Computational Fluid Dynamics
Simulation of A Scroll Compressor

Ji Ke

A thesis submitted in partial fulfillment of the requirements for the degree
of
BACHELOR OF APPLIED SCIENCE

Supervisor: Markus Bussmann

Department of Mechanical and Industrial Engineering
University of Toronto
March 2008
Abstract

A Computational Fluid Dynamics (CFD) simulation of a scroll compressor leakage model is developed. The simulation uses an existing commercial CFD code CFX, with a number of new routines written to allow it to simulate the conditions and motions involved in an moving scroll compressor. The meshing and the simulation are done based on results from other researchers. However, no validation of the solution can be performed in time.

First, a simplified scroll compressor model is constructed. This model has the benefit of being very flexible in terms of the simulation variables. These variables include but are not limited to: the simulation of gas leakage, simulation of outlet pressure, and with some additional work, it is able to simulate a full scroll compressor’s efficiency.

A mathematical model of the scroll compressor is also developed. The solution of the mathematical model can be solved numerically, but has not been attempted. The mathematical model, if accurate, can be used to verify some of the CFD solutions.

Finally, a small series of simulations is done using the developed model. The amount of circumferential leakage is the simulation result. Input parameters are the pressure differences between the inlet, the outlet, and the geometry. The trend of the results appears nominal when compared to theory. However, the values of the results cannot be verified due to the lack of physical models to comparison. A list of future work that can improve the validity of this thesis is listed.
I would like to express my gratitude to all those who gave me the possibility to complete this thesis. I want to thank Professor Markus Bussmann for giving me the permission to commence this thesis, and for all his guidance and support along the way. I would also like to thank Yi Ke and Anish Trikha of the University of Waterloo for providing me with the computer resources needed to complete this project. A special thanks to Sagebelle Wu for editing this thesis (a seemly painful task indeed). And of course, to my parents, Enjiu Ke and Ling Li, for being the engineers that I want to be.

Thank you.
## Contents

List of Figures 7

List of Symbols 9

1 Introduction 11
   1.1 Fundamental Principles ................................. 11
      1.1.1 Scroll Compressors Introduction ..................... 11
      1.1.2 Improving scroll compressor’s efficiency .......... 13
   1.2 Project Introduction ................................. 13
      1.2.1 Objective ....................................... 13
      1.2.2 Software Used .................................... 15
      1.2.3 Outline of the Current Work ....................... 15
      1.2.4 On mathematical model of the scroll compressor [1] [2] 15
      1.2.5 On Moving Boundary Meshing [3] ........................ 15
      1.2.6 On CFD Simulation [4] .............................. 16
   1.3 Motivation ............................................. 16

2 Mathematical Models 17
2.1 Conservation Equations .................................. 17
  2.1.1 Mass .................................................. 18
  2.1.2 Momentum .......................................... 18
  2.1.3 Energy ............................................... 19
  2.1.4 Equation of State ................................. 19
2.2 Euler’s Method ............................................ 19
2.3 Turbulence ............................................... 20
  2.3.1 Reynold’s Number ................................. 20
2.4 Mathematical Verification of the CFD results .......... 21

3 Scroll Compressor Modeling ............................... 23
  3.1 Scroll Compressor Layout ............................. 23
    3.1.1 Model Simplification ............................ 23
    3.1.2 Assumptions ..................................... 24
  3.2 Gas Leakage Modeling ................................. 25
  3.3 CFD Model ............................................. 25
    3.3.1 Meshing .......................................... 25
    3.3.2 Boundary Layer ................................. 26
    3.3.3 Wall Boundary Conditions ...................... 26
    3.3.4 Time Controls .................................. 28
    3.3.5 Mesh Motion .................................... 28
    3.3.6 Convergence Parameters ....................... 30

4 Simulation Results ......................................... 32
  4.1 Typical Velocity Profile ............................. 33
  4.2 Results Summary ..................................... 33
4.3 Typical Velocity Profile ........................................ 34
4.4 Typical Pressure Profile ........................................ 34
4.5 Typical Density Profile ........................................ 34

5 Discussion ......................................................... 35
  5.1 Development of a scroll compressor CFD method ............ 35
  5.2 Source of Errors ............................................... 36
    5.2.1 Modeling Errors ........................................... 36
    5.2.2 Simulation Errors ........................................... 37

6 Conclusion ......................................................... 38
  6.1 Computational Fluid Dynamics Modeling ...................... 38
  6.2 Circumferential Leakage Simulation ......................... 38

