Vels Journal Of Mechanical Engineering
Vol-2, Issue-2, July 2015

CONTACT ANALYSIS OF ROLLER BEARING USING FINITE ELEMENT METHOD

T.Gopalakrishnan[1*], Ruban.M[2]

[1,2]Assistant Professor, Department of Mechanical Engineering, VELS University, Chennai . Tamilnadu. India

[1*]gopalakrish185@gmail.com, [2]rubanmurugesan@gmail.com

Abstract- *Roller Bearings are a type of rolling-element bearing that uses cylinders (rollers) to maintain the separation between the moving parts of the bearing (as opposed to using balls as the rolling element). This research study has been carried out for the contact analysis of roller bearing using CAE tools CATIA and Ansys. Through contact analysis the changes could be showed in stress, strain, penetration, sliding distance, friction stress on the roller bearing. Furthermore, the simulation results revealed that the computational values were consistent with theoretical values. The all showed that the model and boundary conditions were correct and rational, and it would provide a scientific basis for optimum design of rolling bearings under complicated loads.*

Keywords— Roller bearing, Contact analysis, Finite Element Method, CATIA, Ansys

I.INTRODUCTION

The term "rolling bearing" includes all forms of roller and ball bearing which permit rotary motion of a shaft. Roller bearings structure is simple and is widely used in engineering field. Rolling bearings are high precision, low cost but commonly used in all kinds of rotary machine. Its main failure mode is contact fatigue spalling of rolling elements. The contact finite element analysis can show bearings' information under contact, such as contact stress, strain, penetration and sliding distance, and so on, which play a significant role in optimum design of complicated rolling bearings. The analysis of contact problems is a major concern in many engineering applications such as ball bearings, gears, rollers and pressure vessel attachments. Many practical engineering problems involve the contact of bodies with non-conforming contact situations, and are loaded by tangential as well as normal forces. The numerical modelling of practical contact problems requires special attention because the actual contact area between the contacting bodies is usually not known in advance. With the change of load, material, boundary condition or the other factors, touch or separation will take place between surfaces, most frictional effects on contact problems are needed to be considered. They may be disordered as well as nonlinear.

ANSYS gives a good blue print for contact analysis which can take friction heat and electrical contact into account. It also has a special contact guide which is conveniently for creating contact pairs. The internal expert system of contact analysis does not require any settings of related contact parameter in a general contact analysis. So it can easily establish contact analysis.

II.THE CONTACT STATE ANALYSIS AND THEORETIC CALCULATION FOR ROLLER BEARING

When ball bearing works, it is usually that more than one rolling ball bears the load. The condition is complex between rollers and rings. When the load is 0, the contact area is a point, i.e., point-contact. When the load increases in running, the bearing inner ring, outer ring and rolling elements bring forth plastic deformation in the contact area, so the point-contact becomes face-contact. Furthermore, contact area gradually becomes ellipse, and generates residual stress. The contact parameters, such as the place, size, shape of contact area, as well as the contact pressure and friction force distribution, will be variable with loads change. These are typical boundary nonlinear problems. On the base of Hertzian theory, deep groove ball bearing's contact stress is:

$$P_m = \frac{1}{\pi e_a e_b}\sqrt[3]{\left(\sum \rho\right)^2 \times \frac{5F_r}{Z\cos\alpha}}$$

In the formula,e_a,e_b is respectively Hertzian contact coefficient; $\sum\rho$ is the sum of main curvature; F_r is the

radial load; and Z is the number of rolling elements; α is contact angle under loads.

III.THE BASIC STEPS IN THE BEARING'S CONTACT ANALYSIS

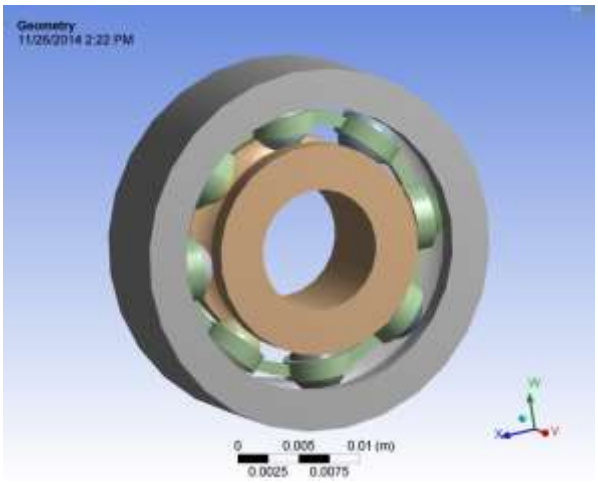
In contact problem involved two boundaries, it is natural that take one boundary as target surface and take the other one as contact surface. Surface-surface contact is very suitable for those problems just as: interference fitting installation, or embedded contact, forging and deep-drawing. Typical surface-surface contact's analysis steps mainly include:

1. Build 3D geometry model and mesh
2. Identify contact pairs
3. Name target surface and contact surface
4. Define target surface
5. Define contact surface
6. Set up element key options and real constants;
7. Define and control rigid goal's movement (only applicable in rigid-flexible contact)
8. Apply the necessary boundary condition
9. Define solution options and load steps
10. Solve contact problems
11. Look over and analyze results.

