STUDY OF IMPACT OF ORBIT PATH, WHIRL RATIO AND CLEARANCE
ON THE FLOW FIELD AND ROTORDYNAMIC COEFFICIENTS FOR A
SMOOTH ANNULAR SEAL

A Thesis
by
AARTHI SEKARAN

Submitted to the Office of Graduate Studies of
Texas A&M University
in partial fulfillment of the requirements for the degree of
MASTER OF SCIENCE

August 2009

Major Subject: Mechanical Engineering
STUDY OF IMPACT OF ORBIT PATH, WHIRL RATIO AND CLEARANCE
ON THE FLOW FIELD AND ROTORDYNAMIC COEFFICIENTS FOR A
SMOOTH ANNULAR SEAL

A Thesis

by

AARTHI SEKARAN

Submitted to the Office of Graduate Studies of
Texas A&M University
in partial fulfillment of the requirements for the degree of

MASTER OF SCIENCE

Approved by:

Chair of Committee,         Gerald L. Morrison
Committee Members,          Je-Chin Han
                            Othon Rediniotis

Head of Department,         Dennis O’Neal

August 2009

Major Subject: Mechanical Engineering
Study of Impact of Orbit Path, Whirl Ratio and Clearance on the Flow Field and
Rotordynamic Coefficients for a Smooth Annular Seal. (August 2009)
Aarthi Sekaran, B.E., Vasavi College of Engineering, India
Chair of Advisory Committee: Dr. Gerald L. Morrison

The study of the effect of different orbit paths and whirl ratios on the
rotordynamic coefficients of a smooth eccentric annular seal, using Computational Fluid
Dynamics (CFD) was performed. The flow was simulated for two different orbits – linear
and circular for orbit speeds ranging from 0 to 1. This was done using the FLUENT CFD
code with a time – dependent solver which allowed the use of dynamic meshing and User
Defined Functions (UDFs). The effect of clearance was also studied by simulating the
flow through an eccentric seal with one-tenth the clearance and comparing the results.

It was seen that the flow field varies significantly with both the change in orbit
and clearance and this in turn affects the forces and rotordynamic coefficients. The linear
orbit showed major changes in terms of both the flow fields and the resulting forces. The
velocities, pressure magnitudes and forces were much larger than the circular orbit.
Another important finding was that the behavior of the flow for the smaller clearance is
viscosity dominated compared to the inertia dominated flow seen for large clearances.
The computation of rotordynamic coefficients for the circular orbits used Childs’ theory
and it was seen that for larger clearances the CFD predictions were not in agreement with
the expected trends from this theory. The smaller clearance simulations, however, show force predictions from which the rotordynamic coefficients obtained match the theory.
To the one person who is behind my every inspiration, my every smile.
I would like to thank my committee chair, Dr. G.L. Morrison who has been a constant source of support and encouragement throughout the course of this research. I would also like to express my gratitude to Dr. J.C. Han and Dr. O. Rediniotis for their guidance and for taking the time to serve on my committee.

Furthermore, I would like to thank Dr S.H. Park for being really patient and answering all my numerous queries during the initial stages of the project. I would also like to acknowledge the support of the Turbomachinery Research Consortium which made this work possible. Thanks also to my friends and colleagues and the department faculty and staff for making my time at Texas A&M University a memorable experience.

Finally, thanks to my family for all their love and encouragement.
NOMENCLATURE

c           Nominal clearance between rotor and stator
C_{xx}, C_{xy} Direct and cross coupled damping coefficients
d           Rotor diameter
e           Rotor eccentricity ratio
K_{xx}, K_{xy} Direct and cross coupled stiffness coefficients
L           Rotor length
M_{xx}, M_{xy} Direct and cross coupled inertia coefficients
P           Pressure
R           Rotor radius
Re          Axial Reynolds number
Ta          Taylor number
U_{axial}   Mean axial velocity
U_{ave}     Bulk averaged axial velocity, 7.4m/s
U_{radial}  Mean radial velocity
U_{theta}   Mean azimuthal velocity
W_{sh}      Surface rotational speed of shaft
\mu         Dynamic viscosity, 7.84 \times 10^{-4}kg/m-s
\nu          Kinematic viscosity, 0.001003kg/m-s
\rho         Density, 998.2kg/m^3
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABSTRACT</td>
<td>iii</td>
</tr>
<tr>
<td>DEDICATION</td>
<td>v</td>
</tr>
<tr>
<td>ACKNOWLEDGEMENTS</td>
<td>vi</td>
</tr>
<tr>
<td>NOMENCLATURE</td>
<td>vii</td>
</tr>
<tr>
<td>TABLE OF CONTENTS</td>
<td>viii</td>
</tr>
<tr>
<td>INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>LITERATURE REVIEW</td>
<td>3</td>
</tr>
<tr>
<td>Experimental studies involving flow fields</td>
<td>3</td>
</tr>
<tr>
<td>Numerical studies involving flow fields</td>
<td>7</td>
</tr>
<tr>
<td>Studies involving rotordynamic coefficients</td>
<td>8</td>
</tr>
<tr>
<td>OBJECTIVES AND METHODOLOGY</td>
<td>12</td>
</tr>
<tr>
<td>CODE DESCRIPTION</td>
<td>16</td>
</tr>
<tr>
<td>Physical models of fluid mechanics</td>
<td>16</td>
</tr>
<tr>
<td>Modeling turbulence</td>
<td>17</td>
</tr>
<tr>
<td>NUMERICAL MODEL</td>
<td>21</td>
</tr>
<tr>
<td>Description of solution method</td>
<td>21</td>
</tr>
<tr>
<td>Linearization</td>
<td>22</td>
</tr>
<tr>
<td>Discretization</td>
<td>23</td>
</tr>
<tr>
<td>Pressure velocity correction</td>
<td>26</td>
</tr>
<tr>
<td>CONSTRUCTION AND IMPLEMENTATION OF DYNAMIC MESHES</td>
<td>29</td>
</tr>
<tr>
<td>Using dynamic meshes in FLUENT</td>
<td>29</td>
</tr>
<tr>
<td>Describing rotor motion using UDFs</td>
<td>32</td>
</tr>
<tr>
<td>Compiling the UDF and using it in FLUENT</td>
<td>41</td>
</tr>
<tr>
<td>RESULTS AND DISCUSSION FOR ANNULAR SEAL OF 50mil CIRCULAR ORBIT</td>
<td>44</td>
</tr>
<tr>
<td>Experimental results and comparisons for a statically eccentric annular seal..</td>
<td>45</td>
</tr>
<tr>
<td>Experimental results and comparisons for an eccentric annular seal at a whirl ratio of 1.0</td>
<td>50</td>
</tr>
<tr>
<td>Effects of varying whirl ratio</td>
<td>53</td>
</tr>
<tr>
<td>Computation of rotordynamic coefficients</td>
<td>57</td>
</tr>
</tbody>
</table>

**RESULTS AND DISCUSSION FOR ANNULAR SEAL OF 50 mil CLEARANCE WHIRLING IN A LINEAR ORBIT**

- Results for eccentric annular seal where the orbit path speed is the same as rotor speed | 61 |
- Effect of varying speed of rotor orbit | 67 |
- Analysis of resulting forces | 69 |

**RESULTS AND DISCUSSION FOR ANNULAR SEAL OF 5 mil CLEARANCE WHIRLING IN A CIRCULAR ORBIT**

- Results for eccentric whirling seal of 5mil clearance at whirl ratio of 1.0 compared against seal of 50 mil clearance | 73 |
- Effect of varying whirl ratio | 76 |
- Impact of varying mass flow rate on flow field | 79 |
- Computation of rotordynamic coefficients | 80 |

**SUMMARY AND CONCLUSIONS** | 83 |

**REFERENCES** | 88 |

**APPENDIX A** | 92 |

**APPENDIX B** | 139 |

**APPENDIX C** | 142 |

**VITA** | 146 |
INTRODUCTION

Turbomachines today are extensively used for a variety of applications in industry which ensures constant efforts to increase their performance and efficiency. Increasing the performance, more often than not, requires operation at higher speeds which in turn leads to the equipment being subject to forces of high magnitude. In order to ensure stable operation of the system, it is essential to control these forces and maintain them in an optimum range. An important component that has a considerable effect on these factors is the annular seal. In addition to influencing the stability of the system, these seals also act as leakage control devices thus minimizing secondary flow in turbomachines.

Annular seals are simple in geometry and resemble plain journal bearings; the clearances for seals however are larger and this coupled with high pressure drops across the seal make for highly turbulent flow. This flow is important to analyze as it determines the performance of the seal as well as the forces generated which act upon the rotor. It has been observed [1] that unbalanced forces result in eccentric positions of the rotor, followed by whirling. Whirling involves the precession of the center of the rotor around the center of the stator where the whirl ratio is defined as the ratio of the precession rate of the rotor to the rotation speed of the shaft. Seals can help stabilize or destabilize the rotor motion depending upon the seal design.

Research performed in this field is so far predominantly limited to the study of the

This thesis follows the style of the Journal of Turbomachinery.
rotordynamic coefficients. Experimental data at the Turbomachinery Laboratory includes a fairly extensive amount of material from studies performed on smooth annular seals using results obtained from LDA measurements. Although these experiments span a range of flow conditions, the limitations of the experimental set-up make it hard to obtain the data for any arbitrary combination of parameters that one might need to investigate. Thus, the need for an alternative method of study is required which can be fulfilled by CFD. The advantages of using a CFD model are further magnified by the fact that it provides all the data important for analysis, i.e. - pressure fields, velocity profiles and turbulence measurements at any position throughout the length of the seal.

The simulations will be run using the flow parameters from the experiments conducted by Johnson [2] and Thames [3], where the (axial) Reynolds number and Taylor number are defined as –

\[ Re = \frac{2Uc}{\nu} \]  

\[ Ta = \frac{cW_{sh}}{\nu} \sqrt{\frac{2c}{d}} \]

where \( U \) is the bulk velocity, \( c \) is the radial clearance of the annulus, \( \nu \) is the kinematic viscosity of the fluid, \( W_{sh} \) is the surface rotational speed of the shaft, and \( d \) is the rotor diameter.
LITERATURE REVIEW

The introduction provides an overview of the importance of annular seals in turbomachinery as well as a brief idea of the work accomplished in the area so far. This section will elaborate on past research as well as illustrate the need for the current study. The literature review has been composed so as to discuss the two major areas under research separately – studies limited to the rotordynamic properties and those that deal with flow fields in annular seals (mainly eccentric annular seals).

Experimental Studies Involving Flow Fields

Research in annular seals dates back to the mid 1970’s when their impact on the operation of turbomachinery was recognized. One of the very first attempts to model and analyze the fluid flow in thin films was by Hirs [4], who developed as bulk flow theory for turbulence in lubricant films. The theory was basically developed for turbulent flow between smooth surfaces and was such that the empirical constants required could be derived from bulk flow measurements, and did not require the determination of flow velocity profiles. Results were found to provide good agreement with the law-of-wall theory for turbulent flows in bearings at high Reynolds numbers but the theory was not very effective for low Reynolds numbers.

Brennen [5] performed a study of the flow in an annulus surrounding a circular cylinder. The goal was to understand the influence of the flow conditions on whirl. The study was an analysis where the dynamic forces on the cylinder, due to the fluid, were computed. The interactions between the rotor and the fluid were found to be dependent on the whirl deflection which was in turn dependent on the forces in the outer annulus.
This indicated the possibility of whirl instabilities for whirl ratios less than one and also that for a high Reynolds number, the mean flow could be unstable to Taylor vortices above a critical Taylor number.

Lessen [6] performed an analytical study of the flow in a dynamically eccentric whirling annular seal within the Taylor vortex regime (Ta>41.3; the Taylor number here is defined as \( Ta = \frac{a \Omega^2 c^3}{2 u^2} \), where \( \Omega \) is the angular velocity of the rotor). The axial flow was modeled as a plug flow which was justified by assuming high cross clearance mixing from the Taylor vortices, which are present along the circumferential direction. One of the more important findings was the presence of Goertler disturbances, along the axial direction (analogous to Tollmein-Schlichting waves) which were induced by the curvature of the boundary surface. Based on their behavior, it was deduced that turbulence in a seal resembles that of a flat plate at high Reynolds numbers. Another important conclusion was that tangential flow circulation did not change when the rotor was offset eccentrically (not considering whirl).

The introduction of the laser Doppler anemometer (LDA) system at the Turbomachinery Laboratory led to extensive experiments to further study the flow field in annular seals. This was initiated by a study by Johnson [2] who performed 3-D LDA measurements to obtain the mean velocity and the full Reynolds stress tensor for concentric and 50% eccentric annular seals (for a non-whirling rotor). Results indicated that for the case of an eccentric seal with no pre-swirl, the tangential velocity in the smallest clearance increased to 30% of the rotor speed at exit. At the largest clearance however, the tangential velocity did not show as large an increase and reached about 20%
of the rotor speed. The analysis of the flow field indicated that the flow near the inlet was turbulent and it was also near the entrance that the radial velocity decayed to zero in the first x/c=1. The development of anisotropic turbulence was also observed as the flow moved downstream.

Thames [3] followed this study by measuring the flow field inside a 50% eccentric annular seal (whirl ratio of 1.0) using the same 3-D LDA system. The existence of a region of high axial momentum, which at the inlet is on the suction side and migrates to the pressure side at the exit, was observed. This confirmed the results of Johnson’s experiments and was called the ‘saddle back’ effect. Other significant findings were a vena contracta induced recirculation zone on the pressure side of the rotor and no significant increase in tangential velocity with the presence of a whirling rotor motion.

Das [7] was one of the first researchers to investigate the effect of the eccentricity ratio on the flow field at the Turbomachinery Laboratory test-rig and this was done by comparing the results of Johnson and Thames to a seal operating at 10% eccentricity and a whirl ratio of one. The ‘saddle back’ effect was observed again but was not as pronounced. An analysis of the radial velocities also indicated the presence of a recirculation zone due to the vena contracta effect near the seal inlet as in case of the 50% eccentric seal. Winslow [8] completed this study by measuring the dynamic pressure and stator wall shear stress for the cases of 0, 10 and 50% eccentricity at a whirl ratio of one and observed that for small eccentricities there were small entrance and exit effects while larger eccentricities showed a peak in the mean pressure followed by decay and an increase again, near the exit. It was also seen that the shear stress does not contribute to
conditions at the wall of the stator thereby invalidating the assumption that the axial
velocity is a maximum at the maximum clearance location.

Robic [9] carried out a study on the effect of pre-swirl on the pressure field of a
whirling annular seal. One of the significant results of this study was that for negative or
no pre-swirl, the overall moment tends to axially pitch the rotor, closing off the exit flow
and opening the entrance on the right side of the minimum clearance (the pressure side).
Another recent study was that by Suryanarayanan [1] who investigated the impact of
whirl ratio for a 50% eccentric annular seal and also measured the wall pressure
distribution for the same. The ‘saddle back’ effect observed earlier was found to switch
sides from the pressure to the suction sides at a whirl ratio of 0.8 and 0.9 for positive
whirl ratios and between -0.7 and -1.0 for negative whirl ratios.

The most recent work at the Turbomachinery Laboratory which involves the
measurement of wall pressure distributions for eccentric annular seals was carried out by
Cusano [10]. The study was for a 25% eccentric whirling annular seal at different
Reynolds numbers for a range of whirl ratios from 0.1 to 1.0. The presence of whirl orbits
which were almost elliptic were observed, which seemed to distort further from a circular
path with an increase in the whirl ratio. For Re=24000 and Ta=3300, Taylor-Goertler
vortices were observed which showed an increase in size and azimuthal movement
around the seal with increasing whirl ratio. The case of Ta= 6600 did not show
pronounced vortices for whirl ratios between 0 and 0.4 and longitudinal vortices were
observed for a whirl ratios of 0.5 and higher. The switch in the pressure distribution as
observed by Robic and Suryanarayanan was confirmed and was seen at a whirl ratio of
0.7 for Ta=3300 and 0.4 for Ta = 6600.
Numerical Studies Involving Flow Fields

One of the first numerical studies on eccentric annular seals was by Tam et al. [11] who conducted a numerical and analytical study of fluid dynamic forces in seals and bearings. This study used a model of fluid forces based on average fluid circumferential velocity ratio which in turn was based on the assumption that the dynamic forces were rotating at the rotor’s precession speed. The simulation was performed using PHOENICS-84 and involved a grid of 12x16x16 in the tangential, radial and axial directions respectively. Turbulence modeling was accomplished using the Prandtl mixing length theory considering its relative simplicity as well as the fact that the mixing length was directly given by the clearance of the seal. The simulations were performed for a range of pressure drops, shaft speeds, eccentricity ratios and whirl ratios with bromotrifluoromethane as the working fluid. Results indicated large changes in local values of seal dynamic forces and large tangential separation zones. Recirculation zones were observed along the stator wall which depended on shaft and precession speeds. The effects of pre-swirl and fluid injection into the clearance were also studied.

