Analysis of Finite Length Squeeze Film Damper using CFD

B. Venkata Narayana
Research Scholar, Department of Mechanical Engineering
S.V.U. College of Engineering, Tirupati – 517502 India

Dr. P Venkara Ramaiah
Associate Professor, Department of Mechanical Engineering
S.V.U. College of Engineering, Tirupati – 517502 India

Dr. G. Krishnaiah
Professor, Department of Mechanical Engineering
S.V.U. College of Engineering, Tirupati – 517502 India

Abstract- The traditional method of lubrication analysis of squeeze film dampers involves the complicated numerical calculations to evaluate performance characteristics of fluid film. This paper presents a methodology to develop a 3D model of fluid film for finite squeeze film damper using Computational Fluid Dynamics (CFD). Here, the equations are solved under steady conditions with the commercial software ANSYS14.2 Version Fluent Software. The pressure distribution is obtained for half Sommerfeld boundary conditions operating at different L/D ratios with various eccentricity ratios. The results show reasonable agreement in general with the analytical methods.

Keywords – Finite length SFD, CFD, ANSYS Fluent

I. Introduction

Squeeze Film Damper (SFD) is considered to be one of the most important technological component for all rotating high speed rotating machinery. These are widely used to suppress sub synchronous instabilities and attenuate vibration levels developed in high speed machineries. It comprises of a stationary casing, a non-rotating journal attached to the shaft, generally a roller bearing is show in Figure 1. An SFD comprises a thin lubricant film within the annular clearance gap between the housing and journal. The fluid film produces a pressure field due to the dynamic motions of the journal and effectively reduces the whirling amplitude of motion. Several researchers [1-8] have attempted to improve the performance characteristics of SFD for various configurations by different solution methods.

Traditional method of lubrication analysis on bearing fluid film involves development of complicated numerical calculations and complex programmes. Computational Fluid Dynamics (CFD) is used to simplify the fluid model, to predict the how fluids flow and to solve the equations of fluid flow with the specified conditions on the boundary of that region. The CFD codes make the analysis effective when complex flow geometries are involved or when solutions that are more detailed are needed. The most common method which is used to solve these equations is known as the finite volume method. The computational domain is discretized into finite volumes and then for every volume the governing equations of fluid film are solved.

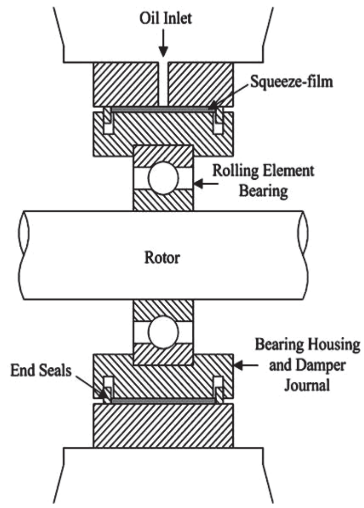


Figure 1. Typical representation of SFD

A significant number of works have been published to simulate the pressure filed and calculate the performance characteristics of journal bearings with the application of CFD. However, the researchers like Tucker et al. [9] and Chen et al. [10] applied the CFD codes to make the analysis effective when complex flow geometries are involved. Ravindra and Sandeep [11] analyzed the pressure distribution of the hydrodynamic plain journal bearing lubricated with oil under steady state conditions using COMSOL Multiphysics software. Shenoy et al. [12] presented a methodology to model and simulate the Overall Elasto-Hydrodynamic Lubrication of a full journal bearing using the sequential application of CFD. Gertzos et al. [13] were derived the performance characteristics of a hydrodynamic journal bearing lubricated with a Bingham fluid by means of three-dimensional computations fluid dynamics (3-D CFD) analysis. The FLUENT software package was used to calculate the hydrodynamic balance of the journal using the dynamic mesh technique.

Based on the above approaches, this paper presents a method to model and simulate the finite length SFDs for half Sommerfeld boundary conditions using the latest simulation software like ANSYS Fluent software.

II. THEORY

This paper presents a methodology to simulate a 3D model of fluid film for finite length squeeze film damper using Computational Fluid Dynamics (CFD). Here, the equations are solved under steady state conditions with the commercial software ANSYS14.2 Version FLUENT Software. The pressure distribution is obtained for half Sommerfeld boundary conditions operating at different length-to-diameter (L/D) ratios by varying eccentricity ratios ($\in$). The results of the CFD model with the Newtonian fluid are compared with the theoretical analyses.

A. Equations to be solved-

In fluid dynamics, the film's pressure and velocity distributions are governed by the coupled continuity equation and the momentum equations. The continuity equation derives from the basic law of mass conservation. The momentum equation is simplified by neglecting the small terms. For all flows, CFD solves mass and momentum conservation equations. Instead of the generalized Reynolds equation, it is better to use the Navier-Stokes equation since Reynolds equation has certain limits to obtain the solution for different problems.

