고속 회전중인 볼베어링 내 공기 유동특성에 관한 연구

CFD Analysis of Air Flowfield Characteristics Inside a Ball Bearing at a High Speed Rotation

*김동주[1], 오일석[1], 홍성욱[1], #김경진[1]
* D. Kim[1], I.-S. Oh[1], S.-W. Hong[1], # K. Kim (kimkj@kumoh.ac.kr)[1]
[1] 금오공과대학교 기계공학부

Key words : Ball bearing, High speed rotation, Numerical simulation, Computational fluid dynamics (CFD)

1. Introduction

A ball bearing is the essential element in many high speed applications such as aero engines and high precision machine tools. When operated at high speed rotation, a large amount of frictional heat occurs between the rolling elements and the raceways and it can cause an excessively high temperature rise in ball bearings. Unless treated properly, frictional heat generation has been one of the primary reasons for the bearing failure and it adversely affect the bearing life and reliability. Therefore, it is important to predict the bearing temperature with a reasonable degree of accuracy in order to maintain the bearing within the recommended range of operating temperature.

Once the frictional heat is generated inside the bearing, there are several heat dissipation modes including the conduction through the raceways and eventually to the housing and the shaft, and the convection by air and lubricant flows. To this date, there have been several attempts in the modeling of heat dissipation, but the most approaches frequently used still rely on the empirical formulas, especially for the estimation of convective heat transfer[(1,2)].

In an effort to understand the convection by the air flow, which is expected to be quite a complex phenomena, this study focuses on the air flow around the fast rotating and orbiting bearing balls. A simplified model of air flows inside the bearing is analyzed by CFD and the air flow characteristics at high speed bearing rotation is presented in this paper.

2. Model and Analysis

2.1 Computational Model

Although the geometry inside a ball bearing is quite complex and it varies greatly depending on the specific bearing type, it basically includes the inner and the outer raceways, the balls, and the cage that secures the balls in place. For the present CFD analysis, the internal geometry is simplified greatly as shown in Fig. 1. The inner and the outer raceways of ring shape are approximated to be flat in axial direction, even though the each raceway actually has the curvatures that guide the ball rolling around the raceways. At this point, the cage is removed in the computational model in this study. Present model seems to be quite oversimplified but note that this study deals with the fundamental flow analysis and further study is required to have the more realistic inner geometry of bearing.

The geometrical data is adopted from the SKF 7218 ball bearing. The inner (d_i) and the outer (d_o) raceway diameters are 102.79 and 147.73 mm, respectively, while their width is 33.71 mm. The ball diameter (D) is 22.23 mm. The number of the balls (Z) is 16 in the actual bearing, but the case with the ball number of 8 and 12 are also investigated.

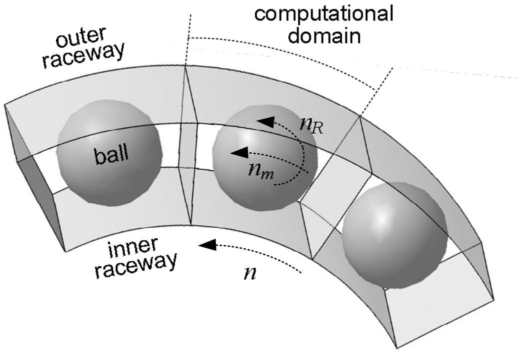

Fig. 1 Simplified geometry and computational domain for the flow simulations inside the ball bearing.

2.2 Method of CFD Analysis

By taking advantage of a periodic layout of geometry, the computational domain includes only one ball and the inner space with the cut angle of 360°/Z, as shown in Fig. 1. In reality, the inner raceway moves with the given rotational speed n (rpm) and the outer raceway does not move, while the balls are in orbital and rotational motions. Assuming the perfect ball rolling on the both raceways, the orbital ball speed around the bearing center and the rotational ball speed around the ball center is expressed as[(1)]

$$n_m = n(1-\gamma)/2 \quad \text{and} \quad n_R = n(1-\gamma^2)/2\gamma\,, \qquad (1)$$

respectively, where γ is D/d_m and d_m is the pitch diameter given as $(d_i + d_o)/2$. In this study, the range of bearing rotational speed between 10,000 to 100,000 rpm is considered.