7 Figures .......................................................... 41
List of Figures

7.1 Basic scroll components 1 ........................................ 41
7.2 Basic scroll components 2 ........................................ 42
7.3 Basic scroll operation .............................................. 42
7.4 Two Leakage Types ................................................. 43
7.5 Reynold’s number as a function of gap width and flow rate .. 43
7.6 CFD simulation of a a Concentric Couette flow solution .... 44
7.7 Simplified Simulation Model ....................................... 44
7.8 Sweeping Meshing .................................................. 45
7.9 Meshing of the Simplified Model .................................. 46
7.10 Boundary Types .................................................... 47
7.11 Boundary Types .................................................... 48
7.12 Results Summary .................................................. 49
7.13 Typical Velocity Profile .......................................... 49
7.14 Mean Velocity Across the Gap .................................. 50
7.15 Volume Flow Rate Across the Gap .............................. 50
7.16 Density Across the Gap .......................................... 51
7.17 Mass Flow Rate Across the Gap ................................. 51
7.18 Velocity profile for 0.06mm gap size, 1.25bar outlet pressure,
zoomed in .................................................. 52

7.19 Pressure profile for 0.06mm gap size, 1.25bar outlet pressure,
zoomed in .................................................. 53

7.20 Pressure profile for 0.06mm gap size, 1.25bar outlet pressure,
zoomed out .................................................. 54

7.21 Density profile for 0.06mm gap size, 1.25bar outlet pressure,
zoomed in .................................................. 54
List of Symbols

Bold face indicates vector version of the normal faced symbol.

- \( l \): Mesh size
- \( \alpha(t) \): Angular offset between the orbiting scroll and the fixed scroll
- \( \delta \): Gap size between the center of the orbiting scroll and the center of the fixed scroll
- \( \Delta \): Minimum gap size between the two scroll profiles
- \( \Delta_\text{local} \): Local gap size between the two scroll profiles
- \( \Gamma \): Mesh stiffness value
- \( d \): Distance between the node and the moving boundary
- \( \chi \): Displacement relative to the previous mesh location
- \( m \): Mass
- \( \rho \): Density
- \( v \): Velocity
• \( \mu \) : Dynamic viscosity

• \( h \) : Enthalpy

• \( g \) : Gravity (9.81 m/s\(^2\))

• \( z \) : Height

• \( n \) : Outward normal vector

• \( a \) : acceleration

• \( V \) : Volume

• \( R \) : Gas Constant

• \( r \) : radius

• \( T \) : Temperature

• \( C_p \) : Constant pressure specific heat

• \( \Phi \) : Simulation Variable

• \( d \) : Characteristic length

• \( \omega \) : Angular Velocity
Chapter 1

Introduction

The scroll compressor, when compared to other compressors, is known to operate more smoothly, quietly and reliably. As with any machinery, it is the desire of the designer to improve the efficiency of the machine; and the modern tool of choice when coming to improving efficiency is to create a computational model. This thesis is aimed to create a computational fluid dynamics (CFD) model for the scroll compressor in hopes that it will benefit future attempts in modeling a scroll compressor.

1.1 Fundamental Principles

1.1.1 Scroll Compressors Introduction

A regular scroll compressor is a type of fluid machinery that compresses a fluid by constantly changing its internal volume. A scroll compressor usually consists of a housing, a stationary scroll fixed on the housing, a driving crankshaft supported by bearings on the housing, and an orbiting scroll
driven by the crankshaft (figure 7.1, 7.2). The orbiting scroll is constrained by an anti-self-rotating mechanism: the orbiting scroll can only orbit, not rotate, with respect to the stationary scroll. The volumes formed between the stationary scroll and the orbiting scroll change with the orbiting movement of the orbiting scroll, and cause the fluid in the volumes to be compressed (figure 7.3).

Due to the intricate and precise nature of the scroll vanes, it was only until the 1980s (75 years after the first scroll patent) that the scroll compressor could be economically manufactured. Scroll compressors are more advantageous than traditional compression techniques such as reciprocating and screw compressors in the following ways:

- more efficient
- more reliable
- lower noise
- lower vibration
- more compact

Scroll compressors have a low noise and low vibration characteristic due to the low line speed of the compressor. In addition, scroll compressors have fewer moving parts than other types of compressors which, theoretically, should improve reliability. Furthermore, scroll compressors do not require discharge valves and suction valves, which are some of the most vulnerable parts of a reciprocating compressor. However, scroll compressors are more
vulnerable to debris within the compression chamber, which is usually caused by the wear of the scroll involutes.

