EXPERIMENTAL AND NUMERICAL ANALYSIS OF FLOW AND PRESSURE FIELDS INSIDE A VARIABLE DEPTH SINGLE POCKET HYDROSTATIC BEARING

A Thesis
Presented to
The Graduate Faculty of The University of Akron

In Partial Fulfillment
of the Requirements for the Degree
Master of Science

Frank Horvat
August, 2008
ABSTRACT

This paper studies the experimental and numerical development of flow patterns and pressure profiles inside a variable depth pocket of a hydrostatic journal bearing. The parameters used in this study consist of the pockets aspect ratio, restrictor inlet conditions and shaft velocity. The investigation will be conducted at both low and high shaft rotational speeds.

The flow visualization method uses a full flow field tracking Lagrangian method to track microsphere particles injected in the main stream of pocket’s feedline and thus reconstruct the flow pattern in the pocket. A high intensity pulsed laser is used to create a thin sheet of light that locates and illuminate the particles in a plane, while a long distance microscope (LDM) video camera is used to record digitally the flow images.

The experimental endeavor is supplemented by a numerical simulation which uses CFD-ACE+ (of ESI Corporation, Huntsville, AL) as the computational engine. This package utilizes the full three-dimensional Navier-Stokes equations applied for a steady-state incompressible Newtonian fluid with constant properties flowing in a hydrostatic pocket. The results offered herein present both numerical velocity vector and pressure fields and experimental qualitative flow patterns for the shallow and deep pockets.
ACKNOWLEDGEMENTS

I would like to thank all the people who supported me in this academic endeavor. Including Dr. Braun, who gave me the opportunity to work on this project. I especially would like to thank my mother, Luba and father, Eugene, who I owe a great deal for all their encouragement.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>LIST OF FIGURES</th>
<th>ix</th>
</tr>
</thead>
<tbody>
<tr>
<td>NOMENCLATURE</td>
<td>xiii</td>
</tr>
</tbody>
</table>

## CHAPTER

### I. LITERATURE REVIEW

1.1 Experimental Literature Review .....................................................1

1.2 Numerical Literature Review .........................................................4

### II. SCOPE OF WORK

2.1 Numerical and Experimental Simulations of a Variable depth Pocket .......9

### III. NUMERICAL ALGORITHM

3.1 General Introduction to CFD-ACE+ ..................................................11

3.2 Numerical Methodology Adopted by CFD-ACE+ ......................................12

3.2.1 Discretization ...............................................................12

3.2.2 Velocity-Pressure Coupling ...................................................18

3.2.3 Boundary Conditions .........................................................20

3.2.4 Solution Methods ...........................................................22

### IV. NUMERICAL RESULTS

4.1 Introduction .................................................................26

4.2 Geometry and Boundary Condition Applications ................................27

4.2.1 Geometry .................................................................27
4.2.2 Boundary Condition Applications .......................................................... 29

4.3 Gridding of Geometry .................................................................................. 31

4.4 Results for the Case [Pocket Depth = 0.700, Flow rate = 1.23gpm] .............. 33

  4.4.1 Visualization of Flow Patterns, AR = 1.000 .............................................. 33

  4.4.2 Pressure Profiles, AR = 1.000 .......................................................... 37

4.5 Results for the Case [Pocket Depth = 0.700, Flow rate = 0.615gpm] .............. 38

  4.5.1Visualization of Flow Patterns, AR = 1.000 .............................................. 38

  4.5.2 Pressure Profiles, AR = 1.000 .......................................................... 41

4.6 Results for the Case [Pocket Depth = 0.250, Flow rate = 1.23gpm] .............. 42

  4.6.1 Visualization of Flow Patterns, AR = 0.357 .............................................. 42

  4.6.2 Pressure Profiles, AR = 0.357 .......................................................... 46

4.7 Results for the Case [Pocket Depth = 0.250, Flow rate = 0.615gpm] .............. 46

  4.7.1 Visualization of Flow Patterns ............................................................ 47

  4.7.2 Pressure Profiles .................................................................................. 49

4.8 Results for the Case [Pocket Depth = 0.060, Flow rate = 1.23gpm] .............. 50

  4.8.1 Visualization of Flow Patterns, AR = 0.086 .............................................. 50

  4.8.2 Pressure Profiles, AR = 0.086 .......................................................... 52

4.9 Results for the Case [Pocket Depth = 0.060, Flow rate = 0.615gpm] .............. 53

  4.9.1 Visualization of Flow Patterns, AR = 0.086 .............................................. 53

  4.9.2 Pressure Profiles, AR = 0.086 .......................................................... 56

4.10 Results for the Case [Pocket Depth = 0.030, Flow rate = 1.23gpm] .............. 56

  4.10.1 Visualization of Flow Patterns, AR = 0.043 .............................................. 58

  4.10.2 Pressure Profiles, AR = 0.043 .......................................................... 59
4.11 Results for the Case [Pocket Depth = 0.030, Flow rate = 0.615gpm] ..........60

4.11.1 Visualization of Flow Patterns, AR = 0.043 .......................................60

4.11.2 Pressure Profiles, AR = 0.043 ..........................................................62

4.12 Results for the Case [Pocket Depth = 0.015, Flow rate = 0.615gpm] ..........63

4.12.1 Visualization of Flow Patterns, AR = 0.021 .......................................63

4.12.2 Pressure Profiles, AR = 0.021 ..........................................................66

4.13 Results for the Case [Pocket Depth = 0.015, Flow rate = 0.615gpm] ..........66

4.13.1 Visualization of Flow Patterns, AR = 0.021 .......................................68

4.13.2 Pressure Profiles, AR = 0.021 ..........................................................69

V. DESCRIPTION OF EXPERIMENTAL INSTALLATION .............................70

5.1 Introduction ....................................................................................................70

5.2 Test Loop .......................................................................................................70

5.3 Drive System ..................................................................................................70

5.4 Test Section (Apparatus) ..............................................................................71

5.5 Variable Depth Pocket ..................................................................................72

5.6 Data Acquisition system ..............................................................................74

5.7 Visualization System ....................................................................................75

5.7.1 Creation of the Laser Sheet .................................................................75

5.7.2 Visualization of the Test Section ............................................................76

5.8 Working Fluid ...............................................................................................77

VI. CALIBRATIONS .............................................................................................78

6.1 Calibration of the Pressure Transducers .......................................................78

6.2 Calibration of Motor .....................................................................................80
LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1</td>
<td>Three dimensional view of control volume (Computational Cell)</td>
<td>12</td>
</tr>
<tr>
<td>3.2</td>
<td>Two dimensional computation cell (Control Volume)</td>
<td>14</td>
</tr>
<tr>
<td>3.3</td>
<td>Computational cell located next to a boundary</td>
<td>21</td>
</tr>
<tr>
<td>3.4</td>
<td>Flowchart for computational calculation</td>
<td>23</td>
</tr>
<tr>
<td>4.1</td>
<td>Matrix of numerical parameters</td>
<td>27</td>
</tr>
<tr>
<td>4.2</td>
<td>Nodal points and pocket geometry</td>
<td>28</td>
</tr>
<tr>
<td>4.3</td>
<td>Overall view of geometry and boundary conditions</td>
<td>28</td>
</tr>
<tr>
<td>4.4</td>
<td>Axial and circumferential view of the pocket</td>
<td>30</td>
</tr>
<tr>
<td>4.5</td>
<td>Gridding of geometry</td>
<td>32</td>
</tr>
<tr>
<td>4.6</td>
<td>General view of 0.700” pocket depth (AR = 1.000) showing flow patterns within the pocket. This case examines a flow rate of 1.23gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm</td>
<td>35</td>
</tr>
<tr>
<td>4.7</td>
<td>General view of Pressure profiles for a 0.700” pocket depth (AR = 1.000) with a flow rate of 1.23gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm</td>
<td>36</td>
</tr>
<tr>
<td>4.8</td>
<td>General view of 0.700” pocket depth (AR = 1.000) showing flow patterns within the pocket. This case examines a flow rate of 0.615gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm</td>
<td>39</td>
</tr>
<tr>
<td>4.9</td>
<td>General view of Pressure profiles for a 0.700” pocket depth (AR = 1.000) with a flow rate of 0.615gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm</td>
<td>40</td>
</tr>
<tr>
<td>4.10</td>
<td>General view of 0.250” pocket depth (AR = 0.357) showing flow patterns within the pocket. This case examines a flow rate of 1.23gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm</td>
<td>43</td>
</tr>
</tbody>
</table>
4.11 General view of Pressure profiles for a 0.250” pocket depth (AR = 0.357) with a flow rate of 1.23gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm..................................................................................44

4.12 General view of 0.250” pocket depth (AR = 0.357) showing flow patterns within the pocket. This case is examines a flow rate of 0.615gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm........................................................................48

4.13 General view of Pressure profiles for a 0.250” pocket depth (AR = 0.357) with a flow rate of 0.615gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm..................................................................................49

4.14 General view of 0.060” pocket depth (AR = 0.086) showing flow patterns within the pocket. This case is examines a flow rate of 1.23gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.................................................................51

4.15 General view of Pressure profiles for a 0.060” pocket depth (AR = 0.086) with a flow rate of 1.23gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm..................................................................................52

4.16 General view of 0.060” pocket depth (AR = 0.086) showing flow patterns within the pocket. This case is examines a flow rate of 0.615gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.................................................................54

4.17 General view of Pressure profiles for a 0.060” pocket depth (AR = 0.086) with a flow rate of 0.615gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm..................................................................................55

4.18 General view of 0.030” pocket depth (AR = 0.043) showing flow patterns within the pocket. This case is examines a flow rate of 1.23gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.................................................................57

4.19 General view of Pressure profiles for a 0.030” pocket depth (AR = 0.043) with a flow rate of 1.23gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm..................................................................................58

4.20 General view of 0.030” pocket depth (AR = 0.043) showing flow patterns within the pocket. This case is examines a flow rate of 0.615gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.................................................................61

4.21 General view of Pressure profiles for a 0.030” pocket depth (AR = 0.043) with a flow rate of 0.615gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm..................................................................................62
4.22 General view of 0.015” pocket depth (AR = 0.021) showing flow patterns within the pocket. This case examines a flow rate of 1.23gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm..............................................................64

4.23 General view of Pressure profiles for a 0.015” pocket depth (AR = 0.021) with a flow rate of 1.23gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm..........................................................65

4.24 General view of 0.015” pocket depth (AR = 0.021) showing flow patterns within the pocket. This case examines a flow rate of 0.615gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm..............................................................67

4.25 General view of Pressure profiles for a 0.015” pocket depth (AR = 0.021) with a flow rate of 0.615gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm..............................................................68

5.1 Overall view of test loop.................................................................71

5.2 Cross section and front view of test section........................................72

5.3 Variable depth pocket assembly. ......................................................73

5.4 Sketch of the vision system with LDM-long distance microscope........76

6.1 Calibration curves for Entran pressure transducers.............................79

6.2 Calibration of Baldor variable speed motor.......................................80

7.1 Locations used to center both the housing and rotor..........................82

7.2 Experimental test loop.................................................................84

8.1 Matrix of experimental parameters..................................................86

8.2 Qualitative view of pressure dominated flows within a pocket.............87

8.3 Qualitative view of Couette dominated flows within a pocket..............88

8.4 General view of 0.700” pocket depth (AR = 1.000) showing flow patterns within the pocket. This case examines a flow rate of 1.23gpm with shaft speeds of 0, 200, 1000 and 1500rpm..............................................................91

8.5 General view of 0.700” pocket depth (AR = 1.000) showing flow patterns within the pocket. This case examines a flow rate of 0.615gpm with shaft speeds of 0, 200, 1000 and 1500rpm..............................................................94
8.6 General view of 0.250” pocket depth (AR = 0.357) showing flow patterns within the pocket. This case examines a flow rate of 1.23gpm with shaft speeds of 0, 200, 1000 and 1500rpm.................................................................96

8.7 General view of 0.250” pocket depth (AR = 0.357) showing flow patterns within the pocket. This case examines a flow rate of 0.615gpm with shaft speeds of 0, 200, 1000 and 1500rpm.................................................................99

8.8 General view of 0.060” pocket depth (AR = 0.086) showing flow patterns within the pocket. This case examines a flow rate 1.23gpm with shaft speeds of 0, 200, 1000 and 1500rpm.................................................................101

8.9 General view of 0.060” pocket depth (AR = 0.086) showing flow patterns within the pocket. This case examines a flow rate 0.615gpm with shaft speeds of 0, 200, 1000 and 1500rpm.................................................................103

9.1 Deep pocket conclusion 1.................................................................108

9.1 Deep pocket conclusion 2.................................................................109

9.3 Deep pocket conclusion 3.................................................................110
NOMENCLATURE

Re = Clearance based Reynolds number = $\frac{RoC}{v}$

$u =$ velocity in the circumferential direction, in./sec

$v =$ velocity in the radial direction, in./sec

$x =$ circumferential coordinate, in.

$y =$ radial coordinate, in.

$z =$ axial coordinate, in.

$\rho =$ fluid density, lb-sec$^2$/in.$^4$

$\omega =$ shaft (rotor) rotational speed, rad/sec

$C =$ radial clearance, in.

$r =$ radius of shaft
CHAPTER I

LITERATURE REVIEW

1.1 Experimental Literature Review

The lubrication problems associated with bearing design have been a classical fluid dynamics problem for many years. Researchers have contributed countless time and effort to the optimization and development of hydrostatic and hydrodynamic bearings. Speen (1962) examined the pressure and flow characteristics of an externally pressurized gas lubricated bearing. He compared the axial and radial support parameters for two types of integrated journal-thrust bearings to that of a conventional journal, which is equipped with either a journal of thrust surface. Pressure and flow fields were also analyzed in all 3 cases using fluorescent oil. Loading conditions ranged from light (where light loading was defined as any “loading condition where the gas film pressure acting on the thrust surface is considerable less than the pressure acting on the journal surface”), to moderate and heavy thrust loading. One of the three types of the integrated journal bearing was a basic configuration where the journal and thrust surfaces were connected at 45 degrees. The second type was slightly modified to accommodate a square groove, which was milled into the 45 degree intersection, where the journal and thrust surfaces join. The results showed improved performance in the integrated journal bearing, which has both journal and thrust surfaces, over that of the conventional bearing,
which has either a journal or a thrust surface. There was significant improvement in load support, stability and stiffness. The third tested (square groove design) showed that, if properly designed, stability issues can be avoided and contribute to the overall bearing performance. Betts and Roberts (1968) studied the theoretical and experimental effects of a liquid-lubricated hydrostatic journal bearing. They used a semi-empirical theory to describe the performance of the bearing where the flow was in its turbulent regime. This paper focuses on specific design parameters for cylindrical bearing, which is equipped with two circumferential row of eight, equally spaced jets. The experiment utilized a Reynolds number for flow in the clearance, which is characterized by the equation,

\[ \frac{8 \rho Q}{\pi \eta D} \]

The experiment covered Reynolds numbers between 1700 and 7000. There were two types of working fluids in this case. Paraffin was the first, which is a low viscosity lubricant used to cover Reynolds numbers around 1700. The second type of working fluid used water to obtain measurements within the turbulent region where the Reynolds number was in excess of 7000. The results focused on the dimensionless parameters Ko, which is defined as the ratio of pressure downstream of jet over the inlet supply pressure. They concluded that their semi-empirical study covered both the viscous and turbulent regimes. The optimal load carrying capacity was obtained for a Ko of around 0.5. An additional conclusion is that cavitation needs to be considered for bearing design when Ko values are much lower than 0.5. Koseff and Street (1984) investigated the three-dimensional lid-driven cavity, to study the effects of the end wall and the size of the downstream secondary eddy. In the cavity the size of the downstream secondary eddy (DSE) was predicted by use of a two-dimensional numerical solution and then compared by experimental measurements. Both the numerical solution and the experimental results
showed close agreement, but for a cavity with a spanwise-aspect-ratio (SAR) of 1:1, the experiments yielded opposite results from the numerical work. Their observations showed the DSE decreased in size with increasing Reynolds numbers for spanwise-aspect-ratios of 2:1 and 1:1 and the opposite effect (DSE increased in size) for a SAR of 3:1. Another major structure in the three-dimensional cavity was the development of the ‘corner vortex’. The corner vortex developed from the interaction of the shear and pressure forces acting on the recirculation fluid within the pockets cavity. The last major observation focused on the spiraling motion of the fluid. Originating from the DSE, the counter-clockwise spiral motion moved to the end wall toward the corner vortex for a Reynolds number of 5000 and a SAR of 3:1. The authors concluded that the corner vortices located at the end walls in the near vicinity of the DSE were a major influence on the size of this eddy. Ho and Chen (1984) investigated the pressure distributions inside a six-pocket journal bearing with non-uniform depths. The six equally spaced pockets were subject to variable operating parameters, which include shaft rotational speed and lubricant supply pressure. Results determined as both load and flow increased, pressure increased. The conclusion determined that the bearing acted like a hybrid bearing, where the film pressures exhibited both hydrodynamic and hydrostatic effects. Scharrer and Hibbs (1990) presented results for flow coefficients for the orifice of a hydrostatic bearing. The authors focus was on the effects of pressure ratio and Reynolds numbers on flow coefficients. The assumption of using standard data for the orifice is widely used, but a hydrostatic bearing is considered different. The difference lies due to the large length to diameter ratio as well as the downstream obstruction (the bearings shaft). They also claim that standard data for the loss coefficient is not enough because of the different
and complex flow field within the pocket. Plots of discharge coefficient vs. Reynolds number for oil, Freon and GN2 for different orifice diameters were also analyzed. The author’s concluded that, in both the laminar and turbulent flow, the flow coefficient was highly dependent on Reynolds number.

