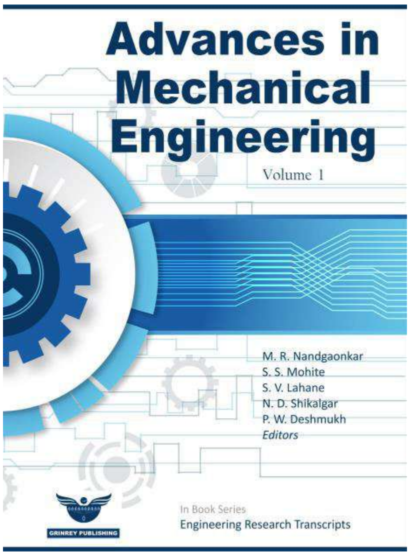


Comparison of Different Algorithms for CFD Analysis of Two Dimensional Journal Bearing Model

Smeet Ashar*, Sounak Chatterjee, Vaishnavi Gaikwad, Abhijeet Chavan, Sandeep Thorat

Department of Mechanical Engineering, MIT ADTU School of Engineering, Pune, India

*Email: smeetashar@gmail.com

Abstract

Journal bearings have wider range of applications in industries owing to their theoretically infinite life span and high load carrying capacity at high speeds. Pressure generation phenomenon in case of hydrodynamic journal bearing is an attraction for researchers. With help of advanced computational facilities and dedicated software tools hydrodynamic action of journal bearing can be predicted more accurately. Deciding the suitability of a particular solution scheme for a given problem is one of the critical tasks, which can save computational time. Present work aims for investigation of two dimensional hydrodynamic analysis of a journal bearing, in order to get the most suitable computational scheme to get the fastest convergence rate for this type of model. A laminar model under steady state conditions has been taken under consideration for present analysis. Numerical investigation is carried out by using software tool ANSYS Fluent. Two dimensional model of lubricating fluid film region for a fixed value of eccentricity is generated and meshed. The pressure profiles for the hydrodynamic journal bearing are computed for a fixed value of viscosity and speed by using various schemes available in the software. Comparison of results for each scheme is made and the most effective scheme is selected by considering the minimum number of iterations & time taken to converge the solution. SIMPLE algorithm is found to be fastest and most suited for such cases with SIMPLEC and PISO also working well.

Keywords
CFD
Hydrodynamic Journal Bearing
SIMPLE algorithm

Cite
Smeet Ashar, Sounak Chatterjee, Vaishnavi Gaikwad, Abhijeet Chavan, Sandeep Thorat, Comparison of Different Algorithms for CFD Analysis of Two Dimensional Journal Bearing Model, Engineering Research Transcripts, 2 (2023) 1-8

DOI: https://doi.org/10.59196/ERT.2

1. Introduction

Journal bearings are one of the most used bearings due to their low maintenance, infinite life and higher load carrying capacities. The growing tendency toward higher-speed, higher-performance, but smaller-sized machinery has pushed bearing working conditions toward superior versions, as well as numerous forms of research and study being done on them.

Relevant work has been done on the computational fluid dynamic analysis of journal bearings by various researchers. D. S. Jang et. al. [1] compared the various algorithms of PISO, SIMPLE, SIMPLEC for a variety of problems in CFD, with the aim of determining the superiority of an algorithm for a given type of problem. Their work also comments on the inconsistent behaviour of the SIMPLE and SIMPLEC algorithms when used along with time steps.

There is adequate literature that shows the various analyses that have been performed on journal bearings using CFD tool. Sahu et. al. [2] and Chauhan et. al. [3] carried out thermo-hydrodynamic analysis of a journal bearing using CFD. They used CFD to predict the performance and three-dimensional study has been done to get the pressure and temperature distribution circumferentially and axially. B. Manhsoor et. al. [4] did CFD analysis using three different turbulent models (Standard k-ε model, Reynolds Stress Model

(RSM) and Realizable k-ε model). They concluded, standard k-ε model is adequate for convergent solution and it is faster than Reynolds Stress Model (RSM) and Realizable k-ε model. Few design parameters were considered, and their analysis is done for different L/D ratio. Similarly, U. Manojkumar et. al. [5] examined the pressure field and deformation of bearing coupling surfaces of an elastic hydrodynamic model using fluid dynamics and fluid surface interaction (FSI) methods.