1.1.2 Improving scroll compressor’s efficiency

There are a number of ways to improve scroll compressor efficiency. The most successful methods include axial and radial compliance and oil/water injection. Axial and radial compliance is a method to mount the orbiting scroll compliantly on the driving shaft, thus cancelling any misalignment caused by machining tolerance and deformations. Oil/water injection is done by directly injecting a fluid into the compression chamber to seal any machining tolerance and deformations [5]. Both of these methods are aimed to provide better sealing between the fixed scroll and the orbiting scroll, thus limiting the external and internal leakage and improving the efficiency.

1.2 Project Introduction

1.2.1 Objective

The objective of this thesis is to develop a method to accurately simulate the behaviour of a scroll compressor. The secondary objective of the present thesis is to use the developed method to simulate the internal leakage of the scroll compressor.
Simulation Model Construction

In order to simulate a scroll compressor accurately in the given time period, a simplified CFD model was constructed. The objective of this model is to accurately portray the inner working of a scroll compressor and be flexible enough to add and modify simulator parameters easily.

Internal Gas Leakage Simulation

As with most compressors, internal gas leakage is a major cause of efficiency decrease. In scroll compressors, there are two types of internal leakage: axial and circumferential leakage (figure 7.4). Both are based on the concept of air in high pressure regions tending to move to regions of low pressure. Axial leakage occurs when air escapes between the axial gap to different compression chambers. The effect of the axial leakage is small by comparison, since the axial gaps are usually limited by tip seals. The second type is circumferential leakage, where the air in a high pressure chamber leaks to the adjacent low pressure chamber. The circumferential leak is the larger of the two internal leakages because scroll compressors are required to have some clearance between the fixed and orbiting scroll walls to allow no contact motion and to provide clearance for thermal expansion and machining allowance. The majority of internal leakages occur at this clearance, which serves as the focus of this thesis.
1.2.2 Software Used

Several commercial CFD solvers were examined at the start of the project for their suitability in solving transient simulations with moving boundaries. ANSYS CFX was selected for its user friendly interface and robust help system. Considerable amounts of time were spent on learning the software.

1.2.3 Outline of the Current Work

As serious work on scroll compressor research has started only since the 1980’s, relatively few academic research publications were available. The following are a few references that served as the basis of the present thesis.

1.2.4 On mathematical model of the scroll compressor [1] [2]

A mathematical model of a scroll compressor was constructed by Yu Chen for his Graduate studies thesis. From this thesis, the mass and energy conservation differential equations were analyzed. This document also provided insight on the initial test values for several CFD variables, including the timestep size. This document was invaluable in helping with the understanding of the mathematical modeling of a scroll compressor.

1.2.5 On Moving Boundary Meshing [3]

Although a completely different field, Kho and Garland’s study on 3D animations provided insight on how to mesh and animate a geometry in such a
way that the result keeps the skeletal appearance of the original mesh. The theories behind this document helped with animating a scroll compressor meshing that has a variable thickness of differing magnitudes.

1.2.6 On CFD Simulation [4]

The PHD thesis by Glenn Horrocks on the CFD modeling of a Rotary Valve Internal Combustion Engine helped with the understanding of how to apply CFD on a deforming solid/liquid interface. The detailed use of CFX in Horrocks’ PHD studies provided insight on how to perform similar experiments on a scroll compressor.

1.3 Motivation

The scroll compressor is a relatively new form of compression technology; little research has been done on the scroll compressor. To the author’s best knowledge, this is the first research paper on the methods of simulating the internal workings of a scroll compressor.

In addition, due to the desirable traits of the scroll compressor, it is the prime candidate for the fuel cell compressor/expander module. However, because compressors used in the fuel cell’s scroll compressor/expander module consumes about 20-30 % [6] of the power generated by the fuel cell system, it is reasonable to conclude that any increase in efficiency of a compressor will have a noticeable increase in the efficiency of the fuel cell.
Chapter 2

Mathematical Models

The mathematical model used to solve the velocity and pressure of the air is a transient, isothermal, displacement diffusion model. The transient scheme is of second order backward Euler convergence. The governing equations are the transient compressible Navier Stokes Equations. The governing equations are incorporated into a commercial CFD code, Ansys CFX, which was used to solve the present thesis scenario.

2.1 Conservation Equations

The compressor leakage model, like all other physical models, follows the conservation of mass, momentum and energy. The conservation equations serve as the basis for the Navier Stokes Equations. The Navier Stokes Equations are a set of nonlinear partial differential equations, and its conservation is an application of Newton’s second law. The compressible Navier Stokes
Equations are defined as follows:

\[
\rho \left( \frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} \right) = -\nabla p + \mu \nabla^2 \mathbf{v} + \mathbf{f} + \left( \frac{4\mu}{3} + \mu^{\nu} \right) \nabla (\nabla \cdot \mathbf{v})
\]

### 2.1.1 Mass

The compressor model is an open thermodynamic system in which mass enters and leaves the system. The amount of mass transferred represents the amount of gas leakage. The mass conservation equation for a deformable control volume is defined as:

\[
\left( \frac{dm}{dt} \right)_{\text{syst}} = 0 = \frac{d}{dt} \left( \int_{\text{CV}} \rho dV \right) + \int_{\text{CS}} \rho (\mathbf{v}_r \cdot \mathbf{n}) DA
\]

Which states the total mass difference of the control volume is equal to the rate of change of the content of the control volume plus the net rate of flow through the control volume.

