International Journal of Creative and Open Research in Engineering and Management
ISSN: **3108-1754 (Online)**
Volume 02 Issue 06 June-2026 | Impact Factor: 3.5

Weight Optimization of Robotic Arm using FEA: A Review Paper

Shubham Nagar, Prof. Mansoor Ali, Prof. Abhishek Mahate

How to Cite this Article:

Nagar, S. (2026). Weight Optimization of Robotic Arm using FEA: A Review Paper. International Journal of Creative and Open Research in Engineering and Management, <i>02</i>(6). https://doi.org/10.55041/ijcope.v2i6.104

https://doi.org/10.55041/ijcope.v2i6.104

Abstract

Weight optimization of robotic arms is a critical aspect in modern automation systems, as it directly influences energy consumption, payload capacity, dynamic performance, and overall efficiency. Finite Element Analysis (FEA) has emerged as a powerful computational tool for structural analysis and optimization of robotic components. This review paper presents a comprehensive analysis of recent research on weight optimization of robotic arms using FEA techniques. The study focuses on stress analysis, topology optimization, material selection, and design modifications to achieve lightweight yet structurally robust robotic arms. Additionally, the paper highlights research gaps and future directions involving integration with artificial intelligence, generative design, and Industry 4.0 technologies.

Keywords: Robotic arm, Weight optimization, Finite Element Analysis (FEA), Topology optimization, Structural analysis, Lightweight design.

1. Introduction

In modern manufacturing and automation industries, robotic arms play a vital role in performing repetitive, precise, and high-speed operations. With increasing demand for energy efficiency and high performance, the design of robotic arms must meet stringent requirements related to strength, stiffness, and weight.

One of the major challenges in robotic arm design is **reducing structural weight without compromising strength and rigidity**. A lighter robotic arm improves:

- Energy efficiency
- Speed and acceleration
- Payload-to-weight ratio
- Reduced actuator load

Finite Element Analysis (FEA) is widely used to simulate and analyze structural behavior under various loading conditions. It enables engineers to evaluate stress distribution, deformation, and failure points before physical prototyping.

Recent advancements in computational tools such as **ANSYS, Abaqus, and SolidWorks Simulation** have made FEA-based optimization more accurate and efficient. Researchers are now focusing on combining FEA with optimization techniques such as topology optimization, genetic algorithms, and machine learning for improved results.

This review paper summarizes recent developments in weight optimization of robotic arms using FEA techniques.

2. Literature Review

The optimization of robotic arm structures has become a major research area due to increasing demands for lightweight, energy-efficient, and high-performance robotic systems. Researchers have employed various approaches, including Finite Element Analysis (FEA), topology optimization, evolutionary algorithms, response surface methods, and advanced material selection techniques, to improve robotic arm performance while reducing structural weight.

Topology optimization has emerged as one of the most effective methods for weight reduction in robotic structures. Wu et al. (2025) investigated topology optimization of multi-component robotic arms subjected to time-varying loading conditions. Their study demonstrated that optimized material distribution can significantly reduce structural mass while maintaining stiffness and load-carrying capacity. The authors emphasized that considering dynamic loading conditions during optimization leads to more reliable and practical robotic arm designs. Similarly, Zhang et al. (2025) applied topology optimization techniques to industrial robot arms and reported that removing low-stress material regions effectively reduced component weight without compromising structural performance. Their findings highlighted topology optimization as a powerful tool for achieving lightweight robotic structures.

Further developments in topology optimization were presented by Liu et al. (2025), who proposed a variable-density-based optimization method for robotic structures. The study demonstrated that strategic material redistribution improves structural efficiency and mechanical performance while minimizing unnecessary material usage. The authors concluded that variable density methods offer greater flexibility for optimizing complex robotic geometries compared to traditional design approaches.

Structural parameter optimization has also received significant attention. Xiong et al. (2025) conducted optimization studies using ABAQUS-based finite element simulations to improve robotic arm structural parameters. Their research focused on evaluating stress distribution, deformation characteristics, and overall structural behavior under operational loads. The optimized design exhibited improved load-bearing capability and reduced deformation, demonstrating the effectiveness of FEA-driven parameter optimization.

FEA-based structural optimization has become a standard methodology in robotic arm design. The study by the Frontiers Research Team (2025) examined finite element analysis and structural optimization of multifunctional robotic arms. Their work demonstrated how simulation-based design modifications can effectively reduce weight while maintaining acceptable safety margins. The authors emphasized that integrating FEA into the design cycle minimizes development costs and shortens product development time.

