Finite element analysis of thrust angle contact ball slewing bearing

Biao Deng, Yuan Guo*, An Zhang, Shengjin Tang
Xi'an Institute of high technology, Xi’an 710025, China

*Corresponding author e-mail: orchard1993@126.com

Abstract. In view of the large heavy slewing bearing no longer follows the rigid ring hypothesis under the load condition, the entity finite element model of thrust angular contact ball bearing was established by using finite element analysis software ANSYS. The boundary conditions of the model were set according to the actual condition of slewing bearing, the internal stress state of the slewing bearing was obtained by solving and calculation, and the calculated results were compared with the numerical results based on the rigid ring assumption. The results show that more balls are loaded in the result of finite element method, and the maximum contact stresses between the ball and raceway have some reductions. This is because the finite element method considers the ferrule as an elastic body. The ring will produce structure deformation in the radial plane when the heavy load slewing bearings are subjected to external loads. The results of the finite element method are more in line with the actual situation of the slewing bearing in the engineering.

1. Introduction
Slewing bearings are widely used in hoisting and conveying machinery, mining machine, port machinery, wind power, radar and missile launchers and other large rotary device, It can also bear larger axial load, radial load and overturning moment load, stress is large, the speed is slow, the characteristics of dynamic load can be ignored. In recent years, the mechanical calculation of turntable bearings has attracted the attention of relevant researchers. For small bearings, usually under the assumption of rigid ferrule, that only the ball and the raceway contact with local elastic deformation, in the Hertz contact theory on the basis of the bearing to study the contact stress and deformation [1] [2]. Potonik P [3]uses the vector to represent the geometric structure of the double-row, four-point contact ball turntable bearing, establishes the static equilibrium equation of the contact load distribution, and obtains the rolling body load by solving the equation. Srecko Clodez [4]establish a turntable bearing static model basing on the vector representation, and calculating the contact surface stress. These studies work under the assumption of rigid ferrule to establish the hydrostatic equation of the turntable bearing, and obtain the load of the rolling element by solving the system of equations.

The development of computer technology has led to the application of finite element technology in the analysis of turntable bearings. In order to overcome the shortcomings of the rigid ferrule theory on the turntable bearing analysis, the researchers use finite element analysis software to analyze and calculate the turntable bearing. Daidie [5] proposed using the elastic rod element instead of the rolling element to analyze the bearing internal load distribution. Smolnicki [6] uses the supercell to represent
the role of "raceway - rolling body - raceway" to analyze the effect of support structure deformation on bearing load distribution. MA Jianhai [7] proposed a method to calculate the contact load considering the elastic deformation of bearing rings and their supporting structures, and improved the computational efficiency of the finite element method. Rui Xiaoming [8] established the deformation equation of the turntable bearing ring, and the method of solving the distribution load of the turntable bearing is given by the simultaneous nonlinear transcendental equations. In the finite element analysis of the turntable bearings [9], the rolling body is simplified as a nonlinear compression spring to solve the bearing internal load distribution. Li Yunfeng [10] [11] uses the finite element method to analyze the turntable bearing and calculate the load distribution of the bearing. These studies effectively simplify the finite element model of turntable bearings.

The purpose of this paper is to use the finite element analysis software ANSYS to establish the finite element model of the thrust angular contact ball bearing, and calculate the force condition of the bearing.

2. Finite element analysis of turntable bearings

Figure 1 shows the thrust angular contact ball turntable bearing structure diagram.

Fig. 1 Specification of turntable bearing structure

2.1. Solid geometry modeling

In order to reduce the size of the finite element calculation of the turntable bearing as much as possible, to improve the calculation efficiency, in the finite element modeling for the following considerations: Using the symmetry of the turntable bearing structure and the symmetry of the load, a 1/2 model of the turntable bearing was established; Ignore the bolt connection hole and drive structure on the ferrule. In the ANSYS software, the process of finite element modeling of the ferrule is as follows: First, in the axial section of the turntable bearing, create a number of key points on the profile outline. Secondly, in the axial section of the turntable bearing, connect the defined adjacent key to obtain the contour of the axial cross section of the ferrule. Finally, the inner ring section around the bearing axis rotation sweep 180° to generate the inner ring entity; the outer ring section around the bearing axis rotation sweep 180° to generate the outer ring entity. When modeling a steel ball, move the work plane to the center of the first ball and create a sphere. In the column coordinates in accordance with a certain circumference to copy all the ball.