### 2.1.2 Momentum

Similarly to conservation of mass, the rate of change of the linear momentum of the system is equal to the rate of change of the content of the control volume plus the net rate of flow through the control volume [7]:

\[
\left( \frac{d}{dt} \right) (m\mathbf{v})_{\text{syst}} = \sum \mathbf{F} = \frac{d}{dt} \left( \int_{\text{CV}} \rho \mathbf{v} dV \right) + \int_{\text{CS}} \rho (\mathbf{v}_r \cdot \mathbf{n}) DA
\]
2.1.3 Energy

The scroll compressor model is assumed to be isentropic, adiabatic and laminar. Although this assumption is unlikely to be true, it greatly simplifies the compressible flow calculation in terms of energy conservation. The energy conservation equation used in Ansys CFX is the Reynold’s averaged energy equation. [8]:

\[
\frac{\partial \rho h_{tot}}{\partial t} - \frac{\partial p}{\partial t} + \nabla \cdot (\lambda \nabla \tau) + \nabla \cdot (U \cdot \tau)
\]

The turbulence flux term, \( \rho u h \), and the entropy term are eliminated due to the assumptions.

2.1.4 Equation of State

To solve this system of equations, two relations are required. First is the equation of state, which for air as an ideal gas is defined as:

\[ PV = mRT \]

Second is the constitutive equation of enthalpy:

\[ h = C_p T \]

2.2 Euler’s Method

The general discrete approximation of the transient term for the \( n^{th} \) timestep is:
\[
\frac{\partial}{\partial t} \int_V \rho \Phi dV \approx V \frac{1}{\Delta t} \left( \frac{3}{2} (\rho \Phi) - 2 (\rho \Phi^o) + \frac{1}{2} (\rho \Phi^{oo}) \right) [8]
\]

This equation is second order accurate in time, which will minimize the discretization errors associated with the first order Euler’s approximation. Due to the aggressive deformation of the boundary layers of the orbiting scroll, the second order accuracy will also help to minimize the diffusion of steep temporal gradients. Finally, second order accuracy is also recommended for laminar solutions [8].

### 2.3 Turbulence

For a more accurate simulation and to examine the effects of wall roughness, a turbulent model should be used to simulate the compressor. However, as turbulent simulation will add significant CPU overhead, its benefit must first be quantified before it becomes a factor in the simulation.

#### 2.3.1 Reynold’s Number

The critical Reynolds number indicates the transition between laminar and turbulent flow. The critical Reynolds number for this scroll compressor can be found by experimentation and it is typically given as follows.

\[
Re = \frac{\rho v^2_s/L}{\mu \nu_s/L^2}
\]

The variable L is the characteristic length.

In the present experiment’s case, the characteristic length is a convention,
and it is arbitrarily defined as the smallest gap between the orbiting and fixed scroll. The assumption for this calculation is the flow rate (amount of leakage), which dictates the velocity of the flow.

Two simulations were run with identical variables. The difference is that one simulation is run with the assumption of laminar flow only while the other one has turbulence flow along with laminar flow. The differences between the two simulation results are minimal, and the simulation with turbulence took more time to simulate. To keep the simulation time down, it was decided to assume laminar flow only in the actual simulations.

Figure 7.5 shows a range of Reynold’s number that was used in the simulation.

2.4 Mathematical Verification of the CFD results

The present problem can be described mathematically as a concentric Couette flow. Coincidentally, the concentric Couette flow is also one of few exact solutions for the Navier-Stokes equation citation [7]. Therefore, Couette flow solution is very useful in validating the CFD results. To simplify the calculation, the following assumptions are made:

- Incompressible flow
- Laminar flow
- Isothermal flow
The conservation equation is therefore:

\[ \nabla \cdot \mathbf{V} = 0 \]

Which in cylindrical coordinates, it becomes:

\[
\frac{1}{r} \frac{\partial}{\partial r} (rv_r) + \frac{1}{r} \frac{\partial}{\partial \theta} (rv_\theta) + \frac{1}{r} \frac{\partial}{\partial z} (rv_z) = 0
\]