Arghir and Frêne [12] performed simulations for a test case of a 50% eccentric synchronously whirling annular seal. This was accomplished using a specific rotordynamic method based on the perturbed form of the averaged full Navier-Stokes equations. The calculated quantities of stator pressures and shear stresses, distributed forces, velocity and turbulent kinetic energy distributions were found to be in good agreement with the experimental data from Morrison et al. [13, 14] and Morrison and Winslow [15]. The average pressure distribution also matched experimental data except
at the entrance and exit regions. It was also concluded that perturbation technique could be used for eccentricities as large as 50%.

**Studies Involving Rotordynamic Coefficients**

One of the very first studies investigating whirl was by Newkirk and Taylor [16] who referred to the action of whirl instabilities as ‘oil-whip’ and observed that these vibrations occurred at shafts running above their critical speeds. The relation of the whirling motion to the lubricant was also observed and recommendations were made to prevent the occurrence of oil whip in running machinery. Hori [17] performed a theoretical and experimental study of the same and compared the results to those of Newkirk and Taylor. In this study, oil whip was treated more like a problem of dynamical stability of the rotor and Hori showed that the problem of instability could be avoided by assuming zero pressure instead of negative pressure in the oil film. Stability with in terms of vibrations was also discussed and a significant conclusion was that stabilization occurs when either the eccentricity is increased or when the oil force is increased.

Fritz [18] extended the work done by Hori by analyzing the hydrodynamic mass and damping coefficients of a liquid in a thin annulus surrounding a vertically oriented rotor assembly. Theoretically it was seen that dynamical stability was reached if the rotor speed is less than twice the critical speed. Another important study that was conducted around the same time was by Black [19] who developed a simplified, linear theory, considering squeeze forces to compute the stiffness and damping coefficients for plain annular seals. From the results obtained, it was concluded that damping forces associated with squeeze actions are large and for seals with a high length to clearance ratio, this
could cause negative stiffness. It was also seen that whip instability at high speeds may result from the rotation of fluid within the seal.

Allaire et al. [20] conducted a study on the dynamics of short axial seals with a high axial Reynolds number. This study implemented the bulk flow theory by Hirs and gave an analytical solution of semi-empirical bulk flow equations for an eccentric seal. It was observed that for a small axial Reynolds number, the pressure drop through the seal was due to friction but as the leakage rate increased, the ‘Bernoulli’ effect became stronger leading to high pressure gradients. An important deduction was that stiffness and damping were strong functions of the eccentricity ratio. Another significant study of rotordynamic coefficients was by Nordmann et al [21] where a finite difference method was used to compute the coefficients. This involved carrying out a perturbation analysis of the Navier-Stokes equations coupled with the $k-\varepsilon$ turbulence model to obtain a set of differential equations which could be solved by a simple finite difference method. The results obtained were better than those obtained using the simple bulk flow models but the computation time involved was also longer.

Childs [22] derived expressions which defined the dynamic coefficients for turbulent annular seals and was able to analytically solve for the rotordynamic coefficients of high pressure annular seals with correction coefficients for inlet swirl conditions. This approach assumes a fully turbulent flow in the seal and applies perturbation analysis to equations obtained using Hirs’ theory. The results were found to match those of Black and Jenssen [23] for zero swirl but results for flows including swirl were not physically supportable hence leading to the conclusion that Hirs turbulence model was not suitable for short seals with a significant swirling flow.
This was followed by a study by Chen and Jackson [24] who performed an experimental study to determine the effect of eccentricity and misalignment on the rotordynamic coefficients for high pressure annular seals. Relationships were developed for the reaction force and leakage versus the eccentricity, misalignment and rotation rate. A concentric tapered seal was used as a model for an eccentric seal and it was seen that the effect of eccentricity or misalignment was not very pronounced when the flow was fully turbulent. It was observed that the relation between the direct damping coefficient and cross-coupled stiffness was preserved through all eccentricity ratios and also that the dynamic coefficients changed significantly for high eccentricity ratios when the perturbation direction coincided with the minimum clearance.

Kanemori and Iwatsubo [25] conducted an experimental study of the dynamic reaction force in an axially long annular clearance between a concentric rotor and stator. Results showed that the reaction force depended on the whirl velocity and reached a minimum for a whirl ratio of 0.5. It was also seen that the tangential component of the fluid force acts as a destabilizing component when the rotor whirls forward and the whirl ratio is between 0 and 0.5. For small axial Reynolds numbers, the stiffness coefficient $K_{xx}$ also acted as a destabilizing force while $K_{xy}$ and the damping coefficient $C_{xx}$ behave similarly and increase with Reynolds number. It was also seen that the effect of the Reynolds number on the added masses $M_{xx}$ and $M_{xy}$ is negligible.

One of the more recent studies in the determination and analysis of rotordynamic coefficients for seals is that by Xi and Rhode [26] who studied the rotordynamics of labyrinth seals which undergo an axial shifting of the rotor. This study used a CFD perturbation model, the use of which was initiated by Dietzen and Nordmann. This
perturbation model is similar to the bulk flow model but differs in the treatment of the governing equations and the basic idea was to reduce the computation time that would be required by a full 3-D CFD model, at the same time obtaining a physically feasible solution. The study realized that the effect of swirl velocity at the inlet was not negligible as was assumed until then. The results from this research corresponding to the relations between the radial and tangential forces and the rotodynamic coefficients were used for the computation of the rotodynamic coefficients in the present study.
OBJECTIVES AND METHODOLOGY

The present work intends to determine the effect of the orbit path of the rotor on the flow field and the rotordynamic coefficients of a seal by means of a numerical study. The study also aims to investigate the effect of clearance on these parameters by comparing simulations for two different clearances of 1.27mm and 0.127mm with the same orbit paths. The two orbit paths that will be simulated include a circular path and a linear (back and forth) path, which were selected based upon previous experimental results. The models used here are full 3-D models (based on seal geometry as used in prior experiments) and the orbit simulation involves the use of the deforming mesh capability of FLUENT in order to reproduce the fluid flow as closely as possible. Numerical simulations of this nature will not only aid the study of turbulent annular seals but also reduce testing and development costs. This technique could be used to further study elliptic orbit paths as obtained during Cusano’s experiments; the simulations could thus serve as a more thorough database and be useful in determining and comparing rotordynamic coefficients for a range of operating conditions.

The current analysis utilizes the experimental work carried out by Johnson [2], Shresta [27], Thames [3] and Cusano [10]. The initial goal was to simulate the flow through the seal using the conditions from Johnson’s and Thames’s work so as to validate the CFD analysis. This involved the construction of a 3D model in Gambit (Ver. 2.4.6) while the flow was simulated and solved in FLUENT (Ver.6.3.26). The model is that of a smooth annular seal with the following dimensions: \( L = 37.3 \text{mm} \), \( R = 82.05 \text{mm} \) and clearance \( c = 1.27 \text{mm} \) (50mil) and the whirl orbit radius was one half of \( c \). The rotor was maintained at an eccentricity of 50% where the eccentricity ratio is defined as \( e/c \) (e –
off-centeredness of the rotor). Also, a swirl ring was added as a part of the model so as to provide an inlet boundary condition for the annular clearance. Fig. A-1 shows a sectional view of the seal test rig used at the Turbomachinery Laboratory while Fig. A-2 shows the dimensions of the rotor used.

The entire assembly was meshed using a hexahedral scheme, comprised of approximately 1,000,000 nodes. The node distribution possessed a very fine mesh for the clearance regions and maintains a reasonable mesh quality for the rest of the seal. It is essential to find a balance between the number of elements required and economy of the solution. A representative mesh is shown in Fig. A-3. In order to capture the turbulence characteristics of the flow and the velocity gradients close to the wall accurately, very fine meshes are employed in the boundary layers along the rotor and stator walls. This ensures a higher mesh density where necessary without greatly increasing the number of nodes used.

Once the creation of the model was complete, the next step was to simulate the whirling motion of the rotor. This was done using the dynamic mesh capability of FLUENT which accommodates changes in the shape of the domain, at each time step. It is also essential to ensure that the spring remeshing scheme for hexahedral meshing is enabled. This is done using the ‘rpsetvar’ command. The orbit paths were then simulated using User Defined Functions (UDFs) in FLUENT which are basically programs written in C++ that specify the motion of the rotor.

The working fluid used is water with the properties as in the FLUENT database (density 998.2 kg/m³ and dynamic viscosity 0.001003 kg/m-s). A shaft speed of 3600
RPM (corresponding to a Taylor number of 6600) was used and the whirl ratios will vary from 0 to 1 at intervals of 0.2. The flow was specified by a mass flow rate of 4.87 Kg/s (corresponding to an axial velocity of 7.4m/s, Reynolds number of 24,000) for the 1.27mm clearance case. The turbulence model used was the standard k-ԑ model with an enhanced wall treatment and pressure gradient effect.

Since the k-ԑ model is primarily valid for turbulent core flows (i.e. flows away from the wall), it is necessary to take the required steps to make suitable accommodations for the near-wall flows. This is done by incorporating the enhanced-wall treatment where the moving wall – i.e. the rotor creates turbulent flow. However in order to use the enhanced wall treatment it is necessary to ensure a very fine grid (y+ ≈ 1) close to the wall, so that it can capture the innermost viscous sub-layer. It is necessary to have y+ values below 5 and this has will be ensured throughout the model except in the entrance region next to the walls where the viscous sub layer is extremely thin. A plot of the typical y+ values for the rotor wall is shown in Fig. A-4.

The pressure gradient effect was also used along with the enhanced wall treatment. The enhanced wall treatment functions are capable of incorporating influences of both pressure and thermal gradients and this is controlled by the options in the k-ԑ model. The pressure gradient is taken into account to accommodate the effect of the larger clearance at the plenum where the flow exits.

A preliminary case was run for a circular orbit of an eccentric seal (eccentricity 0.5) for whirl ratios ranging from 0 to 1. Once the results from the run for whirl ratio 0 were obtained, they were directly compared to the data from Johnson’s and Shresta’s
work, which include the LDA measurements for an eccentric smooth annular seal at 3600 RPM (no whirl). The results for whirl ratio 1 were compared to Thames’s work. Once the models were validated, the cases for the other orbits and whirl ratios were ready to be investigated.

The effect of the clearance of the seal on the rotordynamic coefficients is also studied. This is accomplished by using the same seal model but with a reduced clearance of 0.127mm (5mil), which is done by increasing the rotor radius and keeping the other dimensions the same. The mass flow rate used for this case is 0.487 kg/s (i.e. 1/10 the mass flow rate used earlier). The other flow parameters and the solver are the same, which results in a Reynolds number of 2400 and Taylor number of 208.71.

Thus a complete database of flow variables can be obtained for different orbits, whirl ratios and clearances which would enable comprehensive analysis and conclusions. The present work can also be built upon in order to simulate a range of orbits and clearances which could complement experimental study.
CODE DESCRIPTION

Physical Models of Fluid Mechanics

The basic goal of conducting a numerical study is to obtain an accurate depiction of the performance without investing the time or the cost involved in an experimental procedure. This involves obtaining the flow field which could then be analyzed in order to determine the effect of the geometry or flow parameters. The simulations for this study were performed using the CFD solver FLUENT 6.3.26 which essentially solves the Reynolds Averaged Navier-Stokes (RANS) equations along with a suitable turbulence model. Before going into the details of the solver, it is essential to have an idea of the equations being solved as well as the theory behind the equations being used. This section will discuss the governing equations of fluid dynamics while the numerical methods used to solve them will be discussed in the next section.

For any flow problem, the equations of conservation of mass and momentum are to be solved. The continuity equation or the equation of conservation of mass is given by

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m$$  \hspace{1cm} (3)

here, $\rho$ is the fluid density and $\vec{v}$ represents the velocity vector. The source term $S_m$ accounts for the mass added to the continuous phase from any secondary phase such as vaporization of a liquid or from a user defined source.

The equation of conservation of momentum, better known as the Navier-Stokes equation is given by
where $p$ is the static pressure, $\bar{t}$ is the stress tensor, and $\rho \vec{g}$ and $\vec{F}$ are the gravitational and external body forces respectively. The stress tensor is further defined by

$$
\bar{t} = \mu \left[ \nabla \vec{v} + \nabla \vec{v}^T - \frac{2}{3} \nabla \cdot \vec{v} \bar{I} \right]
$$

where $\mu$ is the dynamic viscosity, $\nabla \vec{v}$ is the gradient of the velocity vector, $\nabla \vec{v}^T$ is its transpose, and $\bar{I}$ is the identity tensor. This includes the use of Stokes hypothesis in order to represent the bulk viscosity as $2/3$’s the dynamic viscosity. For simulations involving a deforming or moving mesh, the dynamic viscosity is held constant.

### Modeling Turbulence

In order to handle turbulent flows, FLUENT 6.3.26 has the option of using both Large Eddy Simulation (LES) and RANS models which introduce additional terms in the governing equations that are then modeled to obtain closure. Although the LES model is a relatively new solution technique and could provide better results at times, the cost and the time that go into employing the technique are much higher compared to the conventional RANS modeling. The current study used the RANS approach which basically represents the transport equations for the mean flow quantities alone, and this can be used with any of the existing turbulence models such as the Spalart Allmaras model, the $k - \varepsilon$ model or the $k - \omega$ model.

Therefore, the equations being solved are

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

(6)
\[
\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right] + \frac{\partial}{\partial x_j}(-\rho \overline{u'_i u'_j})
\]

(7)

While using the Reynolds averaged approach, it is essential to make sure that the Reynolds stress term \(\overline{\rho u'_i u'_j}\), is accurately modeled. One of the most common ways to do this is using the Boussinesq hypothesis which is used with the Spalart Allmaras, \(k - \varepsilon\) and \(k - \omega\) models. The relation used here is

\[
\rho \overline{u'_i u'_j} = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3} \left(\rho k + \mu_t \frac{\partial u_k}{\partial x_k}\right) \delta_{ij}
\]

(8)

The Boussinesq hypothesis performs well for most practical applications but for flows which are more complicated, an alternative Reynolds Stress Model approach could be used where transport equations are solved for each of the terms of the Reynolds stress tensor.

FLUENT has the option of using the Spalart-Allmaras, \(k - \varepsilon\) or \(k - \omega\) models each of which can be used depending on the application. The model used here is the standard \(k - \varepsilon\) model along with an enhanced wall function. The \(k - \varepsilon\) model was chosen owing to the fact that it has been proven to be effective for a range of turbulent flows and has been used successfully for many industrial applications. This is basically a semi-empirical model which is based upon model transport equations for the turbulence kinetic energy and dissipation rate. The main assumption here is that the flow is fully turbulent. The turbulence kinetic energy, \(k\), is modeled as

\[
\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho ku_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_p - \rho \varepsilon - Y_M + S_k
\]

(9)
While the turbulence dissipation $\varepsilon$ is modeled as

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_3 \varepsilon G_B) - C_{2\varepsilon} \rho \varepsilon^2 + S_\varepsilon$$

(10)

Here, $G_k$ represents the generation of turbulence kinetic energy due to mean velocity gradients, $G_B$ is the generation of turbulence kinetic energy due to buoyancy and $Y_M$ is the contribution of fluctuating dilation in compressible turbulence to the overall dissipation rate. $C_{1\varepsilon}, C_{2\varepsilon}$ and $C_{3\varepsilon}$ are constants while $\sigma_k$ and $\sigma_\varepsilon$ are the Prandtl numbers for $k$ and $\varepsilon$ and $S_k$ and $S_\varepsilon$ are user defined source terms. The turbulent viscosity is further computed as

$$\mu_t = \rho C_{\mu} \frac{k^2}{\varepsilon}$$

(11)

The default values for the constants were retained which are

$$C_{1\varepsilon} = 1.44, C_{2\varepsilon} = 1.92, C_{\mu} = 0.09, \sigma_k = 1.0, \sigma_\varepsilon = 1.3$$

The $k - \varepsilon$ model, including its variations such as the standard, renormalization group (RNG) and the realizable models though relatively effective, are largely valid for turbulent core flows, i.e. flows away from walls. It has been observed that there are significant effects that must be taken into account for flows close to walls. This includes the no-slip condition for velocity being satisfied at the wall, low tangential velocity fluctuations close to the wall and rapid growth of turbulence in the outer part of the near wall region (owing to large gradients of mean velocity).