1.2 Numerical Literature Review

Numerical simulation of a lid driven cavity has been a classical numerical problem which has been investigated by many researchers. Donovan (1969) studied the unsteady flow in a two-dimensional square cavity. The purpose of this study was to compare the solution of the steady state Navier-Stokes equation with that of the unsteady Navier-Stokes equations for large time periods. In his numerical solution, the finite difference method was employed for a constant density model. The results were presented in both numerical and visual comparisons. The numerical simulation for the unsteady flow inside a square cavity used both a Reynolds number of 100, which was primarily based on the length and velocity of the moving surface of the lid, and the viscosity of the fluid. The results of the numerical simulation were compared to the experimental portion of this paper. The comparison focused on the position of the vortex center, velocity profile and pressure profile within the square cavity. For large times, the velocity profiles for the unsteady equations were compared to steady state equations. For this comparison, the center position of the vortex inside the square cavity showed excellent agreement with one another, which led to the conclusion that the unsteady solution is numerically correct for large times. The same was also apparent in the comparison for the velocity and pressure profiles. Braun, Choy and Zhou (1993) examined the parameters of a hydrostatic pocket aspect ratio, supply orifice position and
attack angles on steady-state flow patterns. The influence of the Reynolds number and jet strength on flow patterns as well as both pressure and shear on the moving lid were also investigated. This study shows the effects of both the dynamic and geometric parameters that create the hydrostatic pockets flow patterns behavior. This study proposes to solve a combination of the three basic cases, flow in a closed lid driven cavity, flow in an open cavity driven by a shear layer and flow in a cavity with a penetrating jet. The numerical analysis consists of evaluating the aspect ratio, jet position and inclinations as well as the Reynolds number, jet strength, flow patterns, pressure and shear characteristics. They found that the magnitude of the static pressure increases with increasing jet force, which causes a combined effect creating a back step in pressure in the upstream portion and a Rayleigh step in the downstream portion of the pocket. Another parameter studied was the position of the jet angle. The position and angle of the jet strongly affected the lid driven shear layer and the pressure effects within the hydrostatic pocket. It caused recirculation zones within the pocket to change size, shape and position. It was found that, as the parameter K (jet angle) was changed, that the normal forces on the driving lid were strongly affected, while the jet position had little or no effect. As the aspect ratio increased (AR>1) the combined effects of the shear layer and the jet became separated. The shearing forces acting on the driving lid increased exponentially for aspect ratios less than 0.1 and decreased asymptotically to a finite value for aspect ratios greater than 1.0. Another important finding for the cavity aspect ratio of 0.1, was that a decrease in the lid clearance by a factor of 2, caused an increase in pressure acting on the lid by a factor of 5. This does not hold true for the deep cavity, since the jets effects cannot travel up the pocket to impact the driving lid. Braun, Zhou
and Choy (1994) observed the transient flow patterns and pressure characteristics in a hydrostatic pocket. The pocket is a square, lid driven open cavity with a jet penetrating from the bottom. The main focus of this paper is to solve three cases, jet penetration, time dependent motion of the lid associated with the time varying clearance as well as the interaction of the shear layer within the cavity and clearance. Besides time there are four other parameters that are included: the jet strength, the vertical vibration of the lid which will be a function of the vibration amplitude and the Reynolds number based on the lid velocity. It was observed that for the flow driven into the pocket by the dominate pressure forces with no shaft rotation caused the location of the upstream secondary eddy (USE) and the downstream secondary eddy (DSE) to create a symmetrical velocity profile. Other observations showed how the shearing effects overtake the Poiseulle effects in the time span of $t=0.0$ to $t=1.0$ seconds. Plots of the transient development of the velocity profiles and the flow profiles are shown for the pocket in the time varying lid clearance analysis. Braun and Dzodzo (1995) studied the three dimensional flow and pressure patterns in a single pocket of a hydrostatic bearing. The shallow pocket coupled with a long slender restrictor was used to observe both the secondary vertical cells (SVC) and the modified vertical cells (MOVC). The physical analysis involved a high shear Couette flow interacting with a low pressure, Poiseuille flow. The result showed a modified vortical cell directly in front of the restrictor exit, greatly influencing the local flow coefficient and the effective diameter of the restrictor. The upstream portion of the pocket showed no fluid exiting the upstream clearance due to a fluid turn around zone. This turn around zone (TAZ) developed due to the large shear layer, caused from the shaft rotation. The shear layer entrains the fluid exiting the upstream portion of the
pocket. This entrainment caused the fluid the physically change direction and flow with the shaft toward the downstream exit. Braun, Dzodzo and Lattime (1996) expanded their investigation to include some qualitative and quantitative comparisons for flows in deep pockets. Due to flow exiting the restrictor and the shear layer carried by the shaft, the flow is unmistakably three dimensional. The quantitative portion is obtained through the use of the full three dimensional Navier-Stokes equation. The Poiseuille effect comes from the pressure differential between the inlet of the restrictor and the outlet of the pocket. The Couette flow is caused by the shear layer generated by the rotation of the shaft. The shear layer from the upstream exit is responsible for the recirculation (turn around zone) the fluid experiences. When the restrictor inlet pressure is increased (Poiseuille dominated) the jet from the restrictor, impinges the shaft more powerfully. Due to this increase in the restrictors jet effect, a small recirculation zone under the shaft can be seen next to the turn around zone in the upstream portion of the pocket. In Couette dominated flow, since no fluid exits the upstream clearance, there is a Rayleigh effect which forms in downstream exit of the pocket due to the high volume of fluid exiting the pocket. In the shallow pocket, the assumption for constant pressure across the pocket is not valid. However for the deep pocket, constant pressure throughout the pocket is valid. Braun, Dzodzo and Hendricks (1996) studied the change in the radius of curvature of three different pocket/land joints (PLJ) inside a shallow hydrostatic pocket. The parameters used were Reynolds numbers based on the shaft velocity and jet strength (restrictor). The interaction of the shear layer with the cavity flow as well as the flow patterns, pressure profiles, inertial pressure drops and velocities inside the pocket were analyzed. They found the largest resistance and highest pressure inside the pocket came
from the square exit of the PLJ. When the PLJ was rounded there was lower pressure inside the pocket. As for the flow pattern on the downstream portion of the land, the round profile of the PLJ caused the recirculating vortex to decrease in height. On the upstream portion of the land the fluid turn around zone moved upward toward the rotating shaft. As for the square PLJ, the fluid turn around zone was further from the shaft and closer to the bottom of the land. For the Poiseuille dominated flow the pressure inside the pocket was relatively constant with a peak pressure from the impinging jet coming from the pockets restrictor. The Couette dominated flow produced a pressure ramping effect throughout the pocket until the downstream clearance of the pocket. The inertial pressure drop was smaller for the Couette flow than the Poiseuille flow, which is about 4.5 times larger. The combination of the impinging jet and Couette dominated flow created a Main vortical cell inside the pocket. Where, as in the Poiseuille dominated flow, there was no Main vortical cell inside the pocket.
CHAPTER II

SCOPE OF WORK

2.1 Numerical and Experimental Simulations of a Variable depth Pocket

The overall goal of this work is to further investigate the dynamics of flow through a single, variable depth pocket of a hydrostatic journal bearing. Both experimental and numerical work is done to compare qualitatively and quantitatively the velocity and pressure profiles present within the pocket. The experimental work is meant to both validate the numerical simulations and provide additional physical insight into the flow in a hydrostatic pocket. The bearing clearance region and both upstream and downstream lands were the major areas of interest. Two major phenomena were investigated. The first, involved the transition from Poiseuille dominated flow to inertial (Couette) dominated flow. The second, dealt with the study of flow patterns on the position and movement of vortical cells developed within the pocket due to variable parameter changes.

Numerical analysis was performed using the computational engine CFD-ACE+, (ESI corporation, Huntsville, AL). The simulations visualized both the velocity and pressure profiles within the pocket. These two variables were analyzed while varying system parameters which included, the inlet conditions (varying the restrictor inlet flow), pocket depth and shaft speed.
The experimental portion of this thesis used a test fixture equipped with an adjustable depth pocket piston insert. Using a pulsed laser combined with a system of lenses and mirrors, a thin plane sheet of light was created. The mirrors could be adjusted to scan the circumferential cross section of the pocket. In the axial direction a full field particle tracking method is used to follow the trajectories of the seeded particles within a thin illuminated plane. This plane allows for the formation of a clear two-dimensional image of the flow inside the pocket in different axial sequential planes.

Two types of methods will be used in the analysis of the variable depth pocket. The first is a qualitative analysis, which will focus on the flow patterns within the pocket. The flow patterns will be analyzed for different inlet conditions (varying the restrictor inlet flow and pressure), pocket depth and shaft speed. The second is a quantitative analysis, which will focus on two parameters. The pressures within the pocket including the upstream and downstream lands (clearance) and the velocity profile within the pocket itself. The pressures and inlet restrictor velocity in the pocket cavity will be measured using pressure transducers and a positive displacement flow meter. Both the pressures and velocity profiles will be discussed for the same conditions as in the qualitative analysis. Both of these methods and the comparison of numerical and experimental work will allow a greater understanding of the dynamics within the variable depth, single pocket hydrostatic bearing.
CHAPTER III

NUMERICAL ALGORITHM

3.1 General Introduction to CFD-ACE+

The numerical results are obtained using CFD-ACE+ (ESI corporation, Huntsville, AL), a commercial computational engine used for multi-physics computational analysis. The program has a powerful preprocessor, CFD-GEOM, which allows the input of complicated geometries as well as a post processor CFD-VIEW through which results can be presented in two or three dimensional format. The CFD-ACE+ algorithm, is employed to solve the full, compressible Navier-Stokes equations in a Cartesian system of coordinates using body fitted coordinates. The solution domain is divided into many cells called control volumes. Using a finite volume approach, the differential equations are turned into a system of algebraic equations and are numerically integrated over each of the computational cells using a collocated cell-centered variable arrangement, where all dependent variables and material properties at the cell’s center are stored. For the momentum equations, a first order upwind scheme is used. For pressure calculations CFD-ACE uses the SIMPLEC scheme which was originally proposed by both Van Doormal and Raithby (1984) which is an enhancement of the SIMPLE algorithm of Patankar and Spalding (1972). The equation for pressure correction is obtained from the continuity equation, and the scheme of velocity and pressure calculations is fundamentally iterative in nature. The equations for pressure correction
will be dealt with in this chapter. The residuals of continuity and momentum equations are required to be below $10^{-4}$. To improve convergence the under-relaxation factors were adjusted individually for each one of the variables $(u,v,w,p)$. For both velocity and pressure the typical under relaxation factors vary between 0.1 and 0.7. Because the convergence criteria were not satisfied, these relaxation factors were chosen to help obtain a convergent numerical solution.

3.2 Numerical Methodology Adopted by CFD-ACE+

The Numerical Methodology discussed in these next sections reference the CFD-ACE+ Users Manuals (2002).

3.2.1 Discretization

The computational engine CFD-ACE+ utilizes a finite volume approach to solve each of the governing differential equations. The solution domain is integrated into a number of cells which are defined as control volumes, Figure 3.1. Each one of the governing PDE’s are integrated over each cell (control volume), where all dependent variables are stored in the geometric center of each cell denoted by point $P$.

Figure 3.1 Three dimensional view of control volume (Computational Cell)
The governing PDE’s are expressed in the form of a generalized transport or momentum equation for a generic quantity, $\phi$.

$$\frac{\partial \rho \phi}{\partial t} + \nabla \cdot (\rho \vec{V} \phi) = \nabla \cdot (\Gamma \nabla \phi) + S_\phi$$  \hspace{1cm} (3.1)

The above momentum equation has all four terms active, which are the transient, convection, diffusion and source term. CFD-ACE+ denotes this equation as the generic conservation equation for a quantity $\phi$.

CFD-ACE+ utilizes the finite volume approach, where the governing equations are numerically integrated over each of the computational cells. After the finite volume integration is applied to Eq. 3.1, Eq. 3.2 is formed.

$$\int_{\delta} \frac{\partial (\rho \phi)}{\partial t} d\delta + \int_{\delta} \nabla \cdot (\rho \vec{V} \phi) d\delta = \int_{\delta} \nabla \cdot (\Gamma \nabla \phi) d\delta + \int_{\delta} S_\phi d\delta$$  \hspace{1cm} (3.2)

The integrated transient term of Eq. 3.2 is discretized to yield Eq. 3.3

$$\int_{\delta} \frac{\partial (\rho \phi)}{\partial t} d\delta = \frac{\rho \phi \delta - \rho^0 \phi^0 \delta^0}{\Delta t}$$  \hspace{1cm} (3.3)

The integrated transient term includes superscript “0” terms which denotes the old time step, while terms without the superscript denote the current time step. The term $\delta$ represents the cell volume and may change with respect to time if moving grids are used in to computational domain.

The convection term within the momentum equation becomes discretized to become:

$$\int_{\delta} \nabla \cdot (\rho \vec{V} \phi) d\delta = \int_{A} \rho \phi (\vec{V} \cdot \vec{n}) dA = \sum_{c} \left( \rho_c \phi_c V_{c} n \right) A_c = \sum_{c} C_c \phi_c$$  \hspace{1cm} (3.4)
Where the term \( C_e \) denotes the mass flux term and the subscript term “e” is indicative of an arbitrary cell face (Figure 3.2). The fact that the dependent variable \( \phi \) is located at the cells center, interpolation schemes need to be applied to determine the cell face values at \( e \).

For ease of visualization Figure 3.2 shows a two dimensional computational cell, which will be used to transform the governing differential equations to their discretized counterparts. Points “P” and “E” are defined as the cell centers of two adjacent cells where all the dependent variables information are stored.

Some types of interpolation or discretization schemes utilized by CFD-ACE+ include the First-order upwind, central difference and the second-order upwind scheme.

In implementing the First-order upwind scheme, the computational cell face \( \phi_e \) is taken to be equal to the generic value \( \phi \) in either the upstream (\( \phi_P \)) or downstream (\( \phi_E \)) direction. This depends on the direction of the flow at the cell face whether it is in the normal direction or opposite the normal direction. Referencing Figure 3.2, if the
direction of flow is normal to face e, then the information at cell face “e” will be equal the upstream node point “P”, \( \phi_{e}^{UP} = \phi_{P} \). \( \phi_{e} \) is expressed as:

\[
\phi_{e}^{UP} = \begin{cases} 
\phi_{P} & if \nu_{e} > 0 \\
\phi_{E} & if \nu_{e} < 0 
\end{cases}
\]  

(3.5)

The First-order upwind scheme has only first order accuracy, but it is one of the most stable schemes.

In the Central Difference Scheme, the cell face \( \phi_{e} \), is evaluated arithmetically by averaging the values at the cell centers “P” and “E”. The major assumption states that the value for \( \phi \) varies linearly between the two cells centers. The linearity allows a set of linear equations to be developed for the dependent variables. These set of equations are solved using Cramer’s rule. In the end we end up with a type of shape function described mathematically in the following equation, Eq. 3.6.

\[
\phi_{e}^{CD} = \gamma_{e} \phi_{P} + (1 - \gamma_{e}) \phi_{E} 
\]  

(3.6)

Where \( \gamma_{e} \) is described as the geometrical weighting function at cell face \( e \).

Conventional central difference schemes may give rise to non-physical oscillations in the numerical solution and may become unstable in the evaluation of convection terms within the discretized momentum equation. To compensate for this misfortune, artificial or numerical damping has been added to Eq. 3.6 to improve upon the stability of the scheme. The preceding equation thus becomes,

\[
\phi_{e}^{CD} = \alpha \phi_{e}^{UP} + (1 - \alpha) \phi_{e}^{CD}
\]  

(3.7)

Where \( \alpha \) is denoted as a blending factor. This blending factor is what combines both the first-order upwind scheme and the central difference scheme to produce a more stable solution. This equation has an order of accuracy between 1 and 2. This is due to how the
blending factor is chosen. If $\alpha$ is set equal to zero, then the preceding equation becomes a conventional second-order accurate, central difference scheme. However if $\alpha$ is set equal to one, then the preceding equation results in a first-order accurate upwind scheme.

Second-Order Upwind Scheme uses linear interpolation between two upstream cells, instead of one cell as mentioned in the previous schemes. For implementation of this scheme to unstructured grids quadrangle cells are treated differently from the triangle cells. The second-order upwind scheme gives,

$$\phi_e^{SUD} = \begin{cases} f(\phi_{e3} \phi_p) i fV_e^n > 0 \\ f(\phi_{eE} \phi_p) i fV_e^n < 0 \end{cases}$$

(3.8)

As in the case for the central difference scheme and blending factor is added for numerical damping needed for a stable solution.

$$\phi_e = \alpha \phi_e^{UP} + (1 - \alpha) \phi_e^{SUD}$$

(3.9)

The diffusion terms within the general momentum equation becomes discretized in the following manner.

$$\int_\delta (\nabla \phi) \cdot d\delta = \int_A \nabla \phi \cdot \vec{n} dA = \sum_e \frac{\partial \phi}{\partial n} A_e$$

(3.10)

Where the three unit vectors $\vec{n}$, $\vec{e}$ and $\vec{\tau}$ are defined in figure 3.2,

$$\frac{\partial \phi}{\partial n} = \frac{1}{\vec{n} \cdot \vec{e}} \left( \frac{\partial \phi}{\partial e} - \vec{e} \cdot \vec{\tau} \frac{\partial \phi}{\partial \tau} \right)$$

(3.11)

When substituted into Eq.3.10, the diffusion term becomes,

$$\int_\delta (\nabla \phi) \cdot d\delta = \sum_e \frac{\Gamma_e}{\vec{n} \cdot \vec{e}} \left( \frac{\partial \phi}{\partial e} \right) A_e - \sum_e \frac{\vec{\tau} \cdot \vec{e} \Gamma_e}{\vec{n} \cdot \vec{e}} \left( \frac{\partial \phi}{\partial \tau} \right) A_e$$

(3.12)
The discretized form of the partial derivatives become:

\[
\left( \frac{\partial \phi}{\partial e} \right)_{e} = \frac{\phi_{E} - \phi_{P}}{\delta_{P,E}} \tag{3.13}
\]

\[
\left( \frac{\partial \phi}{\partial \tau} \right)_{e} = \frac{\phi_{C2} - \phi_{C1}}{\delta_{C2,C1}} \tag{3.14}
\]

Where \( \delta_{P,E} \) and \( \delta_{C2,C1} \) represent the linear distance between cell centers “E” and “P”, and “C2” and “C1”, respectively.

