Numerical simulation of elastohydrodynamic and hydrodynamic lubrication using the Navier-Stokes and Reynolds equations

Torbjörn Almqvist

Department of Mechanical Engineering
Division of Machine Elements

2001:33 • ISSN: 1402 - 1757 • ISRN: LTU - LIC - - 01/3 - - SE
Numerical simulation of elastohydrodynamic and hydrodynamic lubrication using the Navier-Stokes and Reynolds equations

Torbjörn Almqvist

Division of Machine Elements
Department of Mechanical Engineering
Luleå University of Technology
SE-971 87 Luleå, Sweden

2001
PREFACE

We know that the benefits of lubrication were known as early as 3500 BC from illustrations showing Egyptians using water to lubricate the sledges used to transport enormous building blocks. However, the mechanisms of hydrodynamic lubrication were not explained until the 19th century when Osborne Reynolds (1886) made his pioneering work. However, even though more than a century has passed, a lot of work remains to be done in order to fully understand the lubrication process.

The first time I heard the word Tribology (Tribology comes from the Greek word tribos which means rubbing) was in the spring of 1998 when my supervisor to be contacted me, and wondered if I would be interested in post graduate studies at the Division of Machine Elements in Luleå. The research area was elastohydrodynamic lubrication (EHL, a sub-area of Tribology). After a couple weeks of thought, I decided to give it a try.

Having worked for two and a half years in the field of EHL, I now understand what an interdisciplinary subject it is. In order to work theoretically with EHL, knowledge of several other disciplines is required; fluid mechanics, solid mechanics, thermodynamics and rheology. In addition, fields such as materials science, chemistry, physics, mathematics, machine design and performance and reliability are also closely related to the subject.

A number of my colleagues are working in the field of EHL or with closely related subjects at the Division of Machine Elements. This, in addition to the great atmosphere, gives me hope of being able to make and important contribution of the final goal of solving the whole EHL phenomenon.

The people I want to thank are firstly, and most importantly, my wife Maria and my sons David and Gustav who have supported me during this work. My supervisor, Roland Larsson who has a broad knowledge of EHL and who has inspired me when the work looked like a hopeless struggle. And finally, all my colleagues at the Division of Machine Elements.

I have also had the opportunity to be a student of the National Graduate School in Scientific Computing (NGSSC), a national programme, which has over 50 graduate students from different areas of science. The programme is funded by the Swedish Foundation for Strategic Research (SSF).

I would also like to thank the Swedish Council for Engineering Sciences (TFR) for financial support.

Torbjörn Almqvist, June 2001.
ABSTRACT

The work presented in this thesis concerns computer simulations of the lubrication process. The main subject of interest is elastohydrodynamic lubrication (EHL) and, to some extent, hydrodynamic lubrication (HL). The thesis comprises an introductory section and three papers; referred to as A, B and C.

Simulation of EHL is an inter-disciplinary task, incorporating the fields of fluid mechanics, solid mechanics, thermodynamics and rheology. In almost all numerical simulations of lubrication performed today, the hydrodynamics are modelled using the Reynolds equation. This equation is derived from the equations of momentum and continuity and using a thin film approximation. However, the assumptions made when deriving this equation limits the size of the computational/spatial domain and the equation cannot predict pressure variations across the lubricating oil film.

The subject of papers A and B are numerical simulations using the full equations of momentum and continuity, (Paper B), and the equation of energy (Paper A). The main aim of the work was to investigate the possibilities of carrying out numerical simulations based on the above equations. The rheology was assumed to be Newtonian; the equations are then commonly referred to as the Navier-Stokes equations (N-S). The second aim of the work was to investigate the possibilities of using a commercial software, CFX 4.3 [1], to carry out the numerical simulations.

The results in Paper A show that it is possible to simulate thermal EHL line contacts up to pressures of approximately 1 GPa. The limitations of the approach are due to a singularity that can occur in the equation of momentum when a critical shear stress is reached. With a more complete rheological model (non-Newtonian rheology) it should be possible to perform simulations at even higher contact pressures.

Paper B presents the results of isothermal simulations comparing the N-S and Reynolds equation approaches. The result show that there may be some discrepancies between the two approaches; although only small discrepancies have been observed in the smooth line contact simulations made.

The characteristics of the EHL-contact with a wide range of scales and large gradients in pressure, viscosity and temperature make developing accurate numerical simulations to a difficult task. The computational cost is high due to the small under-relaxations factors that must be used in order to obtain converged numerical solutions.

The work to date has shown that it is possible to use the extended approach in conjunction with a commercial software, CFX 4.3 [1]. This approach makes it possible to extend the computational domain in future in EHL-simulations, where the Reynolds approach is not valid.

Paper C presents the results of simulations of a lubricated pivoted thrust bearing. The objective of this study was to verify a thermo-hydrodynamic (THD) model for this type of bearing. The model developed handles three-dimensional temperature distribution in the oil film and pad, as well as two-dimensional temperature variation in the runner. The viscosity and density are treated as functions of both temperature and pressure.

Experiments have been performed in a test rig consisting of two identical equalising pivoted pad thrust bearings. Experimentally measured power loss, runner temperature and pressure profiles as a function of load and rotational speed were compared with the theoretical investigations.

The results showed fairly good agreement when the oil inlet temperature and heat transfer coefficients were modified in order to obtain the same runner temperature in both theory and experiment.
CONTENTS

1 INTRODUCTION .................................................................................................................. 1

1.1 Background .................................................................................................................. 1
1.2 Objectives ................................................................................................................... 3
1.3 CFD .............................................................................................................................. 5
1.4 Outline ......................................................................................................................... 5

2 ELASTOHYDRODYNAMIC LUBRICATION ....................................................................... 6

2.1 Physics of EHL-contacts ............................................................................................. 6
2.2 Characteristics of EHL ................................................................................................. 7
2.3 Rheology ....................................................................................................................... 8

3 MATHEMATICAL FORMULATION ................................................................................ 10

3.1 Governing equations .................................................................................................. 10
3.2 The singularity ............................................................................................................ 12

4 NUMERICAL SIMULATIONS ......................................................................................... 14

4.1 Discretisation ............................................................................................................. 14
4.2 Solution Algorithm .................................................................................................... 14
4.3 Error analyses ........................................................................................................... 16
4.4 Cavitation ................................................................................................................... 17

5 SUMMARY OF PAPERS AND CONCLUDING REMARKS ............................................ 19

6 FUTURE WORK .............................................................................................................. 21
APPENDED PAPERS


B. Almqvist T, Larsson R. Comparison of Reynolds and Navier-Stokes Approaches for Solving Isothermal EHL Line Contacts, to be presented at the 2001 WTC Conference, Vienna, Austria.

1 INTRODUCTION

In this section, a brief introduction to the work is given. The section consists of a background, objectives, a short explanation of the subject CFD and finally an outline of the thesis.

1.1 Background

Tribology is the science of interacting surfaces in relative motion. Tribology includes friction, wear and lubrication. Friction is the tangential component of the contact force between two bodies i.e. the force that must exerted in order move a body against a plane, see Fig 1.1.

![Figure 1.1. Friction (F_t) and normal force (F_n) acting on a box when moving with a velocity U against a plane.](image)

Wear is a destructive process whereby surface material is removed, and often results in damage to machine components. Lubrication is a way of controlling both the friction and wear. However, a lubricant can have other effects such as cooling, insulation, transport of contaminants, protection against corrosion and damping.

**Hydrodynamic lubrication (HL)** occurs when the pressure induced in the lubricating fluid separates or partly separates the surfaces in motion. **Elastohydrodynamic lubrication (EHL)** occurs when the component surfaces deform due to the fluid pressure.

An example of an EHL-contact (soft EHL) is in the contact between a tire and a wet road. The pressure induced in the water film between the tire and the road will both deform (in addition to the deformations of the tire when driving on a dry road) and tend to lift the tire. When the tangential velocity of the tire rises above a critical value, the pressure will be high enough to separate the tire from the road, resulting in aquaplaning.

In machine components EHL occurs in contacts such as gears, cams and bearings. There is, of course, an economic interest in preventing damage to machine components, especially in situations where failure of a machine component, such as a bearing in an electric motor, can result in a long stop in production.

This kind of problem forces the designers of machine components to try to predict the running conditions the component will experience, which of course is a difficult task.
In order to ensure long life, it is not uncommon to over dimension a component at a higher cost and weight than is needed.

A typical component where EHL is present is a roller bearing, see Fig. 1.2.

![Figure 1.2. Roller bearing.](image)

Thrust bearings and journal bearings are examples of HL-bearings and, in most cases, the contact pressure is not high enough to deform the contact surfaces. However, if the shear rates are high, heat will be generated in the oil film which can lead to thermal crowning of the surfaces.

Typical applications for thrust bearings are in hydro-electric power-plant stations, where demands on durability are extremely high. A typical tilting pad thrust bearing can be seen in Fig 1.3.

![Figure 1.3. Thrust bearing.](image)

In hydro-electric applications, a large amount of oil is used and if leakage occurs, serious pollution of the downstream river can result. For this reason, changing to more environmentally friendly lubricants can reduce the potential of damage if leakage occurs. This raises the question of whether it is safe to change the lubricant whilst maintaining safe running conditions?

Numerical simulation is an important tool both for predicting the behaviour of lubricated machine components, design optimisation and helping choose appropriate lubricants. In order to simulate and predict the performance of the lubricated components, reliable computational models must be developed.
The modelling of bearing performance is an interdisciplinary problem, which means the model must include hydrodynamic-, thermodynamic- and structural dynamic equations. The correct choice of constitutive equations is also of major importance in order to model the physics correctly.

Finally, readers who wish to penetrate the subject of lubrication more general, see Hamrock [2] and Bhushan [3].

1.2 Objectives

There are both technical and economic interests in predicting the performance of lubricated bearings or machine components. Increasing demands on operating performance and durability forces designers to find answers to questions such as:

- What is the minimum size for a bearing which preserves safe running conditions?
- What demands are placed on a lubricant used in this application?
- How does surface roughness influence the performance?
- What are the maximum pressures or temperature gradients?
- Can operating conditions result in thermal damage to the components?
- Can a reduction in frictional losses be obtained if other lubricants are used?
- How do contaminants influence the lubrication?

Trial and error tests can be performed, but are both expensive and time consuming. The use of numerical simulations can be more effective but, of course, must be verified by experimentally obtained data.

To date, simulation of the performance of EHL-contacts have been limited to a very small computational domain, see the domain labelled contact in Fig. 1.4. One reason is the small contact area which carries the load. But also due to boundary conditions and assumptions made when deriving the governing equation that models the flow. This equation is known as the Reynolds equation and is a combination of the momentum equation, with inertia neglected, and the equation of continuity. In the derivation, the rheology is assumed to be Newtonian and the length scales across the oil film is assumed to be three orders of magnitude smaller than in the other directions.

If simulations of how contaminants in a lubricant pass through or is dragged into the contact, or the flow in the region between the rollers in a bearing are to be carried out, the equations of momentum and continuity have to be solved separately (as well as the thermodynamic and deformation equations for the solids).

In this work, the equations of momentum and continuity are solved separately and the rheology is assumed to be Newtonian. In this case the equations are then referred to as the Navier-Stokes equations (N-S hereafter).
Figure 1.4. Computational domain compared to the size of the flow domain in a ball bearing, from Larsson and Jacobson [4]. The circle named contact is the small part of the ball in contact to the outer race of the bearing. Approximately, this is the size of the domain where EHL-simulations are performed today.

The real contact surfaces in an EHL-contact are rough. If the contact roughness has a periodicity and amplitude comparable with the thickness of the lubricating film, it is uncertain that Reynolds equation can model the physics of the flow correctly.

To date, no simulations of EHL contacts using the full N-S equations have been presented. Much of the work presented in this thesis has been to investigate whether it is possible to simulate EHL contacts using N-S equations and if this can be done using a commercial software, a computational fluid dynamics (CFD) package, in this case CFX 4.3 [1].

The key research questions can thus be summarised as:

- Is it possible to use the full N-S equations for simulating EHL?
- Can a commercial CFD-software be used for this simulation?

If the answer to one or both of these questions is yes, further possibilities then emerge for the investigation of EHL such as:

- How does surface roughness influence flow?
- Can the computational domain be extended at both the inlet and outlet regions?

In the long term potential goals are to:

- Obtain a numerical model, which, given correct input data, can accurately predict friction, oil-film-thickness, pressure and temperature in machine components where EHL is present.
- Transform the results into easily applicable formulae for predicting the above mentioned effects.
1.3 CFD

The numerical method used to simulate the flow in the present work is known as computational fluid dynamics (CFD). The behaviour of flows, thermodynamics and related phenomena can be described by integro-differential equations which, in most cases, cannot be solved analytically.

One way of solving these problems is to approximate the equations by a discrete formulation, where the equations are approximated over finite domains in both space and time. The continuous integro-differential equations will transform into an algebraic system of equations, which can be solved by a computer. The result from the computations will be discrete solutions of the variable in both space and time.

1.4 Outline

The thesis begins with a survey which introduces the field of hydrodynamic lubrication (HL) and elastohydrodynamic lubrication (EHL) and the need for numerical simulations. The introductory survey is brief but references are given for the reader who wants to study the subject of HL or EHL more deeply.

The remaining parts of the thesis deal with EHL. In section 2, the physics of an EHL-contact is described. A separate part of this section deals with rheology (the constitutive equation describing the connections between deformation, time and force in a fluid or solid).

In section 3 the governing equations are presented, the intention being not to derive them, but rather simply present them in order to give the reader an insight into the equations which lie behind the numerical simulations. This section also deals with the limitations of the Navier-Stokes (N-S) approach which are due to a singularity that can occur in the momentum equation.

In section 4, the numerical work is described. This section covers most of the work that has been done, and is intended to give a brief overview of the numerical matters. Again, further references are given for the interested reader.