ANSYS supports surface contact elements of rigid-flexible or flexible-flexible. The elements form contact pairs by using target and contact surface. For the rigid-flexible contact, it can be chosen as contact surface such as convex surface, dense meshing or little size surface, otherwise chosen as target surface.

IV.FINITE ELEMENT MODEL

By use of CATIA The model is built and imported into ANSYS, in theory, it is feasible that model can be changed with model's parameters change such as material property, restrained displacement and applied load, and so on. Roller bearing is mainly composed of inner ring, outer ring, rolling elements and cage. Due to the different conditions, some bearings have snap ring groove, shield, seal, bearing peak thread, and soon. Because bearing's some structures have a little influence on stress distribution, such as chamfer, edges, snap ring groove, shield, seal and bearing peak thread, which can be ignored in modeling .



The imported model is then meshed using ANSYS.

Fig: 1 Finite Element Model

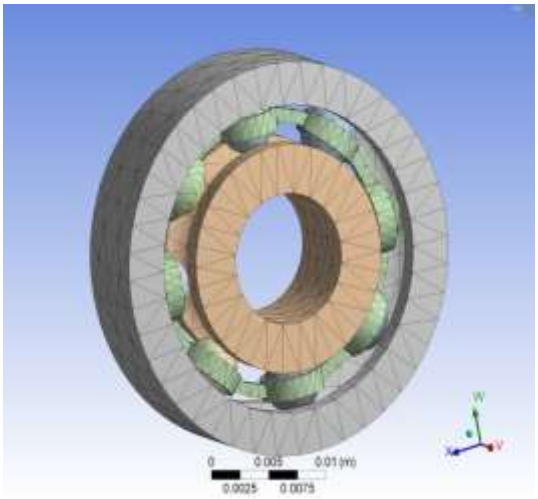


Fig 2: Meshed model

V.DEFINE CONTACT AND CHECK CONTACT ELEMENT

Taking separately the raceway's groove surface of inner and outer ring as target surface, and taking correspondingly half sphere surfaces of balls as target surface, two contact pairs can be built. It is necessary that to make sure the contact is rigid-flexible contact between rolling element and inner or outer ring, to choose CONTA174 as contact element type which has 8 nodes and quadrilateral include middle node, and to choose TARGE170 as target element type which has 3 nodes without middle-node, to set 0.1as normal penalty stiffness (FKN) value of each contact pair (if the value is excessive, it will cause some problems which contact

analysis doesn't convergent), to set 0.01 as initial contact closure(ICONT) value and 0.003 as friction factor value.

After meshing contact element, it is important to check the outer normal direction. To 3D element, the outer normal direction is decided by nodes' number and the right rule. The outer normal direction of contact surface must face to target surface. Otherwise, at the beginning of analysis and calculation, program maybe believes that the contact surface has exceeded penetration, and it will be difficult to find initial solution, so as to program stop executing. If the direction is wrong, it must be changed by reversing the wrong nodes number (Command: ESURF,,REVE or GUI：Main Menu>Preprocessor>Create>Elements>Surf to Surf),or to redefine element normal direction (Command: ENORM or GUI：Main Menu> Preprocessor> Create> Move / Modify > urf Normals).

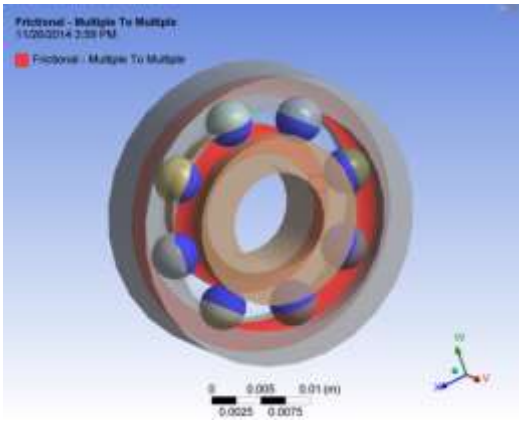


Fig 3: Frictional consideration of bearing model

VI.SET BOUNDARY CONDITION AND APPLY LOADS

Boundary condition restrained the all degrees of freedom(DOF) of bearing outer ring's cylinder surface, fixed the displacement of the axial direction （UZ in Cartesian coordinate system） and circle direction(UY in cylindrical coordinate system) of groove surface and bore surface of outer ring, groove surface and cylinder surface of inner ring, ball and cage's nodes in pitch diameter of ball set, added constraint of the axial direction and radial direction(UX in cylindrical coordinate system),and applied radial loads to the lower part's nodes of bore surface in inner ring.

\

VII.ANALYZE CONTACT STRESS RESULTS

By means of simulation, the contact change status can be got, such as contact stress, strain, penetration and sliding distance, contact friction stress, contact pressure, among the inner and outer ring, balls and cage, as well as bearing's displacement at the same time. Fig. shows the vonMises total strain. The bigger contact displacement mainly concentrated on the lower part of inner ring, and the lowest ball had the largest strain. The biggest total displacement and strain of bearing respectively is DMX=0.09335, SMX=1.6647e10; Specifically, the inner ring had the largest total displacement, and the ball had the biggest total strain, which were consistent with the fact. From Fig, we can also know that the contact area had an approximate ellipse shape in contact area of inner ring and rolling element, which was consistent with the Hertzian contact theory.

