Static characteristics and influencing factors analysis of azimuth axis of a radar antenna pedestal

WEN Guang¹, CHENG Ming²*, TANG Xianchun², JIANG Xinhui², ZHANG Xuerong²

¹School of Intelligent Manufacturing, Chengdu Technological University, Chengdu 611730, China
²Chengdu Radio Special Equipment Factory (state owned 633 factory), Chengdu 611730, China
Wg66360336@126.com

Abstract: Azimuth axis is an important part of radar antenna support. Static characteristics were analyzed with the help of finite element software. And the stress and deformation distribution of the structure under the normal working load were obtained, and the influence of the size parameters of the azimuth axis structure on its static characteristics was also investigated. Through the analysis, it is found that the maximum stress value of the azimuth axis under the working load is less than the allowable stress of the material, and the strength meets the use requirements. With the increase of size parameter \( l_3 \) and \( d_{11} \), the maximum stress of the shaft decreases gradually, and increases with the increase of size parameter \( d_{32} \). The analysis results provide data reference for further optimization of the azimuth axis.

1. Introduction

As one of the important parts of radar antenna support, azimuth axis is mainly used to adjust the radar azimuth. In radar target searching, the azimuth axis bears periodic torsion load. Under the action of torsion load, the azimuth axis may have excessive internal stress, which will lead to crack initiation and propagation, and then lead to serious fracture accidents. Therefore, for the designers of azimuth axis, the analysis of static characteristics such as structural strength and deformation is one of the problems that must be solved. A lot of practical experience shows that the static analysis of structure is one of the important contents of structural design analysis. In literature [1], taking the output shaft of a reducer as the research object, the static characteristics were analyzed by using finite element software. In literature [2], the static characteristics of a motorized spindle rotor bearing system were analyzed by using the finite element software. According to the calculation results, the influence of the position of the front and rear bearings and the aperture on the static performance of the motorized spindle was analyzed. In literature [3], taking a new type of rubber coupling as the research object, the static characteristics of the coupling were studied by using ABAQUS software, and the influence of the number of wedges and hole structure on the structural strength and static characteristic parameters of the coupling was analyzed. In literature [4], based on the analysis of cutting conditions, loads and constraints, the static characteristics of the supports of horizontal lathe were studied by using finite element method, and the relationship between the geometric structure types of the supports and their static characteristics was summarized. In literature [5], the static characteristics of a direct drive turntable under different working conditions were analyzed by using the finite element method.
Through the analysis, it is confirmed that the static stiffness of the turntable meets the design requirements. In literature [6], the static characteristics of a machine tool bed were analyzed by SolidWorks and ANSYS Workbench software, and the tool displacement value caused by the bed deformation was obtained through the analysis. In literature [7], taking a self-designed spindle of a micro CNC lathe as the research object, the static characteristics of the spindle were analyzed by using the finite element software ANSYS Workbench. The analysis results laid a foundation for the subsequent optimization design of the spindle. In literature [8], the strength and stiffness of a direct drive CNC turntable spindle were analyzed by using the traditional analysis and calculation method and the finite element analysis theory. The analysis results show that the strength and stiffness of the spindle meet the requirements. In literature [9], the static characteristics of a hydrostatic radial gas bearing were analyzed by numerical analysis method. It is found that the speed of the journal has an important influence on the static characteristics of the bearing. Based on the finite element analysis theory, the static characteristics of the frame of a tobacco stalk drawing crusher were studied in literature [10], and the optimal design of the frame was completed on the basis of the analysis results. In this paper, based on the analysis of the forces acting on the azimuth axis of a radar antenna support, the static characteristics of the azimuth axis were analyzed by using the finite element software, and the influence of structural size parameters on its static characteristics was also discussed.

2. Structural sketch of the azimuth axis

Azimuth axis is an important part of radar antenna support, which is mainly used to adjust and determine the radar azimuth. Its structure is shown in Fig. 1. The azimuth axis is a step shaft, which mainly bears the torque caused by the motor starting in the working process. The shaft is made of Q345 steel, and its material characteristic parameters are shown in Tab. 1. The yield strength of the material is 345 MPa. When the safety factor is 1.34, the allowable stress of the material is 257 MPa.

![Fig. 1 Structural diagram of azimuth axis](image)

Tab. 1 Material characteristic parameters of Q345

| Characteristic parameters | Density/Kg/mm³ | Elastic modulus/MPa | Poisson's ratio |
|--------------------------|----------------|---------------------|----------------|
| Values                   | 7.85×10⁻⁶      | 2.1×10⁵             | 0.3            |

3. Static characteristics analysis of the azimuth axis

Firstly, the finite element model of the azimuth axis was established in the finite element software ANSYS, as shown in Fig. 2. When modeling, the small structures such as undercut and chamfering on the axis were ignored. Considering that the axis structure is a symmetrical body of revolution, the sweeping meshing method was adopted to mesh the axis, and the solid 45 element was used to simulate the axis, and the whole axis was divided into 18008 elements with 25212 nodes.
After the establishment of the finite element model, the load and constraint were carried out according to the actual working state of the azimuth axis. The specific constraints and loads are as follows: the full constraint was applied on the fixed end face of the azimuth shaft, and the torsional load was applied at the joint of the output shaft of the azimuth drive motor.