In section 5, a summary and concluding remarks are given and in section 6, suggestions for further work. The remainder of the thesis consists of three papers A, B and C.
2 ELASTOHYDRODYNAMIC LUBRICATION

The physics of elastohydrodynamic lubrication is difficult to understand. The high pressures, temperature variations, deformation of what may be thought of as rigid surfaces, and the small scale involved make it difficult to build an intuitive model of the behaviour of the flow. Even scientists who have spent many years studying EHL do not understand the physics completely.

To date, no complete experimental visualisations of the flow in an EHL-contact have been presented and of course, this is one of the reasons why some mechanisms of the flow have not yet to be fully understood. However, some of the mechanisms that govern the flow are understood and presented below.

2.1 Physics of EHL-contacts

The physics of EHL-contacts involves fluid dynamics, structural dynamics, thermodynamics as well as rheology. In Fig. 2.1, a simplified 2D EHL-model is presented.

In the figure, the lubricating fluid interacts with the moving surfaces through adhesive forces between the fluid and the surfaces. In the central region of the contact, viscous forces develop between adjacent layers in the fluid and viscous forces balance the forces created by the pressure. In this region the inertial forces are several orders of magnitude smaller than the viscous forces, $Re << 1$ ($Re$ denotes the Reynolds number).

However, in the inlet and outlet region of the contact, inertia is much more important. If a simulation which covers the inlet, contact and outlet is to be developed, these terms become important.

When positive load is applied to the roller, the gap between the contact surfaces decrease, which results in an increase in the shear rate. This increasing shear rates result in turn in an increased shear stress in the converging gap at the inlet region.

The pressure increases to maintain the balance between shear stresses and pressure.
If the pressure increases, the viscosity and density of the lubricant will also increase due to the pressure dependency of these variables. The pressure in the central region of the contact is so high that local elastic deformation of the contact surfaces occurs.

A temperature rise occurs in the inlet and contact regions due to viscous dissipation and compressive heating. This will tend to decrease the viscosity and density at the same time as the increasing pressure tends to increase them.

There will, however, be a maximum shear stress that the lubricant can resist before ruptures in the lubricant film occurs. Above this maximum stress, the fluid will not be affected by further increases in shear rate, or at least become less affected by a shear rate increase.

Many lubricants contain of long molecular chains which will orient themselves in the direction of flow in the contact and also become elongated. This preferred orientation will cause the viscosity to decrease (shear thinning effects) and the stretching of the molecules results in an elastic response in the fluid.

At the outlet of the contact, where the contact surfaces separate and the pressure falls rapidly, cavitation will occur.

As can be seen, the EHL-problem involves several coupled effects. For this reason it is dangerous to omit any of the effects mentioned above, for example thermal effects, in any numerical simulation.

### 2.2 Characteristics of EHL

The magnitude and range of some of the key characteristics of EHL in bearings are presented below.

- High pressures in the central region \( \sim 0.1 - 3 \) GPa
- Contact length \( \sim 0.1 \) mm
- Thin lubricant film \( \sim 10-1000 \) nm
- Temperature variations \( \sim \) two order of magnitude \( K \)
- Elastic deformations \( \sim 1-10 \) \( \mu m \)
- Viscosity variations \( \sim \) seven order of magnitude \( \text{Pa s} \)
- Density variations \( \sim 30 \% \)

A good way of describing the difference between the size of the contact length and film thickness is:

- *Magnify the area of the EHL-contact so that it covers the surface of a paper in A4 format. The film thickness is then approximately of the same order as the thickness of the paper.*
2.3 Rheology

The Greek philosopher Heraclitus (540-480 B.C) once said 'panta rei' with means everything floats. In an infinitely long perspective this is true, however, the word has also given the name to the science of rheology.

The common way of treating the dynamic part of the stress tensor in a fluid is to assume that the stress tensor is linear proportional to the rate of strain tensor i.e. Newtonian behaviour. This assumption is valid for most engineering flows.

However, the constitutive behaviour of a lubricant is far more complex. Newtonian rheology works well only up to a limiting shear stress and the assumption of Newtonian rheology is a rough simplification when the pressure or shear rate is high.

The stresses produced in contacts taking the above assumption give unrealistic values for traction in the contacts (traction is the integrated tangential shear stress along a surface) and more complete rheological models must be used.

A more complete rheological model takes a yield stress criterion into account, where the fluid does not respond to increasing shear rates, or at least has a weaker dependency on shear rate. Two of these models are the Bair-Winer limiting stress model and the Ree-Eyring model. These models also include an elastic term, but when this term is assumed to be small, the models gives a as visco-plastic behaviour and can then be formulated in an equivalent viscosity, see Khonsari and Hua [5].

The Bair-Winer limiting stress model and the Ree-Eyring model (no elastic contribution), here formulated in the equivalent viscosity $\mu^*$, are:

$$\mu^* = \tau_0 - \exp\left(\frac{\mu \gamma}{\tau_0}\right)$$ (2.1)

$$\mu^* = \tau_0 \sinh^{-1}\left(\frac{\mu \gamma}{\tau_0}\right)$$ (2.2)

The shear rate is defined by:

$$\dot{\gamma} = \sqrt{2 \epsilon_{ij} e_{ij}}$$ (2.3)

The summation convention is used here. The symbol $\tau_l$ denotes the limiting shear stress where the fluid ceases to respond to increasing shear rate. The symbol $\tau_0$ represents the shear stress when the fluid starts to behave nonlinearly when stress is plotted against the shear rate. The dynamic viscosity is denoted by $\mu$ and the rate of stain tensor $e_{ij}$ is defined as:

$$e_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$$ (2.4)

Where the velocity components are denoted by $u$. 
In Fig 2.2, a Newtonian model is compared with the Bair-Winer limiting stress model. The almost flat region in the Bair-Winer model occurs when the shear stress ceases to respond to increasing shear rate.

![Graph showing shear stress vs. shear rate for Bair and Winer and Newtonian models.]

*Figure 2.2. Bair-Winer limiting stress model compared with a Newtonian model.*

The physics of the flow in an EHD-contact, with a characteristic time for the flow of the same order of magnitude as the relaxation time, suggests however, that elastic effects can be of importance. If these effects are included, the model becomes a visco-elasto-plastic model.

In this work the loads and stresses are, however, in the region associated with Newtonian rheology which will thus be used in these first attempts to use CFD for simulating EHL contacts.
3 MATHEMATICAL FORMULATION

The governing equations used in this work are presented below. No attempt is made to derive these equations, but rather to present them and discuss their properties. The flow is assumed to be laminar and therefore only laminar equations are considered.

3.1 Governing equations

The equations are written here in a co-ordinate free notation. The equations 3.1-7 can be seen in CFX User Guide, see CFX [1].

The momentum equation is given by:

\[
\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = \mathbf{B} + \nabla \cdot \sigma
\]  

(3.1)

The velocity field is denoted by \( \mathbf{u} \) and the body force \( \mathbf{B} \) is assumed to be zero. The symbol \( \otimes \) denotes the vector product and \( \rho \) denotes the density.

The total stress tensor, \( \sigma \), for a Newtonian fluid is given by:

\[
\sigma = -p\delta + (\zeta - \frac{2\mu}{3})\nabla \cdot \mathbf{u} \delta + \mu\left(\nabla \mathbf{u} + (\nabla \mathbf{u})^T\right)
\]  

(3.2)

The bulk viscosity is represented by \( \zeta \) and is assumed to be zero and the hydrostatic pressure is denoted by \( p \) and the unit tensor by \( \delta \).

A combination of equation (3.1) and expression (3.2) results in the momentum equation used in this work:

\[
\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla p + \nabla \cdot \mu\left(\nabla \mathbf{u} + (\nabla \mathbf{u})^T\right)
\]  

(3.3)

The equation of continuity:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0
\]  

(3.4)

and the equation of energy:

\[
\frac{\partial (\rho H)}{\partial t} + \nabla \cdot (\rho \mathbf{u} H) - \nabla \cdot (k \nabla T) = \frac{\partial p}{\partial t} + \phi
\]  

(3.5)

Where \( H \) is the total enthalpy and \( T \), temperature, the viscous dissipation is denoted by \( \phi \) and \( k \) is the thermal conductivity.
Expression (3.6) and (3.7) give the dissipation and total enthalpy:

\[
\phi = \nabla \cdot \left( \mu \left( \nabla u + (\nabla u)^T - \frac{2}{3} \nabla \cdot u \delta \right) u \right) \tag{3.6}
\]

\[
H = C_v T + \frac{p}{\rho} + \frac{1}{2} u^2 \tag{3.7}
\]

Where \( C_v \) denote the specific heat at constant volume.

The expression for the elastic deformations in line contacts are given by:

\[
d(x) = \frac{2(1-\nu)}{\pi E} \int_{-\infty}^{\infty} p(x') \ln \left| \frac{x-x'}{x_0} \right| dx' \tag{3.8}
\]

Where \( d(x) \) is the deformation and \( x_0 \) distance where the deformation is zero. Poisson’s ratio is denoted by \( \nu \) and \( E \) is the modulus of elasticity, see Hamrock [2].

The expression used for the viscosity is the Roelands’s expression: see Roelands [6].

\[
\mu(p,T) = \mu_0 \exp \left\{ \left[ \ln(\mu_0) + 9.67 \right] - 1 + \left( 1 + \frac{p}{P_0} \right)^{c} \left( \frac{T-138.0}{T_{in}-138.0} \right)^{3.0} \right\} \tag{3.9}
\]

The viscosity at ambient pressure and temperature is denoted by \( \mu_0 \) and \( T_{in} \) is the inlet temperature. \( P_0 \) is a constant and \( S_0 \) and \( z \) are thermo-viscous and pressure-viscosity parameters.

The density is modelled by the Dowson-Higginson’s expression, see Dowson [7] with a thermal extension:

\[
\rho(p,T) = \rho_0 \left( \frac{C_1 + C_2 p}{C_1 + p} \right) \left[ 1 - \epsilon(T-T_{in}) \right] \tag{3.10}
\]

Where the constants \( C_1 = 5.9 \times 10^8 \) and \( C_2 = 1.34 \). The density at ambient pressure and temperature is denoted by \( \rho_0 \) and coefficient of thermal expansivity coefficient is denoted by \( \epsilon \).
3.2 The singularity

When the viscosity is pressure dependent, a singularity may exist in the momentum equation. In order to show the presence of the singularity (which was first shown by Renardy [8]), a simplified analytical analysis of the momentum equations was made and the following simplifications introduced:

- No time dependency
- No body forces
- No compressibility
- Isothermal conditions

Most of the viscosity expressions concerning EHL have an exponential dependence on pressure. The model for the viscosity used in the analytical investigation is therefore:

$$\mu = \mu_0 e^{f(p)} \quad (3.11)$$

Where $f$ is a function of pressure, e.g. the exponent $\alpha p$ ($\alpha$ is the pressure-viscosity index) in the Barus expression, see Hamrock [2], or, the exponent in the Roelands’s expression, see expression (3.9) (with no dependency in temperature).

An order of magnitude analysis of the momentum equation (3.3) in addition to the assumptions above gives the following momentum equation for the high-pressure region of the contact:

$$\nabla p = \mu' \nabla p \cdot (\nabla u + (\nabla u)^T) + \mu \nabla^2 u \quad (3.12)$$

The derivative of viscosity with respect to pressure is here denoted by $\mu'$. An expansion of equation (3.12) into Cartesian component form and some algebra will give the following equations:

$$\frac{\partial p}{\partial x} = \frac{2\mu' e_{xy} \nabla^2 v + (1 - 2\mu' e_{xy}) \mu \nabla^2 u}{1 - (2\mu')^2 (e_{xy}^2 - e_{xx} e_{xy})} \quad (3.13)$$

The rate of deformation tensor, $e_{ij}$, is defined according to expression (2.4). Bair et. al. [9], describes the derivation of the equations in (3.13). As can be seen, the same denominator occurs in the both momentum equations in (3.13). There will be a singularity in the pressure gradients if the denominator approaches zero (and the nominator is not zero).

A rewriting of the denominator in (3.13) will give an expression for the principal shear stress when a singularity (singular shear stress) can occur in the momentum equations:

$$\tau = \frac{1}{f'(p, T)} \quad (3.14)$$
Where $\tau$ represents the singular shear stress and $f'$ denotes the derivative with respect to pressure in the exponent in expression (3.11).

The presence of the singularity is important for CFD simulations. If a time dependent problem is solved, passing through the singularity will result in exponentially growing oscillations in time and a rapid divergence will occur. In the case of steady state solutions, non-constant periodic solutions in space may exist. For a more detailed explanation see Renardy [8].

With the assumption of isothermal Newtonian rheology, this singularity can easily be reached. However, it is not thought to be possible to pass through the singularity since thermal effects and/or non-Newtonian rheology will prevent this.

Figure 3.1 shows a result of a simulation where the presence of the singularity can be observed.

![Figure 3.1](image)

**Figure 3.1. Navier-Stokes and Reynolds solutions for two different values of the pressure-viscosity coefficient, $\alpha$, in a converging gap geometry experiment. The solid line is the solution obtained by CFD-simulations and the dots are the solution given by the Reynolds equation approach.**

Two numerical simulations with different approaches have been carried out. The model geometry is a converging gap geometry with the lower surface given a constant velocity of 0.5 m/s whilst the upper surface was held fixed.

The solid line is the result from a numerical simulation using a N-S approach whilst the dotted line are the results from the Reynolds equation approach. The difference between the two sets of results is due to the singularity, see paper B.
4 NUMERICAL SIMULATIONS

The approach presented in this work uses the full equations i.e. the N-S equations (Newtonian rheology assumed). This section does not describe the whole numerical treatment, which is lengthy and described elsewhere, but rather presents a brief introduction to the FV-discretisation and the algorithm used for simulating EHL-problems numerically.