Simulation can be easily handled when we use a moving reference frame by having the reference point at the bearing center. Two planes in rotational direction are connected as the periodic condition and the side planes are set to be pressure condition of zero gage pressure. Three-dimensional steady incompressible air flow simulations have been carried out using FLUENT, the commercial CFD software. Due to high speed bearing rotation, the flow is assumed to be turbulent and k-ε turbulence model is utilized. The numbers of grid cells used in the simulation are approximately 430,000 and 270,000 for the cases of Z = 8 and 16, respectively.

3. Results and Discussion

Pressure and flow velocity vector fields for the case of Z = 16 and n = 10,000 rpm are shown in Fig. 2. The pressure values near two contact points between the ball and the raceways are found to be very different from each other and the pressure minimum is found near the contact point on the outer raceway. This pressure minimum becomes more significant with increasing rotational speed, although this observation is irrespective of the number of balls. Fig. 2(b) shows the flow velocity vectors near the contact point on the outer raceway, which are viewed from the moving reference frame. Due to the clockwise rotation of the ball, downward motion of air is induced near the contact point. As the bearing rotates at higher speed, the velocity of the induced flow increases.

Fig. 3 shows the distributions of pressure and wall shear stress on the ball surface facing toward the outer raceway and the ball surface area around the contact point has the minimum pressure

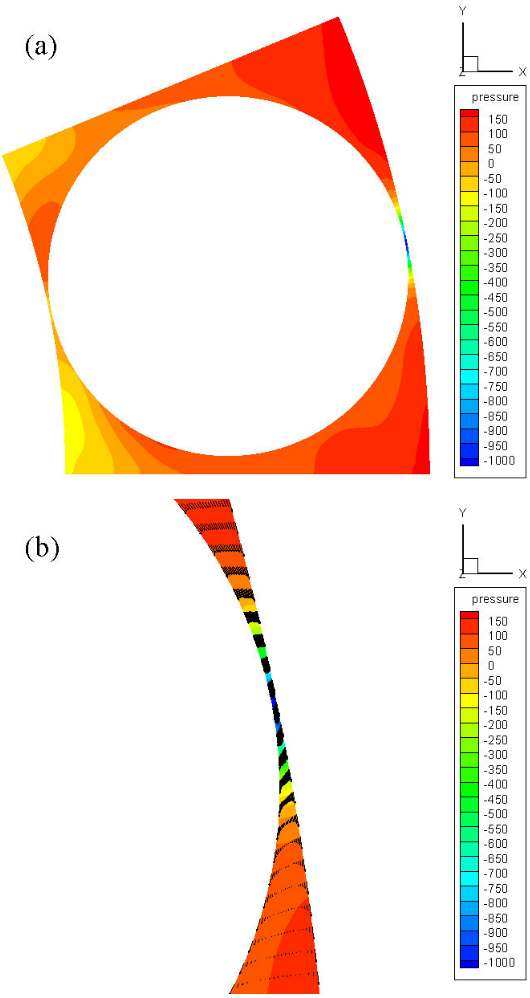

Fig. 2 Flowfields on the z-mid plane: (a) pressure; (b) relative flow velocity vectors from the moving reference frame.

and maximum shear stress. In order to investigate the vortical structure of air flow around the ball, flow velocity vectors on the y = 0 and 1.1 mm plane are drawn in Fig. 4. Clearly, three different vortex pairs (A, B, and C) are formed right behind the ball for Z = 16 and n = 10,000 rpm (Fig. 4a). Among them, the central vortex pair B seems to be very similar to that found in the uniform flow around a sphere(3). Note that the location of the vortex pair B is off-centered toward the outer raceway. The other vortex pairs A and C are believed to take place due to the wall effect of the inner and outer raceways. The vortices A and B quickly disappear and only the vortex C is observed at the y = 0 plane. Similar flow patterns are also found for the case of Z = 8, although not shown in this paper.