Mass Conservation Equation-

The equation for conservation of mass, or the continuity equation, can be written as

$$\frac{\partial \rho}{\partial t} + \nabla.\left(\rho \vec{v}\right) = 0 \tag{1}$$

This equation is the general form of the mass conservation equation and is valid for incompressible as well as compressible flows.

Momentum Conservation Equation

Conservation of momentum in an inertial (non-accelerating) reference frame is described by

$$\frac{\partial}{\partial t}\left(\partial\vec{v}\right)+\nabla.\left(\rho\vec{v}\vec{v}\right)=-\nabla p+\nabla.(\bar{\bar{\tau}})+\rho\vec{g}+\vec{F} \qquad (2)$$

All investigations that referenced in the present work solve the fundamental Reynolds Equation or in its modified form

$$\nabla.\left(\frac{h^3}{12\mu}\nabla p\right)=\nabla.\left(h\bar{U}\right)-V \qquad (3)$$

Where U is the velocity of journal surface parallel to the film and V is the squeeze film velocity.

B. Assumptions

The following assumptions for the bearing model are used in the present work.

- A rigid aligned bearing with geometry is considered
- Steady state operation is assumed
- The flow is laminar and the lubricant is an incompressible Newtonian fluid with a constant uniform density ‘ρ’ and viscosity ‘μ’ throughout the film
- Gravity and inertia forces are negligible
- The pressure across the film is constant
- There is no slip between the lubricant and bearing surface
- The film is held in between the annular space isothermally

C. Boundary Conditions

Consider the bearing wall as stationary and the journal as rotating wall. The sides of the lubricant volume are assigned with a zero pressure condition, with this the lubricant is free to flow to the sides. When the film pressure drops down to the atmospheric pressure due to heavy external load or high operating rational speed, rupture of the film or cavitation occurs. Taking into account the above situations, the following boundary conditions are used which applies the non-sub-atmospheric pressure constraint:

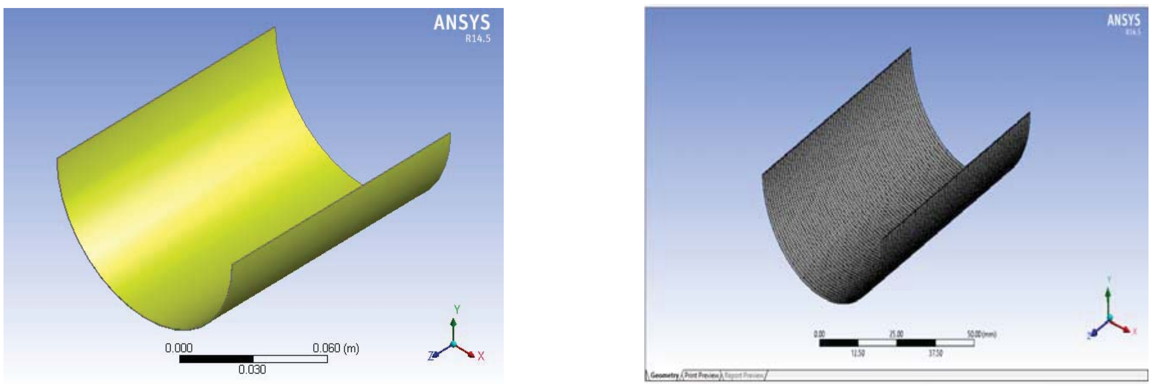
$$p-p_a>0 \text{ at } \pi\leq\theta\leq 2\pi,\ z=0,\ z=L \qquad (4)$$

The Reynolds boundary condition makes the pressure curve to drop in parallel with θ-axis just after 180^o and could be used instead of half Sommerfeld boundary conditions. In some cases, the Reynolds boundary condition gives more accurate results than the half Sommerfeld boundary conditions. Although it is relatively accurate, the Reynolds boundary condition is still an approximation to the transition from full fluid flow to flow with cavitations. It is very difficult to modify the Navier Stokes equations to include the Reynolds boundary conditions. Therefore, the Navier Stokes equations are solved with half Sommerfeld boundary conditions instead of Reynolds equations.

III. CFD Modelling and Analysis

In the present work, a 3-D simulation model is developed using the CFD package ANSYS Fluent. The pre-processor ANSYS mechanical desktop is used for the development of film geometry and grid generation. The mesh is transferred to FLUENT, where the boundary conditions, model properties etc. are set. Many positions of the eccentricity are necessary to complete the diagrams presented here. The clearance size of SFD is very small compared to journal diameter and length which enforces the use of hexahedral cells.