4.1 Discretisation

The equations of momentum, continuity and energy can be expressed in a general conservation law. The general form of the conservation law for a scalar quantity can be expressed as:

\[
\frac{\partial}{\partial t} \int_U dV + \int_{\partial U} F \cdot dS = \int Q dV
\]

(4.1)

The scalar quantity is here expressed as \( U \) whilst \( F \) is the flux vector and \( Q \) the volume sources. The discrete form of this equation can be expressed as:

\[
\frac{\partial}{\partial t} \left( U_j \Omega_j \right) + \sum_{sides} F \cdot dS = Q_j
\]

(4.2)

Where \( U_j \) and \( Q_j \) are the discrete variables for the quantity and volume sources and \( \Omega_j \) is the volume of the discrete control volume. In order to obtain a discrete formulation, CV-volumes and areas have to be computed, as well as sources and fluxes.

The software used in this work uses a body fitted grid where the computational nodes are placed in the centre of each CV. For a more complete description, see Hirsch [10].

4.2 Solution Algorithm

The full N-S equations were solved numerically by a commercial CFD-software. The solution algorithm is based on the scheme given in Fig. 4.1. The expressions for viscosity and density have to be implemented in the user routines USRVIS and USRDEN. To implement the deformations and displace the geometry so that the force balance will be fulfilled, the user routine USRGRD is used. A new grid has to be created during each outer iteration.

The code uses a finite volume discretisation of second order accuracy in the diffusive terms and upwind interpolation was selected for the convective terms. Because of the dominance of the diffusive terms in the contact region, the scheme is assumed to be of second order accuracy.

The pressure correction algorithm used was SIMPLE. The relaxation algorithm for the momentum equations was based on Stone’s method and for the pressure correction ICCG (conjugated gradient method) was used. For details of the user routines or numerical algorithm see CFX 4.3 [1].
Create an initial grid and set boundary conditions and initial values on pressure and temperature.

Computation of $u$ and $v$-velocity.

Computation of pressure.

Computation of viscosity and density.

Computation of temperature.

Convergence check on mass and $u$, $v$-momentum residuals.

If convergence

Yes

Quit computations and post processing

No

Compute force balance and deformation equations

Update the grid and iteration matrices

**Figure 4.1. Solution scheme.**

The solution algorithm used by the code is sequential, which can lead to instabilities if it is attempted to reach a solution too quickly. This demands very small under-relaxation factors and the result is a very slow convergence rate. The meshes used in the simulations were structured non-uniform meshes with a higher mesh density in the high-pressure region.

The computations were assumed to be converged when the mass residual $m_{res} < 10^{-4}$ of the initial mass residual.

Figure 4.2 shows the result of a numerical simulation of the temperature distribution in a line contact. The surface velocities of the upper and lower boundary are 0.6 and 0.1 m/s respectively and the radius of the roller is 10 mm. The load applied on the roller is 150 kN.
Temperature distribution [K]

Figure 4.2. Temperature distribution from a line contact simulation where the surface velocities of the upper and lower boundary are 0.6 and 0.1 m/s respectively. The radius of the roller is 10 mm and the applied load 150 kN.

The flow in the contact is here from left to right. The geometry is much magnified in the film thickness direction. The dimensions in the streamwise direction and minimum distance across the contact are approximately 0.5 mm and 0.1 μm respectively.

4.3 Error analysis

Error analysis is extremely important in all numerical simulations. Two types of error should always be estimated when checking the validity of a numerical simulation; the convergence error and the discretisation error.

These error are defined as:

- Convergence error: The difference between the exact and the iterative solution of the algebraic system of equations.

- Discretisation error: The difference between the exact solution and the exact solution of the algebraic system equations.

The error and residual are related by:

$$Ae^n = p^n$$

(4.3)

Where $p^n$, $e^n$ is the residual and error after $n$ iterations.
The convergence error is proportional to the residuals, thus if a zero initial guess is used, the error is the same as the converged solution itself. This means that a reduction of the residual by four orders of magnitude results in approximately the same reduction in the convergence error. A reduction of four orders of magnitude gives an error of approximately 0.01% of the solution, see Ferziger and Peric [11].

The discretisation error, denoted by \( \varepsilon_h \) (error in the mesh), is estimated as:

\[
\varepsilon_h = \frac{\varphi_h - \varphi_{2h}}{1 - 2^r}
\]

(4.4)

Where \( \varphi_h \), \( \varphi_{2h} \) are the solutions of the finer and coarser mesh respectively and \( r \) is the order of the discretisation. The diffusive terms are of second order accuracy and the convective terms are of first order; although the dominance of the terms in the central region of the contact implies that the order of scheme is of second order accuracy.

In this work the mean discretisation error \( \varepsilon_{hm} \) was computed by:

\[
\varepsilon_{hm} = \frac{1}{3N} |\varphi_{h,i} - \varphi_{2h,i}|
\]

(4.5)

The number of discrete values in the coarser mesh is denoted by \( N \).

However, it is not the formal order of the scheme which is of importance, but rather the reduction of error when the mesh is refined.

4.4 Cavitation

One of the crucial points of a CFD-simulation is that pressure is a result from other equations, i.e., equations of momentum and continuity and an equation of state if the flow is compressible.

In the exit region of the line contact, cavitation will occur. One way of simulating this in the N-S approach is to modify the density. The method used in this work is to model the density using the Dowson-Higginson’s expression, see (3.10), when the pressure is above a specified cavitation pressure \( P_{cav} \). When the pressure falls below \( P_{cav} \), a second order polynomial was used to interpolate the density down to zero. With this approach the density will decrease in the exit region of the contact where the surfaces diverge. If not modifying the density, large negative pressures occurs, which of course, is not physically correct.

Figure 4.3 shows the density distribution obtained from the numerical simulation. According to the figure. The pressure would be negative in the exit region of the contact if not modifying the density. Large negative pressures are of course not physical correct since cavitation will occur before.
Density distribution [kg m⁻³]

Figure 4.3. Density distribution in a line contact simulation, where the surface velocities of the upper and lower boundary are 0.6 and 0.1 m/s respectively. The radius of the roller is 10 mm and the applied load 150 kN.

Using this technique no modifications to the equations have to be made in order to satisfy continuity. The pressure is, in fact, a result of continuity.
5 SUMMARY OF PAPERS AND CONCLUDING REMARKS

The work in the thesis comprises three papers A, B and C. The first two papers present investigations into the possibilities of using the N-S equations for simulating lubrication in the EHL-regime, and to investigate whether it is possible to use a commercial software, CFX 4.3 [1].

In the numerical simulations a smooth line contact geometry and a converging gap geometry were used (line contact geometry see Fig. 2.1).

The following conclusions from the investigations can be made:

- It is possible to use the N-S approach and a commercial software for simulating thermal EHL-contacts when the geometries are smooth. Thus far, pressures up to approximately 1 GPa have been simulated, see Fig 5.1. All deformations have here been added to the upper surface; the lower surface being considered rigid.
- The computational cost is very high. The code uses a sequential solution algorithm, which easily leads to instabilities if it is attempted to reach a solution too quickly, see Fig 4.1. That leads to very small under-relaxation factors and the result is a slow convergence rate.
- The approach used to model cavitation, modifying the density, works well. With such an approach, continuity will be enforced at the same time as the negative pressures disappear, see Fig 4.3.
- There may be differences between the Reynolds equation and the N-S approach due to a singularity in the momentum equation. This is seen in the converging gap simulations, see Fig 3.1. No influence of the singularity has so far been observed in the line contact simulations.
- If passing through the singularity the computations will fail.

Figure 5.1. Pressure and film thickness. The upper graph with a small peak represents the pressure distribution and the lower graph with a flat central region is the film thickness.
As can be seen in Fig 5.1, no negative pressures occur in the exit region of the contact due to the way in which cavitation has been handled.

The last paper in the thesis, Paper C, presents a numerical simulation of a pivoted thrust bearing used to verify a thermo-hydrodynamic lubrication model (THD). The numerical simulation was compared with experimentally measured data of the bearing. The numerical model used was based on the Reynolds equation for the lubricating flow and the equation of energy for the oil film temperature.

Correlation between the theoretical and experimental results was encouraging and it may be concluded that agreement is quite acceptable despite the fact that elastic and thermal deformations of the bearing components were not taken into account. However, for good results to be obtained, the following conditions had to be met:

- Adjustment of secondary parameters such as hot oil carries over layer thickness and heat transfer coefficients according to experimental results.
- An accurate description of lubricant property parameters was necessary.

Good agreement between theoretical and experimental pressure profiles was obtained. The small discrepancies observed can be attributed to the elastic and thermal deformations of the runner and pads. Power loss is slightly overestimated even if the model is adjusted to more closely reflect measured temperatures.

The theoretical model has been shown to be accurate enough to be used for simulations of thrust bearings operating in the laminar regime. Even if the absolute magnitude of power loss, temperatures etc. cannot be determined exactly for general cases, the model is very useful for investigations concerning how bearing operation is influenced by lubricant properties, geometry parameters and operating conditions.
6 FUTURE WORK

The geometries used in the simulations are greatly simplified. The real surface of a bearing is rough. A not untypical surface roughness of ~ 1 µm is large compared to an EHL film thickness of ~ 0.1 µm and may have a significant influence on the behaviour and performance of the contact.

The assumption of Newtonian rheology is probably not valid when the contact pressures exceed 1 GPa, and unrealistic values of friction in the contacts, compared with experimental data are seen.

Future work must therefore include the following topics:

- Simulations with rough surfaces.
- Time dependent simulations.
- Implementation of a more realistic rheological model, which includes viscous, plastic and elastic behaviour (generalised Maxwell model).
- Simulations covering an extended computational domain (expansion of the computational domain to include more of both the inlet and outlet regions).
- 3D-simulations.

The last point requires more powerful computing resources since it is not possible to perform 3D simulations on a single workstation.
Nomenclature

\( u \)  Velocity field, components \((u,v,w)\)  \([\text{m s}^{-1}]\)

\( t \)  Time  \([\text{s}]\)

\( x, y, z \)  Spatial co-ordinates  \([\text{m}]\)

\( \rho \)  Density  \([\text{kg m}^{-3}]\)

\( \rho_0 \)  Density at ambient conditions

\( \text{B} \)  Body force  \([\text{N m}^{-3}]\)

\( \varepsilon \)  Coefficient of thermal expansion  \([\text{K}^{-1}]\)

\( \sigma \)  Total stress tensor

\( p \)  Pressure

\( \mu \)  Dynamic viscosity

\( \mu_0 \)  Dynamic viscosity at ambient conditions

\( \mu' \)  Derivative of viscosity with respect to pressure  \([\text{s}]\)

\( \mu^* \)  Equivalent viscosity

\( \varepsilon \)  Bulk viscosity

\( \alpha \)  The pressure viscosity index

\( H \)  Total enthalpy  \([\text{J kg}^{-1}]\)

\( T \)  Temperature  \([\text{K}]\)

\( T_{in} \)  Inlet temperature  \([\text{K}]\)

\( k \)  Thermal conductivity  \([\text{W m}^{-1} \text{K}^{-1}]\)

\( E \)  Modulus of elasticity

\( x_0 \)  Distance to a position where the deformation is zero  \([\text{m}]\)

\( \varepsilon_{ij} \)  Rate of deformation tensor  \([\text{s}^{-1}]\)

\( \gamma^* \)  Shear rate  \([\text{s}^{-1}]\)

\( \tau \)  Singular shear stress

\( \tau_0 \)  Shear stress in the Ree-Eyring model

\( C_v \)  Specific heat at constant volume  \([\text{J kg}^{-1} \text{K}^{-1}]\)

\( P_{cav} \)  Cavitation pressure  \([\text{Pa}]\)

\( \phi \)  Viscous dissipation  \([\text{J m}^{-3} \text{s}^{-1}]\)

\( d \)  Deformation  \([\text{m}]\)

\( S \)  Surface area  \([\text{m}^{2}]\)

\( V \)  Volume  \([\text{m}^{3}]\)

\( C_1 \)  Constant in the Dowson-Higginson's density expression  \([\text{Pa}]\)

\( v \)  Poisson's ratio  [-]

\( \varepsilon_h \)  Discretisation error  [-]

\( \varepsilon_{hm} \)  Mean discretisation error  [-]

\( \varepsilon^* \)  Error after \( n \) iterations  [-]

\( \rho^* \)  Residual after \( n \) iterations  [-]

\( f \)  Function in viscosity expression  [-]

\( f' \)  Derivative with respect to pressure  [-]

\( N \)  Number of discrete values in the coarser mesh  [-]

\( \varphi_h \)  Solution variable, finer mesh  [-]

\( \varphi_{hm} \)  Solution variable, coarser mesh  [-]

\( P_0 \)  Constant in the Roelands's viscosity expression  [-]

\( S_0 \)  Temperature-viscosity index in the Roelands's viscosity expression  [-]

\( z \)  Pressure-viscosity index in the Roelands's viscosity expression  [-]

\( C_2 \)  Constant in the Dowson-Higginson's density expression  [-]
### Scalar variable
- $U$  
- $F$  
- $Q$  
- $r$

#### Flux vector

#### Volume sources

#### Order of the numerical scheme

#### Discrete variables

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\varphi_{h,i}$</td>
<td>Solution variable, finer mesh</td>
<td>[-]</td>
</tr>
<tr>
<td>$\varphi_{2h,i}$</td>
<td>Solution variable, coarser mesh</td>
<td>[-]</td>
</tr>
<tr>
<td>$U_j$</td>
<td>Scalar variable</td>
<td>[-]</td>
</tr>
<tr>
<td>$\Omega_j$</td>
<td>Control volume</td>
<td>$[m^3]$</td>
</tr>
<tr>
<td>$Q_j$</td>
<td>Volume sources</td>
<td>[-]</td>
</tr>
</tbody>
</table>

#### Mathematical symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\otimes$</td>
<td>Vector product $(u_i, u_j)$</td>
</tr>
<tr>
<td>$\bullet$</td>
<td>Scalar product</td>
</tr>
<tr>
<td>$\nabla$</td>
<td>Gradient operator $(\partial/\partial x, \partial/\partial y, \partial/\partial y)$</td>
</tr>
<tr>
<td>$\nabla^2$</td>
<td>Laplacian operator $(\nabla \bullet \nabla)$</td>
</tr>
<tr>
<td>$\delta$</td>
<td>Unit tensor</td>
</tr>
<tr>
<td>$(\cdot)^T$</td>
<td>Transpose</td>
</tr>
<tr>
<td>$</td>
<td></td>
</tr>
</tbody>
</table>
References


THE NAVIER-STOKES APPROACH FOR THERMAL EHL LINE CONTACT SOLUTIONS.

