FEM Analysis of Contact Characteristics of the Joint Bearings

Shuo Han[1], Baogang Wen[2]*, Qingkai Han[1]

1. School of Mechanical Engineering and Automation, Northeastern University, Shenyang, China 110819
2. School of Mechanical Engineering and Automation, Dalian Polytechnic University, Dalian, China 116034

Abstract: Joint bearing is a kind of sliding bearing with spherical surface, which can swing in any direction within a certain angle range. It is widely used in various mechanical equipment, aerospace, industrial robots and other fields. The load of the joint bearing is large than that of rolling bearing, its running speed is low, and its rotation angle is small. The contact characteristics of the spherical bearing under load reflect its bearing capacity and dangerous area. In order to analyze the mechanical characteristics of the joint bearing under the radial load, the finite element model of the joint bearing is established by using the finite element software ANSYS. The stress field distribution of joint bearing under radial load is analyzed. The results show that the maximum von Mises stress appears at the spherical edge of the outer ring of the joint bearing under the radial load, and the maximum von Mises stress of the outer ring is greater than that of the inner ring.

Keyword: Joint bearing; Finite element; Contact analysis

1 Introduction

The joint bearing is characterized by simple structure and small volume. The contact surface between the inner and outer rings of the joint bearing is spherical, so the contact area of the inner and outer rings is large when it works, and it can adjust the center automatically. Therefore, it is suitable for swing motion, tilt motion and rotary motion with low concentricity and high surface pressure and low speed. It is widely used in engineering, lifting, agricultural machinery, trucks and other fields, especially in recent years, the joint bearing has been widely used in aerospace fields, such as Boeing series aircraft control transmission system of Boeing company in the United States The use rate of the joint bearing is almost 100%. The joint bearing is also widely used in the control transmission system and engine system. At present, the demand for high performance and high reliability joint bearings is very urgent at home and abroad.

* Corresponding author (wbg_dlut@163.com)

In the view of the contact characteristics of the joint bearing, scholars at home and abroad have made a lot of efforts and research work in this field by using the finite element method. Liu Liujing,et al.[1] take the large-scale thrust joint bearing as the research object, applied the ANSYS finite element software to establish the bearing's finite element model. According to the different working conditions of the joint bearing, the stress-strain distribution of the inner and outer ring of the thrust joint bearing and the contact pressure distribution are studied under different loads (axial, radial and composite loads). Xiang Dinghan, et al.[2] use ANSYS finite element method to optimize the structure of the radial joint bearing gez101es. A set of finite element models are established by changing the size of the contact sphere of the inner and outer rings, and the maximum equivalent stress and the maximum contact pressure between the inner and outer rings are calculated. By comparing the calculated results, the spherical dimension with the lowest pressure value is obtained. In literure[3], the contact problem between cylindrical conformal surfaces, modelling for instance a fastener joint, is studied. A closed form solution is obtained for the case of elastic similarity, improving the solution obtained by Persson. Oswald[4] investigated the effect of internal clearance on radially loaded deep-groove ball and cylindrical roller bearing load distribution and fatigue life for four clearance groups defined in the bearing standards. It is quite difficult to accurately calculate the reduction of radial clearance of radial spherical plain bearings after assembly. Based on empirical data, a numerical approximation method is used to provide a quick evaluation method of the effect of coordination on radial radial clearance of radial spherical plain bearings[5].

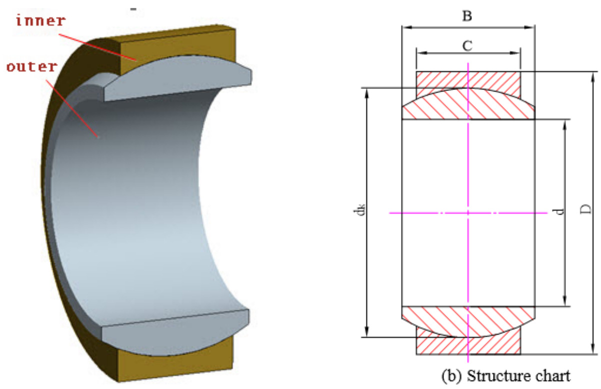
In this paper, the contact characteristics of the joint bearing are studied based on the finite element simulation analysis. The finite element model of the joint bearing is established by the three-dimensional software Pro / E and the finite element software ANSYS, and the static contact is calculated. Finally, according to the calculation results, the influence of the contact characteristics such as the stress and strain of the joint bearing is analyzed.