After the analysis, the stress and deformation calculation results were extracted from the general post processor, as shown in Fig. 3 and 4.

Fig. 2 Finite element model of the azimuth axis

Fig. 3 Nephogram of stress of the azimuthal axis
It can be seen from Figure 3 that the maximum working stress of the azimuth shaft under the action of torsion load is 73.321 MPa, which occurs at the top of the azimuth axis. Compared with the allowable stress value of the material, it can be found that the maximum working stress value is less than the allowable stress of the material. Therefore, the static strength of the azimuth shaft meets the use requirements.

It can be seen from Fig. 4 that the maximum deformation of the azimuth axis under the action of torsion load is about 0.044 mm, which appears at the top of the azimuth axis. Generally speaking, the deformation distribution presents the following trend: the deformation of the axis decreases from the top to the bottom of the azimuth axis.

### 4. Influence analysis of structural size parameters

In order to analyze the influence of structural size parameters of azimuth axis on its static characteristics, and provide data reference for subsequent structural optimization design, the model of the azimuth axis with different values of relevant size parameters (as shown in Tab. 2) was selected in this paper (when the value of one parameter is changed, the value of other parameters remains unchanged). And the loading analysis was carried out according to the above constraints and loads. After the analysis, the maximum working stress of azimuth model was extracted, and the curve of maximum stress changing with parameters was drawn, as shown in Fig. 5.

| Size parameters | $d_{11}$ | $d_{12}$ | $l_{3}$ |
|-----------------|----------|----------|--------|
| Values 1        | 98       | 66       | 578    |
| Values 2        | 100      | 68       | 580    |
| Values 3        | 102      | 70       | 582    |
| Values 4        | 104      | 72       | 584    |
It can be seen from Fig. 5 that with the increase of size parameter $d_{11}$, the maximum working stress of azimuth axis decreases. With the increase of size parameter $d_{32}$, the maximum working stress of azimuth axis increases. With the increase of size parameter $l_3$, the maximum working stress of azimuth axis decreases unevenly.
5. Conclusions
In this paper, the finite element software ANSYS was used to analyze the static characteristics of the azimuth axis of a radar antenna support, and the influence of the structural size parameters of the azimuth axis on its static characteristics was also studied. Through the research, it is found that:

(1) Under the normal working load, the maximum working stress of the azimuth axis is less than the allowable stress of the material, and its static strength can meet the use requirements, and the deformation distribution presents the following trend: the deformation of the axis decreases from the top to the bottom of the azimuth axis.

(2) When the structural size parameters of azimuth axis change, the maximum working stress value is positively correlated with the size parameter $d_{32}$, and negatively correlated with the size parameters $d_{11}$, $l_3$.

Acknowledgments
This research was supported by the Major R & D project of Sichuan Science and Technology Department under the contact number 18ZDYF1080, the National Natural Science Foundation of China under the contact number 51705041 and Introduction of Talents to Start Scientific Research Projects of Chengdu Technological University under the contact number 2017RC009.

References
[1] Wen G., Liu P.P., Su R., et al. (2019) Static Optimization Design of a Reducer Output-axle based on Sensitivity Analysis. Journal of Chengdu Technological University, 22, 2: 1-4.
[2] Xu J.W., Li W.B.. (2015) Analysis on Static and Dynamic Characteristics of Electric Spindle Based on ANSYS. Machinery Design & Manufacture, 9: 9-11+17.
[3] Qiu Y.T., Duan Y., Gu X.D.. (2020) Design and Static Characteristic Analysis of a New Rubber Coupling. Journal of Mechanical Transmission, 44, 8: 103-107.
[4] Dong H.M., Wang H.Y., Wang D.S., et al. (2013) On Structure and Performance for Machine Tool Lightweight Design-Structural and Static Characteristics of Components in Horizontal Lathe. Modular Machine Tool & Automatic Manufacturing Technique, 6: 1-5.
[5] Jin L., Du Y.K., Gou W.D., et al. (2014) Dynamic and static characteristic research on direct drive rotary table considering axial stiffness and radial stiffness. Journal of Machine Design, 31, 6: 72-76.
[6] Shao Y.H., Hong R.J., Yu C.J.. (2020) Static characteristics analysis of machine tool bed performance. Journal of Nanjing Tech University (Natural Science Edition), 42, 1: 110-114.
[7] Tan F., Yin G.F., Fang H., et al. (2015) The Finite Element Analysis of the Desktop CNC Lathe’s Spindle Based on the ANSYS Workbench. Modular Machine Tool & Automatic Manufacturing Technique, 4: 29-32+36.
[8] Luo Y., Gao Z.C., Li L.J., et al. (2019) Analysis of Static Characteristic for Direct-driven CNC Rotary Table Spindle. Machine Tool & Hydraulics, 47, 8: 42-47.
[9] Wang X.K., Xu Q., Zhang L.X., et al. (2019) Numerical Analysis of the Static Performance of Aerostatic Journal Bearings. Machinery Design & Manufacture, 5: 14-17+21.
[10] Shu C.S., Zhang D.B., Zhang Y.F., et al. (2019) Tobacco stalk pulling and shredding machine frame lightweight design and dynamic and static characteristic analysis. Modern Manufacturing Engineering, 6: 55-61+67.