Proceedings of the 2023 International Conference on Machine Learning and Automation
DOI: 10.54254/2755-2721/33/20230238

# Finite element analysis and optimal design of robotic arm

**Fanghao Ding**

School of Hefei NO.1 High School, Hefei, Anhui Province, China, 230091

3612796079@qq.com

**Abstract.** With the rapid development of modern industry and Robotics, industrial robots are widely used in manufacturing. However, because of the high price of industrial robot systems on the market, industrial robots are limited to the production of small and medium-sized enterprises. Against this background, this paper fully investigates the development process and research status of industrial manipulators at home and abroad, and designs a six-degree-of-freedom economic industrial manipulator. ANSYS Workbench software is used to analyze the statics of the arm and the whole machine, and the equivalent stress cloud map and the displacement and deformation cloud chart are obtained. The static strength and stiffness of the large arm and the whole machine are analyzed, and the reliability of the structure is verified. Then adopting the optimization module in the software, the multi-objective optimization design of the large arm is carried out. The size of the structure is reduced and the quality is reduced by 14.7% in the case that the large arm meets the requirements of the allowable stress and the maximum displacement. The target of the optimization design is realized.

**Keywords:** robotic arm, finite element analysis, optimal design.

## 1. Introduction

The emergence of industrial robots has significantly improved human production levels and efficiency. Industrial robots can adapt to hazardous work environments [1-4]. To meet the needs of small and medium-sized enterprises, there has been a focus on lightweight design for industrial robots [5, 6]. Extensive research has been conducted on mechanical arm structure design and structural optimization by scholars and institutions worldwide. For instance, Wang Yang from Beijing Jiaotong University has designed a modular, lightweight mechanical arm and performed topology optimization based on structural analysis [7, 8]. Similarly, Lin Yizhong and colleagues from Guangxi University have conducted finite element analysis and sensitivity analysis on a welding robot's upper arm, optimizing its structure for weight reduction and improved performance [9-12]. The optimization of mechanical arm structures is crucial for enhancing the production efficiency and adaptability of industrial robots. This paper aims to explore the methods and techniques of structural optimization, contributing to the advancement of industrial robots.

DOI: 10.54254/2755-2721/33/20230238

## 2. Statics analysis of the high arm

### *2.1 Basic theory of finite element analysis*

To import a 3D model, create a large arm 3D model in SolidWorks. Simplify the model by removing unnecessary circular holes and chamfers. Import the simplified 3D model into ANSYS Workbench software and set the material properties.

In the past, steel and its alloys were used to ensure robotic arms had sufficient stiffness and strength. However, with the development of lightweight robotic arms, lightweight materials (Al, Mg, Ti alloy and CFRP) have become mainstream [13]. Table 1 lists the density, specific stiffness and specific strength values of these materials. Mg alloy has low density and high specific strength, but relatively low specific stiffness. CFPR has low mechanical property density, but high cost and difficult forming. Al alloy material has excellent light weight and is more widely used in robotic arms. Define the requirements for lightweight and economic properties of the material, and select Al alloy as the boom material.

**Table 1.** Density(ρ), specific stiffness (Rs) and specific strength(Rσ) values of the material.

| Material | $\rho$(kg/m$^3$) | $R_s$(MPa·m$^3$/kg) | $R_\sigma$(MPa·m$^3$/tone) |
|---|---|---|---|
| Steel | 7930 | 25 | 44 |
| Ti alloy | 4400 | 26 | 205 |
| Al alloy | 2700 | 25.5 | 79.6 |
| Mg alloy | 1780 | 23 | 100 |
| CFRP | 1700 | 42 | 326 |

Grid division is a crucial step in creating a finite element model. Start by controlling the global mesh and selecting a tetrahedral mesh division. Set parameters like global cell size, correlation center, smoothness, and mesh transition. Then, refine the local mesh.

### *2.2 Boom statics analysis*

By solving and calculating with ANSYS Workbench software, the equivalent stress cloud map and displacement cloud map of the boom were obtained, as shown in Figure 1. It can be seen from Figure 1. that due to the effect of external load, the maximum deformation occurs at the front end of the manipulator, and the maximum displacement is 0.0431mm, which is a small value, may cause significant displacement at the end of the robotic arm due to its series structure.