T. Almqvist *, R. Larsson
Department of Mechanical Engineering,
Division of Machine Elements,
Luleå University of Technology,
SE-971 87 Luleå, Sweden.
* Corresponding author. Fax: +46-920-91047
E-mail: torbjorn.almqvist@mt.luth.se

ABSTRACT

The complicated nature of the EHL-problem has so far forced researchers to develop their own computer codes. These codes are ultimately based on the Reynolds equation, and if thermal EHL-simulations are required, a simultaneous solution of the equation of energy also has to be performed. To date only a few attempts to solve the full equations of momentum and continuity as well as equations of energy have been performed. However, such an approach will give extended possibilities of simulating EHL-contacts; i.e. the computational domain can be expanded and it will be possible to simulate the flow, not only in the contact but also around the contact. Another possibility is to investigate how the altering length scales of the surface roughness influence the behaviour of the flow in the contact. However, the aim of the work presented in this paper is to investigate the possibilities of using a commercial CFD-code (computational fluid dynamics code) based on the above-mentioned equations for simulating thermal EHL. The rheology is assumed to be Newtonian and the equations of momentum and continuity are then commonly referred to as the Navier-Stokes equations (N-S equations). The geometry chosen for the simulations is a smooth line contact geometry, for which the results from the simulations show that it is possible to use the N-S equations for thermal EHL for contact pressures up to approximately 1.0 GPa. The code used in this work is the commercial CFD software CFX 4.3 [1]. There is a limitation in the N-S approach due to a singularity that can occur in the equation of momentum when the principal shear stresses in the film become too high. However, a thermal approach makes it possible to simulate EHL-contacts at higher loads compared with an isothermal approach, due to the reduction of the viscosity in the former approach. The singularity is not present in the Reynolds approach.

Keywords: Elastohydrodynamic lubrication, Fluid mechanics, Cavitation
## NOMENCLATURE

<table>
<thead>
<tr>
<th>Parameters/ Variables</th>
<th>Meaning</th>
<th>Dimension</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u$</td>
<td>Velocity field</td>
<td>$m , s^{-1}$</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
<td>$kg , m^{-3}$</td>
</tr>
<tr>
<td>$T$</td>
<td>Time</td>
<td>$s$</td>
</tr>
<tr>
<td>$B$</td>
<td>Body force</td>
<td>$N , m^{-3}$</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>Thermal expansivity coefficient</td>
<td>$K^{-1}$</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Total stress tensor</td>
<td>$Pa$</td>
</tr>
<tr>
<td>$P$</td>
<td>Pressure</td>
<td>$Pa$</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Dynamic viscosity</td>
<td>$Pa , s$</td>
</tr>
<tr>
<td>$\zeta$</td>
<td>Bulk viscosity</td>
<td>$Pa , s$</td>
</tr>
<tr>
<td>$H$</td>
<td>Total enthalpy</td>
<td>$J , kg^{-1}$</td>
</tr>
<tr>
<td>$T$</td>
<td>Temperature</td>
<td>$K$</td>
</tr>
<tr>
<td>$k$</td>
<td>Thermal conductivity</td>
<td>$W , m^{-1} , K^{-1}$</td>
</tr>
<tr>
<td>$\tilde{H}$</td>
<td>Thermodynamic enthalpy</td>
<td>$J , kg^{-1}$</td>
</tr>
<tr>
<td>$H$</td>
<td>Total enthalpy</td>
<td>$J , kg^{-1}$</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Poisson's ratio</td>
<td>-</td>
</tr>
<tr>
<td>$E$</td>
<td>Modulus of elasticity</td>
<td>$Pa$</td>
</tr>
<tr>
<td>$x_0$</td>
<td>Distance to a position where the deformation is zero</td>
<td>$m$</td>
</tr>
<tr>
<td>$\varepsilon_{ij}$</td>
<td>Rate of deformation tensor</td>
<td>$s^{-1}$</td>
</tr>
<tr>
<td>$\tau$</td>
<td>Singular shear stress</td>
<td>$Pa$</td>
</tr>
<tr>
<td>$\Delta$</td>
<td>Discretisation error</td>
<td>-</td>
</tr>
<tr>
<td>$Q$</td>
<td>Heat flux</td>
<td>$W , m^{-2}$</td>
</tr>
<tr>
<td>$C$</td>
<td>Specific heat</td>
<td>$J , kg^{-1} , K^{-1}$</td>
</tr>
<tr>
<td>$U$</td>
<td>Surface velocity</td>
<td>$m , s^{-1}$</td>
</tr>
<tr>
<td>$P_{cav}$</td>
<td>Cavitation pressure</td>
<td>$Pa$</td>
</tr>
<tr>
<td>$\phi$</td>
<td>Viscous dissipation</td>
<td>$J , m^{-3} , s^{-1}$</td>
</tr>
<tr>
<td>$F$</td>
<td>Function of pressure and temperature</td>
<td>-</td>
</tr>
<tr>
<td>$D$</td>
<td>Deformation</td>
<td>$m$</td>
</tr>
<tr>
<td>$N$</td>
<td>Number of control volumes, coarser mesh</td>
<td>-</td>
</tr>
<tr>
<td>$\varphi$</td>
<td>Solution variable</td>
<td>-</td>
</tr>
</tbody>
</table>

### Sub- and superscript
- $'$: Derivative with respect to pressure
- $h$: Finer mesh
- $2h$: Coarser mesh

## 1 INTRODUCTION

The usual way of computing the fluid flow and temperature in EHL-contacts is to solve the Reynolds equation and the equation of energy. The former is a PDE derived from the equations of momentum and continuity by utilizing the thin film approximation, and the latter is derived from the first law of thermodynamics.
The approach used in this work involves the full equations of momentum, continuity and energy, with the assumption of a Newtonian rheology. The equations of momentum and continuity are commonly referred to as the Navier-Stokes (N-S) equations. The approach gives extended possibilities of simulating EHL. Some of these possibilities are:

- Expanded computational domain at both inlet and outlet.
- Simulations of particle transport in a bearing.
- Influence of the altering length scales in the contact.

Solving EHL problems on the basis of N-S equations offers the possibility of expanding the computational domain in both the inlet and the outlet regions of the contact; i.e. no thin film approximations have to be performed and the inertia terms are also retained. The replenishment of the contact may, for example, be investigated. The extended approach also provides the possibility of performing particle transport simulations; i.e. it will be possible to determine whether particles enter the contact region or are flushed sideways in the inlet. Another possibility is that of investigating if the scales of the asperities in the contact are so different that the thin film approximation is questionable.

The scope of this work, however, does not include an investigation of the above-listed effects, which are specified to justify the extended N-S approach.

Bair et al. [2] have pointed out that the validity of the Reynolds equation is questionable in the high-pressure region (Hertzian region) of the contact. They showed by using the incompressible inertia less momentum equations that a singularity might be present in the momentum equation under normal EHL-conditions. This singularity (as will be explained in greater detail later) may cause a change in the pressure-gradient along the contact, as well as create a gradient across the oil film. These singularity phenomena cannot be predicted by the Reynolds equation.

Schäfer et al. [3] showed by solving the Stokes equations, i.e. the N-S equation with the inertia terms neglected, that there can be pressure variations across the oil film in the order of several MPa when a high degree of slip is introduced in a smooth EHL line contact. In the work performed by Schäfer et al. a pre-pressure build-up occurred before the normal pressure spike. However, they used a Newtonian fluid model and assumed isothermal conditions, which caused unrealistically high shear stresses in the lubricant film.

A complication of the N-S approach is that pressure is not an independent equation; i.e. the pressure-gradients will contribute to the three momentum equations. The way of dealing with this problem is to construct a pressure field so that continuity is enforced, see Ferziger and Peric [4]. Therefore, the approach of solving the N-S equations for the contact problem is a computer-intensive task. Instead of solving one equation for the pressure, as in a 3D case for the Reynolds approach, one is forced to solve three equations for the velocity and one equation for the pressure correction, see Ferziger and Peric [4].

In the N-S approach we cannot use the method of letting the negative pressure be equal to zero, as is done in the Reynolds approach.

The way of dealing with the problem in CFD (computational fluid dynamics) is to modify the density, and the advantage of such an approach is that mass conservation is enforced at the outlet or between asperities in the contact region where the surfaces diverge.

The aim of this paper is to investigate the possibilities of solving a thermal compressible EHL smooth line contact problem with the commercial CFD-software CFX4.3 [1]. The analysis and the numerical models in this work are limited to 2D cases.
2 METHOD

The governing equations and their numerical solutions will be described here. The treatment of the cavitation in the outlet is also described.

2.1 Governing equations

The governing equations for the full approach are the equations of momentum and continuity and the equation of energy and these equations read as follows:

The equation of continuity

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0
\]  

(1)

The density is here denoted by \( \rho \) and \( \mathbf{u} \) is the velocity field. Time is denoted by \( t \).

The equation of momentum:

\[
\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = \mathbf{B} + \nabla \cdot \mathbf{\sigma}
\]  

(2)

The body force is denoted by \( \mathbf{B} \), \( \mathbf{\sigma} \) is the total stress tensor and the symbol \( \otimes \) denotes here the vector product.

The constitutive equation for the total stress tensor for a Newtonian fluid is:

\[
\mathbf{\sigma} = -p \delta + (\zeta - \frac{2\mu}{3}) \nabla \cdot \mathbf{u} \delta + \mu \left( \nabla \mathbf{u} + \left( \nabla \mathbf{u} \right)^T \right)
\]  

(3)

where the bulk viscosity \( \zeta \) is neglected with the aid of the Stokes assumption, the dynamic viscosity is denoted by \( \mu \) and the pressure is denoted by \( p \). \( \delta \) denotes the unit tensor.

The energy equation expressed in total enthalpy \( H \), is:

\[
\frac{\partial (\rho H)}{\partial t} + \nabla \cdot (\rho \mathbf{u} H) - \nabla \cdot (k \nabla T) = \frac{\partial p}{\partial t} + \phi
\]  

(4)

\( T \) and \( \phi \) denote the viscous dissipation and the temperature, and \( k \) is the thermal conductivity.

The dissipation and total enthalpy are:

\[
\phi = \nabla \cdot \left( \mu \left( \nabla \mathbf{u} + \left( \nabla \mathbf{u} \right)^T - \frac{2}{3} \nabla \cdot \mathbf{u} \delta \right) \right)
\]  

(5)
The thermodynamic enthalpy is here denoted by $h$ and the specific heat at a constant volume by $C_v$.

Most of the viscosity expressions have approximately exponential growth, e.g. Barus and Roelands's relationships, see Hamrock [5], so that the model used for the analytical approach is an exponential expression, where $f(p, T)$ is a function of the pressure and temperature.

$$\mu = \mu_0 e^{f(p, T)}$$  \hspace{1cm} (7)

where $\mu_0$ denotes the viscosity at an ambient pressure and temperature.

The Boussinesq expression is used for the elastic deformations in the contact:

$$d(x) = \frac{2(1 - \nu)}{\pi E} \int_{-\infty}^{\infty} p(x') \ln \left| \frac{x - x'}{x_0} \right| dx'$$  \hspace{1cm} (8)

where $d(x)$ is the deformation and $x_0$ the distance to a position where the deformation is zero. The Poisson's ratio is denoted by $\nu$ and $E$ denotes the modulus of elasticity.

### 2.2 The singularity

In order to show the presence of the singularity (which was first shown by Renardy [6]), a simplified analytical analysis of the momentum equations was performed. Some simplifications were introduced.

- No body forces
- No compressibility
- No time-dependency
- Isothermal conditions

An order-of-magnitude analysis of the momentum equation (2) in addition to the assumptions above will now give the following momentum equation for the high-pressure region of the contact:

$$\nabla p = \mu' \nabla p \cdot \left( \nabla u + (\nabla u)^T \right) + \mu \nabla^2 u$$  \hspace{1cm} (9)

The derivative of viscosity with respect to pressure is here denoted by $\mu'$. An expansion of equation (9) into the Cartesian component form and some algebra will give the following equations:
The rate of deformation tensor, $e_{ij}$, is defined as:

$$e_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$$  \hspace{1cm} (11)$$

The steps of the derivation of these equations are described by Bair et al. [2]. As can be seen, the same denominator will occur in the two momentum equations in equations (10). There will be a singularity in the pressure gradients if the denominator approaches zero and the nominator is different from zero.

A rewriting of the denominator in equations (10) will give an expression for the principal shear stress where a singularity (singular shear stress) can occur in the momentum equations and reads as:

$$\tau = \frac{1}{\dot{f}(p,T)}$$ \hspace{1cm} (12)$$

Where $\tau$ represents the shear stress where a singularity may occur (singular shear stress) and $\dot{f}$ denotes the derivative with respect to pressure in the exponent in expression (7). In expression (12) $T$ is assumed to be constant.