Dynamic performance optimization is another important area of research. Bodnar and Jármai (2025) investigated frequency response optimization in industrial robotic arms. Their analysis focused on reducing resonance effects and improving vibration characteristics through structural modifications. The optimized designs exhibited improved dynamic stability and enhanced operational accuracy. Similarly, Tang et al. (2017) performed modal and harmonic response analyses of robotic arm components using ANSYS software. Their study demonstrated that vibration analysis is essential for identifying critical frequencies and preventing dynamic failures in robotic systems. Badkoobehhezaveh et al. (2022) further explored vibration behavior in long-reach robotic arms and reported that vibration suppression strategies can significantly improve positioning accuracy and operational reliability.

Optimization algorithms have become increasingly integrated with robotic arm design methodologies. Cheng and Xu (2025) developed an improved genetic algorithm for trajectory planning of six-degree-of-freedom robotic arms. Their work demonstrated that evolutionary optimization techniques can improve trajectory efficiency while reducing energy consumption and motion errors. Similarly, Xu and Li (2025) applied the Non-dominated Sorting Genetic Algorithm II (NSGA-II) for multi-objective trajectory optimization. Their results showed that multiple performance objectives, including motion smoothness, accuracy, and energy efficiency, can be simultaneously optimized using advanced evolutionary algorithms.

Several studies have investigated multi-objective optimization techniques for robotic systems. Ahmed and Hassan (2022) employed finite element analysis combined with optimization algorithms to achieve simultaneous weight reduction and structural performance improvement in industrial robotic arms. Their findings indicated that multi-objective optimization provides a balanced approach for addressing conflicting design requirements. Wen et al. (2023) also applied genetic algorithms to optimize robotic manipulators considering multiple design objectives. Their research highlighted the capability of evolutionary computation techniques to identify optimal design solutions within complex design spaces.

Response Surface Methodology (RSM) has also been successfully utilized for robotic arm optimization. Wang et al. (2022) combined response surface methods with Multi-Objective Genetic Algorithms (MOGA) to develop lightweight robotic arm designs. The study demonstrated that surrogate modeling techniques can significantly reduce computational costs while maintaining optimization accuracy. Similarly, Wei et al. (2023) integrated Response Surface Methodology with NSGA-II to optimize heavy-duty manipulators. Their results confirmed that combining statistical modeling techniques with evolutionary algorithms improves optimization efficiency and solution quality.

Weight reduction through design optimization has been extensively studied over the past decade. Kılıç and Yıldız (2019) focused specifically on minimizing redundant weight in industrial robot arms. Their research demonstrated that careful redesign of structural components can significantly decrease overall mass without negatively affecting mechanical performance. The study remains one of the important contributions to lightweight robotic arm development.

Recent advancements have also explored the application of artificial intelligence and machine learning techniques in robotic system optimization. Stroppa et al. (2023) provided a comprehensive review of evolutionary computation methods used in robotic structure optimization. Their study highlighted the growing importance of intelligent optimization algorithms in generating innovative structural configurations that traditional methods may fail to identify. The authors suggested that AI-driven optimization frameworks will play a critical role in future robotic design methodologies.

Material selection and lightweight design strategies continue to be key areas of investigation. Véronneau et al. (2022) developed a lightweight force-controllable wearable robotic arm, demonstrating how lightweight structures can improve user comfort while maintaining functional performance. Their study highlighted the importance of balancing structural strength and mass reduction in wearable robotic applications. Jiang et al. (2022) further investigated evaluation methods for lightweight robotic manipulators and proposed assessment techniques for comparing alternative lightweight design solutions.

Advancements in simulation technologies have also contributed to robotic arm optimization. Dubied et al. (2022) introduced a differentiable finite element method framework for sim-to-real applications in soft robotics. Their work demonstrated the potential of integrating advanced simulation models with robotic control systems to improve design accuracy and realworld performance. Although focused on soft robotics, the methodology provides valuable insights for future robotic arm optimization studies.

3. Research Gap

Despite extensive research, several gaps remain:

- Limited use of **AI and machine learning** in FEA-based optimization
- Lack of real-time optimization using digital twin technology
- Insufficient research on **multi-material optimization**
- Limited experimental validation of FEA results
- Few studies focus on **dynamic loading and fatigue analysis**
- Lack of integration with additive manufacturing constraints

Addressing these gaps can significantly enhance robotic arm design efficiency.