The source term is a function of \( \phi \) and it is linearized in the following manner.

\[
S_{\phi} = S^{U} + S^{P} \phi \tag{3.15}
\]

Where \( S^{P} \) is defined as a negative value. Both \( S^{U} \) and \( S^{P} \) are considered functions of \( \phi \). These terms are considered to be the actual value of \( \phi \) at the previous time step. If the linearized source term is integrated over the control volume the results is,

\[
\int_{V} S_{\phi} d\zeta = S^{U} + S^{P} \phi_{P} \tag{3.16}
\]

Where \( S^{P} = S^{P} \zeta \) and \( S^{U} = S^{U} \zeta \)

Any further details regarding source term linearization are referenced from Patankar(1980).

When all of the numerically integrated transient, convection, diffusion and source terms from above are put together, these terms would result in a linear equation of the form:

\[
\left( a_{p} - S^{P}_{p} \right) \phi_{p} = \sum_{nb} a_{nb} \phi_{nb} + S^{U}_{p} \tag{3.17}
\]
Where the subscripts ‘nb’ denote the values at adjacent cells and are known as link coefficients. This finite difference equations (FDE) form a nonlinear system of equations due to the link coefficients because they are a function of the dependent variable, \( \phi \). However, if the FDE is derived for each individual computational cell, the would result in a set of coupled nonlinear algebraic equations. Because there is no direct matrix inversion to solve these systems of algebraic equations, an iterative process must be initiated until a convergent solution is obtained.

3.2.2 Velocity-Pressure Coupling

The discretization of the continuity equation, which governs mass conservation, is handled differently due to the fact that it is not written in the same form as the generalized convection-diffusion equation. This conservation equation is then utilized by CFD-ACE+ by a method proposed by Rhie and Chow (1983), to determine the pressure fields.

The mass conservation equation is written as follows,

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \vec{V} = \dot{m}
\] (3.18)

Where \( \dot{m} \) represents the mass flow. If on integrates the above equation over the control volume (cell) in figure 3.2, then:

\[
\frac{\rho \delta - \rho^0 \delta^0}{\Delta t} + \sum_{e} \rho_e V_e^n A_e = \dot{m} \delta
\] (3.19)

Referring to the above equation, \( V_e^n \) is the velocity component in the normal direction at cell face \( e \). This velocity component can be written as,

\[
V_e^n = u_e n_x + v_e n_y + w_e n_z
\] (3.20)
Due to velocity and fluid density information being stored at the cells centers, interpolation between adjoining cell faces is needed. The problem is that linear interpolation decouples the velocity and pressure fields. To circumnavigate this problem, a method of evaluating the mass flux at the cells faces is used in conjunction with averaging the momentum equation at the cells faces, thus relating the cell face velocity to that of the local pressure gradient. This was proposed by Rhie and Chow (1983) and later refined by Peric (1988).

Now, since all three components of the velocity equations are known, pressures must be determined. Since there is no current governing PDE strictly for pressures, a variety of schemes have been developed to determine this unknown variable. For pressure calculations CFD-ACE uses the SIMPLEC scheme which was originally proposed by Van Doormal and Raithby (1984), which is an enhancement of the SIMPLE algorithm of Patankar and Spalding (1972). The equation for pressure correction is obtained from the continuity equation, and the joint scheme of velocity and pressure calculations is fundamentally iterative in nature. The momentum equation can be written in a finite difference form as,

\[
a_p u_p = \left( \sum_{ab} a_{nb} u_{ab} + S_U \right)_p - \left( \sum_{e} P_e A_e n_{xe} \right)_p
\]

(3.21)

The subscript “P” denotes that the equation is written for cell center (see figure 3.2). In the above equation if the pressure field is known then the equation can be solved for the velocity \( u \). Since the pressure profile is not known, an iterative procedure must be employed to determine both the pressure field and the velocity.

\[
a_p u_p = \left( \sum_{nb} a_{nb} u_{nb}^* + S_U \right)_p - \left( \sum_{e} P_e^* A_e n_{xe} \right)_p
\]

(3.22)
In the above equation the term \( P^* \) is a guessed pressure term. Once pressure is guessed then the equation can be solved for \( u^* \) which is the velocity. Since, the pressure \( P^* \) has been a guessed value, \( u^* \) will not satisfy the continuity equation. The basic idea is to find correction terms to add to the guessed terms
\[
\begin{align*}
    u &= u^* + u' \\
    P &= P^* + P'
\end{align*}
\] (3.23) (3.24)

Where \( u^* \) and \( P^* \) is the guessed value and \( u' \) and \( P' \) are the correcting values. An expression for \( u_p^* \) can be obtained by subtracting Eq.3.22 from 3.21 to obtain,
\[
\begin{align*}
    a_p u_p' &= \left( \sum_{ab} a_{nb} u_{ab}' \right)_p - \left( \sum_{e} P_{e}' A_{e} n_{se} \right)_p \\
    &= \sum_{e} P_{e}' A_{e} n_{se} - \sum_{ab} a_{nb} u_{ab}'
\end{align*}
\] (3.25)

If \( u_{nb}' \) is approximated by \( u_p' \) which in turn will give an expression for \( u_p^* \)
\[
    u_p^* = \frac{-1}{a_p - \sum_{ab} a_{nb} \left( \sum_{e} P_{e}' A_{e} n_{se} \right)_p}
\] (3.26)

Then substitute all the corrected velocity components, Eq.3.26 into Eq.3.25, the pressure correction equation is written as,
\[
\begin{align*}
    a_p P_p' &= \sum_{nb} a_{nb} P_{nb}' + S_m
\end{align*}
\] (3.27)

where \( S_m \) represents the mass correction (mass source) in the control volume:
\[
S_m = \frac{\rho^0 \vartheta^0 - \rho_p^* \vartheta}{\Delta t} - \sum_{e} \rho_{e} V_{e}^n A_{e}
\] (3.28)

The SIMPLExC procedure is outlined as follows
1) Guess a pressure field \( P^* \).
2) Obtain \( u^* \), \( v^* \), and \( w^* \) by solving discretized momentum equation (3.17).
3) Obtain \( P' \) by solving Equation (3.22).
4) Calculate \( P \) from Equation (3.19).

5) Calculate \( u, v, \) and \( w \) from Equation (3.18).

6) Solve the discretized equations for other flow variables, such as enthalpy, turbulent quantities etc.

7) Treat the corrected pressure \( P \) as a new guessed \( P^* \), return to step 2 and repeat the procedure until converged solution is obtained.

3.2.3 Boundary Conditions

Figure 3.3 Computation cell located next to a Boundary

A control cell which is adjacent to a boundary inside the calculation domain is shown in Figure 3.3. A node is placed on the wall boundary labeled B, shown above. When the finite-volume method is applied to the equation for node P it becomes,

\[
a_i \phi_P = a_E \phi_E + a_N \phi_N + a_S \phi_S + S + a_w \phi_w
\]  

(3.29)

where the coefficient \( a_W \) is set to zero, this is because the links to the boundary node are incorporated into the source term \( S \). The source term when in its linearized form becomes:
\[ S = S_U + S_p \phi_p \]  \hspace{2cm} (3.30)

If the boundary value \( \phi_B \) is fixed, the source term is modified as,

\[ S_U = S_U + a_w \phi_B \]  \hspace{2cm} (3.31)
\[ S_p = S_p - a_w \]  \hspace{2cm} (3.32)

At zero-flux boundaries, such as in the case of adiabatic walls for heat and symmetric boundaries for any scalar variable, the boundary link coefficients are set to zero without modifying any of the source terms.

3.2.4 Solution Methods

CFD-ACE+ uses a segregated solution method where each set of equations are solved sequentially in an iterative fashion until a convergent solution emerges. Parameters within the computational engine dictate how many times the solution is iterated. This user controlled parameter is called the Number of Iterations (NITER). The NITER is coupled with another variable parameter, which is the desired residual. The desired residual is the minimum error reached before the calculation stops. For this case the residual was set to a value of \( 1 \times 10^{-4} \). At every iteration step, the program calculates the residual for each of the cells variables within the computational domain. The residual, is defined as the sum of the absolute value of the residual for that variable at each computational cell. The absolute value of the residual is the summation of all the discrepancies between the current value and the approximate value for all computational cells, \( \sum_{i=1}^{n} |e_i| \), where \( e_i \) is the discrepancy between the current value and the approximate value. Where as the relative error is defined as true error over the current computed
value. The computational process will end when either the NITER or the input residual is satisfied, refer to figure 3.4

Figure 3.4 Flowchart for computational calculation

Under-relaxation is a method used to constrain the auxiliary or dependent variable within the governing equations from iteration to iteration in order to prevent a divergent solution. In CFD-ACE+ this method is accomplished by modifying Eq.3.17, to obtain:

$$a_p (1 + I) \phi_p = \sum a_{nm} \phi_{nm} + S_u + a_p I \phi_p^*$$

(3.33)
The value of $I$, is what determines how strong the under-relaxation will be. In the above equation $\phi_p^*$ is the current step value of $\phi_p$. If there is no change between $\phi_p^*$ and $\phi_p$ between iterations then the additional terms above (under-relaxation) will have no effect. However, before the solution becomes convergent, $\phi_p^*$ will be highly influenced by $I$ and a new $\phi_p$ will be determined for that iterative step. Values for $I$ can be found, CFD-ACE+ User Manual (2002), using equation, (3.34) located in

$$I = \frac{\rho \delta}{\Delta t_f a_p}$$

Where $\Delta t_f$ is the ‘false’ time step. By changing the value of $I$, the time step will vary, such as if $I = 1.0$, then $\Delta t_f$ would be comparable to the max time step in the explicit time-marching scheme. Values for $I$ range from 0.2-0.8 in most cases. However, for a non-convergent solution, it may be necessary to modify the value of $I$, to obtain a more stable, convergent solution.

For each of the auxiliary variables $\rho, P, T$ and $\mu$, the under-relaxation for each of these variables is defined by $\lambda$ and is applied in the following way,

$$\Theta^{\text{new}} = \lambda \Theta + (1 - \lambda) \Theta^*$$

In the above equation, $\Theta$ is the value of any one of the auxiliary variables if there was no under-relaxation factor. $\Theta^*$ is the value on any one of the auxiliary variables at the current iteration. $\Theta^{\text{new}}$ is the new value of the auxiliary variable due to the influence of the under-relaxation factor.

There are two major types of solvers available in the CFD-ACE+ package. The first is the Conjugated Gradient Squared (CGS) +Preconditioning Solver and the second
is the Algebraic MultiGrid solver (AMG). The solver used for the work presented here is
the Algebraic MultiGrid solver (AMG).

The algebraic MultiGrid (AMG) solver has two major advantages. First, the CPU
time only increases in proportion to the number of unknowns in the equation. Second, is
faster convergence for fully-unstructured grids. The basic goal behind the multigrid
method stems from the fact that the error produced from an iterative computational
process can be analyzed using the Fourier equations. By use of the Fourier equation, the
error associated with the computations results in both high and low frequency errors.
Both the high and low frequency errors determine the rate at which a solution converges.
To account for these frequencies multiple grids are used for the computational domain.
Each grid ranges from a fine grid, which is normally the general computational grid, to a
course grid, which may take every other nodal point in the computation domain. Here
the high frequency errors are associated with the course grid and the low frequency errors
are associated with the fine grid. On the fine grid (original grid) the residual is obtained
after a few iterations. Here iterations are performed on the coarse grid to obtain
corrections, which will be used in the fine grid. Additional theory into the MultiGrid
method can be found in Lonsdale (1993).
CHAPTER IV

NUMERICAL RESULTS

4.1 Introduction

This thesis presents numerical results which are documented according to the following matrix, Figure 4.1. There are two variable operational parameters which will be used throughout this numerical endeavor, (i) shaft (rotor) rotational speed, and (ii) lubricating oil inlet (volumetric flow), and one geometric parameter variable (iii) the pocket aspect ratio. The aspect ratio, Figure 4.2, is defined as a measure of pocket depth to the pocket circumferential chord, $\text{AR} = \frac{D}{L}$ ($D=$pocket depth and $L=$Pocket length (0.700”)).

The pocket depths are varied from 0.700”, 0.250”, 0.060”, 0.030” and 0.015”, with the corresponding aspect ratios becoming respectively 1.000, 0.357, 0.086, 0.043 and 0.021. For each aspect ratio the operational parameters (i) and (ii) are varied one at a time. Thus, shaft speeds of 0, 200, 1000, 1500 and 3000 rpm, respectively are used while the inlet mass flow is kept constant. For volumetric flow variation, the values of 1.23 and 0.615 gpm, respectively are used.

The working fluid used for this numerical case, is a synthetic silicone oil. The fluid has a density of 888 kg/m$^3$ and a dynamic viscosity of 20 cP.
Figure 4.1 Matrix of numerical parameters

Under the physical conditions described above the study concentrates on detailing the patterns of the hydrodynamic flow inside the pocket and the associated pressure patterns.

4.2 Geometry and Boundary Conditions

This section will give a description of the overall geometry created using CFD-GEOM, and the boundary conditions used in conjunction with the CFD-ACE+ numerical solver.

4.2.1 Geometry

The fluid flow within a single, variable depth pocket journal bearing will be investigated using the computational software CFD-ACE+. The geometry is created using a pre-processing geometry building software, CFD-GEOM. Figure 4.2 shows the overall view of the single pocket geometry used for each of the numerical simulations. The geometry was set up in such a way that the number of computational cells populating the circumferential direction was kept to a minimum of 150 nodal points, thus reducing the number of iterations and accelerating computing time. The geometry is constructed as follows:

<table>
<thead>
<tr>
<th>Depth Viscosity (20 cP)</th>
<th>Depth Viscosity (20 cP)</th>
<th>Depth Viscosity (20 cP)</th>
<th>Depth Viscosity (20 cP)</th>
<th>Depth Viscosity (20 cP)</th>
<th>Depth Viscosity (20 cP)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.700</td>
<td>0.250</td>
<td>0.060</td>
<td>0.030</td>
<td>0.015</td>
<td></td>
</tr>
<tr>
<td>Density (Kg/m³) 888</td>
<td>Density (Kg/m³) 888</td>
<td>Density (Kg/m³) 888</td>
<td>Density (Kg/m³) 888</td>
<td>Density (Kg/m³) 888</td>
<td></td>
</tr>
<tr>
<td><strong>Flow (GPM)</strong></td>
<td><strong>Flow (GPM)</strong></td>
<td><strong>Flow (GPM)</strong></td>
<td><strong>Flow (GPM)</strong></td>
<td><strong>Flow (GPM)</strong></td>
<td></td>
</tr>
<tr>
<td>1.23</td>
<td>0.615</td>
<td>0.615</td>
<td>0.615</td>
<td>0.615</td>
<td></td>
</tr>
<tr>
<td>Shaft Speed (RPM) 200</td>
<td>Shaft Speed (RPM) 200</td>
<td>Shaft Speed (RPM) 200</td>
<td>Shaft Speed (RPM) 200</td>
<td>Shaft Speed (RPM) 200</td>
<td></td>
</tr>
<tr>
<td>1000</td>
<td>1000</td>
<td>1000</td>
<td>1000</td>
<td>1000</td>
<td></td>
</tr>
<tr>
<td>1500</td>
<td>1500</td>
<td>1500</td>
<td>1500</td>
<td>1500</td>
<td></td>
</tr>
<tr>
<td>3000</td>
<td>3000</td>
<td>3000</td>
<td>3000</td>
<td>3000</td>
<td></td>
</tr>
</tbody>
</table>
Figure 4.2 Nodal points and pocket geometry

Pocket Depth (D): 0.700", 0.250", 0.050", 0.030" and 0.015"
Pocket Length (L): 0.700"
AR = D/L: 1.000, 0.357, 0.086, 0.043 and 0.021

Figure 4.3 Overall view of geometry and boundary conditions

A) Clearance
B) Variable Depth Pocket
C) Restrictor
of both, inner and outer concentric cylinders, is split into two distinct cylindrical halves. The upper cylindrical half is located opposite the variable depth pocket, while the lower cylindrical half is connected to the variable depth pocket. This type of configuration allows areas of non-interest, such as the flow and pressures profiles within the upper cylindrical half, to become separated from the more important regions, the pocket and adjoining lands. Referring the Figure 4.3, the inner cylinder which is 7.974” in diameter represents a rotating wall, while the outer cylinder which is 8.000” in diameter represents a fixed (non-rotating) wall. Together, both the inner and outer cylinders create a concentric, 0.013” gap in the circumferential direction. The linear extrusion protruding out of the outer (fixed) cylinder is the variable depth pocket. This pocket, in Figure 4.4, has a circumferential length of 0.700”, while the pockets thickness, which is 0.700”, was set to 0.350” for symmetry purposes. Together the pockets circumferential cord length and thickness creates a 0.700”x 0.700” square foot print. The pocket depths (Figure 4.2) were varied from 0.250” to 0.060” to 0.030” and 0.015”, with the corresponding aspect ratios becoming 1.000, 0.357, 0.086, 0.042 and 0.021, respectively. The restrictor is attached to the inlet of the pocket and it has an inlet diameter of 0.200” and an overall length of 3.000”.

4.2.2 Boundary Condition Applications

The geometry in figure 4.3 has boundary conditions which define each of the domains necessary for the numerical computation. The two variable parameters (i) rotating shaft and (ii) mass flow and the one geometric parameter (iii) pocket depth (Aspect Ratio) are define for the geometric model.
Figure 4.4 Axial and circumferential view of the pocket

The inner cylinder, shown in Figure 4.3, is defined as a rotating wall boundary condition, physically relates to the variable parameter (i), the rotating shaft. The shaft will have rotations of 0, 200, 1000, 1500, 2000 and 3000 rpm, respectively. The outer cylinder, which represents a fixed (non-rotating) boundary condition, represents the bushing (housing) within which the shaft is rotating. The axial end of the concentric 0.013” circumferential gap, are defined as outlet boundary conditions. The outlet is set to a fixed, gauge pressure of 0psi.

The fixed inlet velocity boundary condition relates to the second variable parameter (ii), the volumetric flow. Since the diameter of the restrictor is known, the average of weighted inlet velocity can be calculated. Two volumetric flows of 1.23gpm
(3.829 m/s) and 0.615 gpm (1.914 m/s) are used for the volumetric flow boundary conditions. This inlet condition is defined as a Cartesian type boundary condition, where the x, y and z velocity components are specified, and the resulting velocity magnitude is then applied to the boundary. The inlet condition used for these set of numerical simulations is to set both the x and z velocities equal to zero and set the Y velocity component to the prescribed value.