Assuming axial and radial flows are negligible, the following equation will be true:

\[
\frac{1}{r} \frac{\partial}{\partial \theta} (rv_\theta) = 0
\]

With the \( \theta \) -momentum equation:

\[
\frac{\partial v_\theta}{\partial t} + (\mathbf{V} \cdot \nabla) v_\theta + \frac{1}{r} v_r v_\theta = v(\nabla^2 v_\theta - \frac{v_\theta}{r^2})
\]

This equation can be solved numerically with velocity boundary conditions set to the same values as the solid boundary (no-slip). Figure 7.6 shows a CFD simulation of a Couette flow defined in this section.
Chapter 3

Scroll Compressor Modeling

3.1 Scroll Compressor Layout

3.1.1 Model Simplification

To use CFD to simulate something as complex as a scroll compressor will require considerable amounts of time and computer power. Unfortunately, neither was available for the present thesis. Therefore it is necessary to create a simplified model that specializes on the simulation of gas leakage.

The Atlas Copco SF2 oil-free scroll compressor is chosen as the modeling framework. Simplifications are made in the following areas: Geometry, boundary condition and assumptions.

Geometry

As seen from figure 7.7, the present simulation model is a copy of the first involute of an actual compressor in terms of the motion profile and the di-
dimensions. To accommodate the discontinuity of the fluid flow due to the removal of additional involutes, an artificial outlet is inserted into the simplified model. Furthermore, since the present simulation is interested only in circumferential leakage, the axial walls of the compression chamber will not be an factor, therefore it is assumed the compression chamber depth is infinite.

**Boundary Conditions**

The inlet pressure is given as the atmospheric pressure, and the outlet pressure is a simulation variable that can be either inputted by the user or solved by the CFD solver. In the present simulation’s case, it is desired to investigate the behavior of gas between the inlet and the outlet with a given outlet pressure, therefore the outlet pressure is inputted by the user as a fixed value in the range of 0.5atm to 1.25atm.

**3.1.2 Assumptions**

Several assumptions were made to simplify the model:

- Laminar Flow
- Adiabatic
- Flow rate is not a function of scroll depth
- Simulations variables are independent of each other
3.2 Gas Leakage Modeling

The amount of gas leaking from the simulated model is calculated by monitoring the velocity across the smallest gap. It has been found from CFD simulations that the gas will flow from outlet to inlet despite the opposite motion of the orbiting scroll. The motion of the orbiting scroll (thus compression chamber) must be taken into account, so the amount of gas flow across the gap should be calculated from relative velocity between the gas and the orbiting scroll. The following equation is used to calculate the relative gas velocity:

\[ v_{\text{relative}} = v_{\text{air}} - r_{\text{average}} \times \omega \]

\( \omega \) is constant in this case.

And the mass flow rate can then be determined by the following equation:

\[ \dot{m} = \rho A v_{\text{relative}} \]

3.3 CFD Model

3.3.1 Meshing

Meshing influences the accuracy, convergence and speed of the solution. Therefore significant amounts of time was spent on the meshing of the simplified model. Unfortunately, as meshing is more of an art than science, a trial and error method was used until a solution was generated with reasonable
confidence in its accuracy, and obtained in a reasonable amount of time.

As the simplified model can be defined as a swept surface, with the top left face and top right face being the origin and destination respectively, it is therefore beneficial to mesh the model with quadrilateral structural meshing. Sweeping meshing technique, sometimes referred to as 2.5-D meshing, was used on the simplified model. Sweeping works by creating a quadrilateral mesh on the beginning end, then sweeps to the other end which contains a similar topographical geometry. In this case, the origin face was swept to the destination face along the guide curves (figure 7.8). The guide curves are defined by the orbiting and fixed scroll profiles. The final mesh contains 20 layers of quadrilaterals as shown in figure 7.9, and a zoomed in view of the gap meshing can be seen in figure 7.10:

3.3.2 Boundary Layer

3.3.3 Wall Boundary Conditions

Four boundary types were used to define the simulation as shown in figure 7.11:

Symmetry

Since it is assumed the present scroll compressor is infinite in depth, the front and back face of the model is determined to be symmetrical to simulate a 2D mesh (CFX is not capable of simulating actual 2D geometries)
Inlet

The inlet represents the suction port of a scroll compressor. This boundary geometry is not an accurate representation of an actual suction port; this is mainly due to the deforming geometry requirements of the simplified model. As an actual suction port will not permit the deforming of a single involute scroll profile, the result will be the tearing and overlapping of geometries and create fatal error in the ANSYS CFX solver.

