Finite Element Static Analysis of Main Arm for a Special Manipulator

Zhang Weiwei
School of Mechanical Engineering Shandong Huayu University of Technology Dezhou, China
zhangweiwei@huayu.edu.cn
zhang8412860@163.com

Abstract—With the development of modern industry, manipulator gradually replaces manual operation. In the process of grasping, moving and placing the workpiece, the manipulator needs to have high accuracy, and the movement track cannot be changed due to the force deformation. Rigidity refers to the deformation, that is, the main stressed parts of the manipulator should have good rigidity, and the deformation should be as small as possible to ensure the accuracy of the motion track. In this paper, the main arm and the auxiliary arm of the manipulator are analyzed by finite element statics. The results show that the displacement and the equivalent stress meet the requirements.

1. INTRODUCTION
The extension axis of the manipulator directly defines the overall working radius of the manipulator. The bearing capacity of the whole manipulator is closely related to the design of the extension axis. The flexibility and extension speed of the manipulator are very important in the whole design. The expansion shaft of this design adopts the double speed structure. The servo motor transmits the power to the helical rack through the helical gear. The rack is fixed on the arm of aluminum alloy to drive the main arm to do expansion movement. The side of the main arm is equipped with synchronous belt and synchronous wheel mechanism. One end of the synchronous belt is fixed on the base of the expansion shaft, and the other end is fixed on the auxiliary arm of the manipulator. In this way, the main arm is operated at the same time, the jib moves with the main boom. The main boom and the auxiliary boom are made of aluminum profiles. The transmission scheme of the telescopic shaft is shown in Figure 1.

Figure 1. Transmission scheme of telescopic shaft

The connection between the main boom and the auxiliary boom and the cross-sectional structure are shown in Figure 2 and Figure 3.
2. **FINITE ELEMENT MODELING OF MAIN ARM**

2.1. **Structural Modeling of Main Arm**

We use the software to build the three-dimensional model of each part of the manipulator, then import the model into the finite element analysis software ANSYS, and build the corresponding finite element model according to the three-dimensional model.

Due to the problem that the CAE(Computer aided engineering) model cannot be generated if the 3D model of the main arm of the complex manipulator is directly imported, the main arm model should be simplified and modified on the premise that it has little impact on the analysis and calculation, so as to strive for accuracy in the process of CAD(computer-aided design) modeling, so as to truly reflect the static and dynamic characteristics of the structure [1]. However, observing the main arm part drawing, there are many small holes and other structures, which have little influence on the overall structure, but if these solid features are imported into the finite element software, it will greatly increase the finite element calculation amount [2]. Therefore, in the finite element model of the main boom, all the small features such as chamfering, rounding and small holes in the CAD model are ignored; in addition, the component structure with small impact on the whole is ignored. The simplified CAD model is shown in Figure 4.

![Figure 4. Main arm structure of manipulator](image)

2.2. **Finite Element Modeling of Main Arm**

2.2.1. **Unit type selection**

According to the characteristics of solid modeling and structural mechanics of the manipulator [3,4], SOLID285 element is adopted for the main arm of the manipulator, and SOLID285 element is suitable for irregular mesh generation model. Its displacement function is shown in Formula 1, 2, 3. The element is a 4-node tetrahedral element [5], as shown in Figure 5. Each node has three degrees of freedom: the rotation angles in X direction, Y direction and Z direction.
2.2.2. Material properties
The material used for the main boom and auxiliary boom is aluminum alloy (calculated according to aluminum alloy 5182). The material of guide rail and sliding block between the auxiliary boom and the main boom is 45 steel, and its elastic modulus, density and Poisson's ratio are shown in Table 1.

| Material   | E (Pa)        | $\rho$ (kg/m³) | $\mu$ (MPa) |
|------------|---------------|----------------|--------------|
| Main arm   | $7.6 \times 10^{10}$ | 2720           | 0.3          | 145          |
| Guide rail | $2.0 \times 10^{11}$ | 7850           | 0.3          | 355          |

2.2.3. Gridding
The software ANSYS can perform different meshing functions according to the changes of parameters, mainly including the intelligent free meshing of complex models and the local refinement of meshes. The main arm of the manipulator is divided into intelligent free grids.