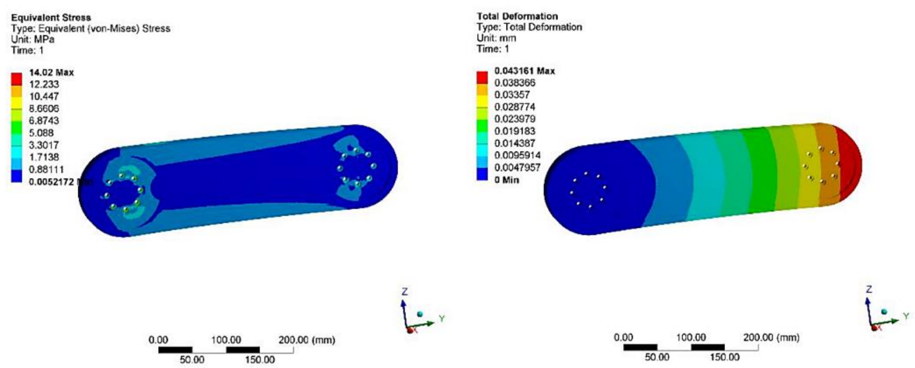


**Figure 1.** The stress and displacement results of arm.

### *2.3. Multi objective optimization design of the boom*

Multi-objective optimization is a design that optimizes parameters by establishing a parameterized model that includes structural size parameters, material performance parameters, load parameters, etc. [14].

DOI: 10.54254/2755-2721/33/20230238

*2.3.1 Sensitivity analysis of the boom.* Under the condition of ensuring that the maximum stress of the boom does not exceed the allowable stress of the material, while also ensuring that the maximum displacement and deformation of the boom are small, the design parameters of the boom structure are optimized.

The sensitivity analysis of the boom is shown in Figure 2. The degree of impact on the quality of the large arm ranges from large to small: the thickness of the main body of the large arm, the thickness of the side rib plate, and the thickness of the convex boss at the joint connection.

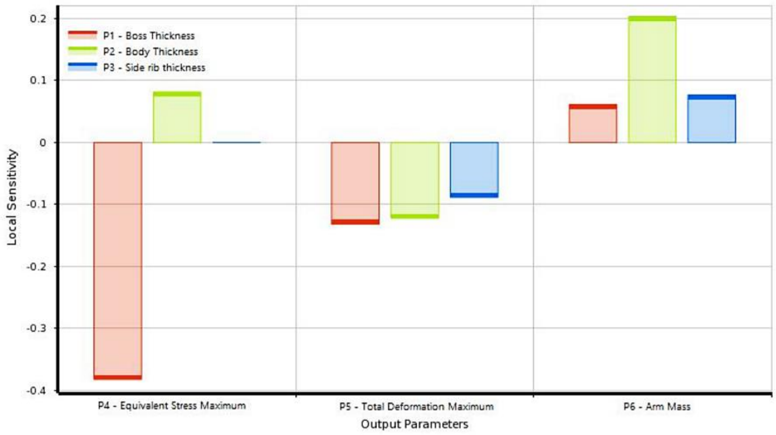


**Figure 2.** The sensitivity analysis of arm.

*2.3.2. Analysis of the relationship between design variables and target variables.* The 3D relationship between the design variables (thickness of the main body of the boom, thickness of the side rib plate, thickness of the joint boss) and the target variables (maximum stress, maximum displacement, mass) in the optimization of the boom structural parameters can be obtained through ANSYS Workbench software, as shown in Figure 3 to Figure 5. It can be seen that the three target variables of the boom are difficult to achieve simultaneously. The maximum displacement and deformation of the large arm are not greater than 0.05mm, which is set as the highest priority.