For the purpose of analysis and based on previous experimental results, it has been seen that the near-wall region can be divided into three layers. The innermost layer
(the viscous sublayer) has dominant molecular viscosity effects and the flow here is almost laminar. For the outer layer, called the fully-turbulent layer, turbulence is the defining characteristic. Finally, there is an interim layer (the buffer layer) between these two layers where both the effects of turbulence and molecular viscosity are seen. There are basically two approaches to modeling the near-wall region – one, where the viscosity affected regions (the viscous sublayer and the buffer layer) are not entirely resolved but treated using semi-empirical relations and another approach (also known as near-wall modeling) is where the turbulence models are modified and the inner layers are resolved fully.

The simulations in the current work have been performed using the enhanced wall treatment which is a near-wall modeling method that combines a two-layer model with enhanced wall functions. It was essential to use enhanced wall functions as the standard wall functions, as mentioned earlier do not resolve the flow in the viscous layers and hence would not work for a situation where the bounding wall moves relatively rapidly or the main flow is subject to a high pressure gradient. While using the enhanced wall treatment, it should be ensured that the mesh by itself is fine enough to resolve the turbulent phenomenon down to the viscous sublayer region. According to the FLUENT manual, this means that $y^+$ values less than five are acceptable as long as the first layer of the mesh is in the sublayer zone. Fig. A- 4 shows a plot of the $y^+$ values for the entire wall region and it is observed that the values are less than 5 almost throughout the seal except at the entrance region where the turbulence kinetic energy is high and the viscous sublayer is very thin.
NUMERICAL MODEL

The previous section discussed the equations that illustrate the physics that went into the problem. This section goes into the details as to how these equations are implemented and the numerical methods that are used to solve them.

Description of Solution Method

FLUENT allows the use of both the pressure-based solver and the density-based solver in order to solve the fundamental equations. For both the methods, the velocity field is obtained from the momentum equations but they differ in the aspect that for the density-based approach, the pressure field is obtained from the equation of state and for the pressure-based approach, it is obtained by solving a pressure correction equation (which is obtained from the continuity and momentum equations). The basic idea is to use a control-volume based technique which involves the following steps – division of the domain into control volumes, integration of the governing equations on the individual control volumes to obtain algebraic equations and the linearization of the discretized equations and solving the system to obtain values for the variables.

The solver used here is the pressure-based solver using the segregated algorithm. This basically means that the governing equations are solved sequentially and the solution loop is iterated until a converged solution is obtained. The steps involved are given below along with a flowchart (Fig. A-5) to graphically show the sequence

1. Fluid properties are updated based on the current solution.
2. Momentum equations are solved, one after another based on the recently updated values and this is followed by solving the pressure correction equation using the solution from the momentum equation.

3. Face mass fluxes, velocity fields and pressure are corrected using the correction from the previous step.

4. Any additional scalars are solved for and source terms (due to interactions between different phases) are updated.

5. Convergence is checked and the loop is repeated until convergence is obtained.

The convergence criteria are described by the user. It should be kept in mind that the governing equations are fully coupled and in order to solve for all the flow properties, many iterations may need to be performed.

**Linearization**

As mentioned earlier, the non-linear governing equations are linearized with respect to different variables in order to obtain algebraic equations. This leads to algebraic equations for every transport equation, for every cell of the domain and the unknowns are the dependent variables of the transport equations. When an implicit formulation is used, each equation has not only the unknown from that cell but from the neighboring cells as well. This, therefore, results in a system of equations which are to be solved simultaneously in order to obtain the unknown quantities. In the segregated approach, this means that each variable equation is linearized implicitly, with respect to itself and the variable field is solved over all the cells.
Since the equations are implicit in nature, the system cannot be solved by direct inversion. They can however be solved easily using any of the existing algorithms and FLUENT uses the Gauss-Seidel solver in conjunction with an algebraic multi-grid method. The Gauss-Seidel method is used as it is one of the most economical in terms of memory requirements. Also, it is proven to be faster in terms of computation compared to direct computation because of the number of zeros present in the coefficient matrix.

**Discretization**

FLUENT uses a control volume based discretization method and allows the user to choose the scheme for the convective terms of each governing equation (the viscous terms are automatically modeled using second order discretization). The first step of this process is to discretize the domain into a collection of cells which is done using grid generation. The next step is to apply the governing equations, in the integral form to every cell in the domain. Once these steps are complete, the discretized equation is linearized and subsequently the linear system is solved. A more detailed account of the entire procedure can be obtained from the FLUENT user’s guide which also includes an example of a scalar transport equation written in the integral form and discretized using finite volumes.

By default, FLUENT stores discrete values of the flow variable only at the cell centers. However, the face values of the flow variables are also required for the convection terms and these are interpolated from the cell center values by an upwind scheme. Upwinding basically means that the face value is derived from quantities in the
cell upstream, or “upwind” direction relative to the direction of the normal velocity $v_n$.

The current research uses the second order accurate upwinding scheme the details of which are discussed in the following section.

**Second-Order Upwinding Scheme**

The second order upwinding scheme uses a multi-dimensional reconstruction approach. Here, higher order accuracy is achieved at cell faces through a Taylor series expansion of the cell-centered solution about the cell centroid. The face value of an arbitrary scalar quantity $\phi$ is thus computed as

$$
\phi_{f,sou} = \phi + \nabla \phi \cdot \vec{r}
$$

where, $\phi$ and $\nabla \phi$ are the cell-centered value and its gradient in the upstream cell and $\vec{r}$ is the displacement vector from the upstream cell centroid to the face centroid. Further, the gradient is computed (using the Green-Gauss theorem) as

$$
(\nabla \phi)_{c0} = \frac{1}{v} \sum_{f} \phi_f \vec{A}_f
$$

here, $\phi_f$ is the value of $\phi$ at the face centroid and the summation is over all the faces enclosing the cell, while $\vec{A}_f$ is the surface area vector for the face. $\phi_f$ can be computed using either a Green-Gauss Cell-Based or Green-Gauss Node-Based approach. This study used a Cell-Based approach, where $\phi_f$ is computed by taking the arithmetic average of the values at the neighboring cell centers.

$$
\phi_f = \frac{\phi_{c0} + \phi_{c1}}{2}
$$
Time Discretization

For transient simulations, such as in this case, the governing equations are to be discretized in both space and time. Temporal discretization involves the integration of every term in the differential equations over a time step $\Delta t$. For a generic variable $\phi$

$$\frac{\partial \phi}{\partial t} = F(\phi)$$  \hspace{1cm} (15)

A second order discretization was again used here, which is basically a finite difference approximation given by:

$$\frac{3\phi^{n+1}-4\phi^n+\phi^{n-1}}{2\Delta t} = F(\phi)$$  \hspace{1cm} (16)

where $\phi$ is a scalar quantity, $n + 1$ is the value at the next time level $t + \Delta t$, $n$ is the value at the current time level and $n - 1$ is the value at the previous time level $t - \Delta t$.

Once the time derivative has been discretized, the time levels that are chosen to evaluate $F(\phi)$ determine the type of formulation, implicit or explicit. The method that has been used in the current work is an implicit, first order scheme, which basically means that $F(\phi)$ is computed at the future time level. This was chosen as the fully implicit scheme is unconditionally stable with respect to time step size. Expressing $F(\phi)$ at the future time level and solving for $\phi$ at future time gives

$$\phi^{n+1} = \frac{4}{3}\phi^n - \frac{1}{3}\phi^{n-1} + \frac{2}{3}\Delta tF(\phi^{n+1})$$  \hspace{1cm} (17)

Many sub-iterations, i.e. iterations per time step are performed before the solution is found to advance in time. This basically means that the entire process of solving the
transport equation is done many times and many intermediate values of $\phi$ are computed before the simulation moves forward to the next time step. It was seen that around 150 iterations per time step were adequate for this simulation.

**Pressure-Velocity Correction**

Pressure-Velocity coupling basically provides an additional condition for pressure by reformatting the continuity equation. The pressure-based solver allows the problem to be solved using either a segregated or a coupled algorithm and provides the option of choosing from the SIMPLE, SIMPLEC, PISO and FSM algorithms. All of these approaches except for the coupled algorithm are based on the predictor-corrector approach. This research work uses the SIMPLEC algorithm with the under-relaxation slightly modified from the default values to aid convergence. The following section will discuss the SIMPLEC algorithm in detail.

**SIMPLEC Algorithm**

The SIMPLEC algorithm is basically a modified version of the SIMPLE algorithm which stands for Semi-Implicit Method for Pressure-Linked Equation. The semi-implicit nature refers to the fact that the velocity correction is explicit while the pressure correction is implicit. The continuity equation is first integrated over the control volume and in the equation so obtained the face values of the velocity are related to the stored values (at the cell centers) to obtain the following equation:

$$ j_f = \rho_f \frac{a_{p,c_0} v_{n,c_0} + a_{p,c_1} v_{n,c_1}}{a_{p,c_0} + a_{p,c_1}} + d_f ( (p_{c_0} + (\nabla p)_{c_0} \cdot \vec{r}_0) - (p_{c_1} + (\nabla p)_{c_1} \cdot \vec{r}_1) ) $$

$$ = \tilde{j}_f + d_f (p_{c_0} - p_{c_1}) \quad (18) $$
here \( J_f \) is the mass flux through face \( f \), \( p_{c0}, p_{c1} \) and \( v_{n,c0}, v_{n,c1} \) are the pressures and normal velocities within the two cells on either side of the face and \( \int f \) contains the influence on the velocities in these cells. The term \( d_f \) is a function of the average of the momentum equation \( a_p \) coefficients for the cells on either side of the face \( f \).

The SIMPLE algorithm starts with an initial guess for the pressure field \( p^* \) and the resulting flux is computed from here using

\[
J_f^* = \hat{J}_f + d_f(p_{c0}^* - p_{c1}^*)
\]

(19)

This does not satisfy the continuity equation and a correction of \( J_f^* \) is added in order to do so i.e.

\[
J_f = J_f^* + J_f'
\]

(20)

For both the SIMPLE and SIMPLEC algorithms, \( J_f' \) is written as

\[
J_f' = d_f(p_{c0}' - p_{c1}')
\]

(21)

where \( p' \) is the cell pressure correction. Further, equations (20) and (21) are substituted into the discrete continuity equation to obtain a discrete equation for the pressure correction \( p' \) of the cell

\[
a_p p' = \sum_{nb} a_{nb} p_{nb}' + b
\]

(22)

where \( b \) is the net flow rate into the cell and is given as

\[
b = \sum_{faces}^N J_f^* A_f
\]

(23)

The pressure correction equation (22) can be solved using the algebraic multi-grid (AMG) method and once this is done, the cell pressure and face flux are corrected using

\[
p = p^* + a_p p'
\]

(24)

\[
J_f = J_f^* + d_f(p_{c0}' - p_{c1}')
\]

(25)
here $\alpha_p$ is the under-relaxation factor for pressure. The corrected face flux thus satisfies the discrete continuity equation during each iteration. The only additional feature of the SIMPLEC algorithm is that the coefficient $d_f$ is defined as a function of

$$\left(\alpha_p - \sum_{nb} a_{nb}\right),$$

which accelerates convergence.
CONSTRUCTION AND IMPLEMENTATION OF DYNAMIC MESHES

Using Dynamic Meshes in FLUENT

One of the more important parts of this numerical study was the simulation of the rotor’s motion within the stator. The experimental apparatus used by Johnson was shown in Fig. A-1. Here, the rotor could be mounted eccentrically on the shaft, producing a whirling motion. It was therefore necessary to use a dynamic mesh to simulate this motion. The CFD simulation being performed begins at the inlet to the swirl ring (Fig. A-6) and ends at a distance of 56.7mm from the exit of the seal (this is done in order to obtain more realistic boundary conditions). At the interface between the swirl ring and the rotor, there is a step whose height varies with the whirling motion.

The changing shape of the fluid domain was thus accommodated using the dynamic meshing capability in FLUENT while the orbit path, i.e. the motion of the rotor was specified using User Defined Functions (UDFs). Dynamic meshing basically enables FLUENT to update the volume mesh for the domain at each time step, based on the new positions of the boundaries. It is, however, necessary to initially have a good quality volume mesh to ensure that all the flow properties are captured satisfactorily. Also, FLUENT requires that the starting mesh contains the moving and non-moving domains grouped by their face or cell zones.

It is seen that as the model contains the swirl ring in addition to the basic seal geometry, there is an interface between the swirl ring and the seal which needs to be addressed. These interfaces are first connected using the ‘Grid Interfaces’ option of FLUENT. As shown in Fig. A-7, the interface zone from the seal and that from the swirl
ring are of different lengths, which results in a non-conformal interface. Thus, while the rotor is in motion, the portion of the interfaces that do not overlap is regarded as a ‘wall’ entity by FLUENT. It is important to note that the interface zone of the seal is actually made up of three separate interface zones (one for the clearance and one for each of the boundary layers along the rotor and stator) while that of the swirl ring is made of a single zone. This requires a three to one interface relationship (which is only available for FLUENT version 6.3 and above) and the use of a multi-layer mesh scheme.

When a dynamic mesh is used, FLUENT uses one of several available mesh motion methods in order to update the volume mesh that is subject to deformation. These methods are classified under smoothing methods, dynamic layering and local remeshing methods. The method that has been used for the simulations in this study is the spring-based smoothing method. Here, the edges between any two nodes are taken as a network of springs and the spacing between the nodes in the initial mesh is taken as the equilibrium state. The force acting at each mesh node is then calculated (taking into account a user-defined spring constant) and the condition that at equilibrium the net force acting should be zero are used to obtain an iterative equation. The force on the mesh node is thus given by

$$
\vec{F}_i = \sum_{j}^{n_i} k_{ij} \left( \Delta \vec{x}_j - \Delta \vec{x}_i \right)
$$

(26)

where $\Delta \vec{x}$ represents the displacement of the node, $n_i$ the number of neighboring nodes and $k_{ij}$ the spring constant between node i and its neighbor j. Using the condition that at equilibrium the net force at a node is zero, the iterative relation obtained is
\[
\Delta \mathbf{x}_i^{m+1} = \frac{\sum_i k_{ij} \Delta \mathbf{x}_j^m}{\sum_i k_{ij}}
\]  \hspace{1cm} (27)

As the displacements are known at the boundaries (from the new boundary positions) a Jacobi sweep is performed to solve (27) on all interior nodes.

At convergence

\[
x_i^{-n+1} = x_i^{-n} + \Delta x_i^{-m, converged}
\]  \hspace{1cm} (28)

where \(n+1\) and \(n\) are the positions at the next and the current time step respectively.

The FLUENT manual recommends the use of the spring-based smoothing method for non-tetrahedral meshes under the following cases: when the boundary of the zone moves predominantly in one direction or when the motion is predominantly normal to the boundary zone. Since these conditions are compliant with the geometry and motion of the simulations to be performed, this was the remeshing method used. The spring based smoothing method is enabled by default only for the tetrahedral meshing scheme. In order to use this in the current simulation where a hexahedral mesh scheme is used, the smoothing method can be enabled for all cell types by using the following command:

\[(rpsetvar 'dynamesh/spring/all-element-type? #t)\]

Alternatively, the following chain of commands can be used:

\[\text{define} \rightarrow \text{models} \rightarrow \text{dynamic-mesh-controls} \rightarrow \]

\[\text{smoothing-parameter} \rightarrow \text{spring-on-all-shapes}\]
Describing Rotor Motion Using UDFs

In order to specify the motion of the rotor for each of the orbit paths to be simulated, the use of FLUENT’s ability to accommodate custom, problem specific changes to the existing code is made. Each different case run required the use of two UDFs, one to specify the orbit path and another to specify the surface velocity for the whirling rotor. A UDF in FLUENT can be used for a variety of applications from the problem definition to the post processing as the need may arise. It is basically a code written using a C programming language which can be dynamically linked to the solver. For the present application, the programs have been written in C++ using a Visual Studio editor. UDFs typically start with the `udf.h` file inclusion directive and are defined using the DEFINE macros that are provided by FLUENT. This program, known as the source code, is saved with a .c extension and can then be interpreted or compiled in FLUENT.

The very first step in using a UDF is to define the problem mathematically, i.e., in the present application, for the motion of the rotor in a certain orbit path, it is necessary to have a set of equations that describe the motion. The next step is to incorporate these equations into a C code. As mentioned in the previous paragraph, FLUENT provides a number of predefined functions or macros which can be used depending on the application. The general format of the DEFINE macro is as follows:

```
DEFINE_macroname (udf_name, passed-in variables)
```
The definitions for the macros used are all contained in the udf.h header file which must be included at the beginning of every UDF source code. This is done using the following compiler directive:

```
#include “udf.h”
```

While programming a UDF for FLUENT, it is useful to know that in addition to the standard data types from C, FLUENT has its own data types to represent the various computational units. Examples of these data types are - Node, face_t, cell_t and Thread. It is also important to know that in a UDF, zones (as in fluid and boundary zones defined in FLUENT) are referred to as threads.