Other than the already discussed literature we can also find studies performed to find the effect of surface texturing as discussed by Chen Y. [6] and S. Cupillard [7], surface texturing of different geometries and shapes have been done on the inner bearing surface and their respective effects on the pressure distribution and load carrying capacities on bearings have been shown by using CFD.

Some work considered different lubricants other than lubricating oils. Gertzoz et al. [8] use Bingham fluids to check non-newtonian lubrication in journal bearing, where the analysis was done on ANSYS Fluent. Various performance characteristics were derived, and Raimondi-Boyd charts were used to present the results. G. Gengyuan et. al. [9] used CFD analysis on a water lubricated bearing using FLUENT for various L/D ratios to propose the design of a transition-arc structure for a bearing bush, which increased the load carrying capacity of hydrodynamic journal bearings owing to the scope of use of water as a lubricant. M. Deligant et. al. [10] worked on a three-dimensional CFD model for a turbocharger journal bearing to find its frictional losses.

Throughout the literature survey it was found that majority of the work is performed on a three-dimensional model of a journal bearing, two-dimensional model being neglected in all the cases. This work aims to present a hydrodynamic analysis on a two-dimensional model of a journal bearing and to compare the best suited scheme for the same.

2. Governing Equation and P-V Coupling Algorithms

The lubricant flow across the clearance space of journal bearing is determined by fluid flow characteristics. The basic laws associated with the flows of fluid are Conservation of momentum, Conservation of energy and Conservation of mass.

In order to get the solution for the Pressure, two-dimensional Reynolds equation is used but ANSYS Fluent solves Navier- stokes equation of continuity and momentum to get the solution for a given CFD problem; here being the pressure distribution for the given fluid film profile. In Fluent these equations are solved for momentum and mass assuming steady state, incompressible & laminar flow. Mass conservation Equation provided by S. Cupillard [8] is represented by equation (1):

$$\frac{\delta\rho}{\delta t} + \nabla\,\rho\vartheta = 0 \tag{1}$$

Here, ρ and ϑ are density and velocity vector. Momentum Conservation Equation is [8]:

$$\frac{\delta\rho\vartheta}{\delta t} + \nabla\rho\vartheta\vartheta = -\nabla P + \nabla\tau + \rho g + F \tag{2}$$

Here, P is the static pressure, τ is stress tensor, ρg is gravitational force & F is external body force. Stress tensor [8] is written as:

$$\tau = \mu\left(\nabla\vartheta + \nabla\vartheta^{T}\right) - \frac{2}{3}\nabla\vartheta I \tag{3}$$

Here, μ is fluid viscosity, I is unit tensor, and another term on right hand side is effect of volume dilation.

2.1 Pressure-Velocity coupling algorithms

The pressure field must be computed to solve the equations, because pressure gradients arise in momentum equations. Pressure-velocity coupling algorithms are used to obtain pressure equations from momentum and continuity equation.

ANSYS Fluent has the following two algorithms for steady flow problems: SIMPLE and SIMPLEC. These schemes are referred to as the pressure-based algorithms. SIMPLE, SIMPLER, SIMPLEC are typically used for steady-state calculations. PISO is recommended for transient calculations.

2.1.1 SIMPLE (Semi-Implicit Method for Pressure-Linked Equations)

This is most commonly used algorithm. Here, in this algorithm the algebraic equation for pressure correction p’ is derived.

$$a_p p' = \sum_{nb} a_{nb} p' + b \tag{4}$$

Term b shows the net flow rate into the cell. In this algorithm pressure correction is applied to the pressure field at each iteration. The challenge is to come up with a good equation for pressure correction as a function of mass imbalance. Most commercial finite volume programmes use SIMPLE as their default algorithm.

