CFD Analysis of a Journal Bearing with a Microgroove on the Shaft

Amir Jourak

M.Sc. in Mechanical Engineering, Hydropower Engineering
CONTINUATION COURSES

Luleå University of Technology
Department of Applied Physics and Mechanical Engineering
Division of Fluid Mechanics
CFD Analysis of a Journal Bearing with a Microgroove on the Shaft

Amir Jourak

Luleå University of Technology
Department of Applied Physics and Mechanical Engineering
Division of Fluid Mechanics
Contents

1 Preface I

2 Summary II

3 Introduction 1
   3.1 Previous Research on Surface Texturing ................. 2
   3.2 Introduction to CFD .................................... 3
   3.3 Thesis Objectives ...................................... 4

4 Theory 5
   4.1 Governing Equations .................................... 5
   4.2 Geometries Used and Parameters Studied ................ 5
   4.3 The Boundary Conditions ................................. 7
   4.4 Transient Simulation Control Parameters ................. 8
   4.5 Meshing Strategy ....................................... 10

5 Results 11
   5.1 Convergence Criteria ................................. 11
       5.1.1 Frequency Analysis of the Load ............... 15
   5.2 Numerical Difference .................................. 15
       5.2.1 Steady State Simulation ....................... 19
       5.2.2 Transient Simulation .......................... 22
   5.3 Load Carrying Capacity ............................... 23

6 Conclusion and Future Work 28
1 Preface

This Master thesis summarizes my work carried out at the Division of Fluid Mechanics at Luleå University of Technology. The supervisor of this research has been Associate Professor Michel Cervantes. This project is performed as a Master Thesis for the Master of Science Program in Mechanical Engineering specializing in Hydropower Engineering.

I would firstly like to thank my supervisor Michel Cervantes for his valuable support and guidance through all my work. I am grateful to Samuel Cupillard at the Division of Machine Elements for providing me some technical help in using ANSYS ICEM and ANSYS CFX. Special thanks go to my colleagues Ida Jansson, Berhanu Mulu Geberkiden and all the staff at the Division of Fluid Mechanics for creating an exciting and enjoyable working environment. And I am also grateful to Gregory F Simmons for his help to check the English grammar of this work.

Finally, special thanks must go to my parents, Ali and Nasrin and my sister, Niloufar who have always supported and greatly encouraged me throughout my academic life.

Amir Jourak
Luleå, February 2008
2 Summary

Journal bearings are used in many applications. An increase of their load carrying capacity and/or decrease of their losses may allow for savings of an enormous amount of energy. One way to enhance the characteristics of journal bearings is to modify the surface texture of the stator. The purpose of the present study is to investigate the influence of a textured rotor on journal bearing characteristics. Special attention is given to the losses and the load carrying capacity under unsteady conditions.

Computational Fluid Dynamics is used to model the flow between the rotor and the journal. The full Navier-Stokes equations are solved under unsteady conditions with the commercial software ANSYS CFX 11. A two-dimensional geometry is used to model the bearing. In order to make a simulation, a separate rotor and stator are made and are attached to each other using a rotor-stator interface. A journal bearing with a dimpled rotor is compared with the same smooth journal bearing.

Firstly, a single geometry for the smooth journal bearing is used to evaluate the possible numerical characteristics created by the interface. The results illustrate no numerical difference between using a single geometry or using two geometries. This result shows the high capability of the ANSYS CFX software to handle moving objects in the flow and allow further investigation on rotor texturing using this method.

Comparison between a dimpled rotor and smooth rotor shows no increase in load carrying capacity of the journal bearing when thermal effects are not taken into account.
3 Introduction

Since the time of ancient Egypt, people have used the energy in flowing water to operate machinery and grind grain and corn. However, hydropower had a greater influence on people’s lives during the 20th century than at any other time in history. Hydropower played a major role in making the wonders of electricity a part of everyday life and helped spur industrial development. Hydropower continues to produce 24 percent of the world’s electricity and supply more than 1 billion people with power. The first hydroelectric power plant was built in 1882 in Appleton, Wisconsin to provide 12.5 kilowatts to light two paper mills and a home. Today’s hydropower plants generally range in size from several hundred kilowatts to several hundred megawatts, but a few big plants have capacities up to and over 10,000 megawatts and supply electricity to millions of people. Worldwide, hydropower plants have a combined capacity of 675,000 megawatts and annually produce over 2.3 trillion kilowatt-hours of electricity, the energy equivalent of 3.6 billion barrels of oil [1]. In Sweden approximately 50% of electricity currently comes from hydropower.