2 Structural features of the joint bearings

The joint bearing is a kind of spherical roller bearings, which is usually composed of an outer ring with concave spherical surface and an inner ring with convex spherical surface. The bearing studied in this paper is ge45es type bearing, its three-dimensional diagram and structural diagram are shown in Fig.1, and its size parameters are shown in Table 1.

Table 1 Bearing size parameters of GE45ES units（mm）

Code	D	d	B	C	d_k
Size	Φ68	Φ45	32	25	60



(a)3D Diagram (b) Structure chart

Fig.1 Structure of joint bearing

3 Finite element analysis of joint bearing

In order to simulate the static experimental conditions of the joint bearing, the mandrel needs to be added to the model, but in general, the strength fatigue failure of the joint bearing does not occur on the mandrel in the actual work (the strength and stiffness are generally large). Therefore, this paper only analyzes the performance of the inner and outer rings of the joint bearing. The force analysis of the joint bearing is a typical non-linear contact problem. The ANSYS has a strong non-linear analysis ability. The calculation results are generally consistent with the actual situation, and can meet the engineering and technical requirements. Therefore, the stress analysis of the joint bearing using ANSYS can obtain more intuitive results. In this section, taking the radial load as an example, the establishment process of the finite element simulation model is introduced.

3.1 Establishment of finite element model

In order to calculate accurately, the full-scale real model is used in this paper, but in order to analyze the effectiveness, it is generally necessary to simplify the original model properly. This paper does not consider some small features of the joint bearing, For example, the transition fillets on the inner and outer rings of the bearing are ignored, because if these fillets are retained in the model, in addition to simply increasing the number of grid cells and nodes, increasing the difficulty of grid division, and wasting computer working time and storage space, the stress field distribution of the joint bearing is basically not greatly affected, and the grooves on the side of the oil groove and outer ring are also ignored.

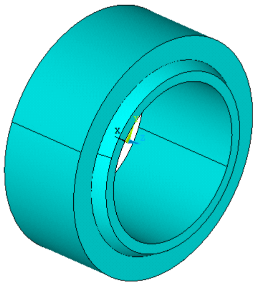


Fig.2 Finite element model of joint bearing

3.2 Definition of units and material properties

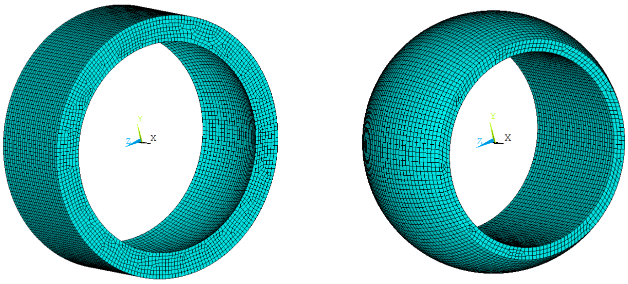
According to Table 1, APDL command flow method is used to establish the finite element model in ANSYS. The solid model is meshed by using 8-node discrete 3D solid element solid45. There is a node at each vertex of the cube, 8 nodes in total, and each node has 3 translational degrees of freedom, i.e. translation in X, Y and Z directions. Set the material properties of each part as shown in Table 2, and set the material as an isotropic linear model.

Table 2 Bearing size parameters of GE45ES

Components	Material	Elastic Modulus （GPa）	Poisson's ratio	Density （kg/m^3）
Bearing inner and outer ring	GCr15	215	0.3	7800
Mandrel	40Cr	206	0.3	7800

3.3 FEM meshing

The model meshing is a very important part in finite element analysis. Whether the mesh division is reasonable or not has a great influence on whether the appropriate finite element model can be established, which directly determines the speed of computer calculation and the accuracy of calculation results. Based on the finite element model of the joint bearing established in this paper, the mandrel can be replaced by a rigid surface. The inner and outer rings contain many circular arc structures, and the hexahedron element can be used, and good calculation results can be obtained. In this paper, solid grid is divided into three-dimensional solid grid by solid45 element, and the inner and outer rings and the mandrel of the bearing are all divided by the method of body scanning. The idea of this method is to scan a section of cross-section mesh through the whole solid so as to make the solid generating unit that has not been divided into mesh. Because only the inner and outer rings of the bearing are analyzed, the grid size of the inner and outer rings is defined as 1mm, and the grid size of the mandrel is 1.5mm. There are 89855 dividing units and 87224 nodes. The meshed model is shown in Fig. 3.