2.1.2 SIMPLER (SIMPLE Revised)

It is an improvised form of simple. Instead of the pressure correction equation used in SIMPLE, the discretized continuity equation is employed in this approach to construct a discretized pressure equation. The intermediate pressure field is obtained in this case without the use of a correction.

$$a_p p = \sum_{nb} a_{nb} p + b \tag{5}$$

Here, more simplified equation was derived as some factors from SIMPLE equation causing the slow convergence of the pressure field.

2.1.3 SIMPLEC (SIMPLE Consistent)

SIMPLEC follows the same procedures as SIMPLE. The main difference is that the momentum equations are modified in such a way that this algorithm neglects terms that are less relevant than those eliminated by SIMPLE. Here, the algebraic pressure correction equation is same as that of SIMPLE.

$$a_p p' = \sum_{nb} a_{nb} p' + b \tag{6}$$

SIMPLEC improves convergence only if the pressure-velocity coupling is a constraint. Often, SIMPLE and SIMPLEC gives identical convergence rates.

2.1.4 PISO (Pressure Implicit with Splitting of Operators)

This algorithm was created to compute unsteady compressible flows in a non-iterative approach. It has also been applied successfully to iterative solutions of steady-state applications. This algorithm is a time-marching technique with a prediction step and one or more corrector steps at each time step. It is considered as an extension of SIMPLE. The algebraic pressure correction equation for PISO is given by:

$$a_p p'' = \sum_{nb} a_{nb} p'' + b'' \tag{7}$$

The algorithm for PISO is same as the SIMPLE. The equations involved are same as that of SIMPLE but as the correction steps are assumed the different notations are used. Once the p' pressure is obtained one more correction step is solved to get the second pressure correction (p'') equation. This algorithm is found to be efficient and fast but more computational efforts are required. The PISO algorithm transfers the repeated calculations done by SIMPLE and SIMPLEC.

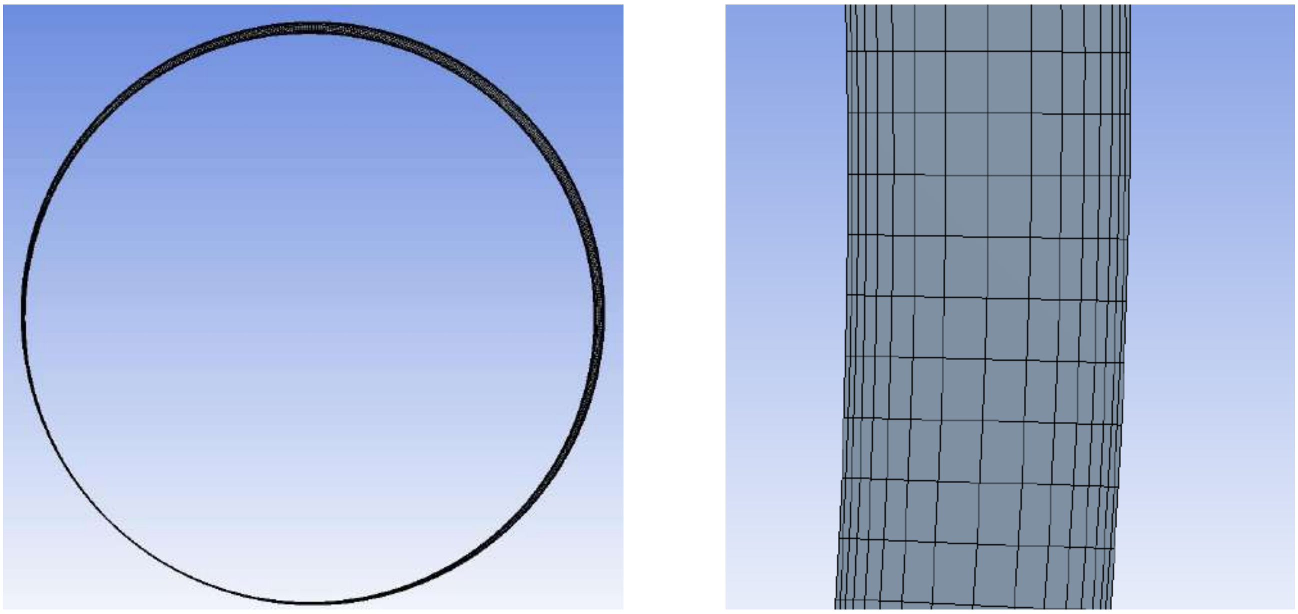
Generally, the SIMPLE is used as the default algorithm in most of the finite volume codes. The SIMPLER, SIMPLEC & PISO are the improved versions of the SIMPLE. All these methods can accelerate convergence since they allow for higher under-relaxation factors than SIMPLE. These algorithms will eventually converge into the same solutions. The variations are in terms of speed and stability. The fastest algorithm is determined by the flow, and no one algorithm is always quicker than the others. The equations and derivations for the algorithm will not be examined in detail here, but they can be easily found in the literature [12].