The global trend is to implement sustainable energy sources such as wind power, solar energy, etc. In order to achieve this goal hydropower can be used as a base energy generation method. Due to availability of hydropower, it is increasingly being used to feed the electricity into the grid at times of high demand. It takes only a couple of minutes to start the machine compared to thermal power plants which take several days. This means the machines have to be started and stopped continuously, something which was not taken into account in the initial design. The consequences for the rotor system are not clear at the present time and need further investigation.

An important part of the rotor system is the journal bearing which is one of the sources of power loss in the system. This kind of bearings works in the hydrodynamic lubrication regime, thereby there is a film of lubricant between the shaft and the journal to separate them. As the shaft rotates in the journal it continuously provides energy to the lubricant. This energy is dissipated, in the lubricant, as a result of fluid shear, friction and increase in the temperature within the lubricant and the bearing. All of which further contribute to reduction of efficiency of the system.

For such a hydrodynamic system, the friction always needs to be
minimized. There are a few alternatives [2] to decrease the friction, such as:

- Reducing the lubricant viscosity.
- Operate the system at lower loads or shear rates.
- Modifying the geometry of the contact.
  - By introducing singularities as a step.
  - By introducing a textured surface in the contact.

This work concentrates primarily on the investigation of the effects of surface texturing of the rotor on the load carrying capacity of the bearing by using CFD (Computational Fluid Dynamic).

### 3.1 Previous Research on Surface Texturing

Surface texturing has been a subject of several theoretical and experimental studies. This is due to the fact that small improvements in bearing performance can be greatly economically beneficial. Amongst all the research which has been carried out on surface texturing, little of it investigated the effect of shaft texturing on bearing performance.

Snegovski and Bulyuk conducted an experiment to study journal bearings with and without micro-grooves on the shaft [3]. Parts of the shaft had surface micro-relief formed by a system of non-intersecting sinusoidal microgrooves obtained by the vibrating roll-forming method. According to the authors the load carrying capacity was increased 1.5–2 times compared to a smooth shaft for sliding velocities ranges from 30 to 60 m/s. For grooves greater than 12 – 15 µm no improvement in bearing performance was achieved. The same bearings were tested in the low speed range. Frictional losses were reduced by 10 to 15% with the dimpled shaft. Bulyuk [4] worked on thermal analysis of sliding bearings with micro-channels on the shaft. He claimed that there is a forced turbulization in the loaded zone. The heat removal from the shaft becomes 1.8 – 4 times greater with micro-channels compared with the smooth shaft.

Arghir et al. in [5] have shown that Stokes equations are inadequate to predict pressure build up with the presence of macro-roughness, as inertia effects can be of importance. Sahlin et al. [6] confirmed this
finding later and presented an optimization of the geometry. Cupillard et al. [2] studied a complete textured bearing using the full Navier-Stokes equations and a cavitation model. As the authors illustrated, the coefficient of friction can be reduced if dimples of suitable width are introduced. According to the authors, this can be achieved either in the region of maximum hydrodynamic pressure for a bearing with a high eccentricity ratio or just downstream of the maximum film for a bearing with a low eccentricity ratio. A new effect of pressure build up has been identified at low eccentricity ratio. The authors in [7] also investigate the mechanism of pressure build up in a convergent gap between two sliding surfaces due to the texture. According to them, as the fluid receives energy from the moving wall, lower losses in the inlet than in the outlet part produce positive variation of the mechanical energy in the inlet part and pressure is built up. It has also been shown that the pressure gradient decreases when recirculation occurs.

Previous research shows that surface texturing is likely to be a way of improving the performance of a journal bearing. All the work which has been done so far on shaft texturing was experimental and a complete study of the effect of the textured shaft on improving bearing performance using the full Navier-Stokes equations has not yet been undertaken. Hence, a numerical study of a journal bearing with a single dimple on the rotor using the Computational Fluid Dynamic (CFD) is carried out.

3.2 Introduction to CFD

CFD is an acronym for Computational Fluid Dynamics, or the technique of using a computer to predict how fluids flow. Numerical methods for the solution of ODEs or PDEs have been conceptually conceived since the time of Newton. However, with the absence of computers at that time there was no way for the application of these techniques. With today's computers, on the other hand, numerical techniques to solve differential equations are readily available.

