A REVIEW OF COMPUTATIONAL FLUID DYNAMICS APPROACH FOR ANALYSIS ON JOURNAL BEARING

G. DINESH REDDY^{1*}, L.S.V. PRASAD² & P.K. SARMA³

¹Assistant Professor, Department of Mechanical Engineering, GMRIT, Andhra Pradesh, India

²Professor, Department of Mechanical Engineering, College of Engineering(A), Andhra University, Andhra Pradesh, India

³Honorary Professor, Department of Mechanical Engineering, College of Engineering(A), Andhra University, Andhra Pradesh, India

ABSTRACT

This paper gives us a brief idea of how the Computational Fluid Dynamics -CFD has helped the researchers in many different forms in different areas. One such area discussed in this paper is about Journal Bearing. Since many years CFD has given quite impressive results and proved to be the most reliable method for a safe analysis. Some of the most common performance predictions of Journal bearings are: Reynolds equation failure, Thermo-hydrodynamic approach, Cavitation modelling, Lubricant grooves, Fluid flow, Heat transfer, Mass transfer, Oil film temperature, 3D Navier Stokes equation, Electro Rheological dynamic mesh, Magneto Rheological dynamic mesh, Fluid structure interaction, Bearing surface deformation, Altitude angle and Load carrying capacity. The results are of high accuracy, which save time and cost. Every industry like rolling, furnaces, boilers, automotive and heavy engineering etc., where Journal bearings are used and quality is the supreme importance can easily choose CFD for various kinds of analysis with cost effectiveness.

KEYWORDS: CFD, Reynolds Equation, Electro Rheological Dynamic Mesh, Magneto Rheological Dynamic Mesh and Thermo-Hydrodynamic Approach

Received: Jun 10, 2020; **Accepted:** Jun 30, 2020; **Published:** Jul 16, 2020; **Paper Id.:** IJMPERDJUN2020356

INTRODUCTION

Journal Bearing is used to support an applied load by reducing the friction relatively in high speed machinery like turbines, electric motors, ship shafting propulsion systems etc. It consists of a fixed sleeve where on the shaft also called as journal rotates. As the metal to metal contact may wear the journal and bearing, so, micro clearance space for the lubricants are provided as they can bear the load of the journal. In the initial stages, the wedge effect will cause the free movement of the journal and as the speed increases the oil film thickness will gain enough support to take the load uniformly. Hence, the factor minimum oil arises for efficient power transmission. Perfect lubrication occurs when minimum oil film thickness is higher than the quantity dependent on the nature of the indiscretions of the communicating surfaces.

Computational Fluid Dynamics (CFD) is an evolution from vast areas of mathematics, physics, mechanics and computers. It is a technique of full three-dimensional approach run on computer software mostly like ANSYS FLUENT etc., where in all kinds of complex equations are solved under static and dynamic conditions along with boundary conditions. Researchers, now a days are very much interested in getting instant results with the help of models and software available with them. This process has also saved costs and time for analysis of complex designs. The usage of CFD is easy to understand with make it much simpler to apply loads, equations, boundary

conditions, material properties etc., Most of results are comparatively similar to that of physical conventional methods and experiments. In CFD, conventional methods and accelerated methods are used to solve the Navier-Stokes equations either in Eulerian or in Lagrangian approach. Also, some methods solve the Boltzmann equations instead of Navier-Stokes equations.

Applications

Cavitation modelling was used to predict sub-ambient pressures by introducing lubricant inlet grooves with conservation of mass and chances of back flow that is based on averaged lubricant vapour properties (P G Tucker and P S Keogh, 1995). It was found by $k-\omega$ turbulence model that internal flow analysis of a dump diffuser used in marine gas turbines and modern aircraft engines by ANSYS FLUENT has no effect on diffuser when diffuser angle is raised (Klein *et al.*, 1995). Through CFD technique, the pressure effect developed on the walls of pipe under different pipe networks namely T-junctions, bends, Y-junction, elbows, expansions, contractions, valves and many other components were studied (Patel *et al.*, 2005). CFD approach, using FLUENT was used to model flow of fluid in a journal bearing with equally spaced three axial grooves which was provided with water from one end was used to compute the stiffness and damping coefficients (Ranjan *et al.*, 2006). The FLUENT software package was used to compute the hydrodynamic balance of the journal using the so-called “dynamic mesh” technique where electro-rheological fluids (ERFs) or A magnetorheological fluids (MRFs) that behave like Bingham fluids, and is used to describe the lubricant and the magneto-rheological (MR) and electro-rheological (ER) fluids behavior of the non-Newtonian fluid flow (Gertz *et al.*, 2008). A software of residual-based variational multiscale modeling procedure for calculation of laminar and turbulent flow of fluid through concentric annular pipe flows was achieved by CFD (Motlagher *et al.*, 2012).