Outlet

The outlet represents the pressure port of a scroll compressor. Like the inlet, the outlet boundary geometry is also not an accurate representation of an actual suction port for the same reason. A note on nomenclature is that although the inlet and outlet in the present simulation are named to represent their counterparts in an actual compressor, as this simulation is a research on the leakage of the compressor, the direction of flow is actually from the outlet to inlet.

Walls (Moving wall, Free moving, Fixed wall)

Wall boundaries are defined as a solid impermeable surface that separates the fluid and solid domain. All walls in the current simulation have a no-slip boundary condition applied, effectively setting the fluid velocity at the wall equal to the velocity of the wall itself. Because heat transfer is omitted in this simulation, the walls are also adiabatic. Also as mentioned in the previous section, the simulation is assumed to be laminar, therefore the wall
roughness will not be a function of the flow rate (when the inlet pressure and
the outlet pressure are fixed), and the walls are defined as smooth.

In addition, to differentiate the fixed scroll and the orbiting scroll, three
mesh motion options are specified:

1. Moving Wall - Orbiting Scroll

2. Free Moving - Median virtual wall between the orbiting scroll and the
   fixed scroll

3. Fixed Wall - Fixed Scroll

3.3.4 Time Controls

CFX uses timestep values to dictate how long the interval is between each
transient calculation. The timestep can be a constant or a variable and due
to the constant RPM of the present simulation, constant timestep was used
to solve the simulation. An optimal timestep for this simulation is a function
of RPM, gap size and mesh size. However for practical reasons, a rough
timestep was estimated based on the mathematical formulas from Yu Chen’s
thesis. The initial estimate has since been improved by methods of trial and
error. The final timestep used for this model is 0.00001 second.

3.3.5 Mesh Motion

The current simulation follows an identical motion path as an actual scroll
compressor. The motion of the orbiting scroll boundary is defined as:
\[ x = \sin(\alpha(t)) \times \delta \]
\[ y = \cos(\alpha(t)) \times \delta \]

However, it is not enough to mobilize only the boundary, as in doing so will create stretched elements within the mesh. The ratio between the original mesh size and the mesh size after half rotation can be defined as:

\[ l = \frac{\delta}{\Delta} \]

As the typical gap size is only about 0.02mm, then the mesh size can increase to more than 100 times the original size if only the boundary layer is in motion.

Thus, it is required to define the motion of each node that is in a certain proximity of the moving boundary. To determine which nodes will move and the magnitude of the movement, each node is assigned a mesh stiffness value:

\[ \Gamma = \left( \frac{1}{d} \right)^6 \]

This solution indicates an exponential increase in mesh stiffness as the distance to the moving boundary decreases. The exponent 6 is experimentally determined to be the best value for the rate of mesh stiffening.

The mesh stiffness values are then used in the displacement diffusion equations to solve for the magnitude of transformation of each node [8]:

\[ \nabla \cdot (\Gamma \nabla \chi) = 0 \]
With this model, the displacements applied on domain boundaries or in sub domains are diffused to other mesh points. The magnitude of motion is exponentially correlated to its distance to the moving boundary.

Although it does require additional calculations to assign mesh stiffness to every node for each frame, the increase in accuracy makes this extra step worthwhile.

It is also interesting to note that for the present model, a node’s translational magnitude based purely on its distance to the moving boundary is not the best solution; a better solution would be to assign a node’s translational magnitude as follows:

\[ \Gamma = \left( \frac{1}{d} \right)^6 \frac{\Delta}{\bar{\Delta}} \]

This solution will provide a mesh stiffness with consideration to the local gap size, resulting in a mesh stiffness profile more aligned with the geometry profile. Unfortunately, the time it would require to program such a function into the simulation will be prohibitively high, therefore the current mesh deformation is governed by the less optimal equation 3.1.

### 3.3.6 Convergence Parameters

Due to the non-linear nature of the present simulation, CFX is only able to approximate a solution by means of iteration, as such, setting up a parameter that controls how many iterations are being calculated is very important. This is true especially for a transient simulation, where a small increase in the numbers of iteration will significantly increase the simulation time.
The present simulation requires a solution in a small localized area; also, the velocity in that area is significantly higher than the inlet. Therefore, in this case, a very high level of convergence is required. However, as previously stated, such an increase in convergence will create a huge burden on the computer, which results in an increase in computational time. For example, when using a 1e-4 residual target, the simulation can be finished in 20 hours, but 5e-5 will require 24 hours, with the simulation results between the two convergence rates differing about 3%. Similarly, if the convergence rate is 5e-4, the simulation will require only 15 hours, but the difference will be off by 10% when compared to the 1e-4 convergence rate. Therefore, in order to maximize the number of simulations done with minimum accuracy loss, the 1e-4 residual target is used.
Chapter 4

Simulation Results

The CFD model was applied to model a typical 3hp scroll compressor design. These simulations were aimed to provide a foundation for future simulations and methods to improve scroll compressor designs by analyzing the gas leakage.