CFD works by solving the equations of fluid flow over a region of interest, with specified conditions on the boundary of that region. The partial differential equations which describe the processes of momentum and mass transfer are known as the Navier-Stokes equations. The most common method which is used to solve these equations is known as the finite volume method. A finite volume method (FVM) discretization is
based upon an integral form of the PDE to be solved (e.g. conservation of mass, momentum, or energy). The PDE is written in a form which can be solved for a given finite volume (or cell). The computational domain is discretized into finite volumes and then for every volume the governing equations are solved [8].

The software used in this work, ANSYS CFX, is based on finite volume technique.

3.3 Thesis Objectives
The objective of the present research is to investigate the influence of the textured rotor on a journal bearing with special attention to the losses and load carrying capacity under unsteady conditions in hydro-electric power generation applications. Evaluation of the CFD codes capabilities of handling moving objects in the flow is also investigated.
4 Theory

4.1 Governing Equations

The Navier-stokes equations are solved using CFX 11. No Reynolds or Stokes assumptions are made in the equations. The equations are applied without body force and with constant properties. They are unsteady and solved in the $x$ and $y$ direction only. The flow is laminar and unsteady and isothermal conditions are applied. With these properties the Navier-stokes and continuity equations can be written respectively,

$$\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_i} (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right)$$ (1)

$$\frac{\partial}{\partial x_i} (\rho u_i) = 0$$ (2)

4.2 Geometries Used and Parameters Studied

A two dimensional model identical to the one used by Cupillard et al. [2] to perform simulations with shallow dimples is used. The lubricant used is also as in [2]. The difference is, in this work only one dimple is used and it is placed on the shaft rather than the bearing. The lubricant used had a density of $840 \text{ kg/m}^3$ and a dynamic viscosity of $0.00127 \text{ Pa s}$. The dimensions used for the journal bearing are as follows: $R_s = 0.05 \text{ m}$ (the shaft radius), $c = 0.145 \text{ mm}$ (the radial clearance), $\epsilon = 0.1$ (the eccentricity ratio) and $\omega = 48.1 \text{ rad/s}$ (the angular velocity).

In order to perform a simulation a separate rotor and stator are made and attached to each other using a rotor-stator interface. Figure 1 shows the dimpled rotor, rotor-stator interface and stator. Figure 2 shows the dimple geometry where, $w = 0.004 \text{ m}$ (the width of the dimple) and $d = 0.00004437 \text{ m}$ (the depth of the dimple). With these dimensions $d/h_{\text{min}} = 0.34$ (ratio of the dimple depth to the minimum film thickness), the dimple according to [2], is categorized as a shallow dimple. A single geometry for the smooth rotor is also used so as to evaluate the numerical difference between using one geometry and two geometries which are connected to each other by an interface.

The load carrying capacity at each time step is calculated from the integration of the pressure acting on the shaft.
\[ L = \sqrt{\left( \int_0^2 \int_0^{2\pi} pr \cos \theta \, d\theta \, dz \right)^2 + \left( \int_0^2 \int_0^{2\pi} pr \sin \theta \, d\theta \, dz \right)^2} \]  

(3)

The average load carrying capacity over a period, \( T \), is then calculated by

\[ \bar{L} = \frac{1}{T} \int_t^{T+t} L(t) \, dt \]  

(4)

Where

\[ T = \frac{2\pi}{\omega} \]  

(5)

Figure 1: Textured rotor model in the journal bearing
4.3 The Boundary Conditions

In this simulation the frame of reference is changed at the interface between rotor and stator thereby one frame change/mixing model should be used to connect the two meshes in the interface. There are three types of frame change/mixing models in ANSYS CFX: Frozen rotor, Stage and Transient rotor-stator [8].

- Frozen rotor: The frame of reference and/or pitch is changed but the relative orientation of the components across the interface is fixed. The two frames of reference connect in such a way that they each have a fixed relative position throughout the calculation. With some account of the interaction between the two frames this model produces a steady state solution to the multiple frame of reference problems. The advantage of this model is it requires the least amount of computational effort of the three frame change/mixing models but the disadvantage of this model is the losses incurred in the transient situation as the flow is mixed between stationary and rotating components is not modeled.