Due to the shape of the geometry for the single pocket bearing, Figure 4.4, shows a symmetry type boundary condition is used for the purpose of saving both iterative and computing time. The symmetry plane is taken through the center of the pocket and is used to cut the geometry in half. Here, the thickness of the pocket is 0.700” and due to the symmetry boundary condition, results in a thickness of 0.350”.

4.3 Discretization

The three-dimensional grid is obtained by extruding the two-dimensional annulus like x-y geometry into the z-direction, Figure 4.5. In the upper cylindrical half in the circumferential direction there are 150 nodal points. In the lower cylindrical half, also in the circumferential direction, there are a total of 90 nodal points on either side of the pocket. The 90 nodal points have been placed, such that the area around the pocket has a larger density of nodal points. The grid spacing technique use to accomplish this task made use of the hyperbolic tangent method. Using this method, nodal points can be spaced accordingly to provide a denser gathering of nodes in the areas of interest. Both
the pocket and restrictor have an array of nodal points which vary between 20-50pts, in the radial direction. Each side of the pocket has a denser collection of nodal points, in the circumferential direction, near the pockets lands and clearance. The bottom of the pocket also has a denser collection of points near the restrictor. These areas of interest allow for better visualization of physical changes which occur due to the physics of the problem. For all the numerical cases, the total number of cells for the entire annulus and pocket varies between 250,000 to 300,000.

CFD-ACE+ uses absolute convergence criteria, which for the pressure field usually, require convergence of the residual of the order of 1.0E-4. For the cases considered here a convergence criterion of 1.0E-4 was used for each of the primitive variable (u, v, w, and
To further improve convergence, the under-relaxation factors (URF) were adjusted individually for each one of these variables. Thus, for velocities and pressure, the typical URFs varied between 0.1 and 0.7.

4.4 Results for the Case [Pocket Depth = 0.700, Flow Rate = 1.23gpm]

This section will focus on varying the shaft rotation. The shaft will rotate at six different speeds: 0, 200, 1000, 1500, 2000 and 3000rpm, respectively. The operational parameter, inlet (volumetric flow) boundary condition will be set to a constant value of 1.23gpm. The one geometric parameter, pocket depth, will also be held constant at 0.700” (AR = 1.000, pocket length (L) = 0.700”).

4.4.1 Visualization of Flow Patterns, AR = 1.000

Figure 4.6 presents a numerical simulation of the flow and pressure profiles for a deep pocket (0.700” depth, AR = 1.000). Note the dynamic viscosity of the lubricating oil was 20 Cp, with a density of 888 (Kg/m³). Streamlines traces are superimposed to better highlight the flow configuration within the pocket. There are two types of Reynolds number based parameters investigated. The first is the Reynolds number based on the clearance, \( \text{Re}_c = \frac{\rho R w c}{\mu} \), where \( \rho \) is the density of the fluid, \( R \) is the radius of the shaft (rotor), \( w \) is the shafts rotation speed, \( c \) is the clearance between the shaft and the fixed housing and \( \mu \) is the dynamic viscosity. The second is the Reynolds number based on the fluid jet, \( \text{Re}_j = \frac{\rho V D}{\mu} \), where \( \rho \) is the density of the fluid, \( V \) is the average velocity of the fluid in the restrictor, \( D \) is the diameter of the restrictor and \( \mu \) is the dynamic viscosity.
For the deep pocket, Figure 4.6a, one can distinguish the formation of two symmetrical vortical cells located in both the left hand side (LHS) and right hand side (RHS) domains of the pocket, centered on each side of the fluid jet exiting the restrictor. Pressure forces dominate inside the pocket due to the dynamic effects of the fluid jet. It is the outer envelop of the jet which forms and sustains both the LHS and RHS vortical cells. Due to these pressure forces all of the fluid exits the pocket circumferentially in both the LHS and RHS clearances. Figure 4.6b, shaft speed is increased to 200rpm. Still, observations of two vortical cell profiles are present within the pocket. Examining the clearance, Couette effects start to be seen due to shaft rotation. As a consequence of the Couette force, a layer of fluid is entrained and then carried by the shaft (figure 4.6b, detail). Here a transition region develops slowly and the Couette forces begin to interact with the dominant pressure force. Inside the LHS clearance of the pocket, the pressure forces dominate thus allowing fluid to exit both the LHS and RHS clearances of the pocket. However, in the LHS clearance, one can see the interaction of fluid entering the pocket, due to the Couette forces and the fluid exiting the pocket, due to the pressure forces. Figure 4.6c, shaft speed is increased to 1000 rpm. For this shaft speed, there still is the observation of the LHS vortical cell dominating the upper portion of the pocket. However, in the RHS region, the vortical cell has decreased in size. The flow in this region becomes entrained by the outer envelop of the fluid jet and flows out of the RHS clearance. The Couette effects begin to dominate the once strong pressure forces. Here more fluid begins to enter the pocket and now only a small amount of fluid
Figure 4.6 General view of 0.700” pocket depth (AR = 1.000) showing flow patterns within the pocket. This case is examines a flow rate of 1.23gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.
Figure 4.7 General view of Pressure profiles for a 0.700” pocket depth (AR = 1.000) with a flow rate of 1.23gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.

is able to exit the LHS clearance(figure 4.6c, detail). In the portion of the clearance, between the pockets land and the jet impingement on the surface of the shaft, one can start to see the formation of a turn around zone (TAZ). In this region, the fluid, which is circulating in the LHS vortical cell becomes entrained by the Couette forces due to the high shaft rotation. It is a consequence of this entrainment that the fluid will physically change directions and begin flowing in the opposite direction (figure 4.6c, detail and Appendix B). Figure 4.6d, e, f, shaft speed is increased to 1500, 2000 and 3000rpm. The vortical cell, which dominates the upstream portion of the pocket has slightly decreased in size and has begun slowly shifting away from the LHS clearance. When Couette forces dominate the pocket, fluid can no longer exit the LHS clearance (Figure 4.6d, e, detail), the fluid is essentially trapped, since the pressure forces are relatively large due to the high flow rate, the outer envelop of the fluid jet sustains the LHS vortical cell. In the LHS clearance (figure4.6d), flow no longer exits the pockets clearance, now all flow is
entering the pocket. In this domain the pressure forces no longer dominate the Couette forces. In the portion of the pocket between the fluid jet and the LHS clearance, the TAZ is no longer in the clearance, but has grown in size and now is situated in the pocket. In the RHS portion of the pocket, there is no longer the presence of a clear vortical cell profile. This is a result of increased shaft speed. As the shaft speed increases, the Couette forces cause the RHS vortical cell to decrease in overall size and eventually disappear. This is caused from the increased shaft speed accelerating the fluid near the surface of the shaft to a higher velocity. This high velocity causes a Bernoulli effect in the upper RHS of the pocket, causing the pressure to drop. The pressure in the region below the RHS vortical cell is higher than in the clearance, this pressure differential causes the RHS vortical cell to shift upward toward the clearance and eventually becomes “pushed” out of the pocket.

4.4.2 Visualization of Pressure Patterns, AR = 1.000

Referring to figure 4.7, shows a variety of pressure profiles for various shaft speeds for a pocket depth of 0.700”(AR = 1.000) and a volumetric flow rate of 1.23gpm. The pressures shown are present at the axial centerline of the bearing, which include the pocket and adjoining lands. We are not presenting pressure profiles for the upper cylindrical half, since there is no build up of pressure. For a zero shaft rotation the pressures due to concentricity in both the LHS and RHS domains of the pocket are symmetric with respect to the circumferential center of the pocket and is a constant value of 17psi. In the radial direction, there is an increase in pressure of 21psi in the area of the impinging jet. This impingement causes a pressure spike, which can be seen in the overall pressure profile. Both the LHS and RHS clearances of the pocket show that
pressure decrease as the flow exits the pockets circumferentially. As the shaft speed increases, the overall pressures within the pocket increases as well. This is due to the Couette forces dominating the pocket. Because of this no fluid exits the LHS clearance of the pocket. To satisfy continuity, more fluid exits the RHS clearance. This increase in flow out of the 0.013” gap, creates a pressure rise inside the pocket. For shaft speed of 3000 rpm, one can see the formation of a small pressure spike on the RHS of the clearance at the exit of the pocket. This is due to fluid acceleration in the small clearance and here a Rayleigh step effect is seen. However the depth of the pocket eliminates (dampens) it to a large extent.

4.5 Results for the Case [Pocket Depth = 0.700, Flow Rate = 0.615]

This section will focus on varying shaft rotation. The shaft will rotate at six different rotational speeds: 0, 200, 1000, 1500, 2000 and 3000rpm, respectively. The operational parameter, inlet (volumetric flow) boundary condition will be set to a constant value of 0.615gpm. The geometric parameter, pocket depth, will also be held constant at 0.700” (AR = 1.000, Pocket Length (L) = 0.700”).

4.5.1 Visualization of Flow Patterns, AR = 1.000

Figure 4.8a, shows the formation of two symmetric looking vortical profiles within the upper portion of the deep pocket. Just as in the previous 1.23gpm case, each vortical cell is centered between the fluid jet exiting the restrictor. In both the LHS and RHS clearances, the fluid is exiting the pocket (figure 4.8a, detail), due to the dominate
Figure 4.8 General view of 0.700” pocket depth (AR = 1.000) showing flow patterns within the pocket. This case is examines a flow rate of 0.615gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.
Figure 4.9 General view of Pressure profiles for a 0.700” pocket depth (AR = 1.000) with a flow rate of 0.615gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.

Pressure forces inside the pocket. These pressure forces are caused from the dynamic effects of the fluid jet impinging the surface of the shaft. As shaft speed increases to 200 rpm, figure 4.8b, still shows both vortical cells dominating the LHS and RHS regions of the pocket. In the clearance, pressure forces still dominate however, one can see the Couette forces have a slightly greater effect over the poiseuille forces for the 0.615 gpm case then the previous 1.23 gpm (figure4.8b, detail). In both domains of the pocket below the recirculating cell, there is a region of fluid only affected by the fluid jet. The jet, which exits the restrictor at the bottom of the pocket, entrains the stationary fluid and carries it to the top of the pocket. Here fluid that does not exit the pocket through the clearance becomes recirculated. Figures 4.8c, d, shaft speed is increased to 1000 and 1500 rpm. Here, vortical cells which dominated in the lower speeds has changed in both overall size and location. The cell center has shifted from its previous location, to the
upper most corner of the RHS clearance. This shift is a direct result of fluid not exiting the LHS region of the pockets clearance. Instead the TAZ entrains the fluid and carries it to the RHS of the pocket. To satisfy continuity, since fluid no longer exits the LHS clearance, flow rate increases in the RHS clearance, allowing all the fluid to exit the pocket. Because of this increase flow out of the pocket, this allows the RHS vortical cell to be pulled toward the clearance. Increase in shaft speed, from figures 4.8c, b, one can see that the RHS vortical cell has dramatically increased in overall size encompassing half the RHS domain. However, as in the previous 200rpm case, there still is a region of fluid which becomes entrained from the fluid jet. This area has reduced in size to accommodate the growing recirculating cell above. The overall shape of the cell has grown to encompass half of the downstream portion of the pocket. For speeds of 1500 and 3000 rpm, Figure 4.8e, f, the LHS vortical cell no longer exists. Instead fluid in this domain becomes entrained by the jet and is carried to the clearance. In the clearance the Couette forces dominate causing the fluid opposing the motion of the shaft to turn around, creating the turn around zone (TAZ). In the RHS portion of the pocket, the vortical cell has encompassed the entire region of the pocket. In the lower RHS of the pocket, one can see the formation of a secondary vortical cell.

4.5.2 Pressure Profiles, AR = 1.000

Figure 4.9, shows a variety of pressure profiles on the surface of the shaft for various shaft speeds corresponding to a pocket depth of 0.700” (AR = 1.000) and a flow rate of 0.615gpm. Also, further details of the pressure profiles within the entire pocket can be seen from figure 4.8. These details overall pressures throughout the entire pocket and clearance using a color coded legend. For zero shaft rotation, the pressures built in
both the LHS and RHS domains of the pocket become a constant symmetric value of 9psi. However, in the immediate vicinity of the fluid jet impinging the surface of the shaft, there is an increase in pressure to a value of 10psi. This impingement causes a pressure spike, which can be seen in the overall pressure profile. Both the LHS and RHS clearances of the pocket show that pressure decreases to zero as the flow exits the pocket and travels around the circumference of the clearance. The range of pressure seen in the pocket for various shaft speeds ranging from no rotation to 3000rpm is 2psi in the pocket.

4.6 Results for the Case [AR = 0.357, Pocket Depth = 0.250”, Flow Rate = 1.23gpm]

This section will focus on varying shaft rotation. The shaft will rotate at six different rotational speeds: 0, 200, 1000, 1500, 2000 and 3000rpm, respectively. The operational parameter, inlet (volumetric flow) boundary condition will be set to a constant value of 1.23gpm. The geometric parameter, pocket depth, will also be held constant at 0.250” (AR = 0.357, Pocket Length (L) = 0.700”).

4.6.1 Visualization of Flow Patterns, AR = 0.357

Figure 4.10a, one can distinguish the formation of two symmetrical vortical cells centered on each side of the fluid jet coming from the restrictor, which occupy both the LHS and RHS region entirely. In the upstream portion of the clearance, the pressure (Poiseuille) forces dominate the pocket. Due to these pressure forces all of the fluid exits the pocket circumferentially in both the LHS and RHS clearances. The upper portion of each cell is elongated toward the pockets clearance. This is due to the dominant pressure force within the pocket, allowing all fluid to fully exit the LHS and RHS clearance. Since the fluid can fully exit the pocket, the upper portions of these cells are drawn

42
Figure 4.10 General view of 0.250” pocket depth (AR = 0.357) showing flow patterns within the pocket. This case is examines a flow rate of 1.23gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.
Figure 4.11 General view of Pressure profiles for a 0.250” pocket depth (AR = 0.357) with a flow rate of 1.23gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.

toward the clearance. The fluid which does not exit the pocket becomes recirculated within the vortical cell. Figure 4.10b, shaft speed is increased to 200rpm. Still, observations of two vortical cells centered between the fluid jet are present within the pocket. Close examination of the clearance shows the Couette effects start to be seen due to shaft rotation. Because of the Couette force, a layer of fluid close to the surface of the shaft becomes entrained and is then carried by the shaft to the RHS of the pocket toward the clearance. Here a transition region develops and the Couette forces begin to interact with the dominant pressure forces from the fluid jet. The pressure forces still dominate, thus allowing fluid to exit both the LHS and RHS clearances of the pocket, circumferentially. However, in the LHS clearance, one can see the interaction of fluid entering the pocket due to the Couette forces, and the fluid exiting the pocket due to the pressure forces. Figure 4.10c, shaft speed is increased to 1000 rpm. There still is the
presence of the LHS cell dominating this portion of the pocket. However, in the RHS region, there no longer is a clear vortical cell profile. Instead, the flow in this region becomes entrained by the jet, exiting the restrictor, which then flows out of the pockets clearance. In the LHS clearance, Couette effects begin to overtake the once dominant pressure force. This can be seen in figure 4.10c (detail), as more fluid begins to enter the pocket through the clearance. In this region of the clearance, between the pockets land and the jet impingement on the shaft, one can see the formation of a turn around zone (TAZ). Here, fluid, which is circulating in the LHS vortical cell, becomes entrained by the Couette forces due to shaft rotation. It is because of this entrainment the fluid will physically change directions and begin flowing in the opposite direction. The elongated upper portion of the LHS vortical cell, is no longer stretched as in the non rotating case (figures 4.10a, b). Because less fluid exits the pocket, this eliminates the drawing of the cell toward the clearance. Figure 4.10d, shaft speed is increased to 1500 rpm. The vortical cell, which dominates the LHS portion of the pocket has slightly decreased in size and the cell center has begun shifting toward the LHS clearance. In the LHS clearance, flow is no longer exiting the pockets clearance. In this domain the pressure forces no longer dominate the inertial forces. The downstream portion of the pocket, there still is no longer any vortical profile. The flow in this area is entrained by the jet and exits the RHS portion of the pocket. Between the LHS clearance and the fluid jet impingement on the shaft, the TAZ is seen in the pocket. Figures 4.10e, f, represent a shaft speeds of 2000 and 3000rpm. On the LHS of the pocket, the dominating vortical cell which encompassed the entire region at lower shaft speeds has decreased greatly in overall size (figure 4.10e). Below the cell, there is a region of fluid only affected by the
fluid jet. The jet, which exits the restrictor, entrains the fluid and carries it to the top of the pocket. Here the fluid that does not exit the LHS clearance becomes recirculated. In the clearance, fluid is only entering the pocket and the increase Couette force from the high speed shaft causes the TAZ to slightly shift downward into the pocket. The RHS of the pocket still yields the region entrained by the fluid jet, which carries the flow out of the pocket.

4.6.2 Pressure Profiles, AR = 0.357

Figure 4.11, shows a variety of pressure profiles for various shaft speeds corresponding to a pocket depth of 0.250” (AR = 0.357) and a flow rate of 1.23gpm. Also, further details of the pressure profiles within the entire pocket can be seen from figure 4.10. For zero shaft rotation, the pressures built in both the LHS and RHS domains of the pocket are larger for this depth than the previous 0.700” deep pocket. The pressures produced on the shaft for this case, results in a constant value of 18psi. However, the pressure spike seen emerging from the constant pressure area of the profile, is largely due to the fluid jet, impinging on the surface of the shaft. This impingement causes a pressure rise, where pressures in this vicinity spike to a value of 22psi. The range of pressure seen in the pocket for various shaft speeds of 0 to 3000rpm is 4psi in the pocket and about 6-7psi on the shaft directly below the fluid jet.

4.7 Results for the Case [AR = 0.357, Pocket Depth = 0.250”, Flow Rate = 0.615gpm]

This section will focus on varying the shaft rotation. The shaft will rotate at six different rotational speeds: 0, 200, 1000, 1500, 2000 and 3000rpm, respectively. The operational parameter, inlet (volumetric flow) boundary condition will be set to a
constant value of 0.615gpm. The geometric parameter, pocket depth, will also be held constant at 0.250” (AR = 0.357, Pocket Length (L) = 0.700”).