To solve the Reynold's equation generalized form, equation of energy and the displacement field, respectively, using two mathematical procedures like Computational Fluid Dynamic (CFD) and Fluid Structure Interaction (FSI). This technique is used to determine the temperature, pressure and velocity fields in the lubricant film and the FSI simulation is used to find the displacement field and stress intensity. The effect of the operational conditions on the fields pressure, displacement, temperature and stress intensity are also analyzed (Panday *et al.*, 2012). CFD simulation software ANSYS was helpful in proposing an idea for establishment of relationship among angular velocity, Reynolds Number and drag coefficient for fluid flow at varying conditions (Prabhakar, 2012). Investigations on few significant works done on modeling of laminar flow and numerical analysis in pipes with the objective to review the various aspects of applications of Computational Fluid Dynamics (CFD) technique in the area of fluid flow through heat exchangers, bifurcated blood vessels and branched-pipes (Ismail and Adewoye, 2012). A numerical study with the aim of realizing the effects of the temperature fluctuations and frictional force on the bearing temperature and viscosity profiles was presented in thermodynamic study of the 3-dimensional plain journal bearing using CFD. As recorded, the pressure distribution on journal surface, not only circumferentially but also axially, with and without considering temperature effect analyzed that increasing the frictional force reduces the viscosity and increases the temperature, as well as the maximum pressure of the lubricant present inside the plain journal bearing (Shahu *et al.*, 2012). Studies on the flow fields of Y-junction and T-junction using shear stress transport (SST) model through ANSYS/CFX software, found that the discrepancy of velocity peak in T-junction with different frequencies and phase-differences (In and Li, 2013).

Computational fluid dynamics (CFD) was done to explore the transition limitations of different flow patterns for discretely viscous oil-water two-phase flow through a straight pipeline (Mandal, 2013). CFD simulation analyzed that

branching junctions especially plays important dynamic and kinematic roles in blood transportation vasculature and industrial flow systems (Sochi, 2013). Utilizing the ANSYS CFX software to investigate the effect of turn/bend for a Y-shape pipe and found that resistance coefficient varies with the change in flow (Hirani and Kiran, 2013). Studies on the effect of pipe diameter, bend angle, Reynold's number and pipe length on coefficient of resistance through CFD analysis and found that coefficient of resistance varies with change in flow parameters (Sehgal *et al.*, 2013). CFD software namely ANSYS was used for point by point study of flow through pipe and also for determination of losses in head due to change in geometry of pipe. Thermo-hydrodynamic analysis was carried out using Computational Fluid Dynamics which solves three-dimensional Navier-stokes equation to analyze the temperature and pressure of the lubricant along the contour of the Journal bearing. (Amit Chauhan *et al.*, 2014). CFD analysis under various Reynolds numbers of laminar single-phase water flow in a helical pipe with hollow cross section was done and found that with rising Reynolds number and creation of centrifugal forces, a high pressure and velocity region occurs between two tubes, at the outer side of the helical pipe with hollow cross section. Besides, they observed that friction factor declines as the turbulence increases (Hosseini *et al.*, 2014). During analysis using CFD, the effect of temperature and pressure variations of journal bearing under steady state condition was found out that maximum pressure tended to occur more often nearer to the region of minimum film thickness (Tiwari, *et al.*, 2014). Utilizing CFD software namely FLUENT, analysis for the flow parameters of a pair of discretely viscous oil and water through horizontal pipeline was done and found that in the annular flow, entire pressure of the combination declines with rise in oil velocity as the pipe cross section is completely wetted with water (Mandal *et al.*, 2014). Equivalent oil film temperature of Hydrodynamic journal bearing was analyzed by using ANSYS software as it is one of the prominent parameter which effects oil temperature, oil viscosity and load carrying capacity of the bearing (Suwarna Torgal and Ruchika Saini, 2015). Models of journal bearing for different eccentricity ratios are simulated for the pressure, deformation and stress of plain journal bearing by CFD interaction approach in fluid structure and figured out that CFD-FSI method delivers a valuable platform to study the mutual result of elastic behavior of the bearing and hydrodynamics (S. Chaitanya Kumar and R. Ganapathi, 2015). Investigations on pressure and temperature effects on viscosity were carried out by varying the L/D ratio, rotational speed and eccentricity ratio on pressure distribution on bearing. Indications of increasing temperature raises the pressure but would lead to considerable decreases in the altitude has been done using computational fluid dynamic "CFD" tool (Panthiet *et al.*, 2015). Computational Fluid Dynamics was defined as advanced engineering method to design, model and solve complex industrial solutions that save time and energy to build sustainable and efficient energy systems. The CFD simulation is coupled with cooling impinging jets used to regulate temperature and heat transfer behaviour of a hot rolling process where cooling water and thermal energy are used more (Lokman Hosaina, b and Rebei Bel Fdhila, 2015). Using the CFD software COMSOL, solution of Navier-Stokes incompressible flow for obtaining the average velocity of fluid in a pipe was measured and compared experimentally (Acharya, 2016). Investigations were done on incompressible, steady fluid flow through a T-junction when fluid passes through it with the help of FLUENT (Nimadge and Chopade, 2017). Computational Fluid Dynamics (CFD) helped in study of MR fluids flow in pipes of circular cross section under the influence of uniform magnetic field (Gedik, 2017). Manifold arrangements were developed with bifurcations/trifurcations which lead to head losses in a fluid system and generated quantitative data of sharp-edged bifurcation loss coefficients obtained through Computational Fluid Dynamics (CFD) (Spooner *et al.*, 2017). A new Computational Fluid Dynamic (CFD) model has been built for the sake of studying oil film pressure profile influenced by many parameters like oil properties, design modification, material properties, and many other factors of coupling future (Nour Marey *et al.*, 2018).