(a)outer ring (b) inner ring

Fig.3 Finite element mesh generation model of the joint bearing

3.4 Contact settings

The contact between the inner and outer rings, the inner and the mandrel of the bearing is a highly nonlinear problem. When the boundary conditions, load types and loading methods change, the contact parts will also change. In this paper, the surface contact element is chosen as the element of bearing contact. Generally, the principles to select the target surface and contact surface are as follows: when the convex surface contacts the concave surface or the plane surface, the concave surface should be selected as the target surface; when a surface with large mesh density contacts with a surface with small mesh density, the rough surface of the mesh unit should be selected as the target surface. According to the selection principle of the target surface and the contact surface, when setting the contact pair for the joint bearing, the inner surface of the outer ring and the outer surface of the mandrel are taken as the target surface respectively, and the outer and the inner surfaces of the inner ring are taken as the contact surface. The contact part is defined by the contact units of target170 and contact174, and two groups of contact pairs are established successively, as shown in Fig. 4. In this paper, the friction type between contacts is set to frictional, the friction coefficient is set to 0.15, and the enhanced Lagrangian algorithm is used as the contact algorithm.

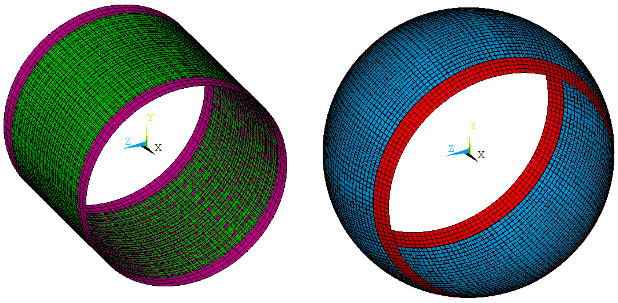


Fig.4 Contact pairs of the finite element model of the joint bearing

3.5 Loading and restraint of bearings

The rated static load 637kn is applied to the axis of the mandrel. In order to make the calculation easily to converge, the concentrated force in the direction of - Y is used to constrain the outer ring of the bearing, and in order to fix the bearing, both sides of the mandrel are fully constrained. In this paper, the contact algorithm is chosen as augmented Lagrange multiplier method, and the load step and time step are set. So far, the finite element contact model of the joint bearing has been established.

Because the journal bearing mainly bears radial load and only bears small axial load, the ratio of FA / FR tends to 0, so the approximate treatment is P = Fr. The rated static load of the bearing is 637kn. Because its static performance is only analyzed, the rated static load is selected as the radial load and applied to the central area of the mandrel in the form of face load. When setting the boundary conditions for the joint bearing, the two ends of the mandrel shall be completely fixed according to the actual working conditions. Since the mechanism has no base support, in order to prevent the mandrel from bending too much, the outer surface of the bearing outer ring shall be restrained by the concentrated force in the direction of -Y and its loading and restraint settings are as shown in Fig. 5. Since the bearing only bears radial load (corresponding to Y direction in the Fig.), all deformation is symmetrical distribution, it is not necessary to set the displacement constraint in X, Y direction of the part.

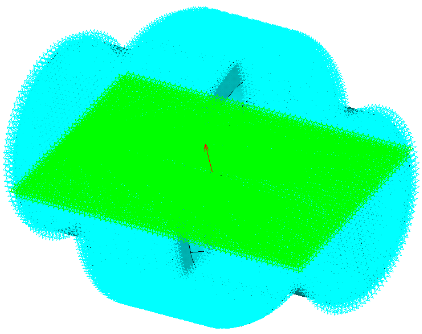


Fig.5 The constrained boundary conditions of the FEM model of joint bearing

4 Results analysis

The finite element model of the above-mentioned joint bearing is simulated by the finite element software ANSYS, and the performance data of inner and outer ring stress distribution, contact pressure and radial displacement are obtained.