2.2. Mesh and touch settings

Mesh division is the key link of finite element analysis. Whether the texture is reasonably divided according to the structural requirements is directly related to the quality of the finite element model, which affects the calculation speed and the result precision. The inner ring and outer ring are all rotating body, contains a lot of circular structure. The use of swept or hexahedral elements to divide the grid will produce irregular and deformed elements, resulting in convergence difficulties. Therefore, the swept division grid and the hexahedral element are not used here. Solid187 unit is a high-order three-dimensional 8-node solid structure unit. This type of unit has a quadratic displacement type function, which can be used in complex models with strong adaptability while ensuring the accuracy of the calculation results. It is suitable for the model of circular boundary. Therefore, this paper chooses the Solid187 unit and meshes the bearing model by using the method of free meshing. And
then specify the number of copies of each contour line unit. The number of parts of the inner and outer ring and the rolling contact parts should be properly encrypted in order to improve the calculation accuracy.

Bearing rolling body and the inner and outer ring is a typical contact analysis problem. The contact area cannot be determined when no solution is made. When the load and boundary conditions change, the area of contact will also change. Since the face-to-face contact unit has both low and high order elements, it allows large deformation, large sliding, cell asymmetry, does not limit the shape of the contact surface, and the contact unit is relatively small. So the ball between the ball and the collar to choose surface-to-face contact unit. The inner and outer ring raceway surface is set as the target surface, using Target170; select the ball surface for the contact surface, using Contal169 unit.

2.3. Set the boundary conditions

In the actual work process, the outer ring of the turntable bearing is fixed on the frame. Therefore, the full diameter limit of the upper end of the outer ring of the turntable bearing is applied to ensure that the displacement is fixed. Due to the 1/2 model of the turntable bearing, it is necessary to apply a symmetrical constraint to the symmetrical cross section of the turntable bearing so that the nodes on the section can not move outside the plane and cannot rotate in the plane. During the analysis, the circumferential displacement of the steel ball is restrained in order to prevent the steel ball from moving in the circumferential direction of the raceway.

In the course of work, the slewing bearings bear both axial load, radial load and overturning moment. In order to facilitate the application of the load and at the same time do not affect the analysis results, all the nodes on the top surface of the slewing ring bearing are coupled so that they have the same degree of freedom. When loading, axial load, radial load and overturning moment load are applied to one node of the coupling node.

3. Example analysis

The structural parameters of thrust angular contact ball slewing bearings used in a launch pad are shown in Table 1. The axial load $F_a=730$KN, radial load $F_r=750$kN, overturning moment $M=338$kN·m when bearing is working. Bearing material elastic modulus $E=2.12\times10^5$Mpa, Poisson's ratio $\nu=0.3$.

| Structural parameter                  | value   |
|---------------------------------------|---------|
| external diameter $D$/mm              | 2175    |
| internal diameter $d$/mm              | 2075    |
| pitch diameter $d_m$/mm               | 2125    |
| ball diameter $D_w$/mm                | 39.5    |
| bearing height $H$/mm                 | 170     |
| radius of curvature radius of inner raceway $f_i$ | 0.542 |
| radius of curvature radius of outer raceway $f_o$ | 0.542 |
| number of steel balls $Z$             | 298     |
| contact angle $\alpha$/°              | 60      |

In view of the working conditions of the turntable bearing, a finite element model of the bearing is established, and the contact between the steel ball and the ferrule is set up in the model, as shown in Figure 2. By solving the finite element model by nonlinear calculation, the contact stress of the raceway is calculated, and the results are shown in figure 3–5.
Figure 3 shows the overall bearing contact stress isogram, can be seen that under the combined action of axial load, radial load and overturning moment, two contact points exist between the steel ball and the raceway, and there are two contact areas; The maximum contact stress occurs at the contact point between the steel ball and outer raceway, its value is 1372.6MPa.

As shown in Figure 4, the contour of the contact stress for the raceway of the inner race shows that the maximum contact stress in the inner race occurs at the contact point between the left raceway of the inner race and the steel ball, the maximum contact stress value is 1372.6MPa.