CONCLUSIONS

Hence, this review helps us to find out the diverse solutions with the help of CFD. Many researchers have been using this CFD to find out the best results before experimenting them, so that modifications can be made to the setup prior conducting the experiment. The result obtained using CFD are mostly in line with the experimental results which encourages the researcher to go for further developments in the research.

REFERENCES

1. Tucker, P.G., & Keogh, P.S. (1995). A generalized computational fluid dynamics approach for journal bearing performance prediction. *Proceedings of Institution of Mechanical Engineers*, 209, 99-108.
2. Ranjan, V., Pai, R., Raghuvir, Hargreaves, Douglas. (2006). Stiffness and Damping Coefficients of 3-axial Groove Water Lubricated Bearing Using Perturbation Technique. In Fillon, M (Ed.) *Proceedings of Futuroscope: 5th Workshop Bearing Behaviour under Unusual Operating Conditions*. Universite de Poitiers, France, Futuroscope Chasseneuil, 1-6.
3. Panday, K.M., Choudhury, P.L. & Kumar, N.P. (2012). Numerical Unsteady Analysis of Thin Film Lubricated Journal Bearing, *IACSIT International Journal of Engineering and Technology*, 4, 185-191.
4. Mukesh Sahu, Ashish Kumar Giri & Ashish Das., (2012). Thermohydrodynamic Analysis of a Journal Bearing Using CFD as a Tool, *International Journal of Scientific & Research Publications*, 2, 1-7.
5. Priyanka Tiwari, & Virendra Kumar., (2014). Analysis of Hydrodynamic Journal Bearing Using CFD And FSI Technique, *International Journal of Engineering Research & Technology (IJERT)*, 3.
6. Arjun Panthi, Jai Balwanshi, Ajay Chandravanshi, & Gourav Gupta., (2015). Design and Analysis of Hydrodynamic Journal Bearing Using Raimondi and Boyd Chart, *International Journal of Core Engineering & Management (IJCEM)*, 2, 109-120.
7. Klein, A. (1995). Characteristics of combustor diffusers. *Progress in Aerospace Sciences*, 31, 171-271.
8. Patel, T., Singh, S.N., Seshadri, V. (2005). Characteristics of Y-shaped rectangular diffusing duct at different inflow conditions. *Journal of aircraft*, 42, 113-20.
9. Ranjan, V., Pai, R., Hargreaves, D.J. (2006). Stiffness and damping coefficients of 3-axial grooved water lubricated bearing using perturbation technique, In: *Proceedings of fifth EDF & LMS Poitiers workshop: bearing behavior under unusual operating conditions*. Futuroscope, 1-6.
10. Gertzog, K.P., Nikolakopoulos, P.G., Papadopoulos, C.A. (2008). CFD analysis of journal bearing hydrodynamic lubrication by Bingham lubricant, *Tribology International*, 41, 1190-1204.
11. Ghosh, Sugato, and ARTHUR MILLER. "Effect using turbulent flow of couple stress fluid in seven and thirteen axial grooves journal bearing." *IJMPERD* ISSN (2013): 209-222.
12. Panday, K.M., Choudhury, P.L., & Kumar, N.P. (2012). Numerical Unsteady Analysis of Thin Film Lubricated Journal Bearing, *IACSIT International Journal of Engineering and Technology*, 4, 185-191.
13. Sahu, M. A., Giri, K., Das, A. (2012). Thermohydrodynamic Analysis of a Journal Bearing Using CFD as a Tool, *International Journal of Scientific and Research Publications*, 2, 2250-3153.
14. Dubey, Harsh K., Satish P. Lokhande, And Rajat G. Kawalkar. "An Analytical Study of Contact Stress and Contact Zone Analysis of Cylindrical (Roller) & Spherical (Ball) Bearings in Epicycloid Condition." *International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)* 8.6, Dec 2018, 865-872
15. Li, X., & Wang, S. (2013). Flow field and pressure loss analysis of junction and its structure optimization of aircraft hydraulic