In order to perform simulation, a separate oil film model is developed for various L/D ratios and at different eccentricity ratios. Figure 2 (a) shows the oil film model for L/D ratio of 1.5 and for the eccentricity ratio of 0.3. The meshing of model for the simulations is done by using hexahedral element in ANSYS. The meshing model for L/D = 1.50 and for the eccentricity ratio of 0.3 is shown in Figure 2 (b). Four divisions are used across the journal bearing film, seventy two divisions are used in the circumferential direction and fifty divisions are used in the axial direction. The total number of cells in mesh is 14,400. Since the calculation of pressure distribution is required at many different eccentricities, the solution procedure is repeated for different eccentricity ratios and length to diameter ratios. To simplify the geometry, one side of the clearance is used as a lubricant inlet and the other as an outlet. The boundary conditions are applied as discussed and set reference pressures as zero at the ends of the geometric model, there by all relative pressures measured with reference to this pressure.



(a) (b)

Figure 2. (a) Typical representation oil film 3-D model (b) Typical representation of Hexa-dominant Mesh

D. Geometries and Parameters

The steady state analysis for plain finite SFD is carried out at different eccentricity ratios with different length to diameter ratios. The data (Table 1) is implemented for the analysis of SFD.

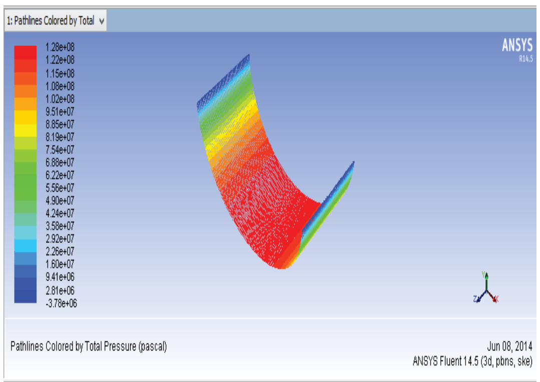
Table 1- Details of Squeeze Film Damper

Parameters	**Dimensions**
Length	25mm, 50mm, 75mm
Diameter of Journal	50mm
L/D ratios	0.5, 1.0, 1.5
Radial Clearance (c)	0.025mm
Eccentricity Ratios (e/c)	0.3, 0.4, 0.5, 0.6
Speed of journal	500 rpm
Dynamic viscosity of oil	0.1934 Pa.s
Density	960 kg-m^{-3}

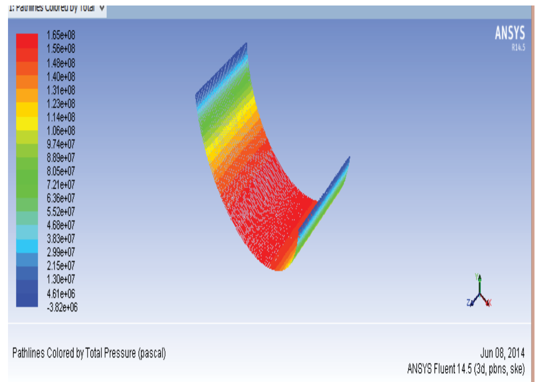
IV. RESULTS AND DISCUSSIONS

The steady state analysis of finite length squeeze film damper is carried out at different eccentricity ratios and L/D ratios. The main characteristics of SFD for CFD analysis are bearing radius (R_b), the radial clearance (c), and the length of the bearing (L). In this analysis, it is assumed that density and viscosity of the lubricant is constant within squeeze film fluid. The flow is considered as laminar and isothermal, and the lubricant as Newtonian fluid.

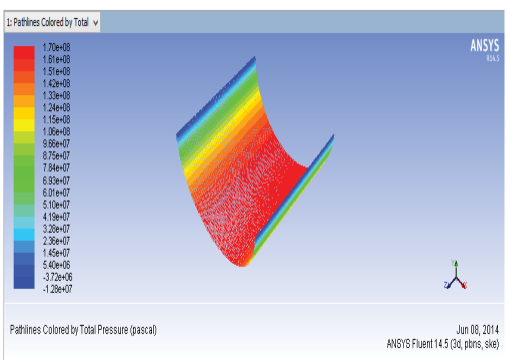
The simulation results from ANSYS Fluent software are shown in Figures 3 at different eccentricity ratios and L/D ratios. Figures shows the pressure distribution on the damper surface when the half Sommerfeld boundary condition is considered. It is clear that the negative pressures are eliminated by considering the half Sommerfeld boundary conditions. The results are compared with the analytical results and there is good agreement between the values. It is observed from the results that maximum film pressures are developed at larger eccentricity ratio and higher L/D ratio. The maximum pressure is developed at the minimum fluid film thickness and at midline in axial direction.