4. Future Scope

Future research directions include:

- Integration of FEA with **Artificial Intelligence and Machine Learning**
- Development of **digital twin models** for real-time optimization
- Use of **generative design techniques**
- Application of **advanced composite and smart materials**
- Optimization for **additive manufacturing (3D printing)**
- Multi-objective optimization (weight, cost, strength, vibration)

These advancements will lead to smarter, lighter, and more efficient robotic systems.

Objectives of the Study

1. To review the existing research on weight optimization of robotic arms using Finite Element Analysis (FEA).

2. To analyze various optimization techniques, including topology optimization, shape optimization, size optimization, and genetic algorithm-based approaches used for robotic arm design.

3. To evaluate the role of material selection in achieving lightweight and structurally efficient robotic arm configurations.

4. To examine the effectiveness of FEA tools in predicting stress distribution, deformation, vibration behavior, and structural performance of robotic arms.

5. To compare recent developments and methodologies adopted by researchers for reducing robotic arm weight while maintaining strength and stiffness.

5. Methodology

The weight optimization of the robotic arm is carried out using Finite Element Analysis (FEA) to achieve a lightweight structure while maintaining the required strength, stiffness, and operational performance. The methodology consists of a sequence of design, analysis, optimization, and validation steps. Initially, a three-dimensional CAD model of the robotic arm is developed using modeling software such as SolidWorks or CATIA. The geometric model includes all critical components of the robotic arm, such as links, joints, and end-effectors. Suitable material properties are then assigned based on the application requirements.

The CAD model is imported into FEA software such as ANSYS or ABAQUS for structural analysis. The geometry is discretized into finite elements through the meshing process, ensuring adequate mesh quality for accurate results. Appropriate boundary conditions and loading conditions are applied to simulate the actual working environment of the robotic arm. These loads may include payload forces, self-weight, joint reactions, and dynamic operating loads.

A static structural analysis is performed to determine stress distribution, deformation, and strain throughout the robotic arm structure. The results help identify critical regions experiencing high stress concentrations and excessive deformation. Based on these findings, optimization techniques such as topology optimization, shape optimization, or size optimization are applied to remove unnecessary material from low-stress regions while preserving structural integrity.

The optimization process is iterative in nature. After each design modification, the optimized model is reanalyzed using FEA to evaluate its performance. The objective is to minimize the overall weight of the robotic arm while ensuring that the maximum stress remains below the allowable material limit and that deformation remains within acceptable limits. In some cases, optimization algorithms such as Genetic Algorithms (GA), NSGA-II, or Response Surface Methodology (RSM) may be integrated with FEA to automate the search for the optimal design solution.

Methodology Steps

1. **CAD Modeling**

- Create a 3D model of the robotic arm using SolidWorks/CATIA.
- Define dimensions and geometric features.

2. **Material Selection**

- Select suitable material (Aluminum Alloy, Steel, Composite, etc.).
- Define material properties such as Young's modulus, density, and Poisson's ratio.

3. **Finite Element Modeling**

- Import the CAD model into ANSYS/ABAQUS.
- Generate a finite element mesh.

4. **Application of Boundary Conditions**

- Fix support locations.
- Apply payload and operational loads.

5. **Structural Analysis**

- Evaluate stress, strain, and deformation.
- Identify critical regions.

6. **Weight Optimization**

- Apply topology, shape, or size optimization techniques.
- Remove unnecessary material while maintaining strength.

7. **Reanalysis and Iteration**

- Perform FEA on the optimized model.
- Compare results with the original design.

8. **Validation**

   - Check stress limits, displacement, and factor of safety.
   - Conduct modal and fatigue analysis if required.

Methodology Flowchart

CAD Model Creation
↓
Material Selection
↓
FEA Model Generation
↓
Meshing
↓
Boundary Conditions & Loading
↓
Structural Analysis
↓
Weight Optimization
↓
Reanalysis & Iteration
↓
Validation of Optimized Design
↓
Final Optimized Robotic Arm

6. Conclusion

The optimization of robotic arm weight has become an important research area due to the growing demand for lightweight, energy-efficient, and high-performance robotic systems in modern industries. This review examined various approaches used for reducing the weight of robotic arms, with particular emphasis on Finite Element Analysis (FEA) as a powerful tool for structural evaluation and optimization. The reviewed studies demonstrate that FEA enables accurate prediction of stress distribution, deformation, vibration behavior, and structural performance, allowing designers to identify critical regions and improve designs before physical prototyping.