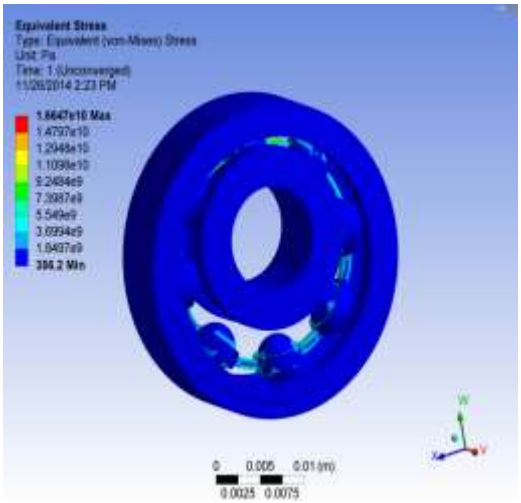


Fig 4: Von misses stress on the bearings

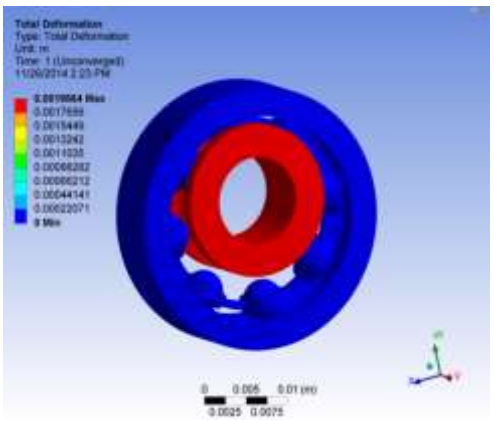


Fig 5: Deformation of the bearing

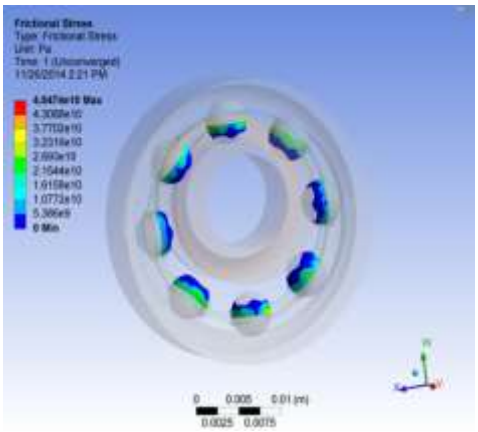


Fig 6: Frictional Stress

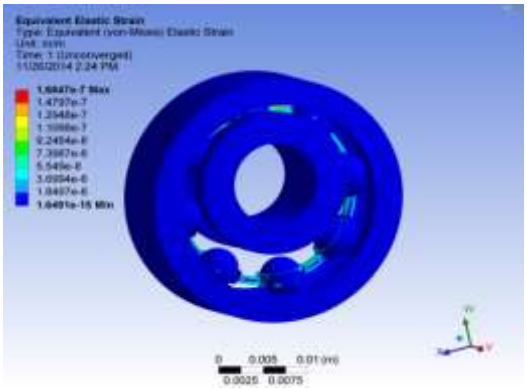


Fig 7: Equivalent elastic strain

Through simulation, the calculation result of the maximal contact stress was 4.847e10MPa, while the Hertzian theory value was 5.021e10MPa. The comparison revealed that there was good consistency between the Hertzian theory solution and finite element solution. Also, ANSYS analysis can acquire bearing parts' other information under contact, such as contact penetration and contact sliding distance, contact friction stress.

VII.CONCLUSION

By using ANSYS to numerically simulate and analyze on stress and penetration during deep groove ball bearing contacts, the ANSYS solution got which had good consistency with the hertzian theory solutions, The contact analysis of this method can easily and intuitively get the stress and strain values as well as their images, which can efficiently understand the parts running information, such as contact penetration, contact stress also.

REFERENCES

1. Xiao Sa Studio. The latest classical courses of ANSYS and ANSYS Workbench. Beijing: Prentice Hall International,Inc.;2004.,6, p. 328.
2. Pavlov V G. Wear calculations for radial ball bearings. Russian Engineering Research, 2008,28(7) , p. 643-650.
3. MA Shi-yao, ZHANG Jin-guo. Contact analysis on rolling bearing by finite element method. Machinery Design & Manufacture,2010,9, p. 8-10.
4. Gangbencunsan, HUANG. Zhiqiang. Design and Calculation of Ball bearing [M] .Beijing: Prentice Hall International, Inc.,2003.
5. Sumit Kumar Dahiya and A.K. Jain analysis of deep groove ball bearing using ansys vsrd International Journal of Mechanical, Civil, Automobile and Production Engineering, Vol. 3 No. 1 January 2013