series: Materials Science and Engineering.

[13] Wang, C., et al. (2023). Optimization of Crane Telescopic Boom Structure Based on Finite Element Software. SAE International Journal of Materials and Manufacturing.

[14] Nagy, B. (2020). Optimization of mobile crane boom using the finite element method. Thesis. **[15]** Yunus, et al. (2024). Design, Analysis, and Optimization of Jib Boom Mechanism in Mobile Cranes. International Journal For Multidisciplinary Research.

[16] Guo-bao, D. (2020). Structure Design and Analysis of Portable Hoisting Equipment. Journal of Physics: Conference Series.

[17] Tas, K. A., et al. (2025). Topology Optimization-Based Lightweight Chassis Design. Dicle Universitesi Muhendislik Fakultesi Muhendislik Dergisi.

[18] Liu, X., et al. (2025). Topology Optimization of Leg Structures for Construction Robots Based on Variable Density Method. ArXiv.

[19] Wu, H., et al. (2025). Structural optimization of the excavator boom under extreme working conditions using EDEM-ADAMS coupled simulation. PLOS One.

[20] Sun, B., et al. (2025). Research on Optimization Design of Excavator Boom Structure Based on Finite Element Analysis. Applied and Computational Engineering.

[21] Suvorov, V., et al. (2023). Weight Reduction of a Ship Crane Truss Structure Made of Composites. Applied Sciences.

[22] Zhuang, J., et al. (2025). Research and Experimental Verification of an Efficient Subframe Lightweighting Method Integrating SIMP Topology and Size Optimization. Applied Sciences.

[23] Mehra, A., et al. (2025). Integrated Topology, Shape and Size Optimization-Application in Automotive Design and Manufacturing. SAE Technical Paper Series.

[24] Sandhya, K. (2024). Structural Analysis of an Off-Road Vehicle Suspension System Using CATIA and ANSYS.

[25] Abbas, Y., et al. (2021). Structural Analysis of a Transport Aircraft Wing.

[26] Anonymous. (2023). Integrated Design and Structural Analysis of Bracket Mounting of Hydraulic Tank of BH 100 Dump Truck. International Research Journal of Modernization in Engineering Technology and Science.

[27] Gaikwad, M. M. (2025). Casting Defect Analysis and Optimization of an Internal Combustion Engine Piston using FEA. International Journal for Research in Applied Science and Engineering Technology.

[28] Amroune, S., et al. (2021). Manufacturing of Rapid Prototypes of Mechanical Parts Using Reverse Engineering and 3D Printing. Journal of the Serbian Society for Computational Mechanics.

[29] Papic, S., et al. (2018). Methodology, Tools and Possibilities of Structural Analysis of the Hook in Assembly of the Lower Pulley in CAD/CAE System CATIA. Masinstvo.

[30] Farayibi, P., et al. (2020). Development of an automated mechanical lift for material handling purposes. African Journal of Science, Technology, Innovation and Development.

[31] Arunkumar, G., et al. (2021). Investigation on design, analysis and topological optimization of hydraulic scissor lift. Journal of Physics: Conference Series.