- Stage: The Stage model is an alternative to the Frozen Rotor model for modeling frame and/or pitch change. Instead of assuming a fixed relative position of the components, the stage model performs a circumferential averaging of the fluxes through bands on the interface. Steady state solutions are then obtained in each reference frame. This model allows steady state predictions to be obtained for multi-stage machines. The Stage model usually requires more computational effort than the Frozen Rotor model to converge, but not as much as the Transient Rotor-Stator model.
• Transient rotor-stator: This model should be used any time it is important to account for transient interaction effects at a sliding (frame change) interface. It predicts the true transient interaction of the flow between a stator and rotor passage. It ultimately accounts for all interaction effects between components that are in relative motion to each other. If one is interested in simulating a periodic-in-time quasi-steady state, then it may be helpful to first obtain a steady state solution using Frozen Rotor interfaces between components.

With all these descriptions for different frame change model available in ANSYS CFX, the Frozen rotor interface is used to obtain a steady state solution because it requires the least amount of computational time to converge and it can also be used as an initial guess for unsteady simulation with Transient rotor-stator interface. The most important thing is that transient rotor-stator interface predicts very good results for all transient interaction of the flow between rotor and stator but it has one disadvantage: the computer resources required are considerable, in terms of simulation time, disk space and quantitative post processing of the data.

After attaching the rotor and stator with the Frozen rotor interface for steady and the Transient interface for unsteady simulation the following boundary conditions are applied: The rotor is set to be rotating with \( \omega = 48.1 \text{ rad/s} \), the stator is set to be stationary and the reference pressure is set to be zero at the stator wall, thereby all the relative pressures are measured with reference to this pressure. The no slip condition is used both for the rotor and the stator wall. The symmetric boundary condition is used for circumferences of the rotor and stator.

4.4 Transient Simulation Control Parameters

Transient simulation of the flow is set up by using the transient rotor-stator interface. Several parameters control the accuracy and convergence of transient simulation, which the most important are: Time Step, Courant Number, Maximum Coefficient Loops and Transient Scheme [8].

• Time step: This provides a way for ANSYS CFX to track the progress of real time during the simulation. If the total simulation time is set to \( T \) and time step to \( \Delta T \), then the ANSYS CFX solver will continue to compute solutions at each time step iteration
until $N\Delta T = T$, where $N$ is the number of time steps actually performed by the ANSYS CFX-Solver. Thereby by reducing the time step the accuracy of the transient simulation will be increased.

- Courant Number: For transient flows Courant number is of great importance which for a one dimensional grid is defined by: $Cr = \frac{u\Delta T}{\Delta x}$ where $u$ is the fluid speed, $\Delta T$ is the time step and $\Delta x$ is the mesh size. The Courant number calculated in ANSYS CFX is a multidimensional generalization of this expression where the velocity and length scale are based on the mass flow into the control volume and the dimensions of the control volume. It is recommended that the Courant number be on the order of unity to accurately resolve transient details.

- Maximum coefficient loops: At each time step in a transient simulation, the ANSYS CFX Solver performs several coefficient iterations or loops, either to a specified maximum number or to the predefined residual tolerance. The maximum number of iterations per time step may not always be reached if the residual target level is achieved first. A value of 3 iterations per time step should be sufficient for most single phase simulations, and values higher than 5 are unlikely to improve accuracy. In multiphase cases, the default value of 10 iterations per time step may be more appropriate. Initially, one should use these values and then adjust the time steps to control the balance between accuracy and solution time. It should be mentioned that improving the accuracy is much more efficiently achieved by reducing the time step than increasing the coefficient iterations.

- Transient Scheme: The transient scheme defines the discretization algorithm for the transient term. There are two schemes available, first order backward Euler and second order backward Euler scheme. The first order backward Euler transient scheme is first order accurate. Its behavior is analogous to the Upwind differencing scheme for advection terms, and suffers from similar numerical diffusion, however it is useful for initial studies. The second order backward Euler scheme is second order accurate and it is applicable for constant and variable time step sizes. For most transient simulations using a second order transient scheme is recommended.
4.5 Meshing Strategy

The generated grid for the simulations is composed of 47040 and 80360 hexahedral elements for the smooth rotor-stator (both one and two geometries) and for the dimpled rotor-stator, respectively. Since a finer mesh is used inside the dimple to resolve the gradients occurring at the inlet and outlet section, the number of elements of the grid in the dimpled rotor-stator is higher than the smooth case.