The presence of the singularity will play an important role in the CFD simulations. If a time-dependent problem is solved, passing through the singularity will result in exponential growing oscillations in time and a rapid divergence will occur. In the case of steady state solutions, non-constant periodic solutions in space may exist. For a more detailed explanation see Renardy [6].

This means that, it is not possible to pass through the singularity and obtain realistic solutions. Shäfer et al. [3] did pass through the singularity and, since the N-S equations are not valid under such conditions, it is likely that their unexpected pressure distribution is erroneous. Thermal equations have to be solved simultaneously, and possibly with the assumption of a non-Newtonian rheology in order to obtain solutions at higher loads.

### 2.3 Numerical solution

The full N-S equations were solved numerically and the inertia and compressibility terms were retained. The N-S equations were solved with the commercial CFD-software CFX4.3 [1], and the solution algorithm is according to the scheme in Fig. 1.
Create an initial grid and set boundary conditions and initial values for pressure and temperature.

Computation of u and v-velocity.

Computation of pressure.

Computation of viscosity and density.

Computation of temperature.

Convergence check on mass and u, v-momentum residuals.

If convergence

Yes

No

Quit computations and post-processing

Compute force balance and deformation equations

Update the grid and iteration matrices

Figure 1. Solution scheme.

The expressions for the viscosity and density have to be implemented in the user routines USRVIS and USRDEN, where these variables are available. To implement the deformations and to displace the geometry so that a force balance will be achieved, the user routine USRGRD is used, where a new grid has to be created in each outer iteration. For details of the user routines, see CFX4.3 [1].

The solution algorithm used by the code is sequential, which easily leads to instabilities if one tries to reach the solution too fast. That leads to large under-relaxations and a very slow convergence rate. The code uses a finite volume discretisation of second order accuracy in the diffusive terms and upwind interpolation for the convective terms. Because of the dominance of the diffusive terms in the contact region, the scheme is assumed to be of second order accuracy.
The pressure correction algorithm was SIMPLE. The relaxation algorithm for the momentum equations was Stone’s method and for the pressure correction ICCG (conjugated gradient method), see CFX4.3 [1].

In the simulations, the Roeland and Dowson-Higginson expressions were used for the viscosity and density, see Hamrock [5].

The meshes used in the simulations were structured non-uniform meshes with a higher mesh density in the high-pressure region. The discretisation error $\Delta$ is computed by:

$$\Delta = \frac{1}{3N} \sum_{i}^{N} |\varphi_{h,i} - \varphi_{2h,i}|$$

where $\varphi_{h}$ and $\varphi_{2h}$ are the solutions of the finer and coarser meshes respectively. $N$ denotes the number of values in the coarser mesh.

2.4 Boundary conditions

The boundary conditions used for the hydrodynamic equations are:

- Specified pressure at both the inlet and the outlet (atmospheric pressure).
- No-slip boundary conditions at the walls (the fluid assumed to be attached to the walls).

The boundary conditions for the thermal computations are:

- Specified temperature at inlet.
- At outlet, the temperature is extrapolated from upstream values.
- At the surface, asymptotic solution of the Carslaw-Jaeger boundary condition is used, see Cheng and Sternlicht [7].

When the surface velocities are in the same direction and in movement, these boundary conditions read as follows:

$$T(x, y) = T_0 + \frac{1}{\sqrt{\pi p_k C U}} \int_{-\infty}^{x} \frac{q(x', y)}{\sqrt{x - x'}} dx'$$

where $T_0$ denotes the ambient temperature of the solids and $q$ is the heat flux across the surface in the normal direction. $U$ and $C$ denote the surface velocity and the specific heat, and in this work the ambient temperature is chosen to be the same as the inlet temperature $T_{in}$.

2.5 Cavitation treatment

One of the crucial steps in a CFD-solution for the EHL-contact problem is the calculation of the pressure. The pressure is computed from the equations of momentum and continuity and an equation of state if the flow is compressible. In the exit region of the line contact, cavitation will occur when the surfaces diverge. In the N-S approach it is not possible to use the cavitation boundary conditions used in the Reynolds equation approach.
One way of simulating the cavitation in the CFD-approach is to modify the density. The method used in this work is to model the density with the Dowson-Higginson expression, see Hamrock [5], when the pressure is above a specified cavitation pressure $p_{cav}$. When the pressure falls below $p_{cav}$, a second order polynomial is used to interpolate the density down to zero. The advantage of such a treatment is that no modifications of the equations have to be performed to satisfy continuity. The pressure is a result of continuity.

There is of course the possibility of replacing the second order polynomial with other choices of functional fits of the cavitation. However, the choice of fit will not affect the final solution much, due to the low pressure in the cavitation region compared with the pressure inside the contact (as long as negative pressures are not allowed and the gradients in the density are sufficiently smooth).

3 RESULTS

The geometry chosen for the simulations was a smooth line contact geometry, and the data for the geometry and other parameters used in the computations are contained in Table 1.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Meaning</th>
<th>Test case 1.</th>
<th>Test case 2.</th>
<th>Dimension</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\rho_0$</td>
<td>Density in ambient conditions</td>
<td>870</td>
<td>870</td>
<td>kg m$^{-3}$</td>
</tr>
<tr>
<td>$\mu_0$</td>
<td>Viscosity in ambient conditions</td>
<td>0.04</td>
<td>0.04</td>
<td>Pa s</td>
</tr>
<tr>
<td>$u_u$</td>
<td>Upper surface velocity</td>
<td>0.5</td>
<td>0.5</td>
<td>m s$^{-1}$</td>
</tr>
<tr>
<td>$u_d$</td>
<td>Lower surface velocity</td>
<td>0.5</td>
<td>0.6</td>
<td>m s$^{-1}$</td>
</tr>
<tr>
<td>$Z$</td>
<td>Pressure-viscosity index</td>
<td>0.5</td>
<td>0.5</td>
<td>-</td>
</tr>
<tr>
<td>Load</td>
<td>Applied outer load</td>
<td>100</td>
<td>150</td>
<td>kN</td>
</tr>
<tr>
<td>$K_l$</td>
<td>Thermal conductivity lubricant</td>
<td>0.15</td>
<td>0.15</td>
<td>W m$^{-1}$ K$^{-1}$</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>Thermal expansivity coefficient</td>
<td>6.5 $10^{-4}$</td>
<td>6.5 $10^{-4}$</td>
<td>K$^{-1}$</td>
</tr>
<tr>
<td>$S_0$</td>
<td>Thermoviscous parameter</td>
<td>1.1</td>
<td>1.1</td>
<td>-</td>
</tr>
<tr>
<td>$C_v$</td>
<td>Specific heat lubricant</td>
<td>2.19 $10^3$</td>
<td>2.19 $10^3$</td>
<td>J kg$^{-1}$ K$^{-1}$</td>
</tr>
<tr>
<td>$k_{su}$</td>
<td>Thermal conductivity upper solid</td>
<td>45</td>
<td>45</td>
<td>W m$^{-1}$ K$^{-1}$</td>
</tr>
<tr>
<td>$k_{sd}$</td>
<td>Thermal conductivity lower solid</td>
<td>45</td>
<td>1.0</td>
<td>W m$^{-1}$ K$^{-1}$</td>
</tr>
<tr>
<td>$C_s$</td>
<td>Specific heat in upper solid</td>
<td>460</td>
<td>460</td>
<td>J kg$^{-1}$ K$^{-1}$</td>
</tr>
<tr>
<td>$\rho_{su}$</td>
<td>Density upper solid</td>
<td>7.8 $10^3$</td>
<td>7.8 $10^3$</td>
<td>kg m$^{-3}$</td>
</tr>
<tr>
<td>$\rho_{sd}$</td>
<td>Density lower solid</td>
<td>7.8 $10^3$</td>
<td>7.8 $10^3$</td>
<td>kg m$^{-3}$</td>
</tr>
<tr>
<td>$R$</td>
<td>Radius of roller</td>
<td>0.01</td>
<td>0.01</td>
<td>m</td>
</tr>
<tr>
<td>$E$</td>
<td>Modulus of elasticity</td>
<td>2.06 $10^{11}$</td>
<td>2.06 $10^{11}$</td>
<td>Pa</td>
</tr>
<tr>
<td>$T_{in}$</td>
<td>Inlet temperature</td>
<td>288</td>
<td>288</td>
<td>K</td>
</tr>
<tr>
<td>$p_{amb}$</td>
<td>Ambient pressure</td>
<td>1.0 $10^5$</td>
<td>1.0 $10^5$</td>
<td>Pa</td>
</tr>
<tr>
<td>$p_{cav}$</td>
<td>Cavitation pressure</td>
<td>1.0 $10^5$</td>
<td>1.0 $10^5$</td>
<td>Pa</td>
</tr>
</tbody>
</table>

The mesh used in the simulations is a non-uniform structured mesh with 400 CV (control volumes) along the contact and 20 CV across the film. A crucial step in the computations is to perform a simulation without passing through the singularity.
In the first numerical experiment, a pure rolling case was simulated. The parameters used in the experiment are contained under case 1, see Table 1. The result of the simulation is shown in Fig. 2.

Figure 2. Film thickness and pressure distribution along the contact in the case of pure rolling. The upper and lower surface velocities are both 0.5 [m/s]. The thermal conductivity of the upper and lower surface is 45 [J kg\(^{-1}\) K\(^{-1}\)].

The temperature distribution is not shown here, and the maximum temperature rise in the contact is only 0.12 K above the inlet temperature. The convergence and discretisation errors in the computations are 1.7\times10\(^{-3}\)% and 6.0\times10\(^{-5}\)% respectively.

In the next experiment, slip was introduced in the contact. The surface velocities and other parameters used in the simulations are contained under case 2, see Table 1. The pressure and film thickness are shown in Fig. 3 and the result from the thermodynamic computations is shown in Fig. 4.
Figure 3. Film thickness and pressure distribution along the contact under sliding conditions. Upper and lower surface velocities $u_u = 0.1$ and $u_d = 0.6$ [m/s] respectively. The thermal conductivity of the upper surface is 45 and that of the lower surface 1.0 [J kg$^{-1}$ K$^{-1}$].

Figure 4. Isotherms in the contact during sliding. Upper and lower surface velocities $u_u = 0.1$ and $u_d = 0.6$ [m/s] respectively. The thermal conductivity of the upper surface is 45 and that of the lower surface 1.0 [J kg$^{-1}$ K$^{-1}$].

The maximum temperature rise in the contact is 28.7 K and occurs just below the centre of the film. The convergence and discretisation errors in the simulation are $1.0 \times 10^{-7}$% and $1.5 \times 10^{-3}$% respectively.
4    DISCUSSION

An interesting aspect of the momentum equations presented in equations (10) is the possibility of a singularity in the pressure gradient. As we can see from equation (12), this singularity can occur for stresses of approximately the same magnitude as those found in EHL; i.e. if isothermal conditions are assumed and the Barus viscosity expression is used with a pressure-viscosity coefficient of $\alpha = 2.0 \times 10^8$, the singular shear stress is 50.0 MPa. This magnitude of the shear stresses is easily reached in highly loaded contacts.

It is not believed that the singularity can be passed through, since non-Newtonian or thermal effects will prevent this. However, if the fluid can resist stresses in the neighbourhood of the singularity, the phase change process of the lubricant (or solidification process) must be much more rapid than that predicted by the Reynolds equation approach.

It is a complicated task to perform simulations near the singularity, because passing through the singularity will lead to a change of type in the momentum equations and the computations will fail. For a detailed explanation, see Renardy 1986 [6].

No singularity will be present in the Reynolds equation due to the assumptions made when deriving the equation. A simulation based on Reynolds equation and an isothermal Newtonian rheology, principal shear stresses will be produced that are high enough to force the momentum equation through the singularity. Bair et al. [2] suggest that the singularity in the momentum equation might be the source of the mechanical shear band seen in their experiments.

It is not believed, however, that the singularity affects the above solutions, since the maximum shear stress in the above slip experiment was 33 MPa and the singular shear stress was 148 MPa. The maximum shear stress in the above slip experiment was confined to the middle of the contact, where the velocity profile is Couette. That means that the nominator in equations (10) is close to zero and the effect of the singularity in the denominator will be damped.

If, however, the principal shear stress gets close to the singular shear stress where a Poiseuille flow component exist, the singularity will probably influence the flow. Such effects may occur in the case of rough surfaces or in more severe viscous heating.

In order to perform computations at higher loads on the basis of the N-S equations, there must be a limitation in the stresses produced by the fluid, so that passing through the singularity is prevented. The inclusion of thermal effects enables simulations at higher loads due to the decrease in viscosity with rising temperature. However, it is possible that a non-Newtonian rheology must be incorporated besides the thermal effects in order to reduce the shear stresses produced in the fluid when higher loads are required in the simulations.

The way chosen in this work to simulate the cavitation works well. It is not believed that an implementation of a more realistic cavitation model would change the solutions to a great extent.

5    CONCLUSIONS

The aim of this work was to investigate if it is possible to use the N-S equations in the solution of EHL-problems, and to investigate the possibilities of using the commercial software CFX 4.3 [1]. In the experiments a smooth line contact geometry was chosen as a model problem and the conclusions from these experiments are as follows:
• It is possible to use the N-S approach and to use a commercial code for solving thermal EHL problems when the geometries are smooth. So far, contact pressures up to approximately 0.7 GPa have been obtained.
• The computational cost is very high. The code uses a sequential solution algorithm, which easily leads to instabilities if one tries to reach the solution too fast. That leads to very small under-relaxation factors and a slow convergence rate.
• The approach used to model the cavitation with a modification of the density works well. With such an approach the continuity will be enforced at the same time as the negative pressures disappear.

ACKNOWLEDGEMENT

The authors gratefully acknowledge the financial support from the Swedish Research Council for Engineering Sciences (TFR) and The National Graduate School in Scientific Computing (NGSSC).