2.2.4. Finite element model
Four node tetrahedron element is used to simulate the jib structure. After discretization, the total number of nodes and elements is 13692 and 47442 respectively. The finite element model is shown in Figure 6 (in the figure, the red on the right is the load, and the direction is downward; the green on the left is the displacement constraint surface).

Figure 6. Finite element model of main arm

3. Finite Element Static Analysis of Main Arm
One end of the main arm is solid and the other end is hollow, as shown in Figure 4.
Carry out strength analysis and handle according to the working extreme position of the main arm, i.e. fix at the solid end of the main arm. According to the calculation, the maximum load on the main
arm is 196N, which acts on the hollow end of the main arm and the direction is perpendicular to the working plane of the auxiliary boom. In the actual calculation, divide the load by the total number of nodes on the edge surface of the main arm, and evenly apply it to all nodes on the edge surface of the main arm.

3.1. Displacement

The ability of parts to resist deformation is called rigidity. If the rigidity of the main arm is larger, the deformation of the main arm will be smaller. Of course, if the rigidity of the main arm is not enough, when it is affected by the vertical load and other forces, the main arm will produce large elastic deformation, which will not only affect the accuracy of picking up the workpiece by the end picker, but also affect the movement route of the end picker. Through the finite element calculation and analysis, the x-direction, y-direction, Z-direction and displacement of the main boom are obtained, as shown in Fig. 7, 8, 9 and 10 respectively.

![Figure 7. X displacement](image1)

![Figure 8. Y displacement](image2)

![Figure 9. Z displacement](image3)

![Figure 10. Total displacement](image4)

It can be seen from the above four figures that the maximum displacement in X direction occurs at the right hollow end, with a value of 0.03645mm; the maximum displacement in Y direction occurs at the edge of the right hollow end, with a value of 0.029738mm; the maximum displacement in Z direction occurs at the right hollow end, with a value of -0.481059mm; the maximum and displacement of the main arm occurs at the right hollow end, with a value of 0.482952mm.
3.2. Stress
As the main boom material is aluminum alloy (calculated according to aluminum 5182), it belongs to plastic material. The fourth strength theory is used to calculate the strength condition. Through the calculation and analysis of the finite element ANSYS software, the equivalent stress diagram of the main boom is obtained. As shown in Figure 11, the maximum stress value of the main boom is 9.03MPa. By looking up the common materials and their main mechanical properties table, the yield of aluminum alloy aluminum 5182 is $\sigma_s = 145$MPa, the safety factor is generally 1.2-2, and the safety factor is $n = 1.3$, then the allowable stress of the main boom is $[\sigma] = \sigma_s / N = 111.5$MPa. Obviously, the equivalent stress of the main boom is far less than its allowable stress, that is, the strength is to meet the design requirements of the manipulator.

![Figure 11. Equivalent stress diagram of manipulator main arm](image)

4. Conclusion
Firstly, this paper introduces the methods of finite element modeling and static analysis. Then, the finite element model and static analysis of the manipulator main arm structure are carried out in ANSYS, and the X, Y direction displacement and displacement nephogram are obtained, and the equivalent stress of the main arm is also obtained, and its analysis is carried out. Finally, the conclusion is drawn that the design and structure of the manipulator main arm meet the use requirements.

REFERENCES
[1] Yi, R. (2002). Structural mechanics analysis with ANSYS. First edition. Beijing: Peking University Press.
[2] Li, L. M. Practical course of ANSYS finite element analysis [M]. Beijing: Tsinghua University Press
[3] Mikiskav Riesder.Richard I.Devries.Finite Element Analysis and Structure Optimization of Vehicle Wheel.SAE Paper830113.
[4] Liu, Y. (2009). Hou Jieru. Fundamentals of mechanical design [M]. Beijing: China Electric Power Press .02:282
[5] Cao, Y.Z.(2004). Altintas Y. A General Method for the Modeling of Spindle –Bearing Systems. Journal of Mechanical Design,12(6): 1089-1104.