3. Two Dimensional Modelling of journal bearing

3.1 Model Description

A 2D simulation model is developed using the CFD FLUENT software. ANSYS Mechanical has been used for mesh generation. The 2D model has been meshed using quadrilateral mesh having 13600 cells. Tri mesh was not used due to its lesser accuracy during computation and a greater number of elements. Face meshing was done with 16 divisions with suitable bias settings. Figure 1(a) shows the complete face of the journal bearing and Figure 1(b) shows a detailed view of the mesh formed. Good mesh quality with an average skewness of 0.069 was achieved.

The bearing used has a journal diameter of 40mm and a clearance of 0.5mm with an eccentricity ratio of 0.8. The lubricant used has a fluid viscosity of 0.057 Pa-S and a fluid density of 872 Kg/m3. The outer wall of the geometry was set as the stationary wall which is the bearing shell and the inner wall was set as the moving wall which is the journal. The model has been solved to find the pressure profile along the bearing surface using various pressure velocity coupling algorithms. The pressure profile is firstly validated using literature and then the same model is simulated using different algorithms which are further compared based on time and the number of iterations taken for convergence.



(a) Face of the journal bearing (b) Detailed view of the mesh

Fig 1: Meshed journal bearing model

3.2 Mesh Independency

Mesh independency is examined to determine the dependability of the mesh type or number of cells on the outcomes. It is obtained when the outcomes are unaffected by changing the number of cells. Table 1 shows the relevant data. The percentage difference shows pressure difference between two adjacent values.

Table 1: Mesh Independency study

Number of face divisions	Nodes	Elements	Max. Pressure (MPa)	%Difference
6	5600	4800	0.164	-
8	7200	6400	0.174	5.7
10	8800	8000	0.179	2.6
12	10400	9600	0.1819	1.4
14	12000	11200	0.1830	0.82
16	13600	12800	0.1840	0.52

4. Results and Validation

4.1 Validation

Results of this work have been extracted for a journal bearing having eccentricity ratio (ε) of 0.8. Taking into account the fact that this work is done on a two-dimensional geometry, no relevant literature is available and thus there is no scope for direct validation. Being a two-dimensional model, this bearing has an l/d ratio of 0, but the l/d ratios available in work by Raimondi and Boyd [11] range from 0.25 to ∞. Thus, for validation purposes, the pressure trend of this model is observed. The numerical data collected by this work matches the general pressure trend of a journal bearing as shown in Figure 2 which shows the plot of max pressure vs. bearing angle. Note that a user defined function is used in simulation to consider cavitation effects in the bearing.

Figure 3 shows the force vectors due to force exerted by the oil film in the region of minimum film thickness. Increasing the eccentricity ratio increases the force exerted by the lubricant i.e. the force exerted on the shaft by the oil film is directly proportional to the eccentricity ratio of the bearing.