6 REFERENCES

COMPARISON OF REYNOLDS AND NAVIER-STOKES APPROACHES FOR SOLVING ISOTHERMAL EHL LINE CONTACTS.

T. Almqvist and R. Larsson
Division of Machine Elements, Luleå University of Technology, SE-971 87, Sweden,
E-mail: torbjorn.almqvist@mt.luth.se

SUMMARY
The Reynolds equation is the dominant approach for simulating the flow in EHL contacts. However, questions have been arisen as to the validity of the assumptions made when deriving the equation i.e. altering length scales. The complicated nature of the EHL problem has so far forced the researchers to develop their own codes, based on Reynolds equation for simulating EHL contacts. The aim of this work is firstly to investigate the possibilities of using a commercial CFD-code for simulating an isothermal EHL line contact based on the Navier-Stokes equations, and secondly to compare the result with a simulation performed using the Reynolds equation. The results show discrepancies between the two approaches when the flow in a simple converging gap geometry was simulated. No discrepancies were observed when a line contact was simulated. The Navier-Stokes approach exhibits a singularity in the equation of momentum, which is not present in the Reynolds equation. This singularity is a source of the discrepancies between the two approaches.

Keywords: CFD, Navier-Stokes, EHL, singularity, cavitation.

1 INTRODUCTION
The usual way of computing the fluid flow in lubricated contacts is to solve the Reynolds equation, which is a PDE derived from the equations of momentum and continuity. However, it is uncertain whether the assumptions made when deriving these equation are valid for running conditions where contact pressures are high, up to several GPa, and surface roughness is of concern.

Bair et al. [1] pointed out that the validity of the Reynolds equation is questionable in the high-pressure region (Hertzian region) of the contact. They showed using incompressible inertia-less momentum equations that a singularity might be present in the momentum equation under normal EHL conditions which may cause a change indicated in the pressure gradient along the contact as well as creating a gradient across the oil film. The behavior of the singularity cannot be predicted by the Reynolds equation.

Schäfer et al. [2] showed using the Stokes equations that pressure variations in the order of several MPa may exist across the oil film when a high degree of slip occurs in a line contact. In this work a pre-pressure build up was shown before the usual pressure spike.

Solving the N-S equations provide a more complete solution to the contact problem but is a computationally intensive task. For the 3D case, it is necessary to solve four equations to obtain the pressure in the contact, three equations for the velocity and one equation for the pressure correction (compared with one equation for the Reynolds approach).

As indicated earlier, an analytical investigation of the momentum equations, show possibilities of singularities in the pressure gradients at quite low shear stresses known for EHL problem.
These singularities are not represented in the Reynolds equation and are a potential source of errors. In addition, to the varying scales that might be present in contacts with rough surfaces.

The aim of this paper is firstly to investigate whether it is possible to solve an EHL problem with a commercial CFD-code CFX4.3 [3]. The second aim is to compare the Reynolds equation solutions with the solutions obtained by the N-S equations, and to see whether any discrepancies occur between the two approaches, which may be due to the singularity.

The analysis and the numerical models presented are limited to the 2D cases. The first geometry studied is a converging gap and the second a smooth surface line contact.

2 \hspace{0.5cm} METHOD

The governing equations and their numerical solution will be described. The treatment of cavitation in the outlet is also considered.

2.1 Governing equations

The governing equations for the full approach are the equations of momentum and continuity. For the case where the rheological behavior of the lubricant follows a Newtonian model, these equations are commonly called the N-S equations:

The equation of continuity is:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j) = 0$$

(1)

The equation of momentum is:

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_i u_j) = \frac{\partial}{\partial x_j}B_i + \frac{\partial}{\partial x_j}\sigma_{ij}$$

(2)

The density and velocity are denoted \( \rho \) and \( u \), time and spatial coordinates are given by \( t \) and \( x \) respectively. The bodyforce is denoted by \( B \) and \( \sigma \) is the total stress tensor.

The constitutive equation for the stress tensor is:

$$\sigma_{ij} = -p \delta_{ij} + (\zeta - \frac{2\eta}{3}) \frac{\partial u_k}{\partial x_k} \delta_{ij} + \eta \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$$

(3)

The bulk viscosity \( \zeta \) is neglected with aid of the Stokes assumption. The viscosity is denoted by \( \eta \) and pressure is denoted by \( p \). Most of the viscosity expressions have exponential growth so the model used has an exponential expression where \( f(p) \) is an arbitrary function of the pressure:

$$\eta = \eta_0 e^{f(p)}$$

(4)

The viscosity at ambient pressure is denoted by \( \eta_0 \). The Boussinesq expression is used for the deformations in the line contact:
\[ d(x) = \frac{2(1-\nu)}{\pi E} \int_{-\infty}^{\infty} p(x') \ln \left| \frac{x-x'}{x_0} \right| dx' \]  

(5)

Where \(d(x)\) is the deformation and \(x_0\) distance where the deformation is zero. The Poisson’s ratio is denoted by \(\nu\) and the module of elasticity is denoted by \(E\).

In order to perform an analytical analysis of the momentum equation, some simplifications have been introduced, namely, no bodyforces, no compressibility and no time dependency. An order of magnitude analysis of the momentum equation (2) with the simplifications indicated above will give the following momentum equation for the high-pressure region:

\[ \frac{\partial p}{\partial x_i} = \eta' \frac{\partial \bar{u}_i}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \eta \frac{\partial^2 u_i}{\partial x_j^2} \]  

(6)

An expansion of equation (6) into its component form along with some algebra will give the following equations:

\[ \frac{\partial p}{\partial x} = \frac{2\eta\eta' e_{xy} \nabla^2 v + (1 - 2\eta' e_{xy}) \eta \nabla^2 u}{1 - (2\eta')^2 (e_{xy}^2 - e_{xx} e_{yy})} \]  

(7.1)

\[ \frac{\partial p}{\partial y} = \frac{2\eta\eta' e_{xy} \nabla^2 u + (1 - 2\eta' e_{xx}) \eta \nabla^2 v}{1 - (2\eta')^2 (e_{xy}^2 - e_{xx} e_{yy})} \]  

(7.2)

\[ e_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \]  

(8)

The rate of deformation tensor is here denoted by \(e_{ij}\) and \(\eta'\) is the derivative of viscosity with respect to pressure. The steps of the derivation of these equations can to be found in Bair et al. [1].

As can be seen, the same denominator will occur in the two momentum equations in (7) and a singularity in the pressure gradients will occur if the denominator approaches zero. By rewriting the denominator in (7), an expression for the principal shear stress when a singularity (singular shear stress) may occur in the pressure gradients in the fluid can be obtained:

\[ \tau = \frac{1}{f'(p)} \]  

(9)

In expression (9), \(\tau\) represents the principal shear stress.

2.2 Numerical solution

The full N-S equations were solved i.e. inertia and compressibility terms were retained, using a commercial CFD-software CFX 4.3 [3]. The solution algorithm was as follows:
1. Create a grid and set initial values of density, viscosity and boundary conditions.
2. Computation of $u$, $v$-velocity
3. Computation of the pressure
4. Computation of the viscosity and density
5. Convergence check, mass and $u$, $v$-momentum residuals, if convergence then quit, otherwise repeat.

The solution algorithm used by the code is sequential, which can easily lead to instabilities. For this reason large under-relaxations must be used, which lead to a slow convergence rate.

The code uses finite volume discretisation with second order accuracy. The pressure correction algorithm used was SIMPLE [3]. The relaxation algorithm for the momentum equation was the Stone’s method [3] and for the pressure correction the ICCG (conjugated gradient method) was used [3].

In the numerical simulation, the Barus, Roelands and Dowson-Higginson expressions were used for the viscosity and density (Hamrock [4]). The meshes used were structured non-uniform meshes with a higher mesh density in the high-pressure region.

2.3 Cavitation
An important factor in the CFD-solution for the line contact problem is that pressure is obtained from other equations, the equations of momentum and continuity and an equation of state if the flow is compressible.

In the exit region of the line contact, cavitation will occur. One way of simulating this is to modify the density.

The method used in this work was to model the density using the Dowson-Higginson expression (Hamrock [4]) when the pressure is above a specified cavitation pressure $P_{\text{ca}}$, and when the pressure falls below $P_{\text{cav}}$, a second order polynomial was used to interpolate the density down to zero.

3 RESULTS
In order to show the existence of the singularity, a simple converging gap experiment was designed. The inlet and outlet heights of the converging gap were $1.6 \times 10^{-7}$, $1.5 \times 10^{-7}$ [m] respectively and the gap length was $9.0 \times 10^{-4}$ [m]. The lower surface of the gap was given a constant velocity of 0.5 m/s while the upper surface was held fixed.

The pressure was prescribed at both the inlet and outlet so pressure boundary conditions were used and ‘no slip’ wall boundary conditions were used for the upper and lower surface. No deformations of the surfaces were allowed.

The incompressible N-S equations were solved. In Fig.1 a comparison of pressure distributions from the Reynolds and N-S equations are shown. The Barus viscosity expression was used and the pressure distribution in the N-S solution was taken along the lower wall of the gap.
Comparison between the Reynolds and N-S solutions was made for two different values of the pressure-viscosity coefficient, $\alpha$. The higher value of the pressure-viscosity coefficient will force the principal shear stress to approach the singular shear stress criterion (9). In the lower graphs a $\alpha$-value of $9.0 \times 10^{-9}$ was used whilst in the upper graphs a $\alpha$-value of $1.0 \times 10^{-8}$ was used.

The maximum principal shear stress in the upper graphs is $7.1 \times 10^{7}$ Pa and the singular shear stress $1.0 \times 10^{8}$ Pa. In the lower graphs, the maximum principal shear stress is $2.5 \times 10^{7}$ Pa and the singular shear stress $1.11 \times 10^{8}$ Pa. In this case the Reynolds equation produces acceptable results. However, Fig.1 indicates there may be discrepancies between the N-S based solution and the Reynolds solution when the principal shear stress approaches the singularity. The maximum pressure difference across the film was less than 1.0 MPa.

The next comparisons were made for a smooth line contact geometry, and including compressibility and deformations of the surfaces in the computations. The viscosity expression used was Roelands and density expression Dowson-Higginson. The data used in the simulations are presented in Table 1 below.

<table>
<thead>
<tr>
<th>Test case:</th>
<th>Case 1:</th>
<th>Case 2:</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\rho_0$</td>
<td>$8.7 \times 10^2$</td>
<td>$8.7 \times 10^2$</td>
</tr>
<tr>
<td>$\eta_0$</td>
<td>$4.0 \times 10^{-2}$</td>
<td>$4.0 \times 10^{-2}$</td>
</tr>
<tr>
<td>$U_a$</td>
<td>$5.0 \times 10^{-1}$</td>
<td>0.0</td>
</tr>
<tr>
<td>$U_b$</td>
<td>$5.0 \times 10^{-1}$</td>
<td>$5.0 \times 10^{-1}$</td>
</tr>
<tr>
<td>$Z$</td>
<td>$5.0 \times 10^{-1}$</td>
<td>$5.0 \times 10^{-1}$</td>
</tr>
<tr>
<td>Load</td>
<td>60.0</td>
<td>92.0</td>
</tr>
</tbody>
</table>

Table 1. Input data for line contact simulation.

In the first experiment, a pure rolling case was tested. The results are shown in Fig.2. The data used in the experiment is given under "Case 1" in Table 1.
Figure 2. Pressure and film thickness for the Navier-Stokes (-) and Reynolds (°) solutions for a smooth line contact in pure rolling. Upper and lower surface velocity of 0.5 m/s.

Comparing the results from the two approaches indicates very small discrepancies. The value of the maximum principal shear stress in the contact is $1.02 \times 10^8$ Pa and the value of the singular shear stress is $1.33 \times 10^8$ Pa.

In order to force the smooth line contact solution closer to the singularity, sliding was introduced. The data used in the simulation is given under “Case 2” in Table 1.

Figure 3. Pressure and film thickness for Navier-Stokes (-) and Reynolds (°) solutions for a smooth line contact in pure slip. The upper surface was fixed while the lower surface has a velocity of 0.5 m/s.
Comparing the results from both approaches again shows small discrepancies, i.e. no influence of the singularity can be seen. The maximum principal shear stress is 9.8×10^7 Pa and the singular shear stress is 1.2×10^8 Pa

4 DISCUSSION
An interesting aspect of the momentum equations presented in (7) is the possibility of a singularity in the pressure gradient occurring. As can be seen from (9), this singularity can occur for stresses of the same order of magnitude as those found in EHL contacts. The numerical simulation presented in Fig.1 clearly show the presence of the singularity.

Performing simulations near the singularity is difficult because passing through the singularity will lead to a change of a type in the momentum equations and diverging solutions will occur. For a detailed explanation see Renardy 1986 [5].

Reynolds equation does not account for the singularity and with a Reynolds based solution with no limiting shear stress function (i.e. Newtonian rheology), principal shear stresses will be produced that pass through the singularity. Bair et al. [1] suggested that the singularity in the momentum equation is the source of the mechanical shear band seen in their experiments. No effects of the singularity have so far been seen in the line contact solutions. The maximum shear stresses in the line contact in the slip experiment presented above were confined to the middle of the contact, where the velocity profile is Couette. That means that the nominator in (7) is zero and the effect of the singularity in the denominator will vanish.

If the principal shear stress in the inlet approaches the singular shear stress, or where a Poisuselle component exists, the effects of the singularity will almost certainly be seen.

Any further increase in the load in the above experiments will force the principal shear stress to pass the singularity and the computations will fail. In order to perform computations at higher loads using the full flow equations, the stresses produced in the fluid must be limited. Thermal effects or non-Newtonian rheology with a yield stress criterion can be used to reduce the stresses in the fluid.

5 CONCLUSIONS
The aim of this investigation was to compare two approaches of solving the lubrication problem under isothermal EHL-like conditions and to investigate the possibilities of using a commercial CFD package CFX 4.3 [3]. The results from this work show that discrepancies may exist between the two approaches.