- pipe system, *Chinese Journal of Aeronautics*, 26, 1080-1092.
16. Motlagh, Y.G., Ahn, H.T., Hughes, T.J., Calo, V.M. (2013). Simulation of laminar and turbulent concentric pipe flows with the isogeometricvariational multiscale method, *Computers & Fluids*, 71, 146-155.
 17. Akhtar, Shahzad, And Tarsem Singh. "Fault Diagnosis Of Self Align Roller Bearing At Various Speed Using Frequency Domain." *International Journal of Automobile Engineering Research and Development (IIAuERD)* 9. 2, Dec 2019, 29–34
 18. Ismail, O.S., &Adewoye, G.T. (2012). Analyses and modeling of laminar flow in pipes using numerical approach. *Journal of Software Engineering and Applications*, 5, 653-658
 19. Prabhakar, R. (2013). *CFD analysis of newtonian fluid flow phenomena over a rotating cylinder* (Doctoral dissertation).
 20. Hirani, A.A., &Kiran, C.U. (2013). CFD simulation and analysis of fluid flow parameters within a Y-Shaped Branched Pipe, *IOSR Journal of Mechanical and Civil Engineering*, 10, 31-34.
 21. Joy, N. I. V. I. N. "Design and Analysis of an Integrated Exhaust Manifold with Turbocharger for Considerable Reduction of Over Heating of Bearing System." *International Journal of Mechanical and Production Engineering Research and Development* 8.2 (2018): 601-608.
 22. Singh, B., Singh, H., Sebgal, S.S. (2013). CFD analysis of fluid flow parameters within a Y-shaped branched pipe, *International Journal of Latest Trends in Engineering and Technology*, 2, 313-317.
 23. Amit Chauhan, Amit Singla, NarenderPanwar&Prashant Jindal. (2014). CFD Based Thermo-Hydrodynamic Analysis Of Circular Journal Bearing, *International Journal Of Advanced Mechanical Engineering*, 4, 475-482.
 24. Priyanka Tiwari&Veerendra Kumar. (2014). Analysis Of Hydrodynamic Journal Bearing Using CFD And FSI Technique, *International Journal Of Engineering Research*, 3, 1210-1215.
 25. SuwarnaTorgal&Ruchika Saini. (2015). Thermo-Hydrodynamic Analysis of Journal Bearing To Find Out Equivalent Temperature, *International Journal of Science Technology & Engineering*, 1, 201-207.
 26. Arjun Panthi, Jai Balwanshi, Ajay Chandravanshi, Gourav Gupta. (2015). Design And Analysis Of Hydrodynamic Journal Bearing Using Raimondi And Boyd Chart, *International Journal Of Core Engineering & Management (IJCEM)*, 2, 109-120.
 27. MdLokmanHosaina, B., &Rebei Bel Fdhilaa, B. (2015). Literature Review of accelerated CFD Simulation Methods towards Online Application. *Energy Procedia*, 75, 3307-3314.
 28. Nimadge, M.G., &Chopade, M.S. (2017). CFD analysis of flow through T-junction of pipe, *International Research Journal of Engineering and Technology*, 4, 906-911.
 29. Gedik, E. (2017). Experimental and Numerical Investigation on Laminar Pipe Flow of Magneto-Rheological Fluids under Applied External Magnetic Field. *Journal of Applied Fluid Mechanics*, 10, 801-811.
 30. Sukhapure, K., Burns, A., Mahmud,T., Spooner, J. (Computational fluid dynamics modelling and validation of head losses in pipe bifurcations, *13th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics*, 489-494
 31. NourMarey, Amman Ali&El-Sayed Hegazy. (2018). Computational Investigation of Oil Film Pressure Profile in Journal Bearings, *Port Said Engineering Research Journal*, 22, 40-45.
 32. EbrahimAhmadloo, NajmehSobhanifar, Fatemeh Sadat Hosseini. (2014). Computational Fluid Dynamics Study on Water Flow in a Hollow Helical Pipe, *Open Journal of Fluid Dynamics*, 4, 133-139.