For the present application, in addition to the basic DEFINE macros, specifying the rotor motion required the use of vector and dimension macros. These macros are basically used to access and manipulate vector quantities and can be used for both two and three dimensional applications. The first DEFINE macro used in this application is the DEFINE_CG_MOTION macro which specifies the motion of the dynamic zones by using the angular and linear velocities at each time step. These velocities are then used to update the node positions on the dynamic mesh zones based on the solid-body motion.

The general syntax for the function is as follows:

```
DEFINE_CG_MOTION (name, dt, vel, omega, time, dtime)
```

The arguments used are

- `name`, which is the UDF name and is specified by the user
- **dt**, which is a pointer to the structure that stores the dynamic mesh attributes. This is passed from FLUENT directly to the UDF.

- **vel**, a real variable which specifies the linear velocity. Its value is passed directly from FLUENT to UDF.

- **omega**, a real variable which specifies the angular velocity. This is passed from FLUENT directly to the UDF.

- **time**, a real variable which specifies the current time. Its value is passed directly from FLUENT to UDF.

- **dtime**, a real variable which specifies the time step. This is also passed from FLUENT directly to the UDF.

In addition to the DEFINE macro, a few other macros were used to handle the vector manipulations in the program. These include the NV macros which operate on vectors, and can be used to perform operations right from the definition of the vector to compute dot and cross products. The program also uses a looping macro:

**begin…end_f_loop** which is used to perform operations over all the faces of a given thread. Another macro used is F_NODE which is used to call vector information such as the coordinate. For this application, this macro is used when the coordinate data for the current node is required within the **begin…end_f_loop** macro. The complete UDF for the circular orbit with a whirl ratio of 1, is as follows:

```c
#include "udf.h"
#include "stdio.h"

DEFINE_CG_MOTION(ROtor_Motion,dt,vel,omega,time,dtime)
{
```
Thread *tf = DT_THREAD (dt);

face_t f;
Node *v;
real NV_VEC (axis), NV_VEC (dx);
real NV_VEC (origin), NV_VEC (rvec), NV_VEC (center), NV_VEC (trans);
real rotation_radii, RPM;
real time1, time2, theta1, theta2;
int n;
/* set deforming flag on adjacent cell zone */
SET_DEFORMING_THREAD_FLAG (THREAD_T0 (tf));
rotation_radii = 0.025*0.0254; //Whirling radius
RPM = 376.99; // rad/sec- 3600rpm: whirling speed
NV_D (axis, =, 0.0, 0.0, 1.0);
NV_D (origin, =, 0.0, 0.0, 0.0);
NV_D (center, =, rotation_radii, 0.0, 0.0);
NV_D (trans, =, 0.0, 0.0, 0.0);
time1 = time;
time2 = time+dtime;
theta1= time1 * RPM;   //t
theta2= time2 * RPM;   //t+
\[\Delta t\]
theta1 -= 6.28318531;
theta2 -= 6.28318531;
trans[0] = rotation_radii*(cos(theta2)-cos(theta1));  //unit in m
trans[1] = rotation_radii*(sin(theta2)-sin(theta1));  //unit in m
Message ("\n whirl angle=%f     time=%f
",theta1*180/3.141592,time);
if (fabs(theta1) > 6.28318531)
    theta1 -= 6.28318531;
if (fabs(theta2) > 6.28318531)
    theta2 -= 6.28318531;
Message ("%f   %f    %f  
", theta1*180/3.141592, trans[0], trans[1]);
begin_f_loop (f, tf)
{
    f_node_loop (f, tf, n)
    {
        v = F_NODE (f, tf, n);
        if (NODE_POS_NEED_UPDATE (v))
        {
            NODE_POS_UPDATED(v);
            NV_V (NODE_COORD (v), +=, trans);
        }
    }
} end_f_loop (f, tf);

The first two lines of the program deal with linking the header files required by
the UDF- udf.h as mentioned earlier and stdio.h which is the “standard input/output
header” and provides functions for input and output operations. The next part involves
the usage of the DEFINE macro which has been given the name Rotor_Motion. Within
the macro, the first line returns a pointer to the face thread for which the dynamic mesh
attributes are specified and this is stored in the pointer *tf. The next few lines deal
with the declaration of the user defined variables used in the program – the vectors
defined are axis, dx, origin, rvec, center and trans while the scalar
variables defined are theta1, theta2, rotation_radii, RPM, time1 and
time2. The deforming flag is then set on the adjacent cell zone which is followed by the
initialization of the variables.

The variable rotation_radii indicates the whirling radius of the seal and
this is given by half the value of the eccentricity of the rotor, about the center of the seal
(the value is multiplied by 0.0254 to account for the conversion from inch to m). The next
variable initialized is RPM which indicates the whirling speed (in revolutions per minute)
– this is the variable whose value is changed for different whirl ratios. Similarly the
variables axis, origin, center and trans are all initialized in the vector form
while the values for the variables time1 and time2 are passed to the function from
FLUENT itself. Within FLUENT, the time step is computed as follows – for the rotor
speed of 3600 rpm, which is 60 Hz, which is 1/60 of a second for 1 revolution; therefore
for a one degree increment, the time step would be \( \frac{1}{360\times60} = 4.6296 \times 10^{-5} \). The values
for theta1 and theta2 are then calculated from the values of the time variables by
multiplying it with the whirling speed.

After the variables are initialized, an if condition is used to ensure that the
values of the angles theta1 and theta2 do not exceed 360°. The next part is to
specify the coordinates of the center of the rotor wall according to the orbit path and this
is computed using the values of theta1, theta2 and the whirling radius. Here, the
first two elements of the trans vector indicate the x and y coordinates of the center respectively and each element is calculated individually. A loop is then set up using the begin…end_f_loop macro in order to loop over the entire thread and within this, the f_node_loop is used to loop over all the nodes of each face of the thread. An if condition is then used within the loop to ensure all the nodes are updated. The use of this UDF for a simple 2-D mesh is shown in Fig. A-8.

Once the UDF for the orbit path was complete, the next step was to create another UDF to describe the surface velocity of the whirling rotor. This new UDF was defined with the DEFINE_PROFILE macro which is basically a model-specific macro that is used to define a custom boundary profile varying with time. The boundary condition specified could range from the velocity (as in this case) to the heat flux or even the wall roughness. The general syntax for this function is as follows:

**DEFINE_PROFILE (name, t, i)**

Where each argument can be described as follows:

- *name*, indicates the UDF name as assigned by the user
- *t*, indicates a pointer to the thread where the boundary condition is to be applied
- *i*, stands for the index that identifies the variable to be identified. This is set when the UDF is hooked with a variable in the boundary conditions panel.