Simulation of the flow in a converging gap indicated that:

- The presence of the singularity in the momentum equations will force the pressure grow faster than predicted by the Reynolds equation, when the maximum principal shear stress approaches the singular shear stress criterion, see (9).
- When the maximum principal shear stress is far from the singular stress no discrepancies between the two solutions can be observed.

For a line contact simulation the following conclusions could be drawn:

- No effects of the singularity were observed.
- It is possible to use a commercial CFD-code but the loads are limited due to the singularity and the computational cost is high.
• The singularity cannot be passed through since thermal and/or rheological effects will prevent this.

Acknowledgement
The authors gratefully acknowledge financial support from the Swedish Research Council for Engineering Science (TFR) and The National Graduate School of Scientific Computing (NGSSC).

6 REFERENCES
The objective of the present research is to verify a THD model of hydrodynamic thrust bearings. The developed model of a pivoted pad bearing, which can tilt both radially and circumferentially, allows for three-dimensional temperature distribution in the oil film and in the pad, as well as two-dimensional temperature variation in the runner. Viscosity and density are treated as functions of both temperature and pressure. Experiments have been performed on a test rig, containing two identical equalizing pivoted pad thrust bearings. Power loss, runner temperature and pressure profiles as a function of load and rotational speed are compared for both theoretical and experimental investigations. Fairly good agreement has been found when the oil inlet temperature and heat transfer coefficients have been estimated in order to get the same runner temperature in both theory and experiment.

1 Introduction

Hydrodynamic thrust bearings are widely used in applications where load capacity and durability are highly important. Classically, Reynolds equation is solved in order to determine the bearing performance, and a solution requires knowledge about lubricant properties such as viscosity and density of lubricant. A more realistic model of the bearing performance treats viscosity and density as both temperature and pressure dependent. That requires a solution of the energy equation, where generation of heat in the lubricant due to viscous dissipation and compression is taken into account.

A number of computer models of hydrodynamic thrust bearing performance have been proposed over the years. Tieu [1] performed a three-dimensional THD analysis of a finite-width thrust bearing, which can tilt circumferentially. Kim et al. [2] presented a three-dimensional THD bearing model taking into account radial tilt, but elastic and thermal distortion of the bearing surfaces was neglected. Viscosity and density were only a function of temperature. Pad oil inlet temperature was assumed to be uniform. In the model proposed by Jeng et al. [3] pad deformations were allowed as well as viscosity variation with temperature. Recently Rodkiewicz and Yang [4] proposed a model for the infinitely long centrally pivoted thrust bearing with both pressure and temperature dependent viscosity and density along with elastic and thermal deformation of the bearing components. The compressive term in the heat equation was also included. A model proposed by Brockett et al. [5] for the fixed geometry thrust bearing includes temperatures and deformations in both the runner and pad. Viscosity was treated as both pressure and temperature dependent. Other approaches to the modeling of thrust bearing performance can also be found in literature (see e.g. Huebner [6], Ettles and Anderson [7]).

However, even today's computer models rely heavily on test data to set up a number of parameters used in the performance calculations and to be verified. Although many theoretical and experimental investigations [8-11] have been undertaken during the last three decades, only a few of published papers contain detailed comparison between theoretical and experimental results.
Such comparison is often carried out for scattered experimental results or for a special “model” bearing. It is therefore difficult to deduce if the theoretical model is suitable for simulation of hydrodynamic thrust bearings under a wide range of operating conditions.

The objective of this work is, therefore, a detailed comparison between a theoretical simulation and an experimental investigation of the industrial “catalogue” hydrodynamic pivoted pad thrust bearing. The model, used in this work, treats both viscosity and density as temperature and pressure dependent. The thermodynamic problem includes runner and pad along with the oil film. The experimental investigation is carried out for two identical thrust bearings. Friction torque, runner temperature and film pressure profiles are measured simultaneously.

2 Theory

2.1 The governing equations. A 3-D variable viscosity and density distribution are assumed. The governing equations for the lubricant are equation of motion, equation of continuity and the energy equation. The Laplace’s equations are used for heat conduction in the pad and runner respectively. The geometry of the problem proposes a cylindrical polar coordinate system.

2.2 Pressure equation. A generalized Reynolds equation [12] can be derived from the equation of motion and the continuity equation. In this equation the rheological behavior of the viscosity assumes to be Newtonian, and time independent. The equation used is:

\[
\frac{\partial}{\partial r} r \frac{\partial p}{\partial r} \int_{0}^{h} \rho \left( F_4 - \frac{F_2 F_3}{F_1} \right) dz + \\
\frac{\partial}{\partial \theta} \frac{1}{r} \frac{\partial p}{\partial \theta} \int_{0}^{h} \rho \left( F_4 - \frac{F_2 F_3}{F_1} \right) dz = r \omega \frac{\partial}{\partial \theta} \int_{0}^{h} \rho \left( 1 - \frac{F_3}{F_1} \right) dz
\]

\( F_1 = \int_{0}^{h} \frac{1}{\eta} dz, \quad F_2 = \int_{0}^{h} \frac{z}{\eta} dz, \quad F_3 = \int_{0}^{h} \frac{1}{\eta} dz, \quad F_4 = \int_{0}^{h} \frac{z}{\eta} dz \)

In equation (1), \( h \) represents the local film thickness, and the cylindrical coordinates are represented by \((r, \theta, z)\). The viscosity and the density are represented by \( \eta \) and \( \rho \).

The bearing assumes to work in the fully flooded lubrication regime, where the boundary conditions are zero pressure at the periphery of the pad and along the trailing and leading edges.

2.3 Energy equation. A thermal variant of the energy equation is used to obtain the temperature distribution in the film. The equation assumes steady flow and constant specific heat \( c_p \) and thermal conductivity \( k \). The viscous dissipation term is reduced by an order of magnitude analysis leaving out less important terms.

The equation used in this work is:
\[
\rho u \frac{\partial T}{\partial r} + \frac{1}{r} \rho v \frac{\partial T}{\partial \theta} + \rho w \frac{\partial T}{\partial z} = \Gamma_T \left[ \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial T}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 T}{\partial \theta^2} \right] \\
+ \frac{\partial^2 T}{\partial z^2} \right] + \frac{\eta}{c_p} \left[ \left( \frac{\partial u}{\partial z} \right)^2 + \left( \frac{\partial v}{\partial z} \right)^2 \right] + \varepsilon T \left( \frac{\partial p}{\partial r} + \frac{v}{r} \frac{\partial p}{\partial \theta} \right)
\]

The flow velocities \((u, v, w)\) and the derivatives of velocity and pressure are determined from the solution of Reynolds equation. The constants \(\Gamma_T, \varepsilon\) are diffusivity and thermal expansion coefficient respectively.

The boundary conditions imposed at the exit planes i.e. inner- and outer radius and trailing edge, are zero heat conduction. Inlet temperature is subdivided in two layers. The layer nearest the runner consists of the mean temperature at the trailing edge of the upstream pad. The second layer has the same temperature as the cold oil from the lubricant supply, Fig. 1.

The thickness of the hot oil carry over layer \(h_o\) is a parameter that can be determined from the experimental results. The parameter can be varied between 0 and 100 percent of the entering oil film thickness.

![Diagram](image)

**Figure 1. Model for the inlet temperature.**

According to Fig. 1, \(T_a\) is the supply oil temperature and \(T_h\) is the mean temperature of the oil from the upstream pad trailing edge.

### 2.4 Heat transfer equations.

The 3-D Laplace’s equation gives the temperature distribution in the pad and has the form:

\[
1 \frac{\partial}{\partial r} \left( r \frac{\partial T}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 T}{\partial \theta^2} + \frac{\partial^2 T}{\partial z^2} = 0
\]

This equation is solved simultaneous with the energy equation. The assumption \(\partial T/\partial \theta = 0\) in the runner gives Laplace’s 2-D equation, i.e. no tangential temperature variations are allowed in the runner.

There are three types of boundary conditions used at the pad and runner:

\[
-k \frac{\partial T}{\partial n} = \lambda (T_B - T_a)
\]

\[
k \frac{\partial T}{\partial z} = k_s \frac{\partial T_s}{\partial z}
\]
\[ k_s \frac{\partial T_s}{\partial r} = 0 \]  \hspace{1cm} (6)

The convection coefficient is denoted by \( \lambda \) and the boundary temperature by \( T_b \). The ambient temperature is denoted by \( T_a \) and has the same value as the entering supply oil in to the bearing. The pad and the runner are indexed \( s \).

Figure 2 represent a cross-section of the bearing, and the type of boundary condition is indicated at each boundary. In Fig. 2 the dashed line represent a virtual boundary to the shaft. It is assumed that the backside of the runner is in contact with air having high contact of oil and that the runner periphery as well as all sides of the pad is in contact with oil.

There are two more boundaries on the pad, according to Fig. 2, surfaces at the pad’s leading and trailing edges. Boundary condition (4) is used at these surfaces.

![Figure 2. Schematic figure over the boundary conditions at the runner and pad (the numbers refers to the applied boundary conditions).](image)

2.5 Viscosity-density expression. The viscosity model is a combination of Roelands and Barus expressions for the temperature-viscosity and temperature-pressure dependence, see e.g. Hamrock [13]. The model used in this work is:

\[
\eta = 10^{\left(\frac{G_0}{T} - 4.200\right)} (1 + 135 S_0) e^{\alpha p} \]  \hspace{1cm} (7)

Where \( G_0 \) is a constant which indicates the viscosity grade of the lubricant. The constant \( S_0 \) is the slope of viscosity-temperature relationship. The coefficient \( \alpha \) describes the pressure-viscosity relationship and is temperature dependent.

The density model has a pressure dependent part according to Dowson and Higginson [14] and the second part is thermal expansion.

\[
\rho = \rho_0 \left(1 + \frac{A_a p}{1 + B_b p}\right) [1 - \varepsilon (T - T_a)] \]  \hspace{1cm} (8)
3 Solution procedure

The Reynolds equation, the energy equation and the heat transfer equations are all solved using finite differences. All derivatives in both Reynolds and the heat transfer equations are discretized by using the CDS (central differencing scheme).

In the energy equation the convective term is discretized using the UDS (upwind differencing scheme). Other derivatives in the energy equation are discretized by CDS. The reason using UDS in the convective term is the high local Peclet number, which leads to instabilities if CDS is used.

In the pressure equation a solution domain of 15 by 15 grid points was used and 15 nodes were used across the film. In order to solve the energy equation a 15x15x15 domain was used. When solving the heat transfer equations a 15x15x15 domain was used for the pad and finally a 15x15 domain was used for the runner solution.

The computational procedure starts by solving the Reynolds equation. When the pressure distribution fulfils moment and force equilibrium equations, the procedure continues with the energy equation and the heat transfer equations. The convergence criterion in the temperature computations is when the dimensionless heat flow from the film into the runner has gone down three orders of magnitude. After inner iterations in pressure and the temperature computations, an outer iteration is made, i.e. back to the pressure computation. Before returning to the inner iteration an update in viscosity and density is carried out. The convergence criterion for the procedure is when the stepwise discrepancy in the film temperature is less than $10^{-4}$.

4 Experiments

4.1 Test rig and instrumentation. A special test rig was developed to study performance characteristics of industrial hydrodynamic thrust bearings. A photograph of the test rig is shown in Fig.3.

![Figure 3. Photograph of the test rig.](image-url)
This rig includes a housing, containing two identical thrust bearings to be tested and a main shaft connected through an intermediate shaft and a belt and pulley system to a 148 kW AC motor with an adjustable speed controller. Maximum operating speed of the motor is 1800 rpm but higher speeds can be obtained with different pulley sections.

The intermediate shaft, supported by two rolling element bearings, is necessary to avoid an influence of the tangential force from the belt on the thrust bearings to be tested. The main shaft with two separate runners is supported by fluid film journal bearings.

Thrust bearings are tested in a balanced pair to accommodate a substantial axial force, resulting from the bearing loading by hydraulic cylinders. The cylinders are located between the tested bearings, which in turn are placed against separate runners. The load on the bearings is calculated from the measured pressure in the hydraulic system and known piston area.

The hydraulic cylinders are connected to an accumulator used to maintain constant pressure in the system during tests. Pressure is created by a hand pump, which is disconnected after loading from the hydraulic system to prevent an influence on the friction torque measurement. Since the housing is situated on four rolling element bearings it is free to be rotated by the action of bearing friction torque. This rotation is prevented by a strain sensor, which shows the actual value of friction in the bearings. It should be pointed out that there is no mechanical contact between the housing and the rotating shaft since labyrinth seals are used. This allows an accurate measurement of bearing power loss.

There are two oil supply systems: one for journal bearings and one for bearings to be tested. The last one includes a 400-litre oil reservoir, a screw pump, a heat exchanger, a flow meter and a filter. Inlet oil hoses are made as a spiral to minimize their stiffness and thus their influence on friction torque measurement. Oil inlet temperature is held constant by adjusting the water flow through a heat exchanger.

Test rig instrumentation encompasses on-shaft sensors and ground-mounted sensors. The on-shaft instrumentation is two high frequency pressure transducers and three thermistors. Pressure transducers and two thermistors are mounted in the runner while one thermistor is placed in the shaft center. The thermistors are installed 1.5 mm under the runner surface. Locations of these sensors are shown in Fig. 4. The raw data from these instruments are transmitted to the data acquisition system with slip rings. The pressure is measured with an accuracy of ±4%, the temperature ±0.3°C, and power loss ±1%.

The ground-mounted instruments except above-mentioned strain and pressure sensors include thermocouples, flow meters and shaft speed meter. Thermocouples are used to control oil inlet and outlet temperatures for tested and journal bearings as well as cooling water temperature. Signals from all instruments are logged by a computer.