4.7.1 Visualization of Flow Patterns, AR = 0.357

Figure 4.12a, show two symmetric vortical cell profiles within the pocket. Just as in the 1.23gpm case, each cell is centered between the fluid jet exiting the restrictor. In both the LHS and RHS clearances, the fluid exits the pocket because of the dominate pressure forces built up inside the pocket due to the fluid jet. As the shaft speed increases to 200 rpm, figure 4.12b, shows again, both vortical cells dominating the LHS and RHS regions of the pocket. However, the cell located in the LHS has decreased in size and below the circulating cell there is an area of fluid which is influenced by the jet. Here, the jet entrains the stationary fluid and carries it to the clearance where it can exit the pocket. Fluid not exiting the pocket becomes recirculated in the vortical cell. In the clearance, still the pressure forces dominate within the pocket; however, one can see the Couette forces have a slightly greater effect over the poiseuille forces for the 0.615 gpm case then the previous 1.23 gpm case. Figures 4.12c, d, shaft speed is increased to 1000 and 1500 rpm. The two vortical cells which previously dominated at lower speeds (0 and 200rpm) have decreased in size and location. Both cell centers have shifted from the center of the pocket to the upper most corner of the LHS and RHS clearance. For speeds of 1500, 2000 and 3000 rpm, Figure 4.12d, e, f, the LHS vortical cell is no longer present. Instead fluid in this domain becomes entrained by the jet and is carried to the clearance. In the clearance the Couette forces dominate causing the fluid opposing the motion of the shaft to turn around, creating a TAZ. In the RHS of the pocket, the center of the vortical cell is centered toward the clearance of the pocket. Just as in the previous shaft speed of
Figure 4.12 General view of 0.250” pocket depth (AR = 0.357) showing flow patterns within the pocket. This case is examines a flow rate of 0.615 gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000 rpm.
Figure 4.13, shows a variety of pressure profiles for various shaft speeds corresponding to a pocket depth of 0.250” (AR = 0.357) and a flow rate of 0.615gpm. Also, further details of the pressure profiles within the entire pocket can be seen from figure 4.12. For zero shaft rotation, the pressures built in both the LHS and RHS domains of the pocket become a constant value of 9psi. However, in the immediate vicinity of the fluid jet impinging the surface of the shaft, there is an increase in pressure to a value of 10psi. This impingement causes a pressure spike, which can be seen in the overall pressure profile. Both the LHS and RHS clearances of the pocket show that pressure decreases to zero as the flow exits the pocket and travels around the
circumference of the clearance. The range of pressure seen in the pocket for various shaft speeds ranging from no rotation to 3000rpm is 2psi. For a shaft speed of 3000 rpm, one can see the formation of a small pressure spike on the LHS of the clearance.

4.8 Results for the Case [Pocket Depth = 0.060”, Flow Rate = 1.23gpm]

This section will focus on varying the operational parameter shaft rotation. The shaft will rotate at six different rotational speeds: 0, 200, 1000, 1500, 2000 and 3000rpm, respectively. The operational parameter, inlet (mass flow) boundary condition will be set to a constant value of 1.23gpm. The geometric parameter, pocket depth, will also be held constant at 0.060” (AR = 0.085).

4.8.1 Visualization of Flow Patterns, AR = 0.085

Figure 4.11a, shows two symmetric vortical cell profiles centered in the upper corners of both the LHS and RHS of the pocket. The symmetric profile seen is very similar to all other cases with no shaft rotation. Because there is no shaft rotation, no shear forces are created, thus allowing all fluid entering the pocket (via the restrictor), to fully exit the pocket in both the LHS and RHS clearances. As shaft speed increases to 200 rpm as shown in figure 4.11b, one can see the inertial effect slowly beginning to entrain a small amount of fluid near the surface of the shaft. However, pressure forces created from the fluid jet are much greater. Because of this, the fluid exits out of the LHS pocket clearance without much resistance. In the pocket, there is still the presence of two vortical cells. Figures 4.11c, d, shows as shaft speed increases to 1000 and 1500 the inertial effects starts to dominate the once strong pressure forces in the clearance of the pocket. At a shaft speed of 1000 rpm, weak pressure forces allow some fluid to exit through the LHS clearance, but due to the strong inertial forces, the majority of the fluid
Figure 4.14 General view of 0.060” pocket depth (AR = 0.086) showing flow patterns within the pocket. This case examines a flow rate of 1.23gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.
Figure 4.15 General view of Pressure profiles for a 0.060” pocket depth (AR = 0.086) with a flow rate of 1.23gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.

enters the pocket. When the shaft speed is increased to 1500 rpm, the inertial forces have completely dominated the pressure forces causing no fluid to exit the LHS pocket. In the clearance a TAZ is formed due the dominating inertial forces acting on the fluid entrained form the fluid jet. The entrained fluid is carried to the LHS clearance where it tries to exit. Because of the strong inertial force, the fluid physically changes direction and is carried to the RHS clearance where it will exit the pocket.

4.8.2 Pressure Patterns, AR = 0.086

Figure 4.12, shows a variety of pressure profiles for various shaft speeds corresponding to a pocket depth of 0.060” (AR = 0.086) and a flow rate of 1.23gpm. Also, further details of the pressure profiles within the entire pocket can be seen from figure 4.11. For zero shaft rotation, the pressures built in both the LHS and RHS domains of the pocket become a constant value of 17psi. However, in the immediate
vicinity of the fluid jet impinging the surface of the shaft, there is an increase in pressure to a value of 22psi. This impingement causes a pressure spike, which can be seen in the overall pressure profile. Both the LHS and RHS clearances of the pocket show that pressure decreases to zero as the flow exits the pocket and travels around the circumference of the clearance. The range of pressure seen in the pocket for various shaft speeds ranging from no rotation to 3000rpm is 10psi in the pocket. For shaft speed of 3000 rpm, one can see the formation of a small pressure spike on the LHS of the clearance.

4.9 Results for the Case [Pocket Depth = 0.060”, Flow Rate = 0.615gpm]

This section will focus on varying the operational parameter (i) shaft rotation. The shaft will rotate at six different rotational speeds. These shaft speeds will range from 0, 200, 1000, 1500, 2000 and 3000rpm, respectively. The second operational parameter (ii) inlet (mass flow) boundary condition will be set to a constant value of 0.615gpm. The one geometric parameter, (iii) pocket depth, will also be held constant at 0.060” (AR = 0.085).

4.9.1 Visualization of Flow Patterns, AR = 0.086

Figures 4.13a, b, yield similar vortical profiles as in the previous 1.23gpm case. Here one can see the two symmetric vortical cells centered between fluid jet exiting the bottom of the pocket. These large cells encompass both the entire LHS and RHS regions of the pocket. Each of the two cells are elongated and stretched to the horizontally across the length of the pocket. These cells re formed from the jet entraining the fluid from both the RHS and LHS domains. The entrained fluid is then carried from the bottom of the
Figure 4.16 General view of 0.060” pocket depth (AR = 0.086) showing flow patterns within the pocket. This case is examines a flow rate of 0.615gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.
Figure 4.17 General view of Pressure profiles for a 0.060” pocket depth (AR = 0.086) with a flow rate of 0.615gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.

Pocket and clearance Pressure (0.060 Depth, 0.615GPM)

Pocket to the top where the fluid not exiting the pocket is forced to recirculate, thus forming these cell profiles. Figure 4.13c, represents a shaft rotation of 1000rpm. Here, the LHS vortical looks similar in profile to the previous 200rpm case. The RHS vortical cell has decreased in overall size. This is caused from the increased flow of the fluid exiting the pockets clearance. Figure 4.13d, e, f, represent a shaft rotation of 1500, 2000 and 3000rpm. On the LHS of the pocket, there is no longer the presence of a vortical cell profile. Instead the fluid is entrained from the fluid jet and carried from the bottom of the pocket to the top where the TAZ is located. The fluid, which is influenced from the TAZ, is carried by the shaft to the LHS region of the pocket, where the fluid exits the clearance of the pocket.
4.9.2 Pressure Patterns, AR = 0.086

Figure 4.14, shows a variety of pressure profiles for various shaft speeds corresponding to a pocket depth of 0.060” (AR = 0.086) and a flow rate of 1.23gpm. Also, further details of the pressure profiles within the entire pocket can be seen from figure 4.13. For zero shaft rotation, the pressures built in both the LHS and RHS domains of the pocket become a constant value of 17psi. However, in the immediate vicinity of the fluid jet impinging the surface of the shaft, there is an increase in pressure to a value of 22psi. This impingement causes a pressure spike, which can be seen in the overall pressure profile. Both the LHS and RHS clearances of the pocket show that pressure decreases to zero as the flow exits the pocket and travels around the circumference of the clearance. The range of pressure seen in the pocket for various shaft speeds ranging from no rotation to 3000rpm is 10psi in the pocket. For shaft speed of 3000 rpm, one can see the formation of a small pressure spike on the LHS of the clearance.

4.10 Results for the Case [Pocket Depth = 0.030”, Flow Rate = 1.23gpm]

This section will focus on varying the operational parameter (i) shaft rotation. The shaft will rotate at six different rotational speeds. These shaft speeds will range from 0, 200, 1000, 1500, 2000 and 3000rpm, respectively. The second operational parameter (ii) inlet (mass flow) boundary condition will be set to a constant value of 1.23gpm. The one geometric parameter, (iii) pocket depth, will also be held constant at 0.030” (AR = 0.043).
Figure 4.18 General view of 0.030” pocket depth (AR = 0.043) showing flow patterns within the pocket. This case examines a flow rate of 1.23gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.
Figure 4.19 General view of Pressure profiles for a 0.030” pocket depth (AR = 0.043) with a flow rate of 1.23gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.

4.10.1 Visualization of Flow Patterns, AR = 0.043

Figure 4.15a, shows a similar symmetric vertical cell profile as in the previous 0.615gpm case. Both the LHS and RHS cells are centered between the fluid jet exiting the restrictor located at the bottom of the pocket. Both cells are centered in the pocket occupying fully both regions. Lack of shear stress, due to no rotation causes the pressure forces to dominate in both LHS and RHS of the clearance. This causes all the fluid the exit the pocket in the circumferential direction (figure 4.15a, detail). Figures 4.15b, c, represent a shaft rotation of 200 and 1000rpm. In the upper part of the LHS clearance, one can see the effect of shaft rotation on the flow pattern in this region. The inertial force, which is due to the shaft rotation, entrains a small layer of fluid on the shaft. This entrained fluid is then carried to the RHS of the pocket where it exits through the clearance. However, for shaft speed of 200rpm, the pressure forces dominate the inertial
forces. The dominant pressure forces cause the majority of the fluid exiting the LHS clearance of the pocket (figure 4.15b, detail). For a shaft speed of 1000 rpm, the inertial forces begin to overtake the previously large pressure forces. This can be seen from the lack of exit flowing through the clearance (figure 4.15c, detail). In the LHS of the pocket, the vortical cell has shifted downward. In the RHS of the pocket, the vortical cell which was defined in the non rotation case, is no longer present. Instead the fluid in this region is entrained from the jet and carried to the RHS clearance where it exits the pocket. Figure 4.15d, e, f, represent shaft speeds of 1500, 2000 and 3000 rpm. In the LHS of the clearance, the inertial force has completely dominated the pressure forces. This is evident from the fluid no longer exiting this portion of the pocket. Still the vortical cell in this region has slightly shifted more towards the bottom of the pocket. In the RHS of the pocket the fluid is still entrained by the jet and exits the clearance.

4.10.2 Pressure Patterns, AR = 0.043

Figure 4.16, shows a variety of pressure profiles for various shaft speeds corresponding to a pocket depth of 0.030” (AR = 0.043) and a flow rate of 1.23gpm. Also, further details of the pressure profiles within the entire pocket can be seen from figure 4.15. For zero shaft rotation, the pressures built in both the LHS and RHS domains of the pocket are larger for this depth than the previous 0.700” deep pocket. There is a noticeable difference in pressure in both the LHS and RHS of the pocket. In the LHS of the pocket the pressure on the shaft are about 18psi, while the RHS of the pocket increase slightly to 19psi. However, the pressure spike seen emerging from the constant pressure area of the profile, is largely due to the fluid jet, impinging on the surface of the shaft. This impingement causes a pressure rise, where pressures in this
vicinity spike to a value of 22psi. As shaft speed increases from 0 rpm to 200, 1000, 1500, 2000 and 3000rpm the pressures produced in the pocket vary, while the deep pocket, pressures were constant. The LHS of the pocket yields lower pressures while the RHS of the pocket show higher pressures. These higher pressures are due to the Rayleigh effect from all the fluid trying to exit the RHS clearance.

4.11 Results for the Case [Pocket Depth = 0.030”, Flow Rate = 0.615gpm]

This section will focus on varying the operational parameter (i) shaft rotation. The shaft will rotate at six different rotational speeds. These shaft speeds will range from 0, 200, 1000, 1500, 2000 and 3000rpm, respectively. The second operational parameter (ii) inlet (mass flow) boundary condition will be set to a constant value of 0.615. The one geometric parameter, (iii) pocket depth, will also be held constant at 0.030” (AR = 0.043).

4.11.1 Visualization of Flow Patterns, AR = 0.043

Figure 4.17a, represents a non rotation shaft speed. Here as in all previous cases is the formation of two symmetric vortical profiles, created by the pure dominant pressure forces developed by the fluid jet. As shaft speed increases from 0rpm to 200, figure 4.17b, shows the LHS clearance where the flow exits the pocket. However in this case, the shear forces developed from the shaft rotation, entrains a layer of fluid, which is then carries to the RHS clearance where it exits the pocket circumferentially. Here one can see the weak inertial forces are stronger in this case than the previous 1.23gpm case. The presences of two vortical cells are still evident in both the RHS and LHS regions of the pocket. As shaft speed is increased to 1000, 1500, 2000 and 3000rpm, shown in figures
Figure 4.20 General view of 0.030” pocket depth (AR = 0.043) showing flow patterns within the pocket. This case examines a flow rate of 0.615gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.
Figure 4.21 General view of Pressure profiles for a 0.030” pocket depth (AR = 0.043) with a flow rate of 0.615gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.

4.17c, d, e, f, one can see in the LHS clearance fluid no longer exits the pocket. This is an indication that the inertial forces have completely dominated the once strong pressure forces. There is no longer a presence of a vortical cell in any of the LHS or RHS portions of the pocket. Due to the strong inertial forces acting on the fluid in the LHS portion of the pocket a TAZ can be seen (figure 4.17d, e, f, details). The TAZ is a direct result of the strong inertial forces overtaking the pressure forces, this in turn, physically changes the direction of the fluid particle. Instead of exiting the LHS clearance, the fluid is entrained by the shaft and carried to the RHS of the pocket were it can then exit.

4.11.2 Pressure Patterns, AR = 0.043

Figure 4.18, shows a variety of pressure profiles for various shaft speeds corresponding to a pocket depth of 0.030” (AR = 0.043) and a flow rate of 1.23gpm.

62
Also, further details of the pressure profiles within the entire pocket can be seen from figure 4.17. For zero shaft rotation, the pressures built in both the LHS and RHS domains of the pocket are larger for this depth than the previous 0.700” deep pocket. There is a noticeable difference in pressure in both the LHS and RHS of the pocket. In the LHS of the pocket the pressure on the shaft is about 9psi, while the RHS of the pocket decreases slightly to 8.8psi. However, the pressure spike seen emerging from the constant pressure area of the profile, is largely due to the fluid jet, impinging on the surface of the shaft. This impingement causes a pressure rise, where pressures in this vicinity spike to a value of 11psi. As shaft speed increases from 0 rpm to 200, 1000, 1500, 2000 and 3000rpm the pressures produced in the pocket vary, while the deep pocket, pressures were constant. The LHS of the pocket yields higher pressures while the RHS of the pocket show lower pressures.

4.12 Results for the Case [Pocket Depth = 0.015”, Flow Rate = 1.23gpm]

This section will focus on varying the operational parameter (i) shaft rotation. The shaft will rotate at six different rotational speeds. These shaft speeds will range from 0, 200, 1000, 1500, 2000 and 3000rpm, respectively. The second operational parameter (ii) inlet (mass flow) boundary condition will be set to a constant value of 1.23gpm. The one geometric parameter, (iii) pocket depth, will also be held constant at 0.015” (AR = 0.021).

4.12.1 Visualization of Flow Patterns, AR = 0.021

Figures 4.19a, b, represents shaft speeds of 0 and 200rpm, here one can see the formation of two symmetric vortical cells centered between fluid jet exiting the bottom of the pocket. These large cells encompass both the entire LHS and RHS regions of the
Figure 4.22 General view of 0.015” pocket depth (AR = 0.021) showing flow patterns within the pocket. This case examines a flow rate of 1.23gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.
Figure 4.23 General view of Pressure profiles for a 0.015” pocket depth (AR = 0.21) with a flow rate of 1.23gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.

Pocket and clearance Pressure (0.015 Depth, 1.23GPM)

Pocket. Each of the two cells are elongated and stretched to the horizontally across the length of the pocket. In the upper LHS of the clearance, the fluid can fully exit the pocket due to the dominant pressure force developed by the fluid jet. For a shaft rotation of 200rpm, the layer of fluid close to the shaft becomes entrained due to the inertial force developed. This entrained fluid is carried to the RHS clearance where it exits the pocket. However, on the RHS of the pocket, the vortical cell is still present. For higher shaft speeds of 1000, 1500, 2000 and 3000 rpm (figures 4.19c, d, e, f) the inertial forces overtake the pressure forces which were once dominate for lower shaft speeds. The fluid in this region, no longer exits the pocket. Because of the inertial force, a TAZ is formed in the upper clearance. As shaft speed increases and the inertial forces become stronger, one can see the TAZ grow from inside the clearance to the upper portion of the pocket (figure#4.19c, e, detail).
4.12.2 Pressure Patterns, AR = 0.021

Figure 4.20, shows a variety of pressure profiles for various shaft speeds corresponding to a pocket depth of 0.015” (AR = 0.021) and a flow rate of 1.23gpm. Also, further details of the pressure profiles within the entire pocket can be seen from Figure 4.19. For zero shaft rotation, the pressures built in both the LHS and RHS domains of the pocket are larger for this depth than the previous 0.700” deep pocket. There is a noticeable difference in pressure in both the LHS and RHS of the pocket. In the LHS of the pocket the pressure on the shaft are about 18psi, while the RHS of the pocket increase slightly to 19psi. However, the pressure spike seen emerging from the constant pressure area of the profile, is largely due to the fluid jet, impinging on the surface of the shaft. This impingement and the small pocket depth produce an extremely large pressure rise, where pressures in this vicinity spike to a value of 25psi. As shaft speed increases from 0 rpm to 200, 1000, 1500, 2000 and 3000rpm the pressures produced in the pocket vary, while the deep pocket, pressures were constant. The LHS of the pocket yields lower pressures while the RHS of the pocket show higher pressures. These higher pressures are due to the Rayleigh effect from all the fluid trying to exit the RHS clearance.