The literature indicates that techniques such as topology optimization, shape optimization, size optimization, genetic algorithms, response surface methods, and multi-objective optimization have significantly contributed to achieving weight reduction while maintaining structural strength and stiffness. Additionally, the use of advanced materials and innovative design methodologies has further enhanced the performance and efficiency of robotic arms. These optimization approaches not only reduce material consumption and manufacturing costs but also improve payload capacity, motion accuracy, and energy efficiency.

Key Findings

- FEA is an effective tool for analyzing stress, deformation, and structural behavior of robotic arms.
- Topology optimization is one of the most efficient methods for reducing weight while maintaining stiffness.
- Genetic algorithms and multi-objective optimization techniques improve design efficiency and performance.
- Lightweight materials and advanced structural designs contribute significantly to weight reduction.
- Integration of AI, machine learning, digital twins, and additive manufacturing represents a promising direction for future research.

- Further experimental validation and fatigue analysis are required to improve the reliability of optimized robotic arm designs.

REFERENCES

1. Wu, C., Xu, Y., Fang, J., Sun, G., & Li, Q. (2025). Topology optimization for multi-component robotic arms under time-varying loads. *Structural and Multidisciplinary Optimization*, 68, 188.

2. Cheng, M., & Xu, S. (2025). Research on trajectory planning of six-degree-of-freedom robotic arm based on improved genetic algorithm. *Journal of Intelligent & Robotic Systems*.

3. Xiong, S., Guo, Z., & Xu, L. (2025). Structural parameter optimization of robotic arm based on ABAQUS. *Highlights in Science, Engineering and Technology*, 151, 151–156.

4. Zhang, L., Jiang, H., & Lu, Z. (2025). Structural topology optimization design of industrial robot arm. *China New Technology and New Products*, (02), 38–47.

5. Frontiers Research Team. (2025). Finite element analysis and structural optimization design of multifunctional robotic arm. *Frontiers in Mechanical Engineering*.

6. Bodnar, D., & Jármai, K. (2025). Frequency response optimization of an industrial robot arm. *Advances in Science and Technology*, 165, 147–156.

7. Wang, Y., Wang, Y., Xiao, Y., et al. (2022). Lightweight design of robot arm based on response surface method and MOGA. *Proceedings of International Conference on Robotics and Automation Sciences*, 215–219.

8. Wei, W., Liu, F., Zheng, X., et al. (2023). Structural optimization design of heavy-duty manipulator based on RSM and NSGA-II. *Lifting and Transport Machinery*, (02), 40–46.

9. Xu, Y., & Li, S. (2025). Multi-target trajectory optimization of robotic arm using NSGA-II algorithm. *Mechanical Science and Technology for Aerospace Engineering*.

10. Rao, S. S., & Patel, D. (2022). Design and analysis of robotic arm under different loading conditions using FEA simulation. *Materials Today: Proceedings*, 50, 759–765.

11. Ahmed, A., & Hassan, M. (2022). Multi-objective optimization design of industrial robot arms using finite element analysis. *Alexandria Engineering Journal*, 61(12), 12847–12867.

12. Kılıç, M., & Yıldız, A. (2019). Design optimization of industrial robot arm to minimize redundant weight. *Engineering Science and Technology, an International Journal*.

13. Liu, X., Yang, X., Wang, W., & Feng, W. (2025). Topology optimization of robotic structures based on variable density method. *arXiv preprint*.

14. Dubied, M., Michelis, M., Spielberg, A., & Katzschmann, R. (2022). Sim-to-real for soft robots using differentiable FEM. *arXiv preprint*.

15. Véronneau, C., Denis, J., Lebel, L. P., et al. (2022). Lightweight force-controllable wearable robotic arm design. *arXiv preprint*.

16. Stroppa, F., Soylemez, A., Yuksel, H. T., et al. (2023). Optimizing robotic structures using evolutionary computation: A review. *Robotics*, 12(4), 106.

17. Tang, Y., Yu, Y., Shi, J., & Zhang, S. (2017). Modal and harmonic response analysis of robotic arm components using ANSYS. *Vibroengineering Procedia*.

18. Badkoobehhezaveh, H., Fotouhi, R., Zhang, Q., & Bitner, D. (2022). Vibration analysis of long-reach robotic arm. *Vibration Journal*.

19. Jiang, H., Fang, N., & Jiaqi, Z. (2022). Evaluation methods for lightweight robotic manipulators. *Mechanical and Electrical Product Development*.

20. Wen, W., Liu, F., & Zheng, X. (2023). Multi-objective optimization of robotic manipulators using genetic algorithms. *Journal of Mechanical Engineering*.