4.2 Test bearing. The test bearing is an equalizing pivoted-pad thrust bearing with steel-backed and babbitt-faced pivoted pads. Pad support is made from hardened steel with a spherical contact surface. The bearing characteristics are given in table 1.
Temperature sensors

Pressure transducers

Figure 4. Sensor locations

<table>
<thead>
<tr>
<th>Table 1. Test bearing characteristics</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inner diameter, mm</td>
</tr>
<tr>
<td>Outer diameter, mm</td>
</tr>
<tr>
<td>Number of pads</td>
</tr>
<tr>
<td>Bearing area, mm²</td>
</tr>
<tr>
<td>Pad angle, degrees</td>
</tr>
<tr>
<td>Pad thickness, mm</td>
</tr>
<tr>
<td>Pivot position (offset), %</td>
</tr>
<tr>
<td>Runner thickness, mm</td>
</tr>
<tr>
<td>Runner diameter, mm</td>
</tr>
</tbody>
</table>

4.3 Test procedure. The basic test procedure was as follows. Test sequence begun by setting the speed to 1200 rpm and specific load to 0.5 MPa. After thermal equilibrium was reached the data were measured. A criterion for the equilibrium was the variation in runner temperature. If this variation didn't exceed ±0.2°C during 15 minutes, equilibrium was judged to be established. The speed was then increased to 1800 rpm and then to 2500 rpm at the same specific load. Experimental data were also generated when speed was decreased to 1200 rpm through 1800 rpm to remove questions about thermal cycling. The difference in measured data was found to be within the accuracy of the measuring instruments. The same procedure was repeated for other specific loads.

All tests were conducted with oil supplied temperature of 50°C. It was held constant within ±0.2°C. Oil supply flow rate of 10.25±0.25 l/min was the same for all operating conditions.

5 Lubricant characteristics

The lubricant used in the tests was a synthetic oil VG46 of polyalphaolefine type. Its characteristics can be found in Table 2. The values of the viscosity, density and temperature-pressure-viscosity relationships are obtained from Duclos [15].
Table 2. Lubricant properties.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \rho_0 )</td>
<td>820 [kg/m(^3)]</td>
</tr>
<tr>
<td>( \eta_{0(323 , K)} )</td>
<td>0.331 [Pa s]</td>
</tr>
<tr>
<td>( \eta_{0(353 , K)} )</td>
<td>0.011 [Pa s]</td>
</tr>
<tr>
<td>( \alpha_{513 , K} )</td>
<td>1.6916 \times 10^{-8} [m(^2)/N]</td>
</tr>
<tr>
<td>( \alpha_{573 , K} )</td>
<td>1.3237 \times 10^{-8} [m(^2)/N]</td>
</tr>
<tr>
<td>( \varepsilon )</td>
<td>6.5 \times 10^{-4} [1/K]</td>
</tr>
<tr>
<td>( c_p )</td>
<td>2190 [J/(kg K)]</td>
</tr>
<tr>
<td>( G_0 )</td>
<td>3.755</td>
</tr>
<tr>
<td>( S_0 )</td>
<td>1.128</td>
</tr>
<tr>
<td>( A_a )</td>
<td>0.6 \times 10^{-9} [m(^2)/N]</td>
</tr>
<tr>
<td>( B_p )</td>
<td>1.7 \times 10^{-9} [m(^2)/N]</td>
</tr>
</tbody>
</table>

6 Results and discussion

The strategy used in this work was to adjust the model according to one separate experimental test run. It is important since heat transfer and heat conduction coefficients as well as the temperature profile of the incoming oil are difficult to deduce theoretically. Empirical values of the convection coefficient from the back and side surfaces of the pad to surrounding oil available in the literature are varied over three or more orders of magnitude. It is also assumed that the air in the housing has a high content of oil. The same value of the convection coefficient for the air as for the surrounding oil was therefore given. Moreover, models for hot oil carry-over do not take into account oil flow rate to the bearing. It strongly influences thermal conditions in the bearing. It was one of the reasons oil flow rate was kept constant in all tests.

To overcome above-mentioned difficulties, thickness of the hot oil carry over layer and thermal parameters were adjusted until an acceptable correlation occurs for one set of operating conditions. It was done for the rotational speed 1800 rpm and bearing load 1.0 MPa. These parameters were then held constant for all load-speed combinations. The parameter \( h_0 \) was adjusted to the fraction 44 percent of the entering oil film thickness, other data on the parameters used in the simulation are listed in Table 3.

Table 3. Thermodynamic properties.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \lambda_{\text{hub}} )</td>
<td>500 [W/(m(^2)K)]</td>
</tr>
<tr>
<td>( \lambda_{\text{shaft}} )</td>
<td>1000 [W/(m(^2)K)]</td>
</tr>
<tr>
<td>( \lambda_{\text{air}} )</td>
<td>500 [W/(m(^2)K)]</td>
</tr>
<tr>
<td>( k )</td>
<td>0.15 [W/(m K)]</td>
</tr>
<tr>
<td>( k_s )</td>
<td>50 [W/mK]</td>
</tr>
<tr>
<td>( T_a )</td>
<td>323 [K]</td>
</tr>
</tbody>
</table>

Figure 5 shows the oil film pressure distribution along the arcs at the positions of 25% and 75% of pad width. The theoretical results and the experimental data show fairly good agreement. Initial parts of the experimental and theoretical pressure profiles at 75% position coincide. However, experimental peak pressure is lower and shifted towards the leading edge. It is believed to be due to the elastic and thermal deformations of the bearing components. Thermal crowning leads to thicker oil film at the outer edge of the pad thus reducing the peak pressure. This effect is not as pronounced at the inner edge. It is confirmed by an excellent correspondence between simulated and experimental peak pressures. Higher pressure in the initial part of the experimental pressure profile can be explained by surface irregularities.
The temperatures in the center of the shaft and in the runner in the positions of 25% and 75% of pad width are presented in Figs. 6-8. The experimental values are denoted by e in these figures.

Figure 5. Pressure distributions along the pad at the positions of 25% and 75% of pad width. Bearing load 1.5 MPa, rotational speed 1800 rpm.

Figure 6. Temperatures in the center of the shaft and in the runner at the positions of 25% and 75% of pad width versus rotational speed. The load is 0.5 MPa.
Figure 7. Temperatures in center of the shaft and at runner at the positions of 25% and 75% of pad width versus rotational speed. The load is 1.0 MPa.

Figure 8. Temperatures in center of the shaft and at runner at the positions of 25% and 75% of pad width versus rotational speed. The load is 1.5 MPa.

As seen in Figs 6-8 the largest deviation between simulated and experimental values are found at high and low rotational speeds. The major reason to such behavior is the choice of experimental data at which the model has been adjusted.

It is likely that parameters such as hot oil carry over and heat transfer coefficients change slightly as the operating conditions change.
In this study there is an opportunity to adjust these parameters to experimental data, but a very careful estimation of the parameters has to be carried out if the simulation should be done for new designs without the possibility to compare with tests.

![Graph](image)

**Figure 9.** Bearing power loss versus rotational speed at the loads 0.5, 1.0 and 1.5 MPa respectively.

![Graph](image)

**Figure 10.** Bearing power loss versus load. Rotational speed is 2500rpm.

Runner temperature varies in the radial direction and this variation increases with higher speed and load. The effect of speed on runner temperatures is higher than that of the load.

Shaft center temperature is only slightly effected by load and speed since it is cooled by incoming supply oil with constant temperature.
One very important factor influencing thrust bearing performance is the rate of oil supply to the bearing. It is easy to adjust in the experiments but very difficult to take into account in the theoretical model. That is why oil flow rate to the bearing is the same in all tests. It is a recommended flow rate for the speed 2500 rpm and load 1.5 MPa. The recommended flow gives a 15-17°C oil temperature rise through the bearing. A decrease in load or speed will result in a lower temperature rise from inlet to discharge. Thus, if the recommended flow is used for each load-speed combination a change in runner temperatures will be greater.

In Fig. 9, bearing power losses are shown as a function of rotational speed for three different loads. The largest deviations occur once again at the higher speeds. One of the reasons is the same as for the temperature. Hot oil carry over layer thickness and heat transfer coefficients are not adjusted for the higher speed case.

As seen in Fig. 9, simulated power loss is higher than the measured loss. This seems to be strange for two reasons. Firstly, the temperature is higher in the simulation and the power loss is then also expected to be lower than measured power loss. Secondly, the measured power loss also includes churning losses at other surfaces where the lubricant is sheared. The power loss is thus overestimated. One reason for this may be the way how the hot oil carry over problem is treated and there are expectations of getting even better results if a more accurate inlet temperature function can be found.

Fig. 10 shows the power loss as a function of load, where the rotational speed is held constant at 2500 rpm. The influence of bearing load on power loss is decreasing at higher loads. Higher loads produce steeper pressure gradients in the oil film thus increasing power loss and film temperatures. At the same time, thinner oil film leads to smaller amount of heat carried away by the oil, which also contributes to higher temperatures in the oil film. Elevated temperatures decrease oil viscosity and, consequently, shearing work being done. Thus, thermal effects compensate an increase in bearing load.

An increasing discrepancy between experimental and theoretical power loss at higher loads is also due to the fact that oil flow rate, optimized for the load 1.5 MPa, does not produce the same cooling effect as for lower loads. This leads to greater deformations of the bearing components and consequently to increasing discrepancy between theory and experiment.

7 Conclusions

The aim of this research has been to verify a pivoted pad thrust bearing computer model against a detailed experimental investigation. The comparison of the theoretical and experimental results is encouraging and one may conclude that the two agree despite of the fact that elastic and thermal deformations of the bearing components were not taken into account.

The prerequisites for such good agreement are:

- adjustment of secondary parameters such as hot oil carry over layer thickness and heat transfer coefficients according to experimental results;
- an accurate description of lubricant property parameters;

A fairly good agreement between theoretical and experimental pressure profiles is obtained. Small discrepancies are attributed to the elastic and thermal deformations of the runner and pads. Power loss is slightly overestimated even if the model is adjusted in order to comply with measured temperatures.
The theoretical model has shown to be accurate enough to be used for simulations of thrust bearings operating in the laminar regime. Even if the absolute magnitude of power loss, temperatures etc. cannot be determined exactly for general cases, the model will be very useful for investigations of how bearing operation is influenced by e.g. lubricant properties, geometry parameters and operating conditions.

Acknowledgement

The authors gratefully acknowledge the financial support of the Swedish National Board for Industrial and Technical Development (NUTEK), ABB Stal AB, ABB Generation AB, Elforsk AB, FMV, Kingsbury Inc., Mobil Oil AB, Skogforsk, Statoil AB, Vattenfall AB and the Swedish Centre for Maintenance Engineering and Management (UTC). The project is also partly funded by National Graduate School of Scientific Computing (NGSSC).

Nomenclature

\[
\begin{align*}
  r, \theta, z &= \text{cylindrical coordinates} \\
  F_{1,4} &= \text{integrand in Reynolds equation} \\
  \rho &= \text{density [kg/m}^3\text{]} \\
  \rho_0 &= \text{density at ambient pressure and temperature [kg/m}^3\text{]} \\
  \eta &= \text{dynamic viscosity [Pas]} \\
  \varepsilon &= \text{thermal expansion coefficient [1/K]} \\
  \alpha &= \text{pressure-viscosity coefficient [m}^2\text{/N]} \\
  k &= \text{thermal conductivity film [W/(mK)]} \\
  k_s &= \text{thermal conductivity pad [W/(mK)]} \\
  \Gamma_T &= \text{diffusivity (k/c_p) [kgW/(mJ)]} \\
  \omega &= \text{angular frequency [rad/s]} \\
  u &= \text{velocity r-coordinate [m/s]} \\
  v &= \text{velocity \theta-coordinate [m/s]} \\
  w &= \text{velocity z-coordinate [m/s]} \\
  h &= \text{film thickness [m]} \\
  T &= \text{temperature [K]} \\
  c_p &= \text{specific heat [J/(kgK)]} \\
  \lambda_{\text{shaft}} &= \text{heat transfer coefficient shaft [W/(m}^2\text{K)}]} \\
  \lambda_{\text{lub}} &= \text{heat transfer coefficient lubricant [W/(m}^2\text{K)}]} \\
  \lambda_{\text{air}} &= \text{heat transfer coefficient of surrounding air [W/(m}^2\text{K)}]} \\
  p &= \text{pressure [Pa]} \\
  \Lambda_a &= \text{constant in density expression [m}^2\text{/N]} \\
  B_b &= \text{constant in density expression [m}^2\text{/N]} \\
  \text{subscripts} \\
  s &= \text{solid} \\
  a &= \text{ambient} \\
  B &= \text{boundary}
\end{align*}
\]

References


The work presented in this thesis concerns computer simulations of the lubrication process. The main subject of interest is elastohydrodynamic lubrication (EHL) and, to some extent, hydrodynamic lubrication (HD). The thesis comprises an introductory section and three papers; referred to as A, B and C. Simulation of EHL is an inter-disciplinary task, incorporating the fields of fluid mechanics, solid mechanics, thermodynamics and rheology. In almost all numerical simulations of lubrication performed today, the hydrodynamics are modelled using the Reynolds equation. This equation is derived from the equations of momentum and continuity and using the thin film approximation. However, the assumptions made when deriving this equation limits the size of the computational/spatial domain and the equation cannot predict pressure variations across the lubricating oil film. The subject of papers A and B are numerical simulations using the full equations of momentum and continuity, (Paper B), and the equation of energy (Paper A). The main aim of the work was to investigate the possibilities of carrying out numerical simulations based on the above equations. The rheology was assumed to be Newtonian; the equations are then commonly referred to as the Navier-Stokes equations (N-S). The second aim of the work was to investigate the possibilities of using a commercial software, CFX 4.3 [1], to carry out the numerical simulations.....(cont.)