4.13 Results for the Case [Pocket Depth = 0.015”, Flow Rate = 0.615gpm]

This section will focus on varying the operational parameter (i) shaft rotation. The shaft will rotate at six different rotational speeds. These shaft speeds will range from 0, 200, 1000, 1500, 2000 and 3000rpm, respectively. The second operational parameter (ii) inlet (mass flow) boundary condition will be set to a constant value of 0.615gpm. The one
Figure 4.24 General view of 0.015” pocket depth (AR = 0.021) showing flow patterns within the pocket. This case examines a flow rate of 0.615 gpm with shaft speeds of 0, 200, 1000, 1500, 2000 and 3000 rpm.
Figure 4.25 General view of Pressure profiles for a 0.015” pocket depth (AR = 0.21) with a flow rate of 0.615gpm for various shaft speeds of 0, 200, 1000, 1500, 2000 and 3000rpm.

geometric parameter, (iii) pocket depth, will also be held constant at 0.015” (AR = 0.021).

4.13.1 Visualization of Flow Patterns, AR = 0.021

Figure 4.21, yields the case of no shaft rotation. Here fluid in both the LHS and RHS of the pocket is entrained by the fluid jet. Unlike the previous cases, there in not a development of a vortical cell profiles. Because the fluid fully exits the LHS clearance, pressure forces are dominant. However, when shaft speeds increase to 200rpm, Figure4.21b, shows the inertial forces are stronger in this 0.615gpm case then the previous 1.23gpm case. Still no presence of a vortical cell is shown, fluid is only influenced by the jet, and exits the pocket in both regions of the pocket. Figure 4.21c, d, e, f, represents shaft speeds of 1000, 1500, 2000 and 3000, show the progression
downward of the TAZ. As the shaft speed increases, the inertial effects increase, allowing the TAZ to grow from the clearance into the pocket (figure 4.21c, e, detail).

4.13.2 Pressure Patterns, AR = 0.021

Figure 4.22, shows a variety of pressure profiles for various shaft speeds corresponding to a pocket depth of 0.030” (AR = 0.043) and a flow rate of 1.23gpm. Also, further details of the pressure profiles within the entire pocket can be seen from figure 4.21. For zero shaft rotation, the pressures built in both the LHS and RHS domains of the pocket are larger for this depth than the previous 0.700” deep pocket. There is a noticeable difference in pressure in both the LHS and RHS of the pocket. In the LHS of the pocket the pressure on the shaft are about 9psi, while the RHS of the pocket decreases slightly to 8.8psi. However, the pressure spike seen emerging from the constant pressure area of the profile, is largely due to the fluid jet, impinging on the surface of the shaft. This impingement causes a pressure rise, where pressures in this vicinity spike to a value of 12psi. As shaft speed increases from 0 rpm to 200, 1000, 1500, 2000 and 3000rpm the pressures produced in the pocket vary, while the deep pocket, pressures were constant. The LHS of the pocket yields higher pressures while the RHS of the pocket show lower pressures.
CHAPTER V

DESCRIPTION OF EXPERIMENTAL INSTALLATION

5.1 Introduction

This section will outline the hardware and lubricating fluid used throughout the experiment. The hardware will include detailed explanations of the testing loop, the drive system, test section, data acquisition system, visualization system and the working fluid.

5.2 Test Loop

The installation Figure 5.1 consists of an oil reservoir connected to a pump which is powered by a Baldor variable speed motor. The test section (Figure 5.2), is connected to the pump by means of flexible lines and is fully pressurized during operation. The oil which returns to the reservoir is thermally conditioned and then restarts the circuit. A positive displacement flow meter and pressure gauges are installed to control pressures and flow at the pocket’s restrictor inlet. A variable speed 5HP Reliance electric motor is connected to the shaft of the test section by means of a pulley-and-belt system.

5.3 Drive System

The drive system is powered by a Baldor variable speed motor. The motor, which is controlled by an external control box, is used to rotate the aluminum shaft mounted inside the test fixture. The motor which is rigidly mounted to the test stand, is connected
to the shaft (rotor) via a Goodyear brand “v” grooved pulley. The external control box utilizes basic operational, increase/decrease features which are used to control the motors rotational speed. The motors rotational speed is calibrated to yield a relationship between the external control box settings and the actual motor (shaft) speed in rpm.

5.4 Test Section (Apparatus)

Figure 5.2 presents a cross section (a), and a frontal view (b), of the test section. The test section enclosure is made out of Lucite housing, milled with an 8 inch inner diameter bore (1) and polished for optical viewing. An aluminum shaft (4) supported by two precision high speed ball bearings (5) is used to turn a Lucite rotor (2) that is 7.974” in
diameter. When assembled the rotor and housing create a 0.013 in concentric clearance that can be changed by adjusting the position of the massive housing (1).

![Diagram](image)

**Figure 5.2 Cross section and front view of test section**

Inserted into the Lucite housing is the Pocket Assembly (3). The pocket assembly, which is made from a piston and piston housing, is what allows the pocket to vary in depth. For this purpose steel linear guide bearings are attached to the bottom of the housing. The housing can then be moved by means of adjusting precision screws and its position can be monitored by means of two diametrically opposite dial gauges (resolution 5E-4 in).

### 5.5 Variable Depth Pocket

The pocket assembly (3) from Figure 5.3 has two main components, the piston and piston housing. The piston housing (A) Figure 6, is manufactured from Lucite material, which was the same material as the test fixture housing (1) in Figure 5.2. The purpose of this material is to allow viewing of the thin laser sheet illuminating the particles seeded within the pocket. The 1.500” circular piston housing was designed with a 0.700” by 0.700” square bore (Figure 5.3c), milled in the center which represents the
four walls of the square pocket. The largest depth that can be achieved is a depth of 1.700” (AR = 2.42). A slightly larger diameter hole is bored into the end of the housing creating an o-ring guide (Figure 5.3, A) for the second component. The housing is fitted with three pressure transducers, two in the circumferential direction and one in the axial direction. These will measure the static pressure in the surrounding lands adjacent to the pocket. The second component, the piston (Figure 5.3, B), was also manufactured from Lucite material. The piston, which was milled to a 0.700” x 0.700” square inserts into the first component, the piston housing. Drilled into the face of the piston is a 0.200” diameter hole (Figure 5.3, B), which serves as the fluid inlet into the pocket. The face of the piston serves as the floor or bottom of the pocket. It is fitted with three pressure transducers, two in the circumferential direction and one in the restrictor (pocket inlet).

Figure 5.3 Variable depth pocket assembly
The piston is inserted into the piston housing to create the variable depth pocket. The piston can be adjusted using a micrometer (Figure 5.2) to adjust the depth of the pocket. The micrometer has a precision of 0.0001in.

5.6 Data Acquisition System

The NEFF System 470 is used for pressure and volumetric flow data collection. This is a stand-alone front end subsystem controlled through an IEEE-488 (GPIB) card by a host computer. Internally the system is controlled by both microprocessor and ROM software. The NEFF system uses up to 16 function cards which can handle both analog and digital I/O signals. For this experiment, two types of function cards were used to handle input signals stemming from both the pressure transducers and flow meter. The pressure transducers used an analog input card (model 470050) which is a 16-channel and 16-bit resolution, differential multiplexer card. The card has a variety of voltage ranges from +/-5mV to +/-10.24V, which are fully programmable for each of the 16 available channels. Its selectable 1 kHz to 10 kHz sampling rate can be adjusted on the function card. For these sets of experiments, a sampling rate of 10 kHz was chosen for all the channels.

Pressures are measured with Entran miniature pressure transducers with a range 0-100 psi and a time response of 100 kHz. The measurement error is of 1% full scale. The volumetric flow is measured by means of a high precision positive displacement flow meter with a range of 0.1-10gal. This flow meter choice makes unnecessary calibration compensation for density or viscosity changes with temperature. The measurement error of this device is also 1% full scale. Both devices were calibrated before starting the experiments and the calibration curves were entered into the databank of the NEFF.
system. The readings of thermocouples installed in the tank storing the working fluid, is used to verify that fluid temperatures remain constant throughout the experiment.

5.7 Visualization System

Figure 5.7, shows a schematic of the visualization system designed to track the full flow field inside the variable depth pocket. The system uses a high intensity 5 kHz pulsed Nd:YLF laser from Spectra Physics. The light train contains a combination of large radius cylindrical lenses combined with a series of highly polished light directing mirrors to create a thin plane sheet of light that is used to scan the circumferential cross sections of the hydrostatic pocket. The light sheet illuminates the seed particles contained in its plane and allows a video camera equipped with a long distance microscope (LDM) to follow their trajectories. Thus the fluid motion is reconstructed in real time in a Lagrangian fashion. The images are digitally recorded and stored through the computer DMA into hard disk storage. The great advantage of this type of storage vs. a VCR based one, is that resolution of acquisition is preserved when viewing single frames.

5.7.1 Creation of the Laser Sheet

The visualization system consists of the 5 kHz pulsed laser, camera and an array of the lenses and the highly polished mirrors, as shown in Figure 5.4. The Merlin laser, which is manufactured by Spectra-Physics, provides a high intensity light source for the visualization of the seeded particles within the variable depth pocket. The series of lens and mirrors together, create a thin planar sheet of light. This thin sheet of light illuminates the seeded particles in the fluid. The objective of the lens is to transform the
laser beam into a thin sheet of light, which its height and can be controlled. The purpose of the polished mirrors, are to allow for the change of direction of the light sheet.

![Figure 5.4 Sketch of the vision system with LDM-long distance microscope](image)

5.7.2 Visualization of the Test Section

The visualization system, Figure 5.4, has five components: An external computer equipped with Insight version 1.39 software written by TSI, a Merlin 5kHz Yag:LF pulsing laser, a laserPulse 610032 synchronizer, which controls the pulsing of the Merlin 5kHz laser, a Particle Image Velocimetry (PIV) camera (S/N111111), an external circuit, to convert the analog signal produced from the synchronizer, to a signal the laser system can use. The Insight software written by TSI allows the user to set up and simultaneously use the PIV camera, synchronizer and the laser to gather images produced by the seeded particles. The light sheet produced by both, the Merlin laser, lenses and mirrors,
illuminates the seed particles contained in its plane and allows a video camera equipped with a long distance microscope (LDM) to follow their trajectories. Thus the fluid motion is reconstructed in real time in a Lagrangian fashion. The video camera used, is a PIV (Particle Image Velocimetry) camera. This camera allows for multiple images to be taken sequentially. There were a total of nine images taken for each case run, at a rate of 800 microseconds between each image. This allowed for the flow patterns inside the seeded pocket to be seen and discard if any images were not clear or fully developed. The images are digitally recorded and stored through the computer DMA into hard disk storage. From there they can be retrieved as needed.

5.8 Working Fluid

This is a colorless, odorless synthetic silicone oil which has a refractive index matched to that of the Lucite (approximately 1.46). The oil has a density of 888 kg/m³ and a dynamic viscosity of 20 cP at room temperature. The fluid is seeded with microsphere light reflecting particles that are 10-20 microns in size. For the timescale of this experiment the particles are practically neutrally buoyant and the Froude numbers under which the experiments were conducted are large enough to ensure that the particles trajectories follow faithfully the fluid flow.
6.1 Introduction

This section will describe the general calibration process for both the Baldor electric motor and pressure transducers. A brief explanation will show how each component is wired into the NEFF System 470 data acquisition (DAQ) board. Both electric motor speed and pressure transducers have calibration curves, which relate the signal voltage to a shaft speed (rpm) or proportional pressure (psi).

6.2 Calibration of the Pressure Transducers

Pressures are measured with Entran miniature pressure transducers with a range 0-100 psi and a time response of 100 kHz. The measurement error is of 1% full scale. The calibration included the use of a pressurized source (air compressor) connected to the pressure transducers bank. The pressure is controlled using a pressure regulator connected between the pressure hose attached to the air compressor and the pressure bank. For calibrations this pressure regulator is also connected to a high precision Heise analog gauge. The pressure transducers are wired in to the NEFF System 470 data acquisition (DAQ) board and used to determine the voltage associated with the calibration pressure. The pressure defines a highly sensitive membrane equipped with a Wheatstone bridge configuration wired inside each transducer. This deformation creates a change in resistance and due to Ohm’s law, proportionally changes the output voltage.
This voltage curve vs. pressures is recorded in the NEFF system. For each pressure transducer, the pressure from the pressure regulator was increased from 0 – 20 psi in increments of 1 psi. For each increment in pressure the corresponding voltage and pressure were recorded. Figure 6.1 shows the linear calibration curve, obtained for each on of the six pressure transducers. Each transducer was calibrated two times to ensure a proper relationship and error evaluation (Appendix A).

Figure 6.1 Calibration curves for Entran pressure transducers
6.2 Calibration of Motor Speed

The motor speed was controlled by a Baldor variable speed inverter controller.

The calibration of the motor speed was done using a stroboscope. A reference mark was placed on the rotating shaft. As the shaft began rotating the strobe light was actuated. As the frequency of the strobe was increased to match the revolution of the rotating shaft, the mark appeared to become stationary. The shaft rotation parameters were varied from 5-20 Hz in increments of 1 Hz. Both the frequency of the stroboscopic lamp and the shaft rotation parameters were documented and graphed. Figure 6.2 shows the linear relationship between both the shaft rotation (Hz) and shaft speed (rpm).

![Graph](Image)

**Figure 6.2 Calibration of Baldor variable speed motor**

\[ y = 58.625x - 49.833 \]

\[ R^2 = 0.9997 \]
CHAPTER VII

EXPERIMENTAL PROCEDURE

7.1 Experimental Start-up

This section will outline the procedure needed to mix and inject (seed) the pocket with particles.

7.1.1 Experimental Installation set-up

The centering of the test installation, Figure 7.1, is a rather iterative task. In order to achieve a 0.013” concentric clearance (gap) between the shaft (rotor) and housing, the Lucite made front panel, which is used to enclose the entire hydrostatic bearing installation must be disassembled. Once the rotor is exposed, a feeler gauge is set to a thickness of 0.013” and placed into the gap at four different locations in the circumferential direction. The four locations are at 0, 90, 180 and 270 degrees. Indexing these four locations will allow for a constant clearance between the rotor and housing. Figure 7.1 shows the four locations used to measure the clearance. To ensure a constant clearance, one has to iterate between the 0 and 270 degree clearances. For this purpose two indicator gauges are set up, one on the back side of the test installation and mounted on the aluminum shaft and the other on the pocket side of the installation. The gauge on the back side of the fixture is set up to make sure the rotor sits aligned with the housing and that no misalignment in the axial direction exists from the front and back ends of the clearance. The gauge on the pocket side allows for the fine adjustment of the clearance.
7.1.2 Particles

For flow visualization it was necessary to seed the fluid with particles small enough to accurately represent the flow and large enough to reflect the light produced by the thin laser sheet plane so that it can be seen by the camera.

Initially, magnesium oxide (MgO$_2$) flakes with an average dimension of 51 $\mu$m were tried, but the flow images were not good. Then polymer micro-spheres with an average dimension of 91 micrometers were used and yielded very encouraging results. Therefore, polymer micro-spheres with 20-30 $\mu$m were used as the tracer (seeding) particles for all the experiments.

Mixing the particulates together required a graduated cylinder filled with 100-150ml of testing oil. Then about a 1/10 cap full of polymer micro-spheres were added to the testing oil and mixed. The solution of oil and particles were mixed until the tracer particles were evenly distributed within the mixture. The mixed particles were then loaded into a 50ml syringe which was used to inject the particles into the flow.
7.2 Experimental Procedure

This section will outline, (i) the procedure used for the test installation during each experiment and (ii) the procedure needed to implement the Particle Image Velocimetry (PIV) camera and software. The first part, will give information on how to run the test installation and describe its basic functions. The second part will give instructions on how to inject the particulate in the test oil.

7.2.1 Experimental Installation

The steps needed to start and run the test installation (reference Figure 7.2) are outlined below

1) The pumps which control the flow of test oil out of the reservoir must be turned on and set to a 0 Hz (no flow) value.

2) The two power supplies for the flow meter and pressure transducers need to be turned on to their appropriate excitation voltage. The flow meter requires a +/-10-12V and the pressure transducer bank requires +/-5V. Once external power has been activated for both the flow meter and pressure transducers, the NEFF system 470 (DAQ) and Mozayik software must be turned on and scanning.

3) There are two external valves that must be in the open position, one, to allow the test oil to exit the test section and flow back into the reservoir. The other allows the oil to flow from the reservoir pumps and into the test section.

4) Baldor variable speed motor, which controls the shaft (rotor) speed on the fixture and the PIV camera must be turned on

5) To activate the Merlin 5kHz pulse laser, an external valve must be open to allow city water to cool the power supply unit. The power unit, which powers the
Merlin laser, must be switched to the on position. The external control unit, which controls the laser, must be set accordingly. To initially turn on the laser, make sure the laser power setting does not exceed 3.0KW and turn on the lens switch and wait a few moments, next, switch on the shutter. At this point the laser will be activated and a green laser beam will be emitting from the unit.

6) Start the reservoir pumps and allow oil to enter and fill the test section. As the oil fills it, an air bubble will form on the top section of the fixture. At this point, the vent holes, which are mounted on top the fixture must be open and allow the trapped air to escape (Figure 7.1, vent holes).

7) Attach the 50 ml syringe and turn the variable speed motor to the desired speed.