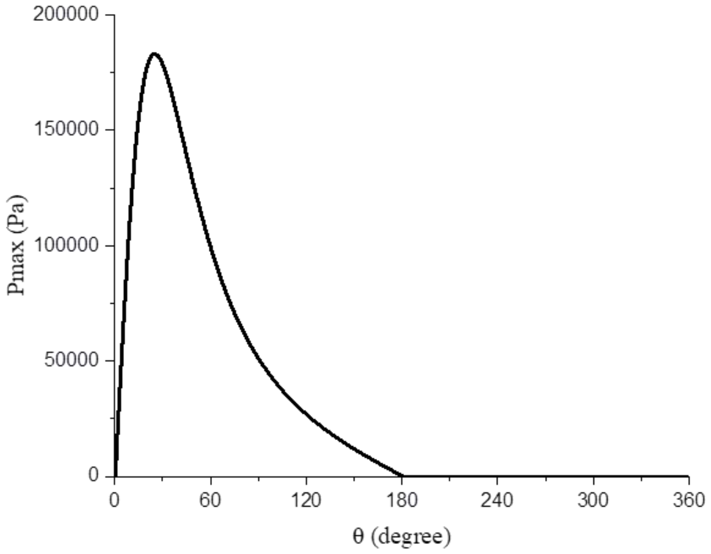


Fig. 2: P_{max} vs. bearing angle

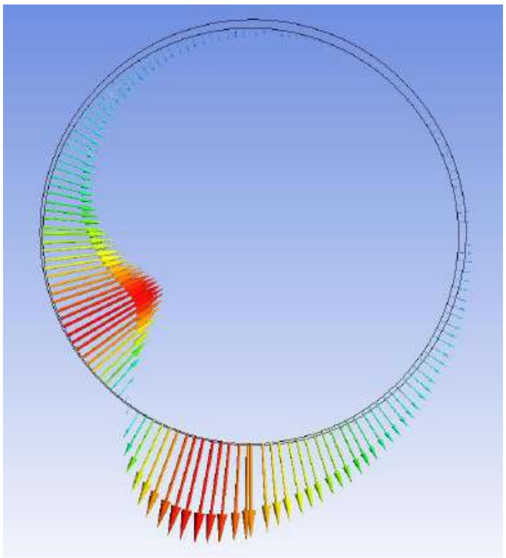


Fig 3: Hydrodynamic force distribution

4.2 Comparison of algorithms

Comparison of various CFD algorithms used in ANSYS Fluent is carried out using basic parameters such as number of iterations taken for the results to converge, time taken, and the result obtained. Convergence of a solution is dependent on the mesh quality, complexity of the problem, residuals, under relaxation factors and many more variables. The residuals were set to a value of 10^{-6} for this work and under-relaxation factors were set to default value for each method respectively. The comparison is shown in table 2.

Table 2: Comparison of CFD algorithms

Method	**Convergence**	**No. of Iterations**	**Time taken* (sec)**	**Max. Pressure (pa)**
SIMPLE	Converged	140	11.98	183046
SIMPLEC	Converged	185	15.66	183046
PISO	Converged*	157	15.26	183046
COUPLED	Not converged	-	-	-

The time taken to obtain results is dependent on the workstation being used and the computational resources available. All the simulations were run on the same workstation on the same day without changing any background settings to keep the resource available similar while every computation.

The SIMPLE algorithm was found to be the most suitable algorithm with fastest convergence. It gives the fastest results for a steady state laminar flow fluid system.

The SIMPLEC algorithm has a skewness correction factor with was set to 0 as default. The solution takes more time and iterations to converge than the SIMPLE algorithm. It was observed that it took longer than this time to converge if a skewness correction factor more than 0 was given.

PISO algorithm has a skewness correction factor and a neighbour correction factor which were set to 1 as default. It was observed to not converge if both the factors were coupled. De-coupling those helped in convergence using the PISO algorithm.

The solution failed to converge using the COUPLED algorithm (specific to ANSYS Fluent) irrelevant of the pseudo transient condition being kept on or off. COUPLED algorithm is mainly used only for transient flow cases, but this study only focuses on laminar flow in a journal bearing, which is why the solution does not converge for this study.

It can be observed from the results that all the methods give same results but vary in computational time and iterations. This model having steady state, laminar, incompressible flow in a two-dimensional geometry is one of the simplest possible cases in this domain and is easily solved using the SIMPLE algorithm.