Due to time constraints, these simulations have not been validated for accuracy. The results were taken after 2 revolutions at 2500rpm. The simulations were also run under several unrealistic assumptions that will hopefully be refined in the future version of the simulation.

A series of 11 simulation runs were performed on 3 different gap sizes. A summary of the variables used in these 11 simulations are shown in figure 7.12.
4.1 Typical Velocity Profile

Figure 7.13 shows three components of a velocity profile: Mesh velocity is induced by the motion of the orbiting scroll, which is driven by the following formula:

\[ V_{mesh} = \frac{2\pi r \omega}{60} \]

The relative velocity, which is the difference between the absolute velocity and the mesh velocity is the velocity of interest, on which all the following data will be based.

4.2 Results Summary

Graphs detailing the mean velocity, volume flow rate, density and mass flow rate of the air across the gap are found on figure 7.14 to 7.17. For the volume and mass flow rate, the unit depth is 1mm.

The results from the simulation appear to be reasonable. The leakage appears to be a function of gap size and pressure difference between the inlet and the outlet. As expected, as the gap size increases, the leakage flow rate increases as well. Similarly, as the pressure difference between the inlet and the outlet increases, the gas leakage also increases. Again, due to time constraints no attempts were made to ensure the accuracy of these results.

There is one anomaly in the resulting data, which states that for 0.06mm gap size with an outlet pressure of 1.0 bar, the mass flow rate is actually lower than that of a outlet pressure of 0.5 bar of the same gap size. This is
probably due to simulation errors that will be discussed in the next section.

4.3 Typical Velocity Profile

Figure 7.18 shows a typical velocity gradient relative to the mesh velocity before and after the minimum gap. The maximum velocity occurs not at the minimum gap, but appears to lag behind by about 3 degrees from the direction of travel (counter-clock wise). This shows the compressibility effects near the gap. Also in the zoomed-in region, the effects of no-slip boundary condition can be observed, as the fluid’s relative velocity at the boundary is zero.

4.4 Typical Pressure Profile

Figure 7.19 shows a typical pressure gradient before and after the minimum gap. The pressure change appears to vary with respect to the local wall distance. Figure 7.20 shows two distinct pressure chambers, one at the lower pressure chamber and one at the higher pressure chamber, it is this pressure difference that facilitated the gas leakage.

4.5 Typical Density Profile

Figure 7.21 shows a typical density gradient before and after the minimum gap. Again, the density change appears to vary with respect to the local wall distance.
Chapter 5

Discussion

The results presented in this thesis are by no means complete. This is an unfortunate consequence of the steep learning curve of CFD and short time span of the project. However, there are some outcomes of the present thesis, which are highlighted below:

5.1 Development of a scroll compressor CFD method

As mentioned in the previous sections, to the author’s best knowledge, there are no known methods on using CFD to simulate the inner workings of a scroll compressor. Therefore, perhaps the greatest contribution of this thesis project is the development of such a method, which include the model simplification, meshing, mesh deformation and simulation of a scroll compressor CFD model. Although the present model can only estimate the mass leakage between two adjacent pressure chambers, the model is flexible enough to be
easily augmented to include additional test variables.

In addition to the trivial variable modifications (inlet pressure, outlet pressure, geometry, orbiting scroll angular velocity), new simulation variables can be easily inserted to observe their effects. These variables include:

- Wall Roughness - needed when simulating turbulence effect
- Heat Transfer - Heat effects play a major role in scroll compressor performance. The amount of heat exchange experienced in a scroll compressor will considerably change the properties of the fluid.
- Axial Leakage - By transforming the 2D geometry into 3D geometry and adding axial seals, axial leakage can be simulated.
- Multiple Compression Chambers - The current model only contains two compression chambers separated by a gap. By adding additional compression chambers, a more accurate representation of the scroll compressor can be realized.

5.2 Source of Errors

5.2.1 Modeling Errors

Many assumptions were made in the creation of the simulation model that might have caused inaccuracy in the results, these include:

- Laminar Flow - Under the laminar flow assumption, wall roughness does not have an effect on the mass flow rate. However, from two test
runs with turbulence flow assumption, wall roughness shows an effect on the mass flow. Therefore, the laminar flow assumption has negative effects on the accuracy of the simulation.

- Adiabatic simulation - There is no heat transfer within the system or with the surroundings. Since temperature plays a major effect on air and boundary layers (e.g. thermal deformation), the assumption of adiabatic simulation is therefore, a source of error.