As shown in Figure 5, the contour of the contact stress of the raceway of the outer raceway shows that the maximum contact stress in the outer race occurs at the contact area between the ball and raceway at the outer right, the maximum contact stress value is 1013.8MPa. It can be seen that the contact stress on the raceway surface of the inner race is greater than that of the outer raceway.
Through the finite element modeling and analysis of the turntable bearing, in addition to calculating the stress distribution inside the ring, the load of each steel ball can also be obtained. The result is shown in Figure 6. The solid line in the figure is calculated by the finite element method in this paper. The dashed line is the result of the numerical method of the traditional bearing theory. The results obtained by the two methods are similar, but there are differences. The results obtained by the numerical method of the traditional bearing theory are smooth curves, which is due to that numerical calculations treat bearing rings as purely rigid bodies, the finite element method considers the ring as an elastic body, and the structure of the ring will deform in the radial plane under the action of the external load. Therefore, the calculated result curve is different from that obtained by numerical method. In the calculation results of finite element method, more steel balls bear the load, and the maximum load of the steel ball is reduced, which is more consistent with the actual force of the bearing.

4. Conclusion
The internal force of thrust angle contact ball slewing bearing was analyzed by using finite element analysis software ANSYS. Calculation results show that:

(1) Under the combined load, there is a point contact between the steel ball and the raceway, and there is a contact area between each steel ball and the raceway.

(2) Compared with the numerical results based on the rigid ring assumption, the results of the finite element method show that more steel balls bear the load and the maximum load decreases.

(3) The contact stress on the raceway of the inner race is greater than that of the outer raceway, which is in line with the actual situation of the project.

(4) The internal force of slewing bearing may also be affected by bolt connection, radial clearance and groove curvature radius coefficient, so further research is needed.
Considering the deformation of the bearing ring structure and the influence of the boundary conditions, a finite element analysis model of the slewing bearing is established. Compared with the traditional bearing theory, the rigid ring assumption is more consistent with the actual bearing force of the slewing bearing.

References
[1] Amasorrain J I, Sagartzazu X, Damián J. Load distribution in a four contact-point slewing bearing [J]. Mechanism & Machine Theory, 2003, 38 (6):479-496.
[2] Zupan S, Prebil I. Carrying angle and carrying capacity of a large single row ball bearing as a function of geometry parameters of the rolling contact and the supporting structure stiffness [J]. Mechanism & Machine Theory, 2001, 36 (10):1087-1103.
[3] Potočnik R, Göncz P, Flašker J, et al. Fatigue life of double row slewing ball bearing with irregular geometry [J]. Procedia Engineering, 2010, 2 (1):1877-1886.
[4] Glodež S, Potočnik R, Flašker J. Computational model for calculation of static capacity and lifetime of large slewing bearing’s raceway [J]. Mechanism & Machine Theory, 2012, 47 (1):16–30.
[5] Daidié A, Chaib Z, Ghosn A. 3D Simplified Finite Elements Analysis of Load and Contact Angle in a Slewing Ball Bearing [J]. Journal of Mechanical Design, 2008, 130 (8):1188-1188.
[6] Smolnicki T, Derlukiewicz D, Stańco M. Evaluation of load distribution in the superstructure rotation joint of single-bucket caterpillar excavators ☆ [J]. Automation in Construction, 2008, 17 (3):218-223.
[7] Mao Fanhai, Zhang Jian, Dong Huimin. Calculation of contact load distribution of slewing bearing with consideration of support deformation [J]. combined machine tool and automatic machining technology, 2016 (3): 20-24.
[8] Rui Xiaoming, Zheng Hui, Raymond Wong. Study on load distribution of variable pitch bearing in wind turbine generator [J]. proceedings of the Chinese society of electrical engineering, 2013, 33 (32): 80-87.
[9] Shang Zhengu, Dong Huimin, Mao Fanhai. Finite element analysis method of turntable bearing with plastic deformation [J]. Journal of agricultural engineering, 2011, 27 (12): 52-56.
[10] Li Yunfeng, Liu Caixia, Chen Hongtao. Finite element analysis of double row four-point contact ball slewing bearings [J]. Harbin Bearing, 2015 (2): 3-6.
[11] Li Yunfeng, Jiang Di. Finite element analysis of roller slewing bearings of six row [J]. bearings, 2015 (5): 7-12.