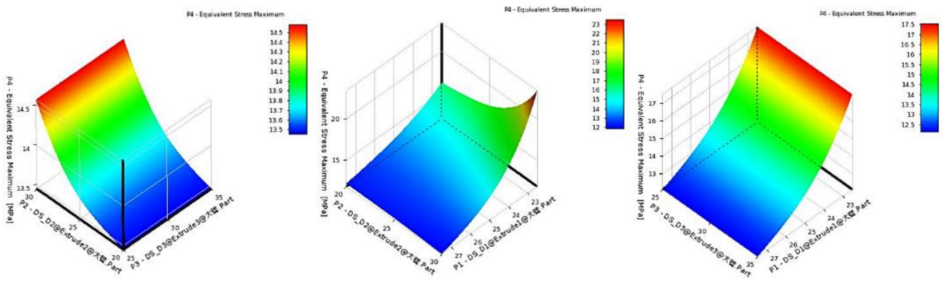

**Figure 3.** The relationship between the design variables and the stress of the arm.

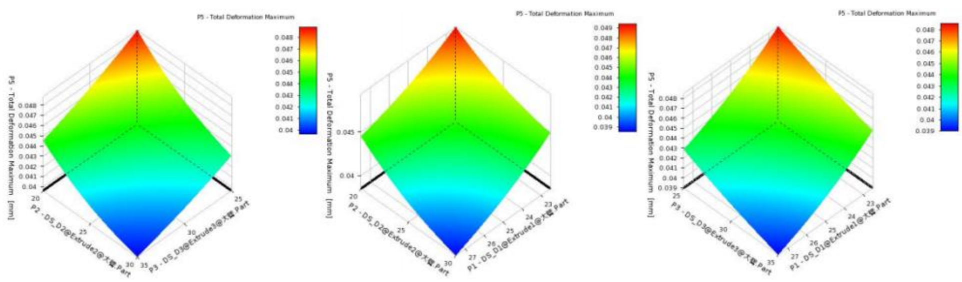

**Figure 4.** The relationship between the design variables and the displacement of the arm.

DOI: 10.54254/2755-2721/33/20230238

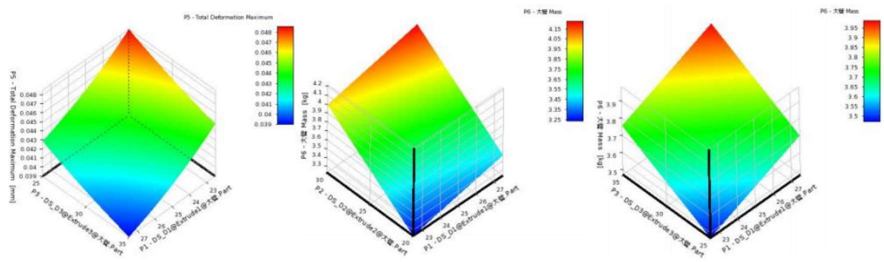

**Figure 5.** The relationship between the design variables and the mass of the arm.

*2.3.3 Multi objective optimization design and analysis of the boom.* By combining the previous sensitivity analysis results with the priority order of optimization objectives, the priority levels of three objective variables are set in the Optimization module of ANSYS Workbench software in order of priority. Through optimization calculation, three optimization result schemes for the boom are obtained.

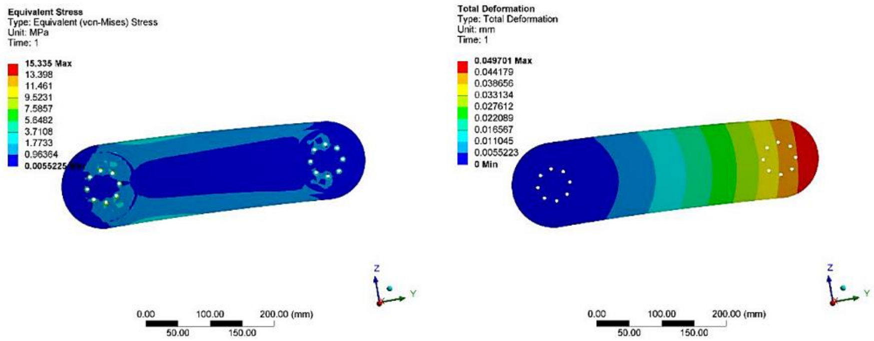


**Figure 6.** The stress and displacement results of the optimized arm.

From the optimization results, it can be seen that the maximum equivalent stress of the optimized boom has increased compared to before, but it is still far less than the maximum allowable stress of the material. After optimization, the maximum displacement of the boom has increased compared to before, but it is less than the required maximum displacement deformation of 0.05mm.

*2.4. Complete machine statics analysis*

Due to the numerous structural details of the robotic arm, the impact of chamfers, threaded holes, and other structures on the overall robotic arm is not significant, so it is simplified during modeling. Import the simplified model of the robotic arm into the ANSYS Workbench software, as shown in Figure 7.

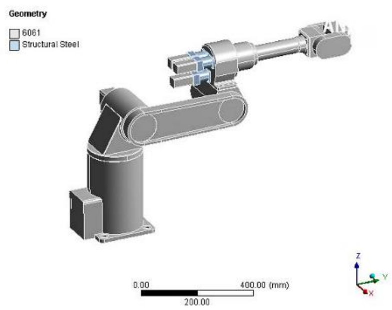


**Figure 7.** The simplified model of whole manipulator.

As shown in Figure 8, the maximum displacement of the entire machine occurs at the forefront of the robotic arm, with a value of 0.1952mm. The displacement deformation at the end of the robotic arm is formed by the accumulation of deformation in each component, so the structure of parts with larger stress and deformation can be improved to improve strength and stiffness. At the same time, the

DOI: 10.54254/2755-2721/33/20230238

structural size of parts with a smaller load-bearing capacity can be reduced to reduce their own weight and improve the positioning accuracy of the robotic arm.