Mesh refinements for this project is exactly the same as in [2]. Therefore it is predicted to have the same numerical error of around 0.5% for the load. In order to completely resolve the gradients occurring at the inlet and outlet of the dimples, refinements are made inside the dimple. Since there is no cavitation with the eccentricity used ([2], Shallow dimples) no refinements are made for cavitation. Figure 3 shows the refined grid for the dimple.

![Figure 3: Grid for the dimple](image)
5 Results

5.1 Convergence Criteria

The simulations started with steady state calculation of the flow with frozen rotor stator interface. The numerical equations were first solved using an upwind advection scheme. After convergence was reached, the advection scheme was set to the second order with the blend factor of 1 which is the most accurate scheme for solving the numerical equations. The convergence control for physical timescale, with the value of $1/\omega = 0.0208$ s, was used as it is appropriate for rotating machinery applications [8].

Convergence was fully achieved for the steady state simulation and the residuals hit the targeted level. This result provided an initial condition for the transient simulation and also a way to compare numerical difference between a rotor and stator attached with a frozen interface and a single rotor-stator geometry. Figure 4 shows the momentum and mass RMS residuals.

![Figure 4: Convergence history curve for steady state simulation](image-url)
As can be seen from the figure 4, after the residuals dropped down the targeted level and the simulation was restarted using a second order scheme, the residuals increased suddenly and declined steadily until they hit the targeted level and convergence occurred. This increase of residuals is due to the fact that this commercial code puts a damping factor on its solver.

The unsteady simulation was started by using the steady state result as an initial guess. The time for the rotor to rotate completely in the stator is 0.1306 second since $\omega = 48.1 \, \text{rad/s}$, thereby the total time of simulation was set to at least 2 second so that the rotor could be swirled in a stator fifteen times and a steady state condition could be obtained. In order to judge the convergence of the simulation, the load carrying capacity of the bearing is monitored through the simulation. Initially, a large time step, with a first order backward Euler transient scheme was used to find a rough solution to the problem. The maximum coefficient loops value of 10 was used through the simulation. The RMS residuals were dropped down by reducing the time step gradually. After the residuals dropped down well below $1.0e-004$ the transient scheme was set to second order backward Euler in order to find the final result. Each time the time step was reduced, RMS Cr number was monitored to check if the time step reduction was sufficient enough for the solver to adequately simulate transient interaction of the flow between the stator and rotor. When the RMS Cr number reached the order of unity no more reduction in time step was applied.

Figures 5 and 6 show the load carrying capacity for dimpled, smooth bearing for two geometries with a load for single geometry and convergence history curve for smooth bearing transient simulation.
Figure 5: Load carrying capacity for a dimpled and a smooth rotor bearing with a load for single geometry, (In figure 5(b), for Cr = 131, the load is between 4150 to 5500 N/m)
As it is shown in figure 6, the RMS residuals keep fluctuating below the magnitude of $1.0e-004$ and it was difficult to judge the convergence from the residuals. However, as illustrated in figure 5, the convergence was fully achieved after the rotor swirled several times in the stator.
5.1.1 Frequency Analysis of the Load

Although the convergence was achieved for the problem, there were some oscillations in the load carrying capacity. A frequency analysis was done using fast Fourier transform. Figure 7 shows the load carrying capacity for Cr = 2.41 and 1 and their frequency contents for the smooth rotor.

As it can be seen from figures 7(b) and 7(c), there are several frequencies in the load. The first frequency but not the main one observed for Cr = 2.41, is 7.61 Hz, is the frequency of rotation of the rotor \((2\pi/48.1 = 7.65HZ)\). The second frequency \((22.83 HZ)\) is the third harmonic of the frequency of rotation. However, for Cr = 1, the first and main frequency is frequency of rotation with the magnitude of 7.645 Hz. The other frequencies as it can be seen from the figures 7(b) and 7(c) are the harmonics of the frequency of rotation.

Figure 8 shows the load carrying capacity for the dimpled rotor and the frequency content of it.

It can be easily observed from the figure 8(a) that there is one main frequency, and that is the frequency of rotation of the dimple. As it can be seen from figures, 7(c) and 8(c), for Cr = 1, the main frequency of the load is exactly the same for the dimpled and smooth rotor with the magnitude of 7.645 Hz. Comparison between figures 7(b) and 8(b) shows exactly the same value of 7.61 Hz for Cr greater than 1.