Apart from the DEFINE_PROFILE macro, this UDF also uses a few other macros. CURRENT_TIMESTEP is one such macro used which returns the current physical time step size in seconds. The time step count is then accessed using the macro N_TIME. The
code also uses the macro F_CENTROID which is used to obtain the real centroid of a face, i.e. the coordinates of the centroid are stored in an array. The UDF written for the whirling of the rotor in a circular orbit, with a whirl ratio of 1 is as follows:

```c
#include "udf.h"
#include "stdio.h"

DEFINE_PROFILE(Vx_of_Rotor,t,i)
{
    real F_Area[ND_ND], x[ND_ND];
    real CG[ND_ND];
    real one[ND_ND];
    real R, w, Theta, del, k;
    face_t f;
    real time, dtime, Omega;
    real a;
    time=N_TIME;
    dtime=CURRENT_TIMESTEP;
    R=0.08205;       // Rotor radius in m
    del=0.000635;     // Eccentricity in m
    Omega=376.99; // Whirling speed in rad/sec
    w=376.99;        // rotor speed in rad/sec
    a=0;
    CG[0]=0.0;
    CG[1]=0.0;

    begin_f_loop(f,t)
    {
        F_CENTROID(x,f,t);
        CG[0] = CG[0] + x[0];
        CG[1] = CG[1] + x[1];
        a = a + 1;
    }
    end_f_loop(f,t)
    CG[0]=CG[0]/a;
    CG[1]=CG[1]/a;

    begin_f_loop(f,t)
    {
        F_CENTROID(x,f,t);
        one[0] = x[0] - CG[0];
        one[1] = x[1] - CG[1];
        R=sqrt(one[0]*one[0]+one[1]*one[1]);
        Theta = atan2(one[1], one[0]);
        F_PROFILE(f,t,i) = -R*w*sin(Theta-3.141592/2);
    }
    end_f_loop(f,t)
}

DEFINE_PROFILE(Vy_of_Rotor,t,i)
{

This UDF involves the use of two DEFINE_PROFILE macros, one for the x-component of the rotor surface velocity and the other for the y-component. For both the components, the computations performed are identical and they differ only in the component of velocity being returned by the function. The x-component of the velocity is computed by the first part of the code and the macro is identified by the name **Vx_of_Rotor**. Within the macro, the initial steps deal with defining the variables – the variables F_Area, x, CG and one are all defined as matrices while R, w, theta, del, k, time, dtime, Omega, and a are defined as scalars. The variable f is
then specified using the face_t data type which identifies a particular face within a face thread. Once all the variables have been declared, the next step is to initialize their values.

The first variable initialized is time and this is done using the N_TIME macro, which gives the number of time steps as an integer. The variable dtime is then initialized using the CURRENT_TIMESTEP macro which gives the current physical time step size (in seconds) as defined in FLUENT. The radius of the rotor is indicated by the variable R and its value is accordingly set to 0.08205 m and similarly the eccentricity (indicated by del) is set as 0.000635 m. Also, the variables a (which indicates the number of faces) and the two elements of the CG matrix (which would hold the position of the centroid) are all initialized as zero.

The begin_f_loop is then started, within which the F_CENTROID macro is used. This part of the code is basically used to compute the centroid for the entire seal geometry. Firstly the F_CENTROID macro is used to obtain the centroid of each face. The value obtained is stored in the x matrix and this is added to the corresponding elements of the CG matrix (which would finally contain the centroid for the geometry), and the value of the variable is simultaneously increased to account for each face. Once this loop is closed, the centroid is computed by dividing the values in the CG matrix by the number of faces. The next part of the code deals with the actual definition of the velocity profile for the rotor. For this, another begin_f_loop is started followed by the F_CENTROID macro. The values of R and Theta are computed from the values of the x and CG matrices and this is followed by using the F_PROFILE macro to set the boundary
condition for each face thread. The \texttt{begin\_f\_loop} ensures that this computation is carried out over all faces for all threads.

The same steps, apart from the boundary profile definition are carried out for the y component of the rotor velocity.

\textbf{Compiling the UDF and Using It in FLUENT}

Once the source code for the UDF is ready, it must be interpreted or compiled before it can be used by FLUENT for the purpose it was written. This basically means that the UDF which is written in a High Level Language needs to be translated to a lower machine language, so it can be directly understood by the computer. While the end result in both cases is the same, the method of compilation varies as in when a UDF is complied, the UDF is grouped into a shared library linked to the FLUENT executable but when the UDF is interpreted, it is read in at run time. Although interpreted UDFs have their advantages such as being compatible with other platforms and not requiring a C compiler, it has a restricted use which led to the use of compiled UDFs for this application. In addition to these reasons, the use of the \texttt{DEFINE\_CG\_MOTION} macro, limits the execution of this UDF to compiling alone. Before compiling or interpreting the UDF, it is, however, necessary to ensure that the source files for the UDF along with the case file for the problem are all contained in the same folder. It is also essential that the \texttt{udf.h} header file be accessible, either by the path followed or by saving a copy of it in the local working directory.
Using the complier leads to building a shared library which is then loaded into FLUENT and hooked to the model as required. Although in this application compiling the UDFs has been done using the Graphical User Interface (GUI), FLUENT also allows the use of a text user interface (TUI) which provides the added option of using a precompiled UDF from alternative sources such as FORTRAN. The process of compiling a UDF is, in essence a two step procedure – building (or the literal compiling of the UDF) and loading. The first step of building the UDF involves the processing of the source (or .c) file into a object file which is written into a shared library, ready for use. From the GUI, this is done by first accessing the Compiled UDFs panel which is located at:

**Define-> User Defined -> Functions->Complied**

From this panel, the source files for the program are selected, and the **Build** button is clicked. The build is complete once FLUENT creates the library (ex- libudf.lib) and the object (ex- libudf.exp) files. The next step is to load this library for use. Loading the UDF basically makes this library accessible to the program being used so it can be selected from the graphics panels of FLUENT. This can be done from the GUI by clicking the load button on the Complied UDFs panel. FLUENT responds to this by opening the built library and making it accessible in the appropriate graphics panels (such as that of the **Boundary Conditions**). Saving the case file at this stage enables the saving of the library with the case file, and thus the library is automatically loaded when the case file is read. This is known as *dynamic loading*.

After the UDF is compiled, it is ready for use within the program and this is done by hooking the UDF to the case file. As this application contains two UDFs each of
which have different purposes, they are hooked to the case individually. The UDF for the orbit path which was defined using the DEFINE_CG_MOTION macro is to be hooked using the Dynamic Mesh Zones panel (located at – Define-> Dynamic Mesh -> Zones). In this panel, each zone that is to move in the orbit path is selected and Rigid Body is selected under the Type. The motion is then specified by selecting the UDF from the drop-down list in the Motion/UDF profile under the Motion Attributes tab. The other UDF which was defined using the DEFINE_PROFILE macro and which specifies the surface velocity on the rotor is then hooked to the program. This is however done from the Boundary Conditions panel (accessed from Define->Boundary Conditions), by selecting the entities associated with the rotor wall and then for each entity the following steps are followed - under the Momentum tab, the Components option is selected and under the Velocity Components on the right the corresponding “udf Vx_of_Rotor” and “udf Vy_of_Rotor” are selected for the X-Velocity and Y-Velocity. Once the UDFs are complied and hooked, the case is now ready to be run.
RESULTS AND DISCUSSION FOR ANNULAR SEAL OF 50 mil CLEARANCE WHIRLING IN A CIRCULAR ORBIT

The path of the fluid in the seal is shown in Fig. A-9 and to understand this in better detail, a complete drawing of the inlet of the test rig is also shown in Fig. A-6. These figures clearly show the different clearances of the plenum, the swirl ring and the seal. The purpose of the plenum was to essentially maintain a uniform flow into the seal as well as keep out debris. The plenum was followed by an inner annular piece to which the swirl ring was attached, producing an annular nozzle which has provisions to produce pre-swirl for the flow entering the seal. The magnitude of swirl angle was increased by placing an annular diffuser (with an inlet clearance of 0.762 and exit clearance of 1.524) after the swirl blades. This slows down the axial velocity and increases the swirl angle. An important feature of the setup was the differences in the clearances at the exit of the diffuser and the inlet of the seal which causes the flow to experience a slight step upon entering the seal. The effect is exaggerated when no swirl ring is used as the difference in clearances is larger.

The section of the test rig that was simulated is shown in Fig. A-6 which also shows an enlarged image of the downward step adjoining the seal region. This step, as mentioned earlier has considerable impact on the flow and was therefore included in the model. The model initially used 15 nodes in the seal clearance along the radial direction and on performing a preliminary run it was seen that this not only showed $y^+$ values beyond the allowed range of 0 to 5 but also high gradients and leakage rate. This initial grid was then built upon by increasing the number of nodes in the clearance region (which is the region of interest here) thus refining the mesh, till grid independence was
attained. The variation in the exit mass flow rate with the number of nodes in the radial direction (in the clearance region) is shown in Fig. A-10. It is seen that increasing the number of nodes from 25 to 35 led to little change in the mass flow rate showing that grid independence has been reached. The final mesh used contains around 1,000,000 nodes with 35 nodes across the clearance. This is significantly larger than the 30x10 measurement grid used by Shresta [27].

Before discussing the results obtained, it is necessary to understand the nomenclature used in the discussion which conforms to that used in the literature so far. The annulus of the seal is divided into four approximately equal regions along its circumference, based on the pressure distribution seen in a journal bearing. As seen in Fig. A-11 (a), these four areas are the pressure side, the suction side, the high side and the low side. The pressure side corresponds to the area constricting in the direction of seal rotation while the suction side refers to the expanding clearance position. The largest and smallest clearances on the other hand are known as the high and low sides of the rotor respectively. The direction of the forces discussed and the line of action (as applicable to this work) are also shown in Fig. A-11(b).

**Experimental Results and Comparisons for a Statically Eccentric Annular Seal**

This section basically recalls the results from the LDA data of Johnson [2] and Shresta [27]. Johnson measured the velocity along the entire length of the seal and the operating conditions were $Re=24000$ and $Ta=6600$. The data used in both Shresta’s and Johnson’s work was the same, the data presented here is Shresta’s work which utilizes an
improved data analysis technique. The measurement grid used with the LDA data is seen in Fig. A-12 while the corresponding mesh used for the simulations is seen in Fig. A-13. From comparing the grids at the minimum clearance positions, it is apparent that the simulations use fine boundary layer meshes which enable a better resolution close to the wall. Also, the overall grid used in case of the simulations is finer than that used for the experimental data.

The contours for the LDA nondimensionalized axial, radial and azimuthal velocities for the four eccentric positions are presented in Fig. A-14 (a)-(d). The results from the LDA data have been nondimensionalized using the average leakage velocity which is 7.32 m/s. It is seen that the axial velocity is largest on the high and suction sides at the entrance but largest on the pressure side at the exit. The inlet of the high side shows small axial velocities in the lower half which are due to the downward step from the inner plug. The lowest velocities are seen on the minimum clearance side which also shows an axial decrease as the flow progresses through the seal. It should also be kept in mind that the LDA was unable to resolve any boundary layers as the boundary layers are thinner than the smallest distance from the seal surface (0.175c) that could be measured. The entire flow field can be explained by studying the flow pattern. As the flow enters the seal, it is first drawn towards the suction side which results in higher axial velocity at the entrance here. As the flow moves downstream, it is pulled to the pressure side due to the wall shear stress of the spinning rotor and this causes high axial velocities to migrate towards this side at the exit.

In case of the radial velocity contours, it is observed that for all the positions, the values of radial velocities lie within \( \pm 0.05 \) of the average axial velocity, except at the
entrance. At the entrance, there is a small region of positive velocity which is higher in
comparison to the downstream values. This could be due to some of the fluid entering the
low side being pushed into the space between the seal and the inner plug and this is
ejected to other eccentric positions by centripetal acceleration. On the maximum
clearance side, a small region of negative velocity is seen close to the entrance which is
again due to the downward step. The azimuthal velocity distributions for all eccentric
positions are similar in terms of the relative magnitudes and trends. The velocities
remained close to zero for most part of the seal except for the region close to the rotor
wall, which shows a layer of high velocities dissipating with distance away from the
rotor. The maximum clearance side also shows a small region of high velocity at the
bottom half in the entrance region which is again the effect of the downward step and it is
seen that at the exit, the azimuthal velocity reaches a magnitude close to 50% of \( U_{ave} \).

Fig. A-15 (a)-(d) show the results from FLUENT simulations for the same case as
investigated by Johnson and Shresta. This simulation used only the seal geometry, mass
flow rate and rotor speed from the experiments as inputs. All of the fluid velocities and
pressures are obtained from the CFD simulations. Comparing the axial velocity contours
at the minimum clearance positions, it is seen that the overall distribution and the
contours are the same in the case of both the experimental data and the simulations. The
location and the size of the maximum velocity region however, differ slightly and this
could be attributed to the fact that the experimental data was taken for a slightly different
angular position of the rotor, i.e. the CFD results were taken when the center of the rotor
was exactly 0.635mm from the center of the stator along the horizontal axis while this
would be hard to maintain for a experimental setup.
It is seen that the region close to the rotor wall shows much lower axial velocities for the experimental data and this could be related to the fact that the LDA was not able to measure this close to the wall; the data was hence obtained by a linear interpolation (which was performed by a graphics program) for the points on the rotor to a distance of 0.175 times the clearance from the rotor surface. The suction side predictions again show a discrepancy in the location and size of the high velocity regions and also in the region close to the rotor wall. The LDA data axial velocity is larger in the center of the clearance and less towards the sides. The maximum disagreement between the simulations and the experimental data is seen in case of the high side. Here it is seen that the simulations over predict the region of maximum velocity to be more uniform and shows an almost constant core with an increasing boundary layer. The pressure side distribution on the other hand, is predicted quite accurately. The distributions are almost identical excepting the region close to the rotor wall, the discrepancy for which has been previously described.

Comparing at the radial velocity contours at the minimum clearance location, it is seen that the predictions show good agreement with the experimental results. The small region of high velocity at the entrance, which is the effect of the different clearances of the plenum and seal, are a slightly over predicted by the simulations. The experimental data also show small regions of negative velocity spread over the entire length which are not seen in the simulated results. This is probably due to LDA alignment errors which were discussed by Shresta. The primary factor that he discussed is that the LDA system is not very competent in measuring the radial velocity distributions, due to laser beam orientations. The suction side comparisons show similar issues as in case of the low side.
The measurements here show the entrance effect spread out over the clearance while the simulations show the region of high velocity contained to a small region at the inlet.

In case of the maximum clearance side, the wake at the lower half of the entrance is seen for both the experimental data and the simulations but the simulations do not predict the high velocity at the entrance. The pressure side radial velocity distribution is almost identical to that of the suction side, in case of the simulations. The experimental data show significant differences between the two with more exaggerated regions of high and low velocity for the suction side. Also, comparison on the pressure side makes it apparent that the LDA is indeed unable to capture the boundary layer (owing to the reason mentioned earlier) which is seen clearly in the predictions.

The last set of comparisons made is for the azimuthal velocity contours from the experimental and the predicted data. The contours, in general, show a highly sheared flow with the minimum velocity region located along the stator wall and is seen as a direct impact of the spinning of the rotor. Comparing the azimuthal velocity distributions at the minimum clearance location, it is seen that although the overall distribution of the velocity is same for both the measurements and the predictions, the simulations slightly over predict the low velocity region. The boundary layers in case of the experimental measurements are larger and this is attributed to the inability of the LDA to take data close to the surface. The suction side contours from the simulations show excellent agreement with the experimental data, except for the region close to the rotor wall, as earlier. The azimuthal velocity distributions for the maximum clearance region from the simulations fairly match that seen in the experimental data. There is however a region of minimum velocity in the upper half of the seal for the experimental data which is not seen
in case of the simulations, also the shear seems to show a more gradual slope for the experimental data. A similar trend is seen in the pressure side comparisons, apart from the fact that here the minimum velocity region is seen to extend further downstream in the predictions than in the LDA measurements.

It is thus seen that the present numerical model predicts the flow field for the stationary seal with a good degree of agreement. The purpose of this comparison was to validate the GAMBIT model used as well as ensure the turbulence model used was performing satisfactorily. The simulation of the flow field upstream of the actual seal allows the inlet condition to be only the mass flow rate through the seal and the outlet to be an outflow condition. Both of these conditions allow the pressure to vary over the inlet and outlet boundaries. Therefore no previous knowledge of the velocity or pressure distributions was required.

The next step was to perform the simulations for a whirling seal. For the first case, a whirl ratio of 1 was chosen as the results could be compared to LDA measurements from Thames. The following section discusses this comparison in more detail.

**Experimental Results and Comparisons for an Eccentric Annular Seal at a Whirl Ratio of 1.0**

The results for these simulations have been presented differently from the previous case and they are presented in a similar format used by Thames in order to enable a one-to-one comparison. These are basically slices of the flow field at different axial positions which are similar for subsequent rotations. The slices have been taken at \( Z/L = 0.036, 0.11, 0.22, 0.77 \) and 0.86 (where \( L \) is the length of the seal section, 37.3
mm) again conforming to the LDA data available. The results are presented in Fig. A-16(a)-(e). The LDA data has been interpolated from the measured values which could not be taken at all angular locations.

At the slice $Z/L = 0.036$, the axial velocity contours are predicted almost exactly by the simulations. The regions of high and low velocities and their locations are reproduced quite accurately, the only discrepancy being that the area of low velocity is slightly larger in case of the predictions. This region of low velocity which is seen at the maximum clearance shows a region of stagnant flow which could be the wake of the sudden step in the inlet region. For the contours of radial velocity, the values are identical in case of both the simulations and measurements, however, the location of the region of low velocity as seen in the simulations is different from that seen in the LDA data. Also, observing radial velocity vectors indicates that they point outward near the rotor and inward close to the maximum clearance position on the stator which is the characteristic of a vena contracta. The tangential velocity is under-predicted by the simulations; the areas of high and low velocity are however reproduced correctly. The maximum discrepancy is seen at the region of maximum clearance which also holds for the predictions for the statically eccentric seal. Also, it is seen that as in case of the zero whirl ratio, the thin boundary layer close to the rotor surface was not captured by the LDA measurements but is clearly seen from the simulations.

The slice at $Z/L=0.11$ follows the same trends as the previous slice. The axial velocity contours from the LDA measurements show a significant decrease in the maximum values but this decrease is not very high in case of the simulations. The profile in general is more evenly distributed owing to the region of maximum velocity