- Assumption that the outlet pressure and gap size is independent - It might not be possible to keep the simulation variables independent as assumed in the current simulation. For example, some manufacturers use the following equations to relate the gap size and pressure difference [1]:

\[
\delta = -9.615 \times 10^{-5} \left( \frac{P_{\text{discharge}}}{P_{\text{suction}}} - 1.67 \right) + 20 \times 10^{-6}
\]

5.2.2 Simulation Errors

The simulation results were taken after only 2 revolutions. In this case, the transient simulation has yet to reach a steady state (In the test run, steady state occurs after the 10th revolution).
Chapter 6

Conclusion

6.1 Computational Fluid Dynamics Modeling

A CFD model of the scroll compressor has been successfully created. The model features a simplified representation of an actual scroll compressor. It is also easy to augment new simulation variables into the new model. The model can be used in future simulations after it has been verified for accuracy.

6.2 Circumferential Leakage Simulation

Several circumferential leakage profiles are simulated using different geometry and pressure differences. The results are again unverified. However the trends of the results fit with the theoretical expectations.
Bibliography


Chapter 7

Figures

Figure 7.1: Basic scroll components
In oder offset 0 degrees
orbit offset 90 degrees
orbit offset 180 degrees
Discharge
orbit offset 270 degrees

Figure 7.2: Basic scroll components 2

Figure 7.3: Basic scroll operation
Figure 7.4: Two Leakage Types

Figure 7.5: Reynold’s number as a function of gap width and flow rate
Figure 7.6: CFD simulation of a Concentric Couette flow solution

Figure 7.7: Simplified Simulation Model
Figure 7.8: Sweeping Meshing
Figure 7.9: Meshing of the Simplified Model
Figure 7.10: Boundary Types
Figure 7.11: Boundary Types
<table>
<thead>
<tr>
<th>Gap Size [mm]</th>
<th>Pressure [bar]</th>
<th>Mean Velocity [m/s]</th>
<th>Volume Flow [m³/s]</th>
<th>Density [kg/m³]</th>
<th>Mass Flow [kg/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.06</td>
<td>1.25</td>
<td>1.02E+02</td>
<td>6.10E-06</td>
<td>2.47E+00</td>
<td>1.51E-05</td>
</tr>
<tr>
<td></td>
<td>1.00</td>
<td>8.83E+01</td>
<td>5.30E-06</td>
<td>2.34E+00</td>
<td>1.24E-05</td>
</tr>
<tr>
<td></td>
<td>0.75</td>
<td>9.35E+01</td>
<td>5.61E-06</td>
<td>2.20E+00</td>
<td>1.24E-05</td>
</tr>
<tr>
<td></td>
<td>0.50</td>
<td>8.45E+01</td>
<td>5.07E-06</td>
<td>1.92E+00</td>
<td>9.76E-06</td>
</tr>
<tr>
<td>0.04</td>
<td>1.25</td>
<td>5.22E+01</td>
<td>2.09E-06</td>
<td>1.83E+00</td>
<td>3.82E-06</td>
</tr>
<tr>
<td></td>
<td>1.00</td>
<td>5.15E+01</td>
<td>2.06E-06</td>
<td>1.80E+00</td>
<td>3.71E-06</td>
</tr>
<tr>
<td></td>
<td>0.75</td>
<td>4.17E+01</td>
<td>1.67E-06</td>
<td>1.72E+00</td>
<td>2.87E-06</td>
</tr>
<tr>
<td></td>
<td>0.50</td>
<td>2.88E+01</td>
<td>1.15E-06</td>
<td>1.55E+00</td>
<td>1.78E-06</td>
</tr>
<tr>
<td>0.02</td>
<td>1.25</td>
<td>1.90E+01</td>
<td>3.80E-07</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>1.00</td>
<td>1.57E+01</td>
<td>3.14E-07</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.75</td>
<td>1.74E+01</td>
<td>3.47E-07</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 7.12: Results Summary

![Velocity Profile Across Gap](image)

Figure 7.13: Typical Velocity Profile
Figure 7.14: Mean Velocity Across the Gap

Figure 7.15: Volume Flow Rate Across the Gap
Figure 7.16: Density Across the Gap

Figure 7.17: Mass Flow Rate Across the Gap
Figure 7.18: Velocity profile for 0.06mm gap size, 1.25bar outlet pressure, zoomed in
Figure 7.19: Pressure profile for 0.06mm gap size, 1.25bar outlet pressure, zoomed in
Figure 7.20: Pressure profile for 0.06mm gap size, 1.25bar outlet pressure, zoomed out

Figure 7.21: Density profile for 0.06mm gap size, 1.25bar outlet pressure, zoomed in