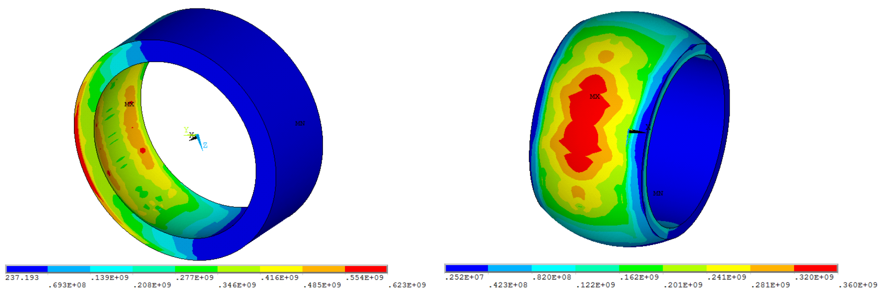


Fig.6 Von Mises stress distribution in inner and outer ring of the spherical bearing

The von Mises stress distribution in the inner and outer ring of the spherical plain bearing is shown in Fig. 6. The maximum von Mises stress of the inner ring appears in the middle of the contact part between the inner ring and the outer ring, with a value of 360MPa; the maximum von Mises stress of the outer ring appears at the boundary of the contact surface of the inner ring and the outer ring, with a size of 632mpa. The main reason for this phenomenon is that when the limit static load is directly applied on the inner ring, the maximum stress of the outer ring appears at the boundary position because the width of the inner ring is wider than that of the outer ring. The outer ring can be regarded as a "simply supported beam", and the contact part between the outer ring and the inner ring will bear a supporting force opposite to the direction of the static load concentration force in addition to the static load. So the equivalent stress of inner ring is larger than that of outer ring.

Fig. 7 shows the displacement distribution of the joint bearing in Y direction, with the maximum displacement of 0.0224mm. Under the action of radial load, the inner ring moves upward, which makes the outer ring and the inner ring produce large elastic deformation in the local contact area.

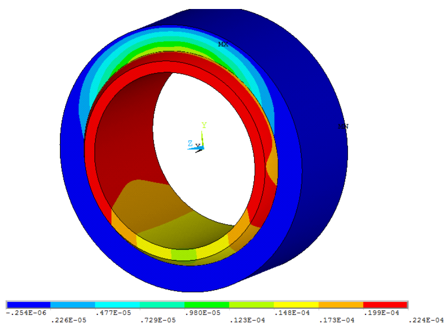


Fig.7 Displacement of joint bearing in the Y-direction

 ISSN 2572-4975 (Print), 2572-4991 (Online)

It is very important to analyze the contact pressure distribution between the inner and outer rings when the joint shaft is used to analyze the strength and optimize the structure of the joint bearing. According to Fig. 8, the maximum contact pressure occurs in the middle of the contact surface between the inner ring and the outer ring, with the size of 609mpa.

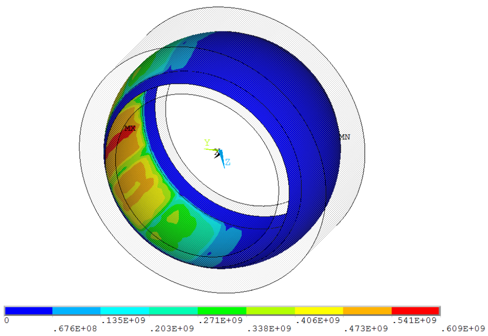


Fig.8 Distribution of contact pressure

5 Conclusions

In this paper, the joint bearing is taken as the research object. Several conclusions are obtained as follows:

(1) The maximum contact stress is 609mpa and the maximum deformation displacement in Y direction is 0.0224mm for a certain type of radial joint bearing under the limit static load without considering the internal clearance of the joint bearing.

(2) It is still difficult to use the analytical method to check the strength and optimize the structure of the joint bearing.

(3) However, ANSYS can be used to solve the stress and strain distribution of the joint bearing under the static load. The calculated results can provide some reference for the strength design and structure optimization of the bearing.

References

1. Liu L J, Gao W, Finite element analysis on structure of the large-scale thrust joint bearing, ***Ship & Ocean Engineering,***2004(2):25-27.
2. Xiang D H, Wang C Y, Dong W F, Structural optimization and friction and wear behavior of a radial spherical plain bearing, ***Tribology,*** 2004, 24(6):564-567.
3. Ciavarella M, Decuzzi P, The state of stress induced by the plane frictionless cylindrical contact. I. The case of elastic similarity,***International Journal of Solids and Structures,***2001,38(26): 4507-4523.

4. Fred B O, Erwin V Z, Joseph V P, Effect of internal clearance on load distribution and life of radially loaded ball and roller bearings,***Tribology Transactions,*** 2012, 55(2): 245-265.
5. Lu J Z, Quick evaluation method of radial clearance change after centripetal bearing assembly, ***Bearing,*** 2012(12):5-6.