moving to a wider clearance, which decelerates the flow. The radial velocity contours again show good agreement while the contours of tangential velocity show an under-prediction as in case of the previous slice. The tangential velocity contours show a slight overall increase due to the rotor tangential shear stress continually accelerating the azimuthal velocity as the flow progresses through the seal.

For the next slice at \( Z/L = 0.22 \), the axial velocity contours from the measured values again show a significant drop in small clearance values as opposed to the more gradual decrease in case of the predictions. Also, the predictions show a small region of zero velocity very close to the rotor which is absent in the LDA data and this could be attributed to the fact that LDA data very close to a surface is hard to obtain. The radial velocity values are relatively lower than the tangential and axial velocities as in the previous observations and these contours show overall agreement except close to the region of minimum clearance where the values are slightly lower for the predictions. The tangential velocity is again under-predicted by the simulations and it is noticed that the LDA is again unable to predict the variations in flow very close to the rotor.

The next axial slice at \( Z/L = 0.77 \) shows little variation in the axial velocity around the circumference for the LDA data. This is however not the case for the simulated results which shows a significant change from the previous slice at \( Z/L=0.22 \) - a region of high velocity is seen to develop on the lower half of the pressure side (close to the maximum clearance region), the magnitude of which is much higher than that seen for the previous slice. The radial velocity comparisons show relatively better agreement except for the small region of slightly high velocity which is not seen in the simulated results. The tangential velocity contour comparisons are more accurate that for
the previous slices and the overall distributions and ranges of velocity are relatively similar.

The same trends as for the slice at Z/L = 0.77 are repeated in case of the slice at Z/L = 0.86. The axial velocity contours are overall under-predicted by the simulations which do however follow through the distributions seen over all the other slices. The radial velocity fields are again well reproduced except for the location of the region of low velocity which is at the minimum clearance in case of the simulations. The tangential velocity contours show better agreement here than in the initial slices, the distributions are almost identical but the measured values do show a slightly higher value of maximum velocity in a few small locations.

Upon studying the flow fields over all the slices, it is possible to make a few conclusions regarding the physics of the flow. It is seen that the axial velocity showed maximum values at the entrance of the seal, on the pressure side and rotates circumferentially to the suction side at the exit. The tangential velocity on the other hand develops with almost a reverse trend, i.e. the values are lower at the entrance and slowly build up towards the exit. Overall, the predictions agree with the data to a fair extent. To obtain highly accurate results, it might be necessary to investigate other turbulence models or use a much finer grid with an LES scheme.

**Effects of Varying Whirl Ratio**

The next step that followed the simulations discussed so far was to perform simulations for varying whirl ratios. This was prompted by the changing of the maximum and minimum velocities and pressure locations observed both experimentally and numerically when the whirl ratio varied between 0 and 1. Similar conditions were
experimentally investigated by Suryanarayanan who investigated the impact of the whirl ratio on measured wall pressure distributions. The flow fields for whirl ratios 0 though 1 at intervals of 0.2 were simulated and the axial, radial and azimuthal distributions have been presented in Fig. A-17 (a)-(c).

Comparing the contours of axial velocity for all the whirl ratios shows the expected trend where the region of high velocity at the entrance slowly moves circumferentially around the seal (from pressure to suction sides via the minimum clearance region) as the whirl ratio increases. For the statically eccentric seal, the region of maximum velocity, seen initially on the pressure side, passes through the minimum clearance region and exits the seal on the suction side, i.e. the region of maximum axial velocity rotates in the direction of the spinning rotor.

This trend persists for a whirl ratio of 0.2 in addition to a wake close to the rotor wall on the maximum clearance side which also moves from pressure to suction sides. At a whirl ratio of 0.4, however, the region of high velocity shows no movement around the circumference but the magnitude of the maximum value seems to decrease rapidly in the first few inlet planes. This trend is repeated for a whirl ratio of 0.6 as well.

Contours at whirl ratios 0.8 and 1.0 show a trend almost completely opposite to that seen for the statically eccentric seal. Here, the region of maximum velocity, initially at the suction side, rotates to the pressure side, through the minimum clearance side. For all the whirl ratios, as observed earlier, the region of high axial velocity dissipates towards the exit of the seal and so does the wake seen at the entrance. The magnitude of maximum axial velocity, begins high for the statically eccentric seal, decreases with the
onset of whirl to a minimum at a whirl ratio of 0.4 and then begins to increase with increasing whirl ratio.

The trends seen for the radial velocity contours differ significantly from those of the contours of axial velocity. The velocity distributions for the statically eccentric case show velocities close to zero almost all through the seal but this changes with the onset of whirl. The regions of high radial velocity are seen to again rotate from the pressure side towards the suction side, but in the opposite direction of the axial velocity via the maximum clearance location and the magnitude remaining almost the same through increasing whirl ratios. Another evident observation is that the radial velocity rapidly decays from having a maximum at the seal entrance to negligible velocities at the exit, and this holds for all whirl ratios. On comparing the radial and axial velocities, it is seen that unlike the axial velocity contours for which the high velocity region was around the minimum clearance region, the high velocity region for the radial velocity contours is located on the pressure or suction sides.

The azimuthal velocity distributions do not show much variation with whirl ratio but does show a significant increase from the entrance towards the exit for all whirl ratios. The amount of swirl present though, decreases from whirl ratio 0 to 0.2, increases at 0.4 and then decreases and stays relatively constant from 0.6 to 1.0. An important observation made here is that for the last few planes towards the exit, the contours are identical for all whirl ratios. In order to provide a more complete picture of the effect of the whirl ratio, the pressure distributions for all the slices at all whirl ratios are provided in Fig. A-17(d). It was not possible to maintain the same contour levels over all whirl
ratios owing to the wide variation of the pressure levels. A detailed analysis of the pressure contours for each whirl ratio is discussed in the following paragraph.

The values of the pressure are seen to be the lowest for the statically eccentric case and understandably increase with the increase in whirl ratio. The location of the region of maximum pressure in this case is closer to the suction side and quickly decreases within the first few axial slices. The minimum pressure on the other hand slowly increases in magnitude and moves from the pressure side to the suction side, through the minimum clearance, tracking the location of maximum axial velocity. The values of pressure for a whirl ratio of 0.2 are slightly higher and also the maximum pressure region is now closer to the minimum clearance location. The minimum pressure follows the same trend as earlier but does not go beyond the minimum clearance position. This is similar to the maximum velocity location which also stops its rotation at the minimum clearance location for this whirl ratio.

The pressure on the entrance plane for the whirl ratio of 0.4 differs from the other two cases so far in the aspect that the minimum pressure region has moved to the maximum clearance position, which corresponds to one of the two maximum velocity locations at the inlet, the other being at the minimum clearance. Also, there are two high pressure regions, on either side of the minimum clearance region corresponding to the regions of minimum velocity. These two regions however merge into one a few planes down and migrate to the maximum clearance position towards the exit as did the minimum velocity locations. Another observation made is that a low pressure region develops on the pressure side of the seal just past the minimum clearance. The pressure distribution for whirl ratio 0.6 is similar to that at whirl ratio 0.2, the values being a little
higher than that at 0.2. There is however a difference in the high pressure region, in the fact that it is more towards the maximum clearance position for the 0.6 whirl ratio.

At whirl ratios of 0.8 and 1.0, it is seen that the minimum pressure at the inlet flips over to the suction side as compared to being located on the pressure sides as in case of whirl ratios 0.2 and 0.6. Also, the pressure distribution becomes relatively uniform in the central regions and the minimum pressure region on the exit plane is on the pressure side. Again, this is consistent with the locations of the maximum axial velocity.

The changing of the pressure distribution around the seal with varying whirl ratio indicates a change in the radial force magnitude and line of action. Fig. A-18(a) presents the magnitude and direction of the force while Fig. A-18(b) represents the change in the line of action of the force with the whirl ratio. From the plot, it is clear that the force direction changes as the whirl ratio increases and there is a significant change in the direction between whirl ratios 0.4 and 0.6. The viscous drag on the rotor is shown in Fig. A-18(c). It is apparent that there is a significant increase in the magnitude of this component as the rotor begins to whirl, but it remains relatively constant for all whirl ratios. It has been observed [5] that it is the tangential force that has a greater impact on the stability of the system. Tangential forces that are in the same direction as the whirl velocity have a destabilizing effect while those in the opposite direction have a stabilizing effect. Thus the tangential forces seen in this case are whirl stabilizing leading to better stability overall.

**Computation of Rotordynamic Coefficients**

The study of rotordynamic forces in seals has been a long researched topic and there are many models that have been developed over the years in order to compute them. For
small radial displacements about the rotor’s arbitrary positions the reaction forces \( F_x \) and \( F_y \) can be modeled as

\[
-\begin{bmatrix}
F_x \\
F_y 
\end{bmatrix} = \begin{bmatrix}
K_{xx} & K_{xy} \\
K_{yx} & K_{yy}
\end{bmatrix} \begin{bmatrix}
x \\
y
\end{bmatrix} + \begin{bmatrix}
C_{xx} & C_{xy} \\
C_{yx} & C_{yy}
\end{bmatrix} \begin{bmatrix}
\dot{x} \\
\dot{y}
\end{bmatrix} + \begin{bmatrix}
M_{xx} & M_{xy} \\
M_{yx} & M_{yy}
\end{bmatrix} \begin{bmatrix}
\ddot{x} \\
\ddot{y}
\end{bmatrix}
\]  

(29)

where \( x, \dot{x}, \ddot{x} \) and \( y, \dot{y}, \ddot{y} \) are the displacements, velocities and accelerations in the \( x \) and \( y \) directions. When the nominal position of the rotor is concentric with respect to the housing, the coefficient matrix becomes simpler and assumes a skew-symmetric form

\[
-\begin{bmatrix}
F_x \\
F_y 
\end{bmatrix} = \begin{bmatrix}
K & k \\
-k & K
\end{bmatrix} \begin{bmatrix}
x \\
y
\end{bmatrix} + \begin{bmatrix}
C & c \\
-c & C
\end{bmatrix} \begin{bmatrix}
\dot{x} \\
\dot{y}
\end{bmatrix} + \begin{bmatrix}
M & m \\
-m & M
\end{bmatrix} \begin{bmatrix}
\ddot{x} \\
\ddot{y}
\end{bmatrix}
\]  

(30)

where \( K \) and \( k \) are the direct and cross-coupled stiffness coefficients, \( C \) and \( c \) are the direct and cross-coupled damping coefficients and \( M \) and \( m \) are the direct and cross-coupled mass coefficients. Among these coefficients, the components \( k \) and \( C \) are important in the determination of rotodynamic stability. A positive value of \( k \) represents a destabilizing stiffness force (acting in the whirl direction) and a positive value of \( C \) represents a stabilizing damping force (acting against the whirl). These two coefficients are also combined together to obtain an effective damping coefficient \( C_{eff} [ = C - \frac{k}{\Omega} \) where \( \Omega \) is the rotor speed] for which a positive value represents a whirl resisting force.

Between the two, \( k \) is the crucial factor that can be used to establish rotor stability and it is hence important to determine its value accurately.

This research basically uses the theory from the CFD perturbation model which was initiated by Dietzen and Nordmann. This model was worked on and improved by many researchers including Kim et al. [29] and Venketesan and Rhode [30]. One of the
most recent studies is by Xi and Rhode [26] and the relations that have been used here can be applied to the present case as well. This study basically involved solving perturbed zeroth and first order governing equations using a finite-volume CFD code that used the SIMPLEC scheme. Once the equations were solved and the resulting tangential and radial force components were obtained, the components were related to the rotordynamic coefficients using

\[
\frac{-F_r}{e} = K + c\Omega - M\Omega^2 \tag{31}
\]

\[
\frac{-F_t}{e} = -k + C\Omega + m\Omega^2 \tag{32}
\]

This was used in conjunction with a least squares curve fitting with respect to whirling speed to obtain the rotordynamic force coefficients.

The radial and tangential force components obtained from the simulations are shown in Fig. A-18 (a) and (b) respectively. It is seen that the radial force initially exhibits a decrease in magnitude from whirl ratios 0 through 0.4 after which an increase is seen from 0.4 through 0.8. This is then followed by a slight drop from 0.8 to 1.0. The initial decrease in force is seen to be consistent with the decrease in pressure variations for these whirl ratios. The increase from whirl ratios 0.4 to 0.8 however, takes place even though the pressure variation remains constant, which indicates that there must be a change in the pressure distribution which causes the change.

The rotordynamic coefficients are obtained by a curve fit as shown in Fig. A-19 (a) and (b) and are consolidated in Table B-1. The plot shows the variation of the radial and tangential impedance \(\left(\frac{F}{e}\right)\) with the whirl ratio in order to ensure that the rotordynamic
coefficients can be directly read off the curve fit. As seen from the plot, the data does not conform exactly to a second degree polynomial. This discrepancy could be attributed to the fact that the simulations may be unable to capture the presence of Goertler vortices which are formed along the axial flow direction and will be investigated further in the following sections.
RESULTS AND DISCUSSION FOR ANNULAR SEAL OF 50 mil CLEARANCE
WHIRLING IN A LINEAR ORBIT

The previous section presented the results and the discussion for the case of an annular seal whirling in a circular orbit. The simulation has been compared to experimental data under similar conditions. During the real-time operation however, at high shaft speeds the seal is likely to move in orbits that are not perfectly circular and that cannot be simulated precisely in the test rig. One of the main objectives of this work is to establish a method that can numerically simulate any orbit, and use the results of these predictions to determine the effect of the orbit in terms of the flow field and the rotordynamic forces. This was initiated by simulating the rotor motion in a linear orbit and analyzing the flow field and the rotordynamic coefficients as before. The results for the condition where the rotor speed and the orbit path speed are equal (analogous to a orbit path speed of 1.0) have been presented in detail in the following section. This is followed by a discussion of the varying orbit path speed on the linear orbit and the analysis of the forces for the same.

Results for an Eccentric Annular Seal where Orbit Path Speed Is the Same as Rotor Speed

The presentation of results for this case of simulations follows the technique of taking axial slices of the seal and magnifying the clearance to see the flow field clearly. Slices have been taken at the same axial positions as before in order to enable a one to one comparison with the results from the circular orbit. In this case, the rotor motion does not assume a steady state solution but is moving back and forth. Thus, it is necessary to
select specific moments (phase or percent of cycle) to present the data. The slices of data will be presented at intervals of 45 degrees, in order to better analyze the field at different positions of the rotor. The changes in the angle have been dealt with as a percentage change of cycle where one complete cycle (of 360 degrees) involves one complete back and forth motion. Fig. C-2 (a)-(c) present the axial, radial and azimuthal velocity distribution fields respectively for these positions of the rotor as it moves along the orbit path. 0% of the cycle here represents the minimum clearance position with the minimum clearance located at the left-most position.

Fig. C-2 (a) presents the axial velocity flow fields for all slices through the seal, for different positions of the rotor. Looking at the flow initially, i.e. when the position of the rotor is at 0% of the cycle, it is seen that there are two regions of high velocity which develop in the planes towards the exit. The gradual development of these two zones shows the initial inertial effects being overcome and giving way to fully developed regions of maximum velocity as the flow moves downstream. The very presence of two separate high velocity regions on the other hand, can be seen as a direct impact of the rotor’s orbit path and this is also supported by the pressure distributions (as seen in Fig. C-2(d)). This can be confirmed by looking at the simulation results for the case of circular orbit, Fig. A-17 (a) where the distribution for a orbit path speed of 1.0 (which would be the same at any angle) clearly indicates just one high velocity region which appears to rotate with the rotor spinning. In addition to this, a small region of relatively high negative velocity is also seen which develops as the flow moves downstream.

As the rotor begins to move in the orbit path, i.e. from left towards right (in the direction of the x axis), the fluid responds by showing slightly larger regions of high and
low velocities which are at almost constant positions throughout the operation. At the next position being analyzed at 12.5% of the cycle, two regions of high velocity on either side of the minimum clearance are seen. Also, it is observed that for almost the entire high side the flow is close to stagnant except for regions close to the rotor. At the next position of 25% of the cycle, the rotor is now concentric with the stator and this is reflected by the two regions of maximum velocity almost merging into one. The high velocities occupy almost the entire minimum clearance region leading to an overall decrease in the magnitudes, while the negative velocities are of much lower magnitudes.

The next two positions of the rotor are taken at 37.5% and 50% of the cycle respectively. As seen from the velocity distributions, the flow field now shows the impact of both the eccentricities as well as the overall development of the cycle in terms of the fact that it now begins to resemble that seen in case of the circular orbit (with respect to both the magnitudes and overall pattern). Physically, by this point in the cycle, the fluid has moved away from the initial region it was confined to (due to the position of the rotor) and the combined effect of the orbit motion and the rotor spinning result in it moving further into the minimum clearance region which is now on the left.

As the rotor begins to complete the other half of the cycle, the flow fields are observed to progress in the opposite sequence as before; the magnitudes of velocities are, however, slightly lower than that for the corresponding position in the forward motion. It can thus be concluded that there is hardly any directional dependence of the path on the flow field. Another observation to be made is that this orbit path produces distinctly higher velocities compared to the circular orbit path. This could lead to greater
magnitudes of forces which could result in having higher impact on the stability and rotordynamic coefficients.

Fig. C-2(b) shows the radial velocity distribution profiles for different positions of the rotor. The initial position of the rotor shows two small regions of high velocity, analogous to the axial velocity distribution, however, unlike the axial velocity regions, these two regions show rapid dissipation within the first few slices. The two regions of negative velocity seen are of almost equal magnitude and follow the same trend, indicating that the fluid at the inlet could squirt out of the seal. Also, an additional region of high velocity is seen on the suction side which begins to develop towards the exit. As the rotor begins to move along its orbit path, it is seen that similar trends follow for the next position in addition to which a region of high velocity is seen to also develop along the minimum clearance region. It is seen that as the rotor moves in the linear path, the effect of the spinning becomes more noticeable which results for the next two positions in the two regions of maximum velocity merging into one and the magnitude steadily reducing.

The backward motion of the rotor however, shows an interesting flip of the location of the maximum velocity region to the maximum clearance side. This could be explained as follows – the axial velocity distributions saw much higher magnitudes of velocity impacting from the orbit path so that the effect of the rotor spin was hardly seen. This is not the case for the radial velocity component, where the magnitudes are small enough to be affected by the spinning motion. As seen for the distributions at 87.5% of the cycle, the region of high velocity seen in the maximum clearance region begins to
spin towards the suction side and the two other regions of high velocity develop thus
tending to the same distribution as the 0% position.

The azimuthal velocity distributions for the linear orbit path of the rotor are
shown in Fig. C-2 (c). The trends seen at the inlet for the 0% position are along the lines
of that seen for the axial and azimuthal velocities and the ranges compare to that seen for
the circular orbit, apart from the presence of the negative velocity region. This region of
high negative velocity is again dominant at the inlet and dissipates within the first few
planes and could be related to the squishing of the fluid in the clearance. As the flow
moves downstream into the seal, an additional region of maximum velocity is also seen