5.2 Numerical Difference

Single geometry and two geometries attached by interface were used to assess the numerical differences and evaluate the CFD code capabilities to handle moving objects in the flow. In other words, this comparison is used to assess the accuracy of using the General Grid Interface [8] between the rotor and stator geometries. Figure 9 presents the part of geometries used. For both cases the total number of elements (47040 hexahedrons), dimensions and properties as described in 4.2 are the same.
Figure 7: Figure (a) - load carrying capacity for the smooth rotor for $Cr = 2.41$ and 1, Figure (b) - Frequency contents of the smooth rotor load for $Cr = 2.41$, Figure (c) - Frequency content of the smooth rotor load for $Cr = 1$
(a) Load carrying capacity for $Cr = 5$ and $1$

(b) Frequency content of dimpled rotor for $Cr = 5$

(c) Frequency content of dimpled rotor load for $Cr = 1$

Figure 8: Figure (a) - load carrying capacity for the dimpled rotor for $Cr = 1$, Figure (b) - Frequency contents of the dimpled rotor load for $Cr = 5$, Figure (c) - Frequency contents of the dimpled rotor load for $Cr = 1$
Figure 9: Part of two dimensional models

(a) Single geometry

(b) Combined geometry with interface
5.2.1 Steady State Simulation

The first comparison was conducted between the steady state simulations. Tangential velocity variation from rotor to stator, tangential velocity contour, tangential velocity profile and load carrying capacity are compared. See figures, 10, 11 and 12.

![Figure 10: Tangential velocity magnitude for single and combined geometry VS Distance from rotor](image)

The magnitude of the velocity decreases linearly from 2.4050 m/s (48.1 rad/s × 0.05 m = 2.4050 m/s) on the rotor to 0 m/s on the stator. As can be seen from figures 10 and 11 there is no difference between the two simulations in this case. Figure 12 shows the velocity profile. As it can be seen from the figure the velocity profile declined linearly for both cases and the profiles look like the same.

Comparison between the load carrying capacity of the bearings shows 1.3 % difference, with the magnitudes of 6885 and 6798 N/m for the single and combined geometry respectively.
Figure 11: Figure (a) - Velocity contour for single geometry, Figure (b) - Velocity contour for combined geometry with interface
Figure 12: Tangential velocity profile
5.2.2 Transient Simulation

In the second comparison, the steady state solution of the single geometry and combined geometry are compared with the unsteady solution of the combined geometry. Figure 13 shows the load carrying capacity for transient simulation of combined geometry and steady state simulation of the single geometry.

![Figure 13: Load carrying capacity for Transient combined geometry and steady state single geometry](image)

The mean value for load carrying capacity for transient simulation of the combined geometry over one period is 6696 N/m which is 1.52 % and 2.82 % lower than of that in the steady state simulations of combined geometry and single geometry respectively. The difference between the load in the transient simulation and steady state simulation is certainly due to the interface as described in 4.3.
5.3 Load Carrying Capacity

Load carrying capacity of the textured rotor was compared with the same smooth rotor for two geometries attached by a transient rotor-stator interface. Figure 14 shows the load carrying capacity of the dimpled and smooth rotor over one period for $Cr = 1$.

As can be seen from the figure 14, there are two main differences between the loads. The first one as discussed in 5.1.1, is the frequency of the loads. While there are several frequencies in the smooth load, there is one main frequency in the dimpled load (See figure 8(c)), which is the frequency of rotation of the dimple. The second difference is the variation of the loads. The load for the smooth bearing varies near its mean value with some oscillations. On the other hand, the load for the textured rotor bearing has only one peak and dip over its period. Therefore, more analysis of the load was performed. Figure 15 shows the load for textured rotor over one period.

As illustrated in figure 15, interestingly, when the load increased from its dip to its peak, the dimple on the rotor rotates from the highest film
Figure 15: Load carrying capacity for dimpled rotor over one period thickness to the lowest film thickness position in the journal bearing. As the load declined smoothly to its dip again, the dimple changed its position to the highest film thickness. Figure 16 shows the pressure profiles for the dimpled and smooth rotor in the highest and lowest film thickness positions.