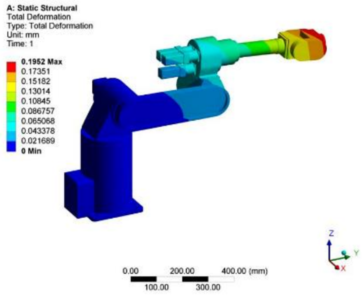


**Figure 8.** The displacement nephogram of whole manipulator.

## 3. Conclusions

ANSYS Workbench software is used to carry out finite element analysis on the boom of the mechanical arm and analyze its static characteristics. The results show that the stress and deformation of the boom meet the requirements, and there is a large margin. On this basis, a multi-objective optimization design was carried out on the boom. While meeting the design requirements, the boom mass was reduced by 14.7%, achieving the goal of optimization. The reliability of the structure is verified by Statics analysis of the whole manipulator.

## Acknowledgments

All suggestions regarding to the topic selection and writing skills from advisors are greatly appreciated. The funding support by my parents are also acknowledged.

## References

[1] Park, C., Park, K., & Kim, D. (2008, April). Design of dual arm robot manipulator for precision assembly of mechanical parts. In 2008 International Conference on Smart Manufacturing Application (pp. 424-427). IEEE.

[2] Abele, E., Weigold, M., & Rothenbücher, S. (2007). Modeling and identification of an industrial robot for machining applications. CIRP annals, 56(1), 387-390.

[3] Madsen, O., Bro Sørensen, C., Larsen, R., Overgaard, L., & Jacobsen, N. J. (2002). A system for complex robotic welding. Industrial Robot: An International Journal, 29(2), 127-131.

[4] Kim, B. J., Yun, D. K., Lee, S. H., & Jang, G. W. (2016). Topology optimization of industrial robots for system-level stiffness maximization by using part-level metamodels. Structural and multidisciplinary optimization, 54, 1061-1071.

[5] Kelmar, L., & Khosla, P. K. (1990). Automatic generation of forward and inverse kinematics for a reconfigurable modular manipulator system. Journal of Robotic Systems, 7(4), 599-619.

[6] Chiheb, Z., Tarek, J., & Aref, M. (2015). Simulations of the Dynamic Behavior of the Robot ADAM with Trunk and Arms Subjected to 3D External Disturbances. In Design and Modeling of Mechanical Systems-II: Proceedings of the Sixth Conference on Design and Modeling of Mechanical Systems, CMSM'2015, March 23-25, Hammamet, Tunisia (pp. 459-468). Springer International Publishing.

[7] Wang Y. (2017). Structural analysis and optimal design of light manipulator. (Doctoral dissertation, Beijing Jiaotong University).

[8] Zheng T. Structural optimization and dynamic analysis of welding manipulator. (PhD dissertation, Zhejiang University of Technology).

[9] Chen C. (2013). Virtual Design and Simulation Analysis of Six Degrees of Freedom Industrial Robot Nanjing University of Information Science and Technology.

DOI: 10.54254/2755-2721/33/20230238


[10] Guan Y S, D X, LI H Z, Y Z N, Wu W G, Jiang L. (2013). Structural Analysis and Optimization of Industrial Robots. Journal of South China University of Technology: Natural Science Edition (9), 6.

[11] Lin Y Z, Liao J F, Liu Q G, Xu J. (2016). Modal analysis and optimization of the arm of a six-degree-of-freedom welding robot. Combined machine tool and automatic processing technology (02), 36-38+42. doi:10.13462/j .cnki.mmtamt.2016.02.011.

[12] Chen Z Q, Liang X H, Lin Y K, Wang N F. (2013). Structural design and finite element analysis and optimization of six degrees of freedom serial robot. Machine Tool and Hydraulics, 41(23), 5.

[13] Hua D Q. (2016). The status and trends of the robot industry and its key materials. New Material Industry (7), 5.

[14] Yeung, S.H., & Man, K.F. (2011). Multiobjective Optimization. IEEE Microwave Magazine, 12, 120-133.