4. Conclusions

Air flow inside a ball bearing rotating at high speed was numerically investigated as a first step to understand the flow and convective heat transfer characteristics. The pressure and shear stress distributions on the ball surface showed common features irrespective of the number of balls. The lowest pressure region was found near the contact point between the ball and the outer raceway, and the pressure became significant with increasing rotational speed. We also found the three-dimensional vortex pairs behind the ball some of which disappeared quickly. These findings would provide us with fundamental understanding of the convection heat transfer from the ball to the air flow inside the bearing.

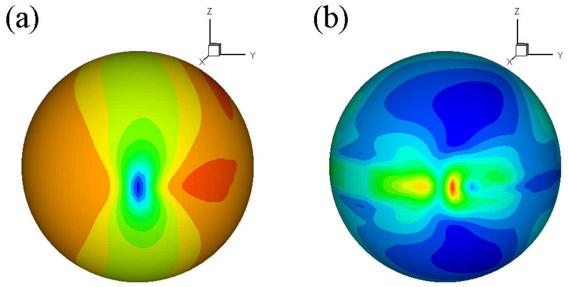

Fig. 3 Distributions of pressure (a) and wall shear stress (b) on the ball surface toward the outer raceway.

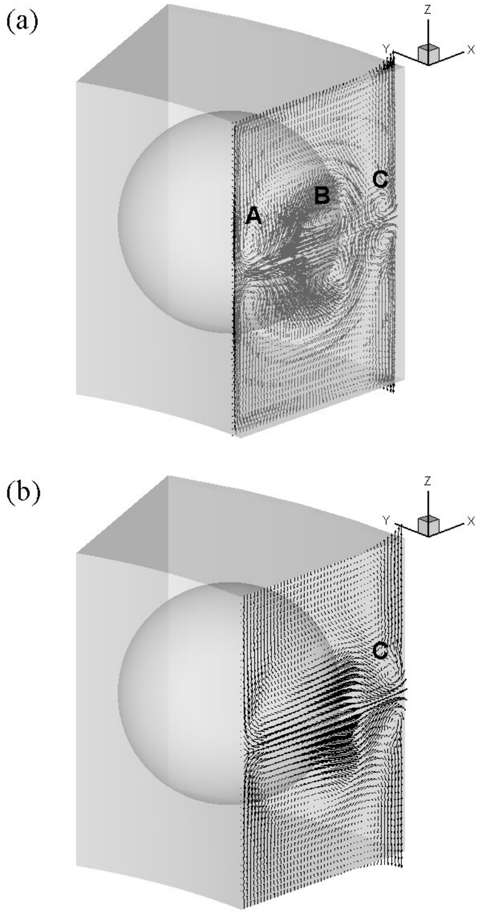

Fig. 4 Flow velocity vectors indicating three-dimensional vortex pairs: (a) y = 1.1mm plane; (b) y = 0 plane.

Acknowledgements

This research was financially supported by Korean Ministry of Knowledge Economy (MKE) under the project "Development of Platform Technology for Machine Accuracy Simulation."

References

1. Harris, T.A., Rolling Bearing Analysis, 4th Edition, John Wiley & Sons, New York, 2001.
2. Bossmanns, B. and Tu, J.F., "A Thermal Model for High Speed Motorized Spindles," International Journal of Machine Tools and Manufacture, 39, 1345-1366, 1999.
3. Kim, D. and Choi, H., "Laminar flow past a sphere rotating in the streamwise direction," Journal of Fluid Mechanics, 461, 365-386, 2002.