to develop close to the rotor wall along the pressure side which extends into parts of the
maximum clearance region. It is observed that as the rotor motion progresses, this region
becomes more predominant until at about 25% of the cycle where it is seen to cover
almost the entire pressure side. It is also seen that the other region of high velocity seen
initially along the suction side rotates closer to the minimum clearance region until this
position which shows the effect of rotor spin (as seen for the radial velocity distribution).

For the next position analyzed at 37.5% of the cycle when the rotor is almost
halfway through the cycle, the presence of the smaller high velocity region on the suction
side altogether disappears as the rotor motion has moved most of the fluid from that
region. Another important observation made here is that the regions of minimum velocity
are the largest at this position and do not dissipate completely even at the exit. The same
trends are seen at 50% of the cycle, apart from the fact that the minimum velocity regions
are smaller. The next position at 62.5% clearly shows the impact of the rotor’s path as it
begins to move from right to left. The fluid is now being forced into the region close to
the suction side where a high velocity region is seen to develop which grows as the flow moves downstream. As the rotor starts to move closer to its original position, the secondary region of high velocity begins to show on the pressure side which becomes more dominant as the rotor moves further into the orbit.

Analyzing the pressure contours helps further understand the characteristics of the flow field. Looking at the initial position of the rotor, the pressure distribution seen justifies the velocity distributions – there are two regions of maximum pressure which dissipate as the flow progresses downstream. This trend is repeated for the next two positions as well the magnitudes of pressure being slightly lower. The next position at 37.5% of the cycle shows the values of pressure beginning to increase; the overall distribution however remains the same. The position at 50% of cycle (when the rotor is at the extreme left) shows a pressure distribution similar to that seen for a circular orbit – the high pressure regions here is limited to the minimum clearance region and tends to spin towards the suction side, which is the direction of rotor spin, as the flow moves downstream. The values of the magnitude of pressure are however much lower than that for the initial position.

As the rotor starts to move back towards the right, the trends shown retrace that seen during the forward motion of the rotor. The presence to two separate regions of high pressure on either side of the maximum clearance region are seen again with the magnitudes initially lower (as in the case of 62.5% of the cycle), then reaching values higher than that seen for the forward motion (at 75% of the cycle) and finally showing slightly lower values.
Effect of Varying Speed of Rotor Orbit

As in case of the circular orbit path, it is important to study the effect of the speed of the orbit path on the flow field for the linear orbit as well. The contours for axial, radial and azimuthal velocities for all orbit path speeds from 0.2 through 1.0 are presented in Fig. A-20 (a) - (c), while those for the pressure are in Fig. A-20 (d). Comparing the contours of axial velocity for all the orbit path speeds, it is seen that the change in orbit speed does cause minor variations in the flow field with respect to the location of the maximum velocity regions. The regions of high velocity for an orbit path speed of 0.2 are located on the pressure and suction sides, close to the minimum clearance region. It can be seen that at even this low orbit path speed, the effect the rotor spin is not very prominent as the maximum velocity zones remain at almost the same location from the inlet to exit of the seal. Moving from the inlet to exit however, does show a gradual dissipation of the maximum velocity, which is slower than that seen for the circular orbit.

The contours of axial velocity at an orbit path speed of 0.4 show a similar trend to that seen for the orbit path speed of 0.2, the significant difference being that the two regions of maximum velocity seem to have moved farther from the minimum clearance region and are now located entirely in the pressure and suction sides. This pattern is confirmed by observed at the contours for an orbit path speed of 0.6 times the rotor speed, where the maximum velocity regions have moved even closer to the maximum clearance region. At an orbit path speed of 0.8 times rotor speed however, the trend seems to reverse itself and the locations of the maximum velocity regions resemble that seen at a orbit path speed of 0.4. Finally, at a orbit path speed equal to the rotor speed, the maximum velocity regions
move back to their positions at a orbit path speed of 0.2, showing little change in the magnitudes throughout.

The radial velocity contours’ overall trends possess very low velocities over almost all the seal except for small regions of high velocity. For an orbit path speed of 0.2, two small regions of maximum velocity are seen, corresponding to the location of the maximum axial velocity regions. These regions however, show rapid decay, resulting in low velocities, farther downstream of the inlet. With an increase in the orbit path speed, another small region of relatively high velocity is clearly seen on the suction side, the magnitude of which seems to slightly increase downstream.

As in case of the circular orbit path, the overall distributions of the azimuthal velocity do not vary greatly with the change in orbit path speed. At an orbit path speed of 0.2, it is seen that there are two regions of maximum velocity, located on the minimum and maximum clearance sides respectively. The magnitudes and locations of these regions remain relatively constant as the flow moves down stream. An interesting observation made, however, is that the regions of stagnation seen in the inlet are dissipated in the same direction as the rotor spinning. This accounts for the fluid inertia being overcome by the centripetal acceleration which is imparted to the fluid due the movement of the rotor. Studying the pressure contours over the different orbit path speeds helps analyze and understand the flow better. As seen in Fig. A-20 (d), the overall ranges of pressure remain almost the same for all the orbit path speeds except for the orbit path speed of 1.0.

The pressure distribution at a orbit path speed of 0.2 explains the locations of the regions of high velocities seen so far. These regions decay gradually towards the exit
until almost uniform values are obtained at the exit plane. A similar trend is seen in the case of the orbit path speed of 0.4. However, the values of pressure seen in the maximum clearance region here are higher than that seen for the previous orbit path speed. Identical trends are maintained for orbit path speeds of 0.6 and 0.8, and for both cases the locations of high pressure regions at the inlet correspond to the high velocity regions. The highest magnitudes of pressure, as expected, are seen for the highest orbit path speed. The ranges shown for these contours could not be kept the same as that for the other orbit path speeds owing to the large variation of pressure values. It is seen that the regions of high pressure are still consistent with the regions of high velocity and they dissipate as the flow moves from the inlet to exit.

**Analysis of Resulting Forces**

The forces exerted on the rotor for this linear motion are expected to be much larger than that seen for the circular orbit, judging from the pressure magnitudes. Since for the linear orbit path, the eccentricity of the rotor is not constant, Childs’ theory cannot be applied as earlier. The stability characteristics must therefore be analyzed based on the variation of the forces over the length of the cycle. Fig. A- 21 (a) shows the variation of the radial and tangential components of the forces along with the line of action plotted against the % of the cycle for the case when the orbit speed is 20% of the rotor speed. Although the radial component is larger than the tangential component, both follow the same trend and show maximum values at two points – around 20% and around 50% of the cycle. It is also observed that the direction of the force hardly changes over the duration of the entire cycle.
Fig. A-21(b) shows the same factors analyzed for 40% of the rotor speed. The trend still shows two maxima but their occurrences are now at 10% and 80% of the cycle. The direction of the force also shows a significant change between 20 to 40% of the cycle which coincides with the minimum force location. The trend for the forces seen here, is similar for the next case of orbit speed, however there is a significant change in the variation of the line of action – the peak for this curve is now seen at close to 60% of the cycle and almost coincides with the location of maximum force. For the next case of 80% of rotor speed as seen in Fig A-21(d) the trends for the forces are seen to roughly follow that seen at 40% of rotor speed. The line of action however shows the same variation as the previous case. Looking at these four cases together it is seen that the variation of the force trends with the orbit speed tend to switch between two patterns which differ with respect to the location of the maximum and minimum forces. The line of action on the other hand, which is an indicator of the direction of force, begins to show development at around 40% of rotor speed and then remains relatively the same for the next two cases.

The last plot in Fig. A-21 (e) is the point when the rotor speed and orbit path speeds are both equal. Here, it is clearly seen that the highest magnitudes for both components occur at two points during the cycle – at around 20% and close to 80%. Looking at the physical interpretation, both 20% and 80% of the cycle are close to the positions when the rotor is almost concentric with the stator. The minimum values on the other hand differ in terms of the positions at which they occur, for the two components – in case of the radial forces the least values are at around 40% and 90% of the cycle while the tangential forces are low at 10% and 60% of the cycle. Overall, the radial forces show a more symmetric behavior and with their magnitudes being much larger that the tangential
component. They have a greater impact on the stability of the system. The changes in the sign on the tangential component again show that it has both stabilizing and de-stabilizing effects at different points in the cycle.
RESULTS AND DISCUSSION FOR ANNULAR SEAL OF 5 mil CLEARANCE WHIRLING IN A CIRCULAR ORBIT

The previous sections presented the results for an eccentric annular seal of 1.27mm clearance whirling in a circular and a linear orbit. This work determined the effectiveness of the method used in predicting the flow and observed the impact of the orbit path on the flow field and rotodynamic coefficients. The validation of the methodology permits the simulation of a seal with a 0.127mm clearance in order to study the effect of the clearance of the seal on the flow field and the rotodynamic coefficients. This will also help overcome some of the difficulties of the LDA system in measuring flow parameters in such small clearances.

The dimensions of the seal were kept the same as in the 50mil case apart from the clearance which was altered to 0.127 mm. The creation of the mesh included establishing grid independence. Fig. A-22 shows the variation of the mass flow rate at the exit as a function of the number of nodes (across the clearance). The plot clearly shows that there is hardly any change in the mass flow rate at 28 and 38 nodes hence proving that grid independence has been obtained. The latter model was used in the simulations. Although there is no experimental data to compare against, the mass flow rate was taken as one-tenth of the 1.27mm case, 0.486 kg/s to keep the axial velocity average the same as the 1.27mm case. The non-dimensional parameters such as the Reynolds number and the Taylor number were 2400 and 208.71 respectively.
Results For Eccentric Whirling Seal of 5 mil Clearance at Whirl Ratio of 1.0 Compared Against Seal of 50 mil Clearance

Fig. A-23 (a)-(e) show the flow fields for both the clearances presented alongside each other so as to enable a direct comparison. The locations of the slices have been kept the same as used in the previous analysis. For the slice at $Z/L = 0.036$, the axial velocity contours show similar ranges of velocity for most parts of the seal. This is to be expected as the mass flow rate for this case has been taken as 1/10 that of the 50mil case which would have a direct impact on the axial velocity. The location of the high velocity region for the 5mil case has however moved close to the maximum clearance region on the suction side with a slightly lower magnitude. The radial velocity contours show the same trends in both cases, the only exception being that the small region of high velocity seen in the 50mil case is not seen in the 5mil case. A significant impact of the change in clearance is seen when the tangential velocity distributions are compared. The initial observation made is that the magnitude of tangential velocities is higher, almost twice the original, in the case of the 5mil seal, which is expected owing to the smaller clearance. The overall distributions also differ with respect to the high velocity regions being located close to the rotor in the 50mil case but in the minimum clearance region for the 5mil case.

Moving further into the seal, Fig. A-23(b) shows the contours on slices at $Z/L = 0.11$. From the axial velocity contours, it is seen that for the 5mil case, the region of maximum velocity has moved slightly closer to the maximum clearance region. It is thus observed that the direction of rotation of this region is the same for both the clearances, i.e., the counter-clockwise direction which is the direction of the rotor spinning. The
radial velocities are again almost identical showing that the change in the clearance has hardly any effect on this component. The tangential velocity distributions for the 5mil case develop further, overcoming inertia to show larger regions of high velocity which continue to maximize at the minimum clearance region. This is however not the case for the 50mil seal which shows little variation between the two slices.

For the axial location at \( Z/L = 0.22 \), the axial and radial velocity distributions show little variation for both cases, except for the maximum axial velocity region which is seen to move slightly closer to the maximum clearance region. The tangential velocities for the 5mil case continue to develop further leading to a high velocity region spanning the entire low side and parts of the pressure and suction sides. The 50mil case also shows a significant increase in the velocity for regions close to the rotor. The next axial location for the 50mil case at \( Z/L = 0.77 \) shows reduced values of axial velocity which are located along same regions as the previous slice. The azimuthal velocity on the other hand begins to show increased values as the rotor drag continues to accelerate the fluid. The contours for the 5mil seal show that the high axial velocity region has moved over to the pressure side after passing through the maximum clearance region. The radial velocities for this slice differ slightly as the minimum velocity region seems to span more area in case of the 50mil case. The tangential velocities for both the clearances are now fully developed and show almost uniform values all over the slice. This makes the difference in the magnitudes for both cases more apparent and the effect of the clearance is clearly seen.

At \( Z/L = 0.86 \) (Fig. A- 23 (e)) the flow is almost at the exit of the seal. The axial velocity distributions for the 5mil case shows hardly any variation but the 50mil case shows a further decrease in the maximum value, for the same trend as before and moved
slightly towards the suction side. The agreement for the radial velocities is maintained apart from a slight discrepancy close to the pressure side. The tangential velocities are almost identical to the previous axial slice for both cases proving that they are indeed fully developed. An interesting observation is made when the overall azimuthal velocity distribution for the 5mil case is compared to that of the 50mil seal - in case of the 50 mil seal, a thin boundary layer was seen close to the rotor surface (the thickness of which was approximately 14% of the minimum clearance); when the clearance was reduced to a tenth of the original, this boundary layer fills the entire clearance (seen clearly in the slices towards the exit) hence causing viscous effects to dominate the flow.

The variation of the pressure along the length of both the seals is also shown in Fig. A-23 (f). The ranges of pressure show a smaller variation in the 5mil case as compared to the 50mil case, however, the 5mil case causes a larger pressure drop in the computational domain owing to the decrease in clearance. Another observation made from these contours is that the variation in the pressure contours is more gradual and smooth in case of the 5mil seal as compared to the 50mil case. This indicates the fact that viscosity effects in the 5mil seal could be more dominant, creating a damping effect. Table B-2 gives a thorough comparison of the pressure variations for both the 5 and 50mil.

In case of the 50mil seal, it was seen that the high pressure regions are on the suction side of the seal and this coincides with the maximum axial velocity locations and the trends seen for both were similar. For the 5mil seal however, this relation does not hold and it is seen that the high pressure regions are on the pressure side and in fact coincide with the regions of minimum axial velocity. This confirms the theory that the flow field is indeed viscous dominated on the basis of the fact that the high pressure is on the pressure
side and maximum axial velocity does not depend on the maximum pressure regions.
The significant differences in the flow fields at 5 and 50mil are thus attributed to the fact
the flow field in case of the 5mil seal is viscosity dominated while the latter is inertia
dominated.

**Effect of Varying Whirl Ratio**

Fig. A-24 (a) shows the axial velocity distributions of the simulations for whirl
ratios 0 through 1.0 for the 5 mil seal. During the analysis of these figures, it should be
kept in mind that the locations of the pressure and suction sides appear to be reversed
compared to that seen in the previous section, owing to the direction of the X axis (which
is into the plane of paper). The overall ranges and the locations of the regions of high and
low velocities show minor variations with whirl ratio. The locations of these extremes are
different from the 50mil case and the magnitude of variation is smaller. The flow field at
zero whirl shows the region of maximum velocity located on the maximum clearance
side. The location of this region seems to rotate from the pressure side towards the
suction side from the entrance to the exit planes, which also seems to be the case for all
other intermediate whirl ratios. With increase in whirl ratio, the maximum velocity region
at the inlet reduces in magnitude and rotates to the suction side with increasing speed.
However, for the large whirl ratios, the region of maximum axial velocity rotates towards
the suction side as the flow progresses downstream. For all whirl ratios, the location at
the exit is almost the same for all whirl ratios. Another observation to be made is that
this region of high velocity which is located almost entirely in the maximum clearance
seems to migrate towards the suction side for whirl ratios beyond 0.6. This could be due
to increase in the centripetal force (due to higher whirling speeds) which is seen in the
high velocity region moving in the direction of the whirling motion. For the distributions of whirl ratio 0.2 and greater, another observation made is that a small region of maximum velocity seems to appear and grow in the planes towards the exit.

The radial velocity distributions are shown in Fig. A-24 (b) for all whirl ratios from 0 through 1. The relative magnitude of this component of velocity is again lower than that obtained for the 50mil case. The trends, however, seem to be similar as there is little variation over the axial slices for all whirl ratios. For the statically eccentric seal, the magnitude and relative trend of the distribution remains almost the same through all axial slices; this was also seen for the 50mil case, with the exception of the inlet plane. With the onset of whirl, a small region of maximum velocity is seen to develop close to the maximum clearance region, the magnitude of which seems to grow with increasing whirl. This is consistent with the location of the high velocity region seen in the axial velocity distributions as well, although the radial velocity plots show faster dissipation of this region.

The tangential velocity distributions, shown in Fig. A-24 (c) follow the trends shown by the radial velocity distributions but the locations of the high velocity regions are a little different. The tangential velocities in for the 5mil case are overall higher than that of the 50 mil case. These high velocities are located close to the minimum clearance region which indicates the effect of the flow accelerating through the small clearance region. In case of the radial velocity profiles, the maximum velocity regions were seen to be almost entirely in the maximum clearance region whereas in case of the tangential velocity profile, it is the minimum velocity regions that are along this area. At a whirl ratio of zero, the trends resemble that seen for the 50mil case initially, but the dissipation
of the low velocity region is enhanced owing to the higher velocity magnitudes. As the whirl ratio is increased, it is seen that higher tangential velocities develop towards the exit as a result of increased centripetal acceleration. The increase in the azimuthal velocity at the exit is also striking and is seen to go from a value of 12m/s for the statically eccentric seal to 30m/s for a whirl ratio of 1.0.

The effect of the varying whirl ratio on the pressure distribution is shown in Fig. A-24 (d). The initial observation made is that the variation of the pressure ranges is not as large as in case of the 50mil seal. For the statically eccentric seal, the location of the region of maximum pressure is close to the pressure side on the minimum clearance region and is seen to decrease rapidly within the next few axial slices. The magnitude of the minimum pressure increases gradually with the onset of whirl and with increasing whirl ratio – this is observed in the exit planes of the seal. The overall trend does not vary greatly with the change in whirl ratio, a variation is however seen in the location of the maximum pressure region which is initially located close to the suction side but moves to the pressure side for whirl ratios greater than 0.6. It should be recalled that in case of the 50mil seal, there was a direct dependence of the maximum axial velocity region on the minimum pressure region for all whirl ratios and this relation clearly does not hold for the 5mil case. The pressure distribution for this case could be compared to that of a journal bearing where a similar behavior of the high pressure regions is observed.