Figure 7.2 Experimental test loop
CHAPTER VIII

EXPERIMENTAL RESULTS

8.1 Experimental Parameters

This section will present experimental results which are documented according to the matrix shown in Figure 8.1. There are two variable operational parameters which will be used throughout this experimental endeavor, (i) shaft (rotor) rotational speed, and (ii) lubricated oil inlet (volumetric flow), and one variable geometric parameter (iii) the pocket aspect ratio. The aspect ratio is defined as a measure of pocket depth to the pocket circumferential chord, \( AR = \frac{D}{L} \) (\( D = \) pocket depth and \( L = \) Pocket length (0.700”)) as shown in Figure 4.2.

The pocket depths are varied from 0.700”, 0.250”, and 0.060”, with the corresponding aspect ratios becoming 1.000, 0.357, and 0.086, respectively. For each aspect ratio the operational parameters (i) and (ii) are varied one at a time. Thus, shaft speeds of 0, 200, 1000 and 1500 rpm respectively are used while the inlet mass flow is kept constant. For volumetric flow variation, the values of 1.23 and 0.615 gpm, respectively are used.

The working fluid for this case is a colorless, odorless synthetic silicone oil which has a refractive index matched to that of the Lucite (approximately 1.46). The oil has a density of 888 kg/m\(^3\) and a dynamic viscosity of 20 cP at room temperature.
Figure 8.1 Matrix of experimental parameters

<table>
<thead>
<tr>
<th>Density (Kg/m³)</th>
<th>Depth viscosity 0.760″ 20 cP</th>
<th>Depth viscosity 0.250″ 20 cP</th>
<th>Depth viscosity 0.060″ 20 cP</th>
<th>Depth viscosity 0.030″ 20 cP</th>
<th>Depth viscosity 0.015″ 20 cP</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flow (GPM)</td>
<td>1.23</td>
<td>1.23</td>
<td>1.23</td>
<td>1.23</td>
<td>1.23</td>
</tr>
<tr>
<td>Shaft Speed (RPM)</td>
<td>200</td>
<td>200</td>
<td>200</td>
<td>200</td>
<td>200</td>
</tr>
<tr>
<td>Flow (GPM)</td>
<td>0.615</td>
<td>0.615</td>
<td>0.615</td>
<td>0.615</td>
<td>0.615</td>
</tr>
<tr>
<td>Shaft Speed (RPM)</td>
<td>1000</td>
<td>1000</td>
<td>1000</td>
<td>1000</td>
<td>1000</td>
</tr>
</tbody>
</table>

There are two types of dominant flow regimes which highly affect the flow patterns within the pocket. The first type, is a pressure (Poiseuille) induced flow. This tends to create symmetrical looking vortical profiles, which become centered on each side of the inlet jet, exiting from the bottom of the pocket. In regions A, B, C and D in Figure 8.2, highlight four major types of flow patterns influenced by the pressure dominated forces. Figure 8.2, regions A and C, shows both the right hand side (RHS) and left hand side (LHS) vortical cells created by pressure forces induced by the jet. Because there is no shaft rotation, no shear forces acts to restrict the motion of the dynamic forces by the fluid jet impinging on the surface of the shaft. Hence, the fluid is free to exit the pocket circumferentially through the both the LHS and RHS clearance. It is the combination of the jet entraining the fluid and the fluid exiting the pocket which forms these recirculating vortical cells. Figure 8.2, region B, shows the fluid jet coming from the restrictor exit, which emerges from the bottom of the pocket. This straight profile looking jet impinges the shaft, causing high pressures to develop on the surface of the shaft. Figure 8.2, region D, show the LHS
Highlights of flow patterns based on dominant flow regimes

(A) Left hand side (LHS) vortical cell created from pressure forces causing fluid to exit the pockets clearance.

(B) Jet exiting restrictor creates dominant pressure forces throughout the pocket.

(C) Right hand side (RHS) vortical cell created from pressure forces causing fluid to exit the pockets clearance.

(D) Fluid exiting upstream pocket through clearance

Figure 8.2 Qualitative view of pressure dominated flows within a pocket
Highlights of flow patterns based on dominant flow regimes

(A) Fluid in the right hand side (RHS) domain entrained by the jet and exits the pocket

(B) Jet exiting restrictor interacts with inertial (shear) forces created by shaft.

(C) Left hand side (LHS) vorticle cell created by the interaction of the turn around zone (TAZ) and inlet jet.

(D) Fluid interacts with inertial effect due to shaft rotation causing a turn around zone (TAZ)

Figure 8.3 Qualitative view of Couette dominated flows within a pocket
clearance when the fluid exits the pocket. It is in the LHS clearance where the direction of flow dictates the dominant force within the pocket. Since all flow exits the pocket, pressure forces dominate the pocket, allowing all fluid to exit.

The second type, is inertial (Couette) dominated flow. It is a result of this Couette force, which overall affects the size and location of the vortical cell configuration, Figure 8.3. Region A, shows particles entrained by the fluid jet exiting the restrictor. This entrainment allows the fluid particles to travel the depth of the pocket and exit through the clearance. Region B, as in the previous, pressure dominated case, shows the fluid jet exiting the restrictor. It is the effect of the jet which causes the pressure forces to compete with the Couette forces developed within the pocket and determines whether or not the fluid enters or exits the pocket through the LHS clearance. Region D, is the location of the LHS clearance of the pocket where the turn around zone (TAZ) forms. In this region, the fluid, which is recirculated in the LHS vortical cell becomes entrained by Couette forces developed due to the shaft rotation. It is this entrainment, which allows the fluid to physically change direction. Region C, shows a LHS vortical cell created from the combined effect of the fluid jet and turn around zone (TAZ). Here outer envelope of the fluid jet entrains the fluid in the pocket, creating the vortical cell profile. Below the recirculating cell, fluid becomes entrained due to the jet as seen in region B, where particles flow from the bottom of the pocket to the clearance where they exit (via the LHS clearance). The Couette forces restrict the flow out of the pocket and the fluid not influenced by the TAZ becomes recirculated.

Under the physical conditions described above the study concentrates on detailing the patterns of the hydrodynamic flow inside the pocket and the associated pressure patterns.
8.2 Results for the Case [Pocket Depth = 0.700, Flow Rate = 1.23gpm]

This section will focus on varying the shaft rotation. The shaft will rotate at three different rotational speeds: 0, 200, 1000, and 1500rpm, respectively. The operational parameter, inlet (volumetric flow) boundary condition will be set to a constant value of 1.23gpm. The geometric parameter, pocket depth, will also be held constant at 0.700” (AR = 1.000).

For the deep pocket, Figure 8.4a, one can distinguish the formation of two symmetrical vortical cells centered on each side of the fluid jet coming from the restrictor exit, which emerges from the bottom of the pocket. These two vortical cells dominate both the upper right hand side (RHS) and left hand side (LHS) domains of the pocket. Due to pressure forces, all fluid exits the pocket circumferentially in both the LHS and RHS clearances. In Figure 8.4b, the shaft (rotor) speed was increased to 200rpm. Still, the observations of two vortical cells are present within the pocket. However, in the LHS clearance of the pocket, inertial effects entrain a layer of fluid allowing it to be carried by the shaft. Here a transition region starts to develop and the Couette forces begin to interact with the pressure effects due to the fluid jet. Figure 8.4c, shaft speed is increased to 1000 rpm. Here the upstream vortical cell still dominates in the LHS portion of the pocket. However, in the RHS of the pocket, there is no longer the presence of a vortical cell profile. Instead, the flow in this region has become fully entrained by the shaft, due to the Couette effects and is carried to the RHS clearance. In the pocket, Couette effects begin to dominate the pressure induced effects, thus allowing more fluid to enter the
Figure 8.4 General view of 0.700” pocket depth (AR = 1.000) showing flow patterns within the pocket. This case examines a flow rate of 1.23gpm with shaft speeds of 0, 200, 1000 and 1500rpm.
pocket (via the LHS clearance). In LHS region of the pocket, adjacent to the clearance, presents a fluid turn around zone (TAZ). In this region, the fluid, which is circulating (via the upstream vertical cell), becomes highly influenced by the inertial forces created by the rotating shaft. Because of this dominant influence, the particle will physically change direction and begin flowing in the direction of the shaft. As a consequence of the TAZ, the LHS vortical cell shifts downward to accommodate the fluids change in direction. In Figure 8.4d, shaft speed is increased to 1500rpm. The vortical cells, which previously dominated the LHS portion of the pocket have slightly decreased in size, where the RHS portion of the pocket, yields no vortical profile, since it is dominated by the Couette shaft effects. There is a drop in pressure due to the accelerating fluid that causes the cell to essentially become “pushed” out of the pocket.

8.3 Results for the Case [Pocket Depth = 0.700, Flow Rate = 0.615gpm]

For the next case, this section will again focus on varying the shaft rotation. The shaft will rotate at six different rotational speeds: 0, 200, 1000, and 1500rpm, respectively. The operational parameter, inlet (volumetric flow) boundary condition will now be changed from the previous 1.23gpm and set to a constant value of 0.615gpm. The geometric parameter, pocket depth, will also be held constant at 0.700” (AR = 1.000).

In, Figure 8.5a, just like in the previous case, one can still distinguish the formation of two symmetrical vortical cells centered on each side of the fluid jet flowing from the restrictor exit, emerging from the bottom of the pocket. These two vortical cells dominate both the upper right hand side (RHS) and left hand side (LHS) domains of the pocket. Due to pressure forces, all fluid exits the pocket circumferentially in both the
LHS and RHS clearances. In Figure 8.5b, shaft (rotor) speed is increased to 200rpm. Still, the observations of two vortical cells are present within the pocket. However, in the LHS clearance of the pocket, Couette effects influence a layer of fluid to become entrained and carried by the shaft. Here a transition region develops and slowly the Couette forces begin to interact with the pressure forces created by the jet exiting the restrictor. In the LHS of the pocket, the pressure forces dominate, thus allowing fluid to exit both the RHS and LHS clearances of the pocket circumferentially. However the not all fluid is exiting the LHS of the pocket. Because of the Couette forces, some fluid is entrained and carried by the shaft causing some fluid to enter the pocket through the clearance. Due to the shear stresses created by the rotating shaft the LHS cell both decrease in overall size and is displaced slightly downward. Figure 8.5c, the shaft speed is increased to 1000 rpm. Here the LHS vortical cell has decreased significantly in overall size and shifted further downward when compared to figure 8.3b, the 200rpm case. This shift downward is caused from the TAZ growing in the LHS portion of the pocket. As the shaft speed increases the TAZ grows in size, due to the Couette forces and essentially “pushes” the LHS vortical cell downwards. In the RHS of the pocket, the vortical cell has grown to encompass the entire domain. As shaft speed increases the RHS vortical cell in side the upper portion of the pocket grows in size. This is caused from the shaft speed accelerating the fluid to a high velocity. Because of the decrease in jet
Figure 8.5 General view of 0.700” pocket depth (AR = 1.000) showing flow patterns within the pocket. This case examines a flow rate of 0.615 gpm with shaft speeds of 0, 200, 1000 and 1500 rpm.
strength, the fluid not exiting the RHS clearance is recirculated into the RHS pocket where it contributes to the growth of the cell. As the shaft speed increases, so does the size of the RHS cell. As the RHS vortical cell grows in size, it traps a secondary vortical cell in the lower RHS corner of the pocket. In the pocket, Couette effects begin to dominate the opposing pressure forces. As a consequence, more fluid begins to enter the LHS of the pocket. In this area of the pocket, adjacent to the clearance, presents a fluid turn around zone (TAZ). In this region, the fluid, which is circulating (via the upstream vortical cell), becomes highly influenced by the Couette forces created by the rotating shaft. Because of this dominant influence, the particle will physically change direction and begin flowing in the direction of the shaft. As a consequence of the TAZ, the LHS vortical cell shifts downward to accommodate the fluids change in direction. In Figure 8.5d, shaft speed is increased to 1500rpm. As shaft speed increases the LHS vortical cell decreases in size until the vortical cell profile is no longer present. This is due to the lower pressure force produced by the jet as a consequence of reduced inlet flow (0.615gpm). Because of the high shaft speed the Couette forces cause the TAZ to grow in size, and the pressure forces from the jet are not enough to sustain the LHS vortical cell.

8.4 Results for the Case [Pocket Depth = 0.250, Flow Rate = 1.23gpm]

For the next case, this section will again focus on varying the shaft rotation. The shaft will rotate at six different rotational speeds: 0, 200, 1000, and 1500rpm, respectively. The operational parameter, inlet (volumetric flow) boundary condition will now be set to a constant value of 1.23. The geometric parameter, pocket depth, will also be held constant at 0.250” (AR = 0.357).
Figure 8.6 General view of 0.250” pocket depth (AR = 0.357) showing flow patterns within the pocket. This case is examines a flow rate of 1.23gpm with shaft speeds of 0, 200, 1000, and 3000rpm.
Figure 8.6a, represents a zero shaft rotation, shows the formation of two symmetrical vortical cells centered between the fluid jet emerging from the restrictor located on the bottom of the pocket. Both vortical cells encompass both the LHS and RHS of the pocket. It is because of the stationary position of the shaft, that pressure forces, due to the fluid jet, dominate the pocket and cause both the symmetric vortical profile and fluid to exit circumferentially in both the LHS and RHS clearance. In Figure 8.6b, shaft speed is increased to 200rpm, as in the no rotation case, this yields similar vortical profiles. However, because of added Couette forces due to the shaft rotation, fluid becomes entrained by the jet exiting the restrictor. In the LHS of the pocket clearance, fluid is both exiting the pocket, due to pressure forces and entering the pocket, due to Couette forces. On the upstream side, pressure forces still dominate the Couette forces, the majority is the fluid exits the LHS pocket. In the pocket, the vortical cell is slightly displaced downward toward the bottom of the pocket by the TAZ. This shift is direct a result of the growing Couette forces which allow the TAZ zone to grow downward into the pocket. Since more fluid now enters the LHS clearance, through continuity, more fluid must exit the RHS clearance of the pocket. Due to this increased flow rate out of the pocket, the RHS of the pocket is entrained by the fluid jet. Particles are then carried to the clearance to exit the pocket.

8.5 Results for the Case [Pocket Depth = 0.250, Flow Rate = 0.615gpm]

For the next case, this section will again focus on varying shaft rotation. The shaft will rotate at six different rotational speeds: 0, 200, 1000, and 1500rpm, respectively. The operational parameter, inlet (volumetric flow) boundary condition will now be
changed from the previous 1.23gpm and set to a constant value of 1.23gpm. The geometric parameter, pocket depth, will also be held constant at 0.250" (AR = 0.357).

Figures 8.7a, b, show similar vortical profiles as in the previous 1.23gpm case. Here one can see the two symmetric vortical cells centered between fluid jet exiting the bottom of the pocket. These large cells encompass both the entire LHS and RHS regions of the pocket. The top portion of the LHS cell is elongated and stretched to the upper portion of the clearance. Neither of the cells are perfectly round, the upper portion of each cell having an elongation which is a consequence of the fluid exiting the pocket and in a sense, pulling the vortical cells toward the clearance. These cells are formed from the jet entraining the fluid from both the LHS and RHS domains. The entrained fluid is then carried from the bottom of the pocket to the top where the fluid exiting the pocket is forced to recirculate, thus forming these cell profiles. Figure 8.7c, presents the flow formations where the shaft is rotating at 1000rpm. Here, the LHS vortical cell still dominates this portion of the pocket. However, the elongation is more defined for this case than the previous, 0 and 200 rpm case. The fluid jet is still seen emerging from the bottom of the jet and impinging the rotating shaft. In the RHS of the pocket, there are two types of phenomena present. First, the vortical cell which previously dominated this region has greatly reduced in size. Now only a small vortical profile can be seen in this upper portion of the pocket near the clearance. The other, is the fluid in the rest of the region is entrained by the jet and exits through the RHS clearance. In Figure 8.5d, represents a shaft rotation of 1500rpm. Here the LHS vortical cell can be seen highly elongated toward the clearance of the pocket as previously motioned before. The fluid
Figure 8.7 General view of 0.250” pocket depth (AR = 0.357) showing flow patterns within the pocket. This case is examines a flow rate of 0.615gpm with shaft speeds of 0, 200, 1000, and 1500rpm.
jet, which exits the restrictor at the bottom of the pocket has a straight profile which impinges the surface of the shaft. In the RHS region of the pocket, the small vortical cell which was seen in the previous 1000rpm case, is almost no longer present. However, there is a presence of a large recirculating region which encompasses the entire RHS region. This large recirculating region is caused from the entrained fluid, which has not exited the pocket, to be redirected into the pocket.

8.6 Results for the Case [Pocket Depth = 0.060, Flow Rate = 1.23gpm]

This section will focus on varying the shaft rotation. The shaft will rotate at three different rotational speeds: 0, 200, 1000, and 1500rpm, respectively. The operational parameter, inlet (volumetric flow) boundary condition will be set to a constant value of 1.23gpm. The geometric parameter, pocket depth, will also be held constant at 0.060” (AR = 0.086).

Figures 8.8a, b., shows a shaft rotation of 0 and 200rpm, respectively. In these figures, the formation of two symmetrical vortical profiles can be seen. Both the LHS and RHS profiles occupy the entire depth of the pocket. The fluid jet, exiting the bottom of the pocket has a straight profile from the exit of the restrictor to the top surface of the shaft. Both the LHS and RHS vortical cells have a round profiles, not seen in the 0.700” pocket depth of Figure 8.8. Figure 8.8c, represents a shaft speed of 1000rpm. In the LHS, the vortical cell becomes elongated and no longer exhibits its round profiles as previously seen in figures 8.8a, b. However, in the RHS region, the vortical cell is almost non existent. For this case the Couette forces are the dominant forces over the pressure forces. At this point, fluid is no longer exiting the LHS.
Figure 8.8 General view of 0.060” pocket depth (AR = 0.086) showing flow patterns within the pocket. This case examines a flow rate of 1.23gpm with shaft speeds of 0, 200, 1000, and 1500rpm.
clearance of the pocket. Instead all fluid now exits the RHS clearance, causing the vortical cell to become displaced upward toward the clearance. Figure 8.6d, represents a shaft speed of 1500rpm. In the LHS of the pocket the vortical cell, which is almost non existent, has decreased greatly in size and shifted downward. This downward shift is caused from the TAZ becoming larger due to the increased inertial forces caused from the higher shaft speeds. In the RHS region of the pocket the vortical cell is no longer present. This is because the fluid jet does not have space to develop its effects and little fluid is stored in the pocket.