5. Conclusion

This study puts forward a comparison of CFD algorithms for solving two-dimensional journal bearing model using ANSYS Fluent software. The model is well meshed and quality checked so as the results are not affected by changes in mesh type. Mesh Independency is also achieved by comparing pressure values from trial runs in simulation.

It is observed that the SIMPLE algorithm was most suitable for this case. SIMPLEC and PISO algorithms gave the same results but with higher computational time and iterations. These algorithms are case specific with PISO being recommended for transient or compressible flow cases. Solution algorithms can be compared for a dynamic case of a similar two-dimensional bearing model for future scope of work.

Nomenclature

a,b : Coefficients
p : Pressure field
P : Static Pressure
F : Force
I : Unit Tensor
ρ : Density
ρg : Gravitational Force
τ : Stress Tensor
ϑ : Velocity
μ : Fluid Viscosity
nb : Grid Locations
CFD : *Computational Fluid Dynamics*
l/d : *Length/Diameter ratio*
PISO : *Pressure Implicit with Splitting of Operators*
2D : *Two Dimensional*
3D : *Three Dimensional*
SIMPLE : *Semi-Implicit Method for Pressure-Linked Equations*
SIMPLEC : *SIMPLE Consistent*
SIMPLER : *SIMPLE revised*

References

[1] D. S. Jang, R. Jetliand, S. Acharya, Comparison Of The Piso, Simpler, And Simplec Algorithms For The Treatment Of The Pressure Velocity Coupling In Steady Flow Problems, *Numerical Heat Transfer*, vol. 10, 1986, 209-228.

[2] M. Sahu, A. K. Giri, and A. Das, Thermohydrodynamic Analysis of a Journal Bearing Using CFD as a Tool, *International Journal of Scientific and Research Publications*, Volume 2, Issue 9, 2012, 1-7.

[3] A. Chauhan, A. Singla, N. Panwar, and P. Jindal, CFD based Thermo-hydrodynamic analysis of circular journal bearing, *International Journal of Advanced Mechanical Engineering, Research India Publications*, 2016, 475-481.

[4] B. Manshoor, M. Jaat, Z. Izzuddin, and K. Amir, CFD Analysis of Thin Film Lubricated Journal Bearing, *The Malasian International Tribology Conference*, Procedia Engineering Vol 68, Science Direct, 2013, 56-62.

[5] U. Manojkumar, N. H. Jagadish, and B. Kirankumar., CFD analysis of hydro-dynamic lubrication journal bearing using castor oil, *Recent Trends in Mechanical Engineering*, Springer, Singapore, 2020. 671-683.

[6] Chen Y., Wang H., Wang X., Performance Analysis and Geometrical Design of Spiral-Grooved Thrust Bearing Employing CFD Model. *J. Inst. Eng. India Ser. C 102*, 2021, 765–775.

[7] S Cupillard, S Glavatskih, M J Cervantes, Computational fluid dynamics analysis of a journal bearing with surface texturing, *Engineering Tribology*, 2008, 97-108.

[8] K. P. Gertzos, P. G. Nikolakopoulos, and C. A. Papadopoulos, CFD analysis of journal bearing hydrodynamic lubrication by Bingham lubricant, *Tribology international 41.12*, 2008, 1190-1204

[9] G. Gengyuan, Y. Zhongwei, J. Dan, and Z. Xiuli, CFD analysis of load-carrying capacity of hydrodynamic lubrication on a water-lubricated journal bearing, *Industrial Lubrication and Tribology*, 2015, 30-37.

[10] M. Deligant, P. Podevin, G. Descombes, CFD model for turbocharger journal bearing performances, *Applied Thermal Engineering*, 2010, 812-819.

[11] A. A. Raimondi and J. Boyd, A Solution for the Finite Journal Bearing and its Application to Analysis and Design: III, *A S L E Transactions, 1:1,* 1958, 194-209.

[12] Patankar, Suhas V.*, Numerical heat transfer and fluid flow*, CRC press, 2018.

[13] ANSYS Inc, *ANSYS Fluent User Manual*, 2020 R1.

,