Comparing Fig. A-24 (a)-(c) with 18(a)-(c) the effect of the change in clearance can be studied. For the axial velocity distributions, the most obvious change observed is that the location of the high velocity region is flipped from the minimum to the maximum
clearance region. The other observation made in terms of the highest velocities being seen in the statically eccentric seal, then decreasing with the onset of whirl until a whirl ratio of 0.4 and then increasing again, holds for the 5 mil case as well. The ranges on the radial velocity distributions are markedly higher than that for the 50 mil seal and the trends are significantly different as well. The 50 mil case shows one region of maximum velocity on the inlet plane which dissipates completely for the next axial plane but this contrasts the trend for the 5mil case where the region of high velocity is constant through all axial planes. This could again be the effect of the clearance in that the larger clearance causes faster dissipation. The significant conclusion that can be made from this comparison is that the flow field in case of the 50mil seal was momentum dominated while that in case of the 5mil seal is viscosity dominated.

**Impact of Varying Mass Flow Rate on Flow Field**

In order to further investigate and understand the viscous flow behavior for the 5mil seal, the mass flow rates were increased and the resulting flow fields studied. This was done for the statically eccentric seal (which was less computationally intensive) for two different mass flow rates of twice and four times the original flow rate of 0.487 kg/s which resulted in a pressure drop of 9.03e5 Pa and 2.74e6 Pa respectively compared to 3.52e5 Pa for the original case. The results presented in Fig. A-27 gives the axial, radial, azimuthal velocities as well as the pressure distributions for the two cases. On comparing these flow fields to that seen for the initial mass flow rate, it is possible to understand the characteristics of the flow fields better.

Comparing the axial velocity distributions, it is clearly seen that the overall locations of the maximum and minimum velocity regions remain unchanged. An impact of
changing the mass flow rate is however seen directly on the magnitude of this velocity component – the simulations with twice the mass flow rate show a maximum velocity of 20m/s (twice that of the initial mass flow rate) while that with four times the mass flow rate show high velocities of up to 40m/s (four times that seen initially). The constant pattern of the flow field indicates that viscosity is still the dominant effect for the new flow rates. The changes in the radial flow fields are minimal. At the seal inlet, for the azimuthal velocity, the location of minimum value rotates towards the suction side and into the minimum clearance region for the maximum mass flow rate. The region of minimum azimuthal velocity persists further into the seal as the residence time of the fluid is less for the larger flow rate. This results in less time for the rotor to impart tangential momentum and hence smaller azimuthal velocities.

The pressure contours show the same viscosity dominated distribution with larger magnitudes. From these results and comparisons, it can be concluded that the discussions made in case of the initial mass flow rate are valid for these new cases as well and the physics of the flow remains the same. It is thus seen that in the case of small clearances, the characteristics of the flow differ significantly from that of larger clearances and would therefore result in different stability behavior.

**Computation of Rotordynamic Coefficients**

As in case of the 50mil clearance, the rotordynamic coefficients are to be computed for the present case using the same theory used earlier. This involved obtaining the radial and tangential forces which FLUENT can directly report, for each whirl ratio. Fig. A-26 (a) shows the radial force variation with the whirl ratio. These components seem to be of a lower magnitude than the ones for the 50mil case but show the same trend
of decreasing forces until a ratio of 0.4. However, unlike the latter case, the expected trend of increase in the forces is seen from whirl ratio 0.4 to 1.0. This shows the impact of the viscosity dominated flow which leads to a smoother variation of the force with increasing whirl ratio. Also, the viscosity seems to create a damping-like effect which led to lesser deviations in the magnitudes of the forces over whirl ratio - for the 50mil case, the magnitude of the forces ranged from 370 N to 560 N while for the 5mil case they ranged from 260N to 360N.

The tangential force distribution is shown in Fig. A-26 (b), and like the radial forces, they initially show the same trend as the 50mil case; the magnitudes however are larger. Past a whirl ratio 0.2 however, the force components which were seen to initially decrease and then increase in case of the 50mil clearance, are seen to uniformly increase for the 5mil case. This is the effect of the larger radial and tangential velocities which were observed in the flow field of the case of the small clearance. The effect of the ‘damping’ was seen in this case as well leading to a smaller variation in the magnitude of forces. The variation of the line of action of force with the whirl ratio is shown in Fig. A-26 (c). As in case of the 50mil seal, it is again seen that there is a flip in the direction between whirl ratios 0.4 and 0.6 which corresponds to the changes in the pressure distribution – the region of maximum velocity was seen to move from its location close to the suction side to close to the pressure side for this change in whirl ratios.

On observing both the radial and tangential force variation, it is seen that they closely follow the expected trends from experiments, as measured by Kanemori and Iwatsubo [25]. Analyzing the work done by Xi and Rhode [26] in detail, it is seen that there are factors which can explain the discrepancies that were seen in case of the larger
clearance of 50mil. The first observation made is that the simulations performed for the former case were for a clearance of 0.394mm as compared to the 1.27mm (50mil) that was done for this study – this indicates that the theory may not hold for large clearances and it might be necessary to investigate alternate methods to determine the rotordynamic coefficients. The other differences seen are in terms of the pressure drops and the axial Reynolds numbers where the values used by Kanemori and Iwatsubo for both these factors were much lower than those present for the 50mil case in the present study.

The rotordynamic coefficients are obtained by using a straightforward curve fit for the variation of the radial and tangential forces as seen in Fig. A-27(a) and (b) respectively. As predicted by Childs’ theory, a second degree polynomial gives an accurate fit for the data and the rotordynamic coefficients are directly obtained from the curve fit equations which are displayed alongside the curve fits. From the equations, the values are obtained for the rotordynamic coefficients are shown in Table B-4.
SUMMARY AND CONCLUSIONS

The present work investigated the impact of orbit path, whirl ratio and clearance on the flow field, and consequently the rotordynamic coefficients for a smooth annular seal. This was done using the FLUENT CFD code that incorporated a deforming mesh and user defined functions in order to simulate the seal motion. Childs’ theory [22] was then applied to the results in order to determine the rotordynamic coefficients.

The initial simulations were performed for a smooth annular seal with a clearance of 1.27mm (50mil) whirling in a circular orbit. The results for the statically eccentric case and that of a whirl ratio of 1 were directly compared to experimental data. A fair degree of agreement was obtained, hence validating the model and the simulation technique used. Varying the whirl ratio at intervals of 0.2 examined the impact of the whirl ratio on the flow field. The ‘flipping’ of the location of maximum axial velocity region is seen at around a whirl ratio of 0.4 as was the case in experimental data. Also, the magnitude of maximum axial velocity began high for the statically eccentric seal, decreased with the onset of whirl to a minimum at a whirl ratio of 0.4 and increased with increasing whirl ratio.

Although the radial velocities were much lower than the axial component, an interesting observation made here was that for this component, the region of maximum velocity rotated in a direction opposite to that of the axial velocity component. The azimuthal velocity component did not show significant variation with whirl ratio towards the exit plane. The contours were almost identical for all whirl ratios. The radial and tangential force components from results of the simulations were then plotted as a
function of whirl ratio. It was attempted to curve fit the values using a second order polynomial. It was seen that the data did not entirely conform to the same. This could be attributed to the fact that the clearances could have been too large to apply this theory. Another possible reason for this discrepancy could be the presence of Goertler vortices which the simulations were unable to capture.

The effect of the orbit path was then studied by simulating a linear, back and forth motion. Analogous to the change in whirl ratio, the rotor orbit speed was varied. The initial analysis consisted of a study of the flow field for different positions of the rotor for the case when the orbit path speed was the same as the rotor spinning speed (equivalent to a whirl ratio of 1). The axial velocity values seen in the case of the linear orbit were substantially larger than that seen in the circular orbit. Also, the locations of maximum velocity region differed in terms of the fact that there were two regions concentrated close to the pressure and suction sides for the linear orbit while the circular orbit showed a single region in the minimum clearance side. These locations were however consistent with the pressure distributions which showed maximum pressure regions along the same locations.

The trend seen in case of the axial velocities in terms of both relative magnitudes and regions of high velocity is repeated for the radial velocity contours as well. The azimuthal velocity contours, in addition to this trend show the effect of the spinning of the rotor where the high velocity region is seen to move in the same direction as the rotor as the flow moves downstream. The impact of the varying whirl ratio was barely seen in case of the axial velocity contours where a slight variation of the locations of maximum velocity was observed. The radial velocity contours on the other hand did show more
noticeable changes with the whirl ratios. A whirl ratio of 0.2 showed a small region of high velocity on the suction side which was seen to develop with increases in whirl ratio till 0.8 and decreased for a whirl ratio of 1.0. The azimuthal velocity distributions show little change with the whirl ratio. One important observation made however is that the area of impact of the thin boundary region close to the rotor wall reduces with increase in whirl ratio and this is attributed to the increase in centrifugal force. Childs’ theory for the computation of rotordynamic coefficients could not be applied here owing to the non-circular motion. The resulting forces were thus analyzed. They were much larger compared to the case of the circular orbit and were seen to be the greatest for the case when the rotor speed was the same as the orbit path speed.

The impact of the clearance was then studied by simulating a seal with one tenth the original and one tenth the mass flow rate from the previous case for the case of a circular orbit. The initial analysis involved comparing the two different clearances at a whirl ratio of 1.0. From the axial velocity contours it was seen that the behavior of the flow field had entirely changed with the change in clearance. The expected trend of a maximum velocity region in the minimum clearance side (as in case of the 50mil) was not observed and this region was in fact located at the maximum clearance region. As the flow progressed downstream, this region was seen to however rotate towards the pressure side, i.e. in the direction of the spinning rotor. The radial velocities were still low over the entire seal. The tangential velocity components showed a major impact of the change in clearance – the magnitudes were almost twice that seen in the 50mil case and show rapid development downstream. This could indicate that the clearance is so small that the entire region is within the boundary layer, leading to a viscosity dominated flow pattern.
The impact of the whirl ratio on the axial velocity is seen in terms of the fact that the regions of maximum velocity seem to rotate towards the suction side with increasing whirl ratio, which is due to the higher centripetal forces here. The radial velocity contours show the highest magnitudes in case of the statically eccentric seal which reduce with the onset of whirl. A small region of high velocity is then seen to develop, which becomes more apparent with increase in whirl ratio. The most significant impact of the change in whirl is seen in case of the azimuthal velocity distributions which show magnitudes more than twice that seem for the statically eccentric case, at a whirl ratio of 1. The computation of rotordynamic coefficients for this case was seen to accurately follow a second degree polynomial. It can hence be concluded that it may be necessary to investigate alternate techniques for computing these coefficients for seals of large clearances. The small clearance seal was clearly viscous dominated, showing characteristics of a journal bearing. The large clearance seal was momentum dominated and did not agree with the rotordynamic bulk flow models, which the small clearance simulations did. The proper characteristics of these two cases and the transition between them need to be studied in the light that turbomachines operate within both regimes.

Through this work it has been determined that there are a few areas that need to be studied more intensively, one of which would be to extend the technique developed here to more arbitrary orbit paths. This would not only provide a more complete stability study but also enable the reduction of experimental testing costs. Another more interesting result of this study was the change in the flow physics for the smaller clearance – this would surely need a more thorough analysis including an investigation of different Taylor and Reynolds numbers and their effect on the flow.
The author hopes that the present study has been a step forward in simulating and investigating annular seals and also that this study has led to a few interesting findings that could hold potential answers.
REFERENCES


Fig. A-1 Cross-sectional view of the test rig with the annular seal installed (Suryanarayanan, 2003)

Fig. A-2 Dimensions of Annular Rotor (Johnson, 1989)
Fig.A- 3 Meshed seal geometry

Fig.A- 4 Wall $y^+$ distribution for a typical simulation.
Pressure-Based Segregated Algorithm

1. Update properties
2. Solve sequentially: $U, V, W$
3. Solve pressure-correction (continuity) equation
4. Update mass flux, pressure, and velocity
5. Solve energy, species, turbulence, and other scalar equations

Fig. A-5 Overview of the pressure-based segregated algorithm

Fig. A-6 Sectional view of test rig highlighting simulated region and the step
Fig.A- 7 Interface zones from the seal and swirl ring

Fig.A- 8 Path of fluid flow in seal (Shresta, 1993)
Fig.A-9 Detailed drawing of the seal entrance region [2]

Fig.A-10 Grid independence study
Fig.A-11 (a) Eccentric positions and their nomenclature [27]

Fig.A-12 Finite differencing mesh used with LDA data [27]

Fig.A-11 (b) Direction of forces and line of action
Fig.A-13 Mesh used in simulation
Fig. A-14(a) Axial, radial and tangential velocity distributions on minimum clearance side (experimental data)
Fig. A-14 (b) Axial, radial and tangential velocity distributions on suction side (experimental data)
Fig.A- 14 (c) Axial, radial and tangential velocity distributions on maximum clearance side (experimental data)
Fig.A-14 (d) Axial, radial and tangential velocity distributions on pressure side (experimental data)
Fig. A-15 (a) Axial, radial and tangential velocity distributions on minimum clearance side (FLUENT simulations)
Fig.A-15 (b) Axial, radial and tangential velocity distributions on suction side (FLUENT simulations)
Fig. A-15 (c) Axial, radial and tangential velocity distributions on maximum clearance side
(FLUENT simulations)
Fig. A-15 (d) Axial, radial and tangential velocity distributions on pressure side (FLUENT simulations)
Fig.A-16 (a) LDA data and FLUENT simulation results for Z/L = 0.036 (whirl ratio 1.0)
Fig.A-16 (b) LDA data and FLUENT simulation results for Z/L = 0.11 (whirl ratio 1.0)
Fig. A-16 (c) LDA data and FLUENT simulation results for Z/L = 0.22 (whirl ratio 1.0)
Fig. A-16 (d) LDA data and FLUENT simulation results for $Z/L = 0.77$ (whirl ratio 1.0)
Fig. A-16 (e) LDA data and FLUENT simulation results for $Z/L = 0.86$ (whirl ratio 1.0)
Fig.A-17 (a) Axial velocity contours for varying whirl ratio (Inlet at top left, exit at bottom right, direction of rotation – counterclockwise)
Fig.A-17 (b) Radial velocity contours for varying whirl ratio (Inlet at top left, exit at bottom right, direction of rotation – counterclockwise)
Fig.A-17 (c) Tangential velocity contours for varying whirl ratio (Inlet at top left, exit at bottom right, direction of rotation – counterclockwise)
Fig. A-17 (d) Pressure velocity contours for varying whirl ratio (Inlet at top left, exit at bottom right, direction of rotation – counterclockwise)
Fig.A-18 (a) Variation of radial force with whirl ratio

Fig.A-18 (b) Variation of line of force with whirl ratio
Fig. A-18 (c) Variation of tangential force with whirl ratio

\[ y = 0.5584x^2 - 0.4938x + 0.8165 \]

Fig. A-19 (a) Variation of radial impedance with whirl ratio
Fig. A-19 (b) Variation of tangential impedance with whirl ratio

\[ y = -0.049x^2 + 0.0725x - 0.1028 \]
Fig.A- 20 (a) Axial velocity contours for varying whirl ratio (Inlet at top left, exit at bottom right)
Fig.A- 20 (b) Radial velocity contours for varying whirl ratio (Inlet at top left, exit at bottom right)
Fig. A-20 (c) Tangential velocity contours for varying whirl ratio (Inlet at top left, exit at bottom right)
Fig. A-20 (c) Pressure contours for varying whirl ratio (Inlet at top left, exit at bottom right)
Fig.A- 21(a) – Variation of forces and line of action over cycle for an orbit path speed 20% of rotor speed

Fig.A- 21(b) – Variation of forces and line of action over cycle for an orbit path speed 40% of rotor speed
Fig. A-21(c) – Variation of forces and line of action over cycle for an orbit path speed 60% of rotor speed

Fig. A-21(d) – Variation of forces and line of action over cycle for an orbit path speed 80% of rotor speed
Fig. A-21(e) Variation of forces and line of action over cycle for highest orbit path speed

Fig. A-22 Grid independence study for annular seal of 5mil clearance
Fig. A-23 (a) – Comparisons of axial, radial and tangential velocity contours for seals of 5 and 50 mil clearances at an axial location of \( Z/L = 0.036 \).
Fig. A-23 (b) Comparisons of axial, radial and tangential velocity contours for seals of 5 and 50 mil clearances at an axial location of $Z/L = 0.11$
Fig. A-23 (c) Comparisons of axial, radial and tangential velocity contours for seals of 5 and 50 mil clearances at an axial location of Z/L = 0.22
Fig. A- 23 (d) Comparisons of axial, radial and tangential velocity contours for seals of 5 and 50 mil clearances at an axial location of Z/L = 0.77
Fig. A- 23 (e) Comparisons of axial, radial and tangential velocity contours for seals of 5 and 50 mil clearances at an axial location of Z/L = 0.96
Fig. A-23(f) Comparison of variations of pressure along the length for 5 and 50mil seals
Fig. A-24(a) Axial velocity contours at varying whirl ratios
Fig. A-24(b) Radial velocity contours at varying whirl ratio
Fig. A- 24(c) Tangential velocity contours for varying whirl ratio
Fig. A- 24(d) Pressure contours for varying whirl ratio
Fig. A-25 Comparison of axial, radial and tangential velocity contours along with pressure contours for mass flow rates twice and four times the original.

Fig. A-26(a) Radial force distribution with varying whirl ratio
Fig. A-26(b) Tangential force distribution with varying whirl ratio

Fig. A-26(c) Variation of line of action of force with whirl ratio
Fig. A-27(a) Least squares curve fit of radial impedance for rotordynamic coefficients

\[ y = 7.3197x^2 - 7.097x + 5.588 \]

Fig. A-27(b) Least squares curve fit of tangential impedance for rotordynamic coefficients

\[ y = -0.2308x^2 + 0.5284x - 0.7465 \]
Table B-1 Rotordynamic coefficients for a smooth annular seal of 50 mil clearance whirling in a circular orbit

<table>
<thead>
<tr>
<th></th>
<th>K (MN/m)</th>
<th>k (MN/m)</th>
<th>C (MN/m)</th>
<th>c (MN/m)</th>
<th>M (MN/m)</th>
<th>m (MN/m)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>-0.5584</td>
<td>-0.049</td>
<td>-0.0725</td>
<td>0.4938</td>
<td>0.8165</td>
<td>0.1028</td>
</tr>
</tbody>
</table>

Table B-2 Pressure comparisons for 5 and 50mil

<table>
<thead>
<tr>
<th>Factor being compared</th>
<th>5mil case</th>
<th>50mil case</th>
</tr>
</thead>
<tbody>
<tr>
<td>Range of pressure</td>
<td>-370,500 to -364,000 pa</td>
<td>-70,000 to 0 pa</td>
</tr>
<tr>
<td>Pressure drop</td>
<td>6500 pa</td>
<td>70000 pa</td>
</tr>
<tr>
<td>Changes over varying whirl</td>
<td>± 3,500 pa</td>
<td>± 35000 pa</td>
</tr>
<tr>
<td>Maximum pressure region</td>
<td>Located around 15% along the entire length, magnitude decreases with increase in Z/L</td>
<td>Low at the inlet, centered at 55%. Location remains the same but value increases towards the exit</td>
</tr>
<tr>
<td>Minimum pressure region</td>
<td>Located along the entire length from 50% onwards. Magnitude decreases with increase in Z/L.</td>
<td>Magnitude lowest at the inlet, increases with increase in Z/L</td>
</tr>
</tbody>
</table>
Table B-3 Rotordynamic coefficients for a smooth annular seal of 5mil clearance whirling in a circular orbit

<table>
<thead>
<tr>
<th></th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$K$ (MN/m)</td>
<td>5.588</td>
</tr>
<tr>
<td>$k$ (MN/m)</td>
<td>0.7465</td>
</tr>
<tr>
<td>$C$ (MN/m)</td>
<td>0.5284</td>
</tr>
<tr>
<td>$c$ (MN/m)</td>
<td>-7.097</td>
</tr>
<tr>
<td>$M$ (MN/m)</td>
<td>7.3197</td>
</tr>
<tr>
<td>$m$ (MN/m)</td>
<td>-0.2308</td>
</tr>
</tbody>
</table>
APPENDIX C
Fig.C-1 Representation of a deforming mesh for a 2D case

Fig.C-2(a) Variation of axial velocity with change in rotor position
Fig.C- 2(b) Variation of radial velocity with change in rotor position

Fig.C- 2(c) Variation of tangential velocity with change in rotor position
Fig.C- 2 (d) Variation of pressure with change in rotor position
VITA

Name: Aarthi Sekaran

Address: Department of Mechanical Engineering,
          Texas A&M University,
          College Station, TX 77840

Email Address: aarthi85@fastmail.fm

Education: B.E., Mechanical Engineering,
            Vasavi College of Engineering, India, 2006

            M.S., Mechanical Engineering,
            Texas A&M University, 2009