8.7 Results for the Case [Pocket Depth = 0.060, Flow Rate = 0.615gpm]

This section will focus on varying the shaft rotation. The shaft will rotate at three different rotational speeds: 0, 200, 1000, and 1500rpm, respectively. The operational parameter, inlet (volumetric flow) boundary condition will be set to a constant value of 0.615gpm. The geometric parameter, pocket depth, will also be held constant at 0.060” (AR = 0.086).

Figures 8.9a,b yield similar vortical profiles as in the previous 1.23gpm case. Here one can see the two symmetric vortical cells centered between fluid jet exiting the bottom of the pocket. These large cells encompass both the entire LHS and RHS regions of the pocket. Each of the two cells are elongated and stretched horizontally across the length of the pocket. These cells are formed from the jet entraining the fluid from both the RHS and LHS domains. The entrained fluid is then carried from the bottom of the pocket to the top where the fluid not exiting the pocket is forced to recirculate, thus forming these cell profiles. Figure 8.9c, represents a shaft rotation of 1000rpm. Here, the LHS vortical looks similar in profile to the previous 200rpm case. The RHS vortical cell has decreased
in overall size. This is caused from the increased flow of the fluid exiting the pockets clearance. Figure 8.9d, represents a shaft rotation of 1500rpm. On the LHS of the pocket, there is no longer the presence of a vortical cell profile. Instead the fluid is entrained from the outer envelope of the fluid jet and carried from the bottom of the pocket to the top where the TAZ is located. The fluid, which is influenced from the TAZ, is carried by the shaft to the LHS region of the pocket, where the fluid exits the clearance of the pocket.

Figure 8.9 General view of 0.060” pocket depth (AR = 0.086) showing flow patterns within the pocket. This case examines a flow rate of 0.615gpm with shaft speeds of 0, 200, 1000, and 1500rpm.
CHAPTER IX

CONCLUSIONS

9.1 Introduction

The numerical results obtained from CFD-ACE+ closely match the experimental results obtained from the test performed and presented in chapter 8. In this section, both a qualitative and quantitative approach was used to describe the physical effects cause from the change in parameters. These parameter changes had a great affect on the overall flow and pressure patterns developed within the variable depth pocket.

9.2 Experimental and Numerical Observations

Parametric studies of flow and pressure profiles were used to compare both the numerical and experimental results. The comparison documented the effect of two operational parameters, (i) shaft rotation and (ii) inlet condition (volumetric flow) and one geometric parameter, (iii) the pocket aspect ratio on the steady state flow within the pocket. Results conclude that there are two major types of flow regimes present, which can be categorized as either pressure (Poiseuille) dominated or inertial dominated forces. Pressure dominated flows allow fluid the exit the LHS clearance of the pocket. While inertial dominated flows, does not allow any fluid to exit the LHS clearance, but only enter the pocket. Both types of flows highly affect the vortical cell profiles within the LHS and RHS of the pocket. For purely pressure driven flows, the velocity profile within the pocket yields a symmetric vortical cell profile centered between the fluid jet exiting the restrictor at the bottom of the pocket. The symmetric profile is due to the lack shaft
rotation, which as a consequence, inhibits the inertial forces. Therefore, allowing all fluid entering the pocket to exit both the LHS and RHS clearances of the pocket circumferentially.

Further conclusions confirm that varying parameters caused vortical cells to shift within the pocket. This shift is partly due to the growing turn around zone (TAZ) developed from the increased inertial forces due to higher shaft speeds. As shaft speed increases the growing inertial forces influence the surrounding layer of fluid near the surface of the shaft. This fluid becomes entrained by the shaft, causing fluid originally destined to exit the LHS clearance, to physically change direction and flow with the shaft. The fluid is then carried to the RHS clearance where it either exits the pocket or becomes recirculated within the vortical cell. As shaft speed increases the TAZ grows from inside the clearance, to the upper portion of the pocket. This growth causes the LHS vortical cell to shift downward into the pocket.

Another parameter which affects the flows within the pocket is the one geometric parameter, (iii) aspect ratio. The aspect ratio, which is a measure of the pocket cord length to pocket depth, greatly influenced the size and location of the LHS, RHS and recirculation zones within the pocket and upper lands. A large aspect ratio of 1.000 (pocket depth 0.700”), has both vortical cell profiles located in the upper corners of the pocket adjacent to the clearance. However, as the aspect ratio is decreased, the vortical cells shift from the upper corners, to the lower region of the pocket near the fluid jet. This shift in cell location can be attributed to the growing TAZ within the small pocket depth. For a large pocket depth, the growth of the TAZ from the clearance into the upper regions on the pocket is rather insignificant. However, for small pocket depth with aspect
ratios of 0.086 and 0.021 (0.030” and 0.060”), any growth of the TAZ which causes any shift in the vortical cell is significant.

Pressures within the pocket vary according to the two operational parameters (i) and (ii) and one geometric parameter (iii). Due to the concentric nature of the shafts, pressures in the clearance, in the circumferential direction are typically zero. However in the regions close to the pocket and within the pocket itself fluctuate greatly. For the deep pocket, pressures are nearly constant due to the large pocket depth. This can be seen as a flat line in the pressure profiles for pocket depths of 0.700”, 0.250” and 0.060”. The fluid jet, exiting the restrictor at the bottom of the pocket, impinges the surface of the jet causing a pressure spike to form in the center of the pressure profile. As shaft speed increases, the pressures within the pocket, as well as the effect of the pressure spike increase. For shallow pocket depths, the pressure in the pocket is no longer constant. For high flow rates (1.23gpm), pressure in the LHS region of the pocket are lower than the RHS of the pocket. The higher pressures in the RHS of the pocket are due to Raleigh effect. Fluid no longer exits the LHS clearance, and through continuity, there is increased flow through the RHS clearance which causes a pressure increase. The fluid jet impingement on the shaft surface is much larger than the previous deep pocket cases. Here the pressures produced by the jet are much larger than the actual pressure inside the pocket. As the inlet flow rate in decrease (0.615gpm), we see an opposite affect. Here the LHS of the pocket yields higher pressures than the RHS of the pocket. The overall pressures within the pocket have decrease, from the reduction of inlet flow.
9.3 Summary of Conclusions

1) For zero shaft rotation, pressure (Poiseuille) forces dominate the pocket.

2) Pressure forces are due to the fluid jet exiting the restrictor at the bottom of the pocket.

3) In the LHS and RHS regions, symmetrical vortical cells are developed during zero shaft rotation and driven by the outer envelope of the fluid jet.

4) As shaft speed increases, Couette forces begin to interact with the pressure forces. The dominant force is determined based on the flow in the LHS clearance of the pocket. If the majority of the flow enters the pocket through the LHS clearance, Couette forces dominate. If flow exits the pocket through the LHS clearance, pressure forces dominate.

5) The pressure profiles within the pocket are constant for deep pockets and vary for shallow pockets.

6) A pressure spike forms in the center of the pressure profile (inside the pocket) due to the fluid jet impinging the surface of the shaft.
7) When Couette forces dominate the pocket, Figure 9.1, fluid can no longer exit the LHS clearance [A], the fluid is essentially trapped, since the pressure forces are relatively large due to the high inlet flow rate [B], the outer envelop [C] of the fluid jet sustains the LHS vortical cell.

![Figure 9.1 Deep pocket conclusion 1](image)

8) As shaft speed is increased, Figure 9.2, the Couette forces cause the RHS vortical cell to decrease in overall size and eventually disappear [D]. This is caused from the increased shaft speed accelerating the fluid to a higher velocity. This high velocity causes a Bernoulli effect in the upper RHS of the pocket, causing pressure to drop. The pressure in the region below the RHS cell [E] is higher then in the clearance, this pressure differential causes the RHS vortical cell to shift upward toward the clearance and eventually to become “pushed” out of the pocket.
9) As shaft speed increases (Figure 9.3) the LHS vortical cell decreases in size [A] until the vortical cell profile is no longer present. This is due to the lower pressure force produced by the jet as a consequence of reduced inlet flow (0.615gpm). Because of the high shaft speed the Couette forces cause the TAZ to grow in size, and the pressure forces from the jet are not enough to sustain the LHS vortical cell.

10) As shaft speed increases the RHS vortical cell (Figure 9.3) inside the upper portion of the pocket grows in size [B]. This is caused from the shaft speed accelerating the fluid to a high velocity. Because of the decrease in jet strength, the fluid not exiting the RHS clearance is recirculated into the RHS pocket where it contributes to the growth of the cell. As the shaft speed increases, so does the size of the RHS cell.

11) As the RHS vortical cell grows in size (Figure 9.3), it traps a secondary vortical cell in the lower RHS corner of the pocket [C].
Figure 9.3 Deep pocket conclusion 3

[A] LHS vortical cell decreases in size due to increased shaft rotation

[B] RHS vortical cell increases in size due to increased shaft rotation

[C] Secondary Vortical Cell
BIBLIOGRAPHY


Krga, Vladimir, (1997), ‘Experimental Investigation of Flow and Pressure Patterns in a Six-Pocket Hydrostatic Bearing’, University of Akron, Department of Mechanical Engineering, M.S. thesis


CFD-ACE+ User Manual, ESI CFD Group, Huntsville, Alabama, 2002
APPENDICES
## APPENDIX A

THE RESULTS OF CALIBRATION OF THE PRESSURE TRANSDUCERS

<table>
<thead>
<tr>
<th>Trial #1</th>
<th>Channels</th>
<th>6</th>
<th>5</th>
<th>4</th>
<th>3</th>
<th>7</th>
<th>2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure (Psi)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>0.0000</td>
<td>0.0007</td>
<td>-0.0002</td>
<td>-0.0026</td>
<td>0.0019</td>
<td>0.0005</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.0009</td>
<td>0.0023</td>
<td>0.0006</td>
<td>-0.0019</td>
<td>0.0030</td>
<td>0.0021</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>0.0015</td>
<td>0.0037</td>
<td>0.0014</td>
<td>-0.0011</td>
<td>0.0040</td>
<td>0.0036</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.0021</td>
<td>0.0051</td>
<td>0.0022</td>
<td>-0.0004</td>
<td>0.0050</td>
<td>0.0050</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>0.0029</td>
<td>0.0066</td>
<td>0.0030</td>
<td>0.0033</td>
<td>0.0060</td>
<td>0.0065</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>0.0036</td>
<td>0.0081</td>
<td>0.0038</td>
<td>0.0011</td>
<td>0.0070</td>
<td>0.0080</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>0.0043</td>
<td>0.0096</td>
<td>0.0047</td>
<td>0.0019</td>
<td>0.0081</td>
<td>0.0095</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>0.0050</td>
<td>0.0110</td>
<td>0.0055</td>
<td>0.0026</td>
<td>0.0091</td>
<td>0.0110</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>0.0057</td>
<td>0.0126</td>
<td>0.0063</td>
<td>0.0033</td>
<td>0.0101</td>
<td>0.0125</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>0.0064</td>
<td>0.0140</td>
<td>0.0071</td>
<td>0.0040</td>
<td>0.0111</td>
<td>0.0139</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>0.0071</td>
<td>0.0155</td>
<td>0.0079</td>
<td>0.0046</td>
<td>0.0121</td>
<td>0.0153</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Trial #2</th>
<th>Channels</th>
<th>6</th>
<th>5</th>
<th>4</th>
<th>3</th>
<th>7</th>
<th>2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure (Psi)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>0.0000</td>
<td>0.0007</td>
<td>-0.0002</td>
<td>-0.0026</td>
<td>0.0019</td>
<td>0.0005</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.0008</td>
<td>0.0023</td>
<td>0.0006</td>
<td>-0.0018</td>
<td>0.0030</td>
<td>0.0021</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>0.0014</td>
<td>0.0036</td>
<td>0.0014</td>
<td>-0.0011</td>
<td>0.0040</td>
<td>0.0035</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.0021</td>
<td>0.0052</td>
<td>0.0022</td>
<td>-0.0004</td>
<td>0.0050</td>
<td>0.0050</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>0.0029</td>
<td>0.0066</td>
<td>0.0030</td>
<td>0.0033</td>
<td>0.0060</td>
<td>0.0064</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>0.0036</td>
<td>0.0081</td>
<td>0.0038</td>
<td>0.0011</td>
<td>0.0070</td>
<td>0.0080</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>0.0042</td>
<td>0.0096</td>
<td>0.0046</td>
<td>0.0018</td>
<td>0.0080</td>
<td>0.0095</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>0.0050</td>
<td>0.0110</td>
<td>0.0056</td>
<td>0.0026</td>
<td>0.0090</td>
<td>0.0109</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>0.0057</td>
<td>0.0125</td>
<td>0.0062</td>
<td>0.0033</td>
<td>0.0101</td>
<td>0.0124</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>0.0063</td>
<td>0.0139</td>
<td>0.0070</td>
<td>0.0040</td>
<td>0.0110</td>
<td>0.0138</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>0.0071</td>
<td>0.0155</td>
<td>0.0079</td>
<td>0.0046</td>
<td>0.0121</td>
<td>0.0153</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Average</th>
<th>Channels</th>
<th>6</th>
<th>5</th>
<th>4</th>
<th>3</th>
<th>7</th>
<th>2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure (Psi)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>0.0000</td>
<td>0.0007</td>
<td>-0.0002</td>
<td>-0.0026</td>
<td>0.0019</td>
<td>0.0005</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.0008</td>
<td>0.0023</td>
<td>0.0006</td>
<td>-0.0018</td>
<td>0.0030</td>
<td>0.0021</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>0.0014</td>
<td>0.0036</td>
<td>0.0014</td>
<td>-0.0011</td>
<td>0.0040</td>
<td>0.0035</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.0021</td>
<td>0.0052</td>
<td>0.0022</td>
<td>-0.0004</td>
<td>0.0050</td>
<td>0.0050</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>0.0029</td>
<td>0.0066</td>
<td>0.0030</td>
<td>0.0033</td>
<td>0.0060</td>
<td>0.0065</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>0.0036</td>
<td>0.0081</td>
<td>0.0038</td>
<td>0.0011</td>
<td>0.0070</td>
<td>0.0080</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>0.0042</td>
<td>0.0096</td>
<td>0.0046</td>
<td>0.0018</td>
<td>0.0080</td>
<td>0.0095</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>0.0050</td>
<td>0.0110</td>
<td>0.0056</td>
<td>0.0026</td>
<td>0.0090</td>
<td>0.0109</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>0.0057</td>
<td>0.0125</td>
<td>0.0062</td>
<td>0.0033</td>
<td>0.0101</td>
<td>0.0124</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>0.0063</td>
<td>0.0139</td>
<td>0.0070</td>
<td>0.0040</td>
<td>0.0110</td>
<td>0.0138</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>0.0071</td>
<td>0.0155</td>
<td>0.0079</td>
<td>0.0046</td>
<td>0.0121</td>
<td>0.0153</td>
<td></td>
</tr>
</tbody>
</table>
APPENDIX B

DETAILED VIEWS OF NUMERICAL SIMULATIONS

0.060” Pocket Depth, Flow rate = 1.23gpm, Shaft Speed =

No flow out of the pocket. Couette flow completely dominates the pressure induced

Layer carried by the

Pressure induced flow dominates the Couette

Flow out of the pocket

Layer carried by the

Upstream Vortical Cell (UVC)

Downstream Vortical Cell (DVC)

Turn Around Zone (TAZ)

Upstream Vortical Cell (UVC)

Fluid entrained by

0.060” Pocket Depth, Flow rate = 1.23gpm, Shaft Speed =
Flow out of the pocket
Layer carried by the shaft

Pressure induced flow dominates the Couette flow

Upstream Vortical Cell (UVC)
Downstream Vortical Cell (DVC)

0.060” Pocket Depth, Flow rate = 0.615gpm, Shaft Speed = 200rpm

No flow out of the pocket. Couette flow completely dominates the pressure induced flow
Layer carried by the shaft

Turn Around Zone (TAZ)
Fluid Entrained by Jet
Large downstream vorticle cell
Secondary vorticle cell

0.060” Pocket Depth, Flow rate = 0.615gpm, Shaft Speed = 2000rpm
0.250” Pocket Depth, Flow rate = 1.23gpm, Shaft Speed = 200rpm

No flow out of the pocket. Couette flow completely dominates the pressure induced flow

0.250” Pocket Depth, Flow rate = 1.23gpm, Shaft Speed = 2000rpm
0.250" Pocket Depth, Flow rate = 0.615 gpm, Shaft Speed = 200 rpm

0.250" Pocket Depth, Flow rate = 0.615 gpm, Shaft Speed
Layer carried by the shaft

Flow out of the pocket

Upstream Vortical Cell (UVC)

Downstream Vortical Cell (DVC)

Pressure induced flow dominates the Couette flow

0.060” Pocket Depth, Flow rate = 1.23gpm, Shaft Speed = 200rpm

No flow out of the pocket. Couette flow completely dominates the pressure induced flow

Layer carried by the shaft

Turn Around Zone (TAZ)

Upstream Vortical Cell (UVC)

Fluid entrained by jet

0.060” Pocket Depth, Flow rate = 1.23gpm, Shaft Speed = 2000rpm
0.060” Pocket Depth, Flow rate = 0.615gpm, Shaft Speed = 200rpm

Couette forces stronger than previous 1.23 gpm case, more fluid entering pocket

Layer carried by the shaft

Pressure induced flow dominates the Couette flow

Flow out of the pocket

0.060” Pocket Depth, Flow rate = 0.615gpm, Shaft Speed = 2000rpm

No flow out of the pocket. Couette flow completely dominates the pressure induced flow

Layer carried by the shaft

Turn Around Zone (TAZ)
0.015" Pocket Depth, Flow rate = 1.23gpm, Shaft Speed = 200rpm

Layer carried by the shaft
Pressure induced flow dominates the Couette flow
Downstream Vortical Cell (DVC)

No flow out of the pocket. Couette flow completely dominates the pressure induced flow
Layer carried by the shaft

0.015" Pocket Depth, Flow rate = 1.23gpm, Shaft Speed = 2000rpm
0.015” Pocket Depth, Flow rate = 0.615gpm, Shaft Speed = 200rpm

No flow out of the pocket. Couette flow completely dominates the pressure induced flow

0.015” Pocket Depth, Flow rate = 0.615gpm, Shaft Speed = 2000rpm