A negative effect on pressure distribution can be observed from figure 16(a). As can be seen from the figure, at the highest film thickness the pressure curve for dimpled rotor stay below the pressure curve for the smooth rotor. Nevertheless, in the lowest film thickness as it is illustrated in figure 16(b), a positive effect on pressure distribution can be observed. Figure 17 shows the velocity profiles at the highest and lowest film thickness positions.

Comparison between the two velocity profiles shows no difference. At the highest and lowest film thickness positions, the velocity decreases linearly from its highest magnitude on the rotor to zero on the stator.
Figure 16: Pressure profiles for dimpled and smooth rotors
Figure 17: Velocity profiles at the highest and lowest film thickness positions

(a) Highest film thickness

(b) Lowest film thickness
In order to judge whether texturing the rotor can increase the load carrying capacity of the journal bearing, the mean value of the loads are compared with each other. The load carrying capacities per unit width are 6696 and 6719 N/m for the smooth and dimpled rotor bearing respectively. With the magnitude of the loads obtained, it is hard to evaluate the effect of the dimple on the load carrying capacity of the journal bearing. The difference between the mean value of the loads is around 0.34 % which can be neglected. Hence, the dimple didn’t change the load carrying capacity of the journal bearing without thermal effects being taken into account.
6 Conclusion and Future Work

Numerical simulations of a journal bearing with a single dimple on the rotor using Computational Fluid Dynamic (CFD) have been performed using the commercial software ANSYS CFX 11. The objective of this project was to investigate the influence of a textured rotor on the load carrying capacity of a journal bearing and also to evaluate the CFD code capabilities to handle moving objects in the flow.

Convergence criteria of the problem and frequency analysis of the load was done in order to understand the possible frequencies of the loads. By monitoring the load during the simulation, convergence judgment was accomplished. The first but not the main frequency observed in the smooth rotor journal bearing load was the frequency of rotation of the rotor. The rest of frequencies were harmonics of the frequency of rotation. On the other hand, there was one and main frequency in the dimpled rotor journal bearing load, which was the frequency of the rotation of the dimple inside the bearing.

Comparison was made between the single geometry and combined geometry by interface to see the possible numerical characteristics created by the interface. The results illustrated no numerical difference between a single geometry and two geometries. This result demonstrates the CFD code capabilities to handle moving objects in the flow.

Finally, Comparison between the textured rotor and smooth rotor showed no increase in load carrying capacity of the journal bearing. Possible reasons are: only one dimple was used, the flow was considered to be laminar, i.e, no increase in speed of rotation can be applied and isothermal condition was applied.

This work mainly shows the possibility of investigation of the effects of texturing the rotor on the increase/decrease of the load carrying capacity of the journal bearing by using ANSYS CFX. There are a few points which deserve future investigation:

- Increasing the number of dimples on the shaft.
- A heat transfer model could be added to the problem to investigate the influence of the rotor texturing on the load carrying capacity of the bearing in non-isothermal conditions.
- A turbulence model could be used to model the effect of shaft texturing on the high speed rotation of the shaft in the journal bearing.
Nomenclature

\( \epsilon \) Eccentricity ratio \((-e/c)\) 
\( \mu \) Fluid dynamic viscosity \(\text{Pa.s}\) 
\( \omega \) Angular velocity \(\text{rad/s}\) 
\( \bar{L} \) Average load per unit of length in one period \(N/m\) 
\( \rho \) Fluid density \(\text{kg/m}^3\) 
\( \theta \) Circumferential coordinate \(\text{rad}\) 
\( c \) Radial clearance \(m\) 
\( Cr \) Courant number 
\( d \) Dimple depth \(m\) 
\( d/h_{\text{min}} \) Dimple depth to minimum film thickness ratio 
\( e \) Eccentricity \(m\) 
\( h_{\text{min}} \) Minimum film thickness \(m\) 
\( L \) Load per unit of length \(N/m\) 
\( l \) Bearing length \(m\) 
\( P \) Fluid pressure \(\text{Pa}\) 
\( r \) Radial coordinate \(m\) 
\( R_s \) Shaft radius \(m\) 
\( T \) Period \(s\) 
\( u_i \) Velocity component in the i direction \(m/s\) 
\( w \) Dimple width \(m\) 
\( x_i \) Coordinate in i direction \(m\) 
\( z \) Axial coordinate \(m\)
References