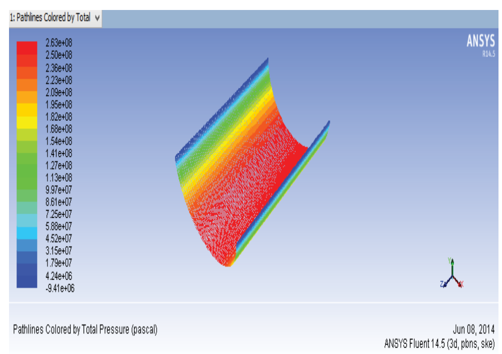
(a)



(b)



(c)



(d)

Figure 3. Simulation results (a) at L/D = 0.5 and $\in$=0.4 (b) at L/D=0.5 and $\in$=0.5 (c) L/D = 1.0 and $\in$=0.3 (d) L/D = 1.5 and $\in$=0.3

IV. CONCLUSIONS

The CFD squeeze film pressures are compared with the pressures obtained from analytical methods The magnitudes of the CFD pressures are higher than those predicted from analytical methods. The comparison is quite decent for both methods, the slight differences in pressures values are due to the analytical assumptions and the software approximations. Hence, solutions obtained by the CFD simulation are to be they are in good agreement.

REFERENCES

[1] Allaire, P. E., Barrett L. E., and E. J. Gunter, 1977, "Variational Method for finite Length Squeeze Film Damper Dynamics with Applications," Wear, 42(1), pp.9-22.

[2] Antunes J., Moreira M and Piteau P, 2003, "Finite Length Bearings and Squeeze Film Dampers: A Two-Dimensional Dynamical Solution," Computational Methods and Experimental Measurements XI.

[3] Della Pietra L., and G. Adiletta, 2002, "The Squeeze Film Damper over Four Decades of Investigations. Part I: Characteristics and Operating Features, Shock Vibration Digest, 34(1), pp. 3-26, Part II: Rotordynamics Analyses with Rigid and Flexible Rotors, Shock Vibration Digest, 34(2), pp. 97-126.

[4] Jung, S. Y., San Andres L. A., and Vance, J.M., 1991, “Measurements of Pressure Distributions and Force Coefficients in a Squeeze Film Damper: Part I – Fully Open Ended Configuration,” STLE Tribology Transaction, 34, No.3, pp. 375-382.

[5] Lin J.R., 1998, “Squeeze Film Characteristics of Finite Journal Bearings: Couple Stress Fluid Model”, Tribology International, 31(4), pp. 201-207.

[6] Naduvinamani N.B. and S. B. Patil, 2009, “Numerical Solution of Finite Modified Reynolds Equation for Couple Stress Squeeze Film Lubrication of Porous Journal Bearings”, Computers and Structures, Elsevier Publications, 87, pp. 1287-1295.

[7] San Andres L.A. and J.M. Vance, 1987, “Effects of Fluid Inertia on Finite Length Squeeze Film Dampers”, ASLE Transactions, 30(3), pp. 384-393.

[8] Zeidan, F. Y., San Andres L., and Vance J., 1996 “Design and Application of Squeeze Film Dampers in Rotating Machinery,” Proceedings of the 25th Turbomachinery Symposium, Turbomachinery Laboratory, Texas A&M University, September, pp. 169-188.

[9] Tucker P., and Keogh P., 1994, “A Generalized Computational Fluid Dynamics Approach for Journal Bearing Performance Prediction”, Proceedings of Institution of Mechanical Engineers: Part J, Journal of Engineering Tribology, 3, pp. 99-108.

[10] Chen P., and Hahn E., 1998, “Use of Computational Fluid Dynamics in Hydrodynamic Lubrication”, Proceedings of Institute of Mechanical Engineers: Part J, Journal of Engineering Tribology, 6, pp. 427-435.

[11] Ravindra M. Mane and Sandeep Soni, 2013, “Analysis of Hydrodynamic Plain Journal Bearing”, Proceedings of the COMSOL Conference, Bangalore, pp. 1-5.

[12] Shenoy B. S., Pai R. S., Rao D. S., and Pai R., 2009, “Elasto-hydrodynamic Analysis of Full 360^{o} journal bearing using CFD and FSI Techniques”, World Journal of Modelling and Simulation, World Academic Press, UK, 5(4), pp.315-320.

[13] Gertzos K. P., Nikolakopoulos P. G., and Papadopoulos, 2008, “CFD Analysis of Journal Bearing Hydrodynamic Lubrication by Bingham Lubricant,” Tribology International, Elsevier Publication, 41, pp. 1190-1204.